# NcStudio V10 Glass Cutting CNC System

## Users' Manual

8th 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 intended for operators. If it's your first time to use Weihong CNC system, we suggest that you read through this manual. If not, you can search for the information you need through table of contents.

With 6 chapters, this manual can be divided into 5 parts, 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: introduction to hardware, including chapter 1. This part gives an introduction to the basic configuration, installation, connection and uninstallation of the NcStudio system.
3) Part 3: introduction to software operation, including chapter 2, 3, and 4. In this part, detailed functions, the corresponding interfaces, operations are introduced. This part will be an intuitional guidance to operators in real practice.
4) Part 4: maintenance, including chapter 5. In this chapter, possible problems and their countermeasures are listed, aiming to help users to respond instantly and take effective measures when possible failure occurs.
5) Part 5: appendix, including chapter 6. This part gives an introduction to NcStudio, system parameters, and the *Software License Agreement*.

## 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 |
|---|---|---|
| 2016.02 | R8 | Contact information updated. |
| 2015.12 | R7 | Document structure updated, including reduction of software interface introduction and addition of operation, function and troubleshooting. |

## Precautions

Precautions can be divided into cautions and warnings 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;

**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 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 tuyere. 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

**WARNING**

cabinet;

- Space between this device and other equipments 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;
- 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

## WARNING

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 (1) 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 Installation

## 1.1 Basic Configuration for Host Computer

◆ **Host Computer**

| | |
|---|---|
| CPU: | Main frequency 1G or above |
| Memory: | 512 M or above |
| Hard disk: | 20G or above |
| Display adapter: | 1024*768 at least |
| Display: | VGA 14" or above |
| CD-ROM: | 4X or higher (optional) |
| Mainboard expansion slot: | More than 1 PCI slot (for Communication Card PM85A) |

## 1.2 Installation and Connection of NcStudio System

NcStudio system consists of two parts: the NcStudio software and a motion control card. Please complete the installation of the software before installing the motion control card.

Please uninstall the old version NcStudio before installing the new version. Regarding how to uninstall the software, please refer to section 1.3. You can also directly install the new version. Below is the installation method.

### 1.2.1 Installation of NcStudio Software

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 **Computer** icon, then double click the CD-ROM icon. Find and double click the icon  . The first dialog box that appears is “Language of the theme". Switchover between languages is supported in NcStudio V10.
3) A dialog box will pop up asking users whether to delete the old version before going on further installation. Click **OK**.
4) A dialog box will pop up asking users whether to save configuration parameters. Click **OK** or **No** according to your needs.
5) Installation begins. The software will be installed under the default directory, C:\Program Files\Naiky. The installation progress bar will show the installation progress.

6) During installation, a dialog box will pop up asking users whether to store generated files and executive files separately. **No** is recommended.

7) After Installation completes, a dialog box will pop up asking users whether to shut down the computer. **OK** is recommended.

**CAUTION**

1) Please skip from step 2) to step 5) if it there is no NcStudio software already installed in the computer.

2) In Win7, please uninstall any NcStudio software drivers before installating the software. And update the driver manually after the installation completes.

## 1.2.2 Installation and Connection of Control Card

The steps to install NcStudio Motion Control Card are as follows:

1) Power off the computer and open the chassis cover. Insert the card into an available and well-matched expansion slot. During installation, gently press the two sides of the card to make sure it is firmly inserted into the slot and well connected with the computer baseboard. Then tighten the screw of the control card, and close the lid.

2) Similar with the above step, please insert external devices or periphery equipment into the available and well-matched slots.

3) Installation completes. Please restart the computer.

The overall wiring diagram is shown as Fig. 1-1:

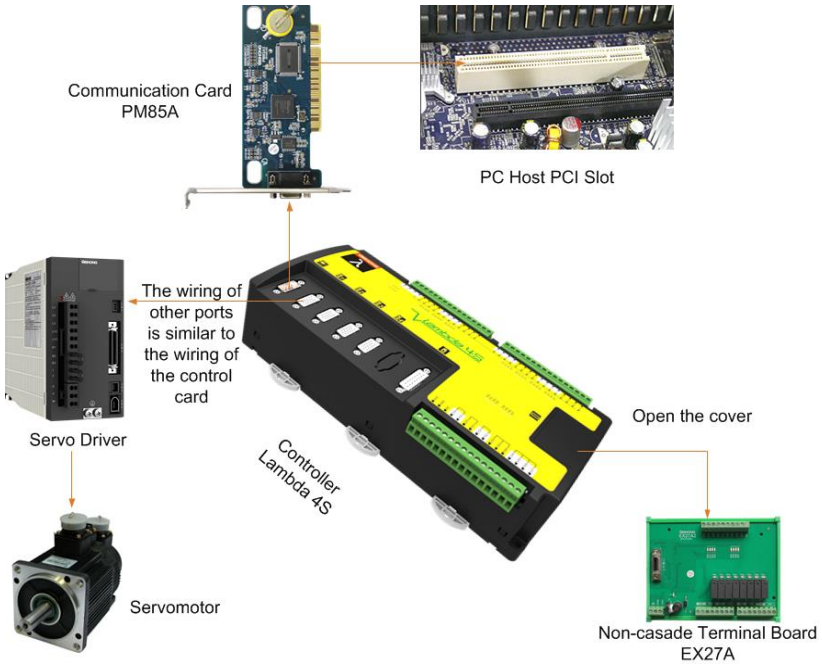



Fig. 1-1 Overall Wiring Diagram

**CAUTION**

The diagram above is only for reference. Please consider the actual situation.

### 1.2.3 Manually Update Hardware Driver

After installations of the software, communication card and other hardware, the user must manually update the hardware driver. Otherwise, the software cannot be launched normally. The specific updating steps are as follows:

1) Right click **Computer**, select **Properties**, and then click **Device Manage**r. Click 数控适配器 and select 维宏数控适配器 **Lambda** 型, right click on it and select **Update Driver Software…** , as shown in Fig. 1-2.

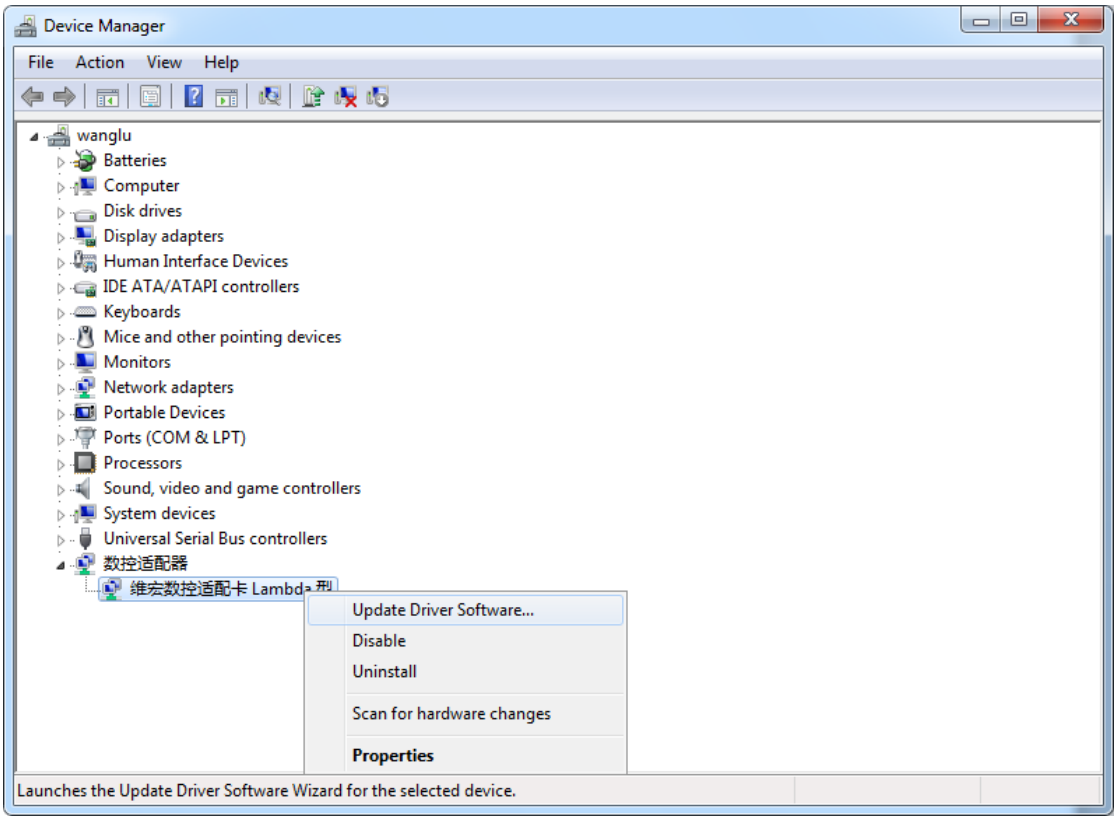



Fig. 1-2 Update Driver Software

2) A dialog box as Fig. 1-3 will pop up. Click “Browse my computer for driver software”.

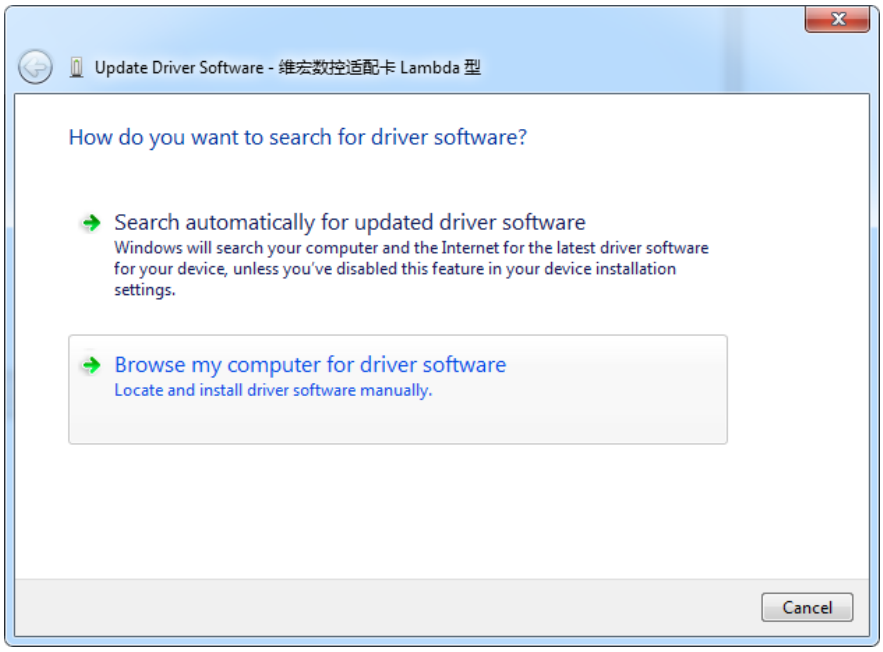



Fig. 1-3 Search for Driver Software

3) A dialog box as Fig. 1-4 will pop up. Click “Let me pick from a list of device drivers on my computer”.

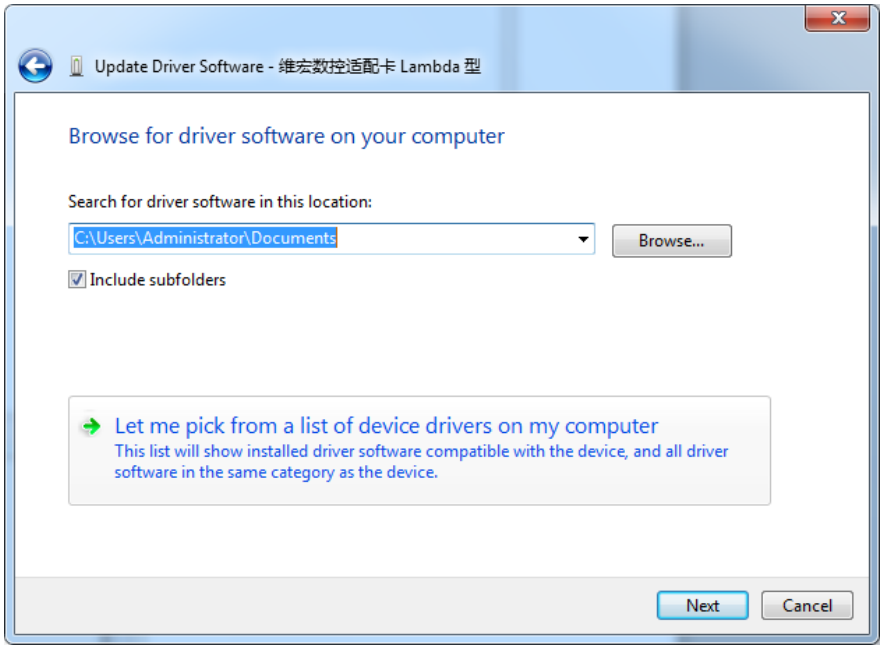



Fig. 1-4 Choose Search and Installation Option

4) Click **Have Disk...** button, as shown in Fig. 1-5. And click **Browse…** button in the next dialog box, shown as Fig. 1-6.

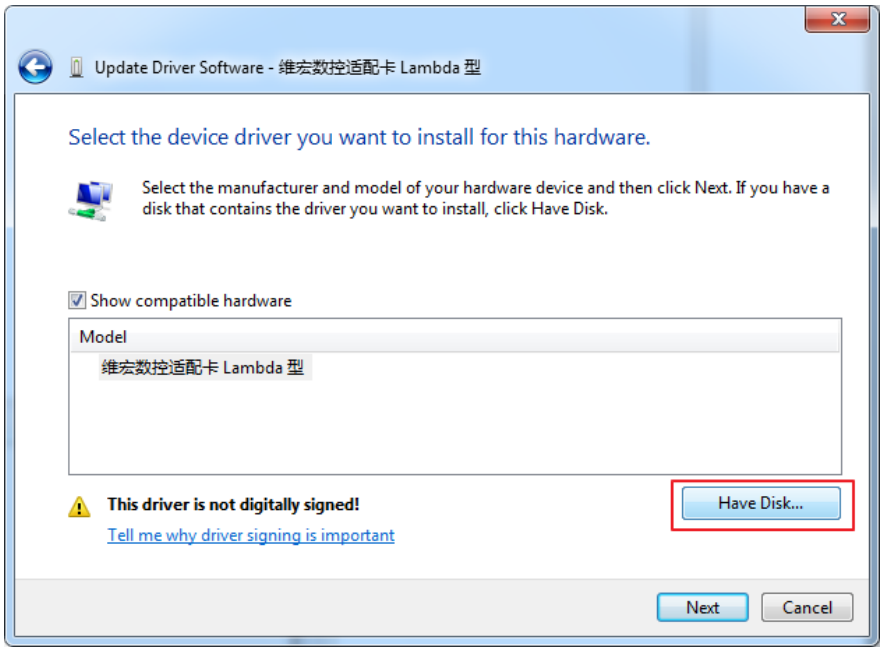



Fig. 1-5 Select the Driver Manually

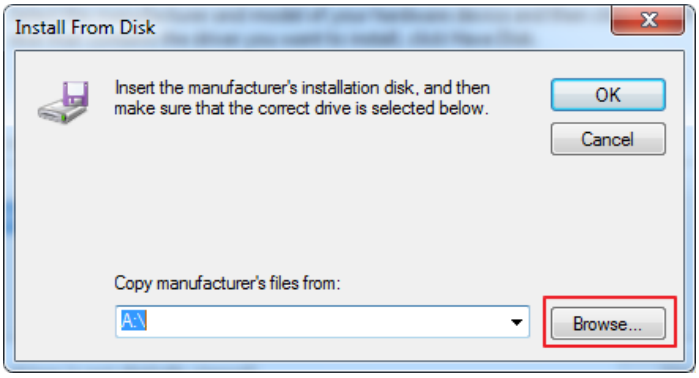



Fig. 1-6 Select the Target File of Driver

5) In dialog box “Locate File”, select the hardware driver Lambda.inf, shown as Fig. 1-7. And click **Open**.

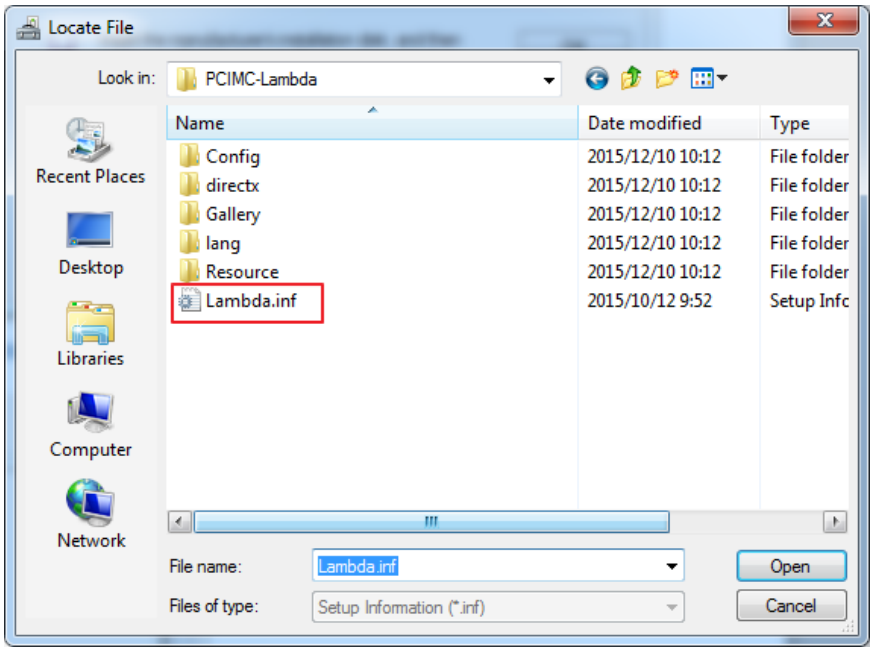



Fig. 1-7 Select the Hardware Driver

6) Then the dialog box “Install From Disk” pops up, showing the directory where the hardware driver is stored, as Fig. 1-8. Click **OK**.

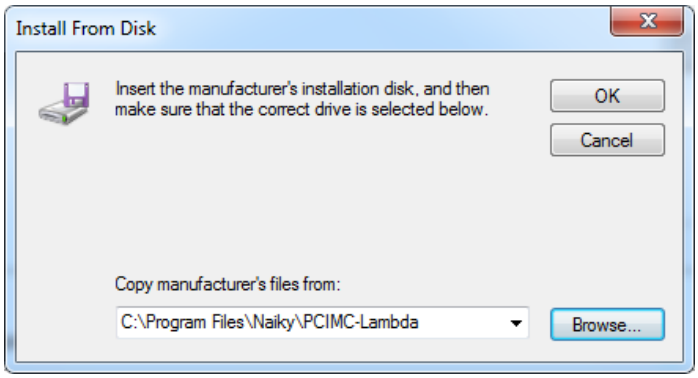



Fig. 1-8 Target File Directory Confirmation

7) Then click **Next** to start updating the Hardware driver.

8) When the updating is finished, click **Close**.

# 1.3 Uninstallation of NcStudio Software

NcStudio is green software. To uninstall it, all you need to do is to delete the folder “Naiky” under directory *C:\Program Files*, and the shortcut icons on the desktop as well as in the **Start** menu.

# 2 NcStudio Interface

NcStudio and NcEditor interfaces will pop up at the same time when the user opens the software NcStudio. NcStudio is for machining operation while NcEditor provides basic editing functions including tool path editing, and auxiliary functions. This chapter gives an introduction to interfaces and menus of the two.

## 2.1 NcStudio Interface

There are three operation mode,i.e. auto mode, manual mode and reference point mode in NcStudio CNC system. Below is a screenshot of the Auto Mode interface, shown as Fig. 2-1.

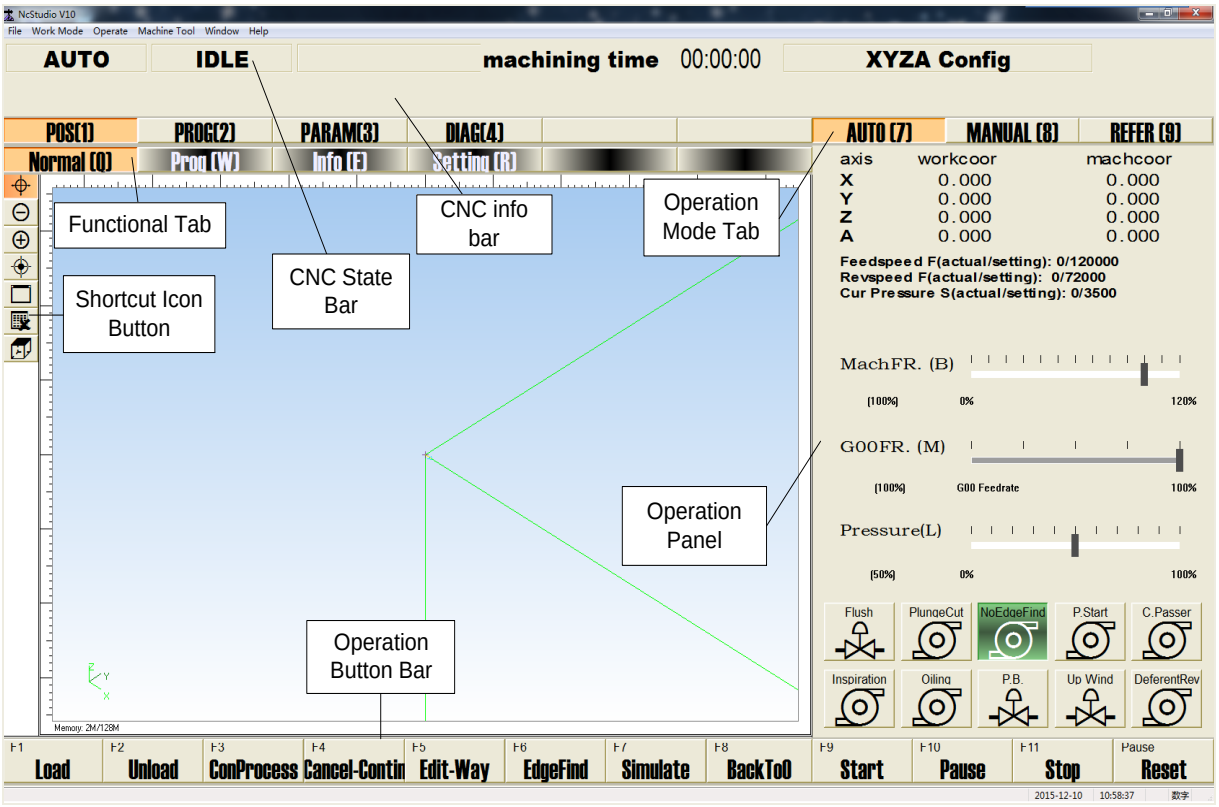



Fig. 2-1 Auto Mode Interface

**CAUTION**

The software interfaces of NcStudio CNC system under different configuration are diffferent, unless otherwise stated, the illustrations in chapter 2 and chapter 3 are based on XYZA Lambda series software.

As shown above, from the top to the bottom of the interface are the title bar, the menu bar, the CNC state bar and the CNC info bar. 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. At bottom of the interface is status bar, which shows the current date and time, and the current state of the capital lock, the number lock, and the scroll lock.

◆ **Title bar**

The title bar shows the software name. Its color shows whether it is currently active or inactive.

◆ **Menu bar**

The menu bar contains several pull-down sub-menu items. Execution of sub-menu items can make the machine execute related function or action, or change the state.

◆ **CNC state bar**

In the CNC state bar, current operation mode, operation state, additional information, machining elapsed time and configuration information are shown in order. User can read directly the operation state and the machining time, based on which the user can make decisions on next operation.

◆ **CNC info bar**

Three kinds of prompts, i.e. normal prompt, warning prompt and error prompt will be displayed in CNC info bar.

- Normal prompt: shows that CNC functions normally. The info bar is in normal color and the prompt texts are in black.
- Warning prompt: alerts the users the warning information. The info bar is in yellow and the texts are in black.
- Error prompt: shows that errors occur in the system and the system cannot run. Error prompts include emergency stop alarms, limit alarms, file error prompt, software error prompt and so on. The info bar is in red and the texts are in white.

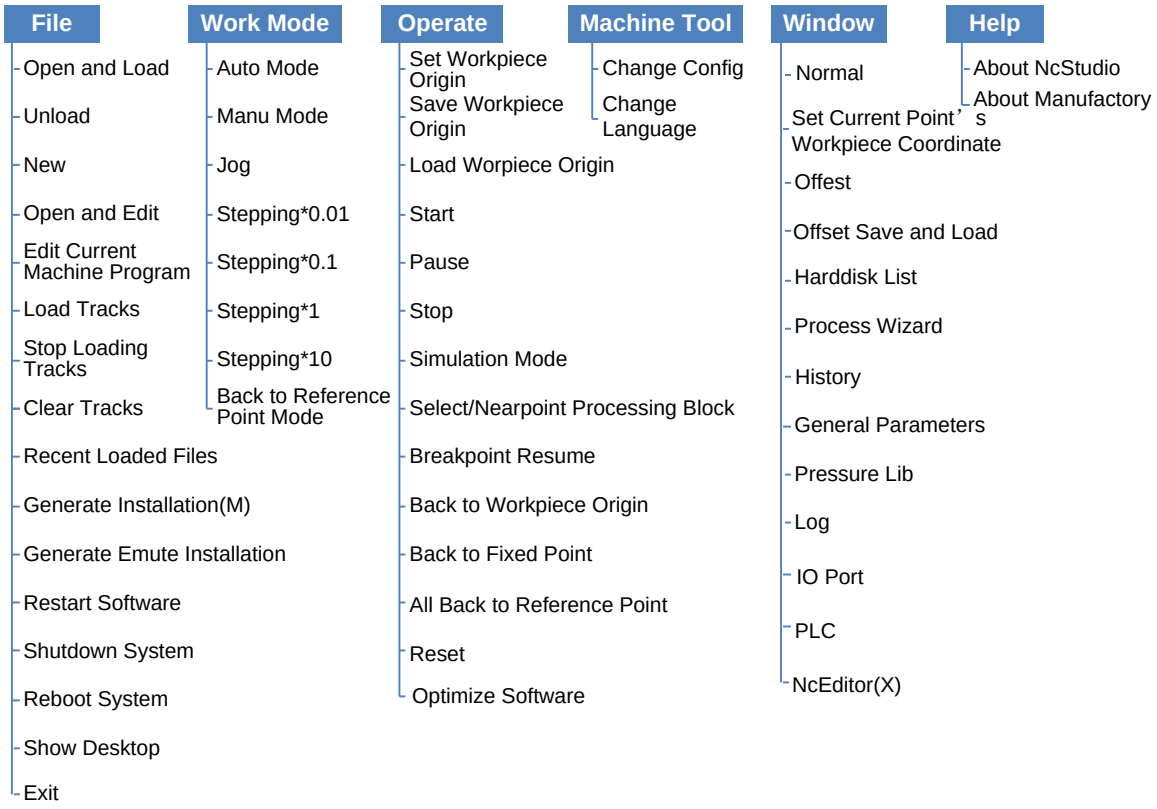
◆ **Operational mode button**

Users can switch among auto mode, manual mode and reference point mode by clicking the operational mode button. In each mode, the function buttons, the operational panel and operation buttons are different.

