NcStudio V10 Milling CNC System

Users' Manual

3rd Edition

(For dual Z axes)

Weihong Electronic Technology Co., Ltd.

Third Edition, February, 2016

XX Printing, XX, XXXX

92 pages

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 8 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: an overview of the product, namely Chapter 1. It introduces the system characteristics and some important concepts such as the coordinate system, operation modes as well as operation status.
3) Part 3: Chapter 2, introducing the basic configuration and installation of the system.
4) Part 4: Chapter 3 & 4, introducing the operational interfaces and menus of the NcStudio V10 software.
5) Part 5: Chapter 5, introducing operation steps and relative precautions.
6) Part 6: including Chapter 6~8. Chapter 6 is about parameter settings, Chapter 7 is the shortcut keys and Chapter 8 is the software license agreement.

Applicable Product Models

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

Product Model	Remarks
NcStudio V10 Milling CNC System	NcStudio V10 milling CNC system is independently developed by Weihong Electronic Technology Co., Ltd., with copyright reserved. This system can directly support G code and PLT code format files generated by various CAD/CAM software, such as UG, MasterCAM, CASMate, Art CAM, AUTOCAD and CorelDraw. The system is used together with NC65A/65B integrated CNC system and PM85A/95A communication cards on most occasions.

Contact Us

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

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

Revision History

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

Date	Edition	Revision
2016.02	R3	Contact information updated.
2015.07	R2	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 for plugging cable connected with the device and the ventilation & heat dissipation in the cabinet;
- Space between this device and other equipments should also be preserved according to the requirements;
- The product should be installed firmly and without vibration. During installing, casting, knocking, striking, or loading on the product is forbidden;
- To reduce electromagnetic interference, power-supply components used should be above AC or DC 50V and the space between cable and CNC device should be preserved above 100mm;
- It will be better if CNC device is installed at a position facilitating debugging and maintenance.

3) Precautions Related to Wiring

- Only qualified people are allowed to participate in the wiring and checking;

TO BE CONTINUED

WARNING

CONTINUE

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

4) Precautions Related to Running & Debugging

- Parameters setting should be checked before running, since wrong setting may lead to accidental movements;
- Modification to parameters should be within the allowable range, or such breakdowns as unsteady running and machine damage will occur.

5) Precautions in Use

- Before power-on, please make sure that the switch is on blackout to avoid occasional start-up;
- Please check the electromagnetic compatibility during electrical design in order to avoid or reduce electromagnetic interference to the CNC device. A low pass filter should be employed 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. About NcStudio

NcStudio V10 milling and engraving CNC system is independently developed by Weihong Electronic Technology Co., Ltd., with copyright reserved. This system can directly support G code and PLT code format files generated by various CAD/CAM software, such as UG, MasterCAM, CASMate, Art CAM, AUTOCAD and CorelDraw.

NcStudio is based on the PC operating system and boasts of stable interfaces which are intuitive and user-friendly.

This system can be applied to various three-axis routers and engraving & milling machines, also can be used in complex mold machining, advertising, decorating and cutting industries, etc.

1.1. System Characteristics

This system has the following functions:

1) It is equipped with four motion axes, namely X/Y/Z1/Z2 axes. The binary input & output points and analog outputs can be further extended. The above configuration varies with different choices of machine tool builders.

2) Auto processing supports G code of ISO standard, HP PLT format, DXF format, JDPaint ENG format and various formats built by popular CAD/CAM software at home and abroad, like UG, Pro/E, MasterCAM, Cimatron, CASMate and ArtCAM, etc.

3) It supports manual operation, like jog mode, stepping (increment) mode and handwheel mode. In these modes, the user is entitled to manipulate the machine with the help of machine tool input equipment like hand-held device or with the help of computer input device such as keyboard and mouse.

4) It supports array processing: the user can repeatedly carry out a program machining according to columns and arrays pre-defined.

5) It supports rotating & mirroring processing. This function can mirror and rotate a program with the workpiece origin as the center.

6) It supports stepping function or incremental feeding: the user can set the precise feeding distance and adjust the step length value.

7) It supports user data input function: the user can input G code online and execute it right away.

8) It supports advanced processing instructions. Inputting a few parameters will suffice to perform bottom milling and frame milling.

9) It supports single step mode, also known as single block function: the user can execute a machining task in single step mode, which can serve as a good support for error diagnosing and troubleshooting.

10) Advanced auto functions such as breakpoint resume (resume machining from the interrupted point) and advanced start (start machining from a pre-defined program line as desired) are supported.

11) High precision during motion axes returning to the machine origin (also known as the reference point).

12) It supports automatic tool measurement function, including fixed calibration, mobile calibration and auto centering, etc.

13) It supports workpiece field save/restore function. In case of sudden power failure, the system has been designed to prevent any system file damage caused by sudden power interruption. Functions like breakpoint resume, backing to machine origin precisely, etc, also guarantee machining field restoration after the system is restarted.

14) It supports real-time adjustment of feedrate override. The user can adjust feedrate override freely during machining. The minimum value is 0, equivalent to a pause in processing and the maximum value is 120%.

15) It supports high-smooth speed connection function. In general, the connecting speed between two G codes usually is a fixed value, such as 0 or a very small value. However, by adopting machining speed adaptive forecasting algorithm, our system can decide the connecting speed between the current code and the next code by taking the connecting speed, its direction, and the maximum acceleration speed into consideration and by employing forward predicting function. This not only improves machining efficiency (from 30% to 300%), but also enhances the machining performance by eliminating speed chatter marks left on the surface of the workpiece.

16) It supports three-dimensional simulation display function. With simple operation, the user can observe the three-dimensional machining result from various orientations to understand it more accurately and more intuitionally.

17) It supports simulation function. This function supports rapid simulation machining which can be finished in an extremely short period of time. Besides the user can check the machining file and result and learn the actual processing time.

18) It supports powerful and agile keyboard operation. The new system strongly supports keyboard operation and thus can fulfill the user’s needs in operation.

19) It supports log function. The powerful log function can help the user check the detailed processing information and system diagnosis.

20) Built-in machining file manager: the user only needs to save the machining files into the designated directory and NcStudio will manage these files in a built-in manager.

21) Built-in file editor: With which, the user can load a machining file into the editor for editing and modifying.

22) It displays file machining information; through simulation or actual machining, machining information window can help the user get such important information as executing time and processing range.

23) It supports auto parameters backup function. The user can back up the parameter settings and recover them when necessary.

1.2. 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.

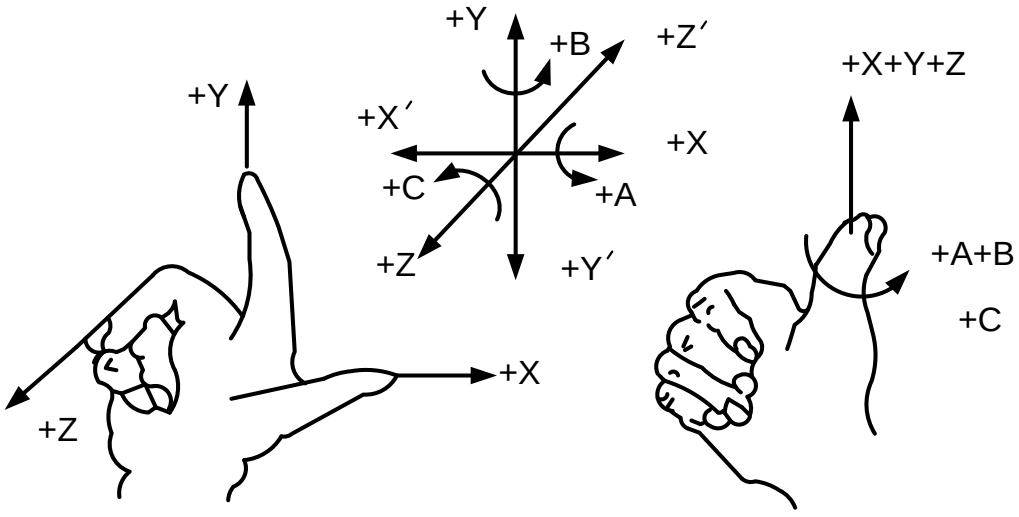


Fig. 1-1 Coordinate system following right-hand rule

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

——Z-axis 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).

——X-axis, Y-axis and Z-axis together constitute the coordinate system following right-hand rule.

1.2.1. Machine Coordinate System (MCS)

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

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

1.2.2. Workpiece Coordinate System (WCS)

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

1.3. Operation Mode and State

1.3.1. Operation Mode

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

Auto Mode

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

Manual Mode

In manual operation mode, there can be further divided into two types, namely jog and stepping.

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 the motion

signal with the help of one of the equipment, for example, clicking the button  on the software interface will make the machine tool move consecutively until the button is released.

In stepping mode, the user also uses 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 the machine tool precisely.

Reference Mode

In reference mode, the user can do settings relative to machine returning to the reference point. For example, it can set whether the machine returns to the reference point or not when the software is launched. It also can set the moving direction and positioning speed when returning to the reference point.

1.3.2. Operation State

In this system, each operation mode can be subdivided into several operation states. The operation mode and the operation state together decide the status of a machine tool.

IDLE

Idle state is the most common one. Under this state, the machine tool does not generate motion, but is ready for any new task.

E-STOP

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

RUNNING

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

PAUSE

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

LOCK

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

2. System Installation and Update

2.1. Basic Configurations of NcStudio

Host Computer Requirement

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. NcStudio Together Used With PM85A/95A

At this time, the system installation covers installation of the software and installation of the communication card. It is recommended to install the software before installing the communication card.

2.2.1. Software Setup

Please install the software as following steps:

1) Power up and start the computer.

2) Insert the installation CD. Double click *My Computer* on desktop to open it and double click the CD-ROM drive. Under the directory, find software installation package (the icon ) and double click it. A dialog box of language selection will pop up, as shown in Fig. 2-1.

Fig. 2-1 Language selection

3) Select [ENGLISH] to begin the installation. To avoid interference of old version software to the installation of the new one, the system prompts that the setup will delete all files of the old version software, see Fig. 2-2.

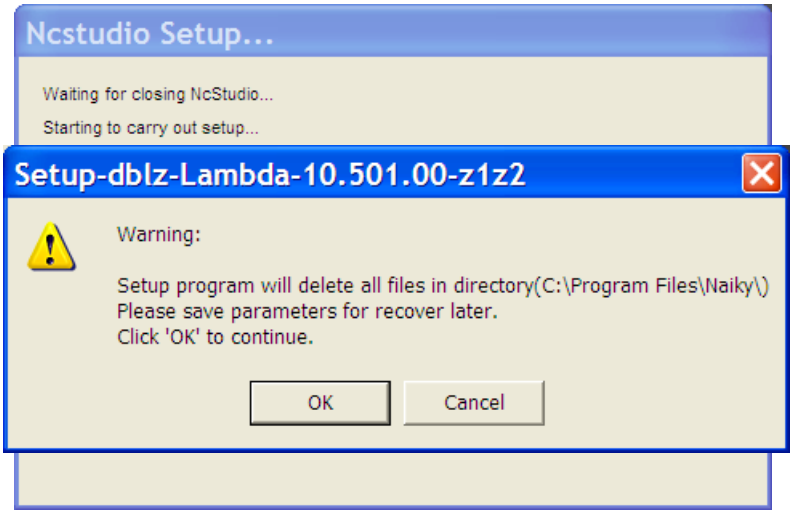


Fig. 2-2 Warning to delete previous files

4) Click [OK] for confirmation. If any old version software has been installed in this PC, the system will prompt to save its parameter settings. If you save the parameter settings here, you can apply all the settings to the current software, needless to set the parameters again afterwards. You can make your own decision and choose [Yes] or [No] to go ahead, see Fig. 2-3. If you install the software of the type for the first time, this step would be absent. Please jump to “Step 5)” to move on.

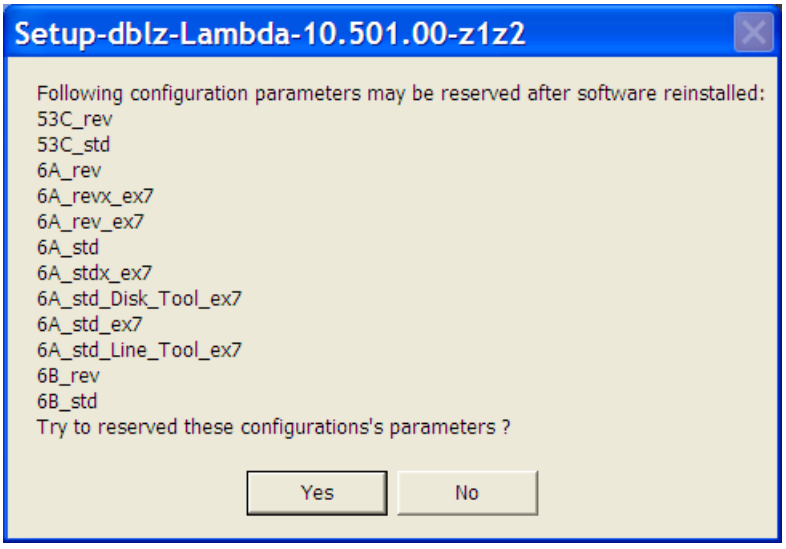


Fig. 2-3 Prompt to reserve parameter settings

5) Installation begins. The NcStudio system will be installed in the directory *C:\Program Files\Naiky* by default. Progressing picture is as shown in Fig. 2-4. With that, a dialog will appear to prompt the user to decide whether to put the generated files and the executive files together or separately, see Fig. 2-5. [No] is recommended here. The next dialog to appear is prompting shutdown of the computer for convenience of communication card installation, and [Yes] is recommended, see Fig. 2-6.

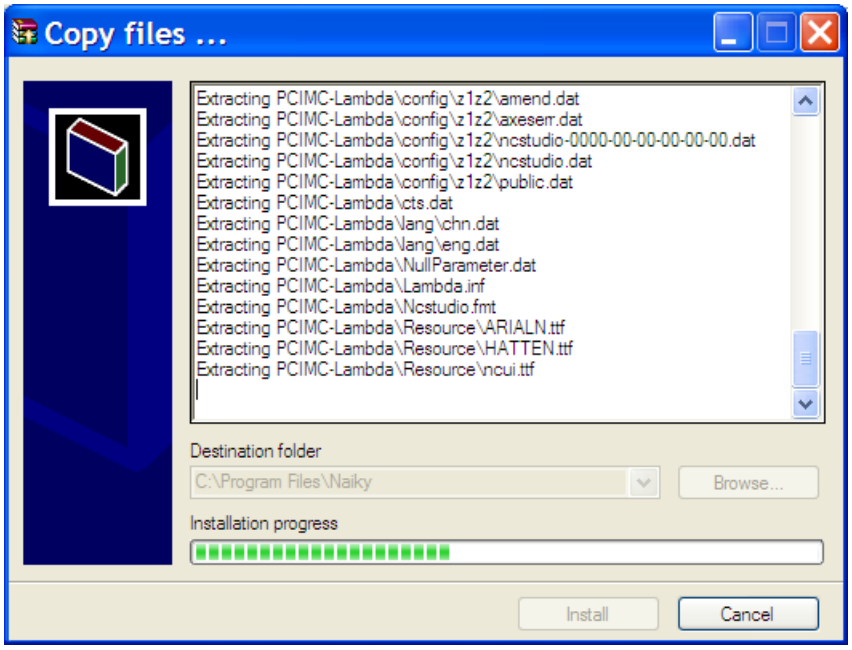


Fig. 2-4 Installing

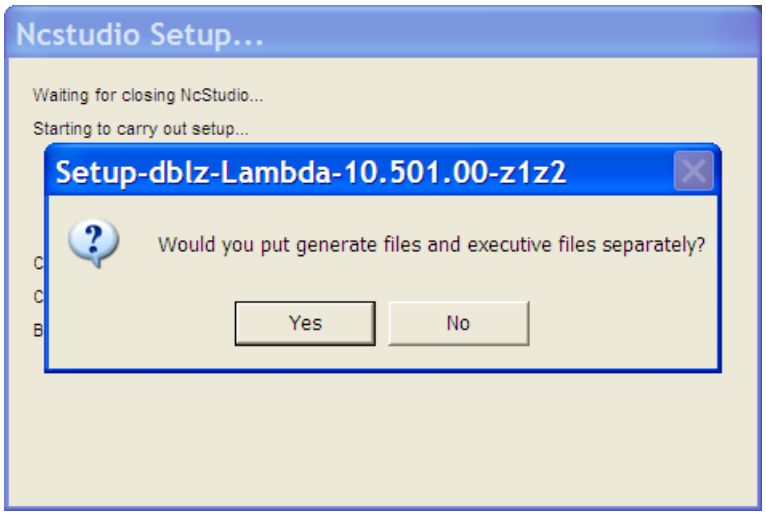


Fig. 2-5 Prompt for file location ([NO] is recommended)

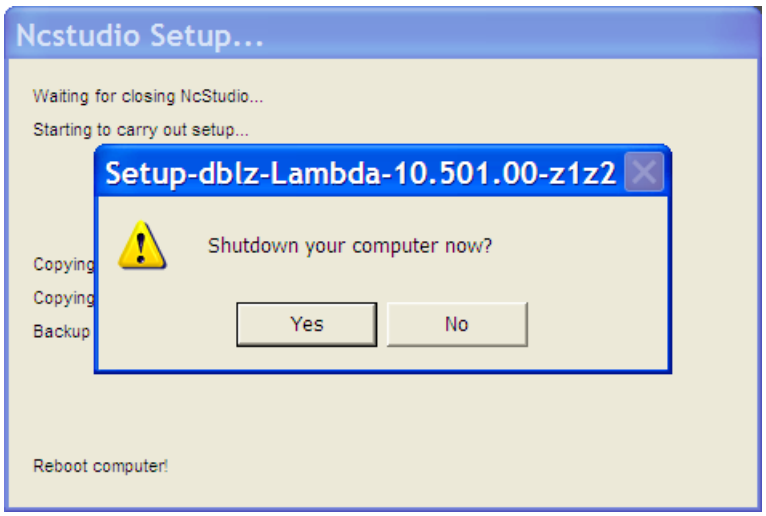


Fig. 2-6 Prompt to shut down the computer ([Yes] is recommended)

6) Software installation completes. The update and re-installation of the software share the same operations with above steps.

2.2.2. Communication Card Installation

Please install the communication card as following steps after software installation is completed:

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

2.2.3. Manually Update Hardware Driver

After installations of the software and communication card, the user needs to manually update the hardware driver. The specific operation steps are as below:

1) Right click *My Computer*, select “Properties”, and then click “Device Manager”. Choose the “CNC Adaptor” item, right click on it and select “Update Driver Software…”
2) A dialog box as Fig. 2-7 will pop up. Select “Install from a list of specific location (Advanced)”, click [Next] to continue.

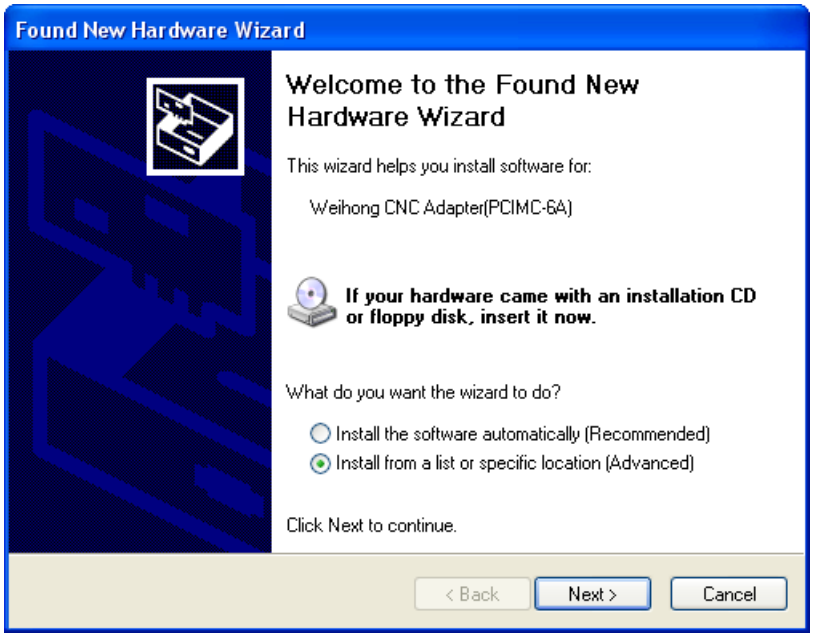


Fig. 2-7 Hardware update interface

3) A dialog box as Fig. 2-8 will pop up. Select “Don’t search, I will choose the driver to install”, click [Next] to continue.

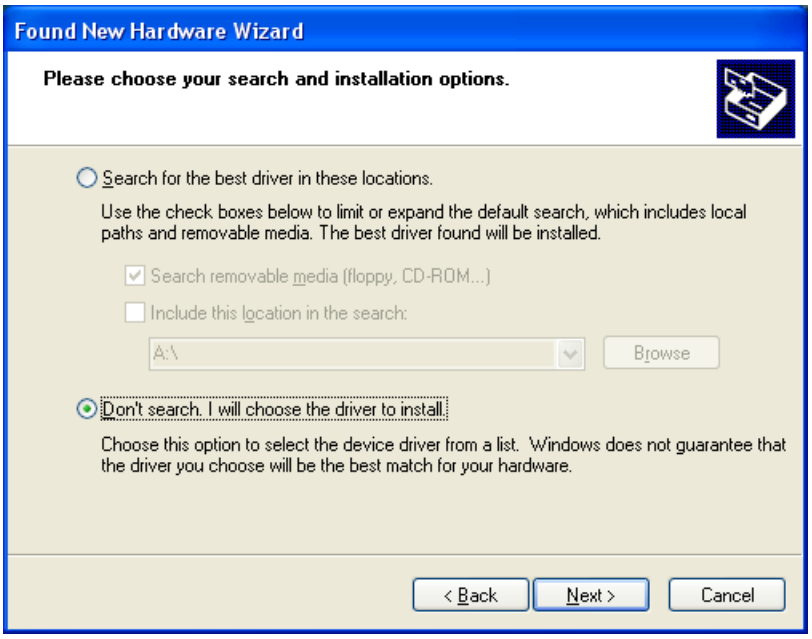


Fig. 2-8 Choose search and installation option

4) A dialog box containing compatible hardware will pop up, as shown in Fig. 2-9, click [Have Disk...] button to open the next dialog box.

