

NcStudio V10 Milling CNC System

Users' Manual

3rd Edition
(For four and five axes)

Weihong Electronic Technology Co., Ltd.

The copyright of this manual belongs to Weihong Electronic Technology Co., Ltd. (hereinafter referred to as Weihong Company). This manual and any image, table, data or other information contained in this manual may not be reproduced, transferred, or translated without any prior written permission of Weihong Company.

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

Preface

About This Manual

This manual is intended for end-users or operators of machine tools. If you use the CNC system for the first time, you need to read through the manual. If you are experienced with the system, you can search for the desired information via the contents.

With 7 chapters, this manual can be divided into 6 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 carefully beforehand to ensure safe operations.
- 2) Part 2: Chapter 1, introducing the software features, the coordinate system, operation modes and operation status.
- 3) Part 3: Chapter 2, introducing software setup and manual update of the driver, as well as installations and connections of hardware. It helps users to properly install the software and start it for the first time.
- 4) Part 4: Chapter 3~5, introducing the operational interfaces, operation procedures, parameter setting and functionalities of the software. It helps users to have a good knowledge of the software, its operational commands, and how-to-use.
- 5) Part 5: Chapter 6, introducing possible problems and situation in real practice, and treatment to different alarms. It helps users to effectively react to the possible trouble and take proper measures to fix it instantly.
- 6) Part 5: Chapter 7, introducing shortcut keys and the software license agreement.

Applicable Product Models

This manual is applicable to four & five axes software of the NcStudio V10 milling CNC system. Refer to the table below for details:

Product Model	Remarks
NcStudio V10 Milling CNC System (For four & five axes)	<ol style="list-style-type: none"> 1) Used together with PM85A/95A communication cards and Lambda 4S/5S controllers. Multiple axes software of the NcStudio V10 is used to control engraving and milling machines with four & five axes, which can be used in complex molding, advertising, decorating and wood cylindrical engraving industries. 2) Lambda 4S/5S controllers can be abbreviated as Lambda controller or the controller.

Contact Us

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

Company Name: Weihong Electronic Technology Co., Ltd.
Headquarters Address: No.1590, Huhang Rd., Fengxian, Shanghai, PRC 201400
Tel: +86-21-33587550
Fax: +86-21-33587519
Website: [http: www.weihong.com.cn/en/](http://www.weihong.com.cn/en/)

Revision History

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

Date	Edition	Revision
2016.12	R3	<ol style="list-style-type: none">1) Software setup steps updated (refer to Section 2.2.1);2) User interface, menus, and operation procedures updated (refer to Chapter 3);3) The parameters related to tool measurement updated (refer to Section 4.2);4) Software configuration updated (refer to Section 4.3.2);5) Machining wizard and program file updated (refer to Section 4.4);6) Simulation and track function updated (refer to Section 4.6);7) The functions, such as wheel heel operation, log, registration, language switch, system maintenance updated (refer to Section 4.7, 4.8, 4.9, 4.10.1, 4.11, 4.12);8) Tool-up and tool-down functions for five-axis XYZAC added (refer to Section 4.13);9) Parameters updated (refer to Chapter 5).
2016.03	R2	Contact information updated.
2015.06	R1	This edition is released for the first time.

Precautions

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



: 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 even this kind of precautions could cause physical injury or machine damage.



: 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

 **WARNING**

for plugging cable connected with the device and the ventilation & heat dissipation in the cabinet;

- Space between this device and other equipment should also be preserved according to the requirements;
- The product should be installed firmly and without vibration. During installing, casting, knocking, striking, or loading on the product is forbidden;
- To reduce electromagnetic interference, power-supply components used should be above AC or DC 50V and the space between cable and CNC device should be preserved above 100mm;
- It will be better if CNC device is installed at a position facilitating debugging and maintenance.

3) Precautions Related to Wiring

- Only qualified people are allowed to participate in the wiring and checking;
- The CNC device should be grounded reliably and grounding resistance should be less than 4 ohm. Neutral line is absolutely not allowed to replace earth wire. Otherwise, it may result in malfunction of the device due to the interference;
- Wiring should be firm and steady, or disoperation 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	INTRODUCTION.....	1
1.1	Operation Mode.....	1
1.2	Operation State.....	2
1.3	Coordinate System.....	2
2	SYSTEM INSTALLATION AND CONNECTION	4
2.1	Host Computer Requirements.....	4
2.2	System Installation, Connection and Uninstallation.....	4
2.2.1	Software Setup	4
2.2.2	Hardware Installation and Connection.....	7
2.2.3	Manually Update Hardware Driver.....	8
2.2.4	Uninstall NcStudio.....	11
3	SOFTWARE OPERATION.....	12
3.1	Software Interface and Menu	12
3.1.1	Holistic Interface	12
3.1.2	Overall Menu List.....	14
3.2	Operation Step.....	16
3.2.1	Choose Configuration	16
3.2.2	Check Polarity of IO Ports.....	17
3.2.3	Manual Operation	19
3.2.4	Reset Machine	20
3.2.5	Load Program File	20
3.2.6	Set Workpiece Origin	21
3.2.7	Set Speed-related Parameters	21
3.2.8	Execute Auto Machining	21
4	SOFTWARE FUNCTIONS.....	23
4.1	Axis Control	23
4.2	Tool Measurement.....	23
4.2.1	Fixed Calibration	24
4.2.2	Mobile Calibration	25
4.2.3	Special Measurement	27
4.3	Offset Management	30
4.3.1	Workpiece Coordinate System (WCS)	30

4.3.2	Offset Setting	31
4.4	Program Management.....	34
4.4.1	Program Wizard	34
4.4.2	Program Files.....	34
4.5	Speed Management	36
4.5.1	Spindle Speed, Feedrate and G00 Speed	36
4.5.2	Jog Feedrate/ Rapid Jog Feedrate	37
4.5.3	Acceleration	38
4.5.4	Reference Circle Speed.....	39
4.6	Simulation and Track	40
4.6.1	Simulation	40
4.6.2	Track	41
4.7	Handwheel Operation.....	44
4.8	Log.....	44
4.9	Register	45
4.10	System Language and Theme	46
4.10.1	Language Switch-over	46
4.10.2	Theme Selection	47
4.11	System Maintenance	47
4.12	Auxiliary Function	48
4.13	Tool Lifting and Plunging for Five Axes	53
5	PARAMETER SETTING	55
6	PRECAUTIONS AND TROUBLESHOOTING	62
6.1	Common Troubleshooting	62
6.1.1	What should users do if the spindle does not rotate?.....	62
6.1.2	What should users do if an axis does not move?	62
6.1.3	What should users do if servo motor Z brake can't be opened?	62
6.1.4	What should users do if machine tool returns to the machine origin abnormally?	63
6.1.5	What should users do if the machine tool motions upward after arriving at the position of tool sensor during calibration?	64
6.2	Alarm Information	64
7	APPENDIX.....	66
7.1	Shortcut Keys List.....	66
7.2	Software License Agreement	68

1 Introduction


NcStudio V10 milling CNC system for four and five axes, which is independently developed and possessed by Shanghai Weihong Electronic Technology Co., Ltd. supports G code, PLT code, ENG code and DXF code generated by multiple CAD/CAM software such as UG, Mastercam, CASmate, ArtCAM, AutoCAD and CorelDraw. In addition, NcStudio V10 designed on the basis of PC operation system is easy to learn and operate.

This system should be used together with PM85A/PM95A communication card and Lambda 4S/5S controller, and the matched software is multiple axes software of NcStudio V10. For the detailed information of PM85A/95A communication card and Lambda 4S/5S controller, please refer to *Lambda CNC System Manufacturer's Manual*.

1.1 Operation Mode

AUTO Mode: In auto operation mode, the machine tool generates motion according to the pre-prepared processing program.

Manual mode: In manual mode, the user can directly control the motion of the machine tool via manual operation equipment, such as computer keyboard, handheld box, and MPG. Manual mode can be further divided into three subdivision modes: jog, stepping (also called incremental mode) and handwheel.

- **Jog mode:** In jog mode, the user can directly control the motion of the machine tool via manual operation equipment, such as computer keyboard, handheld box, and MPG. When the user sends out motion signal with the help of that equipment, the machine tool will continue moving until the pressed button is released. For example, clicking the button  on software interface will make machine tool consecutively move in X-axis negative direction until this button is released.
- **Stepping mode:** In stepping mode, the user can also use manual operation equipment, such as computer keyboard, handheld box and MPG to control the machine tool. However, different from the jog mode, when the user clicks a button once (from clicking the button to releasing it), the machine tool only moves a specific distance (known as the step-size as well). In this way, the user can control the displacement of machine tool precisely.
- **Handwheel mode:** Select "Handwheel" in the software interface to activate the handwheel mode. With enabled, the connected handwheel or MPG can be used to control the motion of machine tool.

Reference Point Mode (Homing): Returning to the machine origin is a required step during startup. In addition, reference point mode is the default operational mode after system startup, in convenience of related operation.

1.2 Operation State

IDLE: Under this state, the machine tool does not generate motion, but is ready for any new task.

E-STOP: This is an abnormal state. Under this state, the machine tool is locked and incapable of moving. 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. 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 starts to generate any motion, the system enters into this state.

PAUSE: When the machine tool is running, if the user implements “Pause” order, or the system parses a M01 command (Wait Command), the system will enter into PAUSE state and wait for the next instruction. At this time, the user can implement “Start” 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.

1.3 Coordinate System

Coordinate system is a terminology describing the motion of a machine tool. For the sake of unification, standard coordinate system adopts right-hand rule, as illustrated in Fig. 1-1:

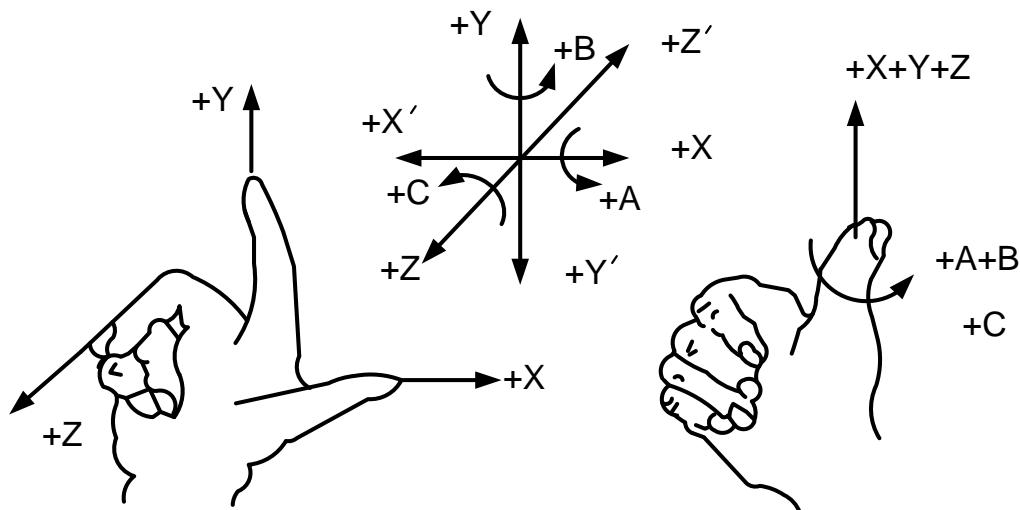


Fig. 1-1 Coordinate system following right-hand rule

For a milling machine, the axes directions of machine coordinate system are decided by both the type of 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 coincides with spindle axis, and the direction of the cutter moving away from workpiece is its positive direction (+Z).

—X-axis is 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).

— A-axis rotates around X-axis, B-axis rotates around Y-axis and C-axis around Z-axis. Counter-clockwise direction is specified as the positive direction of A/B/C axes.

- **Machine Coordinate System (MCS)**

Machine coordinate system is a 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 function of going back to machine reference point. 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 origin (the program origin) to establish a new coordinate system, called workpiece coordinate system, which also abides by right-hand rule. The origin of WCS (namely the workpiece origin or the work zero) is fixed with respect to a certain point on the workpiece, while probably floating with respect to machine origin (home or the machine zero). The choice of workpiece origin should facilitate programming, dimension conversion and minimizing machining error to the greatest extent.

2 System Installation and Connection

2.1 Host Computer Requirements

CPU:	basic frequency 1G or above
Memory:	above 512M
Hard disk:	above 20G
Display adapter:	1024*768 at least
Display:	above 14" VGA
CD-ROM:	4X or higher (optional)
Main board extension slot:	1 PCI/PCIE slot or above


2.2 System Installation, Connection and Uninstallation

If an old version of NcStudio has been installed in your computer, please delete it first before new system setup. Please refer to Section 2.2.4 for corresponding operations.

NcStudio system consists of software and hardware; therefore, system installation covers two parts, namely, software setup, the installation and connection of hardware (mainly communication card and Lambda controller). It is recommended to install the software before installing communication card and Lambda controller.

2.2.1 Software Setup

Please install the software as following steps:

1. Power on the computer, and then insert installation CD. Double click the icon  in "My Computer" to start installing. If this is your first time to install the software, please directly skip to step 4. If not, a dialog box as shown in Fig. 2-1 will pop up.

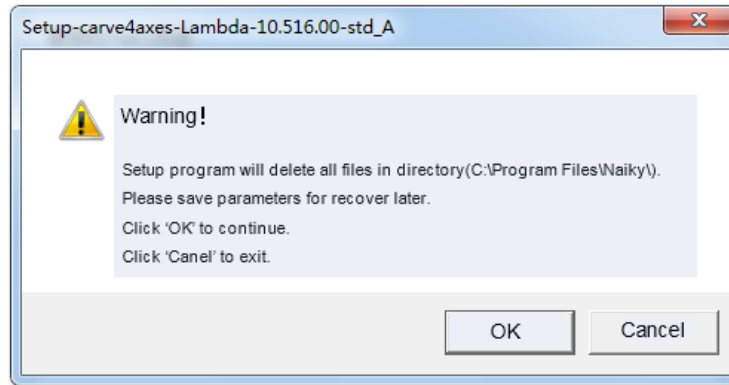


Fig. 2-1 Warning to delete previous files

- Click "OK", and a dialog box as shown in Fig. 2-2 will pop up. If any old version software has been installed in this PC, the system will prompt to save its parameter settings. If you choose to save the previous parameter settings here, you can apply all of them to the current software.

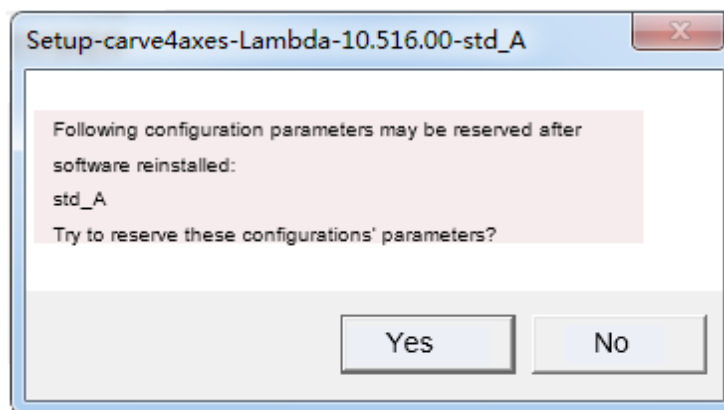


Fig. 2-2 Prompt to reserve parameter settings

- Choose "Yes" or "No" to go ahead.
- Installation begins. The NcStudio system will be installed in directory C:\Program Files\Naiky by default. Progressing picture is as shown in Fig. 2-3 Installing Fig. 2-3.

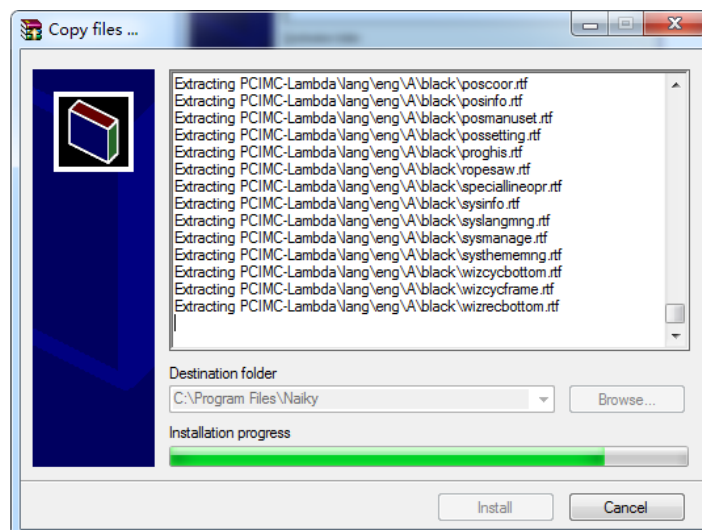


Fig. 2-3 Installing

5. During the installation period, the system will automatically update driver program, as shown in Fig. 2-4. If the auto update failed, you need to update it manually. See Section 2.2.3 for details

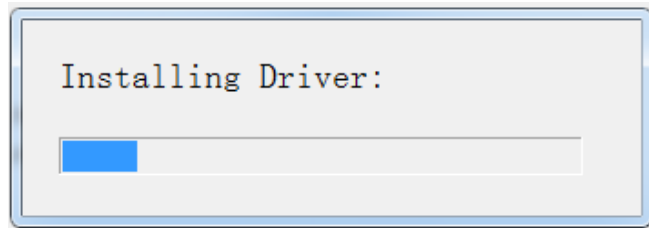


Fig. 2-4 Installing driver

Software installation completes. A dialog box as shown in

6. Fig. 2-5 will pop up. Click “OK” to confirm.

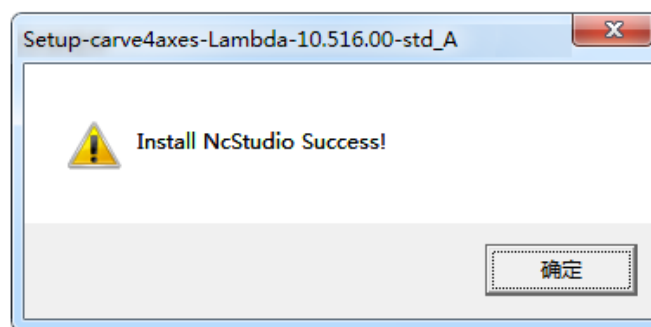


Fig. 2-5 Installation completed



If a dialog box as shown in Fig. 2-6 pops up in installation process, the installation of communication card failed. Refer to the following solutions to resolve this problem.

1. Select “Yes” → Install the communication card → Install the software, and then the system will automatically update driver.
2. Select “No”, and the installation of the software finished. At this moment, the user needs to reinstall the communication card and then manually update driver.

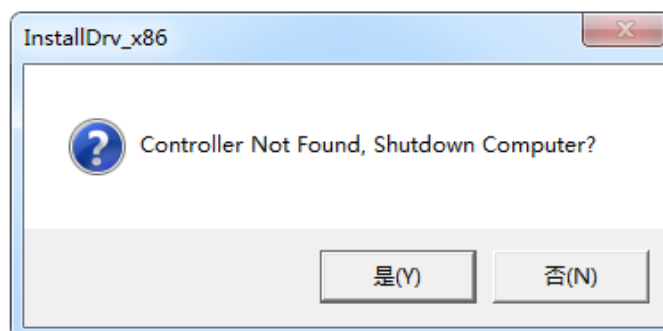


Fig. 2-6 Controller not found

2.2.2 Hardware Installation and Connection

● Hardware Installation

After software installation is completed, power off the computer and install the communication card, the controller and other peripheral devices as following steps:

1. Open the chassis cover, and insert the card into an available and well-matched expansion slot (the PCI slot for PM85A card and PCIE slot for PM95A card). During installation, slightly hold the two sides of the card with your hands to secure that it is inserted into slot firmly and well connected with computer baseboard. Then tighten the screw of control card, and close the lid. The installation of motion control card completes.
2. Similar with the above step, please insert the connection parts of controller and external devices or periphery equipment into its available and well-matched slot.
3. Installation is completed. Please restart the computer.