In auto mode, the system analyzes the loaded machining program files automatically and controls the machine tool to machining on the target workpiece. In manual mode, users control the machine tool manually. The reference point mode is designed to synchronize the internal coordinates in CNC system and the actual coordinates on the machine tool. The default operation mode is reference point mode, and user can execute back to origin operation through this mode conveniently.

# 2.2 NcStudio Menu

There are 6 main menus in NcStdudio, and under each main menu there are sub-menus. The overview diagram of menus is as follows.




**CAUTION**

Menus are different in different operation modes. This diagram is only a reference for menus in auto mode.

# 2.3 NcEditor Interface

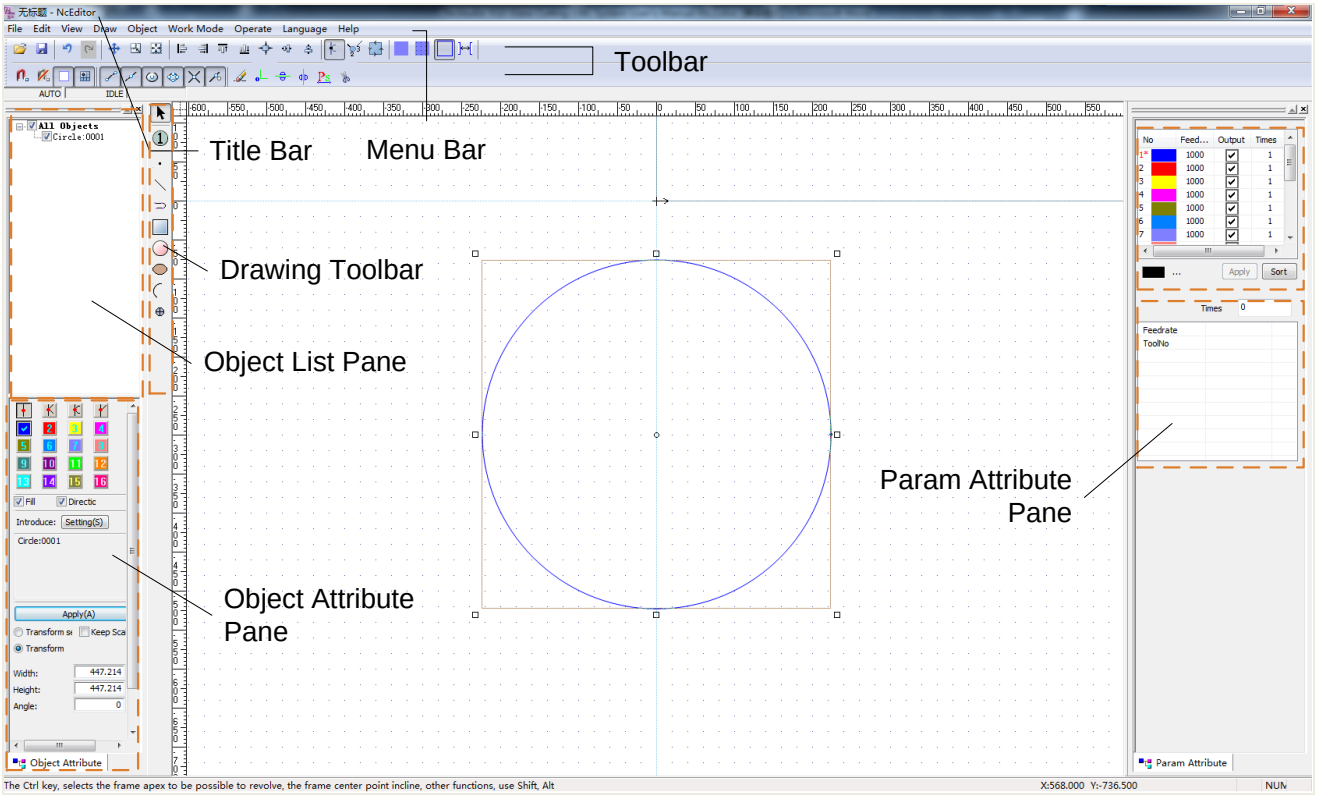



Fig. 2-2 NcEditor Interface

The interface of NcEditor is shown as Fig. 2-2.

Users can choose to open only NcStudio or NcEditor, or both of them by setting the value of parameter N87000. In NcStudio, click **Param** under tab **PARAM**, and then click button **All Param**, and you can find N87000 in Manufacturer parameter.

From the top to the bottom of NcEditor interface are the title bar, the menu bar, and the toolbar. The white area in the center is object editing space. The drawing toolbar on the left of the editing space corresponds to menu **Draw**. The objects in the object list window correspond to the object editing space.

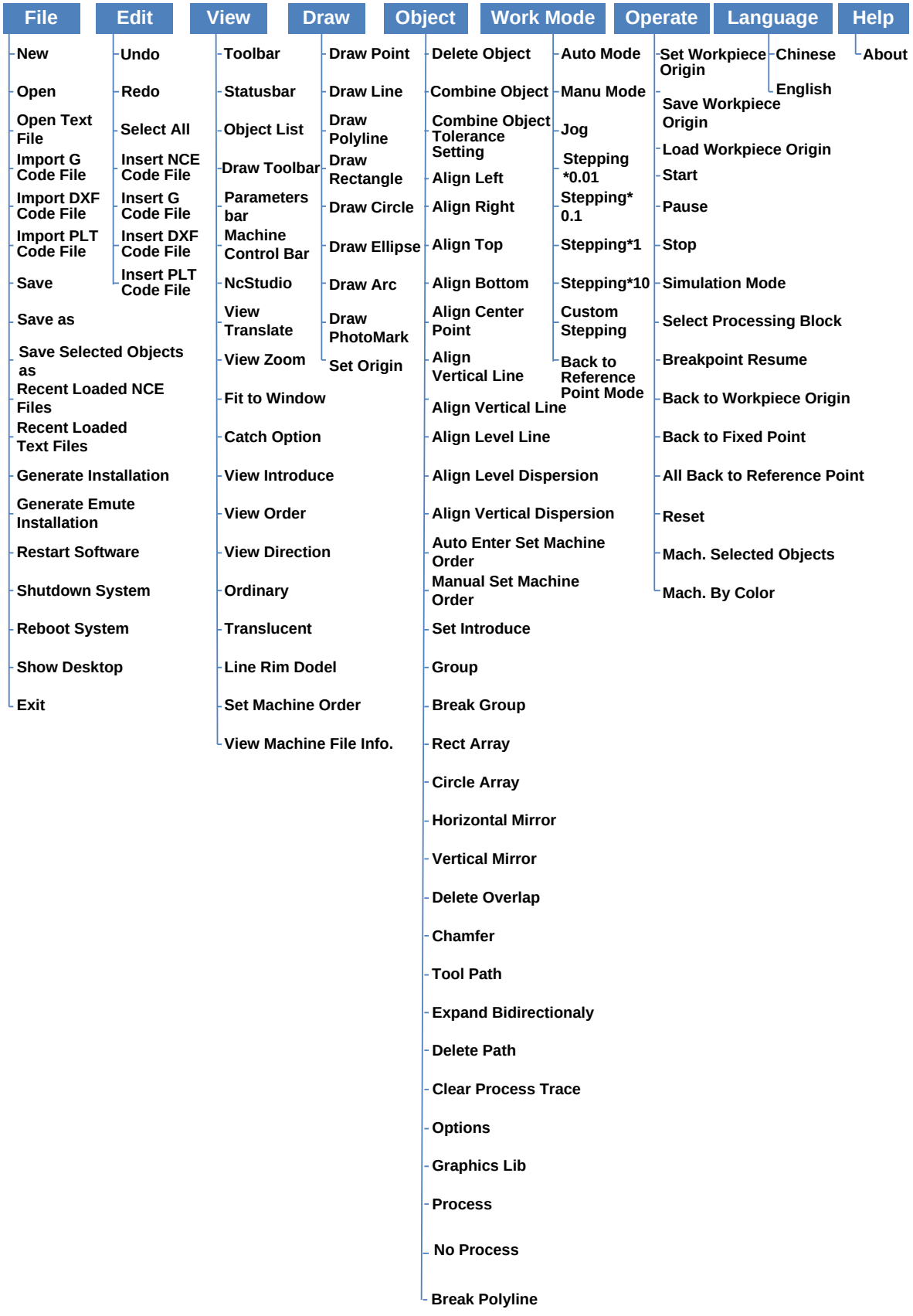
◆ **Object Attribute Window**

The object attribute window is used for setting the attribute of the currently selected object, including setting the tool introduction line, whether to fill the object, setting the precise coordinates of object, setting the machining direction and so on.

◆ **Param Attribute Window**

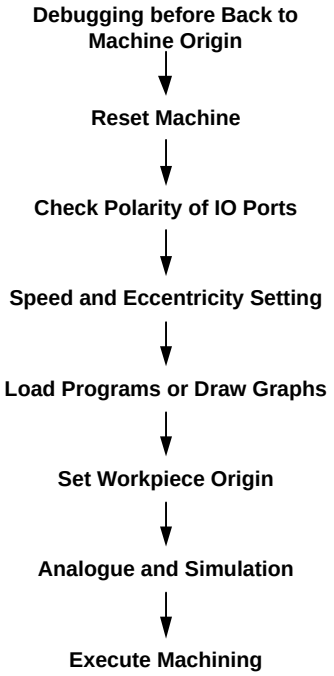
The param attribute window is divided into two parts. The top part is the layer color window and the bottom part is the layer attribute window. In layer color window, there are 16 colors with different number. And each color corresponds to one parameter, including speed, output and tool number. In two-dimension mode in NcEditor, the objects with the same color have same parameters, and if one color's parameter changes, the parameters of all objects in this color will change accordingly.

# 2.4 NcEditor Menu



# 3 Software Operation

After the software and the control card are installed correctly as section 1.2 described, users can open the software and begin debugging. The debugging flow chart is as follows.




## 3.1 Debug before Back to Machine Origin

### 3.1.1 Adjusting Machine Tool Axes

**Determine the positive direction of coordinate axes through the right-hand rule.**

The right-hand rule is illustrated in Fig. 3-1:

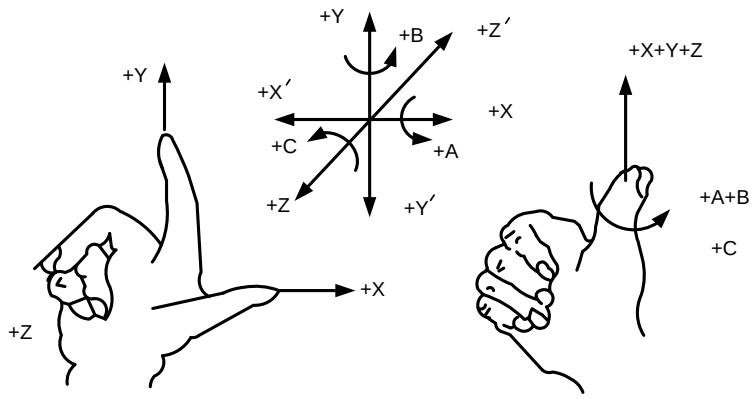



Fig. 3-1 Right-hand Rule Coordinate System

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

Z axis : The axis coincides with spindle axis and its positive direction (+Z) is the direction of the tool moving away from the workpiece, shown in Fig. 3-1.

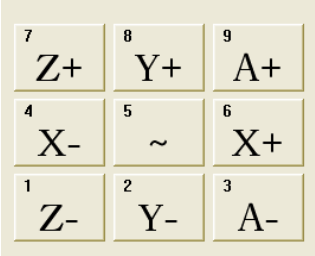
X axis: The axis perpendicular to Z axis and parallel to the clamped surface of workpiece. For a single column vertical milling machine, if the user faces the spindle and looks in the column direction, right moving direction is the positive direction of X axis (+X).

Y axis: Y-axis, shown in Fig. 3-1, together with X axis and Z axis formed the Right-hand Rule coordination system.

"A" axis in NcStudio Software ( instead of the A axis in Fig. 3-1) rotates with Z axis as the centerline, and is equal to C-axis in Fig. 3-1. Counter-clockwise direction is referred to as the positive direction for "A" axis. "X+" direction is the rotating starting position of "A" axis, i.e. zero degree position.

◆ **Adjust Coordinate Axes in Manual Mode**

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.

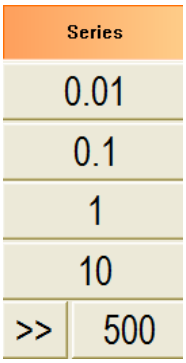



The screenshot of axis buttons is shown on the left. The number on the upper left of represents the shortcut numeric key. "~" button is the high speed button, and it has to be used along with other axis buttons.

For example, to move x axis at low speed in negative direction, click the "X-" button or press the "4" key on the keyboard. To move x axis at high speed in negative direction, press "4" and "5" at the same time on the keyboard.

Manual mode is subdivided into jog mode and stepping mode.

1) **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. 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. Only single-axis motion is supported in stepping mode.




2) **Stepping mode**: 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.

☞ **Parameters Related (Manufacturer Parameters)**

| Parameter | | Range | Default Value | Description |
|---|---|---|---|---|
| N10000 | X Axis Direction | 1; -1 | 1 | The moving direction of X/Y/Z/A axis. |
| N10001 | Y Axis Direction | 1; -1 | 1 | |
| N10002 | Z Axis Direction | 1; -1 | -1 | |
| N10003 | A Axis Direction | 1; -1 | 1 | |
| Through moving the machine tool manually, users can check whether the actual moving direction of the machine tool is consistent with the direction shown on operation buttons. If not, you should change the value of relevant parameters. For example, if you click on the **X+** button, the machine tool moves in negative direction of X axis, you should change the value of N10000 from "1" to "-1". | | | | |

## 3.1.2 Adjusting Pulse Equivalent

**Pulse equivalent (p)**: the moving distance of workbench or rotation degree of rotary axis corresponding to one pulse sent by CNC device, the minimum available distance controlled by CNC system as well.

The smaller the pulse equivalent is, the higher the machining precision and surface quality will be. The larger, the faster feedrate will be. Therefore, lower pulse equivalent should be set under the condition of meeting the demand of feedrate. The relationship between Max. feedrate and pulse equivalent is as follows:

Max. Feedrate = pulse equivalent X 60 X frequency

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

Max. federate =0.001 X 60 X 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.

$$\text{Mechanical Deceleration} = \frac{\text{Reducer Input Speed}}{\text{Reducer Output Speed}} = \frac{\text{Teeth No. of Driven Wheel}}{\text{Teeth No. of Driving Wheel}} = \frac{\text{Motor Speed}}{\text{Screw Speed}}$$

**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 $\theta$ 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:

$$\text{Pulse equivalent}=\frac{5\text{mm}}{\frac{360}{1.8}\times10\times1}=0.0025\text{mm/p}$$

◆ **Servo Motor**

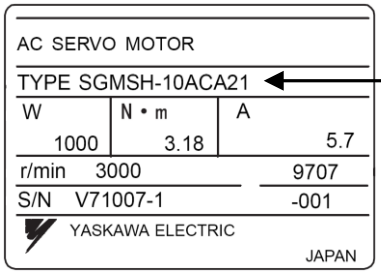
In general, set the default value of pulse equivalent as 0.001mm/p and calculate electronic gear ratio (B/A). Their relationship can be shown as:

$$\text{Electronic gear ratio }\frac{\text{B}}{\text{A}}=\frac{\text{encoder resolution}}{\frac{\text{screw pitch}}{\text{pulse equivalent}}}\times\text{mechanical deceleration ratio}$$

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

**Electronic gear ratio (B/A)**: the parameter of servo driver (take YASKAWA driver as an example, B is PN202 while A PN203). This ratio represents servo scales up or down the pulse frequency sent by CNC system. When B is larger than A, it means scaling up and vice versa. For example, provided the pulse frequency sent by CNC system is 100HZ, if the numerator of electronic gear ratio (B) is set as 1 while the denominator 2, the actual running speed of servo is 50HZ. On the contrary, if the numerator is set as 2 while denominator 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. A label plate of YASKAWA SGMSH type motor is as below, and the 4th character in motor type is the serial encoder specification, so the resolution of this motor is $2^{17}$, i.e. 131072.




Motor Type:
TYPE SGMSH-1 0 A C A 2 1
(The 4th Character)

| The 4th Character: Serial Encoder Spec. | | |
|---|---|---|
| Sign | Spec. | Remark |
| 2 | 17-bit absolute | Standard |
| C | 17-bit incremental | Standard |

Fig. 3-2 Name plate of servo motor-encoder resolution

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.

$$\text{Electronic Gear Ratio } \frac{PN202}{PN203} = \frac{2^{17}}{5/0.0001} \times 1 = \frac{131072}{5/0.0001} \times 1 = \frac{8192}{3125}$$

◆ **Rotary Axis**

The pulse equivalent of rotary axis refers to the rotation degree of the axis clamping the workpiece corresponding to each pulse. The difference of rotary axis movement from linear axis movement lies in that the screw pitch of rotary axis is 360 degrees. Therefore, in calculating rotary axis pulse equivalent, you just need to replace screw pitch with 360.

- **For Stepping Motor**

$$\text{Pulse Equivalent} = \frac{360}{\frac{360}{\text{stepping angle}} \times \text{subdivision} \times \text{mechanical deceleration ratio}}$$

- **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(Manufacturer Parameters)**

| Parameters | | Range | Default | Definition |
|---|---|---|---|---|
| N10010 | Pulse equivalent of X-axis | 1e-009~999 | 0.01 mm/p | It refers to the displacement or angle generated on the relative feed axis per control pulse. |
| N10011 | Pulse equivalent of Y-axis | 1e-009~999 | 0.01 mm/p | |
| N10012 | Pulse equivalent of Z-axis | 1e-009~999 | 0.01 mm/p | |
| N10013 | Pulse equivalent of A-axis | 1e-009~999 | 0.01 deg/p | |

**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) The pulse equivalent of the rotating axis A should be 3 or 5 times that of a linear axis.

## 3.1.3 Setting Travel Bounds of Workbench

The travel of workbench is the effective machining ranges in direction of X/Y/Z axis. Through setting the travel bounds in the software, the machine tool can be protected.

☞ **Related Parameters(Manufacturer Parameters)**

| Parameters | | Range | Default | Definition |
|---|---|---|---|---|
| N10020 | The lower travel bound in X axis | -99999~99999 | 0 mm | The machining coordinates of the lower travel bound in condition that the workbench travel range is checked effective. |
| N10021 | The lower travel bound in Y axis | -99999~99999 | 0 mm | |
| N10022 | The lower travel bound in Z axis | -99999~99999 | -10000 mm | |
| N10030 | The upper travel bound in X axis | -99999~99999 | 3600 mm | The machining coordinates of the upper travel bound in condition that the workbench travel range is checked effective. |
| N10031 | The lower travel bound in Y axis | -99999~99999 | 2600 mm | |
| N10032 | The lower travel bound in Z axis | -99999~99999 | 0 mm | |
| N10040~ N10042 | Check the effectivity of workbench travel range (X/Y/Z) | Y/N | Y | Whether run the workbench travel check |

**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 Reset Machine

By executing rest machine operation, the machine will return all axes to the reference point in order to correct the coordinate system.

When the software is started, operation mode is REF point mode by default, for convenience of execution of the function “Back to Machine Origin”. When the REF mode is active, the user can press F9 to return all axes to the reference points, also can press F1~F4 to return a certain axis to the reference point separately. In addition, the user can select **All back to REF Point** under menu **Operate** to return all axes to the REF point.

On some occasions, for example, when restart the system and continue from the interrupted point after normally shutdown of the system, the user does not have to execute “Back to Machine Origin” operation because current coordinates will be saved when NcStudio is normally closed. Besides, if the user is definitely sure of accuracy of the current position, he/she can enable the “Directly Setting” function.

After all axes have been returned to the reference point, a mark will appear next to each axis on the operation panel.

**CAUTION**

1) It is strongly recommended to return all axes to the REF point first before machining.
2) Only after all axes have returned to the REF point, the following functions can be enabled: soft limit activation, setting the fixed point and change tool.

# 3.3 Check Polarity of IO Ports

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

POS(1) PROG(2) PARAM(3) DIAG(4)
Log (Q) IOPort (W) PLC (E)

| Tag | Pin | P.. | PL... | Input S... | Description |
|---|---|---|---|---|---|
| InPort | | | | | |
| XC | | N | 00... | E,F:4m... | Encoder Zero of X-axis |
| YC | | N | 00... | E,F:4m... | Encoder Zero of Y-axis |
| AC | | N | 00... | E,F:4m... | Encoder Zero of A-axis |
| X2C | | N | 00... | E,F:4m... | Encoder Zero of X2-axis |
| XALM | | N | 00... | E,F:4m... | Axis X Servo Alarm |
| YALM | | N | 00... | E,F:4m... | Axis Y Servo Alarm |
| AALM | | N | 00... | E,F:4m... | Axis A Servo Alarm |
| X2ALM | | N | 00... | E,F:4m... | Axis X2 Servo Alarm |
| X00 | | N | 00... | E,F:4m... | Reference Point of X-axis |
| X01 | | N | 00... | E,F:4m... | Positive Limit of X-axis |

Fig. 3-3 IO Ports of Hardware

The name, polarity, PLC address and functional description of each IO port are shown in Fig. 3-3. The polarity of an input or output port should be set according to the switch type, to put it in other words, for a normally close switch (NC switch), the polarity is P; while for a normally open switch (NO switch), the polarity is N. Wrong polarity setting should be modified timely, otherwise, alarms or other fault prompt may appear.

**To invert the polarity of a port,** firstly select the target port, click button F5"ConvtPol", and enter manufacturer's password. The modification of port polarity takes effect after the software being re-started.

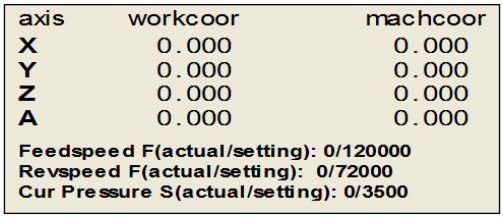
**To set the an input port,** in **IOPort** window, click button **PortAttr** to open a dialog box named "Inport Sampling Setting", where sampling interval, filter function, port enabled, port name and description can be set.

# 3.4 Setting Speed and Eccenricity

## 3.4.1 Setting Speed Parameters

After debugging, you can set the speed parameters.

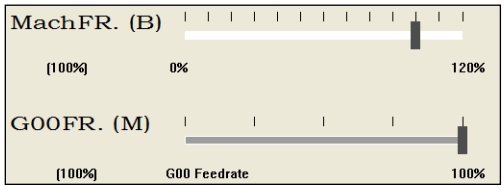
- If you only need to set the general parameters, you can refer to section 4.1.3 for setting the general speed parameters in auto mode and manual mode. When the machine tool works, the speed parameters will show on the operation panel. The screenshot on the right shows the values of speed parameters in auto mode.




**Feedspeed F** shows the actual/setting value of machining and positioning speed of linear axes.

**Revspeed F** shows the actual/setting value of machining and positioning speed of rotating axes.

- You can refer to the following tables to set the whole speed parameters.




During auto machining, speed cannot be adjusted through setting parameters. You can adjust the current multiple ratio of feed speed to control feed speed, as shown on the right.

The Formula is:

The actual speed value = the current feed speed multiple ratio * the setting value of feed speed

The multiple ratio of feed speed is the ratio of machining speed B and the maximum speed M. The current multiple ratio of feed speed shows in "MachFR. (B)" and "G00FR. (M). The value of each scale is 10%. And you can change the scale value by setting the value of manufacture parameter N53008, N53009 and N53010.

How to adjust the multiple ratio of feed speed: click on the multiple ratio scale; drag the sliding block on the scale; select the sliding block and press key PgDn or PgUp.

☞ **Parameters Related to Speed**

| Parameter | | Setting Range | Default |
|---|---|---|---|
| N64000 | The Initial Velocity to Start a Movement | 0~360000 | 0 mm/min |
| N64020 | The Default Speed When Positioning | 0.001~360000 | 20000 mm/min |
| N64040 | The Default Feeding Speed when Machining | 0.001~360000 | 15000 mm/min |
| N64060 | Circumgyrate Axes Permissible Allowable | 0.001~99999 | 8000 rpm |

There is another parameter “The Initial Frequency to Start Movement” whose value is set as 0 in Divers, related to parameter N64000 “The Initial Velocity to Start a Movement”. The Initial Frequency to Start a Movement refers to the maximum frequency to drive the motor to work without accelerating it. Setting an appropriate initial frequency can improve the machining efficiency and avoid the low speed period in which the kinetic characteristic of the motor is not good. The factory parameters of motors generally include the initial frequency to start movement parameter. However, the value of the parameter will change when the machine tool begin to do load movement after it is assembled. Therefore, the parameter should be determined according to actual measured values of the power of the motor and the inertia of the machine tool.

How to determine the value of the parameter N64000: set a small value at first. Then make the machine tool do typical operations and multi-axis movement repeatedly, and increase the value gradually to the maximum value to start movement. The setting value of this parameter is 50% that of the maximum speed to start a movement, generally set as 300 to 400.