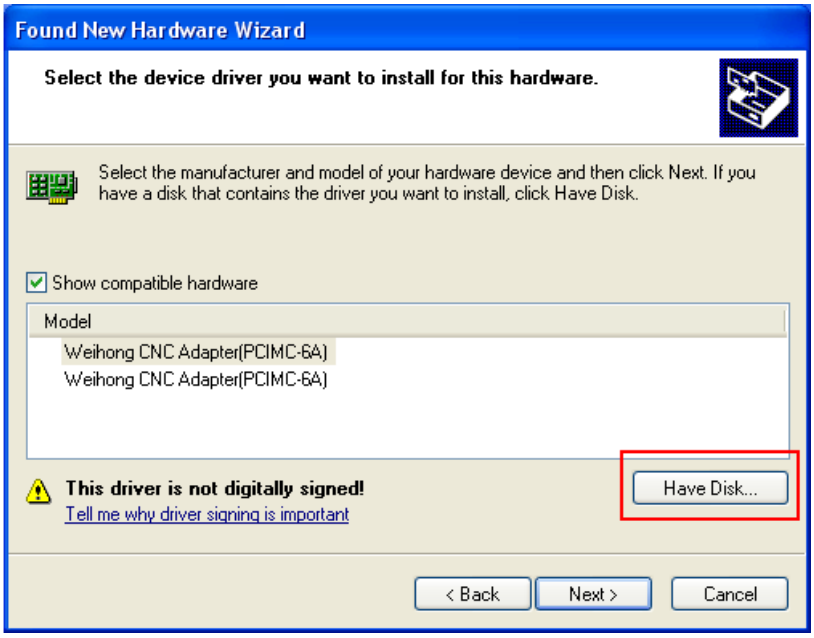


Fig. 2-9 Select the driver manually

5) A dialog box named “Install From Disk” will pop up, see Fig. 2-10. Click [Browse…] button to open the target file of driver.

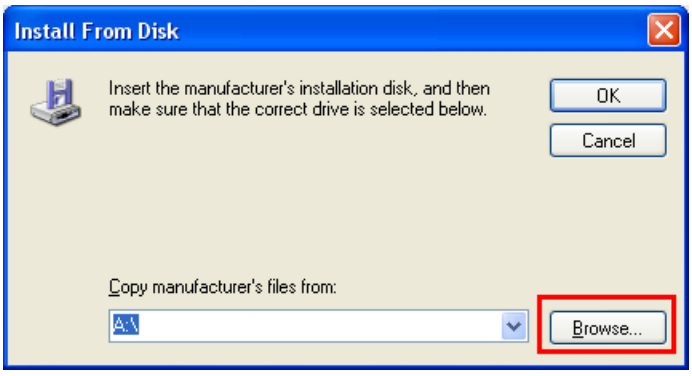


Fig. 2-10 Select the target file of driver

6) In “Locate File” dialog box as Fig. 2-11, select the target hardware drive in the list, Choose the target file “Lambda.inf” under directory “C:\Program Files\Naiky\PCIMC-Lambda”, taking PM85A as an example.

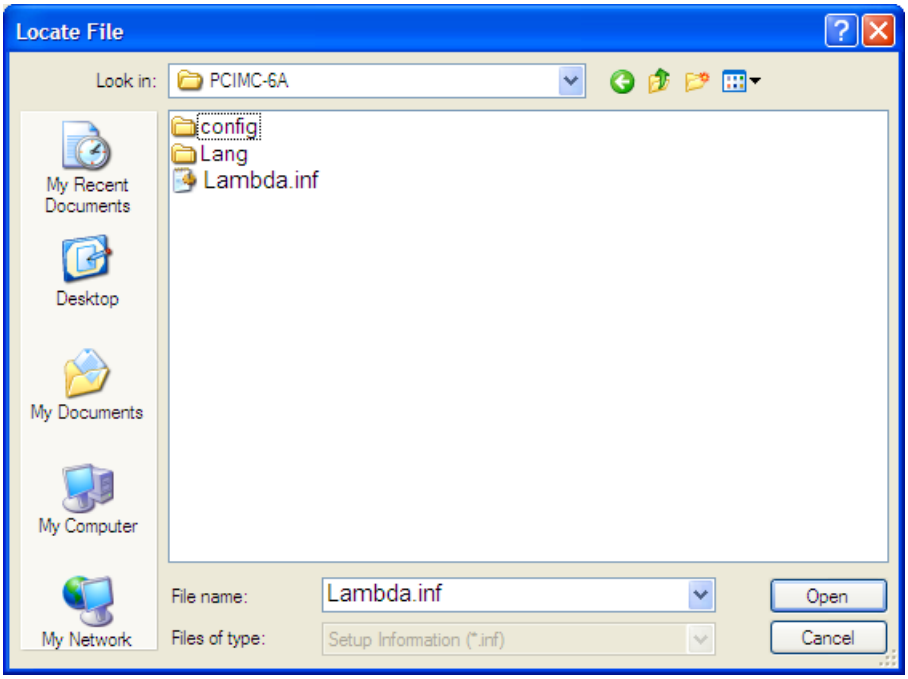


Fig. 2-11 Select the hardware driver

7) After hardware driver being correctly chosen, the interface jumps to the previous dialog box where the target file directory will be displayed under item “Copy manufacturer’s files from:” as shown in Fig. 2-12.

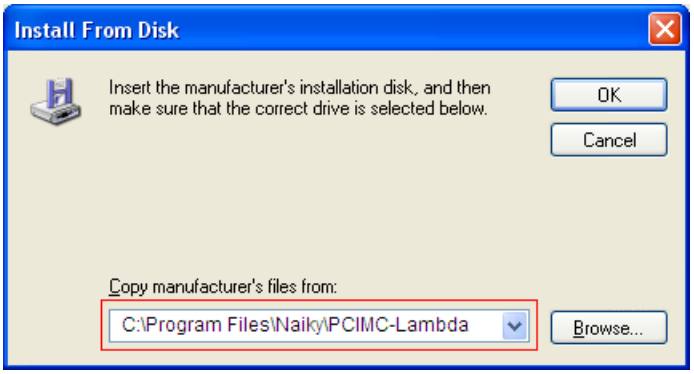


Fig. 2-12 Target file directory confirmation

8) Click [OK] to go back, and then click [Next] to start updating the driver software. The progressing picture is shown as Fig. 2-13.

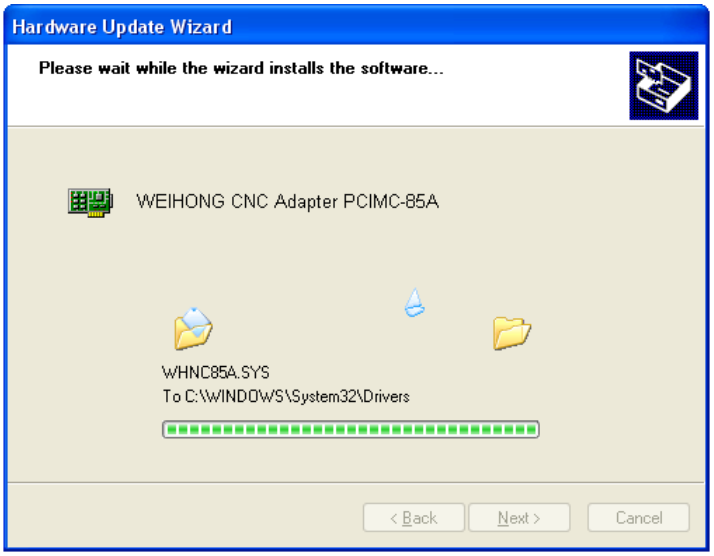


Fig. 2-13 Updating the driver

9) When the updating is finished, a dialog as shown in Fig. 2-14 will pop up. Click [Finish] to complete the update of the hardware driver. Double click the icon  on the desktop or click the icon on the menu *"Start→All Programs"* can launch the software successfully.

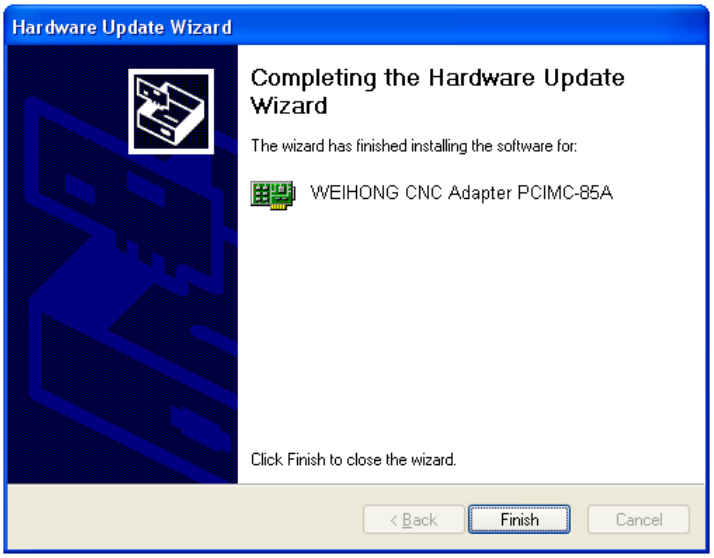


Fig. 2-14 Driver update completed

2.3. NcStudio Used Together With NC65A/65B

When NC65A/65B host leaving factory, it is ready for use with the system and the software all installed.

If you need to re-install or update the software, follow the steps below:

1) Access PC desktop, find out the software installation package and double-click on it. A "Language Selection" dialog box will firstly pop up, as shown Fig. 2-15. When "选择中文界面" is chosen, the

NcStudio system will be run in Chinese later; when “ENGLISH” is chosen, the system will be run in English in use.

Fig. 2-15 Language selection

2) Continue installation. If the NcStudio system have been installed in this PC before, a prompt for deleting the previous files will be shown during installation, as shown in Fig. 2-16, click [OK] for confirmation and continue, click [No] to cancel and exit installation.

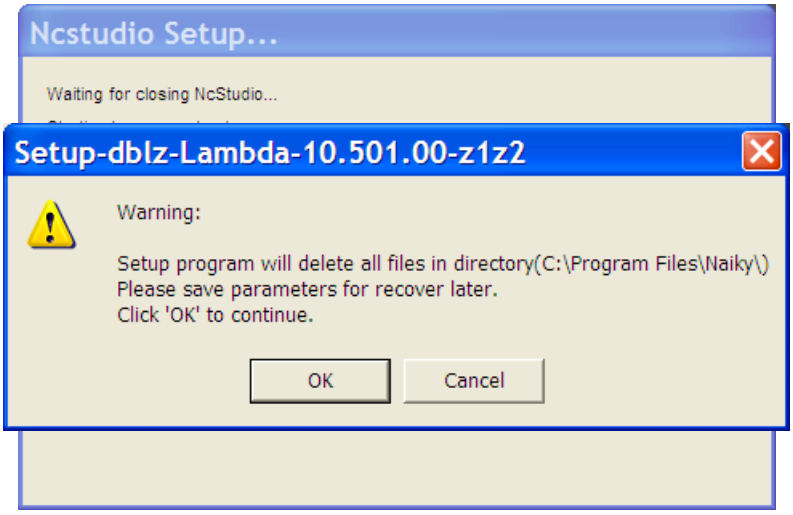


Fig. 2-16 Warning to delete previous files

3) Installation begins. Progressing picture is as shown in Fig. 2-17.

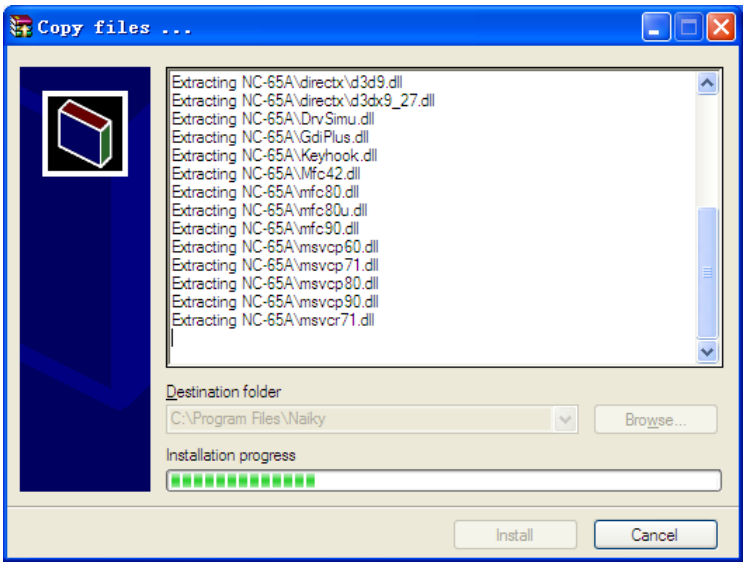


Fig. 2-17 Installing

4) After the software installation is completed, the hardware driver will be installed automatically, see Fig. 2-18.

Fig. 2-18 Installing driver

5) After the driver is installed, the system will be restarted, see Fig. 2-19. the software will be launched automatically after the system restarted. Whole installation finishes.

Fig. 2-19 Driver installation completed

2.4. Uninstall NcStudio

NcStudio is green software which has the following advantages: it can be installed or uninstalled easily and quickly; the installation information will not be written into the registry of the computer; the files under the installation directory can be deleted directly with no remnant files left on the hard disk. Therefore, to delete the NcStudio 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. Operation Interfaces of NcStudio

After the NcStudio software has been installed according to instructions in Chapter 2, the system can be run anytime the user double clicks the shortcut icon on desktop.

The interface of NcStudio consists of the title bar, menu bar, CNC state window, machine control bar and several functional windows.

The holistic interface is shown as Fig. 3-1.

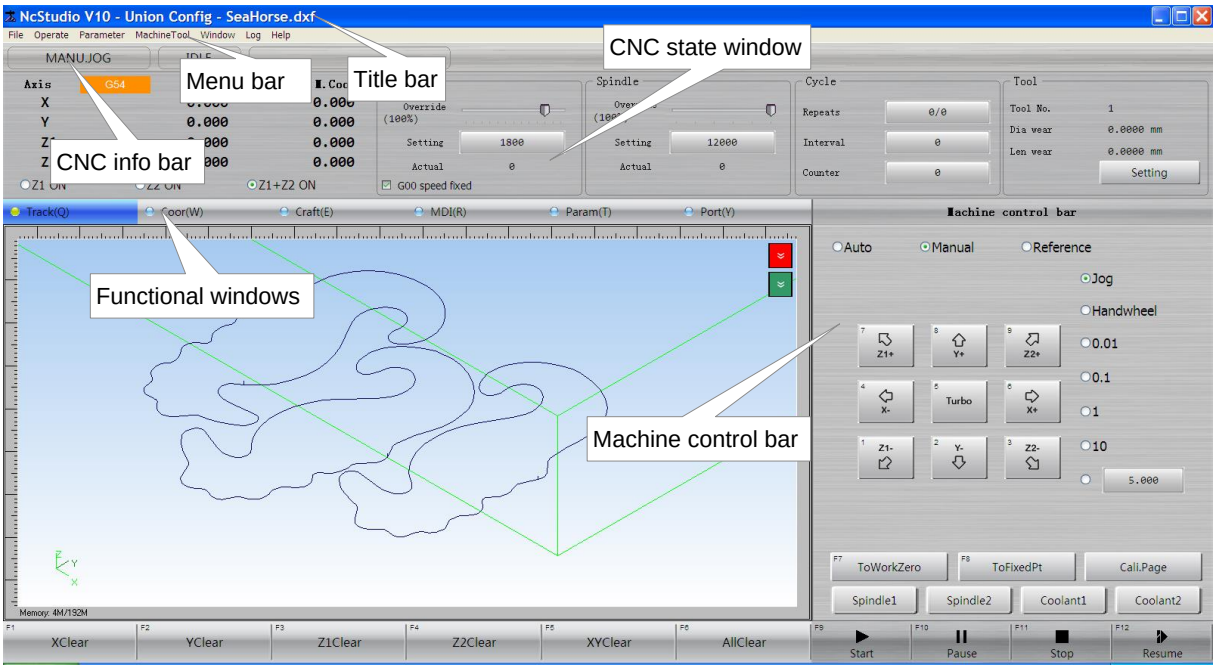


Fig. 3-1 Holistic operation interface of NcStudio V10

3.1. Title Bar

On the top of the NcStudio software interface is the title bar, where software name, the current configuration and name of the loaded program file are displayed. The color of the title bar indicates whether the corresponding window is active or not. A window with blue title bar is active while one with grey title bar is inactive.

Fig. 3-2 Title bar

The icon on the left side of the title bar is system menu box and it can be used to open the window control menu. Clicking this icon or pressing shortcut key “Alt + spacebar” will open a system menu, as shown in Fig. 3-3.

Fig. 3-3 System menu

3.2. Menu Bar

Below the title bar is the menu bar, which includes many normally hidden pull-down menus. Each pull-down menu consists of several menu items and each menu item corresponds to a specific function, action or state. After a menu item is selected, the system will execute the relevant function or action, or alter the system state. The menu item can be selected either by mouse or by keyboard.

3.3. CNC Info Bar

Below the menu bar is the CNC info bar, showing the current CNC state and some other info, including the current operation mode, operation state, running state of machine as well as some possible cautions and alarm events, as shown below. To name but a few, the following picture is only provided as an example.

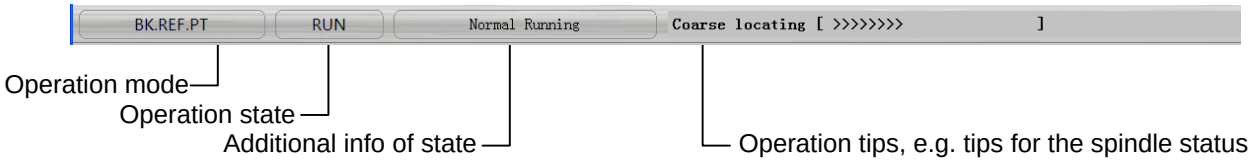


Fig. 3-4 CNC information bar

3.4. CNC State Window

Under the CNC info bar is the CNC state window, which can be divided into 5 areas according to functions, as illustrated below:

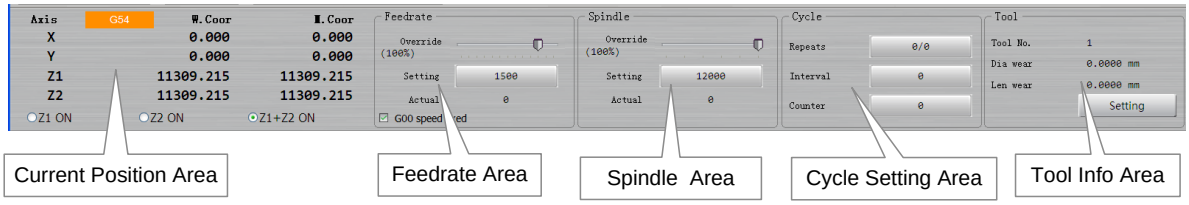


Fig. 3-5 CNC state window

3.4.1. Current Position

To describe various positions, NcStudio supports two types of coordinate system, including workpiece coordinate system (WCS) and machine coordinate system (MCS). As you can see in Fig. 3-6, motion axes of machine tool, machine coordinates, workpiece coordinates as well as the

remaining distance (also known as distance-to-go) are shown in this area. Besides, you can set the current position as the workpiece origin at any moment, which is very convenient to establish a WCS in practice.

A mark indicating “machine coordinates effective” will appear before each axis after “back to reference point” is executed, as illustrated below.

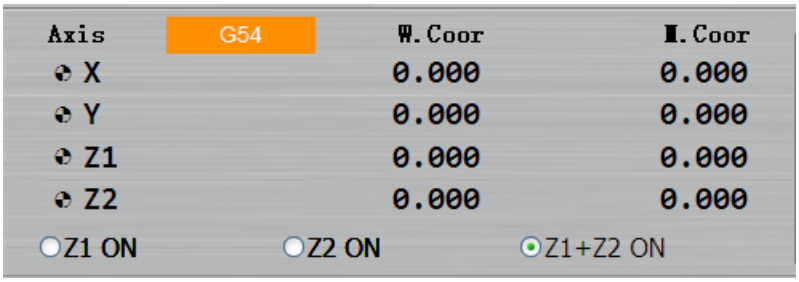


Fig. 3-6 Current position area

3.4.2. Feedrate Area

This area displays information like the setting speed, the actual speed and the feedrate override. The user can also adjust feedrate and feedrate override here.

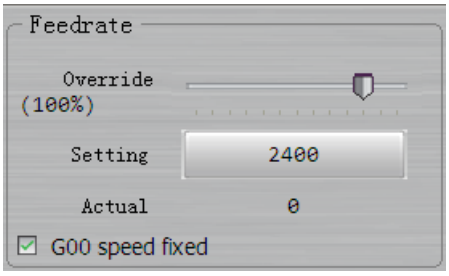


Fig. 3-7 Feedrate area

Sliding Block of Feedrate Override

By dragging the sliding block, the user can regulate the current working speed within a scope of 0~120%. The direction keys on the keyboard can increase or decrease it by 1% while keys *PageDown* and *PageUp* by 10%. Feedrate override is shown in percentage.

$$\text{Actual feedrate} = \text{Setting feedrate} \times \text{Current feedrate override}$$

Setting Value

In auto mode, when the user clicks the figure after “Setting”, a dialog box will pop up for setting machining speed of linear axes.

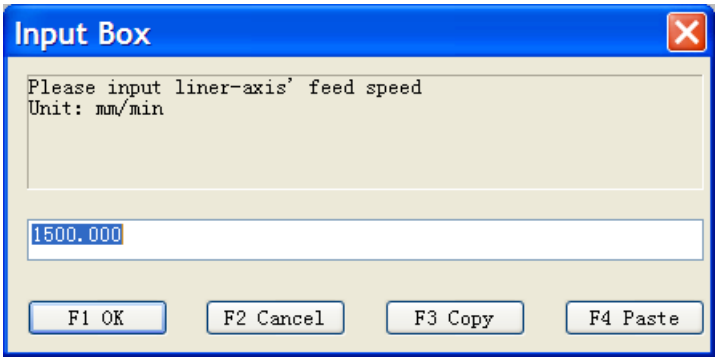


Fig. 3-8 Setting auto speed dialog box

In manual mode, when the user clicks the figure after "Setting", a dialog box as below will pop up for setting jog speed (also known as manual/jog low speed).

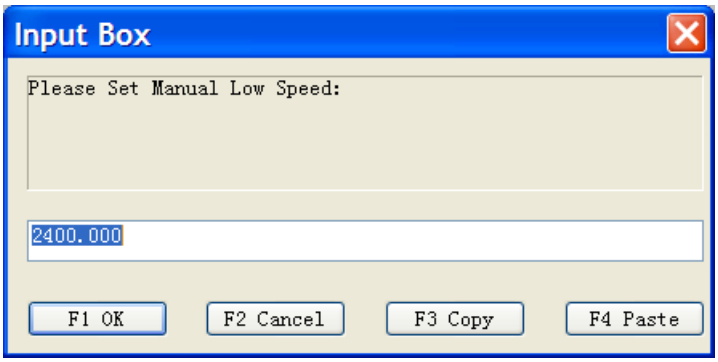


Fig. 3-9 Setting manual speed dialog box

CAUTION

1) The user needs to return four axes to the reference point before using the Manual High/Low Speed, otherwise, speed specified by parameter "N71002 MaxJogFeedrateBeforeBKREF" will be used instead.
2) Pressing number key 5 together with numeric direction keys 4(X-), 6(X+), 2(Y-), 8(Y+), 1(Z-) or 9(Z+) makes the axis run in Manual High Speed, while pressing the number keys alone makes it run in Manual Low Speed.