● Hardware Connection

The following is an overall wiring diagram of the communication card, the controller and periphery equipment:

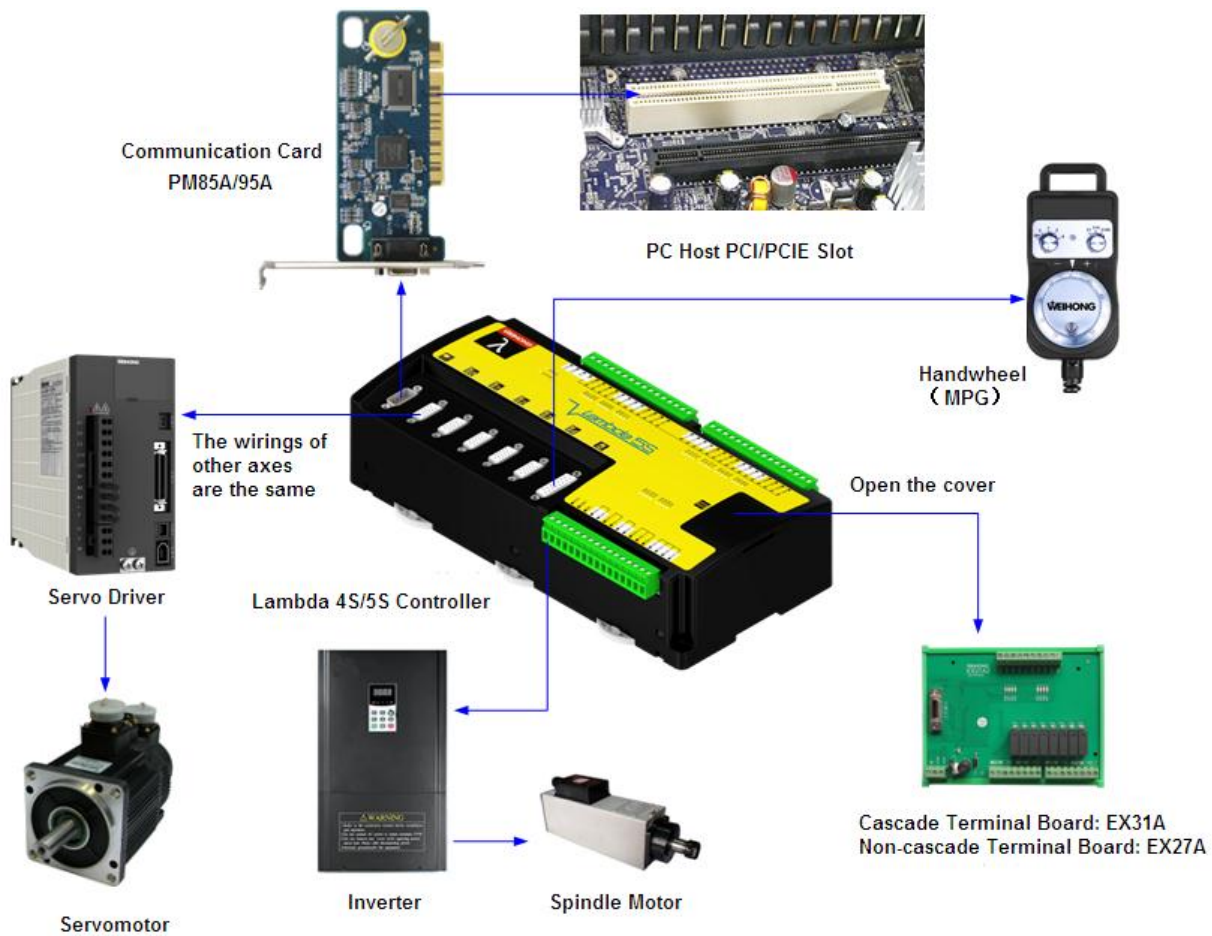


Fig. 2-7 Overall wiring diagram



Please connect according to actual situations. Above picture is provided for reference only.

2.2.3 Manually Update Hardware Driver

1. Right click My Computer, select “Properties”, and then click “Device Manager”. Choose “CNC Adaptor” item, right click on it and select “Update Driver Software...” The hardware wizard dialog box will pop up as shown in Fig. 2-8.

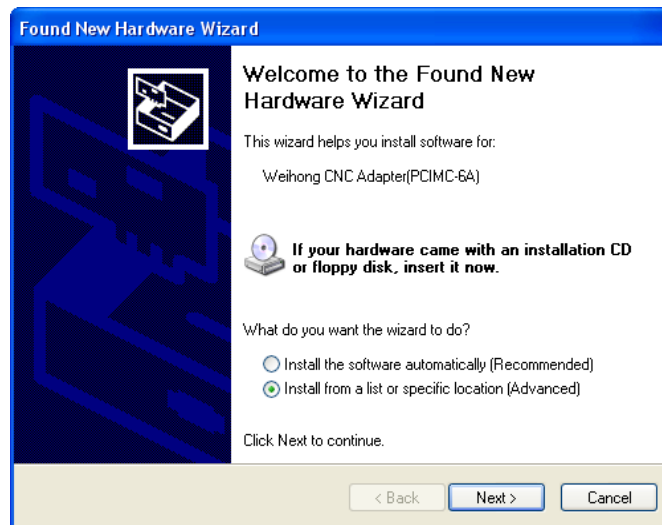


Fig. 2-8 Hardware update interface

2. Select “Install from a list of specific location (Advanced)”, and click “Next” to continue. A dialog box as Fig. 2-9 will pop up.

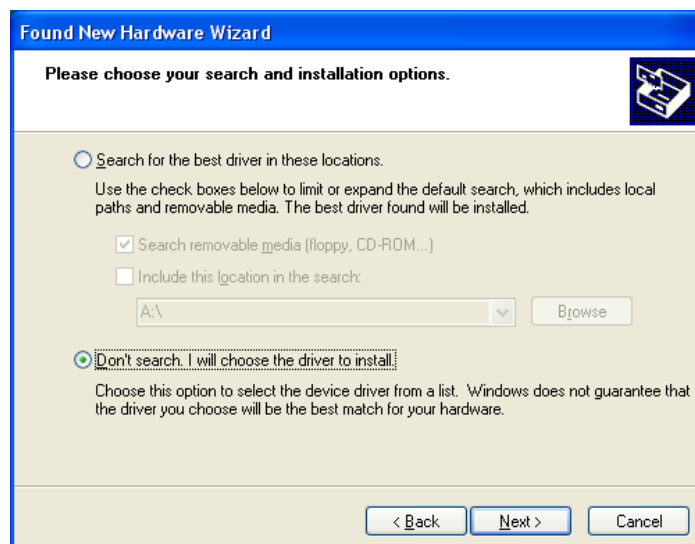


Fig. 2-9 Choose installation method

3. Select “Don’t search, I will choose the driver to install”, and click “Next” to continue. A dialog box as Fig. 2-10 will pop up.

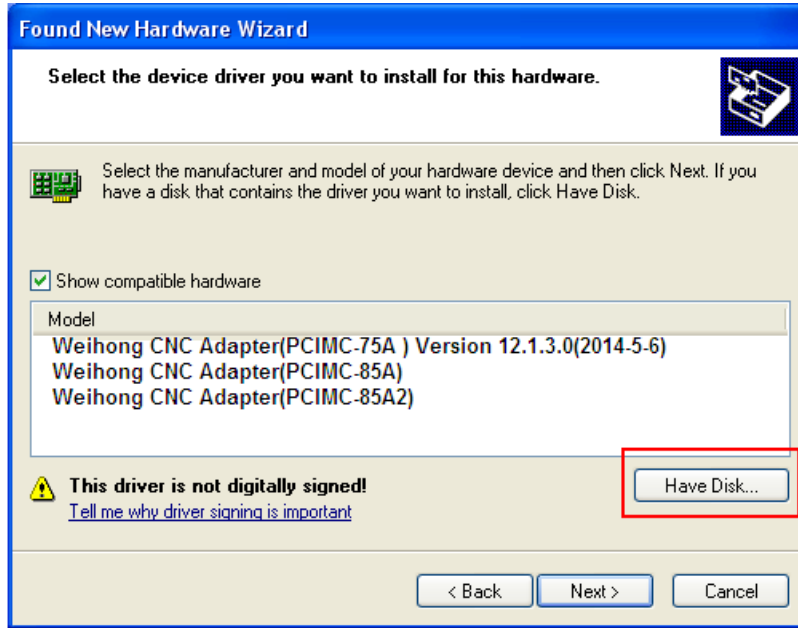


Fig. 2-10 Select the driver manually

4. Click “Have Disk...”, and a dialog box containing compatible hardware will pop up, as shown in Fig. 2-11

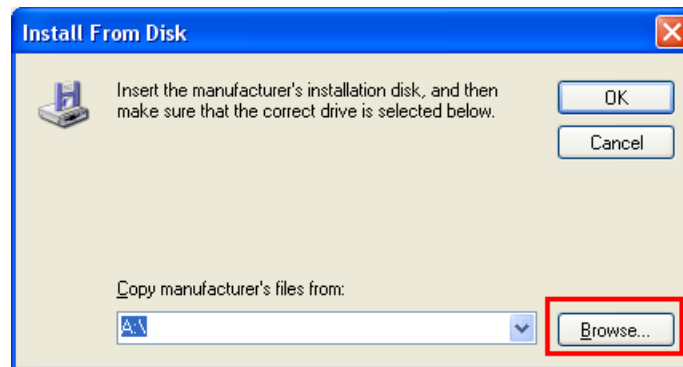


Fig. 2-11 Select the target file of driver

5. Click “Browse...” and choose target file in the dialog box pop up. Taking PM85A as an example, its directory is “C:\Program Files\Naiky\PCIMC-Lambda”, as shown in Fig. 2-12

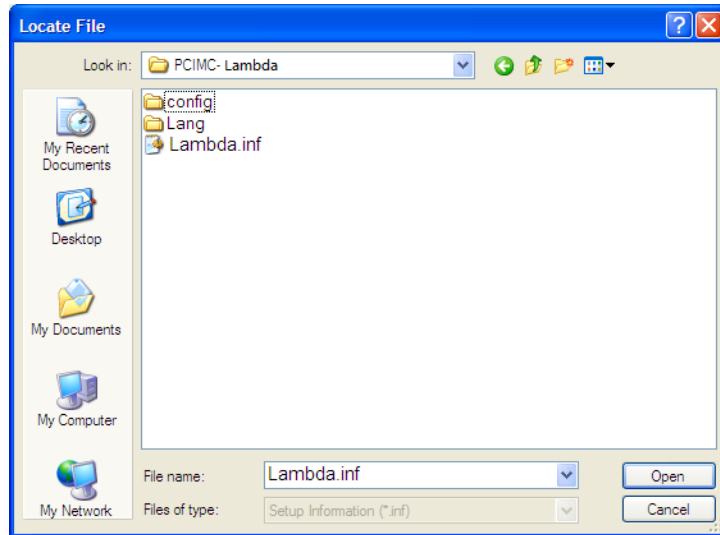


Fig. 2-12 Select the hardware driver

6. After hardware driver being correctly chosen, click “Open” and a dialog box as shown in Fig. 2-13 will pop up.

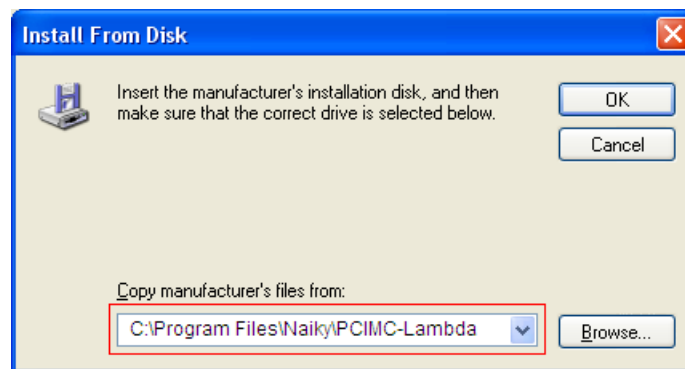


Fig. 2-13 Target file directory confirmation

7. Click “OK”, and the system will go back to the dialog box as shown in Fig. 2-10. Click “Next” to start updating the hardware driver, as shown in Fig. 2-14.

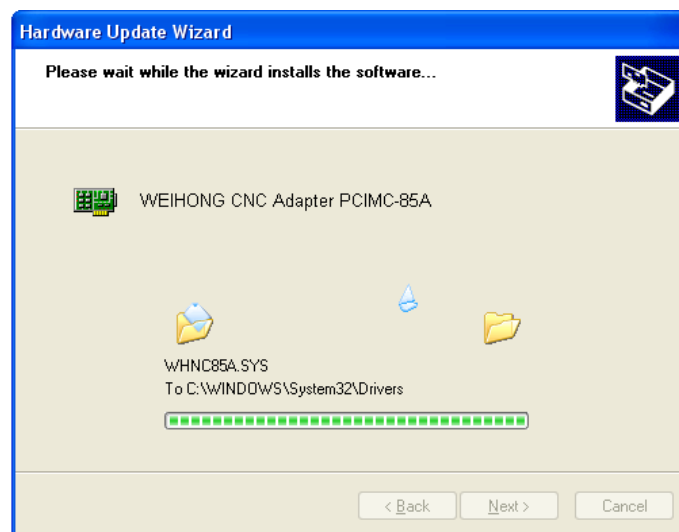


Fig. 2-14 Updating the driver

- When the updating is finished, a dialog as shown in Fig. 2-15 will pop up. Click “Finish” to complete the update of hardware driver.

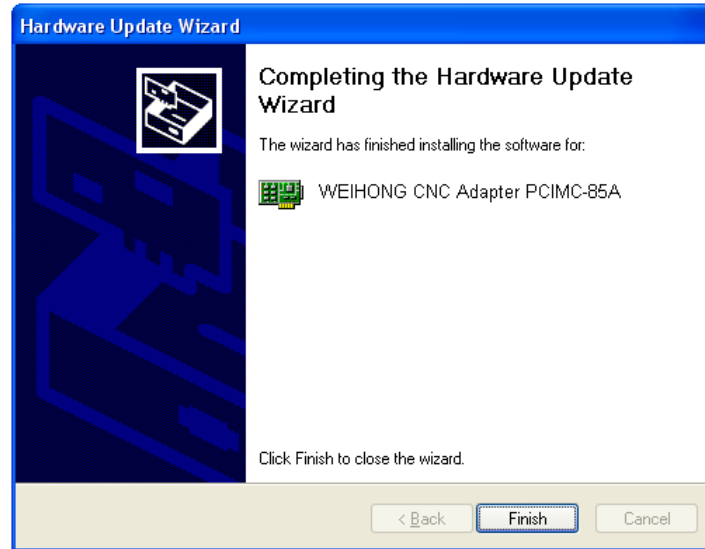


Fig. 2-15 Driver update completed

2.2.4 Uninstall NcStudio

NcStudio is green software. If you want to delete this software, all you need to do is to delete the folder named “Naiky” under directory *C:\Program Files* and delete the NcStudio shortcut icon on the desktop as well as the *Start* menu.

3 Software Operation

In order to help the user know this software quickly, this Chapter focuses on the introduction of software operations. All pictures and information provided in this chapter are taking four axes software as example, if without special notification.

3.1 Software Interface and Menu

3.1.1 Holistic Interface

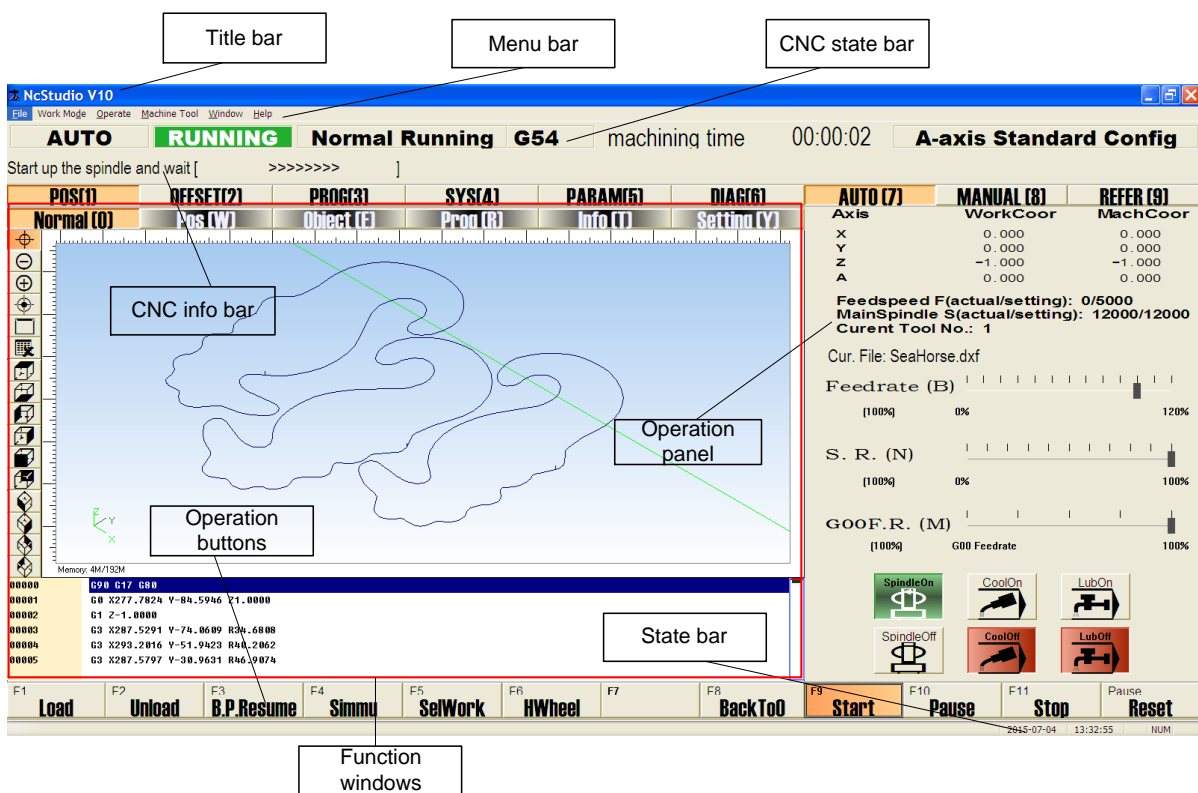


Fig. 3-1 Holistic interface of NcStudio for four axes

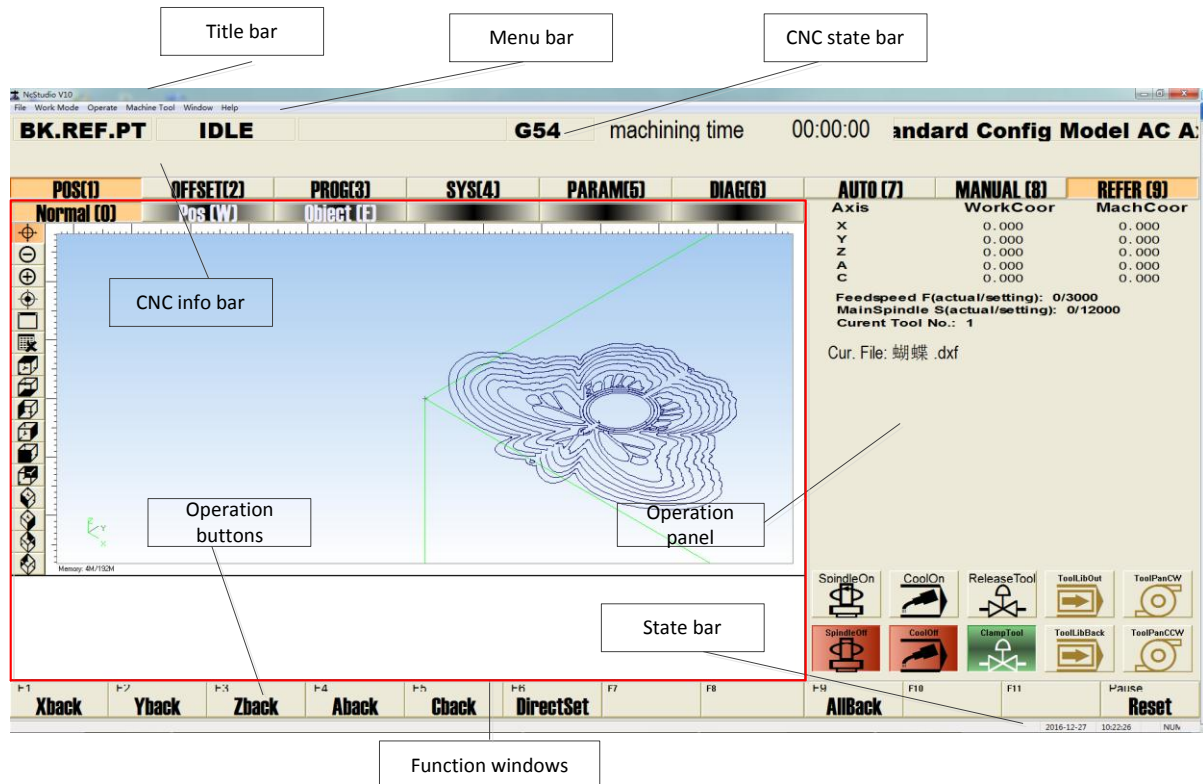


Fig. 3-2 Holistic interface of NcStudio for five axes

Title bar: a bar shows the software name “NcStudio V10”, whose color can tell NcStudio is currently active or inactive.

Menu bar: it contains six main menus “File”, “Work Mode”, “Operate”, “Machine Tool”, “Window” and “Help”, and each menu has several pull-down sub-menu items. Menu selection and functions enabling can be done by mouse or shortcut key. For overall menu items of the system, please turn to Section 3.1.2.

CNC state bar: a bar where current operation mode, operation state, additional information, WCS, machining elapsed time and configuration information are displayed in order.

CNC info bar: three different kinds of prompt information will be displayed, namely, normal prompt, warning prompt and erroneous prompt, and they are corresponding to three different colors respectively, the current interface color, yellow, and red.

Function windows: Six functional interfaces are displayed in this area, including “POS”, “OFFSET”, “PROG”, “SYS”, “PARAM”, “DIAG”, “AUTO” and “MANUAL”, and each contains several secondary functional selection buttons, which may vary because of different system states.

Operation panel: there are three types of panel, corresponding to Auto /Manual /Reference point modes respectively. However, there are some differences among these panels in different modes. Compared with the software for four axes, the software for five axes has several additional functions, such as “Release Tool”, “Clamp Tool”, “Tool Lib Out”, “Tool Lib Back”, “Tool Pan CW” and “Tool Pan CCW”.