☞ **Parameters Related to Acceleration**

| Parameter | | Setting Range | Default |
|---|---|---|---|
| N64080 | The Circumgyrate Axes Angle Acceleration | 0.001~100000 | 4000 deg/s2 |
| N64100 | The Maximum Linear Acceleration of Each Axis When Machining | 0.001~100000 | 3000 mm/ s2 |
| N64101 | The Maximum Linear Acceleration of Each Axis When Positioning | 0.001~100000 | 2000 mm/ s2 |
| N64120 | The Maximum Resultant Feed Acceleration of Adjacent Two Axes | 0.001~100000 | 15000 mm/ s2 |
| N64150 | The Acceleration of the Linear Acceleration Under S-Type Velocity Curve | 0.001~1e+009 | 500 mm/s3 |

[N64100] The maximum linear acceleration of each axis describes the acceleration and deceleration ability of each feeding axis. This index depends on the machine tool’s physical characteristics, such as the mass of the moving parts, the torque of the motor, the resistance, the load in cutting. The value of the parameter is the greater; the time spent in accelerating or decelerating when machining is the less and the efficiency is the higher. Generally, for a servo motor system, the value of the parameter can be 3000. When setting the value, set it small at the beginning. And make the machine tool do all kinds of typical movements and then gradually increase the value, watching carefully whether there are any exceptional conditions. If any,

| Parameter | Setting Range | Default |
|---|---|---|
| decrease the value and leave 50%~100% safety margins.<br><br>[N64120] The resultant feed acceleration of adjacent two axes describes the acceleration and deceleration ability of multi-axes motion. It determines the highest speed of the machine tool in circular motion. The greater the value is, the higher the speed of the machine tool in circular motion will be. Generally, for a servo motor system, the value can be set as 15000. However, if it is a heavy duty machine, the value should be set smaller. When setting the value, set it small at the beginning. And make the machine tool do all kinds of typical movements and then gradually increase the value, watching carefully whether there are any exceptional conditions. If any, decrease the value and leave 50%~100% safety margins. When setting the value, set it small at the beginning. And make the machine tool do all kinds of typical movements and then gradually increase the value, watching carefully whether there are any exceptional conditions. If any, decrease the value and leave 50%~100% safety margins.<br><br>[N64150] The acceleration of the linear acceleration refers to the growth rate of the acceleration, i.e. the increase volume of the acceleration in unit time. The parameter is valid in S-type acceleration or deceleration to alleviate the negative effect caused by sudden acceleration or deceleration. The value of the parameter is suggested to be set as 5000. | | |

☞ **Parameters Related to Others**

| Parameter | | Setting Range | Default |
|---|---|---|---|
| N64203 | To Select an Interpolation Algorithm | 0: triangle;<br>1: S_Type;<br>2: Trapezoid | 1 |
| N64207 | Whether to Limit the Arc-motion Velocity | Yes; No | Yes |
| N64208 | The Maximal Velocity of Circle Motion | 0.001~360000 | 5000 mm/min |
| N64209 | The Minimal Velocity of Circle Motions | 0.001~360000 | 600 mm/min |
| [N64203], the parameter is used to select an interpolation algorithm. The system now supports trapezoid, S shape and acceleration trapezoid algorithms. Trapezoid and S shape algorithms have higher machining efficiency. In the acceleration trapezoid algorithm, the acceleration curve is a trapezoid. The relationship between acceleration and time is: accelerate the acceleration to the maximum acceleration and keep it constant. Then decelerate it to zero in the same way it is accelerated. In the acceleration trapezoid algorithm, parameter [N64203] is set in range 100000~2000009 (mm/s^3). In this algorithm, the flexibility of acceleration and deceleration is good.<br><br>[N64208] The maximal velocity of circle motion: when machining an arc, the machine tool in circular motion will produce a centripetal force, which leads to the vibration of the machine tool. To alleviate the vibration, a limit of the machining velocity is set in the software based on the centripetal | | | |

| Parameter | Setting Range | Default |
|---|---|---|
| acceleration. Take the default as an example. The diameter of the reference circle is 10mm and the maximal linear velocity is 5000mm/min.<br>According to the centripetal acceleration calculation formula:<br>$$a = \frac{v^2}{r}$$<br>Herein, $r = (10/2)mm$; $v = 5000mm/\min$<br>The centripetal acceleration $a$ can be calculated through the formula above. The centripetal acceleration can be applied to machining of other arcs as the maximal centripetal acceleration. If the linear velocity of other arcs is too high, the actual centripetal acceleration will be greater than that calculated out through the above formula. In this case, you should set a limit to the linear velocity of the circular.<br>[N64209], the minimal velocity of circular motion: as for the velocity limit function, the centripetal acceleration $a$ calculated out according to the formula above is not the only criteria.<br>According to formula:<br>$$a = \frac{v^2}{r}$$<br>From the formula, we can know that if the arc is too small, the velocity of the arc will be very low according to formula. This will take too much time to machine the arc at such low velocity. Therefore, the minimal velocity is set. No matter how small the radius is, the machining speed will not be less than the minimal velocity of the circular motion.<br>Generally, considering the driving ability of the servo motor, the friction in machine assembly, and the endurance of machine parts, you can change the maximal velocity of each axis in manufacturer parameters and set a limit to the maximal velocity of the axis in use. | | |

**CAUTION**

Generallly, Maximum Resultant Feed Acceleration of Adjacent Two Axes, i.e. N64120 should be set as two to five times that of The Maximum Linear Acceleration of Each Axis When Machining i.e. N64100.

## 3.4.2 Setting Eccentricity Parameters

Since the cutting point of glass cutting is not the same as the control point. There is a deviation between the two. To solve the problem, you can set parameter Fore-and-aft Eccentricity of the two points, and the system will automatically compensate the deviation in tool path.

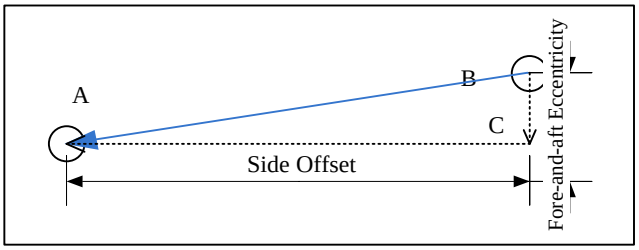



Fig. 3-4 Tools and Eccentricities

- The Fig. 3-4 above shows the characteristic points of tools. It is a top view.
- Control point: A, represents the handle fitted on the machine tool. The point is showed in coordinate system.
- Machine Point: B, the point is where the tool and the workpiece contacts with each other. It is also called cutting point.
- When the value of A is 0, draw a directed line segment from machining point B to control point A. The offset of machining point A and control point B in X axis is “Side Offset”. When the line segment directed in positive direction of X axis, the value of side offset is positive, otherwise it is negative. The offset of machining point A and control point B in Y axis is “Fore-and-aft Eccentricity”. When the line segment directed in positive direction of Y axis, the value of side offset is positive, otherwise it is negative.
- **All in all, side offset is tool’s offset in X axis, and fore-and-aft eccentricity is tool’s offset in y axis.** According to the above definition, we can know that the value of side offset of B to A is negative and the value of fore-and-aft eccentricity of B to A is also negative in Fig. 3-4.

◆ **Measurement of Eccentricity**

For measurement of eccentricity, a bullseye cutting diagram can be used. As shown below, the red track is the tool path, i.e. cutting point track. And the blue track is the control point track. The orange dotted arrow is the directed line segment from the cutting point to the control point. The blue track is only a virtual track and will not show in the machining track.

In the left part of the circle (i.e. when the value of A is 0), draw a directed line segment (the dotted arrow) from the track of the cutting point to the track of the control point. If the direction of the line segment is the positive direction of X axis, the value of the offset is positive, otherwise it is negative. If the direction of the line segment is the direction of Y axis, the value of the offset is positive, otherwise it is negative. In theory, eccentricity is classified into 8 types. The following diagrams will show the different eccentricities.

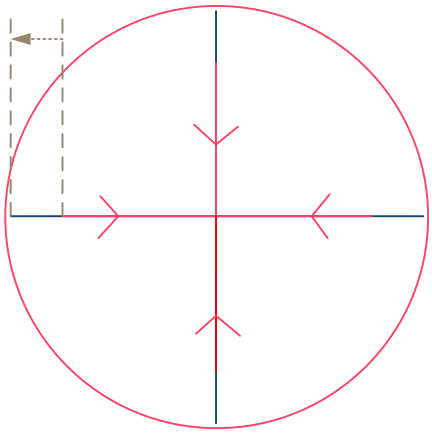


Fig. (1)

As shown in Fig. (1), the side offset exists and its value is negative, while the fore-and-aft eccentricity does not exist.

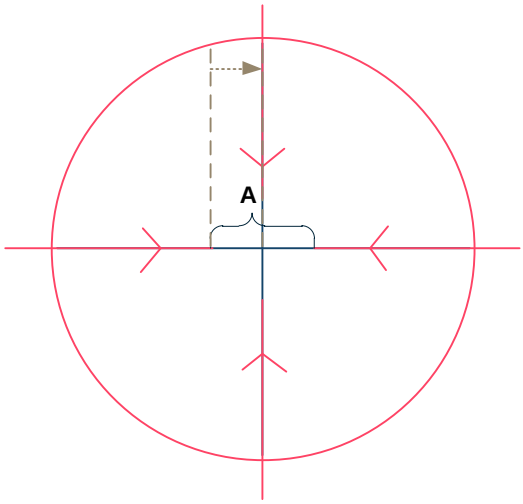



Fig. (2)

As shown in Fig. (2), the side offset exists. Its value is positive and equals ½ the length of line segment A. The fore-and-aft eccentricity does not exist.

Fig. 3-5 Eccentricity Setting Method-1

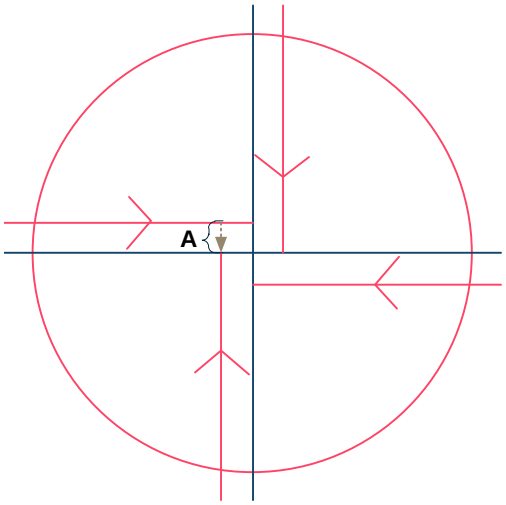



Fig. (3)

As shown in Fig. (3), the fore-and-aft eccentricity exists, and its value is negative and equals the length of line segment A. The side offset does not exist.

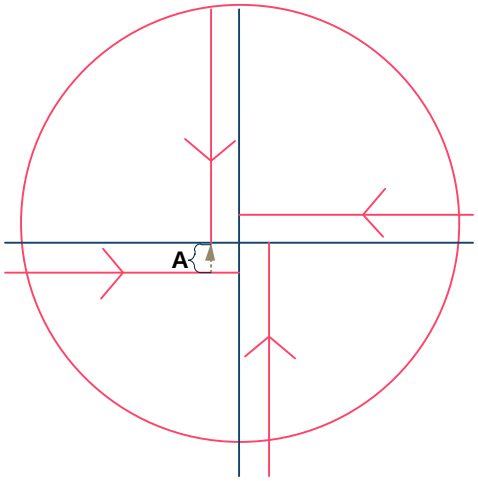



Fig.(4)

As shown in Fig. (4), the fore-and-aft eccentricity exists, and its value is positive and equals the length of line segment A. The side offset does not exist.

Fig. 3-6 Eccentricity Setting Method-2

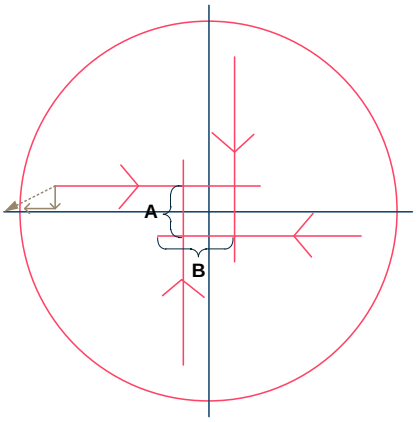



Fig. (5)

As shown in Fig. (5), both the side offset and fore-and-aft eccentricity exist, and their values are both negative. The value of fore-and-aft eccentricity equals the length of line segment A. The value of the side offset equals the length of line segment B minus the length of line segment A.

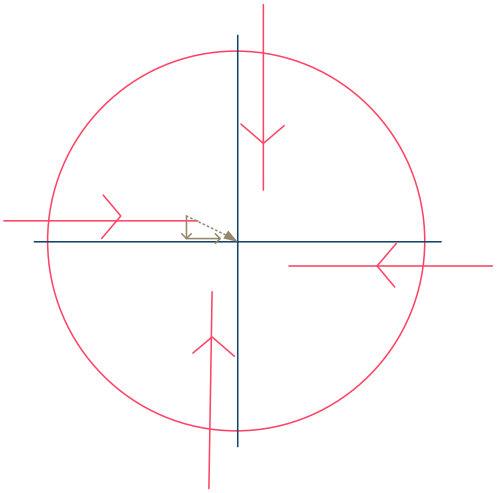


Fig. (6)

As shown in Fig. (6), both the side offset and fore-and-aft eccentricity exist. The value of side offset is positive and the value of fore-and-alt eccentricity is negative.

Fig. 3-7 Eccentricity Setting Method -3

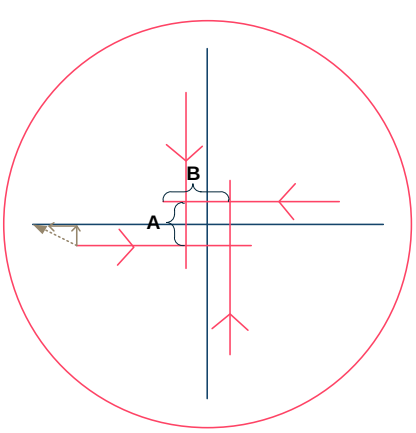



Fig. (7)

As shown in Fig. (7), both the side offset and fore-and-aft eccentricity exist. The value of side offset is negative while the value of fore-and-aft eccentricity is positive. The value of fore-and-aft eccentricity equals ½ the length of line segment A. The value of the side offset equals the length of line segment B minus the length of line A.

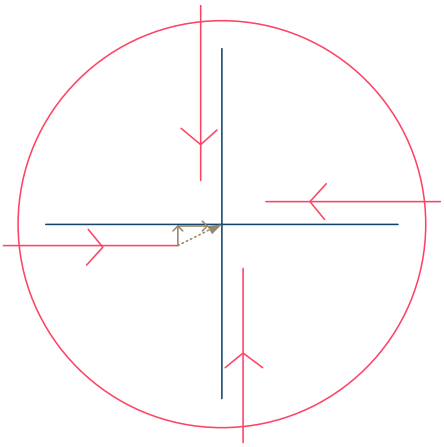


Fig. (8)

As shown in Fig. (8), both the side offset and fore-and-aft eccentricity exist, and their values are both positive.

Fig. 3-8 Eccentricity Setting Method -4

## ☞ Related Parameters (Manufacturer Parameters)

| Parameters | | Description | Default |
|---|---|---|---|
| N83010 | Side Offset | Specify side distance between the positions of cutter center and actual one | 0 mm |
| N83011 | Fore-and-aft Eccentricity | Specify the fore-and-aft eccentricity value relative to the tool control center. In the direction of machining, left side eccentricity has a positive value, otherwise negative value. | 0 mm |

# 3.5 Loading Programs or Drawing Graphs

After setting the parameters, you can load the machining programs. 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.

☞ **Related Parameters**

| Parameter | | Definition |
|---|---|---|
| N65000 | Whether to convert to NCE file when opening files | 0: Not Convert;<br>1: Prompt;<br>2: Convert. |
| If you choose “2”, i.e. “Convert”, the name of non-NCE format file will not change when you load it into NcStudio. But in the software, the file is counted as a NCE file. And if you choose “0”, i.e. “Convert”, the name of non-NCE format file will not change when you load it into NcStudio. It will be parsed in the software. | | |

**CAUTION**

1) The difference in loading files in NcStudio and NcEditor: The file loaded in NcEditor will be saved as “Non-titled.nce” in NCE format automatically; The file loaded in NcStudio can be convert into NCE format file by setting the parameter N65000. Please see the above decription on how to convert in NcStudio.

2) 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.

3) 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 axis in machining programs. Before machining starts, you need to set workpiece origin. Steps to set workpiece origin are shown as Fig. 3-9.

Manually move axis X/Y/Z/A to the position you need to set as worpiece orgin

↓

- In manual mode, click tab **POS**, and press F1/F2/F3/F4/F5 or click operation button **XClear**, **YClear**, **ZClear**, **XYClear**, and **XYZClear**;
- Select **Set Worpiece Origin** under menu **Operate**;
- Select **Set Current Point's Workpiece Coordinate** under menu **Window to open** WorkCoor window. Then press F1/F2/F3/F5/F6 or click operation button **XClear**, **YClear**, **ZClear**, **XYClear**, and **XYZClear**;
- Select Offset under menu **Window** to open Offset window. Then press F1/F2/F3/F5/F6 or click operation button **Set_X**, **Set_Y**, **Set_Z**, **Set_XY**, and **Set_XYZ**;

↓

Through the four methods above, you can reset the coordinate of current point, then the current point will be regard as workpiece origin when machining programs is running.

Fig. 3-9 Set Workpiece Origin

In addition, you can set workpiece origin in NcEditor. Below are two methods of setting origin.

1) Click Set Origin under menu Draw. Dialog box "Set Origin" as shown in Fig. 3-10 will pop out. You can set origin by particular positions or set the X/Y coordinate of the origin directly.

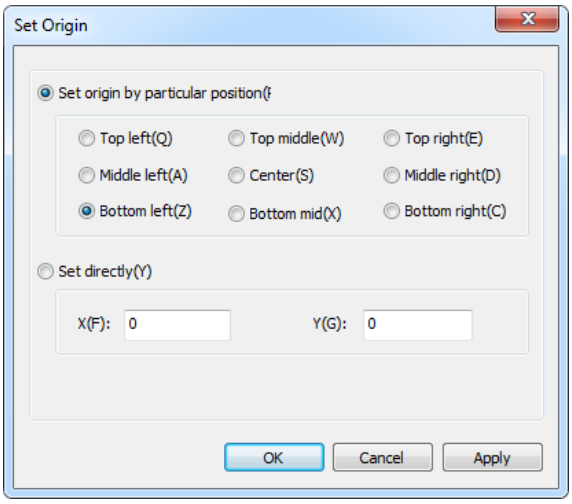



Fig. 3-10 Set Origin

2) The origin for object editing space is also the workpiece origin. There is an icon ⊞ at the intersection point of horizontal scale and vertical scale. Press the center of the icon and drag it to the point you need to set as workpiece origin.

# 3.7 Analogue and Simulation

After setting the workpiece origin, you can go on to execute analogue or simulation. By executing analogue or simulation, you can learn the forms of machine tool motion in advance to prevent the damage of machine tools caused by programming errors. You can also learn other additional information.

◆ **Analogue**

Select sub-menu item **Simulation Mode** under **Operation** menu to enter simulation mode, and click item **Start**. 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.

◆ **Simulation**

In auto mode, click **Simulate** button and then **Start** button on the operation menu bar. The system will process machining in tool path at the default feed speed without tools above the workbench, to verify whether the programmed tool path accords with the expected path.

In simulation process, the air cylinder does not descend, so the glass will not be cut. There is also no output in ports. The machining time in simulation is the same as that in normal machining.

☞ **Related Parameters**

| Parameter | | Range |
|---|---|---|
| N86006 | Whether the simulation buttons can be controlled during machining | 1: the simulation buttons can be controlled during machining;<br>0: the simulation buttons cannot be controlled during machining. |
| The default value of parameter N86006 is “1”, which means the simulation buttons can be controlled during machining. You can press “F7” on the keyboard or click on the operation button to run simulation during machining. If you set the value of the parameter as “0”, the relevant operation button is disabled and simulation cannot be run during maching. | | |

◆ **The Difference Between Analogue and Simulation**

In analogue process, the system only shows the machining path of tools in the tracking display window and does not drive the machine tool to take any machining actions. However, in simulation process, the system will drive the tool to simulate tool path on the XY two-dimensional surface in a safe height without opening the air cylinder and driving Z axis.

# 3.8 Execute Machining

**Start Machining**

Machining can be started by the following three ways:

1) Select **Start** item under **Operate** menu.
2) In auto mode, click button **Start** in the operation button bar.
3) In auto mode, press the 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 button **Pause** in the operation button bar.
3) In auto mode, press the 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** in the operation button bar.
3) In auto mode, press the shortcut key F11.

# 4 Software Functions

## 4.1 General Configuration

The functions introduced in this section are the advantages of general Lambda Series software. Unless specifically indicated, most of the screenshots illustrated in this section show the software under XYZA configuration.

### 4.1.1 Configuration Management

Five different configurations are supported in general Lambda series glass cutting software, i.e. XYA configuration, XYZA configuration, XYA No Edge Configuration, XYZA No Edge Configuration, X1YAX2 configuration.

Select the submenu **Change Config** under menu **Machine Tool**. Dialog box “Config manage” will pop out, shown as Fig. 4-1. You can change the configuration in the drop-down list.

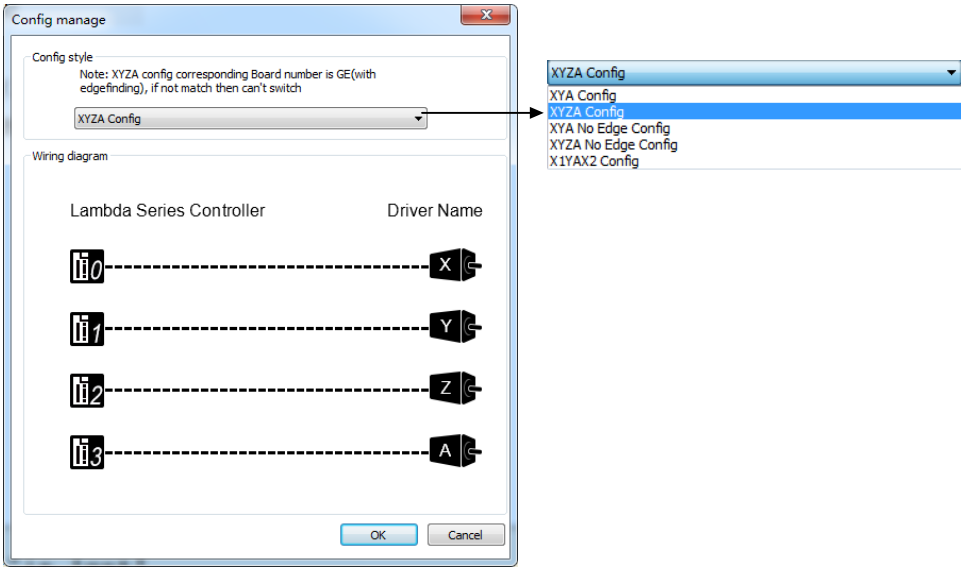



Fig. 4-1 Config Manage Dialog Box

**CAUTION**

Only manufacturers can set the configurations. Uers must not change the configurations.

## 4.1.2 Back to Machine Origin

Back to machine origin operation must be executed before the machine tool starts machining and after emergency stops. The operation button bar in reference point mode is shown in Fig. 4-2. You can make the machine tool back to machine origin with these buttons. Button **REF_DETEC**T is disabled in XYZA configuration. However, in dual-axis configuration, generally the button **REF_DETECT** is enabled and the operation must be executed before backing to machine origin.


Fig. 4-2 The Operation Button Bar in Reference Point Mode

◆ **AllBack**

Press key “F9”, or click on button **AllBack**. And the machine tool will execute backing to machine origins in Z axis, X/Y axis and A axis in order as the system default setting. If Z axis is not included in configuration, the machine tool will execute back to machine origins in X/Y axis and A axis in order.

◆ **Uniaxial Back**

Press key F1/F1/F3/F4, and the machine tool will execute back to machine origin in X/Y/Z/A axes accordingly.

◆ **DirectSet**

Press key “F5” or click button **DirectSet** to set the current state as back to machine origin state.

Function “DirectSet” are generally used in debugging which has low requirements for accuracy. For example, when modifying parameters or public subprograms, the software is repeatedly restarted. In this case, you can use “DirectSet” to achieve software limit protection and save lots of time.

However, to ensure machining accuracy, the suggested operation is to execute back to machine origin operation instead of using the “DirectSet” function each time the software is restarted.

◆ **REF_DETECT**

Before back to machine origin, you should execute **REF_DETEC**T operation to correct the distance error between origins in two axes. Dual-axis control system includes double X axis control system and double Y axis control system. The following operations and diagrams are based on double X axis system.

Assemble an encoder to each X axis of the dual-axis machine tool. The connecting line between the two encoders should be parallel to Y axis, so that the connecting line between origins on the two X axes parallels to Y axis after backing to machine origin operation. Yet the connecting line between the encoders may not be parallel to Y axis, due to inherent errors and assembly mistakes in assembly.

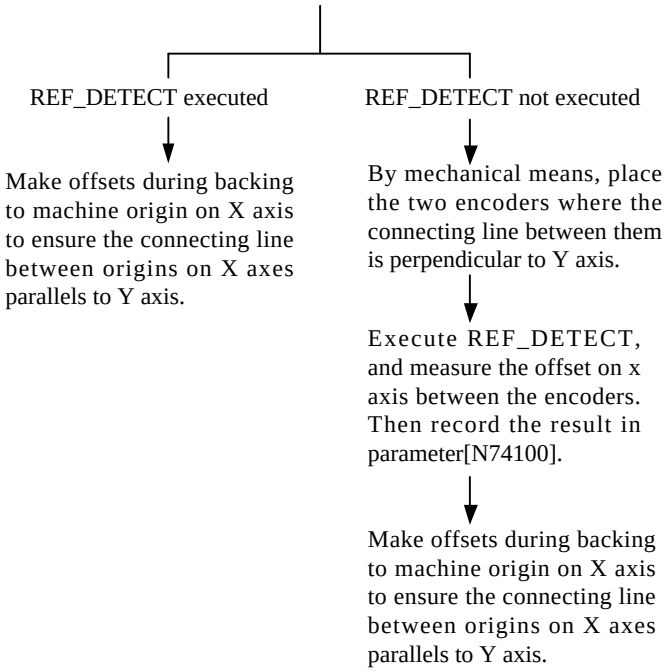



Fig. 4-3 The Principle of X Axis Origin Detection

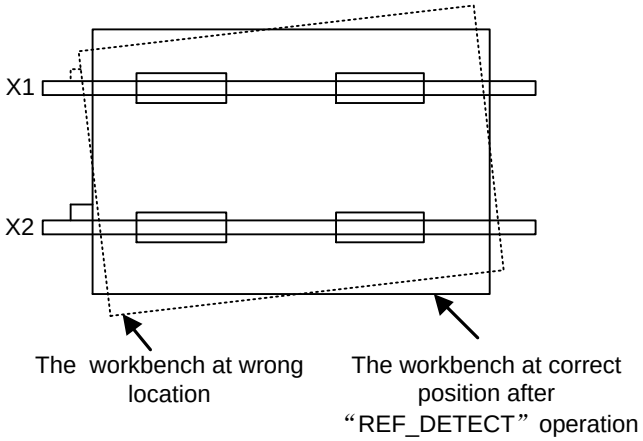



Fig. 4-4 The Diagram of X Axis Origin Detection

◆ **MakeSetup**

Click on button **MakeSetup**, and the software will close automatically and begin making setup package.