Actual Value

It is the instantaneous value of feedrate which alters with the change of setting value, the current acceleration or deceleration condition and the feedrate override.

G00 Speed Fixed

With this menu item selected, the running speed will be fixed at 100% of the setting value of G00 speed, unaffected by the change of feedrate override.

3.4.3. Spindle Speed Area

This area displays information like the setting speed, actual speed and spindle override, etc. The user can also change the setting value and spindle override here. See Fig. 3-10.

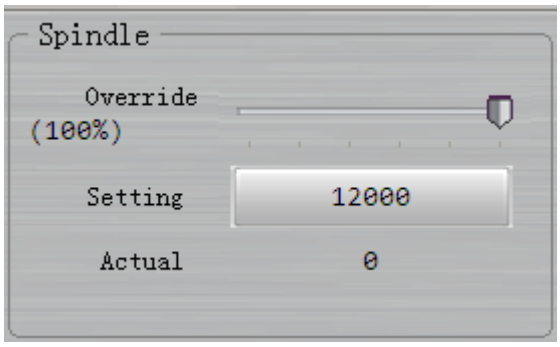


Fig. 3-10 Spindle speed area

Spindle Override Sliding Block

By dragging the sliding block, the user can regulate the current spindle speed within a scope of 0~100%. Parameters “N53002 SRP_MIN” and “N53003 SRP_MAX” can set the minimum and maximum spindle override respectively. The direction keys on the keyboard can increase or decrease it by 1% while keys *PageDown* and *PageUp* by 10%. Spindle override is shown in percentage. The relationship between the actual value and setting value is as below:

$$\text{Actual value}=\text{Setting value}\times\text{Current spindle speed override}$$

Spindle Speed Setting

Clicking [Setting] button will open a dialog box named “Spindle Rev”, where you can set the spindle revolving speed, as shown in Fig. 3-11.

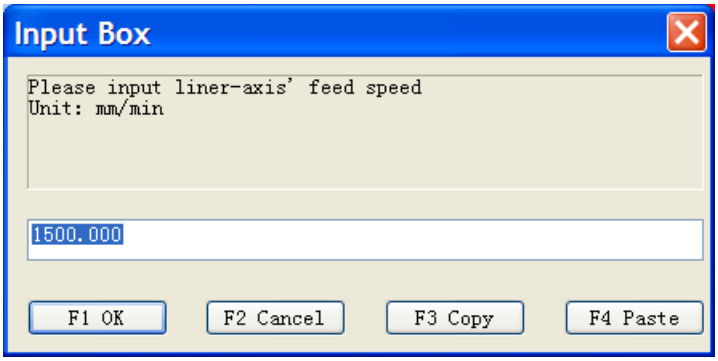


Fig. 3-11 Setting spindle speed dialog box

3.4.4. Cycle Machining Setting

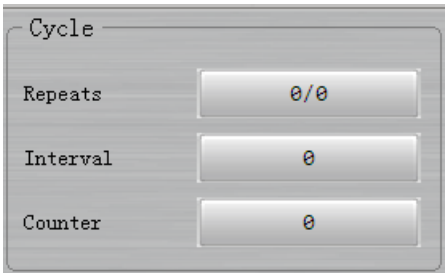


Fig. 3-12 Cycle machining setting area

As shown in Fig. 3-12, the user can set cycle machining in this area.

Repeats

The processing times that has been cycled and the total processing times set. Clicking the figure button next to it can set the total processing times of a program file.

Interval

It refers to the interval time of cycle machining, namely the period of time between the completion of current process and the start of the next process. Clicking on this button can set the interval time.

Counter

It is the number of the finished workpiece or completed parts. Clicking the figure button next to it can clear its count to zero.

3.4.5. Too Info Area

This area displays information including spindle tool number, diameter wear and length wear. In addition, you can set the tool parameters and its compensation here.

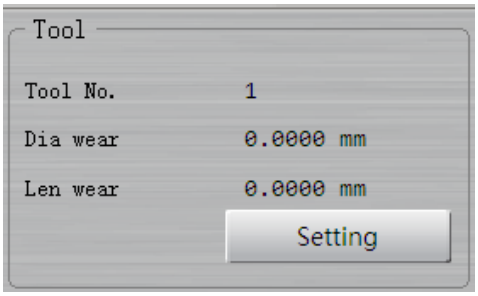


Fig. 3-13 Tool compensation

After tools worn, reshaped or changed, the tool radius will change. On this occasion, the user can make tool compensation by modifying relevant parameters in tool parameters dialog box, instead of modifying previous tool compensation subprogram in machining file.

Clicking [Setting] button will open the parameter dialog box where you can directly set the relevant parameters, as shown below.

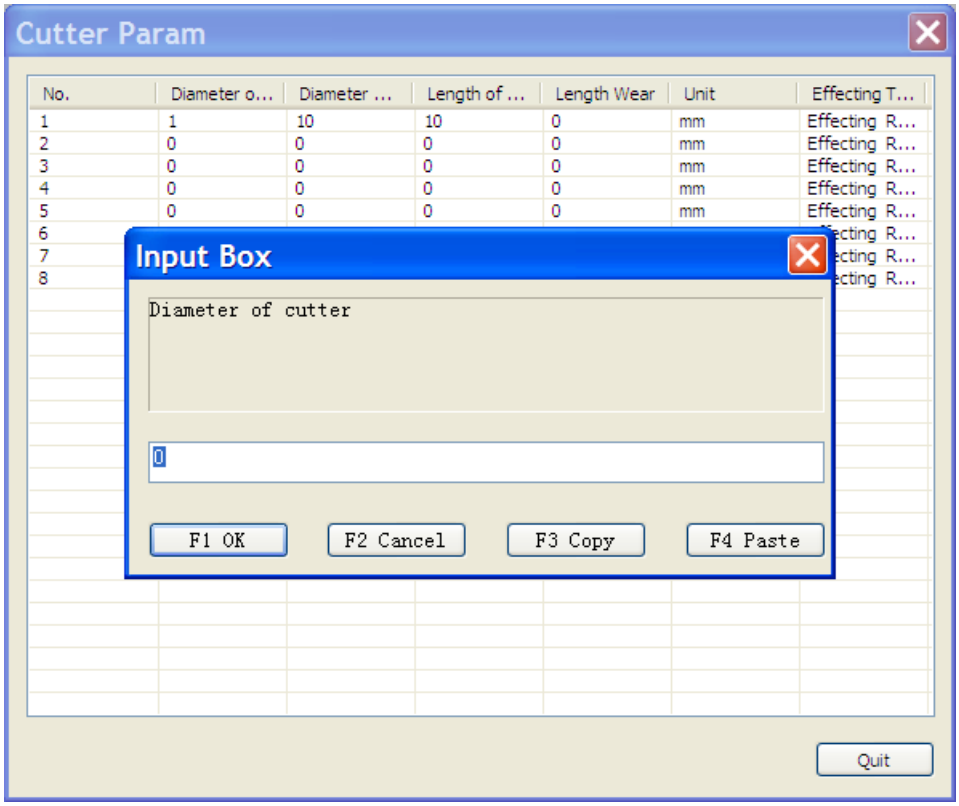


Fig. 3-14 Tool parameters dialog box

3.5. Machining Track

The machining track window shows motion track of the tool in three-dimensional way. When machining or simulation is being performed, the track window will follow the machining track of the tool in real time. In this window, the user then can view the tool path intuitionally so as to ensure the proper implementation and integrity of machining file.

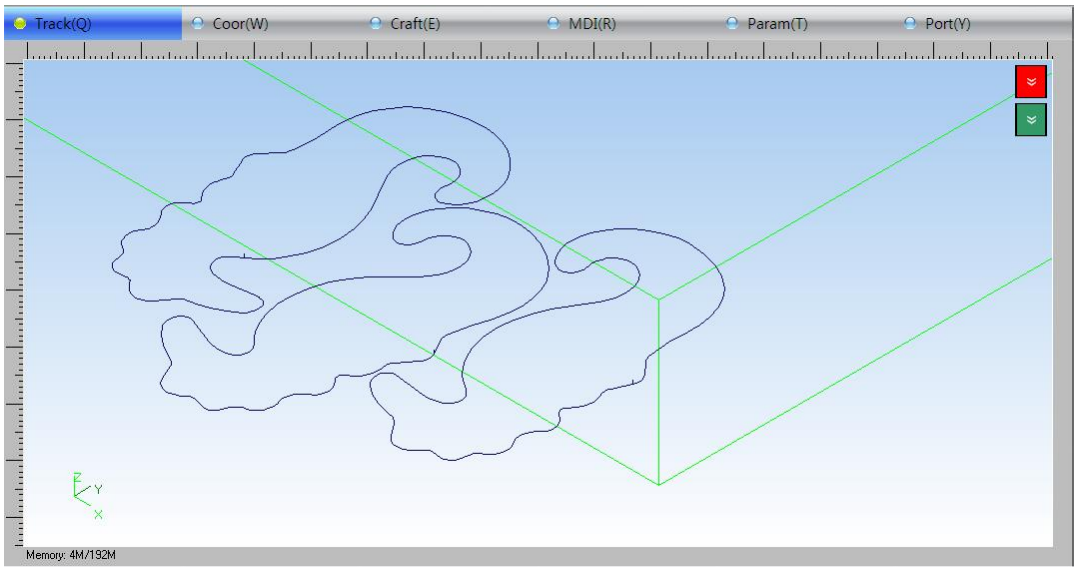


Fig. 3-15 Track window

The “Alarm Event” and “Run Report” mini box are hanging on the upper-right part of the track window. Most of the time, they are folded at the upper right side of the window, and the user can unfold them by clicking on the icon, as shown in Fig. 3-16.

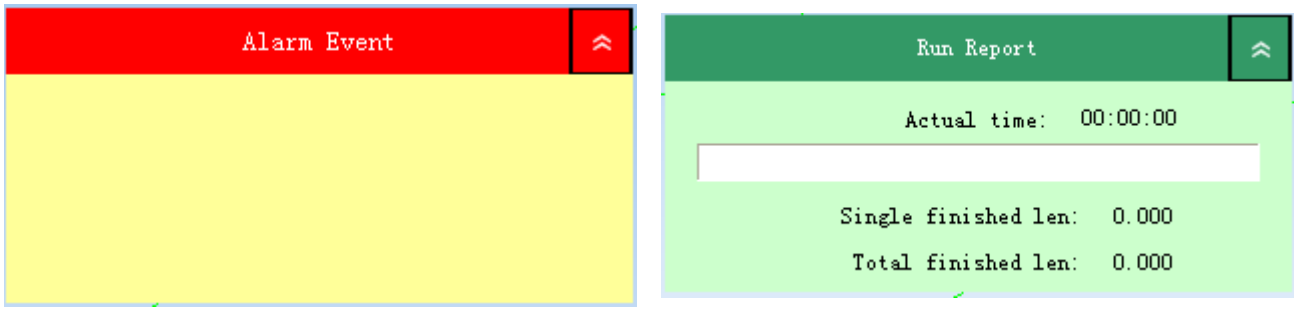


Fig. 3-16 Unfolded Alarm event & Run Report

Right clicking on the window will open a shortcut menu, as shown in Fig. 3-17.

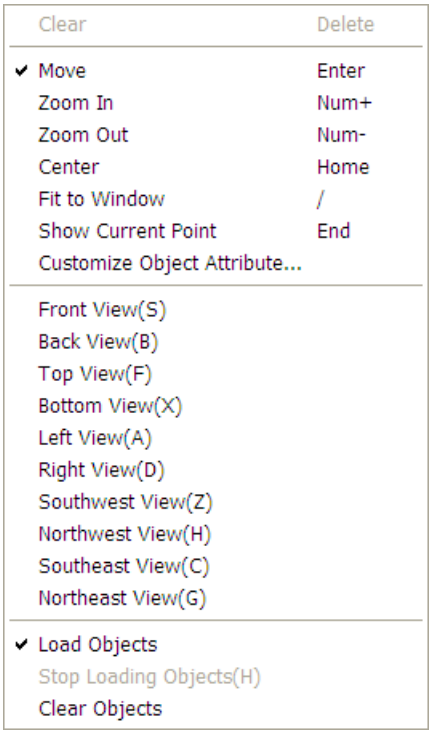


Fig. 3-17 Shortcut menu for track window

Clear

To clear the machining track in the window, both shortcut key and “Delete” key are available.

Move

When the mouse icon turns to , left click the mouse and drag it at the same time to move the track in the window. Numeric direction keys on keyboard share the same effect.

Zoom In/Out

Move the mouse into the *Track* window and click the right mouse button. Select “Zoom in” or “Zoom out” in the pop-up menu to switch between and .

When the Track window is active, turning the mouse wheel for zooming, rolling up to zoom out and rolling down to zoom in.

The user can also zoom in/out the track by pressing “＋” or “－”on the mini-keyboard.

Center

Right click the mouse, and select “Center” item in the pop-up shortcut menu. “Home” key on the keyboard shares the same effect.

Fit to window

It means to show the whole track in the track window, without scrolling the mouse.

Right click the mouse, and select “Fit to Window” item. When the track window is active, you can also press the “/” key on keyboard (only valid in English input method) to fit the track to window size.

Show Current Point

The current machining position will be displayed in the middle of the *Track* window.

Right click the mouse, and select “Show Current Point” item in the pop-up shortcut menu, or use “End” key on the keyboard.

Customize Object Attribute

When the *Track* window is active, right click the mouse and select “Customize Object Attribute” item in the pop-up shortcut menu.

A dialog box named “Custom Parameter” will pop up, where the user can freely define the tracking mode and color, as shown in Fig. 3-18.

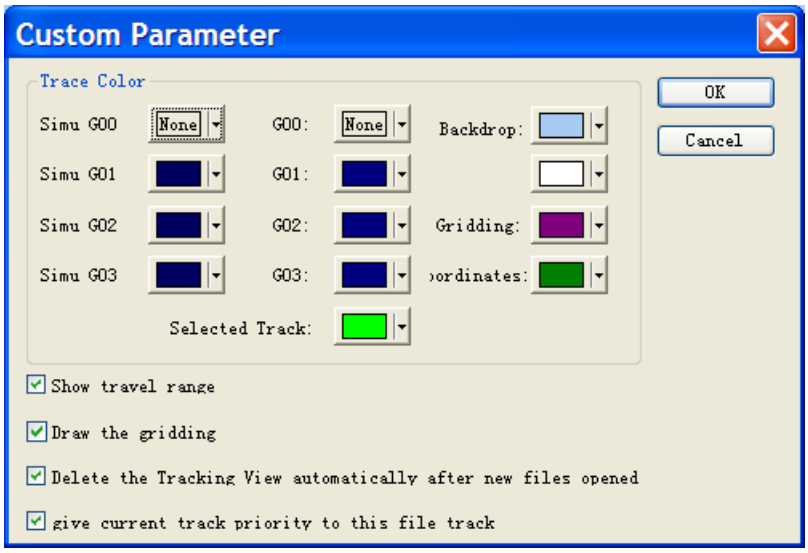


Fig. 3-18 Custom parameter

When “Show travel range” item checked , simulative workbench will be displayed in the *Track* window.

When “Draw the gridding” item checked, grid will be displayed in the *Track* window.

When “Delete the Tracking View automatically after new files opened” item checked, current track will be removed automatically the moment a new program file is loaded into the system.

When “give current track priority to this file track” item checked, a simulative track will be presented with priority; otherwise, the actual machining track will be presented with priority.

Track View

The system provides 10 types of commonly-used views to help the user switch among these views by pressing the shortcut keys on the keyboard.

Front view	Shortcut key: S
Rear view	Shortcut key: B
Top view	Shortcut key: F
Bottom view	Shortcut key: X
Left view	Shortcut key: A
Right view	Shortcut key: D
Southwest equiaxial lateral view	Shortcut key: Z

Northwest equiaxial lateral view	Shortcut key: H
Southeast equiaxial lateral view	Shortcut key: C
Northeast equiaxial lateral view	Shortcut key: G

Load Objects

This item is used to load the track of a program file one time when pre-analyzing the file. While in actual machining, the real-time track will also be simulated. The two kinds of track are displayed in different colors. Selecting this item is to load the track beforehand. When you want to remove it, select the opposite item “Clear Objects”.

Stop Loading Objects

This item is used to terminate the loading of machining track. An over large track will slow down the system running; therefore, there are two options to solve the problem. One, choose this item to stop loading of the track; two, select “Clear Objects” item to remove all loaded track and stop loading the remaining track.

Clear Objects

This item will remove all loaded track and stop loading the remaining track. Note that selecting this item will deselect its opposite item “Load Objects”.

3.6. Coordinate System Window

The coordinate window is where the user can set the workpiece coordinate system (WCS), including set the workpiece offset (also known as part offset), the public offset and conduct fine-tuning or jiggling. Besides, the user can also conduct tool measurement (also called tool calibration) as well as centering here.

3.6.1. Offset Management

At present, the system supports coordinate system G54/G55/G56/G57/G58/G59, in addition, the system also supports the extended WCS of G59, from G59P1 to G59256, altogether 256 groups of extended workpiece coordinate systems, as shown in Fig. 3-19.

To activate the extended WCS, the user needs to firstly set parameter “N61110 Support Extra Workcoors” to “Yes”, and set parameter “N61111 Size of Extra Workcoors” to decide the number of the extended WCS.

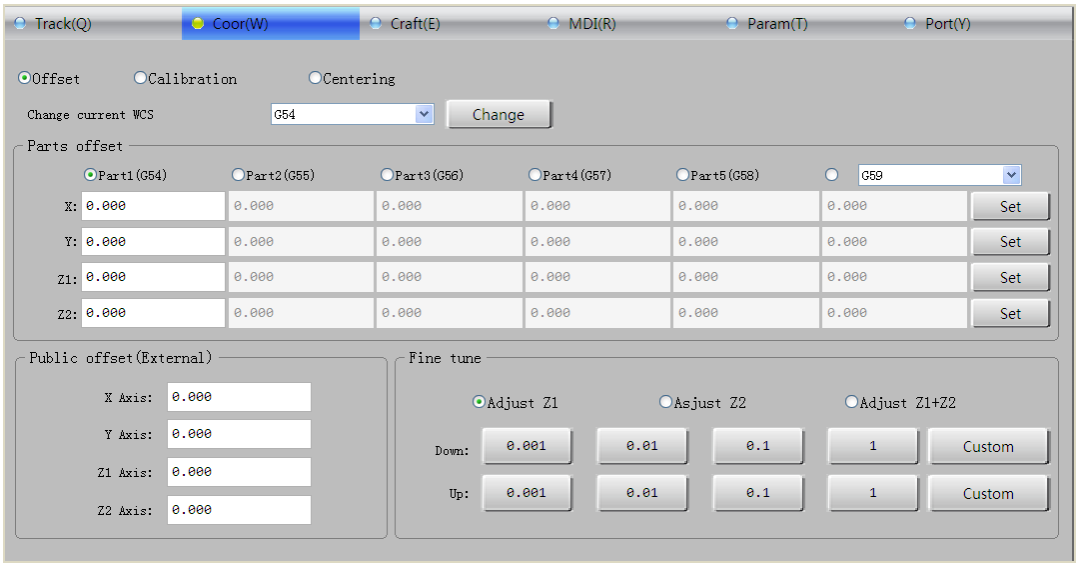


Fig. 3-19 Coordinate system interface

Parts offset

Under “Coor” tab, select “Offset” item to activate the coordinate system interface. Select the WCS to be set under “Parts offset” and make it the current WCS. At this time, the user can set the part offsets of X/Y/Z1/Z2 axes by directly inputting numeric value in corresponding input boxes. Click “Set” button will make the current machine coordinates of X/Y/Z1/Z2 axes the workpiece origin (also known as work zero) of the current WCS.

CAUTION

Differing from setting the workpiece origin by “Clear” operation, setting the part offset by inputting values will not clear the public offset, while “Clear” operation will clear relative public offset, please note that.

Public offset (External)

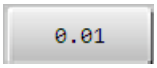
Public offset can be applied to all coordinate systems. It is used to adjust the workpiece origin in X/Y/Z1/Z2 axes without changing the part offsets. The user can set the public offset by inputting values in the input box.

The relationship among part offset, tool offset and public offset is illustrated below:

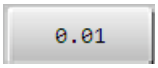
Workpiece coordinate=Machine coordinate-Part offset-Tool offset-Public offset

Fine tune

During machining, the user can fine tune Z axis or adjust Z axis in minor way before restart machining again in PAUSE state.

[Up]/[Down]: the [Up]/[Down] operation aims to adjust the public offset of Z axis, instead of adjusting the position of Z axis. For example, click 

button under [Up] is to ascend the

public offset of Z axis by 0.01mm; while click 

button under [Down] is to descend the public offset of Z axis by 0.01mm.

Custom: with this function, the user can self-define the fine-tune distance of Z axis by manually inputting a concrete number. See Fig. 3-20.

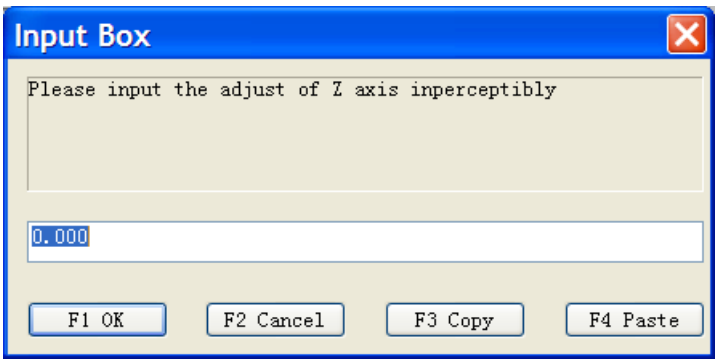


Fig. 3-20 Customize Up/Down distance

3.6.2. Calibration Management

The NcStudio system offers two kinds of tool calibration (also known as tool measurement), namely “Auto measurement mode” and “First and exchange calibration mode”. The user can choose and set the tool calibration mode by setting parameter “N79102 Cali_Mode”. Under “Coor” tab, select “Calibration” item to activate the calibration interfaces where the user can conduct tool calibration.