3.1.2 Overall Menu List

- Four-axis Software

File	WorkMode	Operate	Mach Tool	Window	Help
Open&Load	Auto Mode	Single Block	Main Spindle	Normal	About NcStudio
Unload	Manu Mode	HW Guide	Lubricate	Position	
New	Jog	Set Work Origin..	Coolant	Object	
Open&Edit..	Handwheel	Save Work Origin..	Light	Set Current Point..	
Edit Current Program	Stepping*0.01	Load Work Origin..	Using Calibrate Area	Offset	
Load Objects	Stepping*0.1	Start	Bkerf point and Limit ports setting	Offset Save&Load	
Stop Loading Objects	Stepping*1	Pause		Harddisk list	
Clear Objects	Stepping*10	Stop		Floppydisk list	
Recent Loaded Files..	Custom Stepping	Simulation Mode		Process Wizard	
Generate Installation	Back to REF Point Mode	Select Block..		History	
Generate Emute Instal..		Breakpoint Resume		Config Management..	
Restart System		Circumrotate Mirror Pro..		Interface	
Reboot System		Back to Work Origin		Language of the Theme	
Close System		Back to Fixed Point		System Info	
Show Desktop		All Back to REF Point		General Param..	
Exit		Reset		Param Backup	
		Graduate Process		Param Auto Backup	
				Log	
				IO Port	
				PLC	

Fig. 3-3 Overall menu list

- Five-axis Software

File	WorkMode	Operate	Mach Tool	Window	Help
Open&Load	Auto Mode	Single Block	Main Spindle	Normal	About NcStudio
Unload	Manu Mode	HW Guide	Lubricate	Position	
New	Jog	Set Work Origin..	Coolant	Object	
Open&Edit..	Handwheel	Save Work Origin..	Using Calibrate Area	Setting	
Edit Current Program	Stepping*0.01	Load Work Origin..	Bkerf point and Limit ports setting	Set Current Point..	
Load Objects	Stepping*0.1	Start		Offset	
Stop Loading Objects	Stepping*1	Pause	Set Tool No...	Offset Save&Load	
Clear Objects	Stepping*10	Stop	Set Disk Magazine Pot No...	Harddisk list	
Recent Loaded Files..	Custom Stepping	Simulation Mode	Return Disk Magazine Origin	Floppydisk list	
Generate Installation	Back to REF Point Mode	Select Block..		Process Wizard	
Generate Emute Instal..		Breakpoint Resume		History	
Restart System		Circumrotate Mirror Pro..		Config Management..	
Shutdown System		Back to Work Origin		Interface	
Reboot System		Back to Fixed Point		Language of the Theme	
Show Desktop		All Back to REF Point		System Info	
Exit		Reset		General Param..	
				Param Backup	
				Param Auto Backup	
				Log	
				IO Port	
				PLC	

Fig. 3-4 Overall menu list

3.2 Operation Step

The software can be normally launched after the software is correctly installed, hardware properly connected and hard driver updated according to the Section 2.2. Load a program file and begin debugging. The basic operations during debugging are as below:

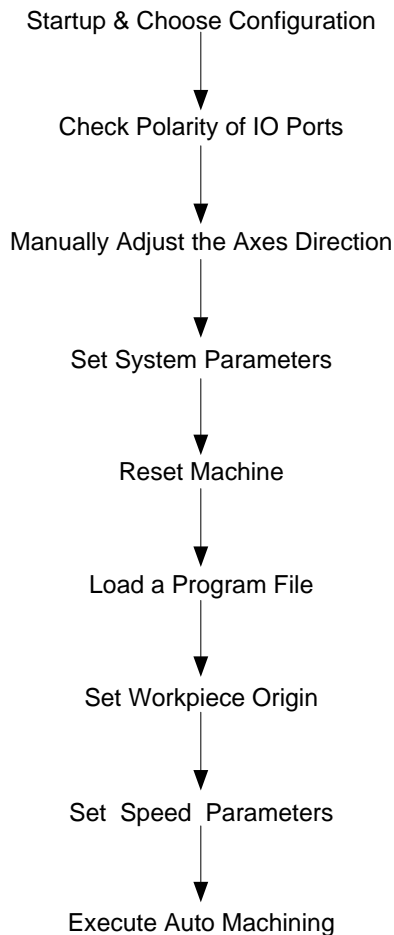


Fig. 3-5 Basic operation steps

3.2.1 Choose Configuration

Before start-up, please make sure that the machine and computer are properly connected. Turn on the



power of the machine and computer, double click the icon **Ncstudio** on the desktop or click it on the “Start” menu list to launch the software NcStudio. If the software is launched for the first time, the user needs to select a configuration according to the machine structure.

There are three ways to access configuration management interface as shown in Fig. 3-6:

1. Choose sub-menu item “Configuration Management...” under “Window” menu.
2. Click “System” functional button and then click secondary button “Configuration Management”.
3. Press NUM key “4” to access “System” functional area, and press alphabet key “Q”.

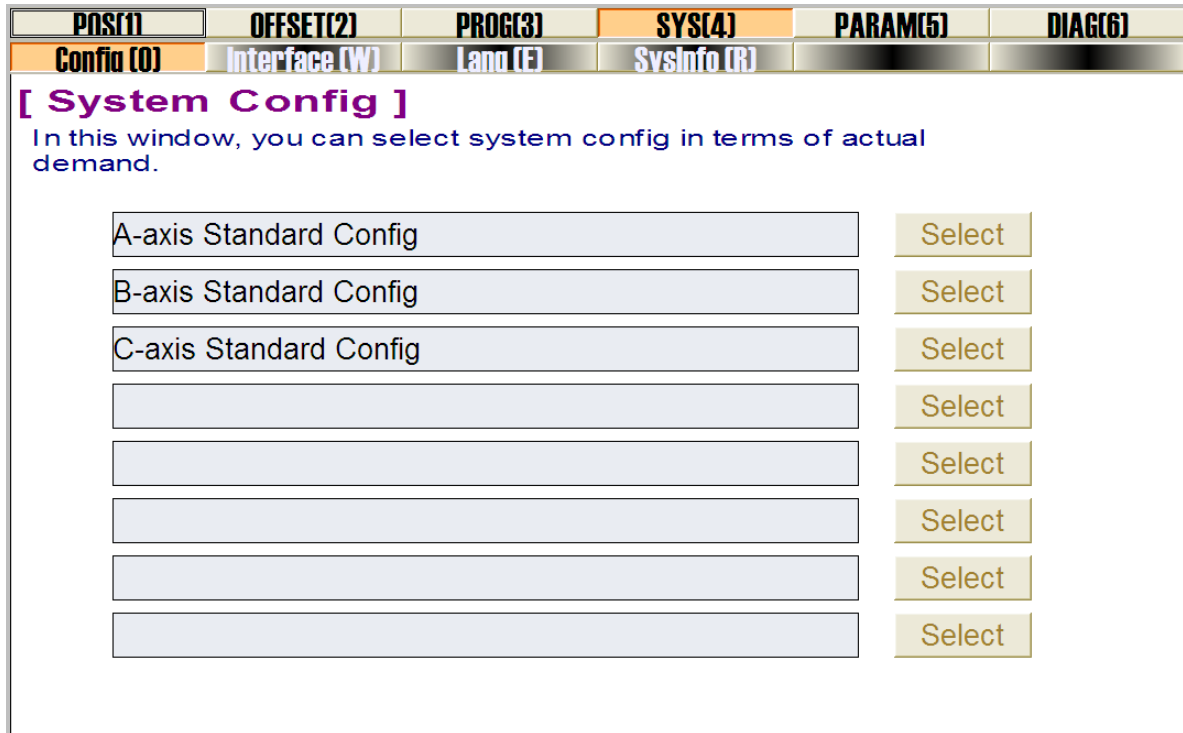


Fig. 3-6 Configuration management window



1. The number and letter next to the functional and sub-functional keys are their corresponding shortcut keys.
2. Number keys on mini NUM keyboard are the shortcut keys corresponding to the axes direction keys on “Manual” operation panel. The number keys on main keyboard are the shortcut keys of functions in functional windows and three operation modes.
3. Generally, configuration is decided by machine tool manufactures, and the end-users cannot change it.

3.2.2 Check Polarity of IO Ports





In “IO Ports” window under “DIAG” functional area, all input and output ports information are displayed, which is used to monitor the system and diagnose malfunctions.

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 prompts may appear.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)																
Lot (0)	IOPort (W)	PIC (E)																			
Tag	Pin	P.. P...	Input ...	Description																	
InPort																					
•XC		N 00...	E,F:4...	Encoder Zero of X-a...																	
•YC		N 00...	E,F:4...	Encoder Zero of Y-axis																	
•ZC		N 00...	E,F:4...	Encoder Zero of Z-axis																	
•AC		N 00...	E,F:4...	Encoder Zero of A-axis																	
•XALM		N 00...	E,F:4...	Axis X Servo Alarm																	
•YALM		N 00...	E,F:4...	Axis Y Servo Alarm																	
•ZALM		N 00...	E,F:4...	Axis Z Servo Alarm																	
•AALM		N 00...	E,F:4...	Axis A Servo Alarm																	
•HX1		N 00...	E,F:4...	Handwheel Ratio X1																	
•HX10		N 00...	E,F:4...	Handwheel Ratio X10																	
•HX100		N 00...	E,F:4...	Handwheel Ratio X1...																	
•HSX		N 00...	E,F:4...	Select X-axis by Han...																	
•HSY		N 00...	E,F:4...	Select Y-axis by Han...																	
•HSZ		N 00...	E,F:4...	Select Z-axis by Han...																	
•HSA		N 00...	E,F:4...	Select A-axis by Han...																	
<table border="1"> <tr> <td>F1</td> <td>F2</td> <td>F3</td> <td>F4</td> <td>F5</td> <td>F6</td> <td>F7</td> <td>F8</td> </tr> <tr> <td>TestOn</td> <td>TestOff</td> <td>CanlTst</td> <td>CanlAll</td> <td>Convtpol</td> <td>DispAll</td> <td>PortAttr</td> <td></td> </tr> </table>						F1	F2	F3	F4	F5	F6	F7	F8	TestOn	TestOff	CanlTst	CanlAll	Convtpol	DispAll	PortAttr	
F1	F2	F3	F4	F5	F6	F7	F8														
TestOn	TestOff	CanlTst	CanlAll	Convtpol	DispAll	PortAttr															

Fig. 3-7 IO ports of hardware

● The I/O ports illustration

<p>Test On (F1)</p> <p>Test Off (F2)</p>	<p>Select port, and press “Test On” or “Test Off”. The signal light before the ports will switch between red and green. When the light is green, this port is active; otherwise, this port is inactive.</p> <p>Note:</p> <p>In case of test mode and actual situation, the state of the signal light is different.</p> <p>Test mode, Green light:  Red light: </p> <p>Actual situation, Green light:  Red light: </p>
<p>Cancel Test (F3)</p> <p>Cancel All (F4)</p>	<p>Press these buttons to cancel analog signal and simulation, namely to use actual hardware signal instead of analog signal.</p> <p>Cancel Test: only cancel the test of selected ports.</p> <p>Cancel All: cancel the test of all ports.</p>
<p>Convert Polarity (F5)</p>	<p>Select port, press this button, and input the manufacture password then click “OK” to confirm. The modification result will take effect after the software restarting.</p>
<p>Display All (F6)</p>	<p>Press this button to show all I/O ports.</p>

Polarity Attribute (F7)

Press this button to open a dialog box named “Inport Sampling Setting” as shown in Fig. 3-8, where sampling interval, port name and description can be set. Filter function and port enabled are checked by default.

Filter function: improving the I/O ports capability of free-interference.

Port enabled: receiving signals from the outer.

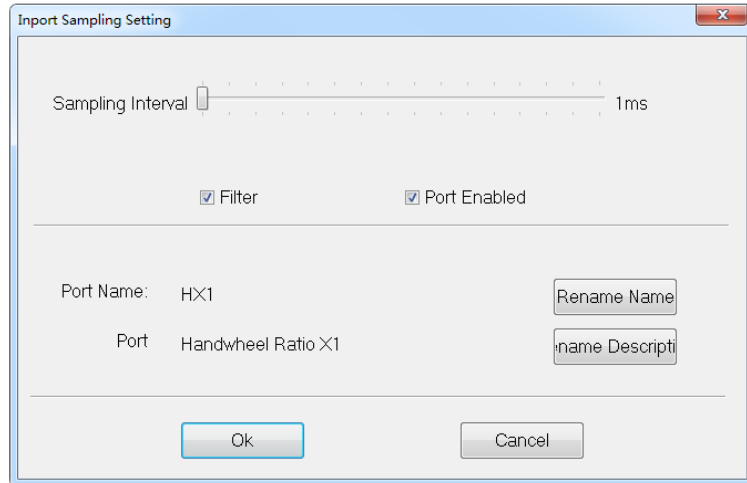


Fig. 3-8 Inport Sampling Setting

3.2.3 Manual Operation

Press numeric key 8 on main keyboard or “Manual” on operation panel to switch to Manual operation panel. The user can manually move the axis by numeric direction keys to confirm the correctness of each axis.

The axis direction keys as shown in Fig. 3-9 and Fig. 3-10 are used to manually move the axes. In manual mode, synchronously press the axis direction key and the rapid motion key “~” (or the corresponding numeric key), the machine tool will move at rapid jog speed. Only press the axis direction key or the corresponding numeric shortcut key, the machine tool will move at jog speed. The axis direction keys of four-axis software are different from the five-axis, but the operation method is same.

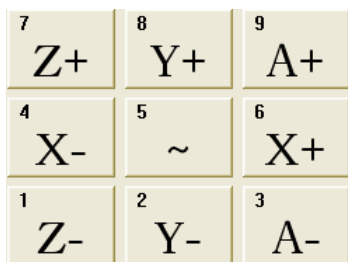


Fig. 3-9 Axis direction key(Four axes)

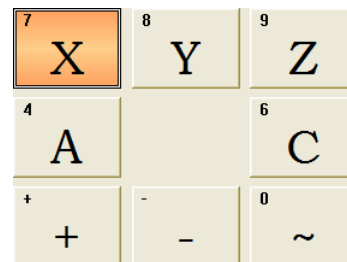


Fig. 3-10 Axis direction key(Five axes)

Manual mode contains three secondary modes, namely, jog, handwheel and stepping. In stepping mode, the user can set the step distance by clicking >>. For more details about manual mode and handwheel, please refer to Section 1.1 and Section 4.7 respectively.

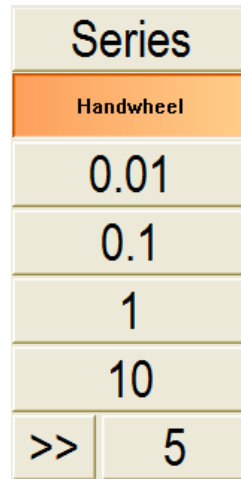



Fig. 3-11 Feed method in manual mode

3.2.4 Reset Machine

With the function “Back to Machine Origin”, machine will return all axes to the reference point and correct the coordinate system.

In the REF mode, the user can press the buttons under software window or the shortcut keys to return all axes or a certain axis to the reference point. In addition, the user can also select “All back to REF Point” item under “Operate” menu 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 the current coordinates has been saved when NcStudio is normally closed. Besides, if the user is definitely sure of the accuracy of current position, he/she can enable “Directly Setting” function.

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



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

3.2.5 Load Program File

Generally speaking, a machining file has to be loaded before machining begins. Otherwise, certain functions related to auto-processing will be unavailable.

Select “Open & Load” from “File” to open a dialog box where you can choose a program file to be processed.

Click “Open” to load the machining file into the system. At this time, the user can press numeric key 1 to switch to Position window and view the program lines in this machining file.

3.2.6 Set Workpiece Origin

The program zero is workpiece origin, or workpiece zero. Before actual machining, the user needs to ensure the actual position of workpiece origin as following steps:

1. In Manual mode, move X/Y/Z/A axes to the intended workpiece origin position.
2. Switch to “POS” or “OFFSET” window, click F5 “All Clear”, or click F1 “X Clear”, F2 “Y Clear”, F3 “Z Clear” and F4 “A Clear” successively to clear the axis coordinate to zero. Of course, in order to set the workpiece origin, item “Set Workpiece Origin” under “Operate” menu can also be used to clear all axes.



The above operations help to set the workpiece origin. The user can also set a more accurate workpiece origin in Z axis with the help of mobile calibration function.

3.2.7 Set Speed-related Parameters

DEFAULT_FEEDRATE: Parameter N64021, it refers to feedrate in machining or feedrate specified by command GXX. If F command is contained in a program file, the setting of parameter N72001 “SPEED_ASSIGN_TYPE” decides whether the feedrate specified by the F command or the setting value of parameter N64021 will be used in machining.

RAPID_TRAVEL_FEEDRATE: Parameter N64020, it refers to G00 feedrate, which is the feedrate of a linear axis in locating or positioning.

REV_RAPID_TRAVEL_FEEDRATE: Parameter N64030, it refers to the feedrate of a rotary axis in locating or positioning.

3.2.8 Execute Auto Machining

Auto machining means that machine tool automatically processes the loaded program file from the first line.

● Start

There are two ways to execute auto machining:

1. Select “Start” item under “Operate” menu list;
2. In auto mode, click button F9 “Start”.

● Stop

During auto machining, there are two 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 F11 “Stop”.

- **Pause**

During auto machining, there are two optional ways to suspend the processing:

1. Select “Pause” item under “Operate” menu list;
2. In auto mode, click button F10 “Pause”.

In addition, the user can use the methods mentioned in “Start Auto Machining” part to continue processing.

4 Software Functions

Due to the similarity operation of four-axis software and five-axis software, this chapter will take the common software for four axes (XYZA) as an example to introduce the software functions, if there is no special notification.

4.1 Axis Control

For four axes system, up to four axes can be supported, and it can be used to control XYZA, XYZB and XYZC type machine. For five axes system, up to five axes can be supported, and it can be used to control XYZAB, XYZAC and XYZBC type machine. Please refer to Section 1.3 for detailed definitions of each axis, and refer to Section 3.2.1 for detailed information of axes configuration.

Motion direction and range of each axis is decided by manufacturer's parameters N1000 "AXIS_DIR", N10020 "WORKBENCH_LOW_LIMIT", and N10030 "WORKBENCH_UPPER_LIMIT". Correct parameters settings can prevent the machine from collision.

Related parameters (are of MFR's access)

Parameter		Description	Setting Range
N10000	AXIS_DIR (X/Y/Z/A)	The direction of axes.	1: positive direction; -1: negative direction
N10020	WORKBENCH_LO WER _LIMIT (X/Y/Z/A)	When workbench range is effective, the machine coordinate of the lower limit of the range.	-99999~99999
N10030	WORKBENCH_UPP ER _LIMIT (X/Y/Z/A)	When workbench range is effective, the machine coordinate of the upper limit of the range.	-99999~99999
N10040	CHECK_WORKBEN CH _RANGE (X/Y/Z/A)	Enable workbench range or not.	Yes: enable; No: disable
If the parameter N10040 is set to "Yes", the software will give yellow alarm prompt and stop machining (if it is in machining) when the axis travels exceeding the lower or upper limit of the range. Please debug according to real situation.			

4.2 Tool Measurement

Tool measurement is a process to set up a workpiece coordinate system in machine coordinate system. In other words, tool measurement aims to set the workpiece origin, or workpiece zero/program zero.

4.2.1 Fixed Calibration

Fixed calibration refers to executing measurement operation at a certain fixed position on the machine. The tool sensor position is decided by parameter N75210 “FIXED_CALI_POS”. In Manual mode, enter into “POS” functional area, and click button F6 “Fix Cali” to conduct fixed calibration operation.

During actual machining, tool length and tool holder position will change after tool change due to breakage or other reasons. On this occasion, the user can conduct fixed calibration to re-confirm the tool length offset. This kind of tool measurement mode is applied in multi-tools mode, namely machine with tool magazine. The sketch map of fixed calibration is as shown in Fig. 4-1:

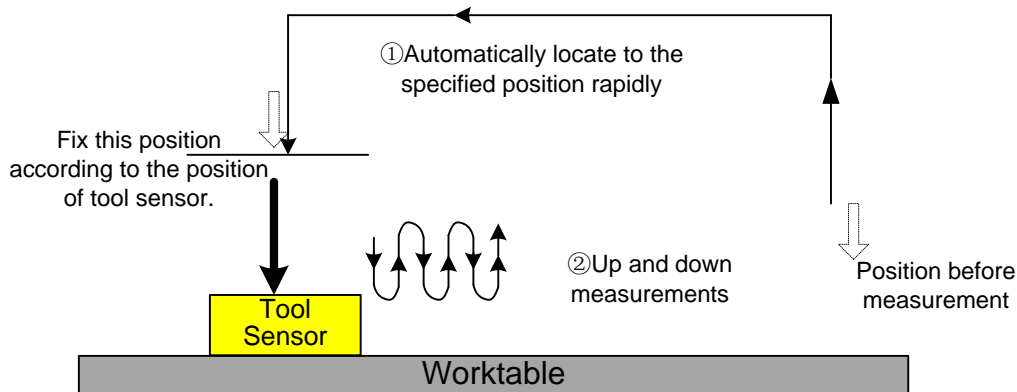


Fig. 4-1 Fixed calibration process

During fixed calibration, the machine coordinate will be recorded when tool nose touching the surface of tool sensor. After tool calibration, the system will automatically make the coordinate subtract from sensor thickness and set it to the tool offset. At the same time, calibration result (the machine coordinate of the tool nose when it touches the sensor surface) will appear in CNC info bar, as illustrated in Fig. 4-2:

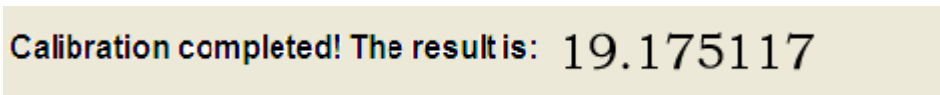


Fig. 4-2 Prompt info after fixed calibration

The relationship among tool offset, calibration result and tool sensor thickness is as below:

$$\text{Tool Offset} = \text{Fixed Calibration Result} - \text{Tool Sensor Thickness}$$

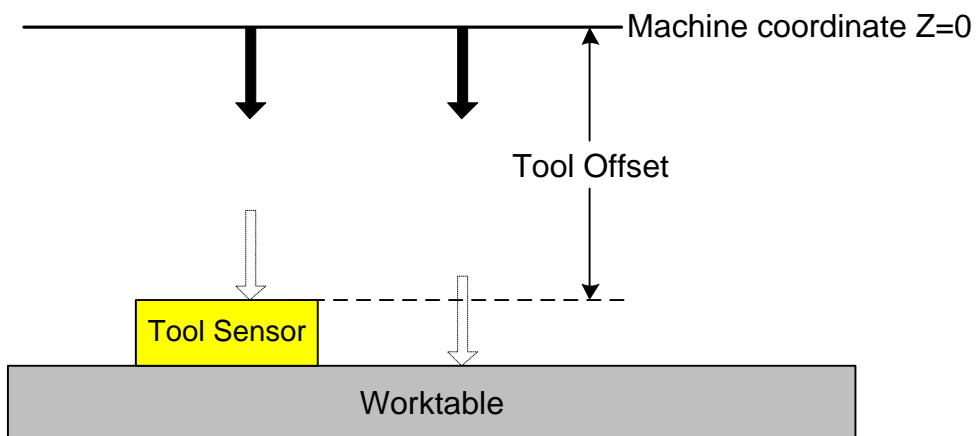


Fig. 4-3 Sketch map of tool offset

Specific calibration steps are as below:

1. Choose a tool by tool number (Txx).
2. Conduct fixed calibration on the selected tool by clicking button F6 “FixCali”.
3. Record value of tool offset.
4. Repeat step 1) and step 2) on each tool.
5. Choose any tool and move it onto the workpiece surface, and execute clear operation by clicking F3 “Z Clear”.