◆ **Reset**

Click on button **Reset** to stop the current operation in case of emergency happened during backing to machine origin.

☞ **Parameters Related to Operation Safety**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N74000 | CANCEL_REFPT_AFTER_ESTOP | Backing to the reference point will be cancelled, when an E-stop occurs. | Yes: cancel;<br>No: do not cancel |
| N74001 | NEED_REFPT_BEFORE_MACHINING | Whether to back to the machine origin before machining. | Yes: must back;<br>No: do not need back |
| N74006 | MACHINE_LOCK | Whether the machine needs to be unlocked to normal use. | Yes: need;<br>No: do not need |
| N74007 | SHOW_BKREF_DLG_TIP | Whether show tips on backing to machine origin when the software starts | Yes: show tips;<br>No: hide tips. |
| Returning to machine origin before machining can avoid machining offset to ensure position precision. It is suggested that both parameter 74000 and 74001 are set as “YES”. Once e-stop operation or reset operation is executed during machining, the current machine origin mark will be cleared, and the system will remind you to execute operation of returning to machine origin. When N74001 is set to “YES”, if there is no backing to machine origin mark “◔” before each axis, the machine is not allowed to move until returning to REF. point is completed. | | | |

☞ **Parameters Related to Backing to Machine Origin**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N74010 | REFPT_COOR | The machine coordinate of the machine orogin, default value is 0. | 0~the upper travel limit of the workbench |
| N74020 | COARSE LOCATING DIR | The moving direction in rough positioning phase when backing to the machine origin. | 1: positive direction;<br>-1: negative direction |
| N74030 | COARSE_LOCATING_VELO | The feeding speed in rough positioning phase when backing to the machine origin. | 0.001~10000 mm/min |
| N74050 | FINE_LOCATING_DIR | The moving direction in accurate positioning phase when backing to the machine origin. | 1: positive direction;<br>-1: negative direction |
| N74060 | FINE_LOCATING_VELO | The feeding speed in accurate positioning phase when backing to the machine origin. | 0.001~10000 mm/min |
| N74080 | BACK_DISTANCE | The additional displacement after accurate locating when backing to the machine origin. Plus value: Positive direction, Minus value: Negative direction. | -1000~1000 mm or deg. |

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N74090 | LocationTimes | The times of accurate positioning when backing to the machine origin. | 1~100000 |
| N74100 | Double Axes Encoder Origin Error | Double Axes Encoder Origin Error. Default is 0. | -10000~10000 |
| N74105 | BKREF_MODE | The mode of backing to machine origins in each axis. Servo mode is applied for X/Y/Z axes, and step mode is applied for A axis. | 1: servo mode;<br>2: step mode |
| In order to establish a correct machine coordinate system (MCS) for machining, at machine start-up, returning to reference point will be executed automatically or manually, i.e. the machine tool will return to its measuring beginning (X, Y, Z=0) to establish machine coordinate system. Reference point can be coincident with machine origin (in default system setting), or not, and the distance between reference point and machine origin can be specified by parameter N74010.<br>When home switches work normally, if the spindle moves away from home switch direction in the process of returning to machine origin (homing), the value of N74020 “COARSE_LOCATING_DIR”, opposite to that in fine positioning stage, should be modified. If the speed of returning to machine origin is too low, properly adjust the value of N74030 “COARSE_LOCATING_VELO”, whose default is 5000mm/min.<br>“Back Off Distance” refers to a certain moving distance away from REF. point to leave the signal sensitive zone of home switches after backing to machine origin completed. | | | |

## 4.1.3 Set Common Parameters

You can set the frequently used parameters as common parameters for convenience.
Select **Param** under tab **PARAM**, and click button **All Param**. A dialog box shown in Fig. 4-5 will pop out.

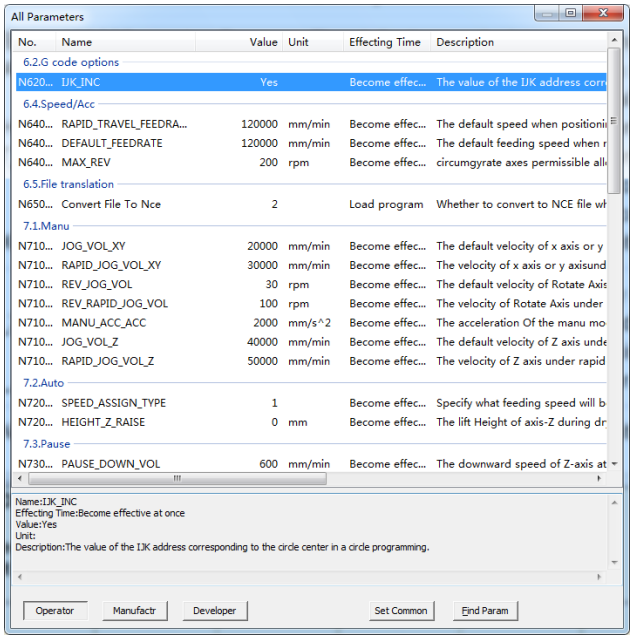



Fig. 4-5 Dialog box “All Parameters”

Select a parameter, then click button **Set Common**. And the parameter will be added in the list under sub tab **Param**.

If you do not need any parameter in the common parameter list, open the **All Parameters** dialog box, select that parameter and click on button **Unset Common,** close the box. The parameter has been deleted from common parameter list.

Under general configuration, the default common parameter list is fixed.

In addition, you can set common parameters in auto mode and manual mode.

◆ **Set Common Parameters in Auto Mode**

In auto mode, select sub tab **Setting** under tab **POS**. The interface of is as shown in Fig. 4-6.

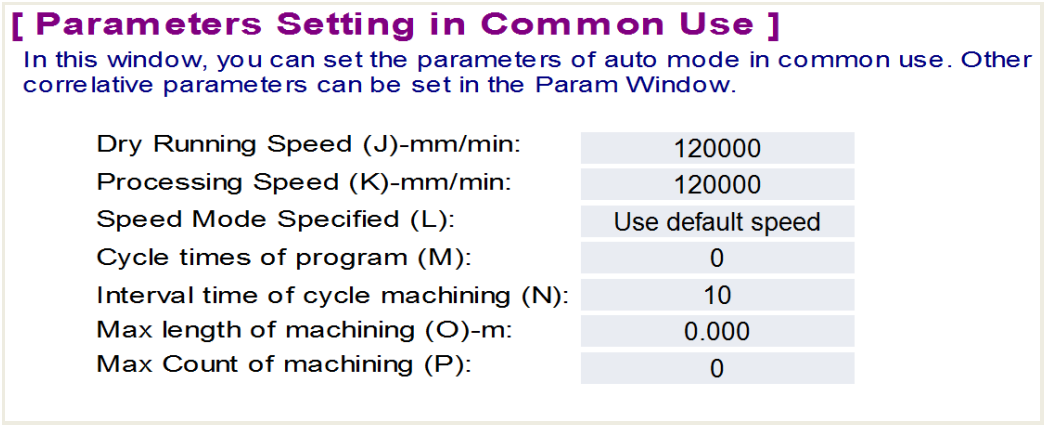



Fig. 4-6 Parameter Setting in Common Use in Auto Mode

- **Dry Running Speed**

G00 speed refers to the running speed when the machine tool positions. The corresponding parameter is N64020 “RAPID_TRAVEL_FEEDRATE”.

- **Processing Speed**

The feed rate refers to the running speed when the machine tool processes. The corresponding parameter is N64040 “EFAULT_FEEDRATE”. Processing speed must be less than dry running speed.

- **Speed Mode Specified**

The corresponding parameter is N72001” SPEED_ASSIGN_TYPE”. If speed mode is specified as **Use Default Speed**, the machine tool will process at **Processing Speed** even if feed rate is specified in the machining file loaded. However, if speed mode is specified as S**P specified by file** and F command exists in the file loaded, the machine tool will process at the speed specified in F command.

◆ **Set Common Parameters in Manual Mode**

In manual mode, select sub tab **Setting** under tab **POS**. The interface of is as shown in Fig. 4-7.

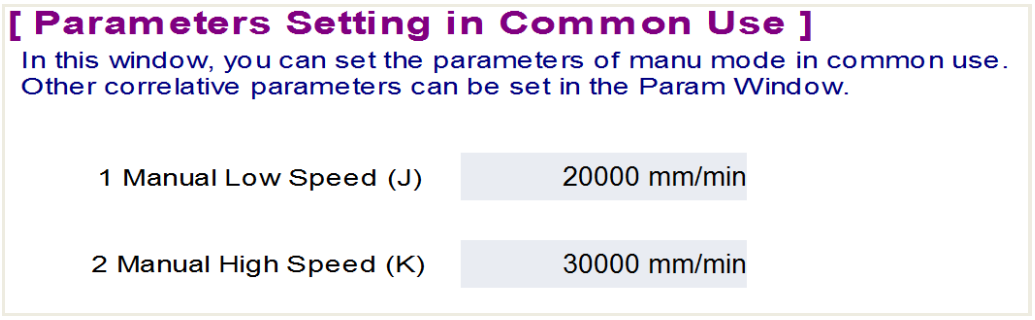



Fig. 4-7 Parameter Setting in Common Use in Manual Mode

- **Manual Low Speed**

The corresponding parameter for manual low speed is N71000 “JOG_VOL_XY”. Manual low speed refers to the running speed of the machine tool in manual mode when only one axis direction key is pressed down. The setting range for manual low speed is 0.001mm/min~ manual high speed.

- **Manual High Speed**

The corresponding parameter for manual high speed is N71001 “RAPID_JOG_VOL_XY”. Manual high speed refers to the running speed of the machine tool in manual mode when one axis direction key and key ~ are pressed down at the same time. The setting range for manual high speed is manual low speed ~360000mm/min.

## 4.1.4 Pressure Library

The software provides pressure library function. With this function, you can set different pressure values for different graphs to improve machining yield.

After loading tool path file, select sub tab **Pressure Lib** under tab **PARAM**, as shown in Fig. 4-8. Add a height in the interface **Pressure Lib** for the current material, and set the pressure value required for glass cutting. In this interface, you can add, modify, delete and use height, save parameters and load parameters.

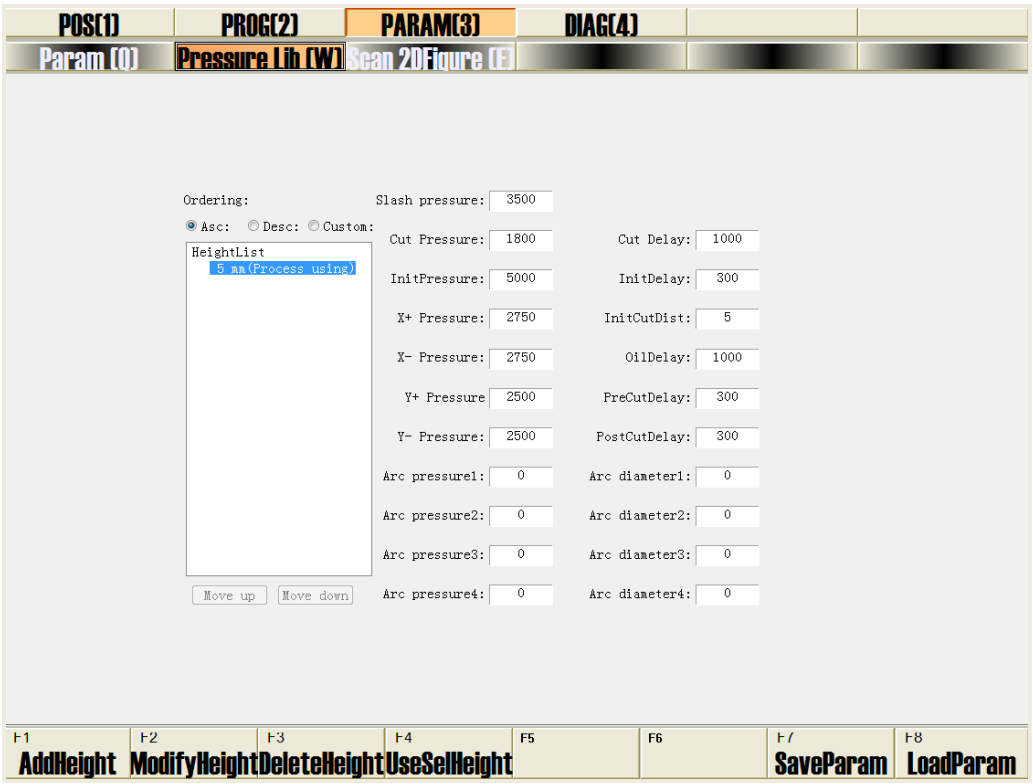



Fig. 4-8 Interface Pressure Lib

Click button **AddHeight**, and enter a proto glass height in range 1~ 100000mm in the pop-out input box.

Select a height in the **Heightlist**, and click button **ModifyHeight**. Then you can modify the height in the input box popping out.

Select a height in the **Heightlist**, and click button **UseSelHeight**. “(Process Using)” will appear behind the height, which indicates that the parameter is being used and cannot be modified or deleted.

Click button **SaveParam** to save the current height list and the corresponding paramters as xml format file. And you can load the xml file the next time you need it by simply clicking **LoadParam**.

Select **Asc**, **Dsc** or **Custom** to set the ordering of heights in the heighlist. When you select **Custom**, you need to click on button **move up** or **move down** to change the ordering of the height you select.

The background for the current selected height is dark blue. In actual machining, the values for parameters such as cut pressure or cut delay must be entered. Different pressures can be set for different heights. The parameters in the right of heightlist are shown as below.

| | |
|---|---|
| Cut Pressure | The cylinder pressure when the tool is at starting point. |
| Cut Delay | The cylinder delay time when the tool is at starting point. |
| InitPressure | The cutting pressure of the cylinder when the tool moves in the initial distance. |
| InitDelay | The cutting delay time of the cylinder when the tool moves in the initial distance. |
| InitCutDist | The cutting distance under InitPressure. |
| OilDelay | The delay time for oiling. |

| | |
|---|---|
| X+/- Pressure | The linear tool pressure in positive/negative direction of X axis. |
| Y+/- Pressure | The linear tool pressure in positive/negative direction of Y axis. |
| Arc Diameter | The cutting arc diameter |
| Arc Pressure | The pressure required for arc cutting. |
| PreCutDelay | The delay time before cutting. |
| PostCutDelay | The delay time after cutting. |
| Slash Pressure | The default pressure of cylinder in slash cutting or ellipse cutting. |

☞ **Related Parameters(Manufacturer Parameters)**

| Parameter | | Description | Setting Range | Default |
|---|---|---|---|---|
| N83000 | Max pressure | Specify the max pressure | 0~100000 | 5000 HP |
| N83100 | CUSTOMIZED_PRESSURE _AVAILABLE | Whether customized pressure is effective. | Yes: effective; No: ineffective | Yes |
| If the value of parameter N83100 is set as “Yes”, X+/= pressure, Y+/- pressure and arc pressure will be applied in line machining and circle machining, and slash pressure will be applied in slash machining and ellipse machining. If the parameter is set as “No”, slash pressure will be applied for machining of all graphics, including line, circle, slash and ellipse.<br>The parameter N83100 is set as default common parameter, so you can modify it directly in the common parameter list under sub tab **Param**. | | | | |

**CAUTION**

1) After entering the values for the paramters in the right of heightlist, press Enter to make the paramter setting effective immediately. Before the values become efffective, you can press Esc to restore the original data.
2) A yellow prompt info “Glass Cut Param Updated!” will appear in the CNC info bar if you have modified the paramters for process using height. A normal prompt info “Pressure Lib Param Uodated!”will appear if you have modified the paramters for non-using height.
3) When you add a new height, the paramters for the new height will be the same as those of the process using height.
4) When the software is running, the **Pressure Lib** interface is diabled.
5) Double click **Heightlist**, and the heights in the list will be folded or unfolded. When you select **Heightlist**, and the process using height will be selected at the same time.

## 4.1.5 Scan 2D Figure

The system provides function “Scan 2D Figure”. Scan 2D figure refers to scanning the glass contour with a laser head and sketch a model contour and generate a DXF format tool path file, which can be loaded into AutoCAD for more precise modification. The function can improve the machining efficiency and save

manually drawing cost for those who do not need high precision machining. Select **Scan 2DFigure** under tab **PARAM**, and the interface is as shown in Fig. 4-9


Fig. 4-9 Interface for Scan 2D Figure

**CAUTION**

The system does not support scanning of group of several workpieces or workpiece with holes.

Scan 2DFigure function can be further divided into normal Scan 2DFigure and advanced Scan 2DFigure. Click button **Advance**, the parameters related to advanced scan 2DFigure function will be shown in the interface, as shown in Fig. 4-10. The parameters are mainly used to handle with exceptional cases, and onsite machining and debugging.

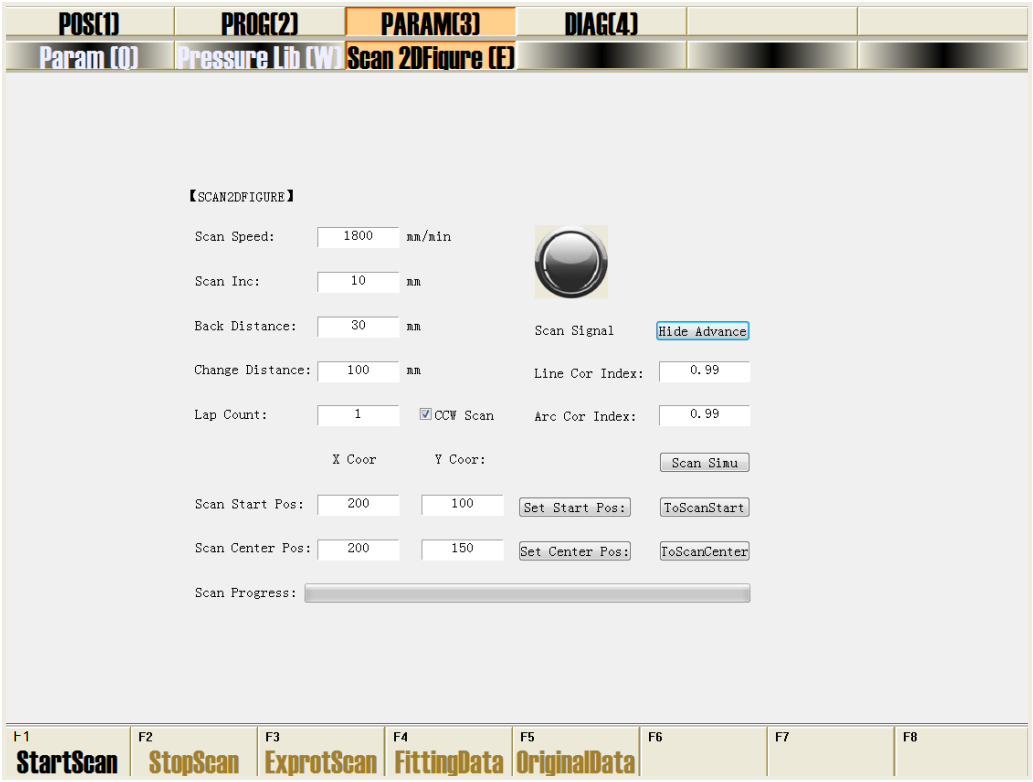



Fig. 4-10 Interface for Advanced Scan 2DFigure

◆ **Parameters for Scan 2DFigure function are as below.**

| | |
|---|---|
| Scan Speed | The moving speed of the optoelectronic switch in scanning process. It is set according to the scanning accuracy of the photoelectric sensor. |
| Scan Inc | The stepping length of scanning contour in scanning process. It is set according to the roughness of workpiece edge. |
| Back Distance | The back distance out of the contour in scanning process. Suggested value for the parameter is 3~5 times the stepping length. The back distance will be modified automatically into 3 times that of stepping length when stepping length is modified. |
| Change Distance | If the laser head does not receive any signal after moving a change distance, it will change direction automatically. Suggested value for the parameter is 3~5 times the stepping length. The change distance will be modified automatically into 10 times that of stepping length when stepping length is modified. |
| Lap Count | You can choose to scan the contour several laps or only one lap. However, for some special 2DFigures, one lap of scanning is not enough. You can modify the lap count to ensure the space between scan start position and scan end position is small enough. |
| CCW Scan | The default scanning direction is CCW. You can cancel the selection of CCW to change the scanning direction into clockwise. |
| Line Cor Index | To fit more points into a line, you can set the parameter as a not less than 0.95 value. To prevent points fitted into a line, you can set the parameter as greater values. |

| | |
|---|---|
| Arc Cor Index | To fit more points into an arc, you can set the parameter as a not less than 0.9 value. To prevent points fitted into an arc, you can set the parameter as greater values. |
| Scan Start Pos | The coordinate of any point in the contour and near the edge of the contour. You can directly enter a coordinate, or manually move the laser head into the position you are going to set as start position and click **Set Start Pos:** to set it as the start position. |
| Scan Center Pos | The coordinate of any point in the contour and near the center of the contour. You can directly enter a coordinate, or manually move the laser head into the position you are going to set as center position and click **Set Center Pos:** to set it as the center position. |

◆ **Basic Operations of Scan 2DFigure.**

1) Place the irregular workpiece on the machine tool, and set scan speed and scan inc according to the rough dimension of the workpiece. The position where the workpiece is placed must have enough space to avoid triggering limits during scanning.
2) Click button **ToScanCenter** to determine whether the current center position is appropriate. If the point is inappropriate, move the spindle of the machine tool until the laser head shines laser into the center of the workpiece, and click button **Set Center Pos**.
3) Click button **ToScanStart** to determine whether the current start position is appropriate. If the point is inappropriate, move the spindle of the machine tool until the laser head shines laser into the edge of the workpiece, and click button **Set Start Pos**.
4) Click button **StartScan**, the software start 2DFigure scan automatically.
5) Export the scan result.

There are three ways to export the scan result.

1) **ExportScan**: by clicking the button you can export a DXF file where all the points in scan result are connected with lines in order.
2) **FittingData**: by clicking the button you can export a DXF file where all the points in scan result are fitted into lines and arcs. The fitting result can be adjusted by modifying parameter **Line Cor Index** and **Arc Cor Index** in advanced scan 2DFigure interface.
3) **OriginalData**: by clicking the button you can export a DXF file which contains all the scanned points. You can modify the scan result graph with other drawing softaware.


1) Before scanning completes, you can click button **StopScan** or **Stop** to end the scanning process. After manually scan stop, the scan result is not be saved, and the scan progress returns to zero,and button **ExportScan**, **FittingData**, and **OriginalData** are disabled.
2) When scanning is completed normally, prompt info “Scan Complete” will show in the CNC info bar and the scan progress bar will be full, button **ExportScan**, **FittingData**, and **OriginalData** are enabled. If you start

another scan after scanning is completed, a dialog box “Reset Last Scan Data, Continue?” will pop out. You can select according to your needs.

## 4.1.6 Following Normal

Generally, following normal function should be applied to cutting devices. For example, in machining a circle, the width of the tool must be calculated. So the direction of cutting edge must continuously change with the circular track. Therefore, in machining a circle, the moving direction of tangent axis, A axis, must be the same as the direction of the tangent of the circle. This is called **Following Tangent** or **Following Normal**. The program which is running will automatically calculate the A axis according to the X and Y axis data in tool path. The green arrow represents the direction of A axis, as shown in Fig. 4-11.

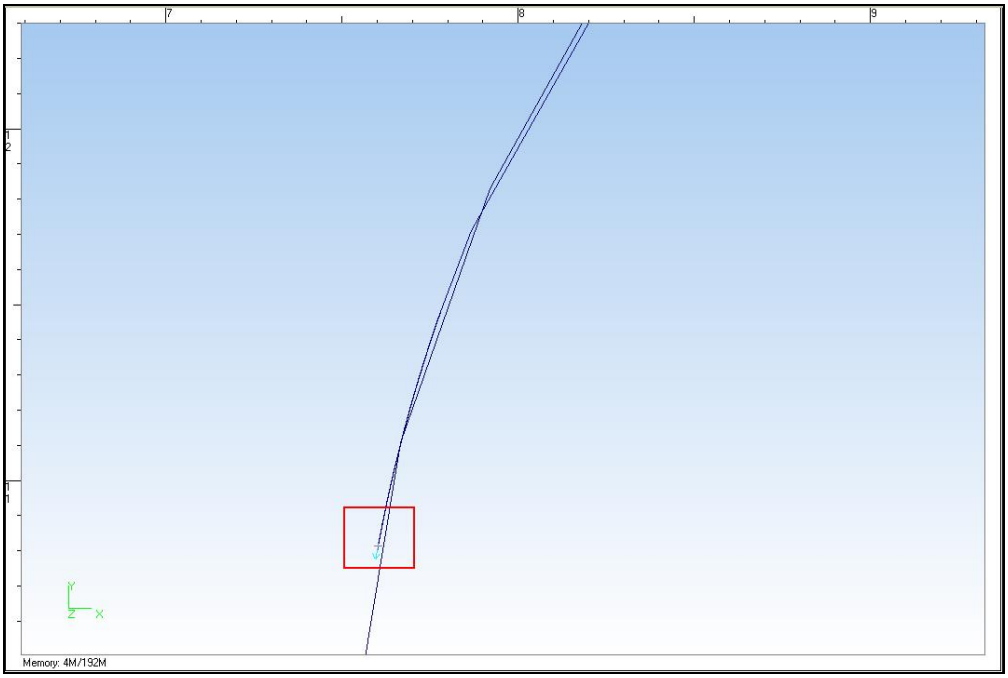



Fig. 4-11 Following Normal

**CAUTION**

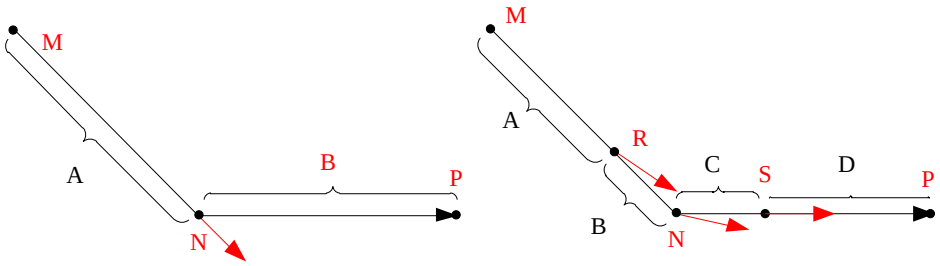
Since following normal function is only effective for .NCE fornmat file, therefore tool path file for glass cutting must be converted to NCE format file to enable the function. The data of A axis is caculated aotomatically with the X and Y axis data, and is not controled by the tool path file.