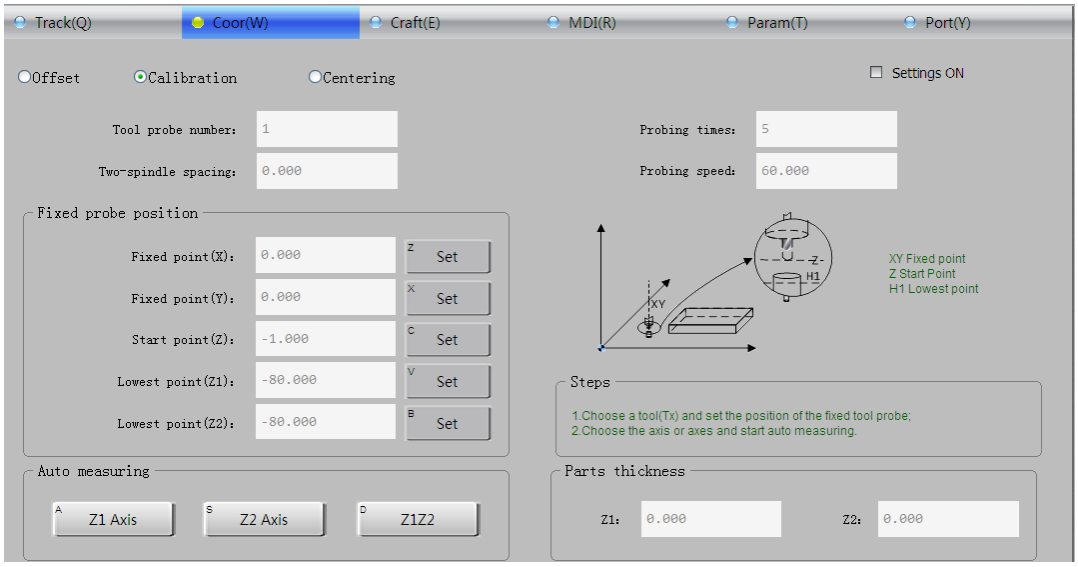


Fig. 3-21 Calibration interface of Auto measurement

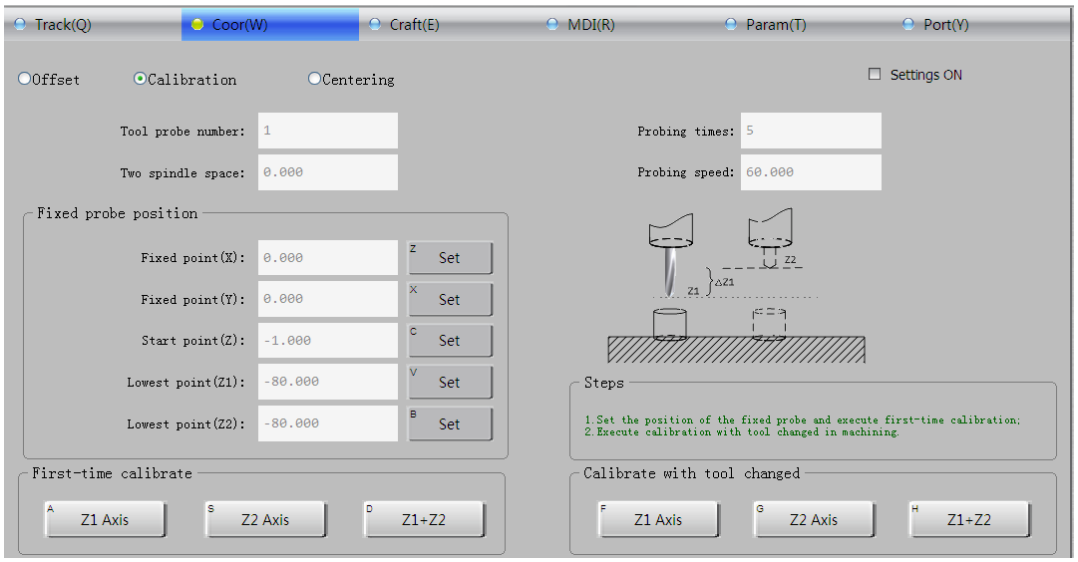


Fig. 3-22 Calibration interface of First/Exchanged measurement

As shown in Fig. 3-21 and Fig. 3-22, the user should firstly check the item “Setting ON” to make all parameters adjustable. Every time the item “Setting ON” be checked, manufacturer’s password will be asked.

Probing times: it refers to the locating times during tool calibration. To make the result more accurate, several times of locating will be conducted and an average value will be calculated and be the final position.

Probing speed: it refers to the speed from the starting point to approaching the tool probe surface during tool calibration.

Fixed point (X/Y): it refers to the X/Y position of the fixed tool probe (also called tool presetter/tool sensor).

Start point (Z): it refers to the position where the spindle starts tool calibration at speed of 60mm/min, while from the highest position to the “Start point”, the spindle moves at speed specified by parameter “N64020 RAPID_TRAVEL_FEEDRATE”.

Lowest point (Z1/Z2): the lowest position the tool can descend to during tool calibration with an aim to protect the tool or tool presetter/sensor from damage caused by too much tool plunging. When the tool travels below this position, an alarm message with yellow background will occur, prompting “It exceeds the lowest position of Z axis, please check!”

Auto Measuring

1) If the software is run for the first time, click 

button below “Auto measuring” to execute tool calibration in Z1 axis. The system automatically records the tool offset.
2) Manually move Z1 axis to the workpiece surface, click 

button to clear the Z1 workpiece coordinate to zero in order to get the Z1 workpiece origin (or work zero in Z1 axis).
3) Auto measurement finishes. The system is ready to actual machining.
4) After tool changed or tool breakage, click 

button to conduct auto measurement. When it finishes, the tool offset after tool changed will be obtained automatically, so as the workpiece coordinate of tool nose in Z1 axis.
5) When auto measurement finishes, the system can continue machining. For Z2 axis, the operation steps are the same as that of Z1 axis as above.

First/Exchanged Calibration Mode

1) Manually move Z1 axis to workpiece surface, and click 

button to set the workpiece origin.
2) Click 

button to execute the first-time calibration. The system will automatically record the Z1 machine coordinate. As shown in Fig. 3-23, the whole process will be completed automatically.

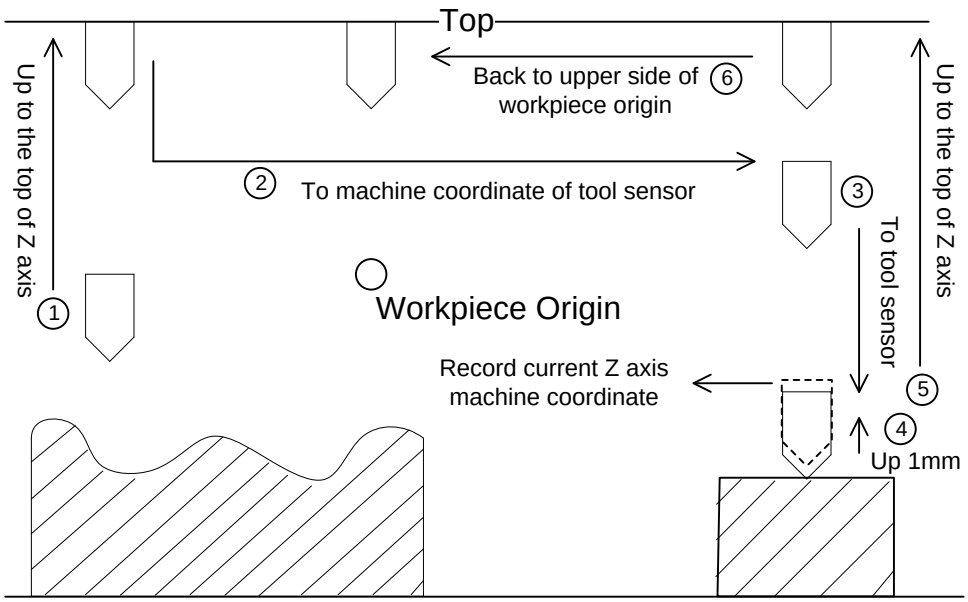
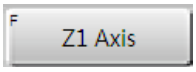


Fig. 3-23 First-time calibration

3) The first-time calibration finishes and the system is ready for actual machining.

4) After tool changed or tool breakage, click 

button under “Calibrate with tool changed” item to conduct tool calibration after tool changed. The system will restore the current Z1 workpiece coordinate. As shown in Fig. 3-24, the whole process will be completed automatically.

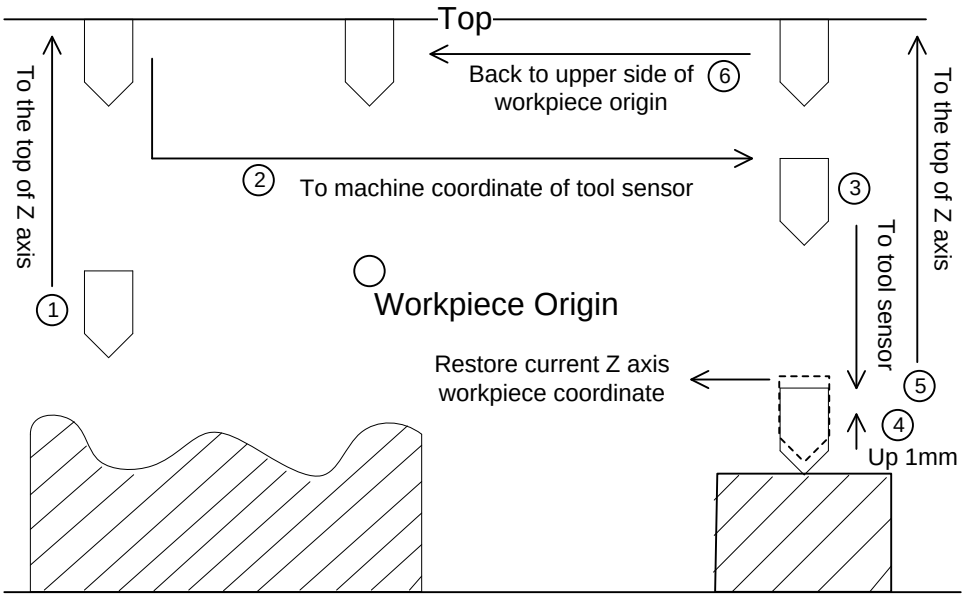


Fig. 3-24 Tool calibration after tool changed

5) When tool calibration finishes, the system can continue machining. For Z2 axis, the operation steps are the same as that of Z1 axis as above.

3.6.3. Centering Management

Centering, as its name implied, is used to set the workpiece origin by locating the center of a workpiece.

Under the “Coor” tab, select “Centering” to open centering interfaces, see Fig. 3-25.

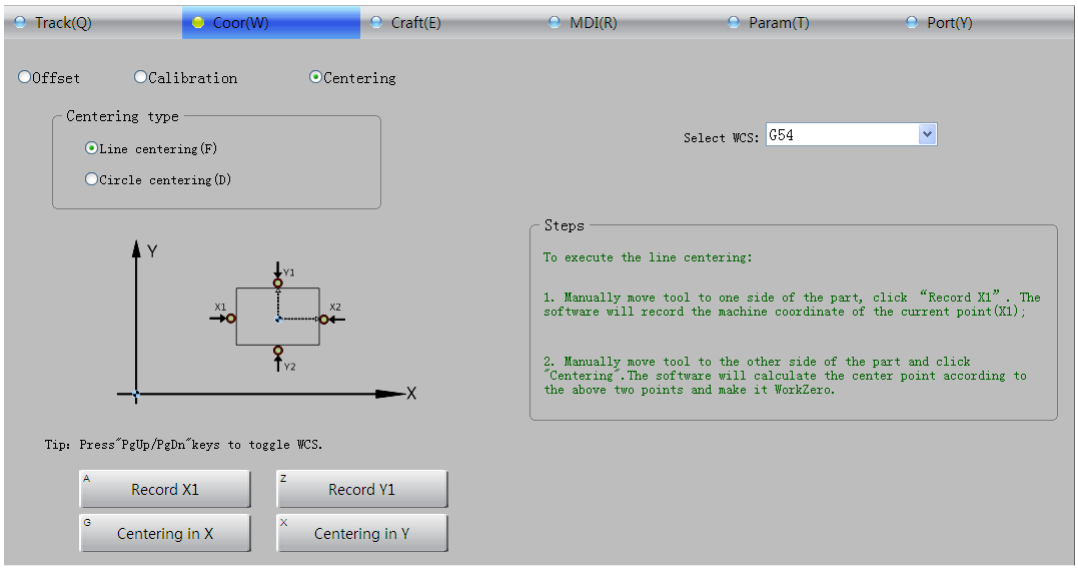


Fig. 3-25 Centering Interface

As shown above, the user needs to choose centering type first, namely choose “Line centering” or “Circle centering”. And select a WCS in the pull-down WCS list.

Line Centering

Taking X axis as example, the centering steps are as followed:

1st, manually move the X axis to one side of the workpiece, click [Record X1] button. The software will get and record the machine coordinate.

2nd, move the axis to the other side of the workpiece, click [Centering in X] button. The software will calculate the machine coordinate of the center point based on the current position and the previous position. The center will be set as the origin in X axis.

Please note that when one axis is being centered, the other axis should be motionless. For Y axis, the centering steps are the same as that of X axis as above.

Circle Centering

Circle centering refers to the automatic calculation of the center point of a circular blank via three positions on the circumference. The center point will be set as the workpiece origin.

Taking X axis as an example, the steps of circle centering are as followed:

1st, manually move the tool to one position on the circumference of circular blank, and click [Record P1]. The software will get and record the machine coordinate of the position 1.

2nd, move the tool to another position on the circumference, and click [Record P2]. The software will get and record the machine coordinate of the position 2.

3rd, move the tool to the third position on the circumference, and click [Centering]. The software will calculate the center point based on the machine coordinates of position 3 and the two recorded ones. This center point will be set as the workpiece origin.

CAUTION

For circle centering, it is suggested to choose three points which are evenly distributed on the circumference; otherwise, the centering result may not be accurate.

3.7. Craft Setting

Under “Craft” tab, the user can specify the feedrate, spindle speed, make workpiece compensation and set array machining parameters.

Enable Default Feedrate

Click button next to the item “Enable default feedrate” to open an input box where the user can choose feedrate to be activated, see Fig. 3-26:

Use speed specified in File:

It means the machining feedrate will be the feedrate specified by F command in program file.

Default speed:

It means the machining feedrate will be the feedrate specified by the system setting.

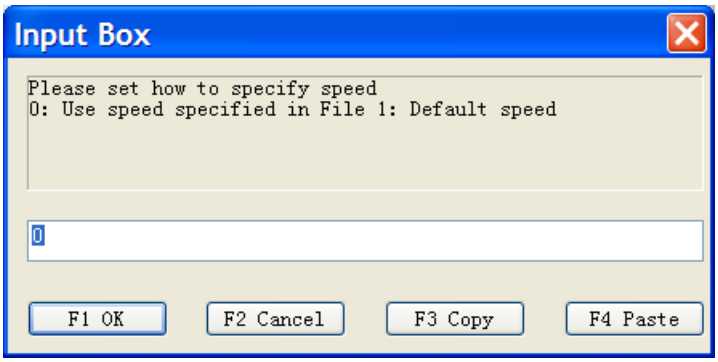


Fig. 3-26 Feedrate specifying box

Enable Default Spindle Revolving Speed

Click button next to the item “Enable default spindle rev” to open an input box where the user can choose feedrate to be activated, see Fig. 3-27:

Yes:

It means the spindle speed will be the speed specified by the system setting.

No:

It means the spindle speed will be the speed specified by S command in program file.

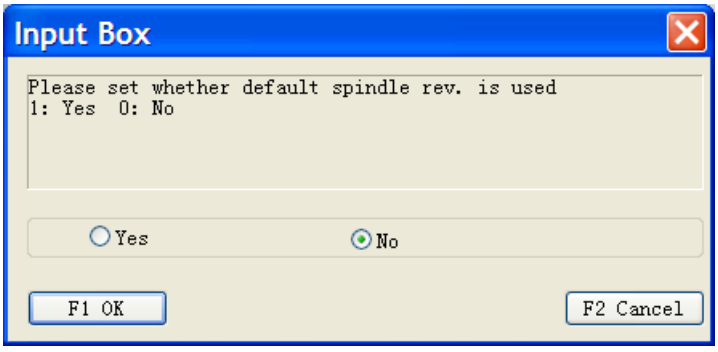


Fig. 3-27 Spindle speed specifying box

Array Machining

This function allows executing array machining on one machining file. Clicking this item will open a dialog box, where the user needs to set row/column number and row/column space (the distance between two workpiece origins in X/Y directions). Besides, the user needs to check the box before the item “Array ON” firstly, as shown below:

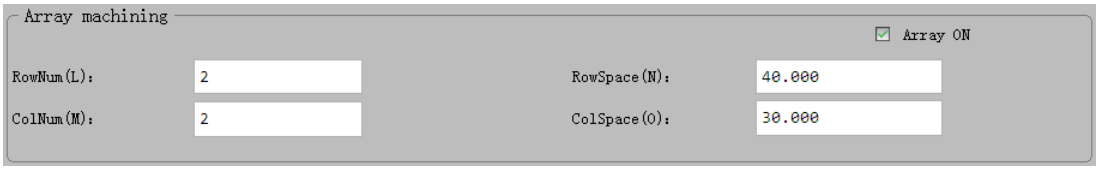


Fig. 3-28 Array machining setting

CAUTION

G codes like G28, G29, G65, G92, M30, M2 and M17 are not supported in array function, neither are subroutines. If they appear in the machining file, the system will prompt to delete them automatically or manually.

Part Compensation Mode

In this dialog box, the user can set workpiece compensation (also called part compensation) related parameters. The compensation mode can be divided into “Single part” and “Array”. The user needs to input values for the workpiece size, choose machining direction and machining order, and set separate compensation values. To finish the operation, check in the box next to the item “IsEnabled” to make all settings effective, see Fig. 3-29.

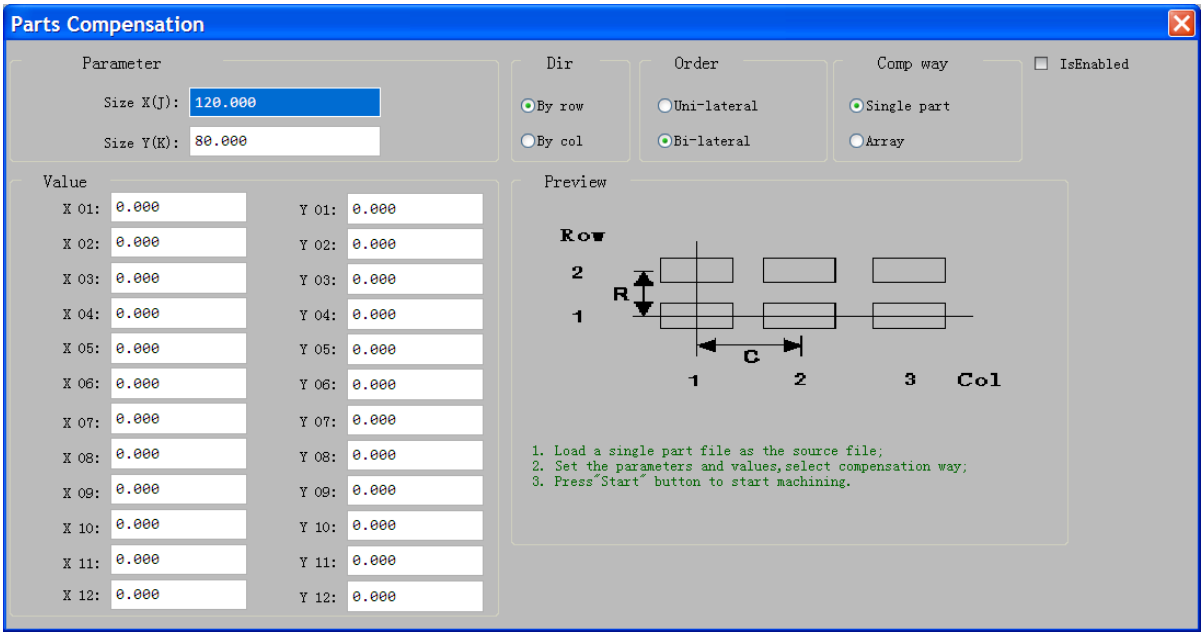


Fig. 3-29 Part compensation dialog

Single part compensation

When this compensation mode is selected, compensation value will be the machining compensation value for each part. For example, items “X01”/”Y01” refer to the compensation distances in X/Y direction for the first part. In this compensation mode, up to 12 parts can be supported.

Array

When this compensation mode is selected, compensation value will be the machining compensation value for one row/column, which decided by the choice of machining direction specified by the item “Dir”. For example, when “By row” is selected, items “X01”/”Y01” refer to the compensation distance for parts in first row; while “By col” is selected, items “X01”/”Y01” refer to the compensation distance for parts in first column in the preview picture. In this compensation mode, 144 (12*12) parts at most can be supported.

CAUTION

Item “Array ON” should be checked before enabling part compensation.

3.8. MDI

In MDI interface, there are 8 input boxes. Click “Edit” button to edit codes, and click “Execute” button to execute the current MDI code. See Fig. 3-30.

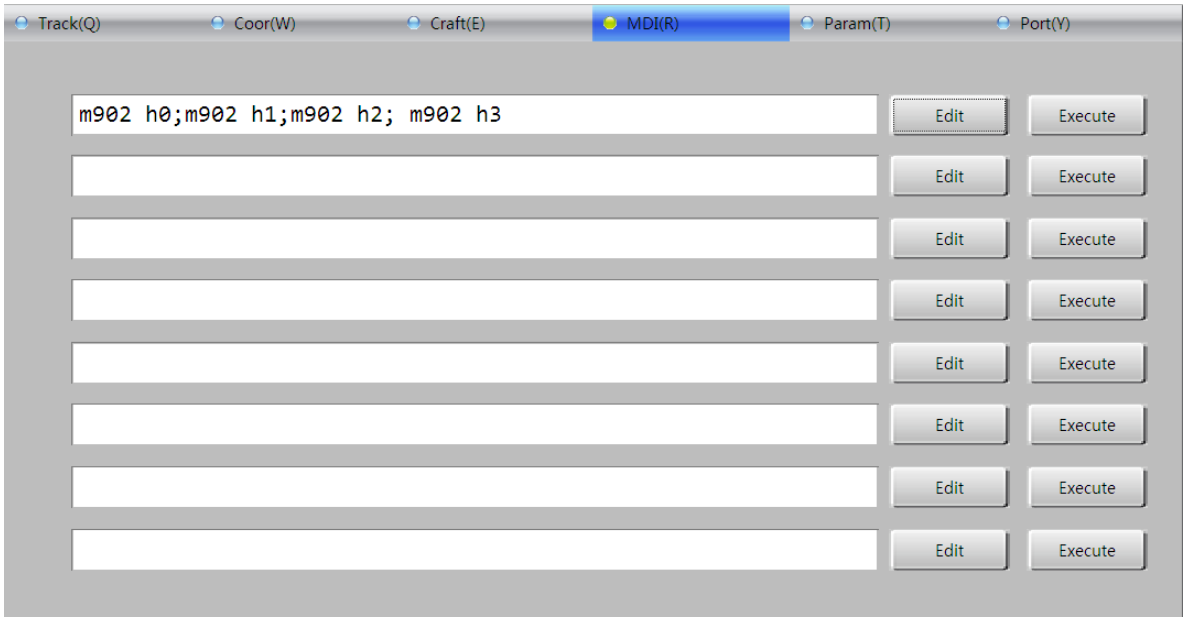


Fig. 3-30 MDI interface

3.9. Parameter

Access to the whole parameters setting is listed in “Parameter” menu on the menu bar. While for those frequently-used or commonly-used parameters, they can be separately arranged under the “Param” tab, for convenience of easy access to those parameters in machining. Please refer to Chapter 6 for detailed information of all parameters.