The user needs to conduct fixed calibration to set tool offset value firstly, and then move the tool nose onto workpiece surface and conduct “Z Clear” operation.

Related parameters (N75210 is of operator’s access and the rest of MFR’s access)

Parameter		Description	Setting Range
N75201	FIXED_CALI_HEIGHT	The height from tool sensor surface to the surface of workbench.	-9999.99~9999.99
N75203	FIXED_CALI_QUICK_SPEED	The speed that the tool moves from the highest position to the calibration-start position in fixed calibration.	0.001~100000
N75210	FIXED_CALI_POS (X/Y/Z)	The machine coordinates of tool sensor (or tool calibrator) in fixed calibration.	-99999~99999

Setting method of the parameter N75201:

- 1) Manually move Z axis to a point above the workbench surface, and slowly move the tool nose downward until it touches the workbench surface. At this time, Z coordinate will be recorded as Z1;
- 2) Lift Z axis, move it to the upper side of tool sensor at the fixed position, slowly move Z axis downward until the tool nose touches tool sensor and obtain tool sensor signal. At this time, Z coordinate will be recorded as Z2:
- 3) Z2-Z1, the difference obtained is the tool sensor thickness. Set the value to parameter N75201.

Parameters N75001/N75002/N75020/N75025/75220/75230/N75240 will possibly be used in fixed calibration. Please refer to Section 4.2.2 for details.

4.2.2 Mobile Calibration

Mobile calibration can be used to set workpiece origin of Z axis by executing measurement at the current position. This measurement type will set the workpiece offset according to the calibration result and the following calculation equation. Relationship between workpiece offset and calibration result is as shown below. Generally, the factory setting value of public offset and tool offset is 0.

$$\text{Workpiece Offset} = \text{Mobile Calibration Result} - \text{Tool Sensor Thickness} - \text{Public Offset} - \text{Tool Offset}$$

After mobile calibration, CNC info bar will show the calibration result and current workpiece offset successively.

The sketch map of mobile calibration is as shown in Fig. 4-4:

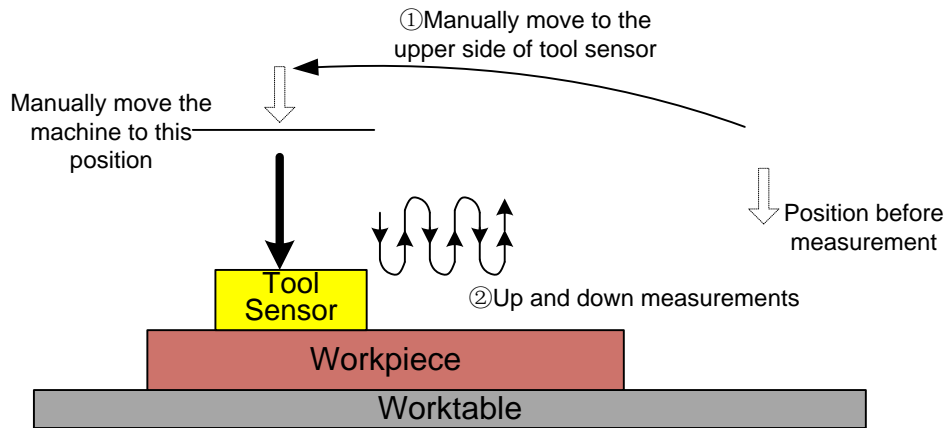


Fig. 4-4 Sketch map of mobile calibration process

The operation of Mobile calibration: In Manual mode, place tool sensor onto workpiece, move Z axis to the upper side of the tool sensor, and click button F7 “MobCali” in “POS” functional area. The system will automatically complete the calibration process and set the workpiece origin in Z axis.

Related parameters (all are of MFR’s access)

Parameter		Description	Setting Range
N75001	CALI_SPEED	The speed of tool when it approaches tool sensor surface in calibration.	0.001~100000
N75002	CALI_TIMES	The times of accurate positioning when tool approaches tool sensor surface in calibration.	1~99
N75020	CALI_TORLERANCE	The maximum tolerance of tool calibration.	0~10
Value of parameter N75020 is the maximum tolerance of tool calibration. Compared with average value of calibration results, when average value is smaller than this value, calibration succeeded, otherwise, calibration failed.			
N75025	CALI_PROTECT_ENABLED	Whether to protect the tool from over-travel in calibration.	No: invalid Yes: valid
N75100	MOB_CALI_THICKNESS	In mobile calibration, the height from tool sensor surface to workpiece origin in Z axis.	-1000~1000

Parameter	Description	Setting Range
Setting method of parameter N75100:		
1) Manually move Z axis to a point above the workpiece surface, and slowly move the tool nose downward until it touches the workpiece surface. At this time, Z coordinate will be recorded as Z1;		
2) Lift Z axis, move it to the upper side of tool sensor on workpiece, and slowly move Z axis downward until the tool nose touches tool sensor and obtain signal. At this time, Z coordinate will be recorded as Z2;		
3) Z2-Z1, the difference obtained is the tool sensor thickness. Set the value to parameter N75100.		
N75220	Cali Work Bench Upper Limit	The machine coordinates of the upper limit of workbench in calibration.
N75230	Cali Work Bench Lower Limit	The machine coordinates of the lower limit of workbench in calibration.
N75240	Cali Area is Valid?	Whether the calibration range is effective or not.
		No: invalid Yes: valid

4.2.3 Special Measurement

Special measurement includes auto measurement of workpiece origin and workpiece boundary. As measurement signal is used, they are called special measurement, exclusively used in Weihong system.

Special measurement is mainly used to calibrate X and Y center of workpiece in order to facilitate machining and generating a machining file, under the precondition that the Z axis workpiece coordinate is confirmed and the worktable of machine tool is insulated.

In manual mode, turn to “Measure” window under “POS” function area, as shown below. In this window, you can conduct special measurement.

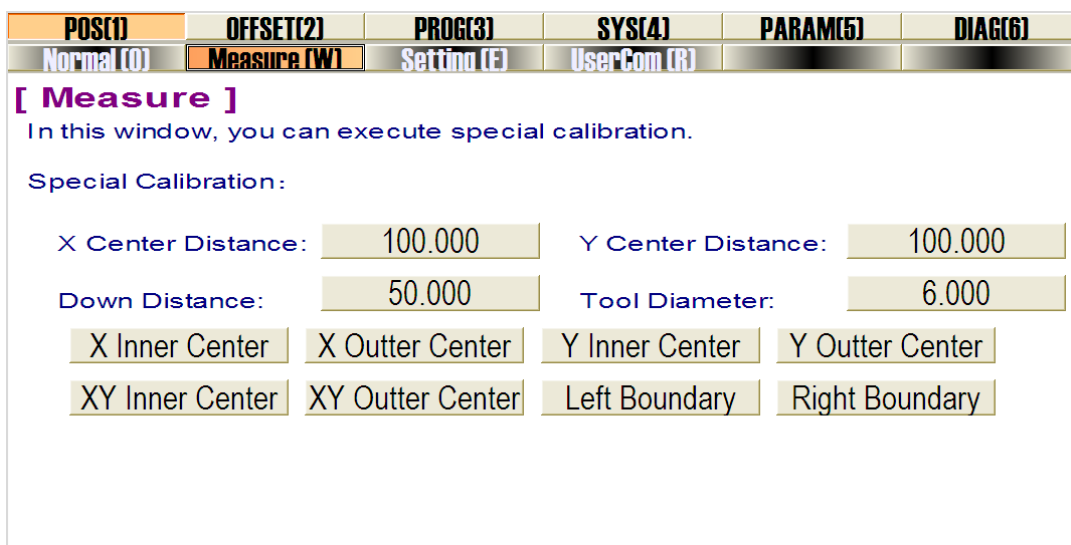


Fig. 4-5 Special calibration

X Center Distance: the pre-estimated distance from workpiece center to X boundary. In outer centering, this distance must be a little larger than the actual value, while smaller in inner centering.

Y Center Distance: the pre-estimated distance from workpiece center to Y boundary. In outer centering, this distance must be a little larger than the actual value, while smaller in inner centering.

Down Distance: tool plunging/lifting distance in tool measurement. In inner centering, this distance must be smaller than the distance from tool nose to workpiece surface, while larger in outer centering.

Tool Diameter: the actual diameter of tool.

● **Centering**

Centering is used to confirm the center point of workpiece blank. In “Measure” window, the user can conduct this function by clicking any one of “X/Y/XY Inner Center”, “X/Y/XY Outer Center”.

Taking the process of “X Inner Center” as an example to introduce centering process, the automatic centering sketch maps are as shown in Fig. 4-6 and Fig. 4-7.

1. Place a conductive workpiece (copper, iron, aluminum) on the insulated workbench, and connect it to the port CUT on terminal board, while the tool to COM port.
2. Before automatic centering, put tool on the predicated center point position, and then click “X Inner Center” to make the tool fall “Down distance”, and translate towards the workpiece a short distance until it touches the conductive workpiece. At this time, the circuit is conducted and signal is sent to the system which will automatically record the current axial coordinate as X1.
3. Then the tool will rise “Down distance”, move horizontally two “X Centre Distance”, move down “Down distance”, and translate towards the workpiece a short distance until it touches the conductive workpiece. As a result, the circuit is conducted and signal is sent to the system which will automatically record the current axial coordinate as X2. The system will then calculate the X coordinate of workpiece center point and move the tool to this center point.

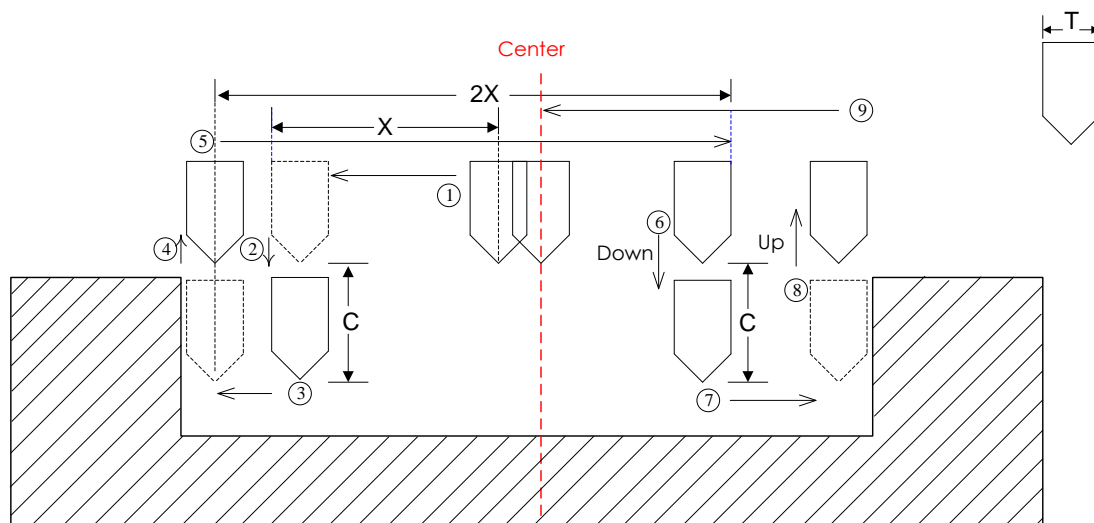


Fig. 4-6 Measurement process of [X Inner Center]

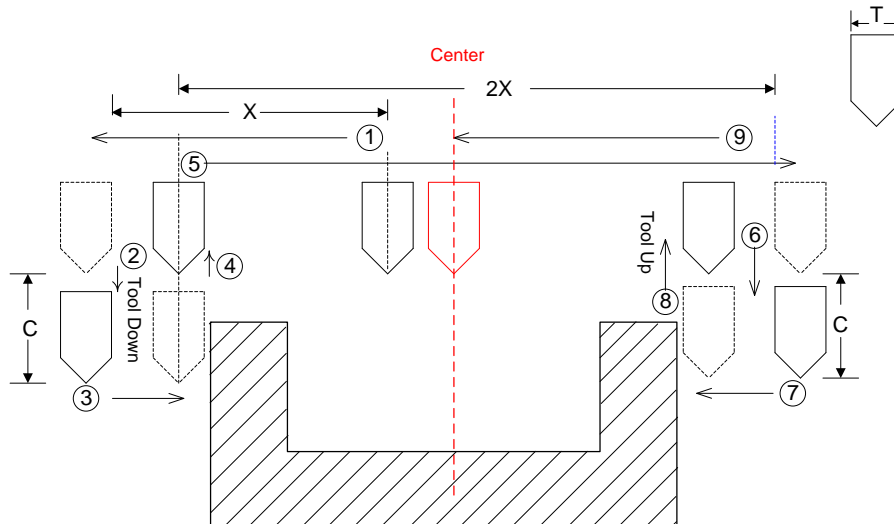


Fig. 4-7 Measurement process of [X Outer Center]



1. Before centering, the user must place the tool nose at the pre-estimated center point, as well as set the value of “X/Y Center Distance”, “Down Distance” and “Tool Diameter”.
2. In inner centering, the “Down Distance” should be smaller than the distance between tool nose and workpiece surface, while larger in outer centering.

● **Boundary Measurement**

Boundary measurement includes calibrating +X boundary, -X boundary, +Y boundary and -Y boundary, which is mainly used to set a boundary point as the workpiece origin. The measurement processes of -X boundary, +Y boundary and -Y boundary are the same as that of +X boundary. The sketch map is as shown in Fig. 4-8.

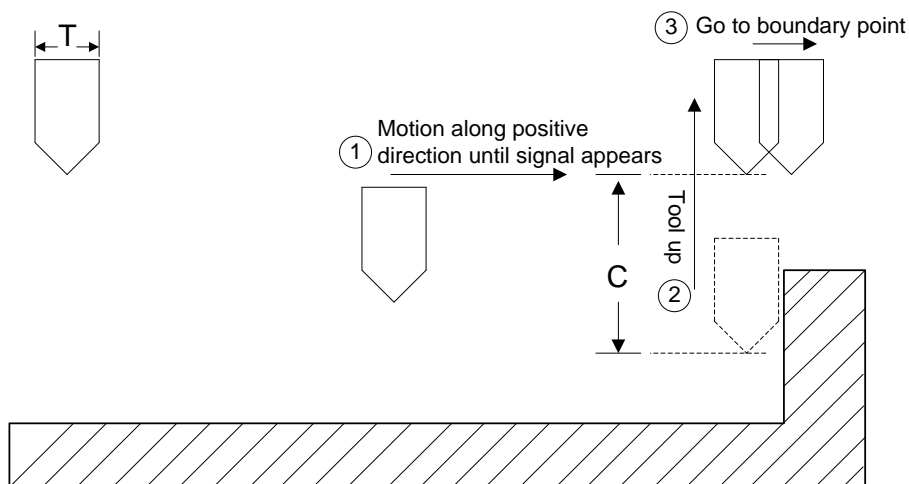


Fig. 4-8 Measurement process of +X boundary

4.3 Offset Management

4.3.1 Workpiece Coordinate System (WCS)

At present, the system supports G54, G55, G56, G57, G58, G59 and the expanded workpiece coordinate system of G59 (G59P1~G59P256).

Before using expanded workpiece coordinate system, the user needs to set parameter N61110 as “Yes”. In addition, the user can set the number of expanded workpiece coordinate system in parameter N61111.

Related Parameters (all are of MFR’s access)

Parameter		Description	Setting Range
N61110	Support Extra Workcoors	Whether the extended WCSs can be supported.	Yes: Support No: Not support
N61111	Size of Extra Workcoors	The number of extra WCSs can be supported.	1~256

N61110 is set to “No” by default. When more than 6 parts need to be setup on worktable at the same time, set N61110 to “Yes” to activate the extended WCSs, and further set N61111 to a desired value.

The relationship of workpiece offset and machine coordinate system is as shown in Fig. 4-9.

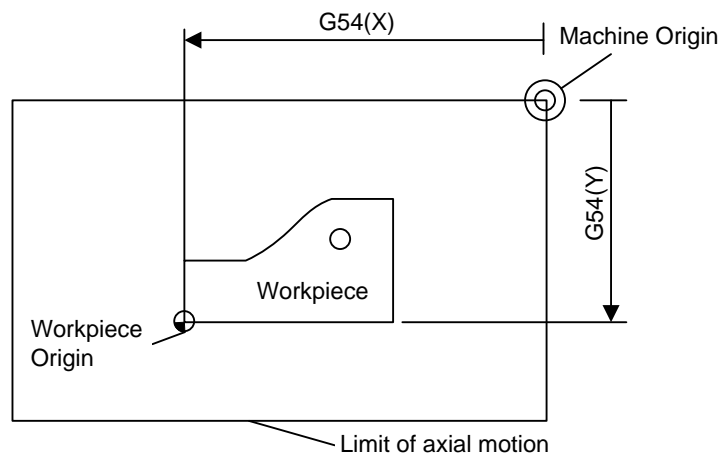


Fig. 4-9 The relationship between workpiece offset and machine coordinate system

One, two or multiple workpiece offsets can be used in machining program. As shown in Fig. 4-10, if three workpieces are installed on the workbench, each workpiece has a workpiece origin relative to G code of WCS. The programming example is as following: drill one hole on each of the three workpieces, with calculation depth Z-0.14.

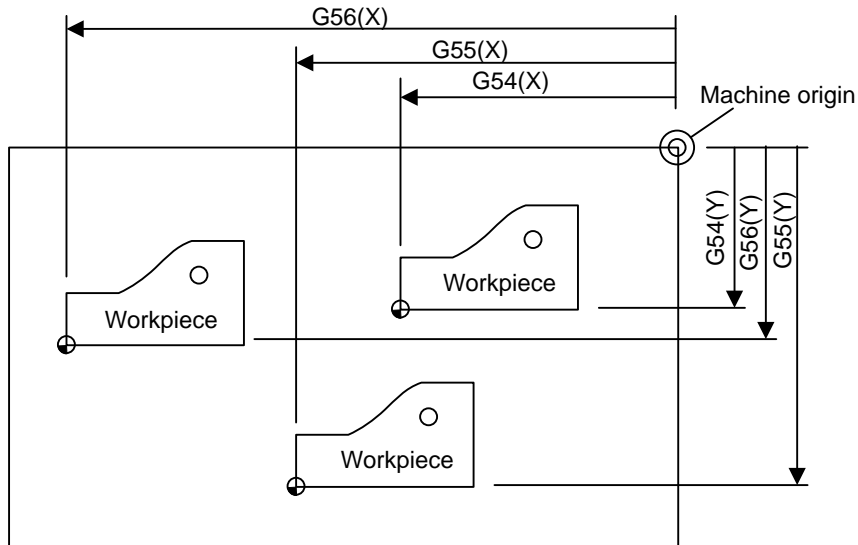


Fig. 4-10 Sketch map

```

O1801
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X5.5 Y3.1 S1000 M03      ( Select G54 )
N4 G43 Z0.1 H01 M08
N5 G99 G82 R0.1 Z-0.14 P100 F8.0
N6 G55 X5.5 Y3.1                        ( Switch to G55 )
N7 G56 X5.5 Y3.1                        ( Switch to G56 )
N8 G80 Z1.0 M09
N9 G91 G54 G28 Z0 M05                    ( Switch to G54 )
N10 M01
...

```

The program segment N3 ~ N5, within WCS of G54, is related to the first workpiece; Segment N6 will drill a hole on the second workpiece of the same batch in WCS of G55, while segment N7 will drill a hole on the third workpiece of the same batch in WCS of G56.

Aiming at all the coordinate systems, public offset is used to adjust the workpiece origin of X, Y, and Z axis, but will not change the workpiece offset value.

The formula of workpiece offset, tool offset and public offset is as below:

$$\text{Workpiece coordinate} = \text{Machine coordinate} - \text{Workpiece offset} - \text{Tool offset} - \text{Public offset}$$

4.3.2 Offset Setting

As shown in Fig. 4-11, the "Offset" window shows the public offset and current workpiece offset of each axis. This dialog box will pop up after clicking "Advanced Setting" in "Offset" interface.