## 4.1.7 Tool Turning or Superposition Turning

Glass cutting is a rapid process. If tools lifted and turned for cutting all angles, machining efficiency will be influenced. Therefore, the system provides tool turning or superposition turning function for different situations.

If the rotation angle is less than the value of parameter N83013 “TURNING_ANGLE”, you need to superpose tool rotation to the tangential direction of the next part of the path. For example, if the value of

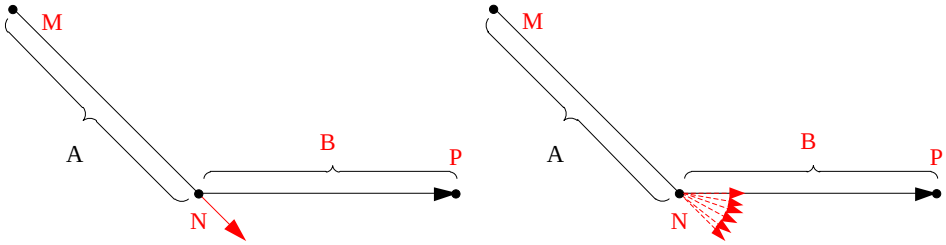
N83013 is 60°, and the rotation angle is 45°, you can use superposition turning function, as shown in Fig. 4-12.




The tool should be rotated at the intersection point(N) of two motion command. However, to save rotation time and avoid influencing the machining efficiency of the other two axes, the tool will be rotated at point R and which is at a distance from point N. Therefore a new command B is generated. The command B makes the direction of the machine tool is same as the angle bisector of the rotation angle. In this way, the machine tool begin rotates from point R and superpose rotation to the direction of NP until it reaches point S.

Fig. 4-12 Superposed Rotation

If the rotation angle is greater than the value of parameter N83013 “TURNING_ANGLE”, you can use turning function. Turning process: lift the tool at point N →rotate the tool to the tangential direction of the next part of the path →start cutting.




Fig(1) Fig(2)

Turning Process: lift and rotate the tool at point N to the direction the tangential direction of the next part of the path and start cutting. The dotted arrow in Fig(2) represents the lifting and turning process at point N. the final direction is NP direction.

Fig. 4-13 Safe Turning

☞ **Related Parameters(Manufacturer Parameter)**

| Parameter | | Description | Setting Range | Default |
|---|---|---|---|---|
| N83013 | TURNING_ANGLE | If the angle of adjacent two lines is greater than this, safe turning. | 0~180 | 60 deg |

## 4.1.8 Process Wizard

For convenience of generating cutting tool path, you can use process wizard function, including line cutting, circle cutting, and multiple region cutting. You just need to set process parameters such as material dimensions to generate a process tool path.

Select **Process Wizard from** menu **Window** or select **ProcessWiz** under tab **PROG**, interface line cutting will be as shown in Fig. 1-1

The process parameters and process types for line cutting, circle cutting and multiple region cutting are introduced as follows.

◆ **Line Cutting**

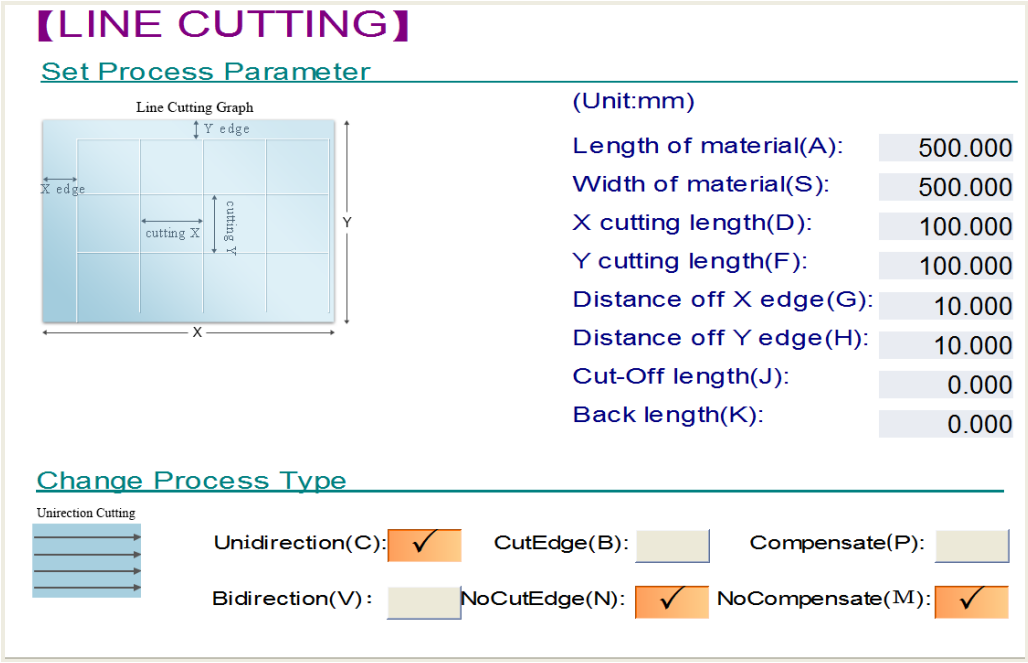



Fig. 4-14 Interface Process Wizard-Line Cutting

| | |
|---|---|
| Length/width of material | The length of material on X/Y axis. Shortcut keys: A/S. |
| X/Y cutting length | The unit cutting length on X/Y axis. Shortcut keys: D/F. |
| Distance off X/Y edge | The distances in X/Y axes between the first horizontal or vertical line and the origin of coordinate. Shortcut keys: G/H. |
| Cut-Off length | The tolerance length of remained material after an effective distance is processed. It is also the end point of tool. It is set to protect the glass in effective distance from being damaged by the tool. Shortcut key: J. |
| Back length | The tolerance length of remained material before an effective distance is processed. It is also the start point of tool. It is set to protect the glass in effective distance from being damaged by the tool. Shortcut key: K. |
| Unidirection | Material is cut in one direction of an axis during machining. |
| Bidirection | Material is cut in the positive or negative direction of an axis in turn during machining. |
| CutEdge | Cut down all the material when cutting the first edge in X axis direction. |
| NoCutEdge | Do not cut edge. Process normally. |

| | |
|---|---|
| Compensate | Compensate the screw error in X/Y axis direction. |
| NoCompensate | Do not compensate the screw error. Process normally. |

◆ **Circle Cutting**

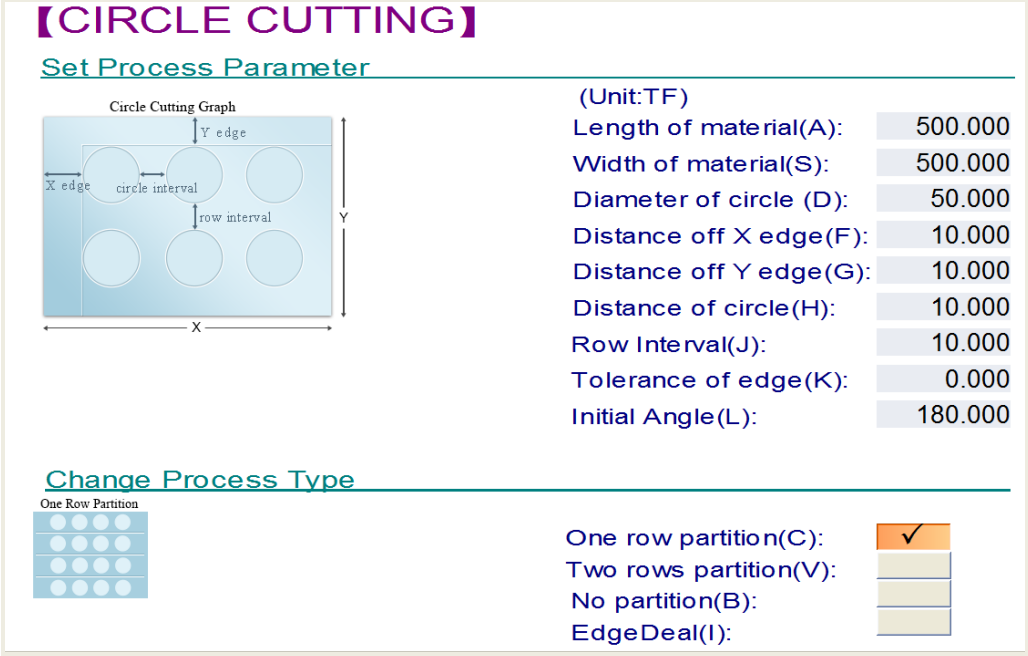



Fig. 4-15 Interface Process Wizard-Circle Cutting

| | |
|---|---|
| Length/width of material | The length of material on X/Y axis. Shortcut keys: A/S. |
| Diameter of circle | The diameter of the circle cut on the material. Shortcut key: D. |
| Distance off X/Y edge | The distances in X/Y axes between the first horizontal or vertical circles and the origin of coordinate. Shortcut keys: F/G. |
| Distance of circle | The distance in X axis between two adjacent circles. It is not the distance of the centers of two circles. Shortcut key: H. |
| Row Interval | The distance in Y axis between two adjacent circles. It is not the distance of the centers of two circles. Shortcut key: J. |
| Tolerance of edge | The tolerance length of remained material in Y axis. If the distance from edge of the last row of circle in Y axis to the material edge is greater than tolerance of edge, a cutting line in x axis direction will be generated. Otherwise, the row cutting line will not be generated. Shortcut key: K. |
| Initial angle | The deviation angle of starting position of tool. Shortcut key: L. |
| One row partition | Circles are cut on the material by rows. Shortcut key: C. |
| Two rows partition | Every two row Circles are cut on the material. Shortcut key: V. |

| | |
|---|---|
| No partition | The circles are cut one by one with no cutting line. The connecting lines of centers of adjacent circles form an equilateral triangle. Shortcut key: B. |
| EdgeDeal | Generate a cutting line in Y axis direction according to the distance of circle. The distance from the line to the left circle equals the distance of circle. Shortcut key: I. |

◆ **Multiple Region Cutting**

Currently, material can be divided into at most 10 regions. Different regions can have different parameter settings to achieve cutting different graphs on different regions and make good use of material.

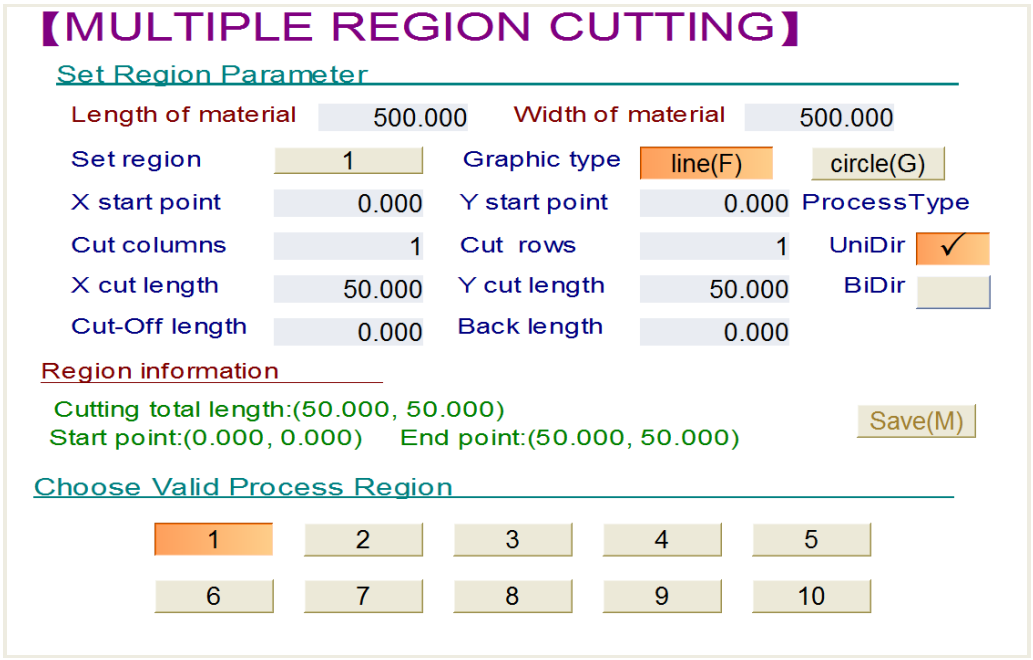



Fig. 4-16 Interface Process Wizard-Multiple Region Cutting

Interface multiple region cutting is as shown in Fig. 4-16. You can click on the button beside **Set region** to set a region number between 1~10 according to the length of material. Select **line** or **circle** as the graphic type. Then set X/Y start points, cut columns and rows, and process parameters and types unique to line or circle cutting. The system will cut the total length automatically. Start point and end point information is shown under **Region information**.

## 4.1.9 Spacing Compensation

Row offset or column offset caused by errors such as screw error and machine tool assembly error needs to be compensated. The steps to set spacing compensate are as follows.

◆ **Steps to Set Spacing Compensate**

1) Access interface **ProcessWiz** according to section 4.1.8. and select **LineCut** to get sub-interface Line Cutting, as shown in Fig. 4-14. Set process wizard of linear cutting, including length/width of material, cutting length in X/Y axis, the distance off X/Y edge, cut-off length and back length. Then proceed with trial cutting.

2) Check **Compensate** under **Change Process Type**, and dialog box “Spacing Compensate” will pop up, as shown in Fig. 4-17. In the dialog box, select X direction, and enter a row No. and the relevant measured data, i.e., data of trial cutting result. Then select Y direction, enter the column No. and relevant measured data. By clicking button **Insert**, you can add a new line for entering data. After entering all the measured data, click button **Save** and **Apply** in turn.

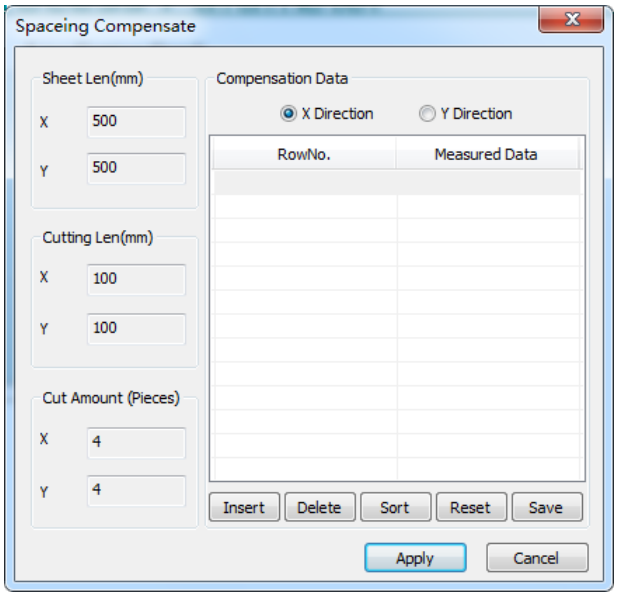



Fig. 4-17 Dialog Box “Spaceing Compensate”

| | |
|---|---|
| Save | Row No. or column No. and the relevant measured data must be saved and then applied. Otherwise, the data will be cleared automatically when you open the dialog box again. If you have not saved the data you entered in X direction, when you switch to Y axis, a prompt “The data of X direction is changed, save it or not?” will pop out. |
| Enter | Row No. or column No. must be less than or equal to cut amount. Otherwise, a prompt “Please enter an integer within [1, Cut Amount] and input value cannot surpass Cut Amount!” |
| Insert | Insert a new line in front of the current line. |
| Sort | Sort the data in ascending or descending order according to row No. or column No. |
| Reset | Reset all the data in X direction or Y direction and save the change. Then all the data in X direction or Y direction will be cleared. |
| Compensation | Cutting length in Y axis is relevant to the compensation in X direction while cutting length in X axis is relevant to the compensation in Y direction. |

The actual cut-off length of a specific row or column can be calculated through the formula below.

Cutting Length in Y Axis – The Measured Data of a Specific Row + Cutting Length in Y Axis = The Actual Cutting Length of the Row

Cutting Length in X Axis – The Measured Data of a Specific Column+ Cutting Length in X Axis = The Actual Cutting Length of the Column

## 4.1.10 Continuous Processing

To operate tasks in batches, in auto mode, select **Normal** under tab **POS** and press F3 or click **ConProcess** button, a dialog as shown in Fig. 4-18 will pop out.

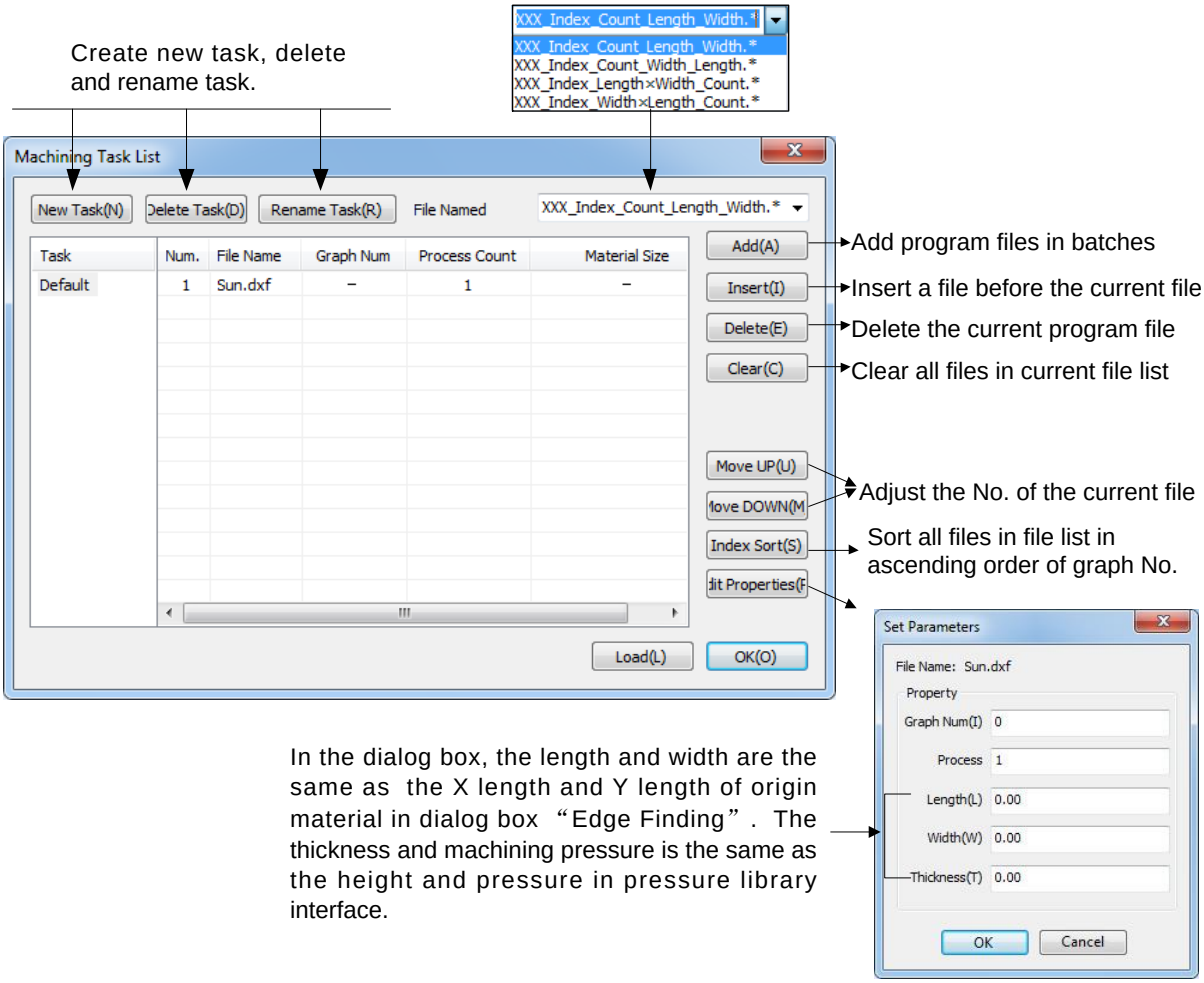



Fig. 4-18 Dialog Box "Machining Task List"

In NcStudio, four models of file name can be identified in the software. If the name of a loaded file is based on one of the four models, the system will automatically record and show graph No., process count, material size and other information. If the file name is not based on any models, you need to click on **Edit Properties** to set the parameters manually.

After entering continued machining task list, click button **Load** to execute machining. If the machining mode is assembly line mode and there is signal of workpiece automatically transmitted after machining finished, tool paths will be machined one by one automatically. Otherwise, you need to manually press key **Start** to machine the next tool path. Diagrams for assembly line and continued processing are shown as Fig. 4-19 and Fig. 4-20.

If you do not need continuous processing, you can press F4 to select **Cancel-Contin** before or when the machine tool is processing the current task. Then the workpiece will be transmitted after the current task is finished, and continuous processing will not be proceeded.

☞ **Related Parameters**

| Parameters | | Description | Setting Range | Default |
|---|---|---|---|---|
| N84000 | UpCylinderDelay | Waiting time in up-cylinder | 0~600000 | 2000 ms |
| N84008 | WaitGlassDelay | Waiting time in Wait Glass | 0~100000 | 2000 ms |
| N84023 | Workpiece is_automatically _transmitted | Whether workpiece is automatically transmitted after finishing machining. | Yes: automatically transmitted;<br>No: not automatically transmitted | No |
| N84024 | Migration GlassDelay | Waiting time in migration glass | 0~100000 | 3000 ms |
| N84027 | JourneyDelay | Waiting time in Journey | 0~100000 | 2000 ms |
| N84040 | With or without A | Whether shield A axis during machining | Yes: without A;<br>No: with A | No |
| N84041 | DelayStartGlass LoadingMachine | Set the waiting time of starting glass loading machine to avoid two pieces of glass knock into each other because of short distance | 0~100000 | 3000 ms |
| N84050~ N84052 | Passer Glass Position X/Y/Z | The Position when passing glass | -100000~100000 | 0 mm |
| For parameter N84040 “With or without A”, if it is set as “Yes”, the system will shield the machining information of A axis. | | | | |

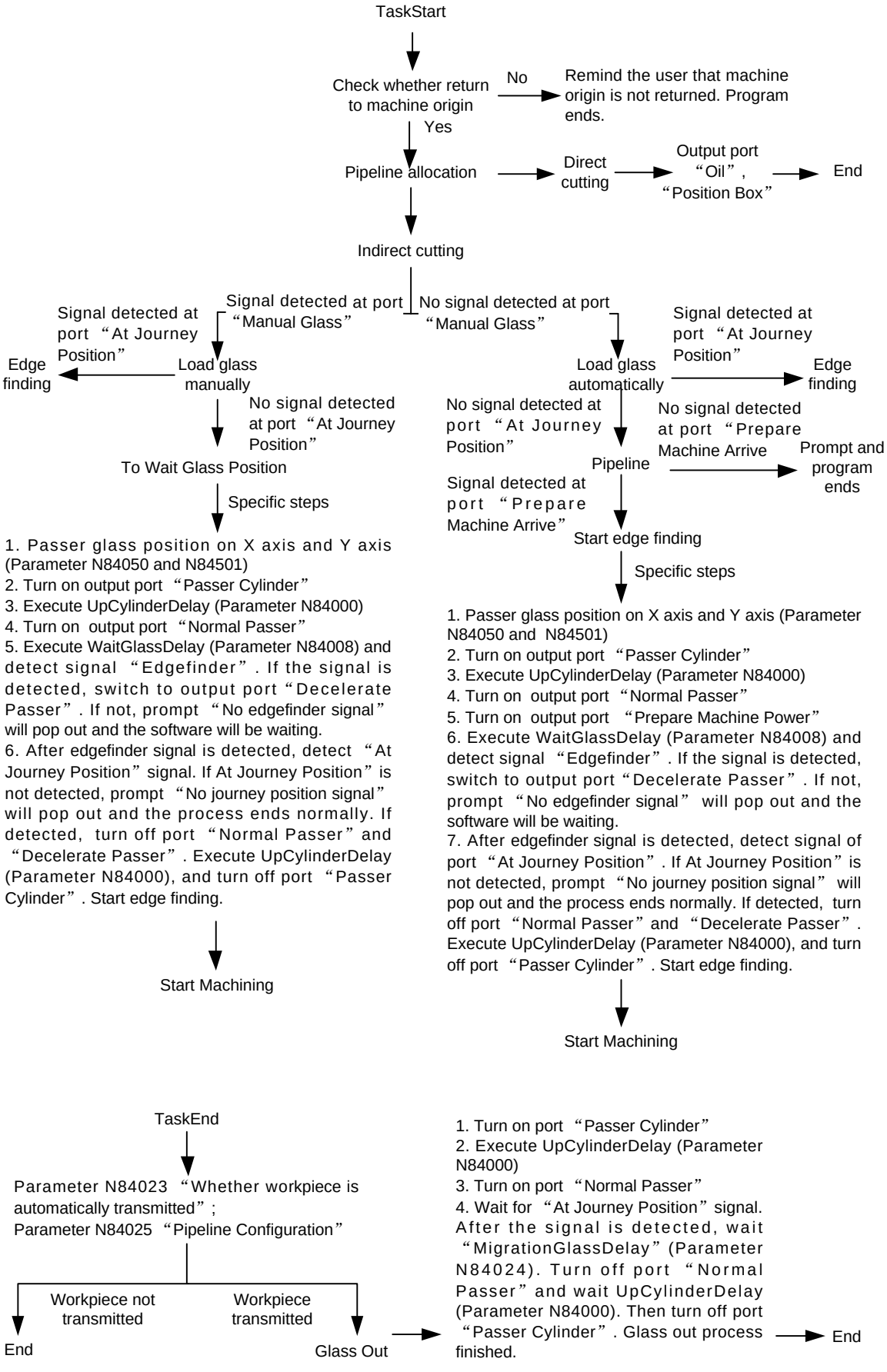



Fig. 4-19 Diagram of Assembly Line

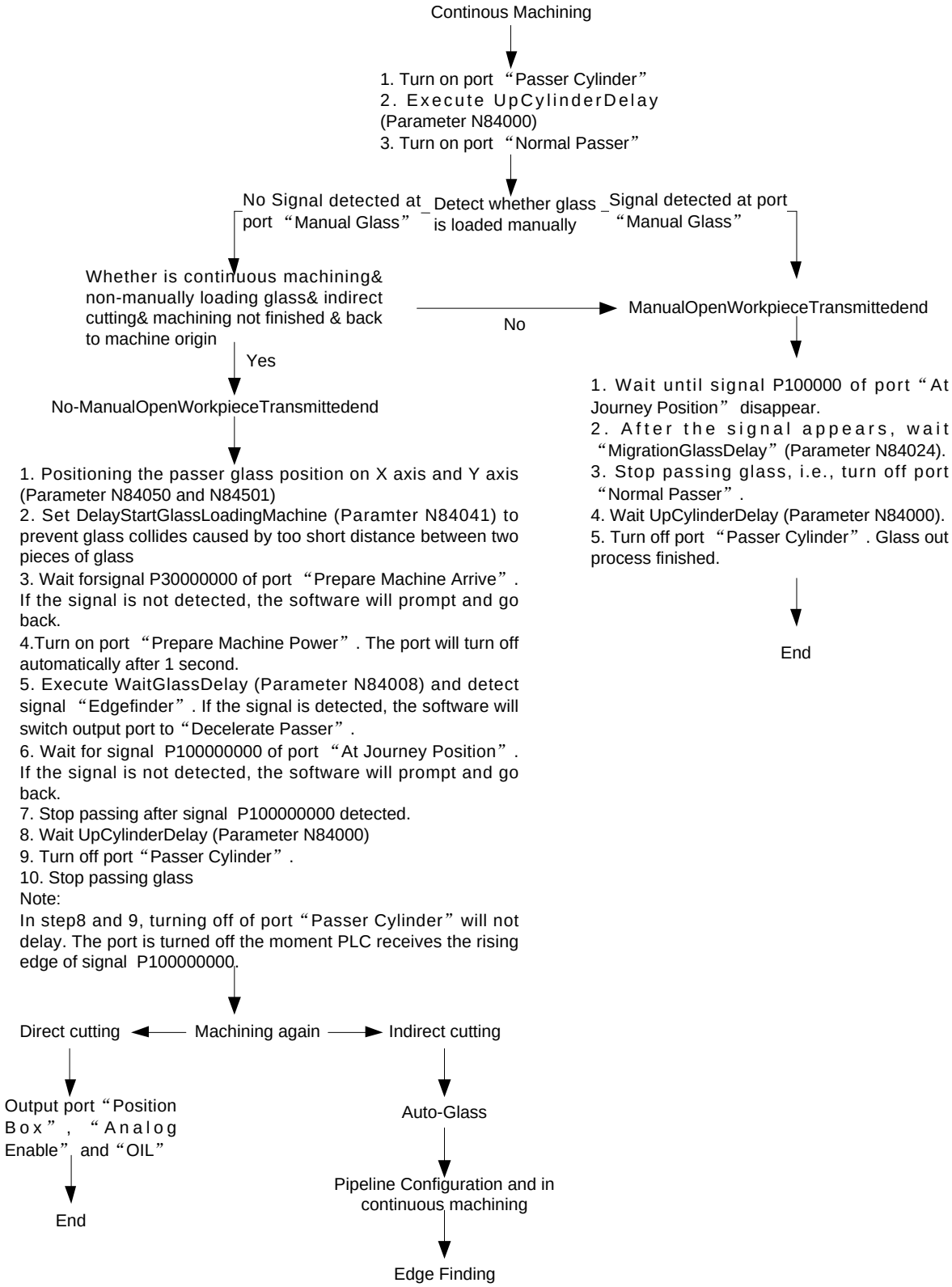



Fig. 4-20 Diagram for Continued Processing

## 4.1.11 Tool Life

You can set the max length of machining and max count of machining, and the system will remind you of tool life. Setting steps are as follows.