Click “SetParameters” item in “Parameter” menu on the menu bar to open the “Parameters” dialog box, select parameters commonly-used in debugging or machining, and click the button “Set Common On”. The parameter then is defined as frequently-used and will be listed under “Param” tab. The user can directly modify them under “Param” tab, instead of accessing the “Parameter” dialog box. See Fig. 3-31.

If the user wants to remove a parameter from the “Param” tab, find it in “Parameters” dialog box and click button “Set Common Off”.

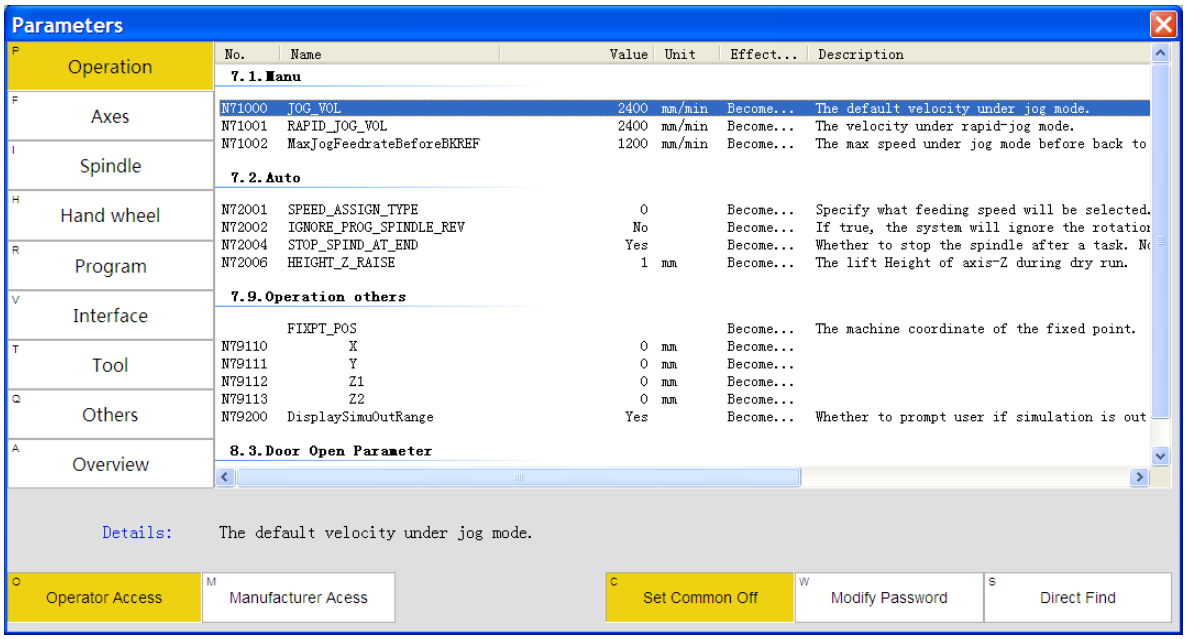


Fig. 3-31 Frequently-used parameters interface

3.10. Port

Under “Port” tab, information about the input and output ports are displayed and corresponding operations can be done. In port info interface as shown in Fig. 3-32, a filled dot before port name stands for an input port while a hollow dot for output port.

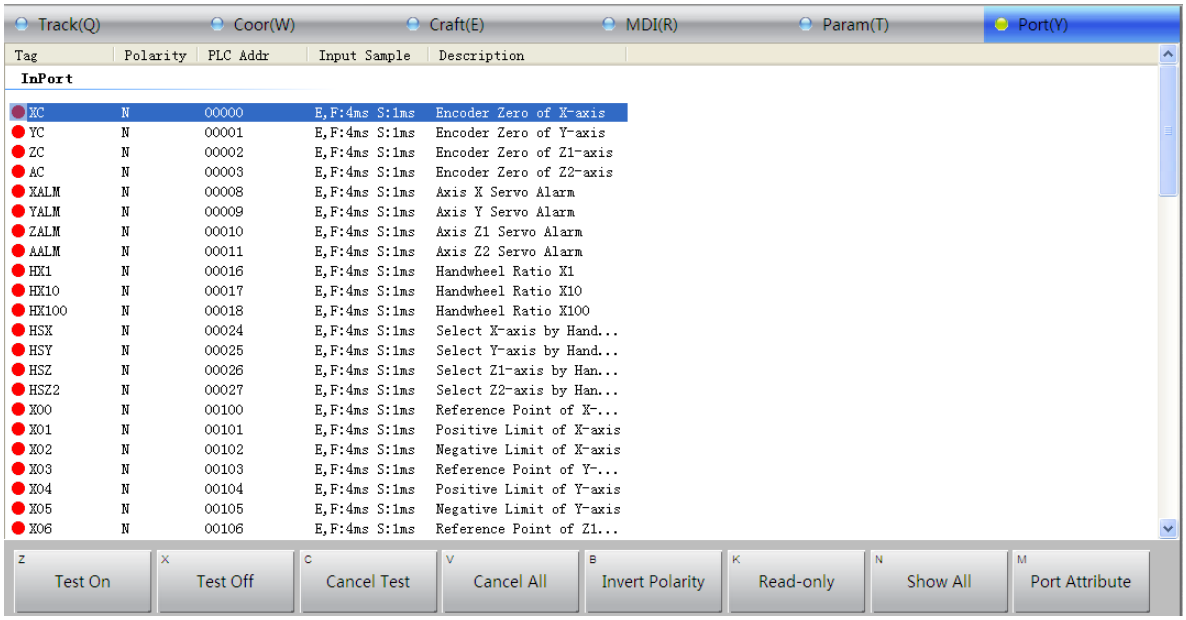


Fig. 3-32 Port info interface

Here is introduction to buttons at the lower part of the interface.

Test On/Off

Clicking [Test On] and [Test Off] buttons can make the color of the dot before the port name switch between green and red. Green dot means the port receives signal while red dot means the port receives no signal.

The two buttons are mainly used to analog hardware signals when conducting simulation test.

CAUTION

Icons for the port indicator are different under testing condition and the real machining condition:

Under testing condition, green indicator is shown as  while red indicator as  .

Under real machining condition, green indicator is shown as  while red indicator as  .

Cancel Test/Cancel All

Clicking the buttons will cancel analog signal and cancel simulation test. And actual hardware signal will take place of the analog signal. Button [Cancel Test] is used to cancel the test on the currently-selected port while button [Cancel All] is used to cancel tests on all testing ports.

Invert Polarity

Please note that polarity of ports for feedrate, spindle override, mode switch, handwheel and encoder zero must be N under any circumstances.

Read-only

After a port set as read-only, test cannot be conduct on the port, namely buttons [Test On], [Test Off] and [Invert Polarity] are unavailable for those read-only ports.

Show All

To show all the ports including hidden ports or reserved ports.

Clicking the button will display all ports and clicking again will hide the used ports.

Port Attribute

After the button is clicked, a dialog box named “Sampling setting” will pop up, see Fig. 3-33.

In sampling setting box, drag the slide to set sampling interval. Checking items “Filter” and “Port Enabled” can enable the function.

In addition, port name and its description can be modified.

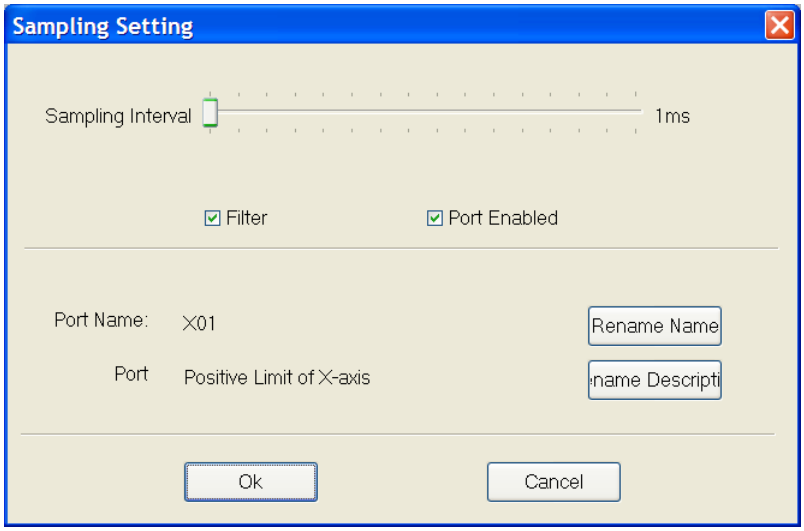


Fig. 3-33 Sampling setting dialog

3.11. PLC

On most occasions, this window is not displayed on functional windows area, the user needs to manually display it. You can click “Customer Interface Order” item in the list of “Window” menu on the menu bar, and make “PLC” upward in the list.

In this window, the user can show the channels, timer and counter. See Fig. 3-34.

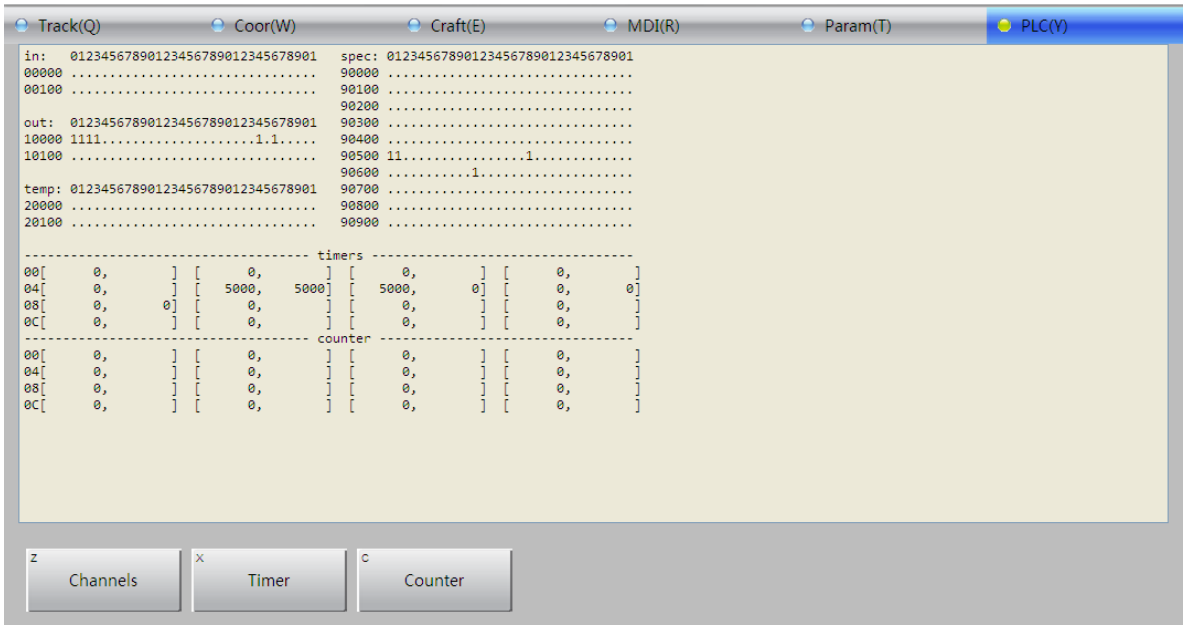


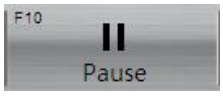
Fig. 3-34 PLC info window

3.12. Machine Control Bar

The machine control bar interface contains three separate subdivisions, which are Auto mode interface, Manual mode interface and Reference point mode interface. Furthermore, manufacturer’s password will be asked to activate settings in Reference point mode interface. The user can press combination key “Ctrl+1/2/3” or select by mouse directly to switchover among three modes.

In Auto/Manual/Reference modes, buttons [Start/Pause/Stop/Resume] are all available. Here is detailed information.

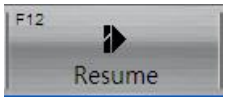
Button [Start], shortcut key F9. After program file loaded into the system and the reference point returned, clicking this button will make the system start machining from the beginning automatically.



Button [Pause], shortcut key F10. Clicking this button will make the system suspend machining and turn it into pause state. The user needs to click button [Start] or relevant menu item to continue machining at this time.

Button [Stop], shortcut key F11. Clicking this button can also stop the system during returning to the reference point.

This button is available in Auto mode or when the user inputs well-working G-code. Clicking this button will make machine terminate machining and conclude the machining task. The system will then turn into idle state. It is a way to normally terminate the task during machining.



Button [Resume], shortcut key F12. Normally, the button is not recommended to use. When power failure or emergent stop occur during machining, or button [Stop] has been clicked, the user can use this button to resume machining on condition that the user can definitely assure the accuracy of the current workpiece coordinate. However, if the user cannot guarantee its accuracy, it is strongly recommended to return to the reference point again firstly before resuming machining. This button is also applicable to simulation.

3.12.1. Auto Mode

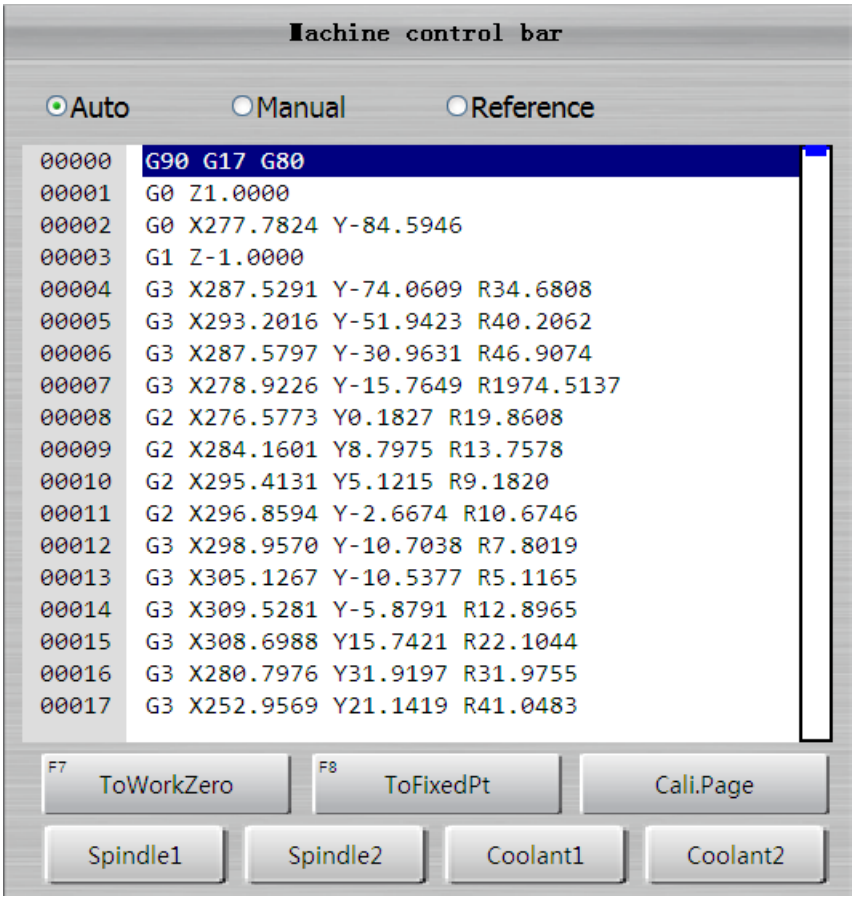


Fig. 3-35 Machine control bar-Auto mode

The Auto mode window shows program lines of the currently-opened file. At present, NcStudio supports the following file formats: G code of ISO standard, HP plotter (HP PLT) format, DXF format, JDPaint ENG format and NCE format exclusively owned by Weihong Company. The user can view and edit the current machining file in this window.

Button [ToWorkZero], shortcut key F7. It can make axes return to the workpiece origin.

Button [ToFixedPt], shortcut key F8. It can make axes return to the position of fixed point, whose machine coordinates are specified by parameter N79110~N79113.

Button [Cali.Page]. Click this button to open the calibration interface. Please refer to Chapter 3.6.2 for more information.

Buttons [Spindle 1/2] are used to turn on/off the spindle 1/2. The button with a sunken appearance means the spindle has been turned on.

Buttons [Coolant 1/2] are used to turn on/off coolant. The button with a sunken appearance means the coolant has been turned on.

The following functional buttons are exclusively available in auto mode, see Fig. 3-36.

Fig. 3-36 Functional keys in Auto mode

Load

Shortcut key F1. It is used for opening program files on the hard disk. Click this button to open a dialog box named “Open and Load”, as shown below:

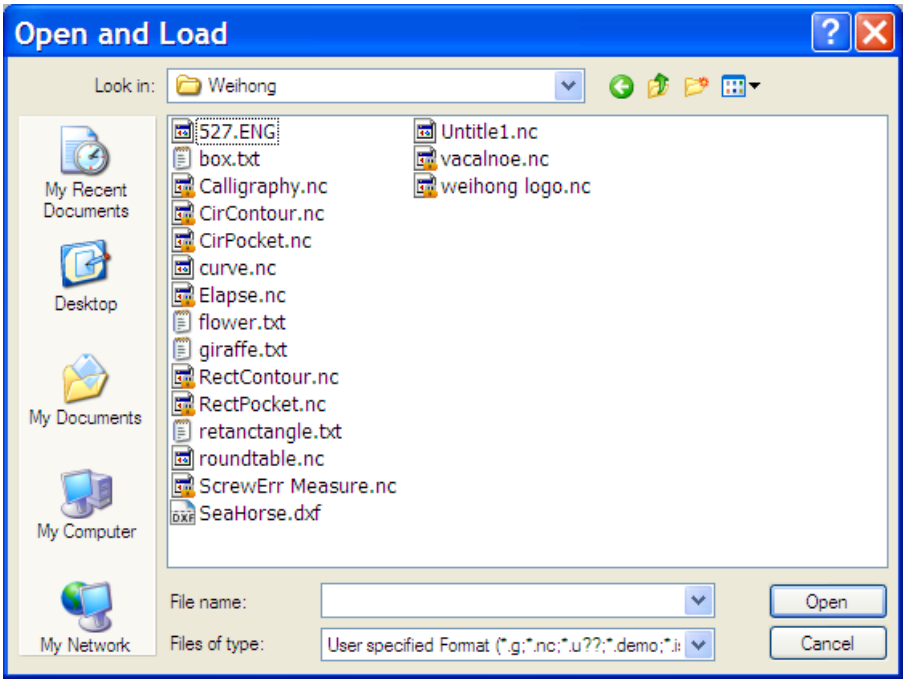


Fig. 3-37 Open and load box

Unload

Shortcut key F2. It is used to unload the current program file.

Edit

Shortcut key F3. Click the button to open the following “Program Editor” box, where the user can do operations such as “Insert” lines, “Delete” lines, “Copy” lines, “Find” and “Replace” lines. After editing, save it and exit.

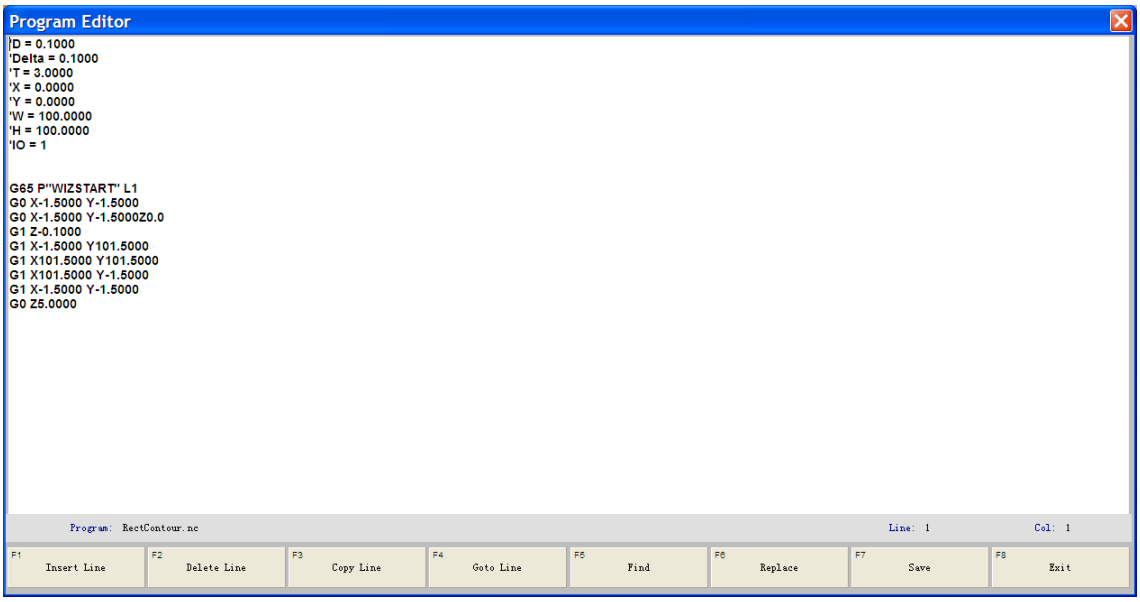


Fig. 3-38 Program editor

GOTO

Shortcut key F4. It is also known as advanced start function. Exclusively available in Auto mode,

This function allows selecting any blocks for machining. The user can choose the processing range in the dialog box according to the type of machining file. Click the button to open a dialog box named “Execute (Advanced Options)” as shown in Fig. 3-39. The function is also applicable to simulation.

There are three ways to define the machining range, as you can see. Make your own choice according to the tips in the dialog box above.

CAUTION

1) If a G code file is loaded, only “Specify the range according to the row number” is available, while the other two unavailable.
2) If an ENG file with version 5.01 and lower is loaded, the three options are all available.

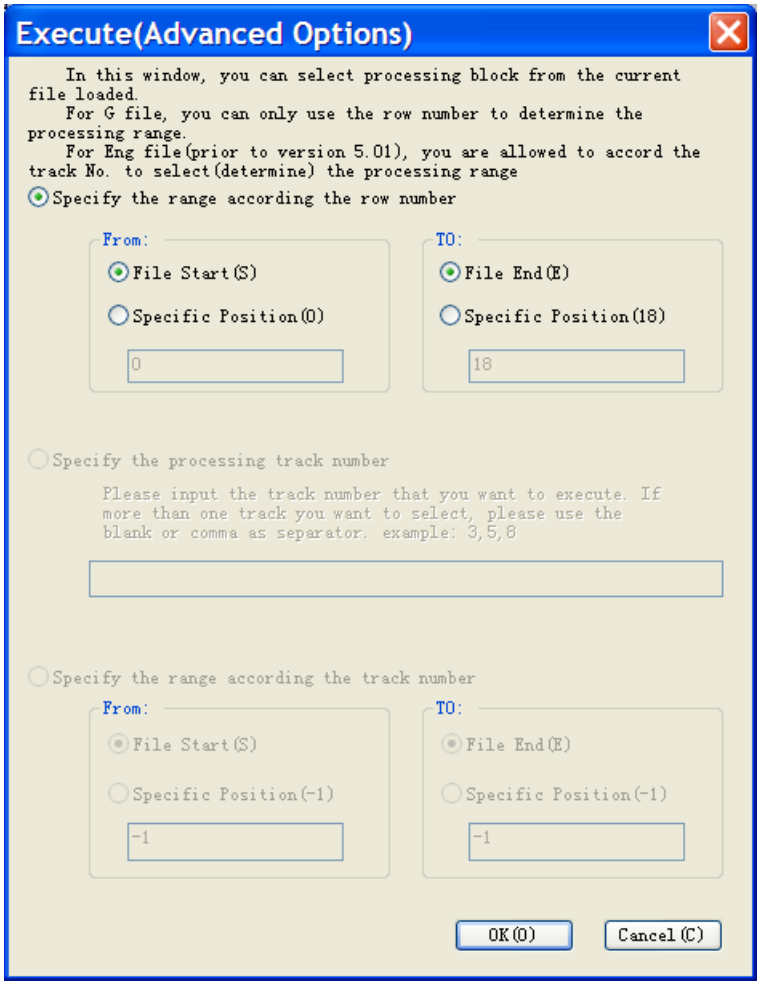


Fig. 3-39 Advanced Options

Selecting option “Specify the range according the row number”, the system will process the program lines within line number defined here.