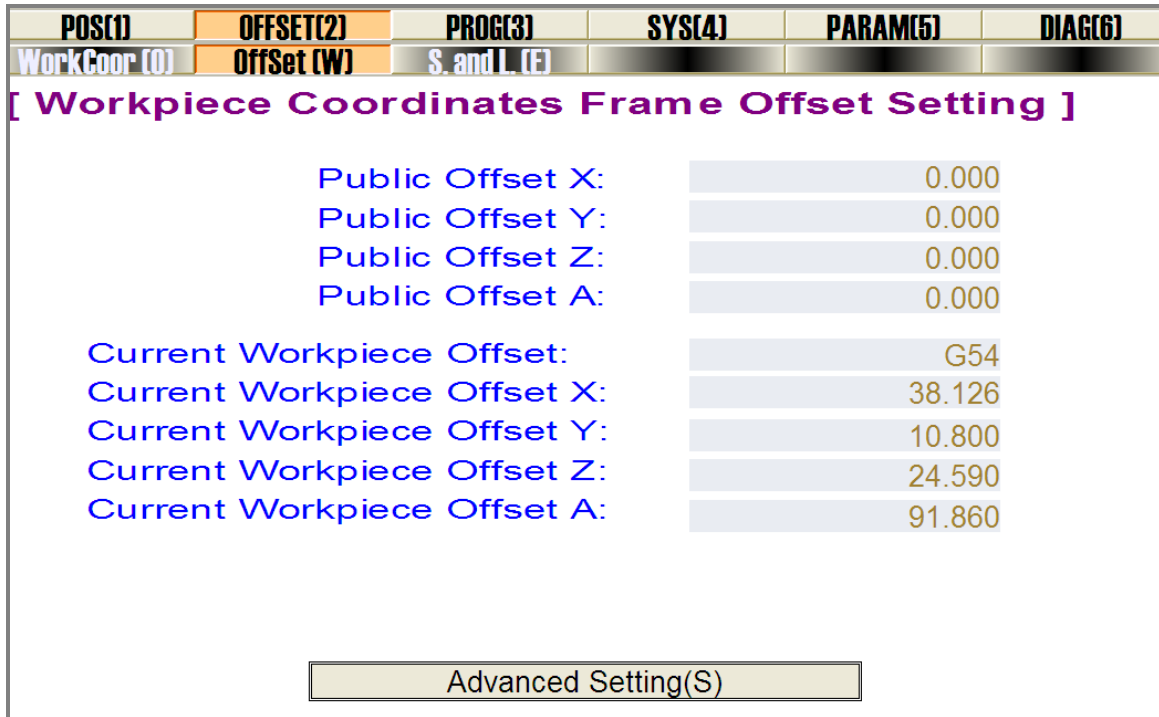


Fig. 4-11 Offset

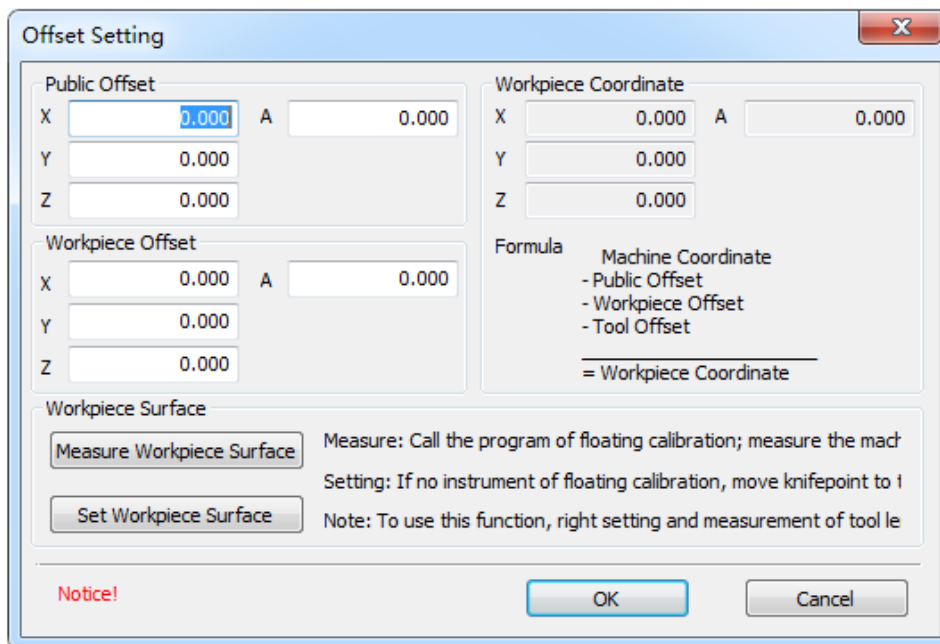


Fig. 4-12 Offset setting dialog box

- **Public offset**

Also called external offset, it is used to record temporary adjustment value of the workpiece origin. Note that this value can only be modified manually.

- **Workpiece offset**

It shows the distance of the workpiece origin relative to the machine origin, that is the machine coordinate of the workpiece origin. You can manually input number to define the machine coordinate of workpiece origin (it is NOT recommended).

- **Workpiece coordinate**

It shows workpiece coordinate of current point. You can clear workpiece coordinate of corresponding axis by clicking F1~F5 “Set X/Y/Z/A/XYZA” buttons on “Offset” window.

- **Workpiece surface**

With this function, workpiece coordinate of Z axis can be cleared to zero.

“Measure Workpiece Surface” refers to calling mobile calibration function, setting machine coordinate recorded when tool nose touches workpiece surface to workpiece offset, and clearing it to zero afterwards. It can be used when a tool sensor is available.

“Set Workpiece Surface” refers to manually moving tool nose to workpiece surface, executing “Set Workpiece Surface” and clearing the workpiece coordinate to zero afterwards. It can be used when there is no tool sensor.



1. Before function “Measure Workpiece Surface” is enabled, you need to firstly measure the thickness of tool sensor, and input the result into parameter N75100 “MOB_CALI_THICKNESS”.
2. After a workpiece origin is set and obtained, if it needs to be frequently used in later operation, you can select the item “Save workpiece origin” under menu “Operate” to save the offset of current workpiece origin. Up to 10 groups of origins can be saved. After a workpiece origin is saved, it can be called and re-used through item “Read workpiece origin” under menu “Operate” afterwards.

4.4 Program Management

4.4.1 Program Wizard

Click “Process Wizard” under “PROG” function area to open “Process Wizard” window, as shown in Fig. 4-13.

The software offers four basic processing program wizards: circular pocket, circular frame, rectangular pocket and rectangular frame. The user just needs to modify the relative parameters to complete milling operation of circular frame and rectangular frame, etc. After setting well, click “Save” and “Load” then the graph set before will be shown in tool path window.

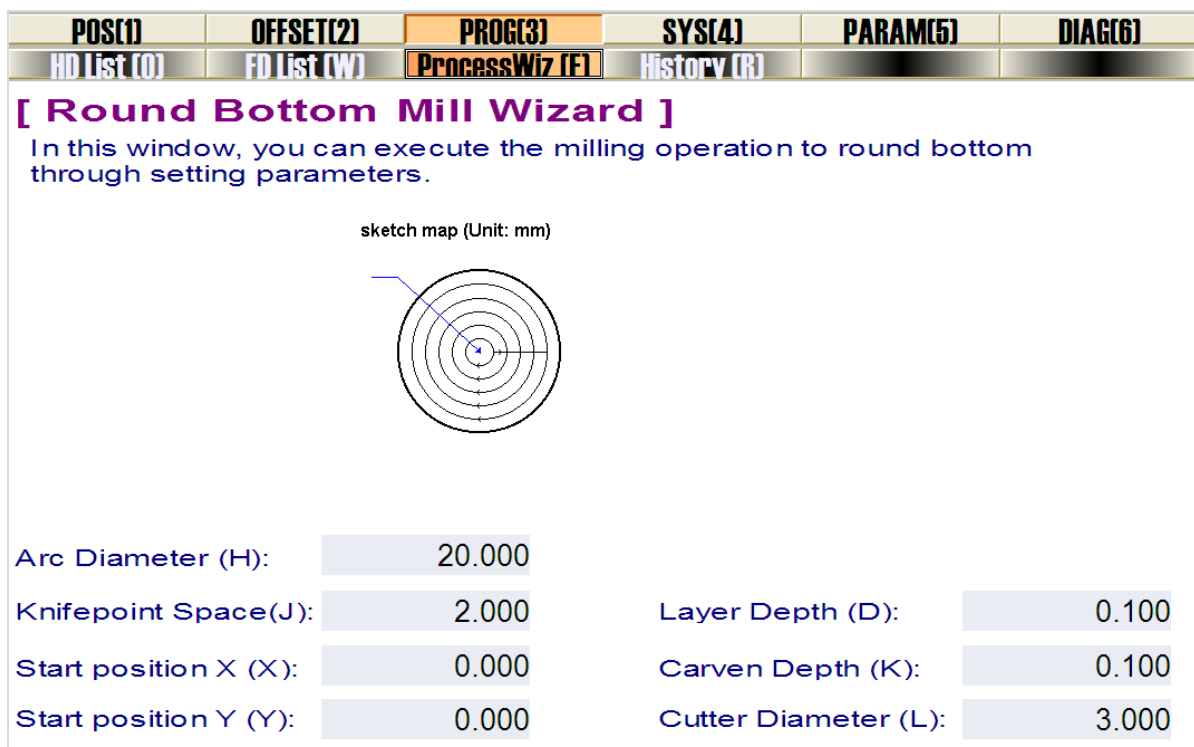


Fig. 4-13 Program wizard-Round bottom mill

4.4.2 Program Files

- **Hard disk list**

Click “HD List” under “PROG” to open hard list window, as shown in Fig. 4-14, which shows all program files and folders under directory D:\NCFILES. The user can click buttons on operation bar at the lower part of window to load, edit, delete, create as well as rename a program.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
HD List (O)	FD List (W)	ProcessWiz (E)	History (R)		
File Name	Size	Last Revised Time			
▢ CirContour.nc	230	2016-12-15 14:20			
▢ ScrewErr Mea...	147	2016-12-15 15:38			
▢ Untitle1.nc	0	2016-12-15 14:30			
Current File folder: D:\NCFILES\ File Name: CirContour.nc Size: 230 When The File Modified Last Time: 2016-12-15 14:20 File Head: 'D = 0.1000 'Delta = 0.1000 'T = 3.0000 'X = 0.0000					
F1	F2	F3	F4	F5	F6
Load	Edit	Delete	New	Rename	

Fig. 4-14 Program files list in hard disk



1. Users can store program files under directory D:\NCFILES, thus the relative operations to those programs can be directly done under this directory.
2. Apart from editing on “Hard disk/Floppy disk list” window, users can also edit the program by clicking item “Open and edit” or “Edit current program” under “File” menu. Note that “Edit current program” item is valid only when the format of loaded program file is G code.

● **Floppy disk list**

Click “FD List” under “PROG” to open floppy disk list window, as shown in Fig. 4-15, where displays program files in removable disk. For example, if there is a file saved under naiky folder of U flash disk, the user can do operations to this file via the operation buttons at the lower part of this window. If several removable disks exist, press F3 “Select Drive” to make a choice.

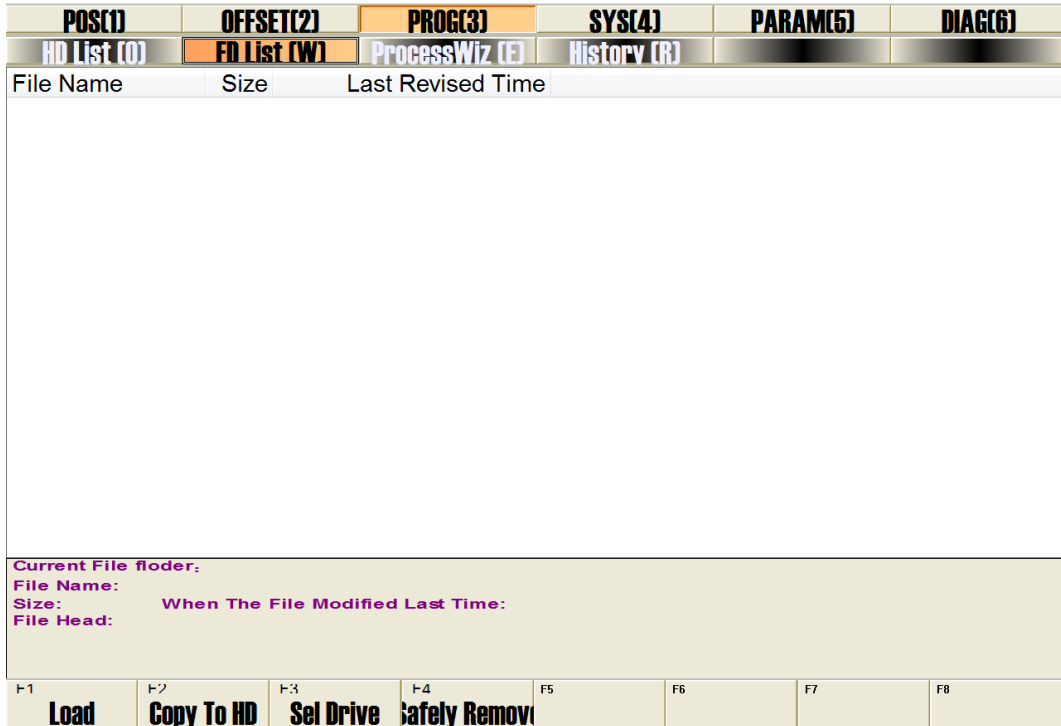


Fig. 4-15 Program files list in floppy disk

4.5 Speed Management

4.5.1 Spindle Speed, Feedrate and G00 Speed

In auto mode, turn to “Setting” window under “POS” functional area, as shown in Fig. 4-16.

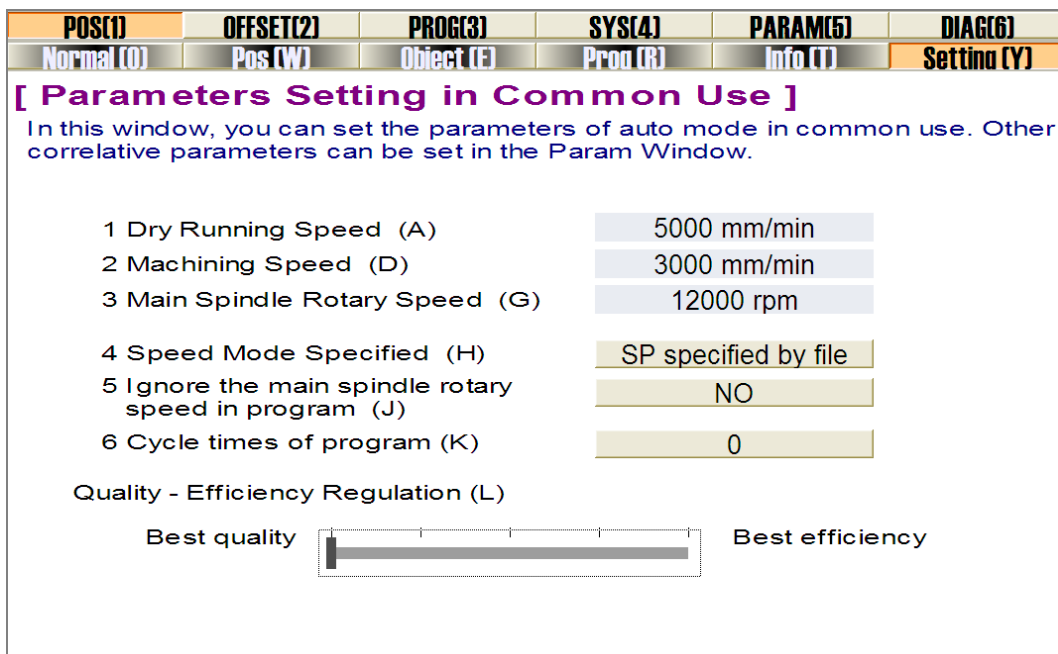


Fig. 4-16 Spindle speed setting in auto mode

● Spindle speed

Click figure button next to each parameter to open the input box, where you can enter desired spindle speed. Modification takes effect immediately.

1. As shown in Fig. 4-16, if parameter “Ignore the main spindle rotary speed in program” (N72002 “IGNORE_PROG_SPINDLE_REV”) is set to “No”, the spindle speed specified by S command in the program file will be enabled if S command exists; otherwise, spindle speed specified by “Main spindle rotary speed” will be enabled in machining.
2. While if parameter “Ignore the main spindle rotary speed in program” is set to “Yes”, spindle speed specified by “Main spindle rotary speed” will be enabled regardless of S command in program file.

● Feedrate

It refers to feeding speed or Gxx speed, the running speed in machining specifically. Following are the method and requirements to set feedrate.

1. Click figure button next to “Machining speed” to open input box, where you can enter a desired value, as shown in Fig. 4-16. Modification takes effect immediately. Or turn to “Parameter” window under “PARAM” function area and modify parameter N64021 “DEFAULT_FEEDRATE”. Modification takes effect immediately.
2. The machining speed (Gxx speed) should be smaller than the dry running speed (G00 speed).
3. If item “Speed Mode Specified” in Fig. 4-16 is set to “SP specified by the file” (corresponding to parameter N72001 “SPEED_ASSIGN_TYPE” is set to “0”), feedrate specified by F command will be enabled if F command exists in the program file; otherwise, feedrate specified by “Machining speed” (corresponding to setup value of parameter N64021) will be enabled in machining.
4. While if the item is set to “Default SP” (corresponding to parameter N72001 is set to “1”), the “Machining speed” will always be enabled regardless of existence of F command in the program file.

● G00 Speed

G00 speed refers to the axial speed when machine is positioning.

1. Click figure button next to “G00 speed” to open input box, where you can enter a desired value, as shown in Fig. 4-16.
2. Turn to “Parameter” window under “PARAM” function area, find parameters N64020 “RAPID_TRAVEL_FEEDRATE” and N64030 “REV_RAPID_TRAVEL_FEEDRATE” and set them to desired values.
3. Dry running speed should be larger than machining speed, namely, G00 speed should be larger than Gxx speed.

4.5.2 Jog Feedrate/ Rapid Jog Feedrate

Jog feedrate (or called manual low speed) refers to the speed when one axis direction button is pressed alone in manual mode, with setting range “1mm/min ~ Jog rapid feedrate”. Rapid jog (or called manual high speed) refers to the speed when axis direction button and “Rapid” button are pressed together in manual mode, with setting range “Jog feedrate ~ G00 speed”.

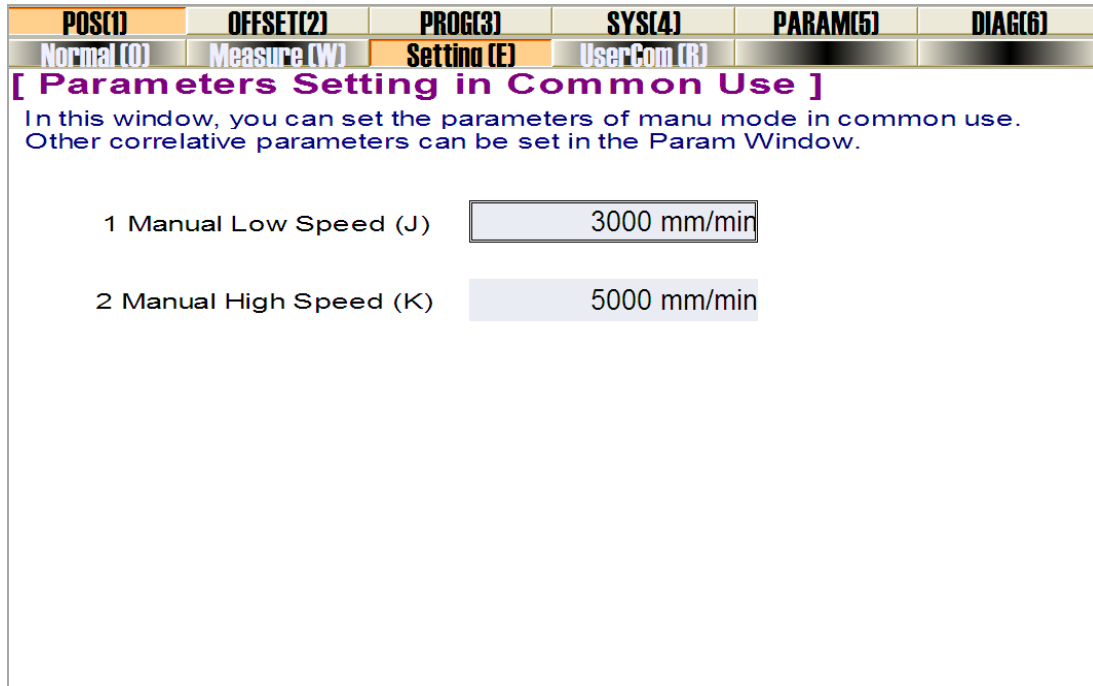


Fig. 4-17 Feedrate setting in manual mode

1. Turn to “Setting” window under “POS” function area in manual mode, as shown in Fig. 4-17. Press number button next to each parameter to open an input box, where you can set an appropriate value.
2. Turn to “Parameter” window under “PARAM” function area, find and modify parameters N71000 “JOG_VOL” and N71001 “RAPID_JOG_VOL”.



Above introduction takes linear axes (X/Y/Z) as an example. Please set parameters N71002 “REV_JOG_VOL” and N71003 “REV_RAPID_JOG_VOL” in the same way if active axes are rotary axes.

4.5.3 Acceleration

It includes angular acceleration of rotary axis, machining acceleration, dry run acceleration, acceleration at corner as well as jerk (acceleration of acceleration). See details in table below.

Parameter		Description	Setting Range
N64080	ANGLE_ACCELERATION	It specifies angular acceleration of rotary axis.	0.001~100000
N64100	AX_LINEAR_MACH_ACC	It specifies the max. acceleration of each linear axis in machining.	0.001~10000000

They are used to describe the acceleration and deceleration capability of each feed axis, with unit “mm/s²”, which is determined by the physical feature of machine, such as the quality of motion part, torque, cutting load and resistance of the feed motor. The larger value of the parameter is, the less time the machine will spend in acceleration or deceleration process, the higher the efficiency is.