1) In auto mode, select tab **Setting** under tab **POS**. then set the max length and count of machining, as shown in Fig. 4-21.

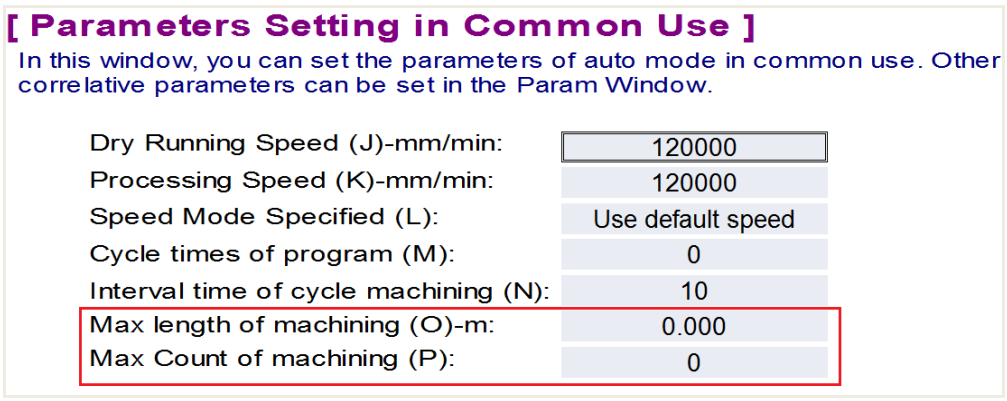



Fig. 4-21 Set the Max Length and Count of Machining

2) After setting the max length and count of machining, load tool path file and begin machining. When the value of **Max length of machining** becomes 1, a yellow prompt,

The wheel will reach its life-span, Please wait for changing another one!, will pop out automatically.

3) If the max length or count of machining has been reached when machining ends, the system will remain in **Pause** state and yellow prompt, The wheel has reached its life-span, change another one please!, will pop out.

4) Click **Start** again, and the machine tool will return to fixed point and the software enters **Idle** state. At the same time, in interface **Info** under tab **POS**, **Machine Length** shows the current machining length, and **Workpieces Count** shows the current machining workpiece count, as shown in Fig. 4-22.

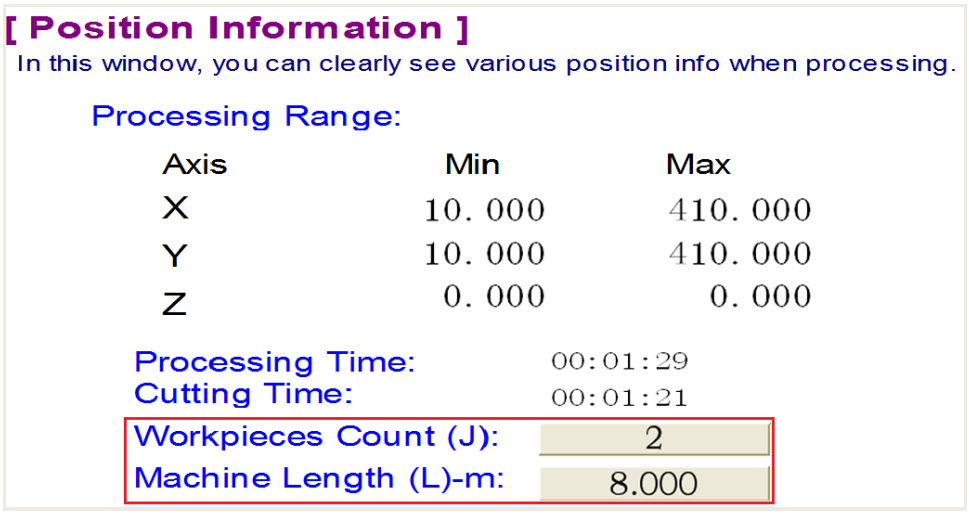



Fig. 4-22 Info Interface

5) If you continue machining, a yellow prompt

The wheel has reached its life-span,please change another one and reset the current machining length. will pop out.

6) Every time an object is cut, the software will enter Pause state and yellow prompt The wheel has reached its life-span, change another one please! will pop out. After machining ends, **Workpieces Count** and **Machine Length** will accumulate according to the current machining. Repeat step 5.

**CAUTION**

1) If the "Max length of machining" is set as "0", no prompts will pop out when maching ends.
2) Click the button **Machine Length** to reset the length machined. Click button **Workpiece Count** to reset the curent workpiece count.
3) Parameter N79311 and N79312 are customized parameter. Under customized configuration, you must set the two paramters before begin the steps above. Parameter N79311, "CHANGE_TOOL_PROMPT" specifies types of changing tool prompt, including: 0: no prompt; 1: prompt by task max num; 2: prompt by mach max len. Parameter N79312, "WHEEL_LIFE_WARNING_LENGTH", specifies the wheel life run out length

## 4.1.12 Clip Overlap

In art glass mirror cutting, there is overlap in dxf formatted graphs. If the overlap is not handled, the machined glass will be scrapped and you cannot break it normally.

- You can clip the overlap in files imported into NcStudio by setting parameter N65107.

**Related Parameters**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N65107 | Clip Overlap | Clip overlap when loading dxf file | Yes: Cilp overlap;<br>No: Do not clip overlap |
| Open **All Param** interface, and click **Manufactr** to open manufacturer parameter list. Set parameter N65107 "Clip Overlap" as "Yes". After setting the parameter, reload files to start machining, and the system will execute clip overlap operation to files loaded automatically during machining. | | | |

- If you need to clip overlap in NcEditor, you can use the following two methods.

Click **Options** under menu **Object**, a dialog box will pop out. Check the box before "Auto-clip the overlap after importing file", as shown in Fig. 4-23. Click **OK** to save the setting. The system will automatically clip overlap after files are imported.

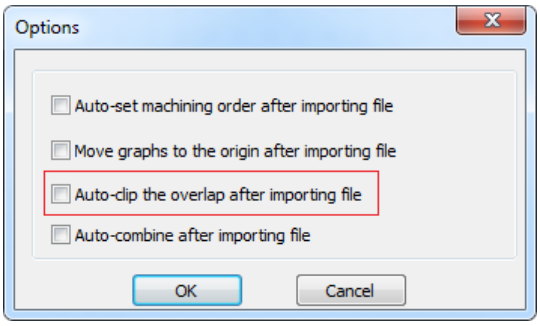



Fig. 4-23 Auto-clip the Overlap after Importing File

In NcEditor, draw or import the machining files, then select all the files, right click and select clip overlap. After clip overlap is finished, a prompt will pop out to remind you that the process is finished.

## 4.1.13 Walk Bound Rectangle

Walking bound rectangle around graphs is supported in the system. With this function, you can view the maximal machining area of the file and determine whether the graphs exceed the material dimensions and whether material is placed properly.

Load the program file. In manual mode, click tab **Normal** under tab **POS**. Then press F6 or click button **WalkBoundRec**. The system will generate a minimal rectangle around the bound of the graphs, as shown in Fig. 4-24. The yellow track is the bound rectangle generated.

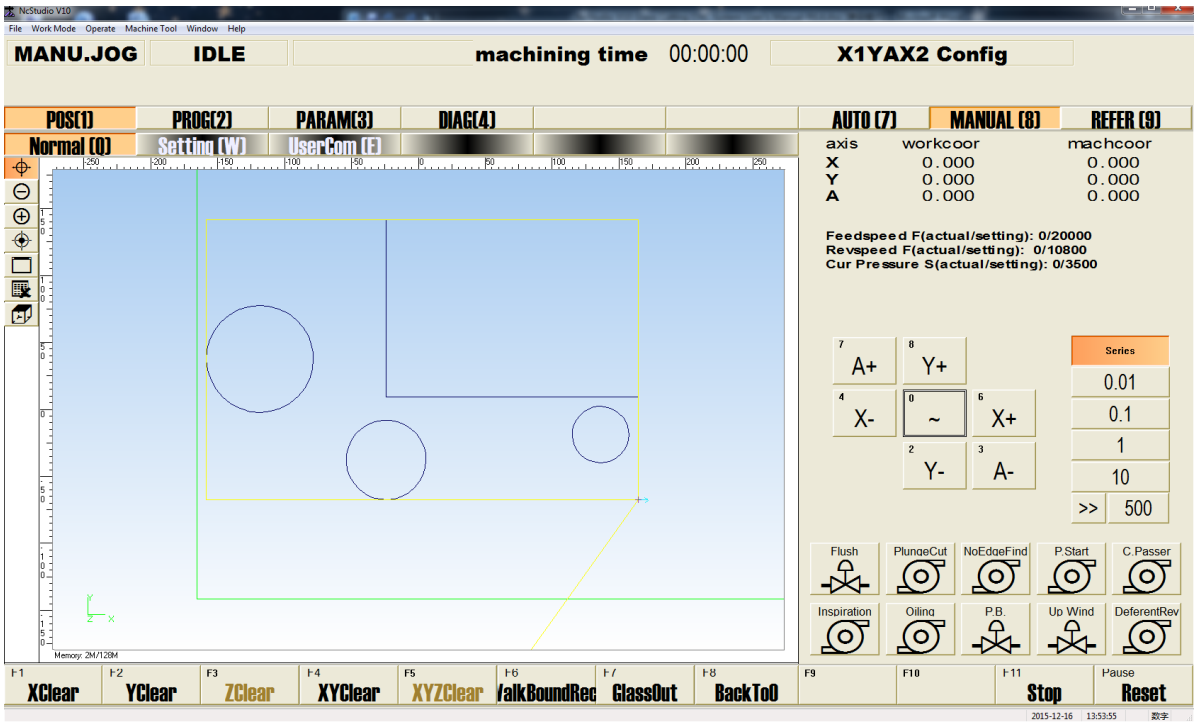



Fig. 4-24 Walk Bound Rectangle Track

**CAUTION**

The speed of walking bound rectangle is manual high speed, and the machine tool is in dry running during walking bound rectangle.

## 4.1.14 Cutoff

To solve the problem of uneven pressures, the system provides clip function. You can cut off polylines and set different pressures and speeds for different parts of the graph according to your needs.

In NcEditor, click the **Cutoff** icon button  in toolbar, and then click on any point of the graph, the dark blue point will appear where you click, as shown in Fig. 4-25.

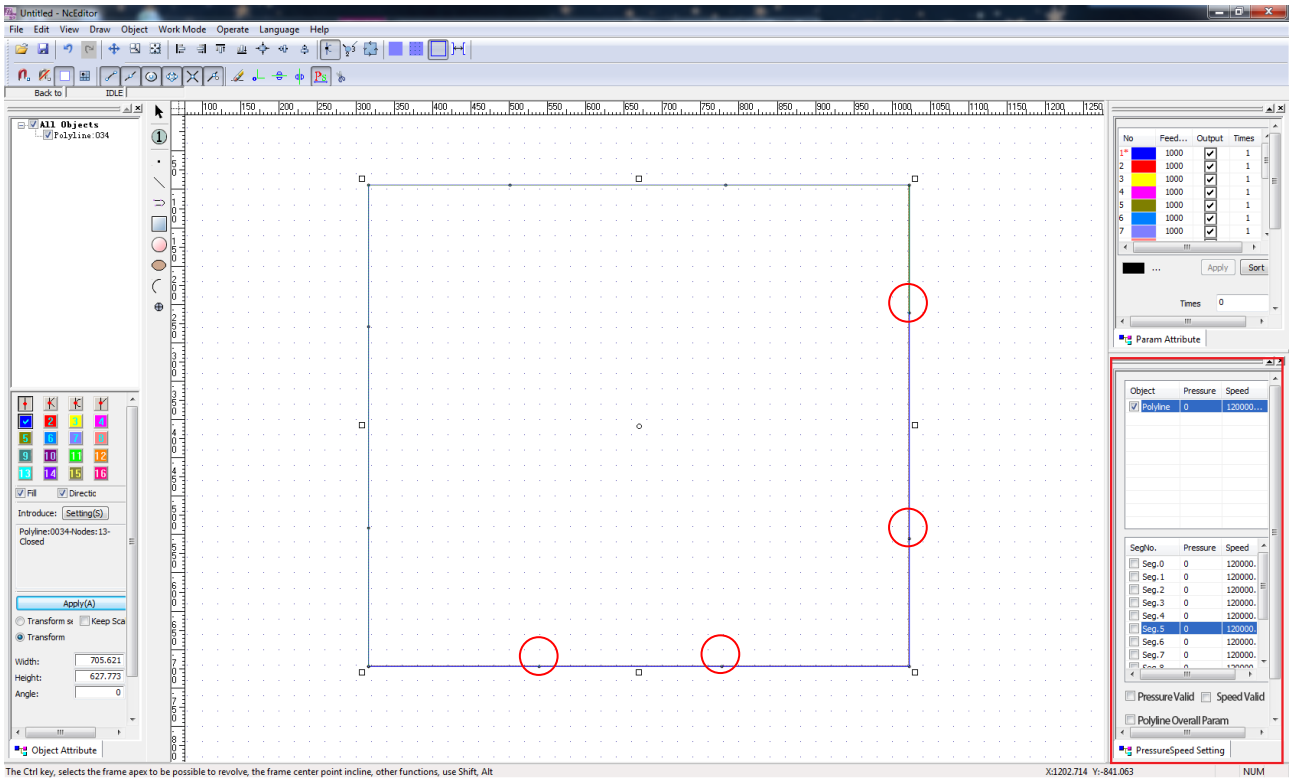


Fig. 4-25 Clip Function

There is no limit in cutoff points on graphs imported or drawn. After setting cutoff point, click the pressure speed icon button  . Then click on any clipped part of the graph, and set the pressure and speed for the part in **PressureSpeed Setting** pane, as shown in the above.

- **Pressure Valid and Speed Valid**

Check the box before **Pressure Valid** or **Speed Valid**, the box before **Polyline Overall Param** will be disabled. The machine tool will machine all parts of the polyline according to different pressure and speed settings.

- **Polyline Overall Param**

Check the box before **Polyline Overall Param**, the boxes before **Pressure Valid** or **Speed Valid** will be disabled. The machine tool will machine all parts of the polyline according to overall pressure and speed settings.

After setting the parameters, click **OK** to save the setting and start machining. During machining, the pressure and speed for each part of the graph will be shown in the software.

## 4.1.15 Optimization Software

The system supports optimization software from different manufacturers. You can set an optimization software directory in the system according to your needs.

After opening NcStudio, select the sub-menu [Optimize software] under menu [Operate]. And set optimization software directory in the prompt dialog box.

**CAUTION**

Take "EDIT-WAY" as an example to introduce optimizzation software. In software "EDIT-WAY", you can input data, optimize layout, create and manage cutting orders and execute other operations. Below is an introduction to the the basic functions of the optimization software.

1) Create new optimization based on new piece list;
2) Directly open or use the optimize results produced in Optima or other systems compatible with Optima;
3) Modify any of the optimization result, for example, changing the position of the piece, add new piece or delete piece;
4) Generate a CNC cutting machine code from the optimize result and send the code directly to full-automatic CNC cutting systems.

## 4.1.16 Edge Finding

Under configurations with edge finding function, press sub-tab **Normal** under tab **POS**. In the interface, press F6 or click button **EdgeFind**, and a dialog box, as shown in Fig. 4-26, will pop out. Edge finding function refers to deflecting the coordinates of the tool path with photoelectric positioning when the material glass is biased or the material is deflected after being transmitted. The function is used to ensure the quality of cutting. In addition, you can also use the function to positioning the glass blank. However, the function is only enabled in machine tools with photoelectric switches.

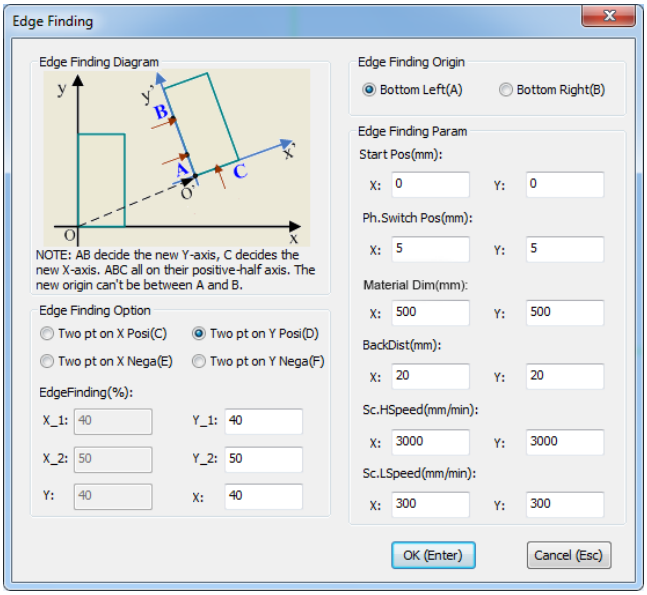



Fig. 4-26 Edge Finding Dialog Box

### ◆ Edge Finding Option

There are four options in edge finding, “Two pt on X Posi”, “Two pt on X Nega”, “Two pt on Y Posi”, and “Two pt on Y Nega”. Two pt on X Posi/Nega refers to positioning two points in X axis to set a new X axis, while Two pt on Y Posi/Nega refers to positioning two points in Y axis to set a new Y axis.

Positive edge finding refers to edge finding from the outside of a workpiece while negative edge finding refers to edge finding from the inside of a workpiece,as shown in Fig. 4-27.

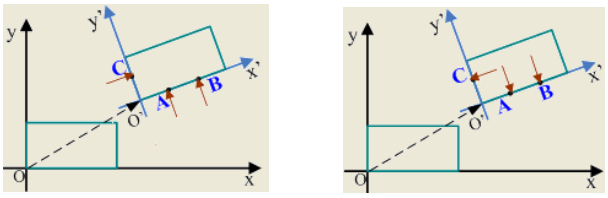



Fig. 4-27 Positive Edge Finding and Negative Edge Finding

Edge finding percent: by multiplying the parameter with the corresponding length, you can get the position of an edge finding start point.

### ◆ Edge Finding Origin

You can set the position of edge finding origin as “Bottom Left” or “Bottom Right” according to the structure of the machine tool.

### ◆ Edge Finding Parameter

| | |
|---|---|
| Start Pos | The start point of positioning in edge finding of X axis and Y axis. In edge finding of X axis, set the start Pos in Y axis, and in edge finding of Y axis, set the start Pos in X axis. |
| Ph.Switch Pos | The position of the photoelectric switch relative to the cutting head. |
| Material Dim | The dimensions of the material glass. The coordinate of Start Pos can be determined by multiplying the edge finding percent and a material dimension. |
| BackDist | After the machine tool receives edge finding signal, it will back off a distance to ensure it stays away from signal zone and then enter the next edge finding phase. |
| Sc.HSpeed | The feeding speed in rough positioning phase. |
| Sc.LSpeed | The feeding speed in accurate positioning phase. |

### ◆ Steps of Edge Finding

Take Two pt on Y Posi edge finding with Bottom Left edge finding origin as an example. Enter edge finding percent and then set edge finding parameters as shown in Fig. 4-26. Click **OK** to save the settings.

1) When the system detects “At Journey Position” signal, Y axis will moves to Start Pos of Y axis, and X axis will moves to edge finding percent X_1 position. Then Y axis will detect the edge of the glass at Y axis Sc.HSpeed. After the machine tool receives Edgefinder signal, Y axis moves forward 1mm from the current position. And then moves backward at Y axis Sc.LSpeed until the Edgefinder signal

disappears. Then the system records point A automatically and Y axis back off a distance according to parameter BackDist.

2) X axis moves to edge finding percent X_2 position and the following steps are same as in step 1). The system records point B automatically.

3) Y axis moves to edge finding percent Y position and the following steps are same as in step 1). The system records point C automatically.

4) Determine the new workpiece zero with point A, B and C. And the software will translate and rotate the original tool path.

**CAUTION**

1) Two pt on X Nega edge finding is a process which can detect Edgefinder signal from no signal to detected signal and then again to no signal.

2) The rotate angle of new wrokpiece cooordinate system relative to the original WCS cannot exeed 15° , otherwise edge finding fails.

3) The new origin cannot be between A and B, otherwise edge finding fails.

4) In traditional edge finding processes, the value of X_1 is less than that of X_2 and the value of Y_1 is less than that of Y_2. However, from the path, if the value of X_1 is greater than that of X_2 and the value of Y_1 is greater than that of Y_2, the total journey will be shorter and the time spent will be less. Therfore, free setting of values of X_1and X_2, Y_1and Y_2 is suppported in this system, except that X_1 is equal to X_2 or Y_1 is equal to Y_2.

# 4.2 Customized Configuration

## 4.2.1 Analog Pressure Control

For users who do not need auto pressure control in machining, you can set the corresponding parameter to control whether to use pressure control in machining.

Click on tab **Param** under tab **PARAM**, and press F1 or click on **All Param** button. Dialog box “All Parameters” will pop out. Find parameter N87002 in operator’s parameter and set the parameter according to your needs.

☞ **Related Parameters**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N87002 | Pressure Control | Whether to need Pressure control when machining | Yes: Use pressure control; No: Do not use pressure control |
| If you set the parameter as “Yes”, after opening the software, you should turn on port “Analog | | | |

| Parameter | Description | Setting Range |
|---|---|---|
| Enable" the first time you make the cylinder output, including start machining and open cylinder output port for the first time, and keep the port "Analog Enable" on until the software is closed.<br>If the parameter is set as "No", after opening the software, there is output at port "Analog Enable" only in cutting process. | | |

## 4.2.2 Loading Glass by Skip Cars

With this function, the skip car will load glass. Therefore, the glass position will not be limited by the position of glass loading machine.

◆ **Operation steps**

Draw or load tool path file which is supported by the software, and press key **Start**. The X axis and Y axis of the machine tool will be positioned to waiting glass position. Turn on the "Skip Car Active" port and control the output of the port.

The software will automatically detect "Glass Load Finish" port signal. If there is not signal, a prompt "No Skip Car Load Glass Finish Signal" will pop out. After the signal is detected, turn off the "Skip Car Active" port.

Wait until signal of port "At Journey Position" is detected, enter normal edge finding process. Start cutting when edge finding ends.

## 4.2.3 Dual Screen Display

To view the dimension of finished glass in breaking process and labeling the glass, you can use **Subscreen** function.

Dual screen includes a main screen and a sub screen. The main screen shows the current software interface and the sub screen can only display by selecting option **SubScreen** under menu **Machine Tool**. When machining ends and glass is transmitted out, the machining graph shown in sub screen will be updated to the original tool path, without machining track. Dimensions will also be shown on sub screen.

◆ **Operating steps**

1) On main screen, select option **SubScreen** under menu **Machine Tool**, and a blank page titled "Sub Screen" will pop out.
2) On main screen, press button **ConProcess** to open dialog box "Machining Task List". Select and load all the tool path files. When machining starts, the main screen will show the current cutting tool path, and the sub screen is still blank.
3) After the cutting of the first tool path is finished, switch to manual mode and press button **GlassOut** on main screen. Screenshot of the first tool path track will be sent to the sub screen.
4) And the second tool path will be loaded automatically on the main screen when the first tool path track is shown in the sub screen. In this analogy, every time machining ends and glass is

transmitted out, the sub screen will update tool path track to the last tool path track finished on the main careen.