Selecting option “Specify the processing track number”, the system will process all program files according to the track number.

Selecting option “Specify the range according the track number”, the system will process the program lines within range defined by the starting and end track number.

Simulation

Shortcut key: F5. The function is only available in auto mode. After program file has been loaded into the system, clicking this button to activate simulation mode and then clicking [Start] button will make the system start machining from the beginning in simulative mode.

In simulation mode, differing from actual processing, the system just displays the tool path in the *Track* window at a high speed, without any actual machine actions. Through simulation, the user can view the machine tool’s movement path in advance to avoid equipment damage possibly caused by programming errors and learn other information.

HW Guide

With this function, machining feedrate can be manually control in auto machining. The user can choose this function to protect the tool from dangers caused by incorrectly loaded program file or other situations.

In Auto mode, with the button clicked, the system will implement the machining file with the turning of handwheel when [Start] button is pressed down. When the handwheel stops turning, machining will also stop. What's more, the machining speed changes with the turning speed of handwheel.

The system supports backward machining (or reverse machining) via HW guide. If any wrong occurs in machining, the user can reversely turn the handwheel to make the system machine reversely.

3.12.2. Manual Mode

The *Manual* window offers an interactive interface for users' manual operations. As shown in Fig. 3-40.

Fig. 3-40 Manual mode window

Numeric Direction Buttons

and


correspond to the positive and negative direction of X axis respectively;

and 

correspond to the positive and negative direction of Y axis respectively;

and 

correspond to the positive and negative direction of Z1 axis respectively;

and 

correspond to the positive and negative direction of Z2 axis respectively;

Selection of Feed Mode

There are three kinds of feed mode in manual operation mode, namely jog, handwheel and stepping. Here is the introduction.

Jog

Check 

to select jog mode.

Press a numeric direction key on the keyboard to control the axis motion. When the key is pressed, the axis moves at normal jog or rapid jog speed; when the key is released, the axis stops.

Handwheel

The user can set parameter “N52022 HW_SELECTION” to choose three axes handwheel or handwheel with more axes.

Check 

to select handwheel mode.

Turn the “Axis Select Knob” to activate an axis to move.

Turn the “Magnification Gear” to decide the magnification ratio for the moving distance of a tool. The moving distance of a tool for one division equals to the minimum increment times the magnification of the gear.

Turn the “Turning Wheel” to manipulate the movement of a tool, with turning direction corresponding to the moving direction of tool.

Fig. 3-41 Handwheel

Stepping

In stepping mode, there are several stepping lengths available. As shown below:

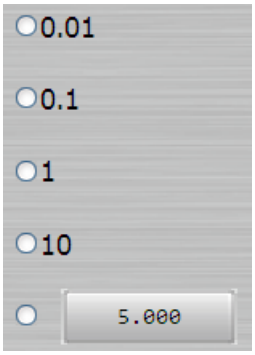


Fig. 3-42 Stepping length

In stepping mode, the machining track in *Track* window is displayed in the color of G01 code.

The user can implement stepping operation via the mouse, keyboard, or operation panel. Once a numeric direction key is triggered, the corresponding axis will move a fixed step length.

Click [5.000] button to open a dialog box where you can set a step length, as illustrated below.

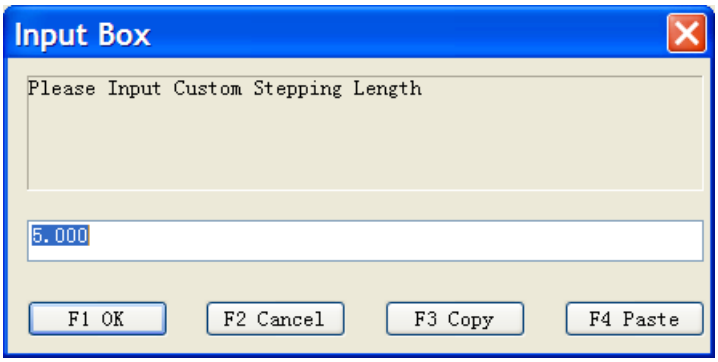


Fig. 3-43 Custom stepping length

CAUTION

1) The customized step length should not be set too large in case of equipment damage caused by mal-operation.
2) As it takes the system a certain period of time to execute each jog order, so please avoid frequent and repeated clicking. Otherwise, the system will give out an error prompt as "Unable to perform the action because the last one is not completed yet."

The following buttons are only available in manual mode.

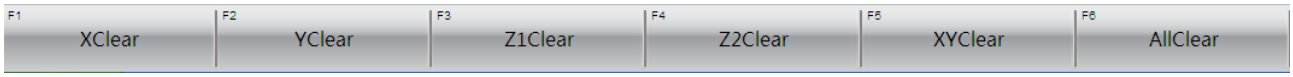


Fig. 3-44 Functional buttons in Manual mode

XClear

Shortcut key F1. With the button clicked, the system will modify offset to make the current X axis workpiece coordinate 0.

YClear

Shortcut key F2. With the button clicked, the system will modify offset to make the current Y axis workpiece coordinate 0.

Z1/Z2Clear

Shortcut key F3/F4. With the buttons clicked, the system will modify offsets to make the current Z1/Z2 axes workpiece coordinate 0.

XYClear

Shortcut key F5. With the button clicked, the system will modify offsets to make current workpiece coordinates of both X and Y axes 0.

AllClear

Shortcut key F6. With the button clicked, the system will modify offsets to make current workpiece coordinates of all axes 0.

3.12.3. Reference Point Mode

In reference mode, settings related to machine returning to the reference point are offered, including automatic returning to the REF point, the returning direction and locating speed. As shown in Fig. 3-45.

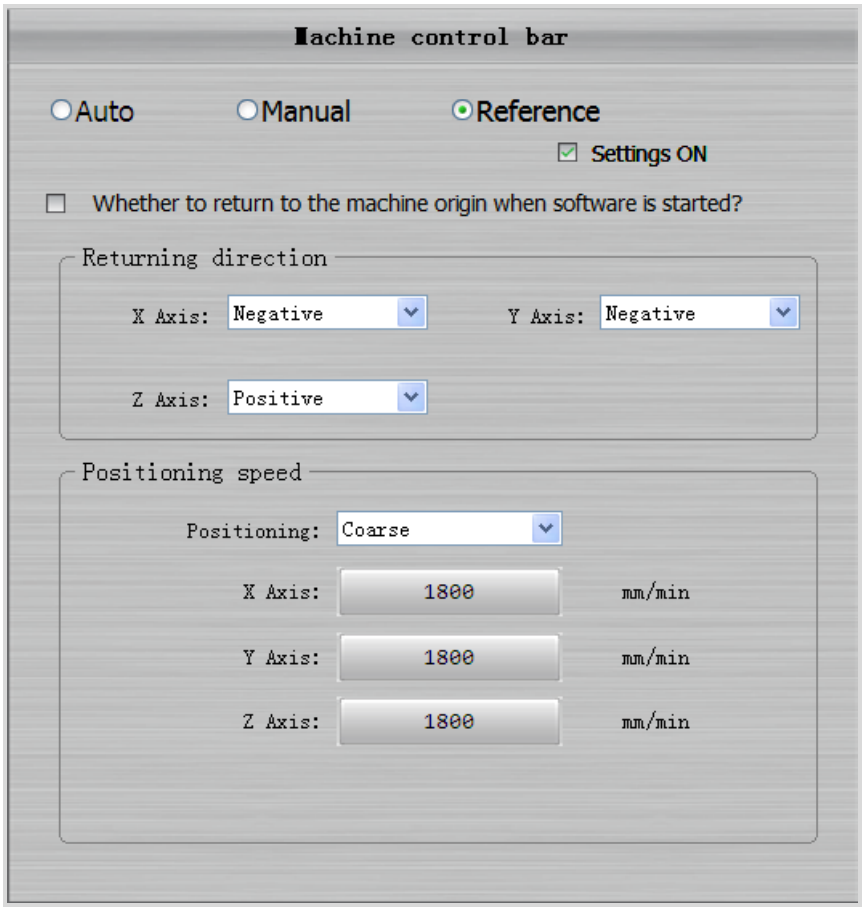


Fig. 3-45 Machine control bar-Reference point mode

As shown above, the user needs firstly check the item “Settings ON”. Manufacturer’s password will be asked at this time. If the item “Whether to return to the machine origin when software is started?” is checked, machine will automatically return to the REF point the moment the software is started.

The following buttons are only available in the reference mode. Here is detailed introduction.

Fig. 3-46 Functional buttons in REF point mode

X/Y/Z1/Z2 Bkref

Shortcut keys are F1/F2/F3/F4 respectively. It will make single axis return to the REF point. It is recommended to return Z axes to the REF point first, otherwise, prompt will pop up as below:

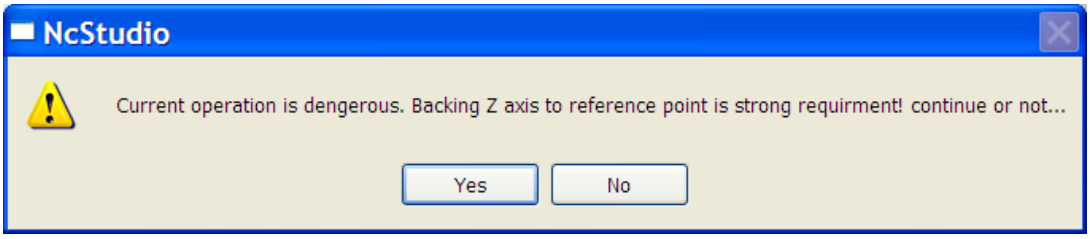


Fig. 3-47 Prompt box

All Bkref

With the button clicked, all axes will return to the REF point in the order of “Z-XY”.

HW Guide

In reference mode, the user can also execute axis returning to the REF point via handwheel.

4. Menu System of NcStudio

4.1. File Menu

The optional items in “File” menu is as following:

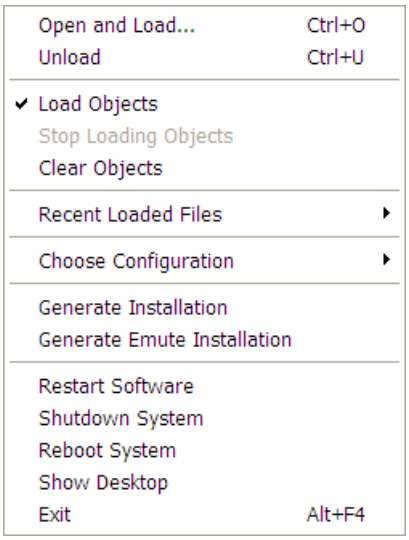


Fig. 4-1 File menu

Open and Load

Shortcut key: Ctrl + O. The item is used for opening a machining program file on the hard disk. Clicking the menu item will open a dialog box titled "Open and Load" dialog box, as shown below:

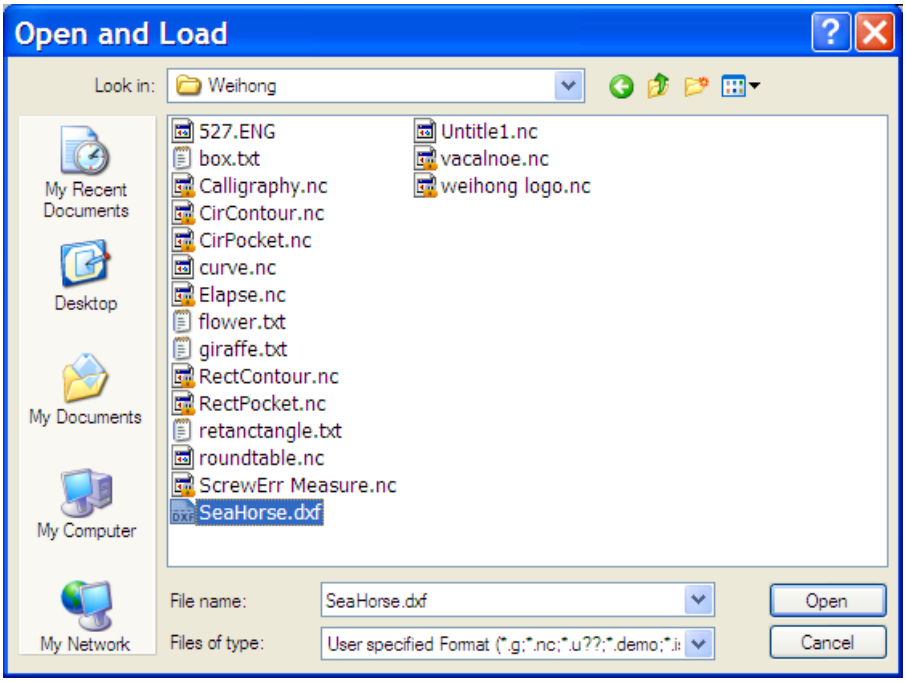


Fig. 4-2 Open and load box

Unload

Shortcut key: Ctrl + U. It is used to unload the current machining file.

Load Objects

This item is used to load the track of a program file one time when pre-analyzing the file. While in actual machining, the real-time track will also be simulated. The two kinds of track are displayed in different color. Selecting this item is to load the track beforehand. When you want to remove it, select the opposite item “Clear Objects”.

Stop Loading Objects

This item is used to terminate the loading of machining track. An over large track will slow down the system running process; therefore, there are two options to solve the problem. One, choose this item to stop loading of the track; two, select “Clear Objects” item to remove all loaded track and stop loading the remaining track.

Clear Objects

This item will remove all loaded track and stop loading the remaining track. Note that selecting this item will deselect its opposite “Load Objects”.

Recent Loaded Files

A sub-menu is included in this item, where recently loaded program files will be shown in the list. The user can quickly load anyone of them into the system again.

Choose Configuration

It contains alternative configuration and linkage configuration. Please note that the configuration can only be chosen by manufacturer to protect machine from possible damage.

Generate Installation / Emute Installation

Clicking the two items, developer’s password will be asked firstly. After privilege confirmed, a language selection dialog box will appear, see Fig. 4-3.

Fig. 4-3 Language selection

This menu item is mainly used for key data backup and restoration, system installation and installation package generation, etc. Selecting this menu item will close the NcStudio for a while and pop out a dialog box titled “System Maintain”, including “Backup key data”, “Key data resume”, “System installation”, and “Generate installation packet”, as shown in the figure below:

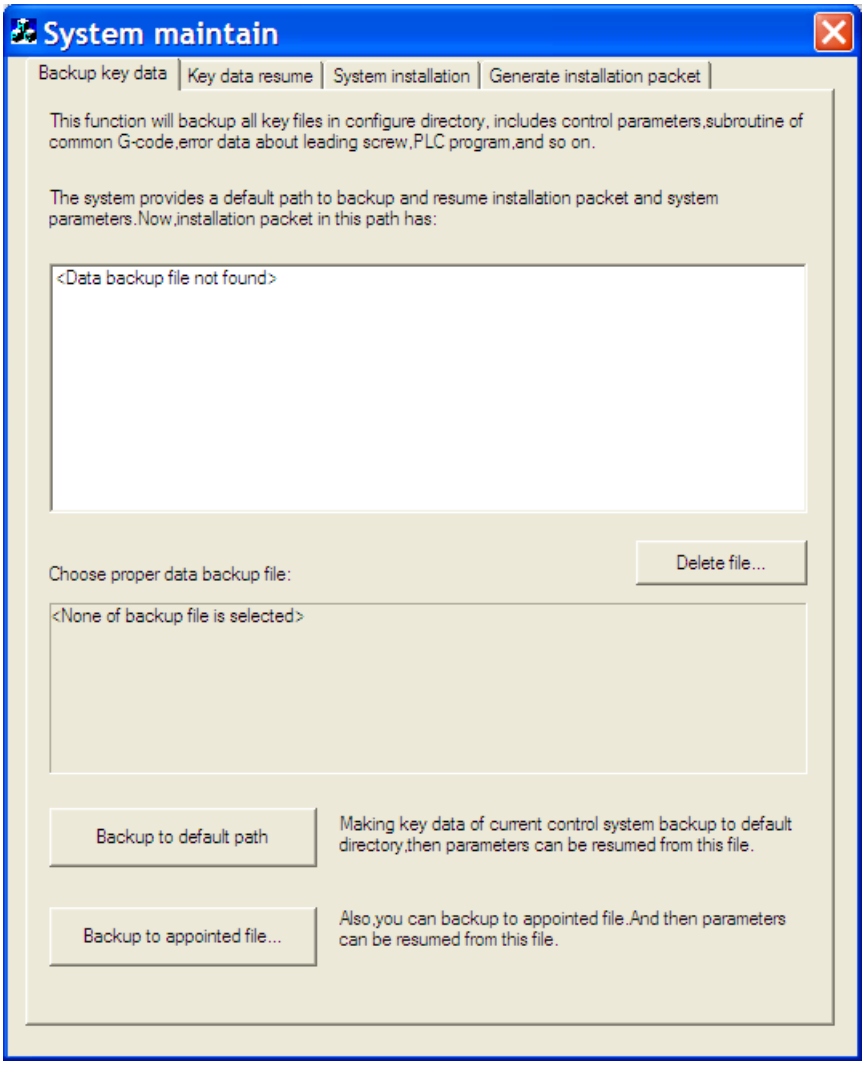


Fig. 4-4 System maintain

When the tab “Backup Key Data” is active, the user can backup all the key files in the installation directory, including controls parameters, subroutines of common-G code, screw error data, PLC program, etc. If there have been backup files under the default directory, they will be displayed in the top white panel. The user can delete or preserve them, and check the backup file property in the window below by clicking the file name. The backup operation buttons are under the indented panel. The user can select whether to save the files into the default path or to the appointed path.

When the tab “Key data resume” is active, the user can resume the key files according to the backup files. If there have been backup files under the default directory, they will be displayed in the top white panel. Clicking the file name, the user can check its property in the indented panel below and execute [Delete] and [Start resume] function. The user can also use the direction keys to select other backup files. If there have been backup files and they are saved in other paths, the user can click [Browse] to search. Clicking the [Start resume] button which is at the bottom of the tab control will start resuming.

When the tab “System installation” is active, the user can reinstall NcStudio system. But it should be paid special attention that after the re-installation, all the former files under the current installation directory will be deleted and the new version won’t keep any data of the old version. So it’s recommended to do key data backup before the reinstallation. If there have been installation packages under the default directory, they will be displayed in the top white panel. Clicking one of the packages, the user can check its property in the indented panel below and execute [Delete] and [Start installation] function. The user can also use the direction keys to select other packages. Clicking the [Start installation] button which is at the bottom of the tab control will start installing.

When the tab “Generate installation packet” is active, the user can create a complete installation package on the basis of current system data, which is useful to backup system files and save a stable version of the system. If there have been packages under the default directory (the same with the installation package path under the tab “System installation”), they will be displayed in the top white panel. Clicking one of the packages, the user can view its property in the indented panel below, and execute [Delete] or check other package properties with the direction keys. At the bottom of the tab control, the user can generate the installation package into the default path or into the appointed path.

The operations to generate emulation installation are the same with operations to generate installation package as above. Please note that emulation installation generation gets a simulation software or emulation software instead of an original one.

Restart Software

With this item selected, the software will be restarted.

Shutdown System

It is used to shut down the system. With this item selected, a prompt dialog box for shutdown confirmation will pop up. Click [Yes] to shut down the system and [No] to cancel it.

Reboot System

It is used to restart the system. With this item selected, a prompt dialog box for restart confirmation will pop up. Click [Yes] to restart the system and [No] to cancel it.

Show Desktop

It is used to switch to the desktop interface.

Exit

It is used to shut down and exit the NcStudio system.

4.2. Operation Menu

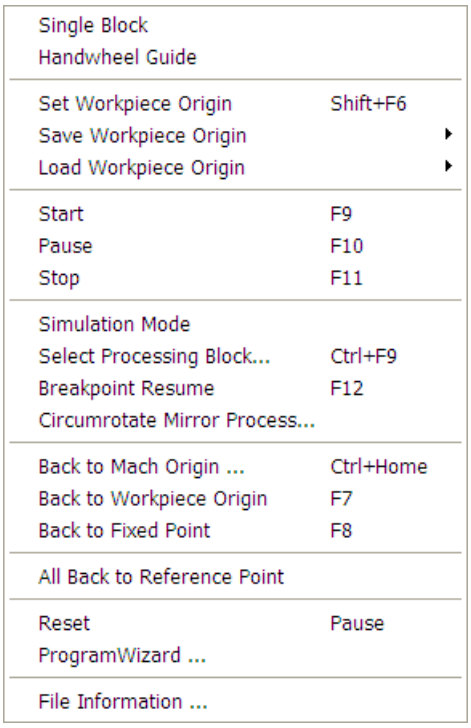


Fig. 4-5 Operation menu

Single Block

With the function activated, every time you click [Start] button, the system runs a program line and pauses; when you click [Start] button again, it will run the next program line.

The user can choose this function before actual machining since it is helpful for error diagnosis and failure recovery.

Please note that when the system is in non-idle state, the user can not enable or disable single block function.

Handwheel Guide

In Auto mode, with the item activated, the system will implement the machining file with the turning of handwheel when [Start] button is pressed down. When the handwheel stops turning, machining will also stop. What's more, the machining speed changes with the turning speed of handwheel.

The user can choose this function before machining to learn that whether the machining file is correct.

Set Workpiece Origin

Shortcut key: Shift+F6. It used to make the current workpiece coordinate the workpiece origin. Please refer to Chapter 3.6.1 for details.

Save Workpiece Origin

This function saves the current workpiece origin as well as the name of the file and the machine coordinates of current point into NcStudio, so that the saved workpiece origin can be easily found later. 10 groups of data can be saved at most.