Parameter		Description	Setting Range
Generally, for servo motor system, the value should be set within “400 ~ 1200”. Set a small value at first, and then repeatedly execute typical motion for a period of time. If there is no abnormal situation, gradually increase the value. If abnormal condition occurs, reduce the value, with “50% ~ 100%” insurance allowance.			
N64101	AX_LINEAR_POST_ACC	It specifies the max. acceleration of each linear axis when machine is positioning.	0.001~10000000
N64120	AX_CON_ACC	The max. resultant feed acceleration of adjacent two axes. 1~2 times of the single axis acceleration is recommended.	0.001~100000
They are used to describe the acceleration and deceleration capability of multiple linkage axes, with unit “mm/s ² ”. The larger value of the parameter is, the higher allowable velocity is in arc motion. Generally, for servo motor system, the value should be set within “1000 ~ 5000”. While for heavy machine, it should be a smaller value. Set a small value at first, and then repeatedly execute typical motion for a period of time. If there is no abnormal situation, gradually increase the value. If abnormal condition occurs, reduce the value, with “50% ~ 100%” insurance allowance.			
N64150	AX_ACC_ACC	The acceleration of the linear acceleration under S-type velocity curve.	0.001~100000



Considering driving capability of the servo motor, friction of mechanical assembly and bearing capability of mechanical components, parameter N64022 “MAX_MACH_FEEDRATE” can be modified to limit the max. allowable velocity of linear axis in machining. When set to “0” (default), the parameter is invalid.

4.5.4 Reference Circle Speed

Reference circle speed includes reference circle max. speed and reference circle min. speed. They are valid on condition that velocity limitation to circular arc is enabled.

Related parameter (are of MFR’s access)

Parameter		Description	Setting Range
N64208	REF_CIRCLE_MAX_VELO	The max. velocity of circle with 10mm-diameter.	0.001~100000
When processing an arc, the vibration will occur due to centrifugal force. To minimize the vibration, the software makes limitation to machining speed when process an arc. Taking default setting as an example, diameter of reference circle is 10mm, and maximum linear velocity is 1800mm/min, then according to formula for centripetal acceleration:			

Parameter	Description	Setting Range
$a = \frac{V^2}{R}$ <p>Where: $r=(10/2)$mm; $v=1800$mm/min;</p> <p>Centripetal acceleration a can be figured out, which will be the maximum allowable centripetal acceleration when machining other arc or circle. If the arc speed is too big, resulting in a larger acceleration value than the formula result, limitation to the speed will be enabled.</p>		
N64209	CIRCLE_MIN_VELO	The minimal velocity of circle motions.
0.001~100000		
<p>According to formula above, the result of a is not the sole factor to decide the activation of speed limitation. According to the formula, it can be concluded that time will be wasted when a circle with a relatively small diameter is being machined. For this reason, this parameter is developed, with which no matter how small the circle radius is, the actual machining speed will not be smaller than the minimum speed specified by this parameter.</p>		

4.6 Simulation and Track

4.6.1 Simulation

The function of simulation provides a fast but lifelike simulated processing environment for users.

Running under this mode, the system will not drive the machine tool to do the according actions but only show the processing trace of the cutter in high speed in the track window. By simulating, users can see the moving form of machine tool in advance, avoiding machine tool damage due to programming mistakes in processing procedure. And they can also know the processing information in “Normal” window.

Below is “Normal” window under “POS” function area in simulation mode.

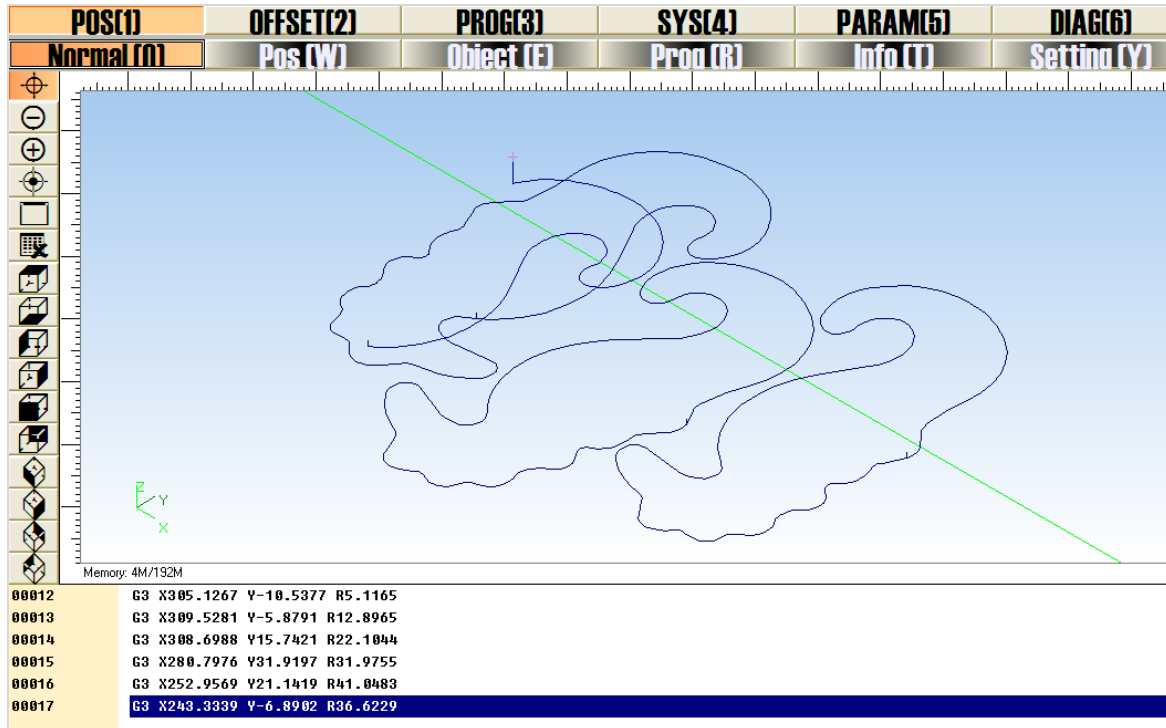


Fig. 4-18 Simulation interface

After a tool path program has been loaded into the system, there are two ways to enable simulation.

1. Clicking “Operate” → “Simulation mode”, simulating machining will be enabled, and the system will start machining the program from the first line. At this time, sub-menus under “Operate” menu change, with “Simulation mode” changing to “Stop simulation and withdraw the simulation mode”, “Start” changing to “Simulation start”, “Pause” changing to “Simulation pause”, and “Stop” changing to “Simulation stop”. If you want to disable simulation mode, click “Stop simulation and withdraw the simulation mode” to exit when simulation finished, or firstly click “Simulation stop” to stop this operation and then click “Stop simulation and withdraw the simulation mode” to exit.
2. In auto mode, turn to “POS” function area, click button F4 “Simulate” first and then click F9 “Start” to start simulation machining from the very beginning. Click F11 “Stop” to stop simulation, and F10 “Pause” to pause. If you want to exit from simulation mode, firstly click F11 “Stop” and then click sub menu “Stop simulation and withdraw the simulation mode”.

4.6.2 Track

In “Normal” and “Object” windows under “POS” function area, different tool path tracks can be displayed, including simulation track, pre-loaded track as well as actual machining track. Apart from the absence of file information display area, “Object” window is the same with “Normal” window.

After program file has been loaded into the system, pre-loaded track will be displayed in these two windows by default. If you want to cancel this default setting, set parameter N81000 “AUTO_LOAD_TRACK” to “No”. Note that if track size of loaded program file exceeds the limit specified by parameter N81001 “AUTO_LOAD_TRACK_LIMIT”, pre-loaded track will not be displayed even though parameter N81000 is set to “Yes”.

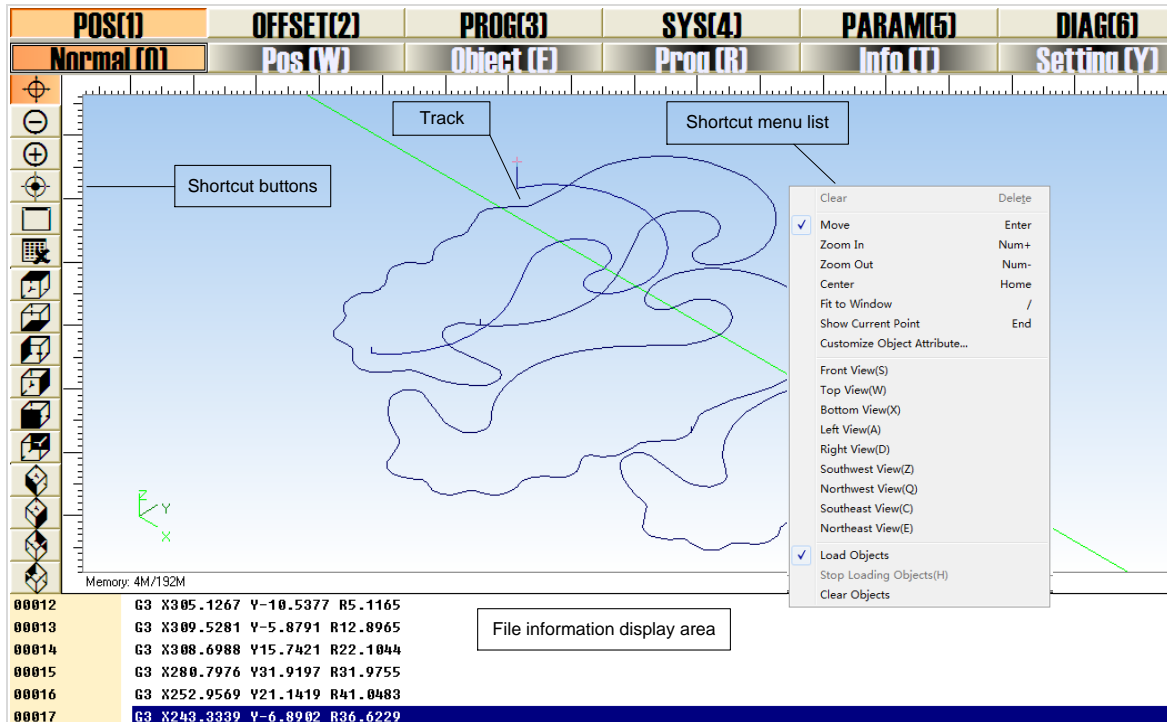





Fig. 4-19 Normal view

As shown in Fig. 4-19, right-click the mouse, and a shortcut menu list will pop up, most of which share the same effect with shortcut buttons on the left side.

Clear it is used to remove simulation track and actual machining track. Note that pre-loaded track cannot be removed by this function. As machining time goes by, track will be more and more complicated and temp file recording the track becomes larger, which will lag down system running and performance. At this time, you can remove the track by this function, with shortcut key “Delete”.

Move it is used to move the track. After this function selected, the cursor will turn to be . At this time, you can drag track to the target position by pressing left mouse button or move it by pressing the direction key on keyboard.

Zoom in/out right click the mouse and select “Zoom in/out” on the shortcut menu, or click  and  on the left side of the window to enable this function. In addition, the scroll of the mouse can also be used to zoom in/out the track.

Center it used to make track at the center of display area, with shortcut key “Home”.

Fit to window it used to show whole track in display area without adjust track position. Shortcut key is “/” on the main board.

Show current point it is used to show current machining position at the center of display area, with shortcut key “End”.

Customize object attribute

it is used to make individual setting for track mode and trajectory color. With this item selected, a dialog box will pop up, as shown in [错误!未找到引用源。](#) .

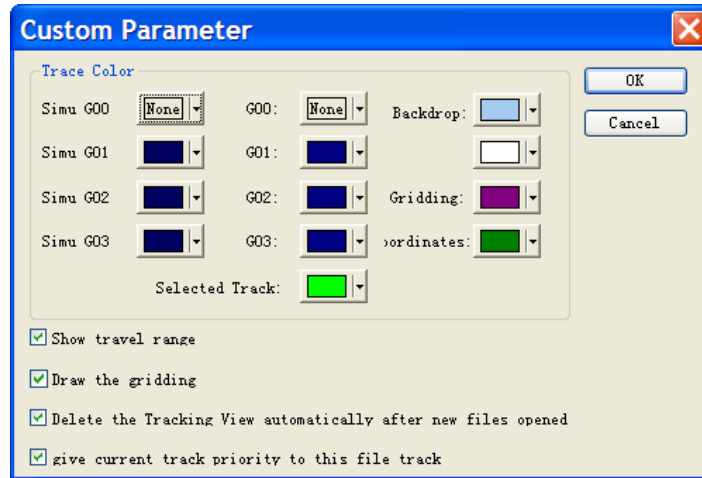
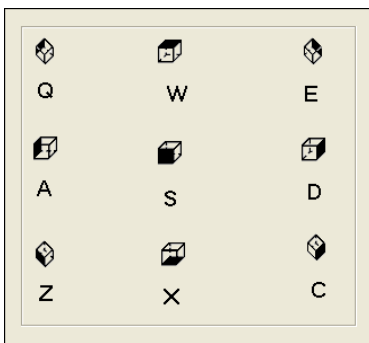


Fig. 4-20 Custom setting dialog box

View function

press shortcut key “S” in “Normal” window, and a dialog box as shown in Fig. 4-22 will pop up. You can switch to different views by corresponding shortcut keys.



View	Shortcut Key	View	Shortcut Key
Front View	S	Right View	D
Back View	B	Southwest View	Z
Top View	F	Northwest View	H
Bottom View	X	Southeast View	C
Left View	A	Northeast View	G

Fig. 4-21 View Shortcut Keys

Load object

it is used to pre-analyze the machining file and load its track in one time. In actual machining, the system will generate a tool path, which will be generally completed during the machining process. These two tracks can be told apart by different colors.

Stop loading object

it is used to stop loading the file. If program file is too big, the system will run slowly. At this time, you can use this function to stop loading. Or select “Clear Object” to clear the pre-analyzed track and stop the system continuing loading.

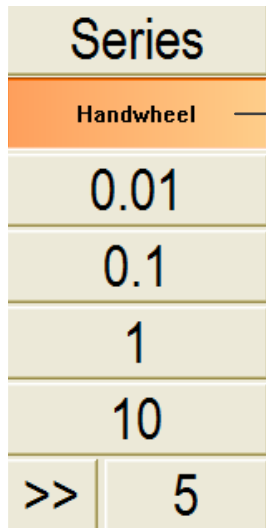
Clear object

it is used to remove the pre-analyzed track and stop the further loading. If this item is checked, “Load Object” will turn to be unchecked.

4.7 Handwheel Operation

- **Handwheel Mode**

Users can select handwheel mode by pressing “Handwheel” button under “Manual” function area, or by selecting “Handwheel” item under “Work Mode” menu.



Handwheel Mode

1. Set the relationship between the handwheel rotation direction and feeding direction by parameter “N52002”.
2. Turn the “Axis selection button” to activate an axis to move.
3. Turn the “Gear selection button” to decide the displacement of moving component (linear axis or rotary axis) against each gear of the handwheel. The displacement is determined by setup value of parameter N52003~N52005.
4. Turn the “Handwheel control rotation disk” to manipulate the movement of a tool.



- **Handwheel Guide**

In auto mode, turn to “POS” function area and click F6 “Handwheel”, or select “Handwheel Guide” item under “Operate” menu to enable this function.

Handwheel guide refers to a way of operation that the automatic execution speed of machining program is manually controlled during auto processing so as to guard against such problem as “tool damage” and dangers caused by wrongly loaded program or inappropriate tool path.

4.8 Log

This function can record all information about operations, alarms and errors. You can browse the log info since this time start-up as well as history log info. The user can enter into “Log” window as shown in Fig. 4-22, by clicking “Log” under “DIAG” menu, or sub-menu “Log” under “Window”.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Log (0)	IOPort (W)	PLC (E)			
Time	Description				
2017-01-05 14...	Find No Flash Disk Currently!				
2017-01-05 13...	Generate installation Failure!				
2017-01-05 13...	Generate installation Success!				
2017-01-05 13...	Successfully load task list!				
2017-01-05 13...	PLC program successfully load.				
2017-01-05 13...	Read and load the dynamic data file...				
2017-01-05 13...	Successfully load last machine task!				
2017-01-05 13...	Nc Studio initiates				
Time: 2017-01-05 13:49:07					
Description: Successfully load task list!					
F1	F2	F3	F4	F5	F6
Refresh	Clear	Info	Warning	Error	System
					F7
					Nowadays
					F8

Fig. 4-22 Log list

- Refresh (F1):** used to refresh the current interface, and update the logs.
- Clear (F2):** used to clear all current logs.
- Show Info (F3):** In default setting, these four buttons are always pressed down; namely the information, warning and error info will be displayed by default since the system started this time. If you want to hide any of them, press the button again.
- Show Warning (F4):**
- Show Error (F5):**
- Show Nowadays (F7):**
- Show System (F6):** used to review the system information.



Please clear logs regularly. When the record file becomes too large in size, it may lag down the system performance and responsive time.

- : Information
- : System log
- : Warning
- : Error log.

4.9 Register

The function is used to limit the usage time of the system.

Click “About NcStudio” item under “Help” menu to open a dialog box named “About NcStudio”, as shown in Fig. 4-23. If the remaining time runs out, please send the “Card No.” to manufacturer to get a registration code. After you received the registration code, click “Register” button in Fig. 4-23, and a

dialog box named “Register” will pop up, as shown in Fig. 4-24. Input the registration code and confirm to register usage time.

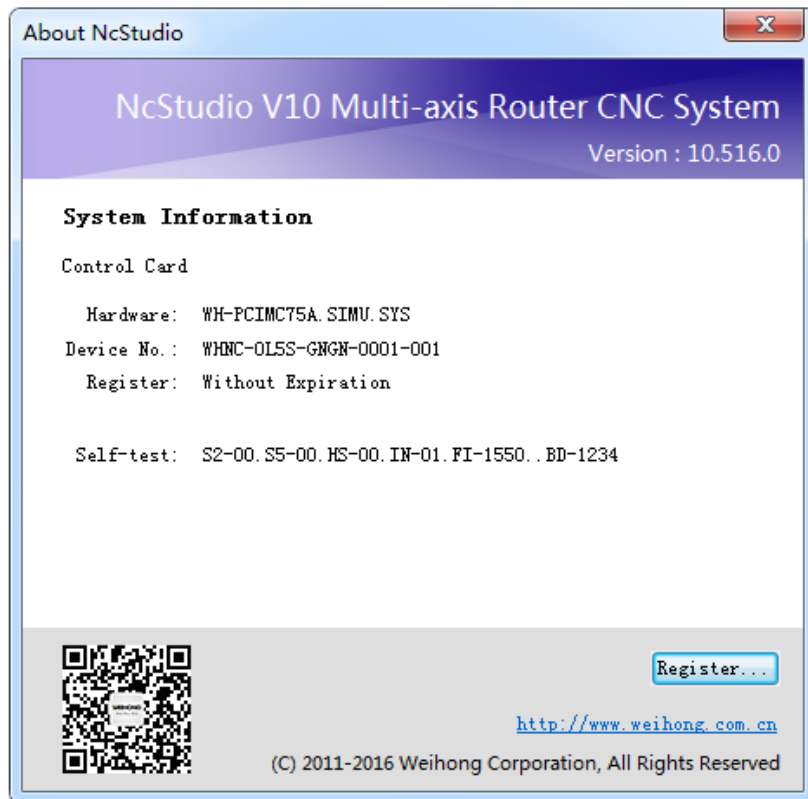


Fig. 4-23 About NcStudio dialog box

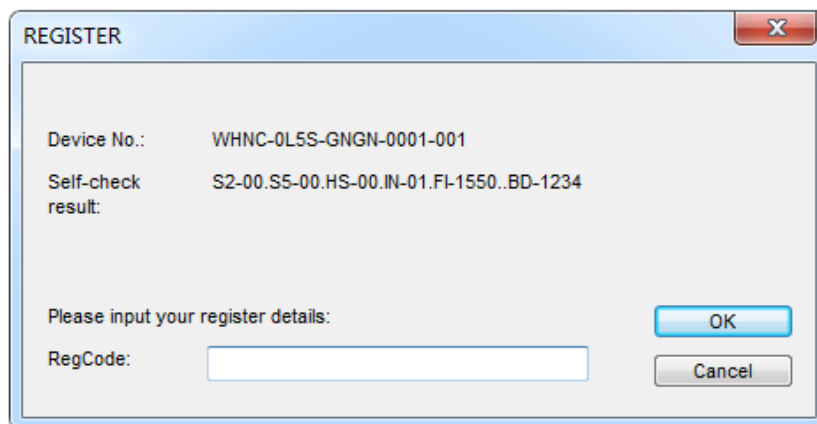


Fig. 4-24 Register dialog box

4.10 System Language and Theme

4.10.1 Language Switch-over

At present, the software supports Chinese and English. Turn to “Lang” window under “SYS” function area. Click “Select” button to set the system language, as shown in Fig. 4-25:

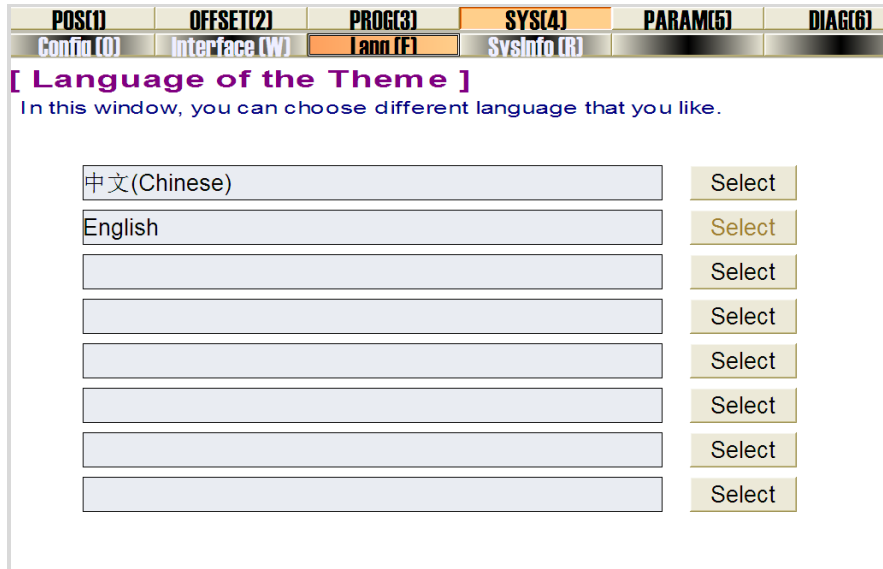


Fig. 4-25 Language selection

4.10.2 Theme Selection

There are two themes offered, black and gray theme, and the gray is the default setting.