**CAUTION**

1) When you move the cursor to the edge of the main screen, you can operate the sub screen, such as maximizing, minimizing or closing the sub screen.
2) You need to install a graphic driver to show the sub screen. After installing the graphic driver, right click on the desktop, and choose option **Screen Resolution**, and change the dispay, multidispay, resolution and orientation. Then the sub sceen will be displayed.

## 4.2.4 Inkjet System

You can use inkjet system to sort products after cutting process ends. The process is as follows.

◆ **Inkjet Process**

1) Inkjet starts.
2) Move to inkjet position 1.
3) Determine whether need to rotate. Rotate if needed.
4) After rotating, the inkjet cylinder outputs. The inkjet cylinder should output after cylinder withdraw in place signal is detected. And detect cylinder popup in place signal.
5) Send a “Trigger Inkjet Signal” to the inject printer.
6) X axis and Y axis start to move according to the inkjet track and jet the content at position 1 until the jetting at position 1 ends.
7) Positioning inkjet position 2 and repeat step 3) to 6) until all the inkjet is completed.
8) Withdraw the inkjet cylinder and detect cylinder withdraw in place signal.
9) Program ends.

**CAUTION**

1) In the software, layer 1 is used for cutting and tool No. for cutting is set as tool 1, as default. Layer 2 is used for inkjet and tool No. for cutting is set as tool 2, as default.
2) The default maching order is executing cutting first and then execute inkjet.

◆ **Layer Only Show Not Machine**

After cutting process ends, breaking glass should be executed. However, you cannot distinguish between glass pieces with similar size or shape and without inkjet mark.

You can solve the problem by marking the pieces after setting layer only show not machine. Set parameter N87005 “Layer Only Show”. And any graphs marked in the layer will not be machined during machining process.

☞ **Related Parameters**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N87005 | Layer Only Show | The Index of Layer only Show Not Machine. | 0~16 |
| N88002 | Inkjet System | Whether for cutting and inkjet system. | Yes: cutting and inkjet system;<br>No: cutting system |
| N88006 | Cut_Inkjet_Option | The options of cut and inkjet, including: 0: cut first then inkjet; 1: inkjet first then cut; 2: cut only; 3: inkjet only. | |
| N88007 | Inkjet Speed | The speed of inkjet. | 1~100000 mm/min |
| N88008 | Ref Height | When Height is greater than Ref Height, the inkjet cylinder cannot out. | 1~100000 mm |

## 4.2.5 Control Oil Analog Output by Adjusting Speed

It is an appropriate way to control oil percent by adjusting speed.

Click sub tab **Param** under tab **PARAM** and then button All Param in the interface. Then select manufacturer parameter N88005 “Max Ref Speed”. The equation is as below.

$$\text{Analog Voltage}=\frac{\text{Current Speed}}{\text{Ref Speed}}\times\text{Current Feed Rate}\times\text{Max Output Voltage(10.6V)}$$

- If the result of the equation is less than the product of Min Oil percent and Max Output Voltage, you need to set the product of Min Oil percent and Max Output Voltage as the current analog voltage.

  For example, if the current speed is 1000mm/min, ref speed is 10000mm/min and the current feed rate is 90%, and Min Oil percent is 10%,

  Analog Voltage =(1000 / 10000)×90%×Max Output Voltage(10.6V)=0.9;

  Min Oil percent ×Max Output Voltage=10%×10=1;

  0.9＜1, therefore, set 1 as the analog voltage for current machining.

- If the result of the equation is greater than the product of Max Oil percent and Max Output Voltage, you need to set the product of Max Oil percent and Max Output Voltage as the current analog voltage.

  For example, if the current speed is 1000mm/min, ref speed is 10000mm/min and the current feed rate is 100% and Max Oil percent is 90%,

  Analog Voltage =(1000 / 10000)×100%×Max Output Voltage(10.6V)=10;

  Max Oil percent ×Max Output Voltage=90%×10=9;

  10＜9, therefore, set 9 as the analog voltage for current machining.

☞ **Related Parameters**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N88003 | Max Oil Percent | The maximal oil percent when machining. | Min Oil Percent ~100 |
| N88004 | Min Oil Percent | The minimal oil percent when machining. | 0~ Max Oil Percent |
| N88005 | Max Ref Speed | Control cutting oil analog output by speed. | 1~100000 mm/min |

## 4.2.6 90 Degree Turning under Config Model 53C

Since A axis do not exist in NcStudio under Config Model 53C, you need to set a port to control the cylinder and rotate the tool with the cylinder.

Set parameter N87004 “Turn 90 deg” as “Yes”. When the program is running, it will determine the tool path is horizontal or vertical. If the tool path turns from horizontal direction to vertical direction, the icon button **90DTurn** in auto mode will turn green. The system will control the output port “90 Degree Turn”. The tool will then rotate 90 degree. If the tool path turns from vertical direction to horizontal direction, the icon button “90DTurn” is grey. The output port “90 Degree Turn” is turned off, and the tool will cut horizontally.

You can also click the button **90DTurn** down to control the output port “Turn 90 deg”, as shown in Fig. 4-28.

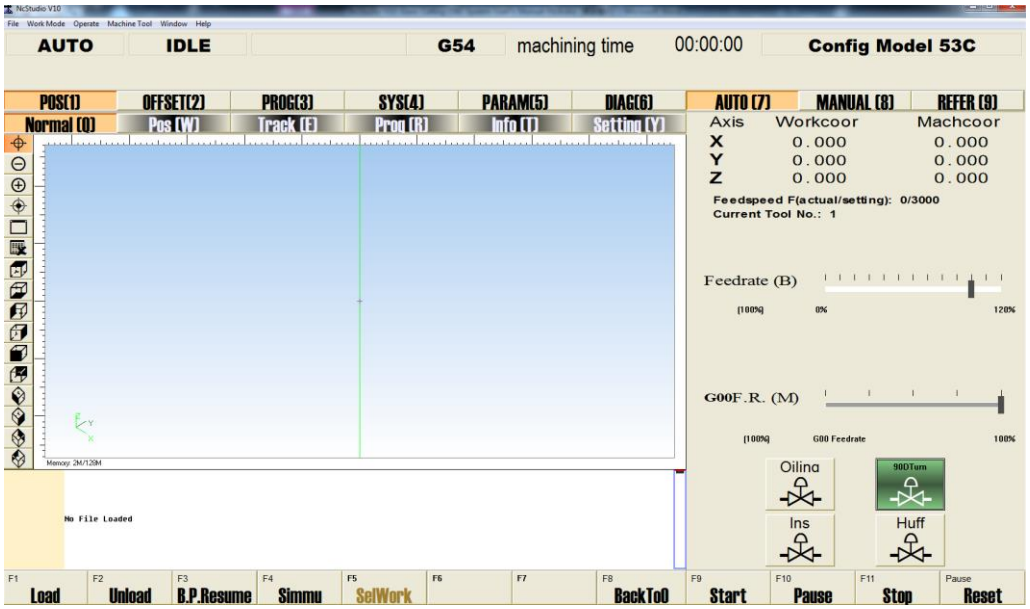



Fig. 4-28 90 Degree Turning

☞ **Related Parameters**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N87004 | Turn 90 deg | Whether to turn 90 deg by cylinder control when machining | Yes: control;<br>No: not control |

## 4.2.7 Sharpening

◆ **Steps of Auto Sharpening**

1) The software detects signal “The spindle start”. If signal detected, machining starts. If not, a prompt “加工前未启动主轴” will appear in the CNC info bar.
2) When max workpiece count is reached, the tool is positioned to “TOOL_COOR”;
3) After waiting for sharpen tool time, the tool reaches “TOOL BACK TO POS”;
4) The software will reset workpiece count automatically after sharpening, as shown in Fig. 4-29.

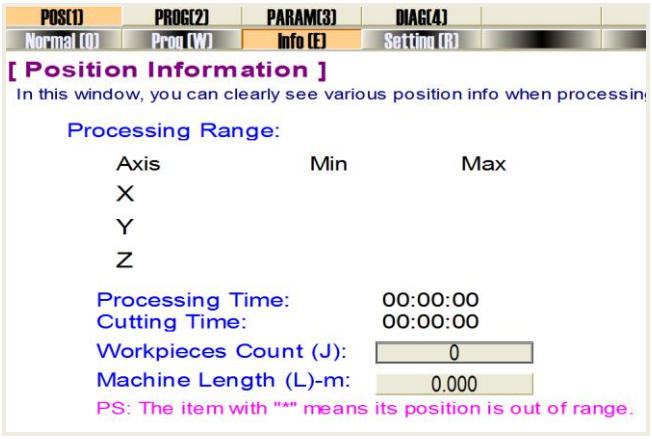



Fig. 4-29 Reset Workpiece Count

**CAUTION**

Sharpening process will not start when actual workpiece count is greater or less than max workpiece count.

◆ **Manually Sharpening**

1) In the current interface, press F6 to select function button **Sharpening**, as shown in Fig. 4-30. The machine tool will reach “TOOL_COOR”;


Fig. 4-30 Manually Sharpening

2) After waiting for sharpen tool time, the tool reaches “TOOL BACK TO POS”;
3) The software will reset workpiece count automatically after sharpening.

☞ **Related Parameters**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N67000 | TOOL_COOR | The machine coordinate of the sharpening tool. Default value is 0. | Depends on actual situation |
| N67008 | SHARPEN TOOL TIME | Waiting time is needed to complete sharpen tool. | 0~100000 s |

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N67009 | TOOL BACK TO POS | Return the machine coordinate after sharpening too. Default value is 0. | Depends on actual situation |
| N67017 | MAX WORKPIECE CUNT | Count more than the master number, need to sharpen tool. Default value is 10. | 0~10000 |

### 4.2.8 High and Low Pressure Output Control

If you use a proportional electromagnetic valve to control pressure instead of using analog pressure control, you will need to control different pressure output ports for different cutting graphs. In linear cutting, you can control the output of high pressure proportional electromagnetic valve with port “Linear Interp Output”. In arc cutting, you can control the output of high pressure and low pressure proportional electromagnetic valves with port “Linear Interp Output” and “Arc Interp Output”.

☞ **Related Parameters**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N87002 | Pressure Control | Whether to need Pressure control when machining | Yes: Use pressure control;<br>No: Do not use pressure control |

**CAUTION**

The version and interface of customized software differs for different users. If you have any questions, please contact with our techinical support center.

## 4.3 Auxiliary Function

### 4.3.1 NcEditor Graph Operation

After importing or drawing graphs, you can set process parameters of the graphs, including combining object, grouping and arraying, and expanding bidirectionally.

#### 4.3.1.1 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.

- **Combine Object Tolerance Setting**

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.

### 4.3.1.2 Catch Option

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. 4-31 will pop up.

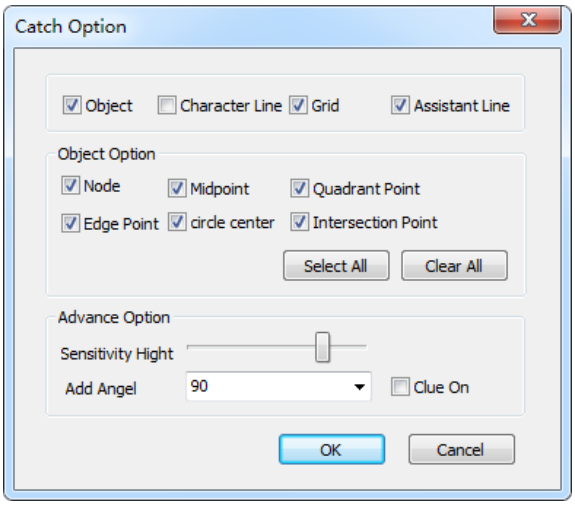



Fig. 4-31 Dialog box "Catch Option"

- **Character Points of Different Graphs**

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 points.

Character line is used to hint a specific position.

Add angle refers to the angle of catching operation. For example, if you set add 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 degrees, the character line will appear on the editing space.

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.

#### 4.3.1.3 Group and Array

- **Group**

To do same edit on 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 the objects or object groups that need to be grouped, and then select **Group** under menu **Object**, or right click and select option **Group**.

- **Break Group**

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**. The objects in the group will go back to the upper group.

- **Rect Array**

In **Object List** window, select one or more objects or object groups, select **Rect Array**, and enter row number, column number, row space and column space in the dialog box popping up. Click **OK** to confirm the setting.

- **Circle Array**

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

#### 4.3.1.4 Chamfer

NcEditor supports two types of chamfer, **Chemfer** and **Fillet**. There are two modes, **The two sides** and **Internal graphics**, for each type of chamfer.

**1) The Two Side Mode**

In the two side mode, chamfer refers to the angle between two edges.

For **Chemfer** type, there are two kinds of size setting, distance setting and angle setting. To draw the 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 the chamfer with angle setting, select **Angle**, and enter the relevant value into input boxes. Click **OK**. Then select two adjacent two edges.

For **Fillet** type, there is only one size setting, radius setting. Enter a value and click OK. Then select two adjacent 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.

#### 4.3.1.5 Expand Bidirectionally

You can set left compensation or right offset in tool path, but you cannot set left and right offset at the same time. Function Expand Bidirectionally can achieve left and right offset setting. Select **Expand Bidirectionally** under menu **Object,** or right click and select option **Expand Bidirectionally**, and a dialog box will pop up. The **Cutter Diameter** and **Chord** is the same as **Tool Dia.** and **Chord** in dialog box “Tool Path”. Inner offset and outer offset are equal to left compensation and right compensation. After setting all the values in the dialog box, click **OK** to confirm the setting.

## 4.3.2 System Management

### 4.3.2.1 Language Choice

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

Select option **Change Language** under menu **Machine Tool**, and switch language between English and Chinese.

### 4.3.2.2 View System Information

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

Select submenu **AboutNcStudio** under menu **Help**, and you can view system information in dialog box “About NcStudio”, as shown in Fig. 4-32.

### 4.3.2.3 Register

“Register” function is used to limit the usage time of the system.

Click “About NcStudio” item under menu **Help** to open a dialog box named “About NcStudio”, as shown in Fig. 4-32, where you can obtain information such as software version, the developer, system information, as well remaining usage time. If the remaining time runs out, please send the “Device No.” to manufacturer to get a registration code. After you receive the registration code, click “Register” button in Fig. 4-32, and dialog box “REGISTER” will pop up. Input the registration code and confirm to register usage time.

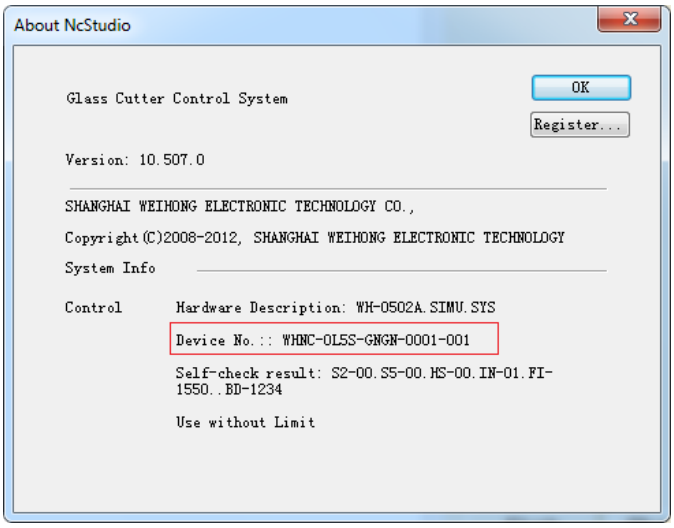



Fig. 4-32 Dialog Box "About NcStudio"

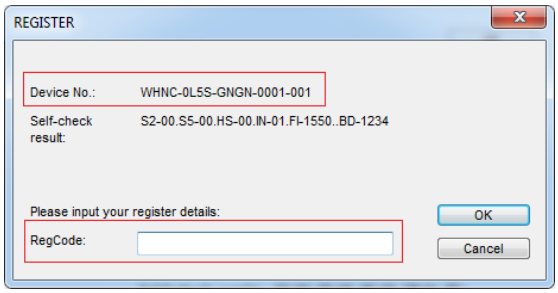



Fig. 4-33 Dialog Box "REGISTER"

### 4.3.2.4 Manufactory Customize

Select **AboutManufactory** under menu **help**, dialog box "Manufactory Customize" will pop up, as shown in Fig. 4-34. You can modify the splash of English or Chinese Interface by selecting **Chinese** or **English** in this dialog box. The picture imported for splash must be BMP format. Restart the software after you confirm the import to complete setting of splash. You will need to login to modify the title and add your company logo on the software. After login, dialog box shown as Fig. 4-35 will pop up. You can set title, splash and LOGO in this dialog box. After setting a title, the software title will be updated immediately. The structure of the title will be "NcStduio icon + NcStudio + the title you set".

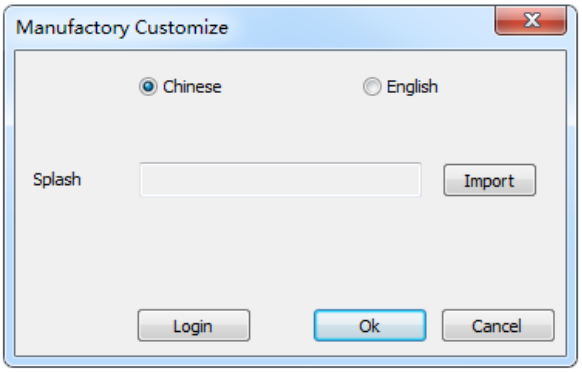



Fig. 4-34 Manufactory Customize-1

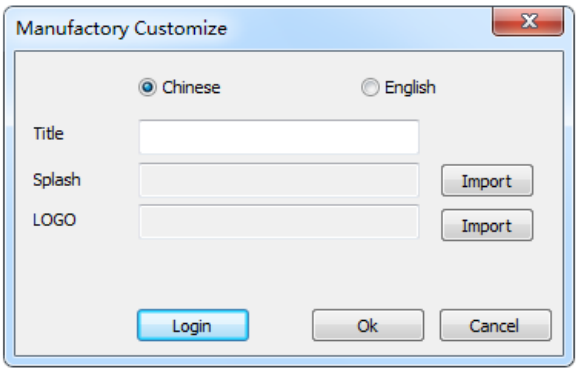



Fig. 4-35 Manufactory Customize-2

## 4.3.3 Shortcuts

Shortcuts include shortcut icon buttons, shortcut menus and shortcut keys.

### 4.3.3.1 Shortcut Icon Button

In any operation mode, access interface **Normal** under tab **POS**. The shortcut icon buttons are on the left of the window.

**Move.** You can achieve the function by pressing keys. When the button is pressed 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 +.

**Show Current Processing Point.** The shortcut key for the icon button 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 button is key Delete.

**Top View.** The shortcut key for the icon button is key W.

### 4.3.3.2 Shortcut Menu

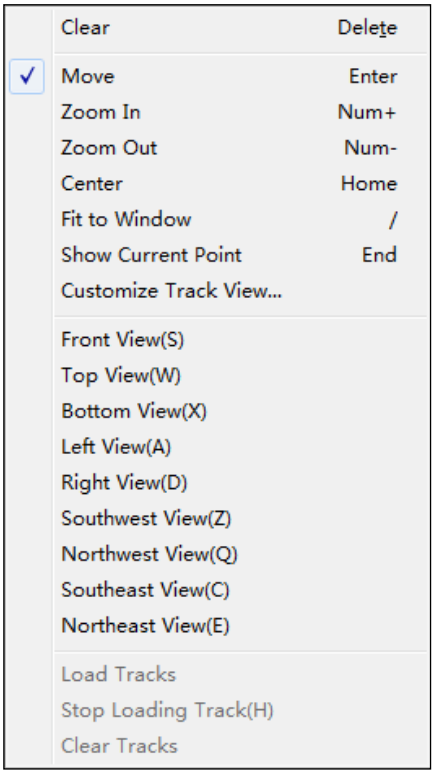
You can also right click on track window under sub-tab **Normal** to gain the shortcut menu.

- **Clear**

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

- **Customize Track View**

Click **Customize Track View**, and dialog box “Custom Parameter” will pop up.




Click a color button, and a color selection box will pop up. A new function preloading track is added into NcStudio V10. Command color is divided into simulation track command color and actual track command color.

In dialog box “Custom Parameter”, check **Show travel rang**, and workbench surface will be shown in the track window. Check **Draw the gridding**, and gridding will be shown in the window. Check **Auto Del Tracking View new files open**, the auto-clear function will be enabled. Click **Show figure’s dimension in G File**, and figure’s dimension in G file will be shown in the window.

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

### 4.3.3.3 Shortcut Keys Overview

| | | |
|---|---|---|
| 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 Manual Mode | Numeric key 1 | Manually move the machine tool in Z- direction |
| | Numeric key 2 | Manually move the machine tool in Y- direction |
| | Numeric key 3 | Manually move the machine tool in A- direction |
| | Numeric key 4 | Manually move the machine tool in X- direction |
| | Numeric key 6 | Manually move the machine tool in X+ direction |
| | Numeric key 7 | Manually move the machine tool in A+ direction |
| | Numeric key 8 | Manually move the machine tool in Y+ direction |
| | Numeric key 9 | Manually move the machine tool in Z+ direction |
| | Numeric key 0 | Accelerate |
| 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 |

**CAUTION**

All view shortcut keys in interface **Normal** will be enabled only after key S is pressed. And after the shortcut keys are enabled, key Q/W/E will be used as view shortcut keys instead of global shortcut keys.

## 4.3.4 Diagnosis

### 4.3.4.1 Log

In **Log** window under tab **DIAG**, important operation and system events can be recorded and displayed. As shown in Fig. 4-36, you can browse the log info since this time start-up as well as history log info.

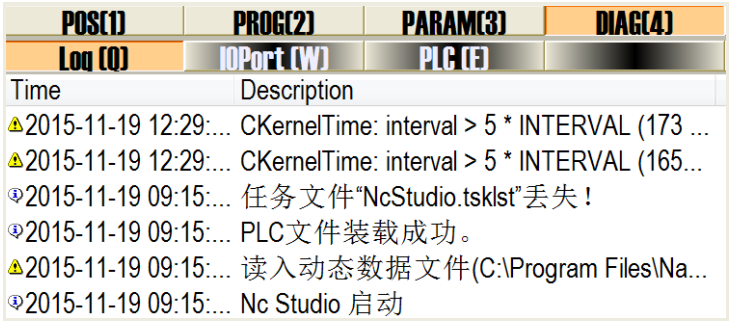



Fig. 4-36 Log

As shown above, the log list contains info like: initialization and shutdown of the system; starting and end of auto machining; changes of workpiece coordinate; system alarms; other information. The manipulation button bar at the lower part of the window is illustrated below:


Fig. 4-37 Operation Button Bar Below Log List

**Refresh**: shortcut key F1, used to refresh the current interface, and update the logs.

**Clear**: shortcut key F2, used to clear all current logs.

**Show Info/Warning/Error/System/Nowadays**: shortcut keys F3~F7. When button is pressed down, information indicated by the button will be displayed in the log list; otherwise, the information will be hidden. In default setting, buttons except F6 "System" are always been pressed down, namely, with convex appearance highlighted in orange. To put it in other words, information, warning and error info since this time start-up will be displayed by default. Click button to cancel the highlight, making the info hidden.

**CAUTION**

1) You can tell the types of info by icon in front. Each information log is marked with the icon , system log with icon , warning with icon and error log with icon .

2) When highlight of 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.

#### 4.3.4.2 Ports on Hardware

Please see Section Check Polarity of IO Ports.

#### 4.3.4.3 PLC

With built-in PLC module of the software, input and output ports can be controlled by PLC running.

You can turn to PLC window under tab **DIAG** to run the PLC. Refer to *Programming Manual of NcStudio* for details.

PLC window is shown as below.

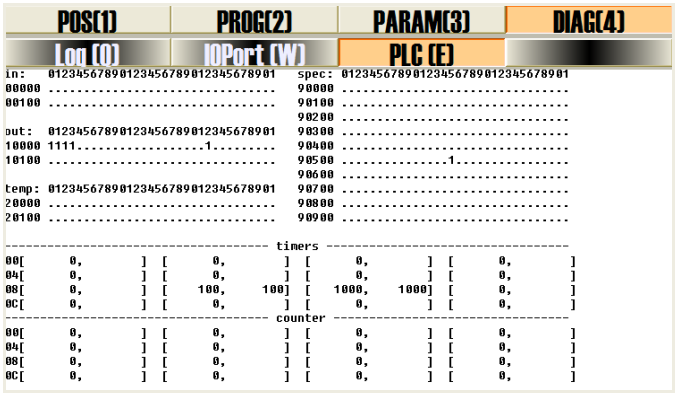



Fig. 4-38 PLC

Buttons on manipulation bar are shown as Fig. 4-39. When buttons are clicked, corresponding dialog boxes will pop out, where the user can input the channel No., timer No. and counter No. which need to be displayed. After it is confirmed, the ports information will be displayed in PLC window.

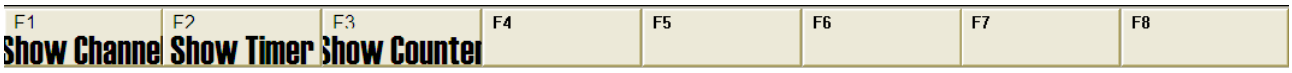



Fig. 4-39 Operation buttons below PLC window

## 4.3.5 User Command

Function “User Command” is supported in the system, and you can input G command online and execute it immediately.

In manual mode, select **UserCom** under tab **POS**, as shown in Fig. 4-40. In this window, you can enter 10 commands, and you can repeatedly enter command in the same line.

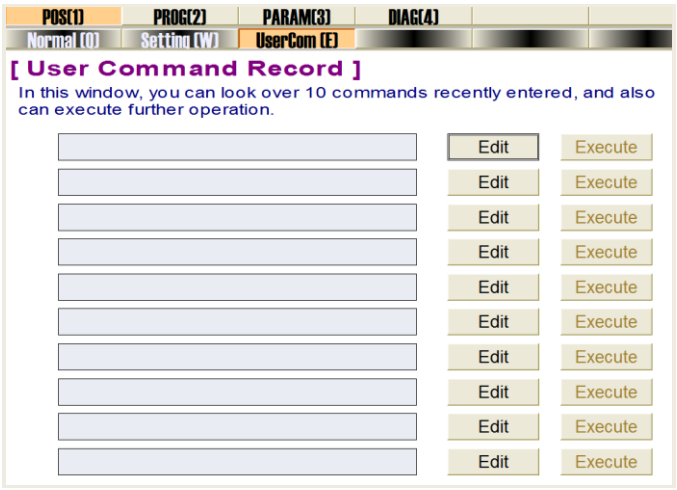



Fig. 4-40 Interface User Command Record

1) In the interface above, when no G command is entered in an entry, the button Execute on the right of the entry is disabled.