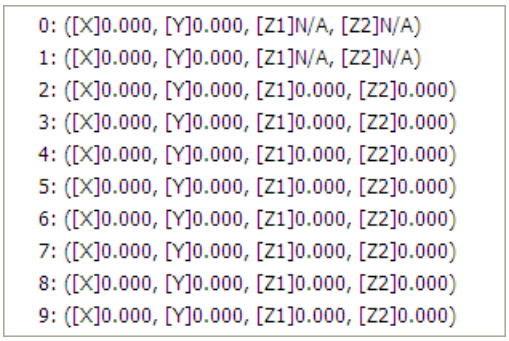


Fig. 4-6 Save workpiece origin

Load Workpiece Origin

It is used to read and load the workpiece origin which has been saved before.

Start/Pause/Stop

Please refer to Chapter 3.12 for details.

Simulation Mode

With the item clicked, the software will enter into simulation mode, namely, all operations are executed in simulation mode.

In simulation mode, differing from actual processing, the system just displays the tool path in the *Track* Window at a high speed, without any actual machine actions. Through simulation, the user can view the machine tool's movement path in advance to avoid equipment damage possibly caused by programming errors and learn other information.

Select Processing Block

Shortcut key: Ctrl+F9. This item is only available in auto mode. Please refer to Chapter 3.12.1 for more information.

Breakpoint Resume

Shortcut key: F12. With the function, the system will continue machining from the interrupted program line.

When power failure, emergency stop occur, this function can make machine directly move to the interrupted point and resume machining, in order to save time. The user can also conduct this operation via [Resume] button in machine control bar.

Circumrotate Mirror Process

Mirror rotate machining refers to generating a mirrored or rotated program path relative to the source program path. With the menu item chosen, a dialog box will pop up, as shown in Fig. 4-7.

After the user selects one of the above options and clicks [OK], the newly-generated machining file will be loaded into NcStudio automatically.

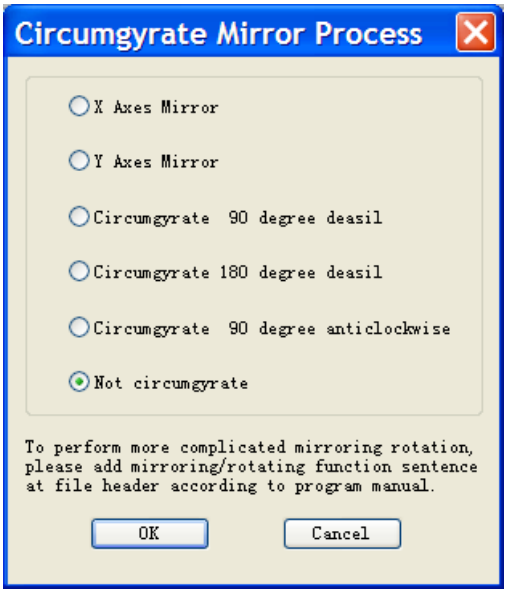


Fig. 4-7 Mirror and rotate

Back to Mach Origin/Workpiece Origin/Fixed Point

Clicking shortcut key “Ctrl+Home” will make machine return to the REF point. When item “Back to Mach Origin” is clicked, a dialog box will pop up, see Fig. 4-8.

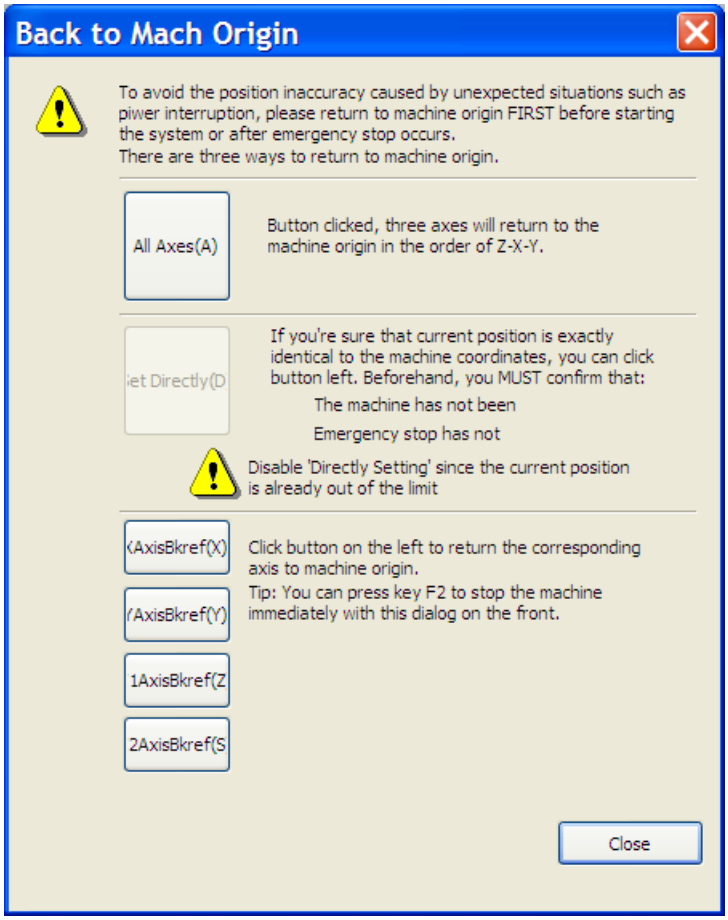


Fig. 4-8 Return to the REF point

[All Axes]: all the axes will go home successively (Z axis first and then X and Y axes.).

[Set Directly]: directly setting the current machine coordinates as correct ones. Before executing this function, the user must confirm that the current X, Y, Z coordinates are right machine coordinates. If the machine tool was turned off or underwent an E-stop before, it is not recommended to execute this function on most occasions.

[X Axis Bkref], [Y Axis Bkref] , [Z1 Axis Bkref] and [Z2 Axis Bkref]: homing the corresponding axis alone.

After all the axes have returned to machine origin, a mark "◔" will appear before each axis in the NC state window.

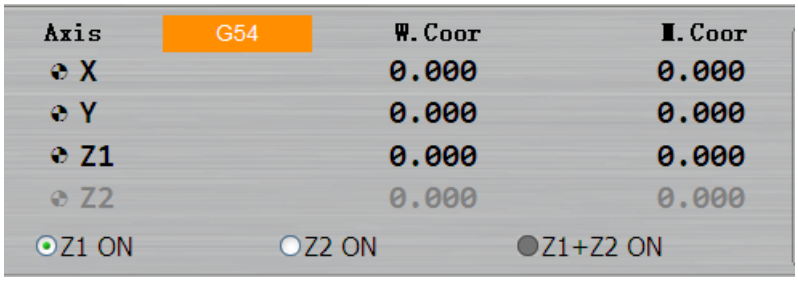


Fig. 4-9 Axis with homing mark

Back to Workpiece Origin

Shortcut key: F7. With the item clicked, when the tool nose is below the safe height, Z1/Z2 axes will firstly move upward to the safe height position before X/Y axes returning to the workpiece origin in an linkage way. When the tool nose is above the safe height, X/Y axes will firstly return to the workpiece origin together before Z1/Z2 axes moving downward to the safe height position. The user can also use button [ToWorkZero] in machine control bar to conduct this operation.

Back to Fixed Point

Shortcut key: F8. With the function, machine will automatically move to the fixed point, where the user can change tool or change workpiece in a convenient way. The fixed point is decided by settings of parameters N79110/N79111/N79112/N79113.

All Back to Reference Point

With the button clicked, all axes will return to the reference point according to the order Z1/Z2-X/Y.

Reset

Shortcut key: Pause. The button is available on all occasions. With it clicked, machine will instantly stop the current machining task (if it is machining), and restore the system from alarming state (if the alarming is ongoing) to idle state.

Program Wizard

With the item selected, an "Program Wizard" dialog box will pop up, including 5 function windows: [Cir Contour], [Cir Pocket], [Rect Contour] and [Rect Pocket], Bottom milling and frame milling can be completed in the first four windows simply by entering the values of relevant parameters ("Inner" and "Outer" are used to specify whether milling the inner part or the contour).

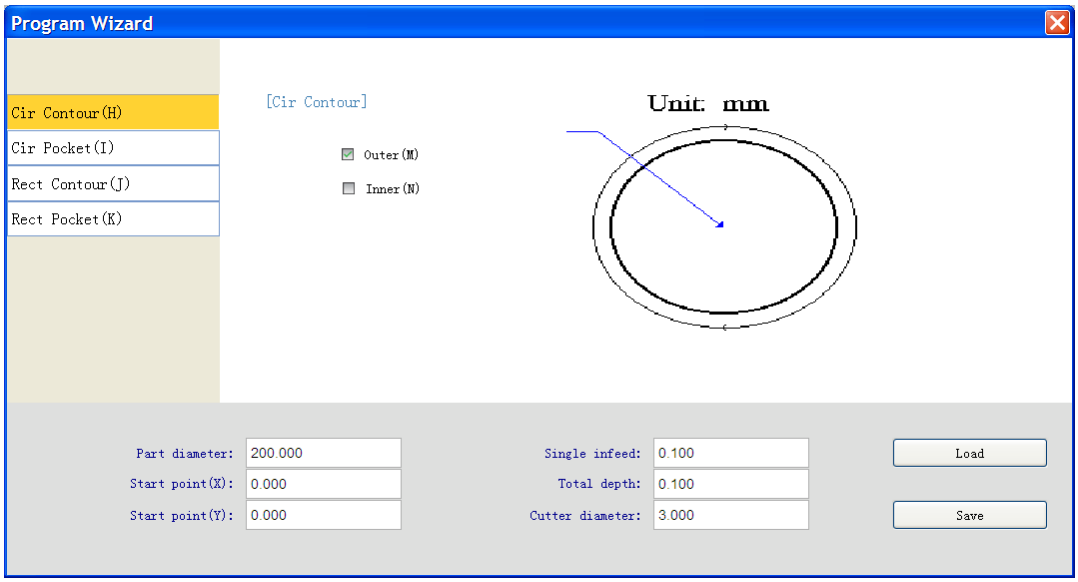


Fig. 4-10 Circle contour

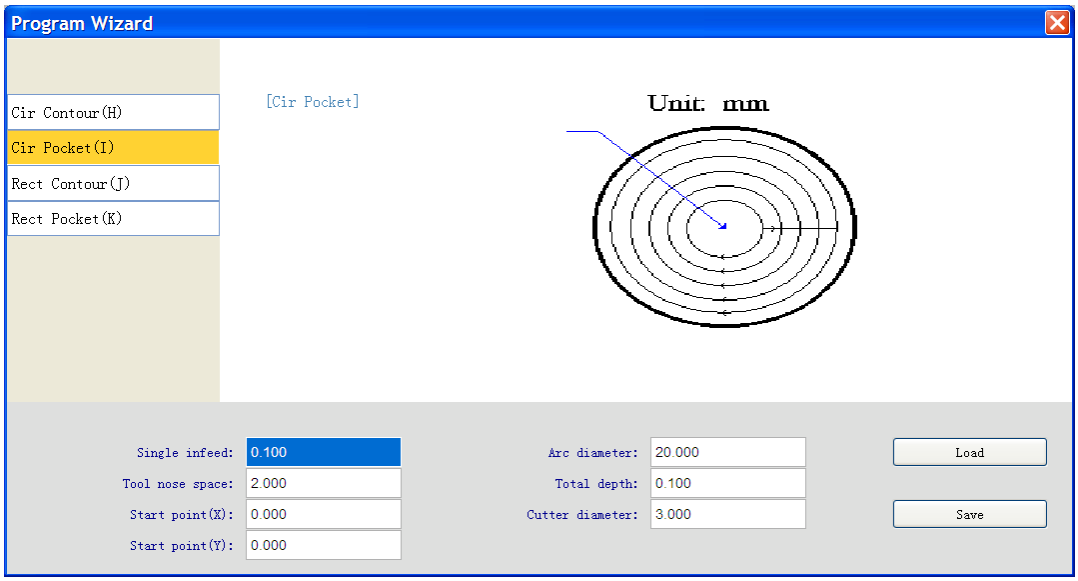


Fig. 4-11 Circle pocket

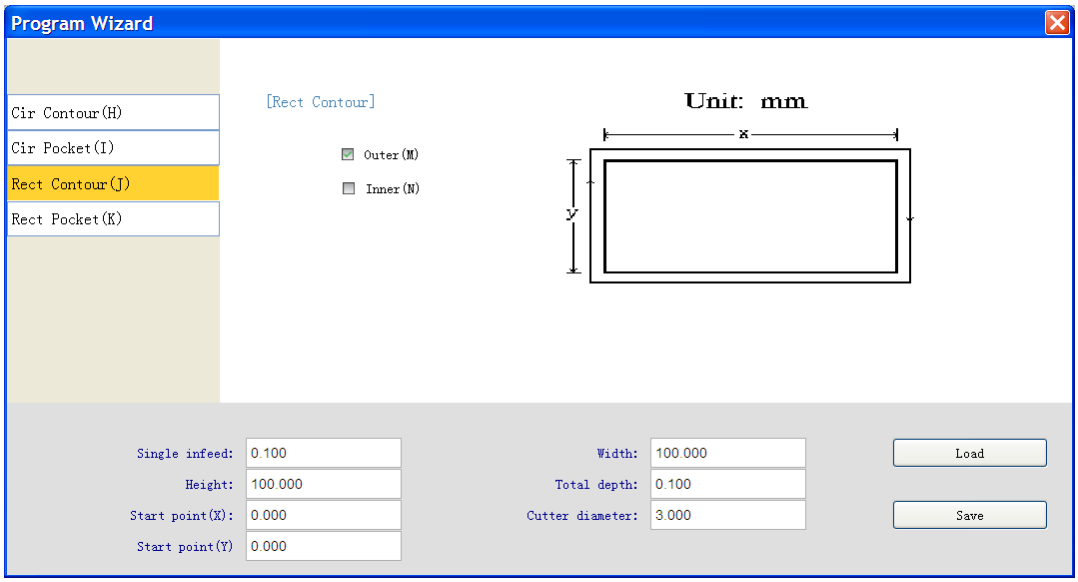


Fig. 4-12 Rectangle contour

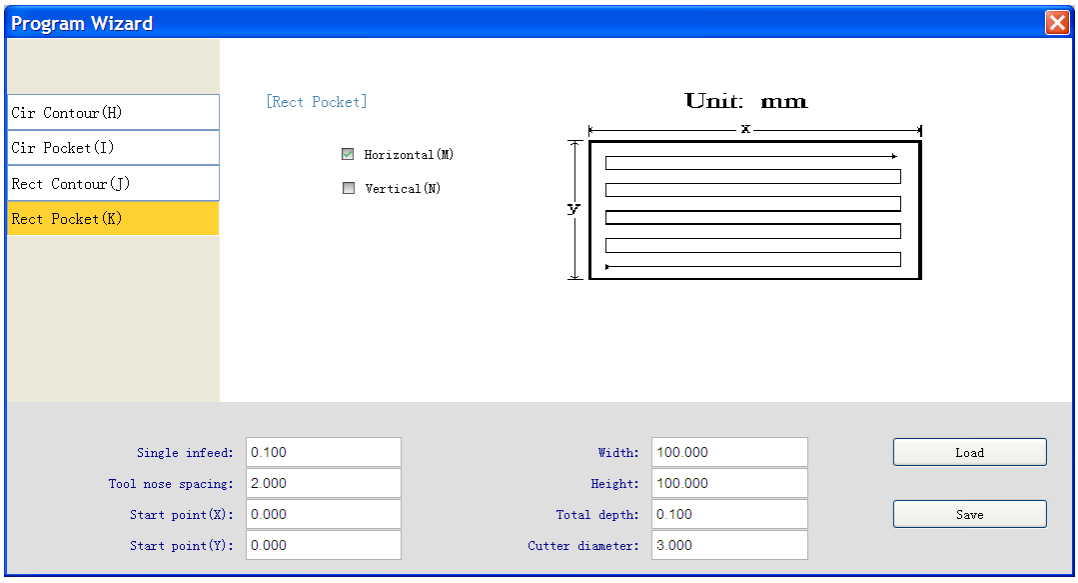


Fig. 4-13 Rectangle pocket

After setting all parameters in corresponding above dialog box, click [Save] button to save parameter setting. Click [Load] button will make the program shape appear in *Track* window.

File Information

The user can obtain information including machining time, travel range as well as cutter moving length, etc. As shown in Fig. 4-14.

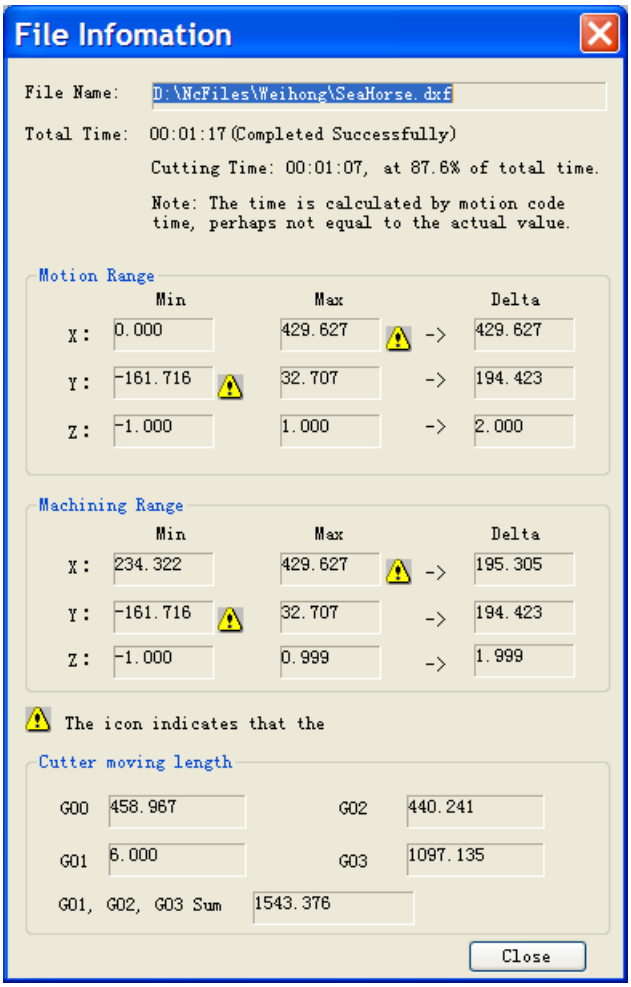


Fig. 4-14 File information dialog

4.3. Parameter Menu

Here is the menu list of “Parameter” menu:

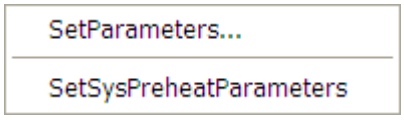


Fig. 4-15 Parameter menu

Set Parameters

Please refer to Chapter 6 for details.

Set System Preheat Parameters

To facilitate machine maintenance and maximize its service, Warm-up and Test-run (also known as machine preheat function) function is developed. The user can enable this function before formal machining or shutting down the system to balance lead screw wear. Machine “Warm-up & Test-run” is completed via repeatedly movement. To enable this function, set parameter “N89360 WhetherToCuttingFluidWhentartMachining” to “Yes”.

In following situations, “Warm-up & Test-run” can be enabled:

1) Shut down the software;

2) Restart the software;

3) Shut down the system;

4) Restart the system;

5) All axes have returned to the reference point.

Before function activation, a prompt dialog will pop up for confirmation, select “Yes” to start machine warming up and test running, and select “No” to cancel it.

Before machine warming up and test running, relevant parameters should be set firstly, which can be found under “Parameter” menu. Clicking the menu item “SetSysPreheatParameters” will open a dialog box as below:

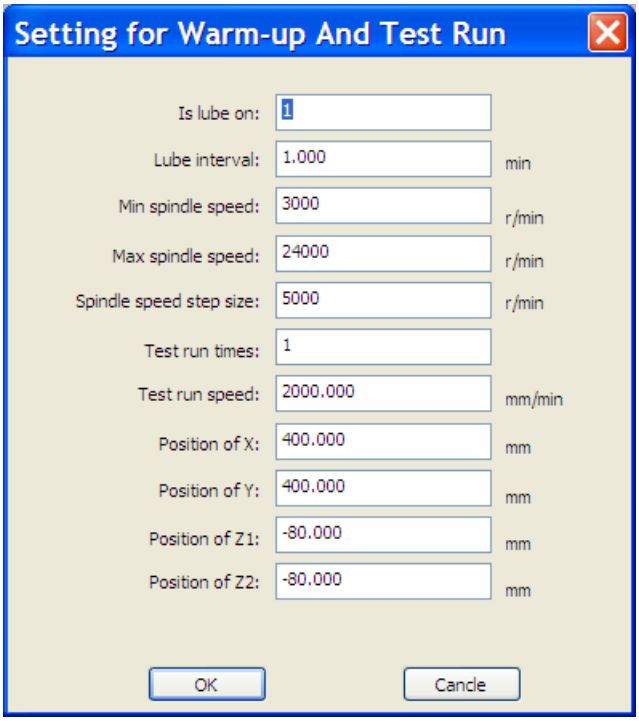


Fig. 4-16 Warm-up and Test Run

As shown in Fig. 4-16, parameters are as below:

Is lube on: 1 for “Yes”, and lube will be turned on before machine warming up and test running; while 0 for “No”, and lube will not be turned on before machine warming up and test running.

Lube interval: it refers to the time interval between two lubricating when parameter “Is lube on” is set to “Yes”. In addition, the duration time of each lubricating is decided by the value of parameter “N41002 LUB_DURATION”.

Min/Max spindle speed: it refers to the minimum/maximum spindle speed used in machine warming up and test running.

Spindle speed step size: when machine is in warming up and test running, the spindle speed will increase by the value each time reciprocating motion (or back and forward motion) is completed. And when the spindle speed reaches the maximum speed, the maximum spindle speed will be used later.

Test run times: the times of reciprocating motion in warming up and test running. A back and forth motion is called once.

Test run speed: it refers to the feedrate used in test running.

Positions of X/Y/Z1/Z2: they refer to the coordinates of the target positions in X/Y/Z1/Z2 axes respectively when machine warming up and test running ends.

4.4. Machine Menu

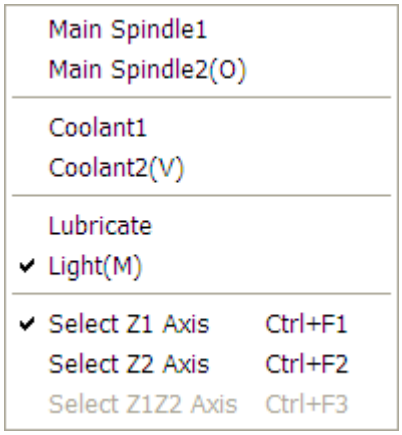


Fig. 4-17 Machine menu

Main Spindle 1/2: used to turn on/off the spindle 1/2.

Coolant 1/2: used to turn on /off the coolant 1/2.

Lubricate: used to turn on/off the lube.

Light: used to turn/off the work light in machine.

Select Z1/Z2/Z1Z2 Axis: used to activate the spindles.

4.5. Window Menu

The “Window” menu is as shown in Fig. 4-18, where the user can switchover among three interfaces under *Machine Control Bar*, switchover among three functional windows under *Coor* window, arrange the customized interfaces as well as select system language.