Turn to “Interface” window under “SYS” function area, and click “Select” button to select the desired theme, as shown in Fig. 4-26.

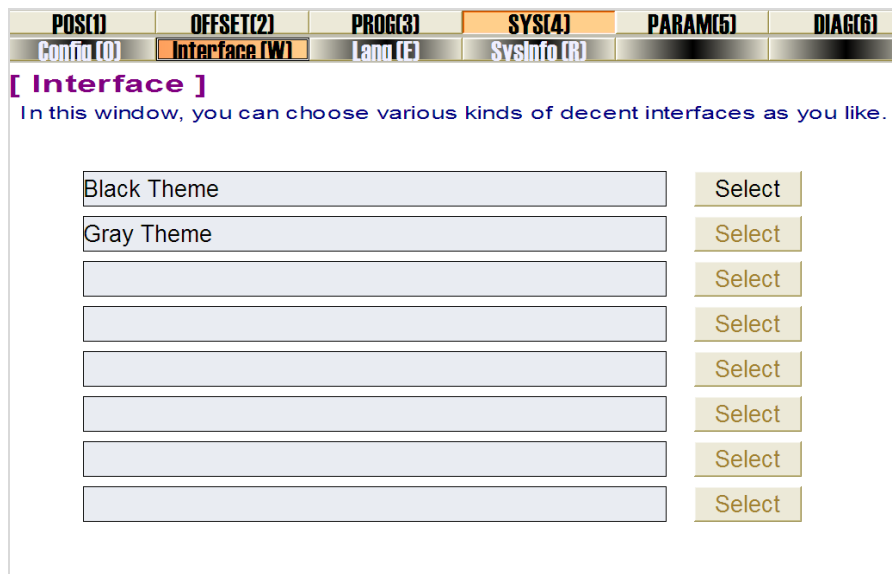


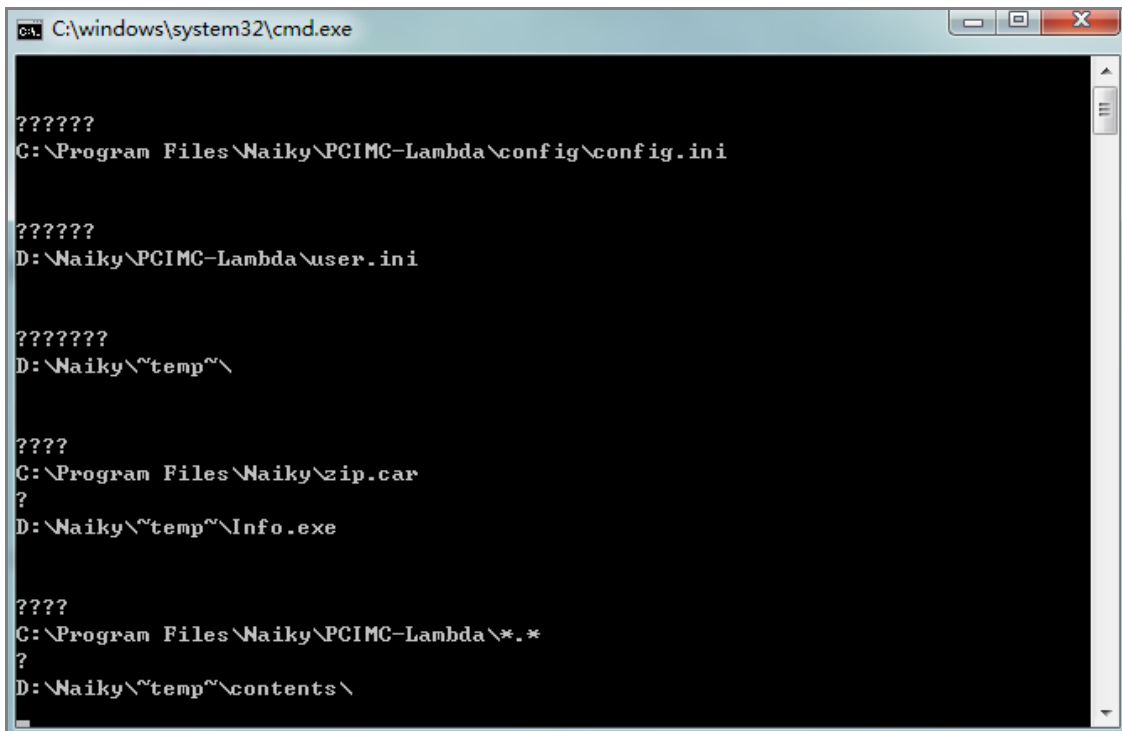
Fig. 4-26 Theme selection

4.11 System Maintenance

The software offers maintenance function, mainly used for key data backup and restoration, system installation and installation package generation, etc.

Select “Generate Installation” or “Generate Emute Installation” under “File” menu, and choose a storage path in the dialog box popped up. Click “OK” to start generate the installation package, as shown in Fig.

4-27. During this process, the system will automatically delete the temporary files and back up all files in the installation directory, including controls parameters, subroutines of common-G code, PLC program, etc.



```
C:\windows\system32\cmd.exe

??????
C:\Program Files\Naiky\PCIMC-Lambda\config\config.ini

??????
D:\Naiky\PCIMC-Lambda\user.ini

??????
D:\Naiky\~temp~\

????
C:\Program Files\Naiky\zip.car
?
D:\Naiky\~temp~\Info.exe

????
C:\Program Files\Naiky\PCIMC-Lambda\*. *
?
D:\Naiky\~temp~\contents\
```

Fig. 4-27 System maintenance dialog box

4.12 Auxiliary Function

- **Single Block**

To enable this function, you can click sub-menu item “Single Block” under “Operate” menu. When executing this function, the system will stop as the program velocity is 0. At this time, you need to click “Start” to continue machining. When the velocity turns to be 0 again, the system will enter into pause mode, namely, the last block finished, and click “start” to goes on for next block.

You can select this function before or in machining, which serves as a good support for error diagnosis and troubleshooting

- **Breakpoint Resume**

To enable this function, you can click sub-menu item “Breakpoint Resume” under “Operate” menu, or turn to “POS” function area in auto mode, click F3 “B.P. Resume”. Once this function is enabled, the system will resume machining from the position where last time machining stops.

When accidental and emergent situations happened such as power interruption, E-stop, the user can select this function to restore machining from where it is interrupted and save time.

● **Select Processing Block**

Select sub-menu item “Select Processing Block” under “Operate” menu, or turn to “POS” function area and click F5 “SelWork” to open a dialog box, as shown in Fig. 4-28, in which the user can do according settings. This function can also be used in simulation.

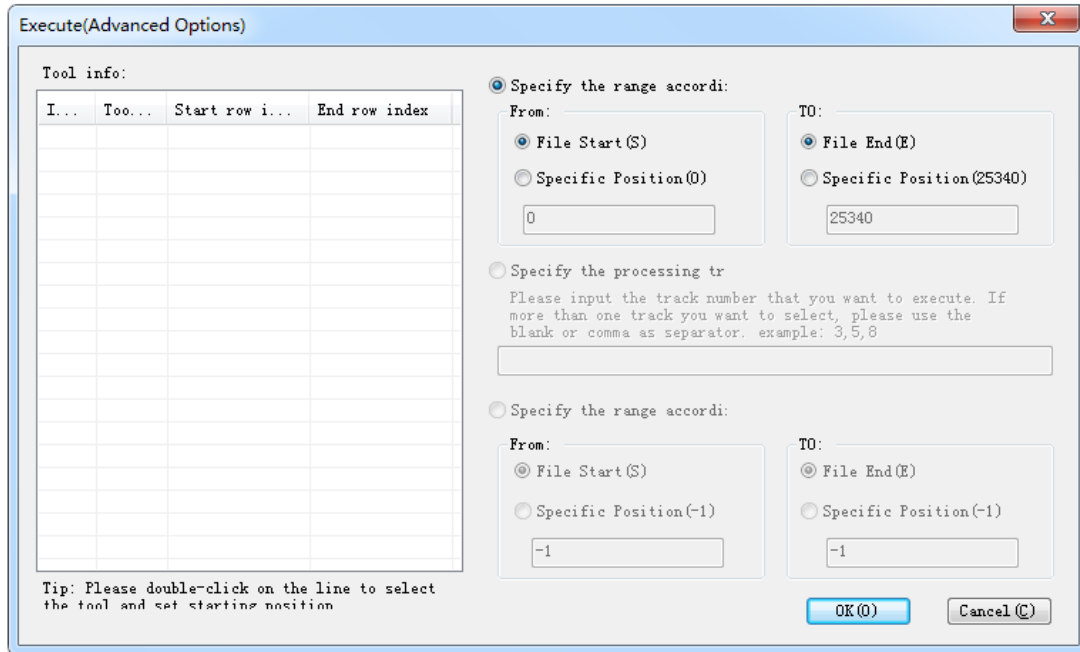


Fig. 4-28 Select processing block

Tool Info

The tool information input must be included in the machining file.

Specify the range according to the program line

If “Specific Position” is checked and the range of program lines to be machined is specified, the system will machine the lines included in this range.

Note:

If the machining file loaded is G code file, such as .nc, .g, .nce, just selection by line can be used.

If the machining file loaded is Eng code file, selection by path No. can also be used.

Specify the path No.

If this function is chosen, the system will machine all files included in the directory.

Specify the range according to path No.

If “Specific Position” is checked and the range of path to be machined is specified, the system will machine the files included in this range.

● **Mirror and Rotate**

This function can generate a mirrored or rotated program and process it.

Click sub-menu item “Circumrotate Mirror Processing” under “Operate” menu, and a dialog box as shown in Fig. 4-29 will pop up. In this dialog box, you can choose an option and enable mirror and rotating machining. The setting will take effect in current machining file.

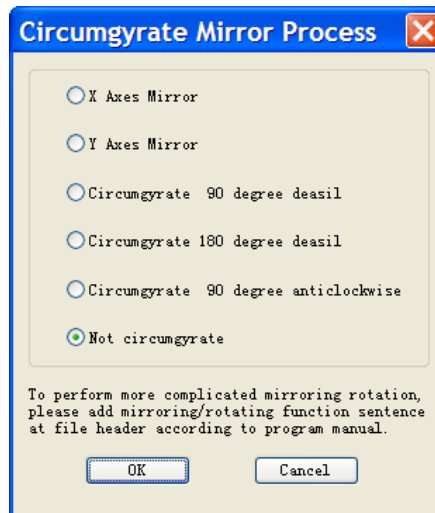


Fig. 4-29 Mirror and rotate

● **Parameter Backup and Auto Backup**

NcStudio system is entitled with automatic backup function of parameter settings. On the one hand, the user can manually back up parameter settings and restore it when it is needed. On the other hand, if the user forgets to manually back up parameter after setting, he/she can use automatic backup function to restore parameter settings from that of last time running and ex-factory setting.

In “Backup” window under “PARAM”, the user can back up the parameter settings and restore it. As shown in Fig. 4-30, after all parameters have been well set, click “Save” button to save the parameter setting. A dialog box where the user can input name for it will pop up. When any one of the parameter settings group is needed to be restored, the user can click “Load” button to read it from the settings and load it into the system.

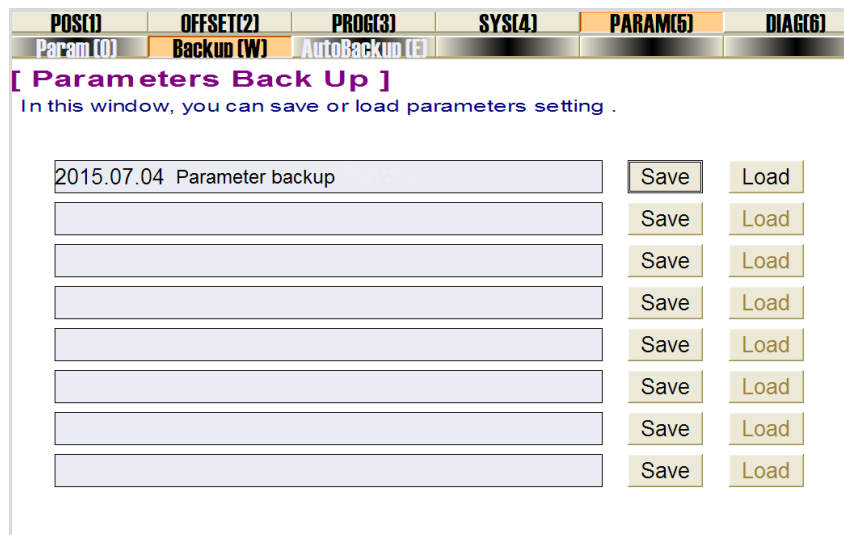


Fig. 4-30 Parameter backup

In “AutoBackup” window under “PARAM”, the user can restore parameter settings which have been saved automatically, as shown in Fig. 4-31.

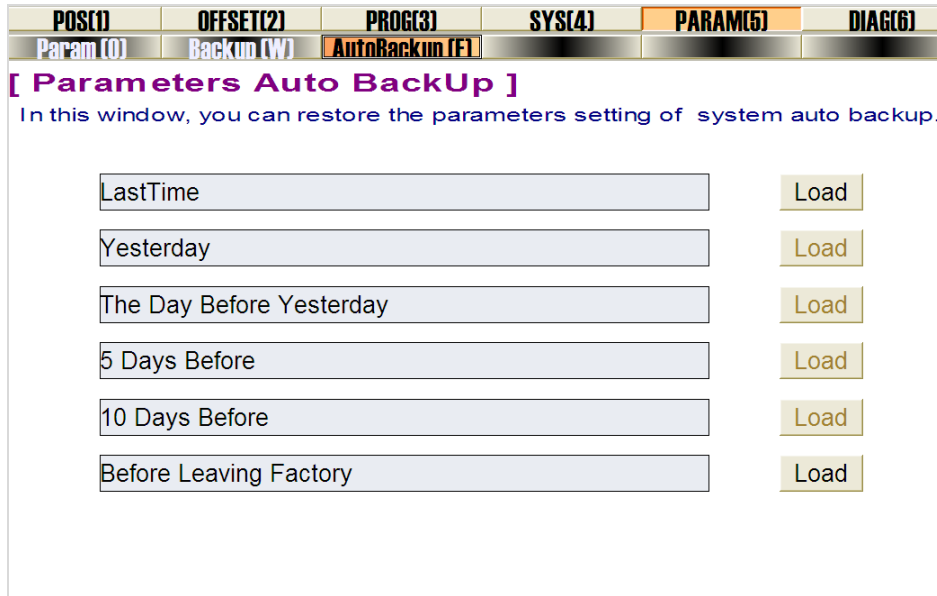


Fig. 4-31 Parameter auto backup

● **Graduation Machining**

Aiming at rotary axis, this function can realize array machining of equivalent angle (known as the graduation) when processing circular arc. The user can set the “Graduate angle”, click button “Browse” to select a source program file (or called original program file), and then click button “Generate file” to generate target file. The system will load the target file automatically after it has been saved. (Note that the tool path program can only contain program blocks for X, Y and Z axes, and the five-axis software does not have this function.)

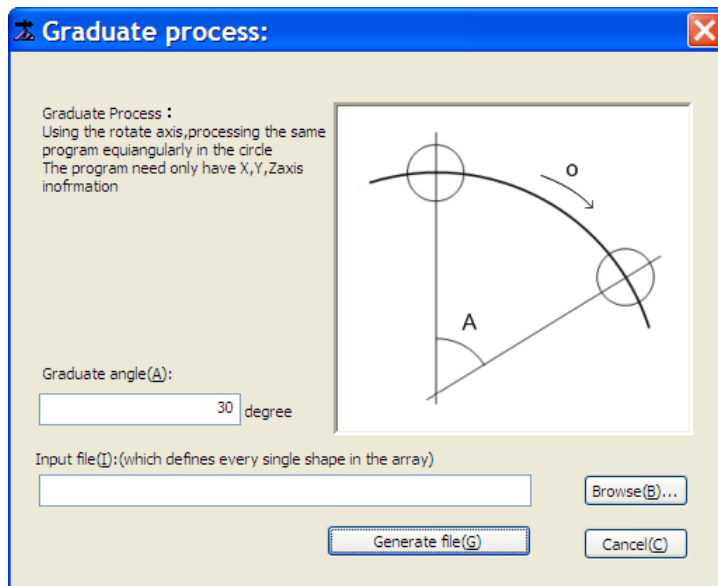


Fig. 4-32 Graduate processing dialog box

- 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 “DIAG” function area to run the PLC. Refer to *Programming Manual of NcStudio* for details. The user can view the desired information via the buttons shown on manipulation bar, including “Show Channel”, “Show Timer”, and “Shown Counter”.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)		
Loc (O)	IOPort (W)	PLC (E)					
in: 01234567890123456789012345678901			spec: 01234567890123456789012345678901				
00000			90000				
00100			90100				
			90200				
out: 01234567890123456789012345678901			90300				
10000 1111.....1..1.....			90400				
10100			905001.....				
			90600				
temp: 01234567890123456789012345678901			90700				
20000			90800				
20100			90900				
----- timers -----							
00[0,] [0,] [0,] [0,]							
04[0,] [5000, 5000] [5000, 0] [0,]							
08[0,] [0,] [0,] [0,]							
0C[0,] [0,] [0,] [0,]							
----- counter -----							
00[0,] [0,] [0,] [0,]							
04[0,] [0,] [0,] [0,]							
08[0,] [0,] [0,] [0,]							
0C[0,] [0,] [0,] [0,]							
F1	F2	F3	F4	F5	F6	F7	F8
Show Channel	Show Timer	Show Counter					

Fig. 4-33 PLC

4.13 Tool Lifting and Plunging for Five Axes

Tool lifting and plunging function only could be applied to XYZAC-type five-axis control system. This function can protect workpiece from possible damages caused by “Pause” or “Stop” during machining.

To use this function, the user needs to set manufacturer parameter “N95101 Five Axis Carrying Knives_Axis Dir”, and set “N95100 Use Five Axis Carrying Knives” to “Yes”. After it enabled, the machine tool will automatically do tool lifting and plunging when the user executing “Select Processing Block” command, or doing some operations such as “Pause” → “Start”, “Stop” → (operations) → “Resume” during normal machining process.

The following part will take the process “Normal machining” → “Pause” → “Start” as an example to briefly introduce this function.

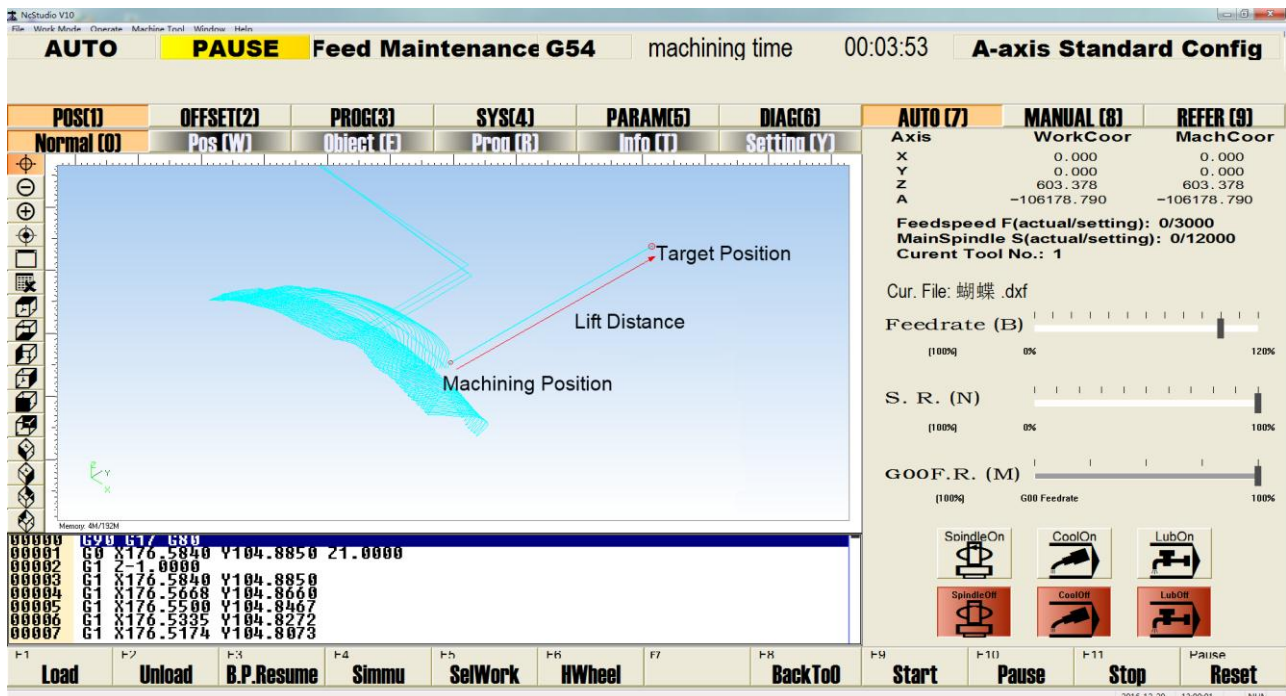


Fig. 4-34 The process of tool lifting and plunging

As shown in Fig. 4-34, when clicking “Pause” during machining process, the tool will lift along the direction set by parameter N95101. The moving distance is the value set by “N73004 Z_OFFSET_ON_PAUSE”. When the “Start” button is clicked again, the tool will move from the lift target point to the machining position on pause to continue machining.