2) Click on button **Edit**, an input box will pop up. You can enter a command in the input box and click button **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.

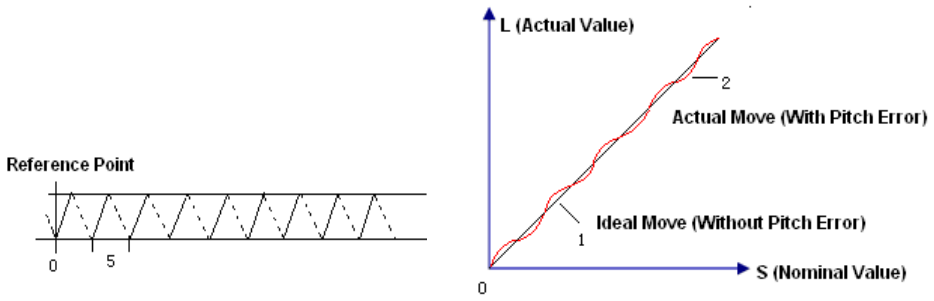
## 4.3.6 Parameters related to Compensation

Screw error consists of screw pitch error and errors caused by backlash. Generally, these two errors don’t need compensation, but backlash compensation is needed in high precision required situation, if higher precision is required, both the two compensations are needed.

◆ **Pitch Compensation**

Pitch error is caused by screw defect and long-term wear, etc. In order to improve precision, pitch compensation is needed to meet the requirement. The sketch of a screw is shown in Fig. 4-41(A). A coordinate system is established, based on “0” point on the screw as the reference point, nominal value as X-coordinate, and actual value as Y-coordinate. Then the ideal moving curve is as curve “1” in Fig. Fig. 4-41(B), however, the actual curve will be curve “2” due to pitch error. That is to say, the Actual value is not the same as its corresponding Nominal value, the actual moving curve deviating from the ideal one, and their difference is called error, i.e.:

Error = Nominal machine coordinate – Actual machine coordinate




(A) Sketch of screw (B) Picture of moving curve

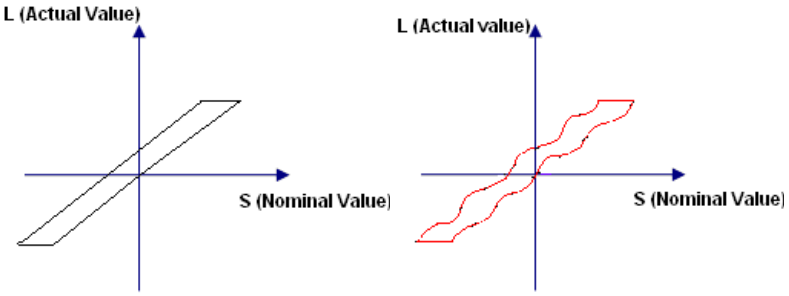
Fig. 4-41 Analysis of Pitch Error

### ◆ Pitch Error Compensation Method

In pitch compensation, generally pitch error value isn't related to feed direction. That is, when the pitch is too small in positive feed, additional pulse is needed, and thus, when negative feed passes the same position, the same amount of feed pulse should be added. But if the pitch is large, deduction of pulse is needed, and neither is the reducing amount related to feed direction. In software compensation, correction of each point on the error curve should be tabulated and saved to the system memory. Then auto compensation for coordinates of each point is available in running, so as to improve machine precision.

### ◆ Backlash Compensation

Hysteresis feature is caused by forward and reverse clearance. Assume that driving shaft drives driven shaft in negative (CW) rotation, servo motor will be idling without moving worktable because of mechanical driving chain backlash, when the driving shaft suddenly begins CCW rotation (positive motion). After staying at a certain position for some time, the worktable will move backward with the driving shaft; when the direction of the driving shaft changes again, the situation is the same, which is called Hysteresis. If pitch error doesn't exist, under ideal condition, the moving curve of worktable is shown in Fig. 4-42(A), in which the curve of horizontal section is during the idling of servo motor without worktable movement. The actual moving curve of worktable is shown in Fig. 4-42(B).




(A) Hysteresis feature (B) Actual moving curve

Fig. 4-42 Analysis of Backlash

The popular explanation is: because spindle is generally fixed on the screw whose outer wire and the inner wire on the outer wire cannot be completely matched, backlash compensation compensates the clearance between the screws of last direction that the spindle needs to finish after reversing its moving direction.

◆ **Measuring Method and Compensation Method for Backlash**

Backlash can be measured by a specialized gauge. Firstly, fix the instrument nearby the spindle. Secondly, make the watch hand at the zero point position (machine origin). Thirdly, manually move “a” millimeter, then move back “a” millimeter, and then see the actual moving distance of watch hand “b” millimeter. Therefore, the backlash is measured, namely (a-b) millimeter.

If one axis moves from positive to negative, “+Q” pulse will be output before reversal; conversely, from negative to positive, “-Q” pulse will be output before reversal (Q is backlash, preset by the program).

☞ **Related Parameters (Manufacturer Parameters)**

| Parameter | | Description | Setting Range |
|---|---|---|---|
| N12000 | ENABLE_LEADSCREW_CMPN | The system enables compensations for the leadscrew errors, including compensations for backlash errors and pitch errors. | YES: enable; NO: disable |
| N12001 | ONLY_BACKLASH_CMPN | If true, the system will only read data from BACKLASH parameters to do compensation. Otherwise, it will read the data of backlash and pitch errors from error data file and take comprehensive compensation. | YES: enable; NO: disable |
| N12010~ N12012 | BACKLASH X/Y/Z | Set the backlash of axis X/Y/Z. | 0~5 mm |
| N12013 | BACKLASH A | Set the backlash of axis A. | 0~5 deg |

## 4.3.7 Breakpoint Resume

In case of accidental and emergent situations such as power interruption, E-stop, to name but a few, breakpoint resume function can help to restore machining from where it is interrupted on condition that current WCS are accurate and precise.

Click submenu **Breakpoint Resume** under **Operate** menu, and the system will resume machining from the position where last time machining stops.

# 5 Troubleshooting

## 5.1 Troubleshooting for Common Problems

### 5.1.1 Troubleshooting for 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 in Fig. 5-1:**

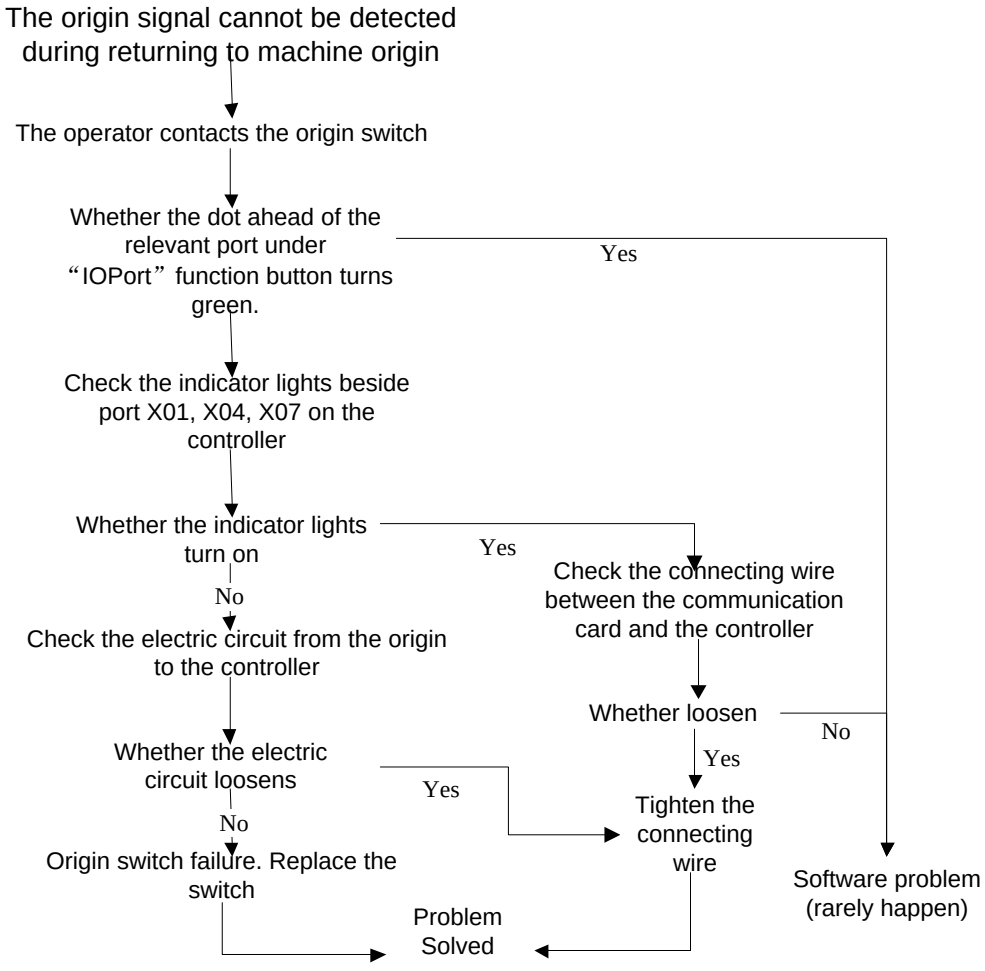



Fig. 5-1 Detection Steps

◆ **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 N74020, “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 N74030, “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.

## 5.1.2 Other Common Problems

◆ **The machine air cylinder does not plunge cut after the button PlungeCut is clicked on the software interface.**

1） In auto mode, check whether the button **PlungeCu**t turns green after being clicked. And then check whether the red dot ahead of output port “Analog Enable” which is under **DIAG** -**IOport** operation button turns green and outputs. Then check whether the output Y02 on the terminal board outputs. If the port outputs, then the software is proved connected with the hardware.

2） Check whether the wiring between the proportional electro-magnetic valve and the output port “analog enable” in the machine tool is correct or not, as shown in Fig. 5-2; If the wiring is correct, then use the multimeter to detect ends of port “Y02” and “GND”, or the analog voltage between “proportional valve signal line” and “proportional valve”.

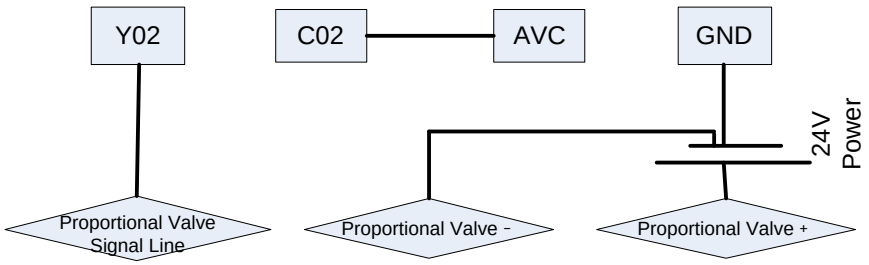



Fig. 5-2 The Wiring between Proportional Electromagnetic Valve and Port Y02

3) Check whether the wiring between the proportional valve and the terminal block loosens. If the wiring loosens, reconnect it.

◆ **A axis rotates wrong during machining tool path.**

1) Modify the value of parameter N83013 “TURNING_ANGLE”. Suggested value is 60;

2) If the angle of the adjacent two lines is about 60° by visual measurement, and the A axis still rotates wrongly, then you should modify the value of the parameter to less than 59° . The boundary angle value of the adjacent lines is vulnerable to errors, for example, the actual angle of the adjacent lines

is 59.99 while it is shown as 60° in the software. Therefore, it is necessary to continually modify the value of the parameter until the problem is solved.

◆ **The machine tool shakes during machining of ellipse.**

When you set the value of parameter N64100, “The Maximum Linear Acceleration of Each Axis When Machining” as a too great value, such as 160000mm/min, the machine tool will shake during machining of ellipse (about 200*150mm). To solve the shake problem, you should set as follows.

1) Set the value of parameter N64120, “The Maximum Resultant Feed Acceleration of Adjacent Two Axes” as twice to five times that of parameter N64100.
2) Increase the value of parameter N64208, “The Maximal Velocity of Circle Motion” to a greater value, such as 5000mm/min or 10000mm/min.
3) Set the value of parameter N64207, “Whether to Limit the Arc-motion Velocity” as “No”.

# 5.2 Warning Information

| Type | Content | Cause | Solution |
|---|---|---|---|
| Warning message | Simulation results show that program range exceeds the machine travel limit. | 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.<br>Check the tool path program file.<br>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) 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 | Check if limit switch works |

| Type | Content | Cause | Solution |
|---|---|---|---|
| | | switch itself. | normally. |
| Servo alarm | Servo alarm of X (Y\Z) 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 button is pressed. | 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. |
| 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;<br>Polarity of port is wrong. Invert the port polarity and restart the software;<br>Analyze possible causes according to the status of indicator “SYSTEM” on Lambda controller;<br>Change another Lambda controller. |

# 6 Appendix

## 6.1 Basic Concepts of NcStudio

### 6.1.1 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 required step during startup. And reference point mode is the default operational mode after system startup, in convenience of related operation execution.

**AUTO Mode**
In auto operation mode, the machine tool generates motion according to the pre-loaded processing program. Therefore, processing programs must be pre-loaded under auto operation mode.

**Manual Mode**
In manual mode, the user 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 users press the manual button, the machine tool move consecutively until the buttons are released.
2) In stepping mode, when users press down the manual button, the machine tool will move a specific distance, known as the step size. The step size has four stages, 0.01mm, 0.1mm, 1mm and 10mm. The user can control the displacement of the machine tool precisely by freely defining the step length in stepping mode.

### 6.1.2 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 the user implements [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. The user 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.

## 6.1.3 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)**

In programming, programmers select a given point on the workpiece as the origin (i.e. the program origin) to establish a new coordinate system, called workpiece coordinate system, which also abides by right-hand rule. The origin of WCS (i.e. the workpiece origin or the work zero) is fixed with respect to a certain point of the workpiece, while probably floating with respect to machine origin (home or the machine zero). The choice of workpiece origin should facilitate simple programming, easy dimension conversion and small machining error to the greatest extent.

# 6.2 Related Dimension Drawing

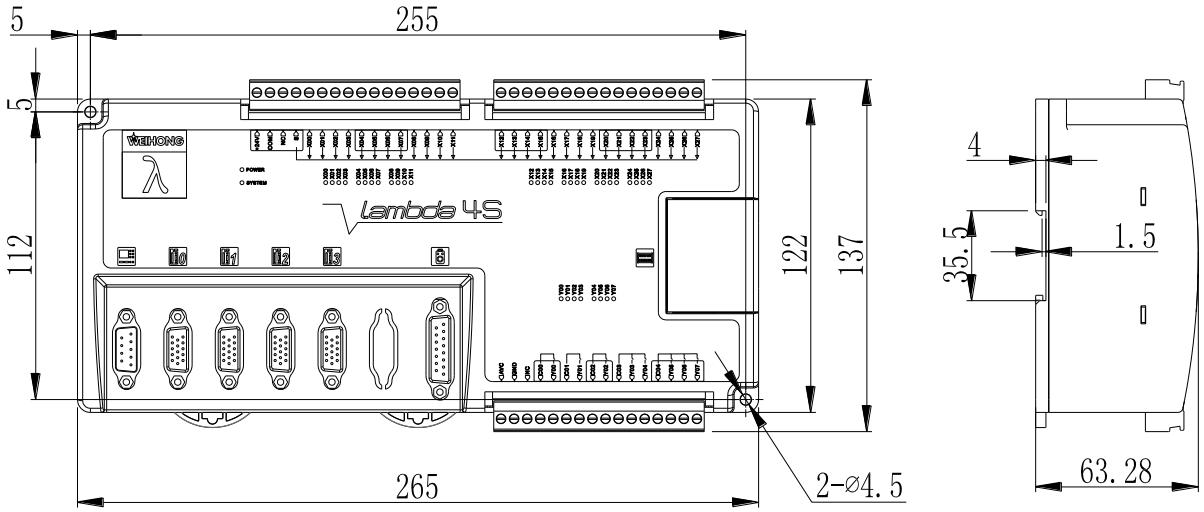



Fig. 6-1 Dimension Drawing for Lambda 4S Controller

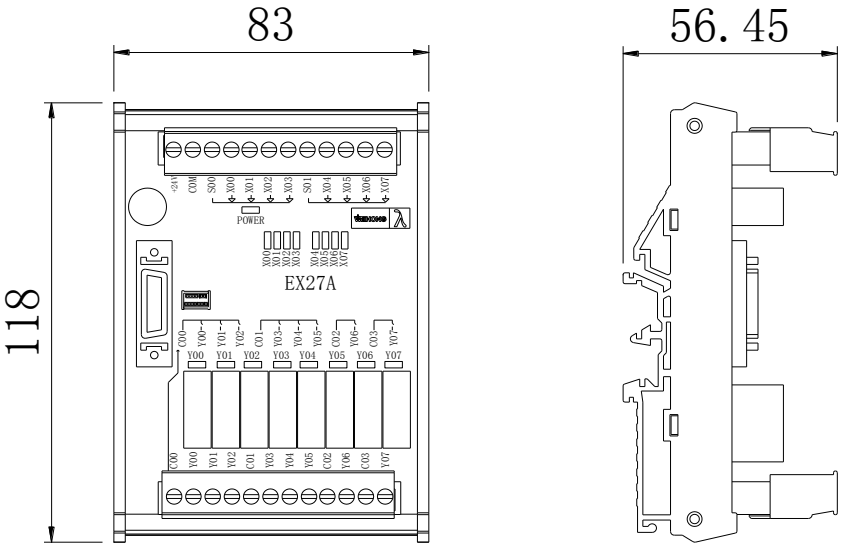



Fig. 6-2 Dimension Drawing for EX27A

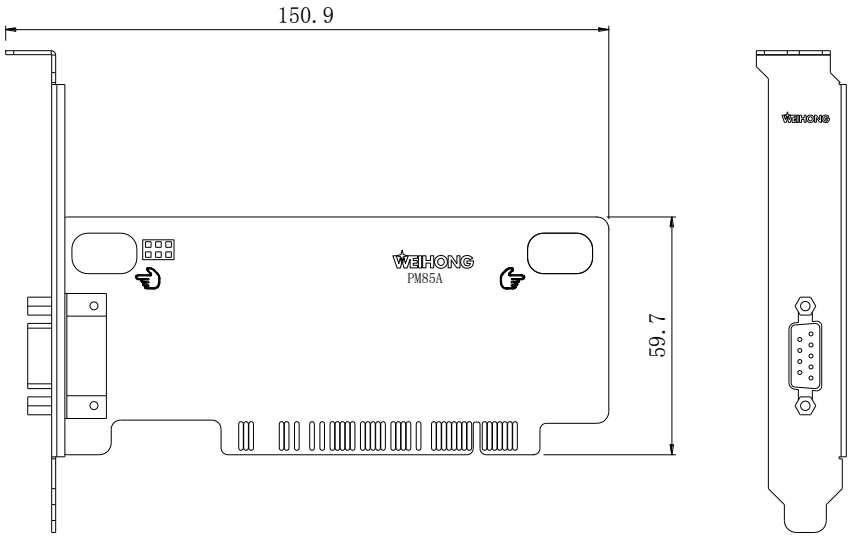



Fig. 6-3 Dimension Drawing for PM85A

# 6.3 System Parameters

According to the authority of the parameters, the system parameters can be classified into operator parameters, manufacturer parameters and developer parameters.

This chapter only gives an introduction to operator authorized parameters. Manufacturer parameters can be accessed only with manufacturer passwords. And developer parameters are for internal use, and can be accessed only with developer passwords.

◆ **How to Modify the Parameters**

Click on tab **PARAM** and then the sub tab **Param**. And then click All Param button in the lower left of the interface. Double click on the operator parameter you are going to modify. Modify the value of the parameter or choose “Yes” or “No” in the dialog box popping up.

◆ **List of [Operator] Parameters**

| Paramter | Name | Setting Range | Default | Effecting time |
|---|---|---|---|---|
| **6.2 G code options** | | | | |
| N62020 | IJK_INC | Yes: Valid;<br>No: Invalid | Yes | Become effective at once |
| **6.4 Speed/Arc** | | | | |
| N64020 | RAPID_TRAVEL _FEEDRATE | 0.001~360000 mm/min | 120000 | Become effective at once |
| N64040 | DEFAULT_FEEDRATE | 0.001~360000 mm/min | 120000 | Become effective at once |
| N64060 | MAX_REV | 0.001~99999 rpm | 200 | Become effective at once |

| Paramter | Name | Setting Range | Default | Effecting time |
|---|---|---|---|---|
| 6.5 File translation | | | | |
| N65000 | Convert File To Nce | 0: Not Convert;<br>1: Prompt;<br>2: Convert | 2 | Become effective at once |
| **7.1Manu** | | | | |
| N71000 | JOG_VOL_XY | 0.001~360000 mm/min | 20000 | Become effective at once |
| N71001 | RAPID_JOG_VOL_XY | 0.001~360000 mm/min | 30000 | Become effective at once |
| N71002 | REF_JOG_VOL | 0.001~360000 rpm | 30 | Become effective at once |
| N71003 | REV_RAPID_JOG_VOL | 0.001~360000 rpm | 100 | Become effective at once |
| N71004 | MANU_ACC_ACC | 0.001~100000 $mm/s^2$ | 2000 | Become effective at once |
| N71023 | JOG_VOL_Z | 0.001~360000 mm/min | 40000 | Become effective at once |
| N71024 | RAPID_JOG_VOL_Z | 0.001~360000 mm/min | 50000 | Become effective at once |
| **7.2 Auto** | | | | |
| N72001 | SPEED_ASSIGN_TYPE | 0: Use file;<br>1: use default speed | 1 | Become effective at once |
| N72006 | HEIGHT_Z_RAISE | -9999999~9999999 mm | 0 | Become effective at once |
| **7.3 Pause** | | | | |
| N73000 | PAUSE_DOWN_VOL | 0.001~360000 mm/min | 600 | Become effective at once |
| N73001 | PAUSE_UP_VOL | 0.001~360000 mm/min | 600 | Become effective at once |
| N73002 | PAUSE_OPTION | 0: lift up a specifies value;<br>1: go to a specified position in workpiece coordinate;<br>2: goto specified position in mach. Coordinate. | 2 | Become effective at once |
| N73003 | Z_WPCOOR_ON_PAUSE | 0~9999 | 0 | Become effective at once |
| N73004 | Z_OFFSET_ON_PAUSE | 0~500 mm | 10 | Become |

| Paramter | Name | Setting Range | Default | Effecting time |
|---|---|---|---|---|
| | | | | effective at once |
| N73006 | Z_MACHCOOR_ON PAUSE | -350~350 mm | 0 | Become effective at once |
| **7.4 Bkref** | | | | |
| N74001 | NEED_REFPT_BEFORE _MACHINING | Yes: back to the machine origin before machining;<br>No: not back to the machine origin before machining | Yes | Become effective at once |
| N74006 | MACHINE_LOCk | Yes: The machine needs to be unlocked to normal use;<br>No: Normal use machine | No | Become effective at once |
| N74007 | SHOW_BKREF_DLG_TIP | Yes: show tips;<br>No: hide tips | No | Become effective at once |
| **7.9 Operation others** | | | | |
| N79000 | Z_DOWN_VELO_OPTION | 0:not disposed;<br>1:direct-Z-motion only;<br>2:general Z-down motion | 0 | Become effective at once |
| N79001 | Z_DOWN_VELO | 0.001~360000 mm/min | 480 | Become effective at once |
| N79003 | SAFE_HEIGHT | 0~1000 mm | 0 | Become effective at once |
| N79005 | SafeToolRaisingFeedrate | 0.001~360000 mm/min | 300 | Become effective at once |
| N79100 | ActionAfterProgramming | 0: Do not Move;<br>1: Back to fix point;<br>2:Back to workpiece origin;<br>3: Back to set point | 2 | Become effective at once |
| N79110 | FIXPT_POS X | -99999~99999 mm | 0 | Become effective at once |
| N79111 | FIXPT_POS Y | -99999~99999 mm | 0 | Become effective at once |
| **8.3 Glass Cutter Parameter** | | | | |
| N83009 | Parrallel A Unchanged | Yes: unchanged;<br>No: changed | No | Become effective at once |
| N83018 | ApproximationLength | 0.001~99 mm | 0.005 | Load Program |
| N83115 | AirPumpDelay | 0~600000 ms | 1000 | Become effective at once |

| Paramter | Name | Setting Range | Default | Effecting time |
|---|---|---|---|---|
| **8.4 Glass Cutter Edge finding Parameter** | | | | |
| N84000 | UpCylinderDelay | 0~600000 ms | 2000 | Become effective at once |
| N84008 | WaitGlassDelay | 0~100000 ms | 2000 | Become effective at once |
| N84023 | Workpiece_is_automatically_transmitted | Whether Workpiece is automatically transmitted after finishing machining Yes; No | No | Become effective at once |
| N84024 | MigrationGlassDelay | 0~100000 ms | 3000 | Become effective at once |
| N84027 | JourneyDelay | 0~100000 ms | 2000 | Become effective at once |
| N84040 | With or Without A | Yes: with A; No: without A | No | Reboot |
| N84041 | Delay Start Glass Loading Machine | 0~100000 ms | 3000 | Become effective at once |
| N84050 | Passer Glass Position X | -100000~100000 mm | 0 | Become effective at once |
| N84051 | Passer Glass Position Y | -100000~100000 mm | 0 | Become effective at once |
| **8.6 Diagnose View** | | | | |
| N86006 | Simulation Control Flag | Whether Simulation machine button is available during run time 0:No; 1:Yes | 0 | Become effective at once |
| **8.7 Show View** | | | | |
| N87002 | Pressure Control | Whether to need Pressure control when machining Yes; No | Yes | Become effective at once |

# 6.4 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.

**Intellectual Property Rights Notice**

The Software Product and all intellectual property rights therein (including but not limited to any all copyrights, patents, trademarks, and publicity rights) are owned by Weihong Company. The Software Product is protected for Weihong Company on the basis of copyright law and international treaty provisions as well as on the basis of other laws and agreements regarding intellectual property. You are not allowed to remove the copyright statement made in the Software Product, and guarantee that you shall copy the copyright statement in all copies of the Software Product or of any part of the Software Product. You are obliged to stop any form of illegal copying of the Software Product and accompanying materials.

**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.