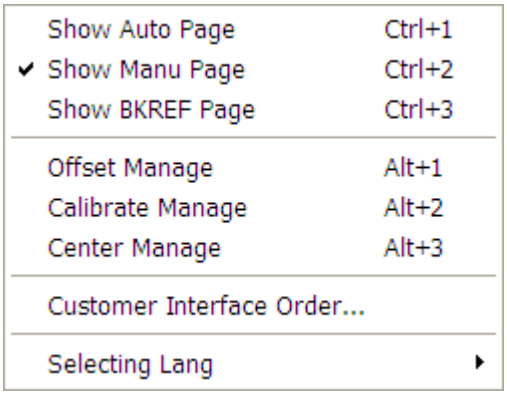


Fig. 4-18 Window menu

In the functional windows area of the holistic interface, only the former six tabs can be displayed while the rest remain hidden on most occasions. The user can customize the interfaces displaying order via the menu item “Customer Interface Order”, as shown in Fig. 4-19.

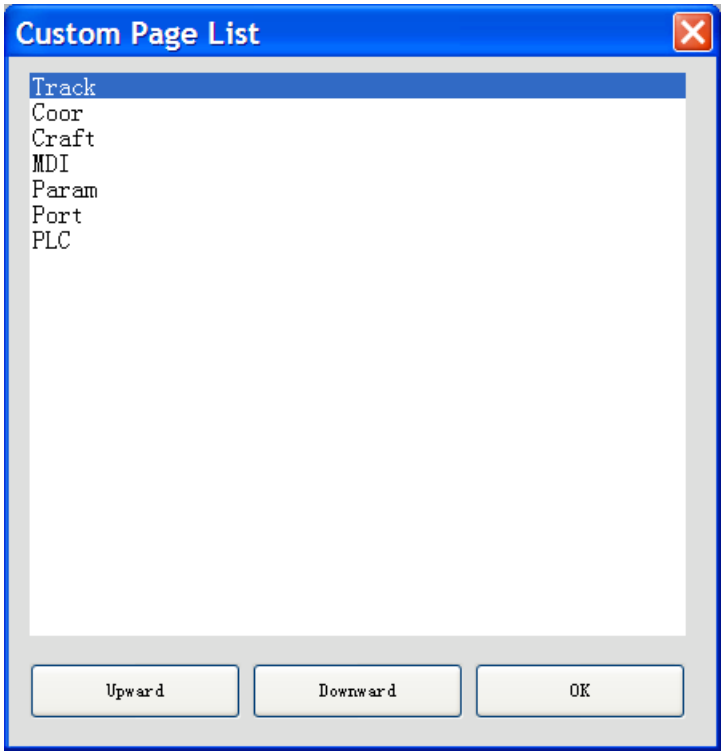


Fig. 4-19 Arrange windows presence

Language Selection

This item is used for choosing software in Chinese or in English.

4.6. Help Menu

Here are the items of “Help” menu:

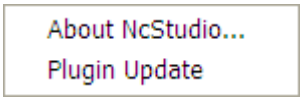


Fig. 4-20 Help menu

About NcStudio

With the item selected, a dialog box will pop up, providing information about NcStudio version, communication card model number and system registration, etc.

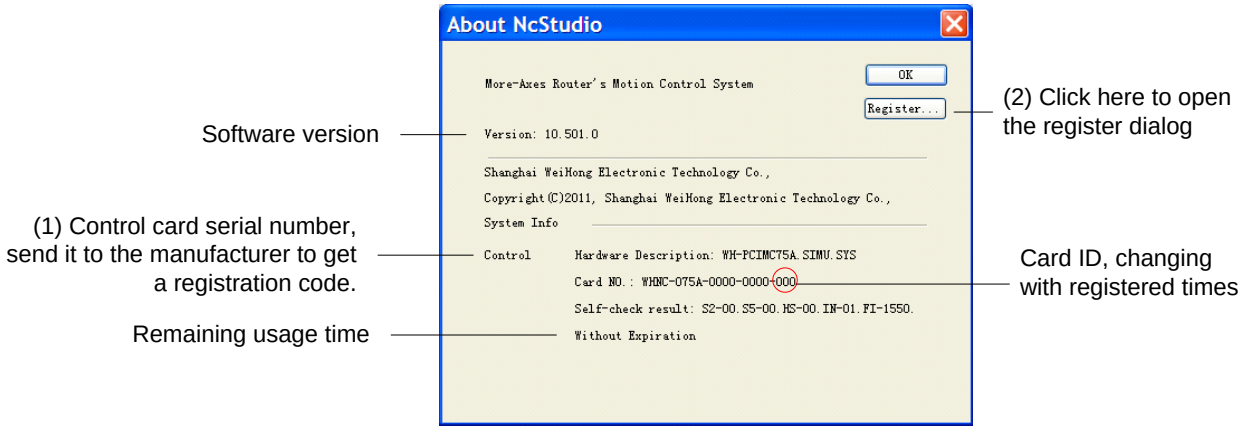


Fig. 4-21 About NcStudio

In the above dialog, the user can view such info as version, communication card, manufacturer, etc.

Register function is used to restrict the system usage time. At the expiration of usage time, the user can write down the Control Card No. (the adapter No.) and send it to the manufacturer to get a registration code. After getting the code, the user needs to click on the button [Register…] to open registration code input box, as shown in Fig. 4-22, input the received registration code, and click “OK” to finish registration.

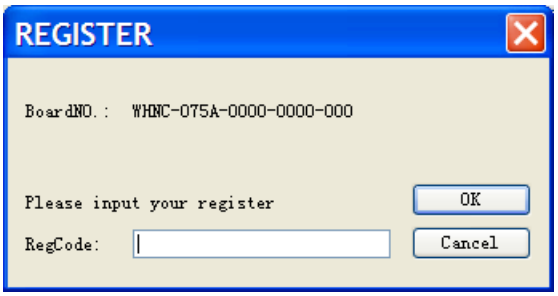


Fig. 4-22 Input registry code

5. Operation Steps

After the communication card and software are installed properly according to chapter 2.2, the system is ready for use. You can follow the flowchart below to conduct debugging and commissioning.

THE FLOWCHART OF OPERATION

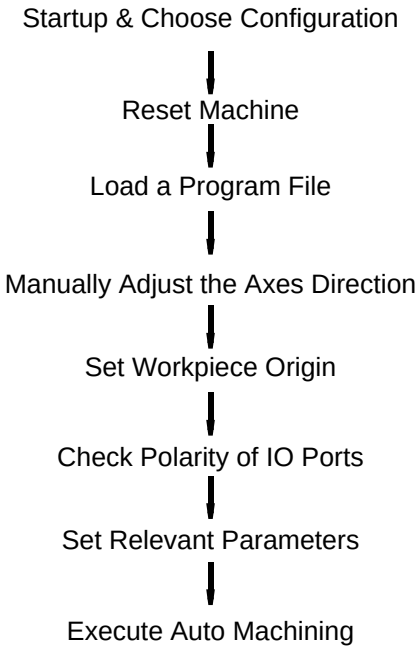


Fig. 5-1 Basic operation flowchart

5.1. Start-up

Before start-up, please make sure that the machine and computer are normally and properly connected. Turn on the power of the machine and computer to start the system and further start the NcStudio CNC system.

5.2. Machine Reset

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

As NcStudio will save the current coordinates if it exits normally, so under certain circumstances, such as restarting the system and resuming the last operation after a normal system shutdown, the user does not have to execute machine reset operation, namely, returning axes to the reference point..

5.3. Load a Machining Program File

Generally speaking, a machining file has to be loaded before machining begins. Otherwise, certain functions related with 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 automatically. At this time, the user can press “CTRL+1” to switch to *Auto* window and view the program lines in the machining file.

5.4. Manual Operation

In *Manual* window, the number keys on the keyboard can be used to operate the machine tool manually to check whether the direction of each axis is right.

The corresponding keys are:

6------Positive direction of X axis

4------Negative direction of X axis

8------Positive direction of Y axis

2------Negative direction of Y axis

9------Positive direction of Z2 axis

3------Negative direction of Z2 axis

7------Positive direction of Z1 axis

1------Negative direction of Z1 axis

Pressing any one of the above keys and the number key 5 simultaneously will make the machine tool move at rapid jog speed.

5.5. Set Workpiece Origin

The workpiece origin is defined as the coordinate origin of X, Y Z1 and Z2 in the machining file. Before machining starts, the workpiece origin should be fixed firstly. The steps are:

Manually move X axis and Y axis to the intended workpiece origin position, and then clear the coordinates of current position by clicking the Clear buttons (X, Y) in the *Coor* window. The system will conduct machining with the current position as the workpiece origin.

Through the above steps the workpiece origin of X axis and Y axis is fixed. To set the workpiece origin of Z axis, more precise operation is needed. Combined with machine tool hardware, the system supports Z1/Z2 axes measurement (calibration) function. Please refer to Chapter 3.6.1 for details.

After all these operations, the workpiece origin is set.

5.6. Check the Polarity of I/O Ports in IO State Window

The IO State window displays all ports on the terminal board which serve as indicators of communication status between hardware and software. The ports are very helpful for system monitoring and troubleshooting. For details, please refer to Chapter 3.10.

To invert the polarity of a port, firstly select the target port, click button [Invert Polarity], a dialog box as below will pop up. The modification of port polarity takes effect after the software being re-started.

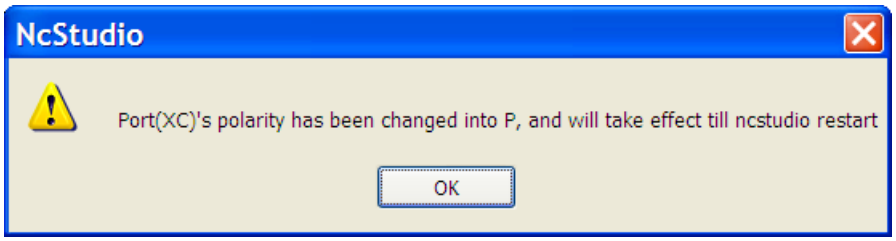


Fig. 5-2 Invert port polarity

5.7. Set Basic Parameters

5.7.1. Feedrate Setting

Feedrate refers to the axial moving speed in machining. When the machining file contains F codes, the user needs to set in *Craft* window to decide whether to use the system default feedrate or not.

5.7.2. Spindle Speed Setting

Spindle speed refers to rotational speed of the spindle in machining. When the machining file contains S codes, the user needs to set in *Craft* window to decide whether to use the system default spindle speed or not.

CAUTION

For more information about feedrate setting or spindle speed setting, please refer to Chapter 3.4.2/3.4.3/3.7.

5.8. Execute Auto Machining

Auto machining means that the machine tool processes the loaded machining file automatically.

Start Auto Machining

Select the menu item [Operate| Start] or click [Start] button at the lower part of the machine control bar or press the shortcut key F9 to activate the function. The machine tool will start machining automatically from the first line of the machining file.

Stop

During auto machining, the user can stop machining through the following three methods: selecting the menu item [Operate| Stop]; clicking [Stop] button at the lower part of the machine control bar, pressing shortcut key F11. With the function activated, the machine tool will stop machining

immediately and the system will enter into “IDLE” state. As the three methods bring the system to a stop with accuracy and in order, they are the recommended ways to stop machining.

When the adaptive connection of high-smooth speed is adopted, the system will stop when the connection speed becomes 0.

Pause

During auto processing, the user can suspend machining through the following three methods: selecting the menu item [Operate| Pause]; clicking [Pause] button at the lower part of the machine control bar; clicking shortcut key F10. To continue machining, choose [Operate| Start] or click [Start] button or press F9.

5.9. Precautions for Multi-tasking

As PC adopts time sharing operation system, generally speaking, while executing auto machining, some other operations or applications can be done or run on the PC, such as editing machining file, but there are two points to pay attention to:

1) It is recommended that the user should consider the computer memory and not open too many windows at the same time.
2) For some application procedures, such as games, VCD player, etc, they might not run stably and smoothly. They are likely to take excessive system resources like memory and CPU during running and at last cause computer crash. Therefore, during processing, in order to avoid processing interruption caused by system crash, it is recommended not to run these applications.

5.10. Precautions for Homing

Homing (backing to reference point/ machine origin or returning to the reference point) may vary with the requirements of different systems. For systems with a high requirement for precision, the process of machine reset will take a long time. Therefore, NC state window should be paid carefully attention to during this process. Do not exit from the “Back To Mechanical Origin” dialog until the system enters into “IDLE” state. Otherwise, the “back to mechanical origin” process will be artificially terminated instead of being normally finished.

If the “back to mechanical origin process” is terminated manually, the consequences will be:

1) Inexact positioning may occur: the calibration function of “back to mechanical origin” is damaged artificially and as a result, the machine coordinates become inaccurate;

2) Software limit function becomes ineffective: as the “back to mechanical origin” process has not been finished, the system will regard the software limit function as ineffective until “back to mechanical origin” process is finished.

6. Parameter Setting

Equipped with abundant machining parameters, NcStudio is competent for various machining tasks. This chapter will introduce operator's parameters only. For manufacture's parameters, refer to Manufacturer's Manual for details.

Parameters in NcStudio can also be divided into the following categories: operation parameters, axes parameters, spindle/handwheel parameters, I/O address parameters, compensation parameters, reference point parameters, cutter parameters, other parameters and parameters overview.

6.1. Parameter Modification Permission

If there is a need to change the password, click [Modify Password] button to open "Modify Password" dialog box, as shown below. After entering the old password and new password correctly, click [OK] to validate the new password.

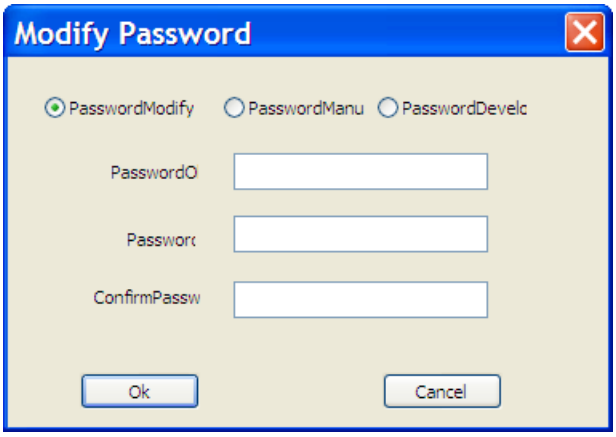


Fig. 6-1 Change password dialog

6.2. Parameter Modification Method

To modify a parameter, double-click on the parameter line and enter the data into the pop-up dialog box.

For "True/False" type of parameters, "1" means "True" while "0" means "False". The user can also directly input "True" or "False" or input the number "1" or "0". Please note that parameters cannot be modified in machining.

6.3. Parameter Inquiry

In "Parameter" dialog box, the user can click button [Direct Find] to directly locate the desired parameter to save time. As shown in Fig. 6-2, the user can inquire a parameter through its number or name in current parameter list or in the parameter overview list.

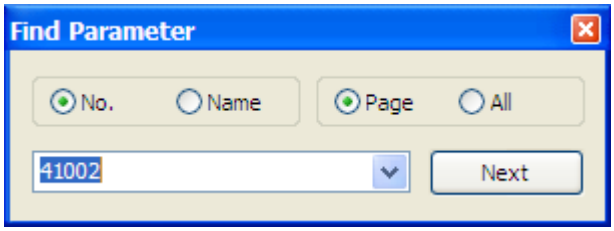


Fig. 6-2 Find parameter

6.4. Operator's Parameters

Here is the list of parameters of operator's access.

No.	Name	Setting Range	Default	Effective
2.0. Spindle general				
N20008	SPINDLE_COL_DELAY_TIME	0～60（mm/min）	0	Immediately
	The delayed time to close the spindle coolant port in order to fully cool the spindle after it stops.			
3.2. Tool setting				
N32020	NAME	-	-	Immediately
	Name of cutter. Maximal length is 127letters.			
N32021	DIAMETER	-	0	Immediately
	Diameter of cutter.			
N32022	LENGTH	-	0	Immediately
	Length of cutter.			
N32023	DIAMETER WEAR	-	0	Immediately
	The amount that is lost by wear in cutter diameter.			
N32024	LENGTH WEAR	-	0	Immediately
	The amount that is lost by wear in cutter length.			
N32030	OFFSET X	-	0	Immediately
N32031	OFFSET Y	-	0	Immediately
N32032	OFFSET Z1	-	0	Immediately
N32033	OFFSET Z2	-	0	Immediately
6.2. G code options				
N62020	IJK_INC	No: Invalid Yes: Valid	Yes	Immediately
	When it set to "No:, the circle center coordinates is relative to the workpiece origin. When it set to "Yes", the circle center coordinates is relative to the starting point of arc programmed.			
N62022	Tool Selection for G Code	No: Invalid Yes: Valid	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 many tools.			

No.	Name	Setting Range	Default	Effective
N62730	G73_G83_SAFE_HEIGHT	-9999～99999（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
	The above orientation stop direction is only valid in G19 (X-Y) plane.			
6.4. Speed/Acc				
N64020	RAPID_TRAVEL_FEEDRATE	0~100000 mm/min	3000	Immediately
	The default speed of beeline axis when positioning.			
N64021	DEFAULT_FEEDRATE	0~100000mm/min	1500	Immediately
	The default speed of beeline axis when machining.			
6.7.				
N67000	MoveToOrigPositionAfterChageTools	0:No 1:Yes	Yes	Immediately
	After tool changed, move back to where the tool stood before tool change. Only used in TURN-Configuration.			
N67001	StartSpindleAfterChangeTools	0:No 1:Yes	No	Immediately
	Whether to start spindle after tool changed. Only used in TURN-Configuration, the spindle can only be started under machining state.			
7.1. Manu				
N71000	JOG_VOL	0～100000(mm/min)	1800	Immediately
	The default velocity under jog mode with the feedrate override being 100%.			
N71001	RAPID_JOG_VOL	0～100000(mm/min)	2400	Immediately
	The velocity under rapid jog mode with the feedrate override being 100%.			
N71002	MaxJogFeedrateBeforeBKREF	0～100000（mm/min）	1200	Immediately
	The maximum speed under manual mode before returning to the reference point.			
N71021	MachTaskEndInformType	0;1;2	0	Immediately
	How to inform operators after the end of machining task. 0: Red lamp is out; 1: Red lamp lights for 3 seconds.			
7.2. Auto				
N72001	SPEED_ASSIGN_TYPE	0: Use file speed; 1: Use default speed; 2: Specify speed proportionally.	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	No: Not ignore Yes:Ignore	No	Immediately
	If set to “Yes”, the system will ignore the rotational speed specified by the file and use the			

No.	Name	Setting Range	Default	Effective
	default speed specified by SPIND_VELO_DEFAULT.			
N72004	STOP_SPIND_AT_END	No: Not stop Yes: Stop	Yes	Immediately
	Whether to stop the spindle after a machining task. Note: if parameter "AutoStopSpindleWhenPause" is set to "Yes", the spindle will be stopped after a task stops regardless of the parameter value.			
N72006	HEIGHT_Z_RAISE	-100000~100000 (mm)	1	Immediately
	The lift height of Z axis during dry run.			
N72040	CoolantAutoStartWhenStartMachining	No: Not turn on Yes: Turn on	No	Immediately
	Coolant auto start when machining starts.			
N72041	CoolantAutoStopWhenFinishMachining	No: Not turn off Yes: Turn off	No	Immediately
	Coolant auto stop when machining finishes.			
7.9. Operation others				
N79110 ~ N79113	FIXPT_POS X/Y/Z1/Z2	-9999~99999mm	0	Immediately
	The position of the fixed point.			
N79200	DisplaySimuOutRange	No: Not display Yes: Display	Yes	Immediately
	Whether to prompt the user if simulation is out of travel range.			
8.1. Position view				
N81000	AUTO_LOAD_TRACK	0: No 1: Yes	Yes	Immediately
	Whether to parse the file automatically after loading the file. .			
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				
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.			

7. Shortcut Keys List

Shortcut Key	Function	Shortcut Key	Function
Global shortcut keys			
Ctrl+1	Show *Auto* window	Ctrl+2	Show *Manual* window
Ctrl+3	Show *Calibration* window	Ctrl+F9	Enable *Advanced Start* Function
Pause	Reset	Ctrl+A	Select all
Ctrl+C	Copy	Ctrl+O	Open and load
Ctrl+S	Save	Ctrl+U	Unload
Ctrl+V	Paste	Ctrl+X	Cut
Ctrl+Z	Undo	Alt+1	Open *Offset* window
Alt+2	Open *Calibration* window	Alt+3	Open *Centering* window
Alt+F4	Exit	Alt+F	Open *File* menu
Alt+O	Open *Operate* menu	Alt+P	Open *Parameter* menu
Alt+M	Open *Machine* menu	Alt+W	Open *Window* menu
Alt+L	Open *Log* menu	Alt+H	Open *Help* menu
F7	Return to the reference point (Homing)	F8	Return to the fixed point
F9	Start	F10	Pause
F11	Stop	F12	Resume machining from the interrupted point
Shift+F6	Set as the WCS Zero		

Shortcut Key	Function	Shortcut Key	Function
Shortcut keys for *Track* window			
Home	Center	End	Show current machining point
+ (mini-keyboard)	Zoom in	- (mini-keyboard)	Zoom out
/ (mini-keyboard)	Fit to window size	S	Front view
F	Top view	X	Bottom view
A	Left view	D	Right view
B	Rear view	H	Northwest view
C	Southeast view	G	Northeast view
Z	Southwest view	Alt+↑ or Alt+↓	Rotate around X axis
Alt+PgUp/ Alt+PgDn	Rotate around Y axis	Alt+→ or Alt+←	Rotate around Z axis
Delete	Clear track		
Shortcut keys for *Maunal* window			
5 (mini-keyboard)	Activate *rapid jog speed*	4 (mini-keyboard)	X- (In jog and increment mode)
6 (mini-keyboard)	X+ (In jog and increment mode)	2 (mini-keyboard)	Y- (In jog and increment mode)
8 (mini-keyboard)	Y+ (In jog and increment mode)	1 (mini-keyboard)	Z1- (In jog and increment mode)
7 (mini-keyboard)	Z1+ (In jog and increment mode)	3 (mini-keyboard)	Z2- (In jog and increment mode)
9 (mini-keyboard)	Z2+ (In jog and increment mode)	F1	Clear X axis coordinate to zero

Shortcut Key	Function	Shortcut Key	Function
F2	Clear Y axis coordinate to zero	F3	Clear Z1 axis coordinate to zero
F4	Clear Z2 axis coordinate to zero	F5	Clear XY axes coordinates to zero
F6	Clear coordinates of all axes to zero		
Shortcut keys for *Auto* window			
F1	Load	F2	Unload
F3	Edit	F4	Advanced start
F5	Simulate	F6	Handwheel guide
Shortcut keys for *Reference* window			
F1	Return X axis to the REF point	F2	Return Y axis to the REF point
F3	Return Z1 axis to the REF point	F4	Return Z2 axis to the REF point
F5	Return all axes to the REF point	F6	Handwheel guide

8. 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.