● Related Parameters (of MER's access)

Parameters		Description	Setting Range
N95100	Use Five Axis Carrying Knives	Enable or not.	Yes; No
N95101	Five Axis Carrying Knives_Aixs Direction	The moving direction when the five axes carrying knives.	1: Positive direction -1: Negative direction
N73002	PAUSE_OPTION	The lifting options of Z axis when pause is specified.	0: Lift up a specified value; 1: Go to specified workpiece coordinate; 2: Go to specified machine coordinate
N73000	PAUSE_DOWN_VOL	The downward speed of Z-axis at previous cutting point after pause.	0~100000
N73001	PAUSE_UP_VOL	The lift speed of Z-axis while entering a pause.	0~100000
N73004	Z_OFFSET_ON_PAUSE	The lift distance of Z-axis when a pause is specified. The parameter is valid only when the value is 0.	0~500



1. In the Normal machining state, clicking “Pause” or “Stop”, the machine tool will automatically execute tool lifting command. While clicking “Start” after “Pause” pressed, or clicking “B.P. Resume” after “Stop” pressed, or choosing “Select Process Block”, the machine tool will automatically execute tool plunging command.
2. In Idle state, when executing “B.P. Resume” and “Select Process Block” functions, the tool will moving to “PAUSE_OPTION” according to G00 instruction. At this time, operators need to ensure the safety of this track. Please note that the action of plunging is opposite to that of the lifting.

5 Parameter Setting

Parameters can be divided into parameters of operator's access, of manufacturer's access and of developer's access according to limits of authority. In addition, manufacturer parameter needs manufacturer password; developer parameter needs developer password, which is only for developer use.

Turn to "Param" window under "PARAM" function area, double click on the parameter to open an input box, and modify the value. Please note all parameters should be modified in idle state.

● Parameter List of Operator's Access

The operator parameters are almost same in four-axis and five-axis software, except the axes number and name. Table below lists all parameters of operator in XYZA four-axis software.

No.	Name	Setting Range	Default	Effective
1.0. Spindle general				
N10069	Rev Axis Prog Unit	0;1	0	Immediately
	The measurement unit of the data in machining files under turn-table mode. 0: Angle (unit: degree); 1: Surface distance of the rotary workpiece (unit: mm).			
N10070	Rotative Workpiece Radius	0~99999(mm)	50	Immediately
	Under turn-table mode, the diameter of the workpiece to be machined.			
3.2. Tool setting				
N32020	NAME	~	~	Immediately
	Name of cutter. Maximal length is 127 letters or the common length of character.			
N32021	DIAMETER	0~9999	0	Immediately
N32022	LENGTH	0~9999	0	Immediately
N32023	DIAMETER WEAR	- 9999~9999	0	Immediately
N32024	LENGTH WEAR	- 9999~9999	0	Immediately
N32030	OFFSET(X/Y/Z/A)	- 9999~9999	0	Immediately
Note: up to 8 tools can be supported in the software, only parameters related to T1 have been listed above, and parameters related to other tools are the same with that of T1.				
4.1. Lube				
N41003	Mach Task End Inform Type	0;1;2	0	Immediately
	The information types after the end of machining task. 0: Red lamp is out; 1: Red lamp lights for 3 seconds; 2: Red lamp lights until any input from mouse or keyboard.			

No.	Name	Setting Range	Default	Effective
6.2. G code options				
N62020	IJK_INC	Yes; No	Yes	Immediately
	The value of the IJK addresses is increment corresponding to the circle center in a circle programming.			
N62022	Tool Selection for G Code	Yes; No	No	After reloading
	When processing G code (or files can be translated to G code except ENG and NCE), tool can be selected if there are multiple tools.			
N62730	G73_G83_SAFE_HEIGHT	-99999~99999(mm)	0	Immediately
	Specifying the retract distance after each feed in the G73_G83 cycle drilling command.			
N62760	DRILL_STOP_DIR	0/1:(G17:+X/-X) 2/3:(G17:+Y/-Y)	0	Immediately
	Only valid in X-Y (G17) plane.			
N62765	T0 Process mode	0;1	0	Immediately
	0: Invalid; 1: Without tool.			
6.4. Speed/Acc				
N64020	RAPID_TRAVEL_FEEDRAT E	0~100000(mm/min)	5000	Immediately
	The default speed of beeline axis when positioning.			
N64021	DEFAULT_FEEDRATE	0~100000(mm/min)	3000	Immediately
	The default speed of beeline axis when machining. Its value should not be larger than that of N64020			
N64030	REV_RAPID_TRAVEL_FEE DRATE	0~100000(rpm)	10	Immediately
	The default speed of rotary axis when positioning.			
6.5. File translation				
PLT_FORMAT_CONVERT_PARAM				
N65000	TOOL_UP_HEIGHT	0~99999(mm)	1	After reloading
	Upward height of the tool while dry run.			
N65001	PLT_UNIT	0.001~99999	40	After reloading
	Length in millimeter (or inch) per PLT unit. 1plt=40mm by default.			
N65002	TOOL_DISTANCE_AT_ARE A	0.0001~99999(mm)	0.025	After reloading
	Distance between tools while machining in PLT-area.			

No.	Name	Setting Range	Default	Effective
N65003	CUTTING_DEPTH	-99999~0	-1	After reloading
	Machining depth of 2D files.			
DXF_FORMAT_CONVERT_PARAM				
N65100	TOOL_UP_HEIGHT	0~99999(mm)	1	After reloading
	Upward height of the tool while dry run.			
N65101	CUTTING_DEPTH	-99999~0(mm)	-1	After reloading
	Machining depth of 2D files.			
N65102	DEPTH_LAYER	-99999~0(mm)	0	After reloading
	Machining depth of each tool layer in 2D files.			
N65103	FIRST_POINT_AS_ORIGIN	1;2;3	1	After reloading
N65104	ENABLE_MACHINE_INDIVI DUALY	Yes; No	No	After reloading
	Machining only one shape each time and go to next shape only after last one finished.			
N65105	ENABLE_MACHINE_BOTT OM	Yes; No	No	After reloading
	[3D Cutting] Valve operation is performed only when cutter arrives at workpiece surface.			
ENG_FORMAT_CONVERT_PARAM				
N65200	TOOL_UP_HEIGHT	0~99999(mm)	1	After reloading
	Upward height of the tool while dry run.			
N65201	ENABLE_CHANGE_TOOL_ PROMPT	Yes; No	Yes	After reloading
	When it meets tool exchange in ENG file, pause and remind to change tool.			
N65202	Recycle Machining Times	1~9999	1	After reloading
	Machining times that needs to be cycled while machining ENG file.			
N65203	Machining ENG File By Tool Number	Yes; No	No	After reloading
	With this function, ENG file machining can be executed by tool selection according to the number specified.			
N65207	Modify By Tool Number	Yes; No	No	After reloading
	With this function, ENG file machining can be executed by specified tool number.			
N65208	Z Up Type After Drill	0;1	0	Immediately
	Z Up Type After Drill, 0: Up to R plane; 1: Up to specified workpiece coordinate position, exclusively allowed in ENG file.			

No.	Name	Setting Range	Default	Effective
N65209	Z Pos After Drill	-1000~1000(mm)	10	Immediately
	Z Up Type After Drill is one, and then lifts up to this workpiece coordinate position.			
6.6. Change tool				
N66000	PROMPT_CT_CMD	Yes; No	No	Immediately
	Whether to suspend the system and to prompt users while meeting the change tool instructions, only using in union mode.			
N66011	CHANGETOOLAHEADPOS (X/Y/Z/A)	-99999~100000(mm)	0	Immediately
	Low speed position before get into the tool slot.			
N66019	Change Tool Speed	0~100000(mm/min)	3000	Immediately
	Spindle speed when changing tool.			
N66020	Change Tool Speed	0~100000(mm/min)	1800	Immediately
	The default speed when Z axis moves from the upper position to lower position in changing a tool.			
N66042	Pause And Prompt While Change Tools	Yes; No	No	Immediately
	Pause and prompt while change tools.			
N66067	Tool Y Dir Space Between (XYZ)	-9999~9999(mm/min)	0	Immediately
Note: up to 21 tools can be supported in the software, only parameters related to T1 have been listed above, and parameters related to other tools are the same with that of T1.				
7.1. Manu				
N71000	JOG_VOL	0.001~100000(mm/min)	3000	Immediately
	The default velocity under jog mode.			
N71001	RAPID_JOG_VOL	0.001~100000(mm/min)	5000	Immediately
	The velocity under rapid jog mode.			
N71002	REV_JOG_VOL	0~100000(rpm)	5	Immediately
	The default velocity of Rotate Axis under jog mode.			
N71003	REV_RAPID_HOG_VOL	0~100000(rpm)	10	Immediately
	The velocity of rotary axis under rapid jog mode.			

No.	Name	Setting Range	Default	Effective
7.2. Auto				
N72001	SPEED_ASSIGN_TYPE	0;1;2	0	Immediately
	Specify what feeding speed will be selected. 0: Use file speed; 1: Use default speed; 2: Specify speed proportionally.			
N72002	IGNORE_PROG_SPINDLE _REV	Yes; No	No	Immediately
	If true, the system will ignore the rotational speed specified by the file and use the default speed specified by SPIND_VELO_DEFAULT.			
N72004	STOP_SPIND_AT_END	Yes; No	Yes	Immediately
	Whether to stop the spindle after a task. Note: if parameter "Auto Stop Spindle When Pause" is set to "Yes", the spindle will be stopped after a task stops regardless of the parameter value.			
N72005	MACH_DEPTH	-99999~0(mm)	-1	Immediately
	Machining depth of 2D file.			
N72006	HEIGHT_Z_RAISE	0.001~99999(mm)	1	Immediately
	The lift height of Z axis during dry run.			
7.3. Pause				
N73000	PAUSE_DOWN_VOL	0~100000(mm/min)	600	Immediately
	The downward speed of Z axis at the previous cutting point after a pause.			
N73001	PAUSE_UP_VOL	0~100000(mm/min)	600	Immediately
	The lift speed of Z axis while entering a pause.			
N73002	PAUSE_OPTION	0;1;2	0	Immediately
	The lifting options of Z axis action when pause is specified. Values: 0: lift up a specified value; 1: go to a specified position in WCS.			
N73003	Z_WPCOOR_ON_PAUSE	0~9999(mm)	10	Immediately
	The value is used to specify the workpiece coordinate of the stoppage position of Z axis when a pause is specified. The value is valid only when PAUSE_OPTION is 1.			
N73004	Z_OFFSET_ON_PAUSE	0~500(mm)	10	Immediately
	When a pause is specified, the value is used to specify the lift height of Z axis corresponding to the previous height before the pause. The value is valid only when PAUSE_OPTION is 0.			
N73005	STOP_SPIND_AT_PAUSE	Yes; No	Yes	Immediately
	Whether the spindle can automatically stop after a pause.			

No.	Name	Setting Range	Default	Effective
N73006	Z_MACHCOOR_ON_PAUSE	-99999~99999(mm)	0	Immediately
	The value is used to specify the machine coordinate of the stoppage position of Z axis when a pause is specified. The value is valid only when PAUSE_OPTION is 2.			
7.4. Bkref				
N74001	NEED_REFPT_BEFORE_MACHINING	Yes; No	Yes	Immediately
	Whether to back to the machine origin before machining.			
N74101	Auto Back To Zero	Yes; No	No	Reboot
	Whether to back to zero automatically.			
7.5. Measure				
N75210	FIXED_CALI_POS (X Y Z)	-99999~99999(mm)	X/Y: 0 Z: -1	Immediately
7.9. Operation others				
N79000	Z_DOWN_VELO_OPTION	0;1;2	0	Immediately
	The options of Z down speed, including 0: not disposed; 1: direct-Z-motion only; 2: general Z-down motion.			
N79001	Z_DOWN_VELO	0~100000(mm/min)	480	Immediately
N79003	SAFE_HEIGHT	0~1000(mm)	10	Immediately
	The safe height to avoid collisions in WCS, which is used after returning to the machine origin or breakpoint resume.			
N79100	IS_FIXED_VALID	Yes; No	Yes	Immediately
	Whether to back to the fixed point after a program.			
N79110	FIXPT_POS X/Y/Z/A	X/Y/Z: -99999~99999(mm) A: -99999~99999(deg)	0	Immediately
N79200	Display Simu Out Range	Yes; No	Yes	Immediately
	Whether to prompt the user if simulation is out of travel range.			
8.1. Position view				
N81000	AUTO_LOAD_TRACK	Yes; No	Yes	Immediately
	Whether to parse the file automatically after load. .			

No.	Name	Setting Range	Default	Effective
N81001	AUTO_LOAD_TRACK_LIMIT	0~100000 KB	1000	Immediately
	The file size limit. A machining file can be automatically loaded only when the file size is less than this value.			
8.3. Door Open Parameters				
N83014	Back to origin mode	0;1	1	Immediately
	Back to origin mode. Available value: 0: Three axes back to origin; 1: All axes back to origin.			
N83015	Z Axis Position when back to origin	0,1	0	Immediately
	Z axis position when return to the reference point. Available options: 0: Back to safe height; 1:Back to position #AREAMAX.Z-1.			

6 Precautions and Troubleshooting

6.1 Common Troubleshooting

6.1.1 What should users do if the spindle does not rotate?

1. Start spindle, and check if the spindle start indicator lamp on the controller is on.
2. If it lights, measure if the SPIN (Y01-C01) port is conducted and the analog voltage output is normal between AVC and GND with a multimeter. If it is conducted and normal, check whether the parameter setting of inverter is right, whether the spindle and inverter have been damaged, or whether the wiring of the spindle and inverter is correct.
3. If not, close the host machine and power off machine tool, and then re-plug the connection cable of terminal board. If it still does not light, please change another Lambda controller or PM85A/95A communication card.

6.1.2 What should users do if an axis does not move?

1. Check if there is output (in green) for “×servo enable” signal of output port in “IO Port” window under “DIAG” functional area. If there is output, the software works normally. Check if the port polarity (it should be NO “N”) is set correctly.
2. Check if the parameter setting of servo driver is correct (like setting control mode as position control, selecting pulse input port for Panasonic driver, etc.).
3. Check if the servo cable of this axis is well contacted at the joint with system host machine and servo driver.
4. Check if the motor is at “SRV-ON”.
5. Check if there is something wrong with servo driver, motor cable, servo cable or control system (e.g. exchange servo cable and servo driver with those of other axes working normally).

6.1.3 What should users do if servo motor Z brake can't be opened?

1. Check if there is signal for Z axis brake input. If not, check whether the servo driver is enabled or not, or parameter related to brake output are correctly set or not.
2. If the signal exists, remove wiring between brake output terminals (Y00-C00), and then start the system and power up machine (ruling out system alarm signal) and test the conduction between the terminals with a multimeter. If not, check the host system; if yes, brake output normally works.
3. Power off the machine tool, re-connect the two wires which have been removed above (connect 24V power supply). Check if the voltage across brake wire of the motor with a multimeter is 24V or not. If yes, the motor is damaged.
4. If the problem remains unsolved, please change another Lambda controller.

6.1.4 What should users do if machine tool returns to the machine origin abnormally?

- Limit alarm or servo driver alarm occurs during returning to the machine origin.
- 1. Observe the port "x machine origin" in "IO Port" window under "DIAG" functional area, and secure that the polarity is consistent with the home switch signal type (N for normally open / P for normally closed).
- 2. Check if the software can receive the REF. point signal of this axis. The method is: trigger the home switch, and then see if the color of the dot before the "x machine origin" changes from red to green in "IO Port" window under "DIAG" functional area. If there is no color change, it indicates the software can't receive the REF. point signal, needing to check if there is an error in the home switch or in the wiring of home switch. To see if the system failure occurs, make the REF. point signal on the controller and COM port into conduction directly with a conducting wire, and then check whether the color of the dot before "x machine origin" changes in "IO Port" window.
- 3. Check whether settings of parameter "N74020 COARSE_LOCATING_DIR" and "N74080 BACK_DISTANCE" are correct or not, whose settings should be opposite to each other.
- 4. Check whether the position of home switch is appropriate to avoid the following three situations: the distance between home switch and limit switch is too small; the home switch is installed behind the limit switch; or the position of home switch is out of the mechanical stroke of machine tool.
- When backing to the machine origin, the machine tool motions towards a certain direction at a relatively low speed (ten percent of the speed of coarse positioning) until limit is triggered.

See if the polarity of "x machine origin" input port is correct in "IO Port" window under "DIAG" functional area. When this home switch is triggered, i.e. if there is signal input; the color of the dot should be green. Otherwise, it is red.

- A certain axis moves a very long distance or keeps moving at a rather low speed towards the reverse direction after coarse positioning during backing to machine origin.



The cause of the above phenomenon is that the system can't detect the encoder REF. point signal of this axis. Do as follows:






1. Move the machine manually and check if the encoder zero signal in "IO Port" is activated or not;
2. See if the servo cable of this axis is well contacted at the joints with Lambda controller and servo driver;
3. Check if there is an error in the driver, motor, encoder cable, servo cable or the CNC system (e.g. exchange servo cable and servo driver with those of other axes able to return to the machine origin normally in turn).

6.1.5 What should users do if the machine tool motions upward after arriving at the position of tool sensor during calibration?

1. View and tell whether the polarity of “Cut signal” is right in “IO Port” window under “DIAG” functional area. The color of the dot before the “Cut presetter” signal is red when the system does not receive tool sensor signal.
2. Test the polarity of “Cut signal” change in “IO Port” window under “DIAG” functional area by manually touching the tool sensor. If there is no change, the tool sensor may be damaged.

6.2 Alarm Information

Type	Warning 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. Check the tool path program file. Modify parameter settings of N10020 and N10030 to enlarge the workbench travel limit. (see chapter 4.1)
	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 “IO Port” window under “DIAG” functional area, and modify the port polarity. (see chapter 3.2.2)
		X axis runs into limit switch directly during motion.	Manually move X axis away from limit switch.
		There is an error in limit switch itself.	Check if limit switch works normally.

Type	Warning Content	Cause	Solution
 Servo alarm	Servo alarm of X (Y/Z) axis	The polarity of X axis servo alarm port is wrong.	Enter "IO Port" window under "DIAG" functional area, and modify the port polarity. (see chapter 3.2.2)
		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 "IO Port" window under "DIAG" functional area, and modify the port polarity. (see chapter 3.2.2)
		The E-stop button is pressed.	Turn the E-stop button clockwise to make it bounced.
 Spindle alarm	Spindle alarm	The polarity of spindle alarm port is wrong.	Enter "IO Port" window under "DIAG" functional area, and modify the port polarity (see chapter 3.2.2)
		There is an error in inverter.	Find the reason based on the alarm type of inverter.
 File error alarm	No NC file. Please load a file into the code interpreter first.	Start file machining with no file loaded in the system.	Load a machining file before start machining.
 Terminal board not connected error alarm	Terminal board not connected.	Cable has not been firmly connected or there is error in Lambda controller.	Re-plug the cable and restart the software to observe the occurrence of the error; Polarity of port is wrong. Invert the port polarity and restart the software; Analyze possible causes according to the status of indicator "SYSTEM" on Lambda controller; Change another Lambda controller.

7 Appendix

7.1 Shortcut Keys List

Shortcut Key	Function	Shortcut Key	Function
Global shortcut keys			
1 (Main keyboard)	Show "POS" functional area	2 (Main keyboard)	Show "OFFSET" functional area
3 (Main keyboard)	Show "PROG" functional area	4 (Main keyboard)	Enable "SYS" functional area
5 (Main keyboard)	Show "PARAM" functional area	6 (Main keyboard)	Show "DIAG" functional area
7 (Main keyboard)	Show "AUTO" operational panel	8 (Main keyboard)	Show "MANUAL" operational panel
9 (Main keyboard)	Show "REFER" operational panel	Q	Show "Normal" window
W	Show "Pos" window	E	Show "Object" window
R	Show "Prog" window	T	Show "Info" window
Y	Show "Setting" window	Ctrl+O	Open and load
Ctrl+U	Unload a program file	Ctrl+N	Create
Ctrl+P	Edit the currently being processed program	Alt+F4	Exit
Pause	Reset	Shift+F6	Set current point as the workpiece zero (WCS zero)
F1~F8	Buttons corresponding to each window	F9~F11	Buttons corresponding to each operational mode
Shortcut keys for "Normal" and "Object" windows			
Home	Center	End	Show current machining point
+	Zoom in	-	Zoom out
/	Fit to window size	Delete	Clear view
S	Front view	W	Top view
X	Bottom view	A	Left view
D	Right view	Z	Southwest view
S --> Q	Northwest view	C	Southeast view
S --> E	Northeast view		

Shortcut Key	Function	Shortcut Key	Function
Shortcut keys for “Manual” window			
1 (mini-keyboard)	Z- (In jog and increment mode)	2 (mini-keyboard)	Y- (In jog and increment mode)
3 (mini-keyboard)	A- (In jog and increment mode)	4 (mini-keyboard)	X- (In jog and increment mode)
5 (mini-keyboard)	Activate rapid jog speed	6 (mini-keyboard)	X+ (In jog and increment mode)
7 (mini-keyboard)	Z+ (In jog and increment mode)	8 (mini-keyboard)	Y+ (In jog and increment mode)
9 (mini-keyboard)	A+ (In jog and increment mode)		
Shortcut keys for [Auto] window			
F9	Start	F10	Pause
F11	Stop	Ctrl+F9	Advanced start (also called selective machining)
Shift+F9	Resume from the interrupted point		

7.2 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.