NcStudio V8 Milling CNC System

Users' Manual

4th Edition

(For three axes / dual Z axes)

Weihong Electronic Technology Co., Ltd.

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

Preface

About This Manual

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

With 13 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, including Chapter 1~2. The two chapters give general description of NcStudio V8 milling CNC system from the perspective of hardware, software as well as installation and wiring, etc.
3) Part 3: Chapter 4~8, introduction to software operation, taking the general three axes software as an example on most occasions. The former two chapters illustrate detailed operations of single functionality and its corresponding interfaces, which will be an intuitional guidance to users and operators in real practice. Chapter 6 lists all the parameters of operators' permission and their modification method. The latter two chapters outline operating and debugging steps as well as precautions.
4) Part 4: including chapter 9 and 10. Chapter 9 introduces the tool magazine configuration while chapter 10 describes the operation of the software with multi-cylinder.
5) Part 5: separate and special introduction to double Z axes software in Chapter 11. It highlights the differences between general three axes software and double Z axes software.
6) Part 6: appendix part, consisting of chapter 3, chapter 12 and chapter 13. You can get the basic concepts of NcStudio, shortcut keys list and the software license agreement accordingly.

Applicable Product Models

This manual is applicable to NcStudio V8 milling CNC system. Refer to the table below for details:

Product Model	Remarks
NcStudio V8 Milling CNC System	Herein referred to V8 as abbreviation, including general three axes series and double Z axes series CNC system for milling machines.

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	R4	Contact information updated.
2015.10	R3	1) Modify the chapter 5.5. 2) Add the chapter 9. 3) Other revisions.
2015.08	R2	1) Delete descriptions about “Auto hardware update” in Chapter 2.2.2. 2) Other revisions.
2015.04	R1	1) Released for the first time as combination of *CNC System NcStudio-V8 for three axes engraving machines users’ manual* and *CNC System NcStudio-V8 for three axes (double Z axes) engraving machines users’ manual*. 2) Update software installation steps as the latest version of software. 3) Add “Configuration selection” function before software installation. 4) Other revisions.

Precautions

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

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

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

WARNING

1) Precautions Related to Storage and Transportation

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

2) Precautions Related to Installation

- Only when this equipment installed in the qualified electricity cabinet can it be used. The construction of the cabinet must reach IP54 grade of protection;
- Paste sealing strips on the joint of the cabinet to seal all the cracks;
- Cable entry should be sealed while easy-to-open on the spot;
- A fan or heat exchanger should be adopted for the heat dissipation and air convection of the cabinet;
- If a fan is adopted, air strainer is a must in air inlet or air outlet;
- Dust or cutting fluids may have access to the CNC device via the tiny cracks and tuyere. Therefore it is necessary to pay attention to the surroundings and air flow direction of the air vent to make sure that the outflow gas is towards pollution source;

WARNING

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

3) Precautions Related to Wiring

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

4) Precautions Related to Running & Debugging

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

WARNING

- 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 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 interface which is intuitive and user-friendly.

Apart from functions of automatic mode, manual mode and backing to the reference point, this CNC system also features the following functions: simulation, real-time motion track displaying, automatic calibration of Z axis, breakpoint resume (advanced start), rotary axis machining, and so on.

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) General three axes series is equipped with three motion axes, namely X/Y/Z axis, while double Z axes series is equipped with four motion axes, namely X/Y/Z1/Z2 axis. The binary input & output points and analog outputs can be further extended. The above configuration varies with different choices of machine tool builders.
2) It supports CNC rotary table (varying with different versions).
3) 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.
4) 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.
5) It supports array processing: the user can repeatedly carry out a program machining according to columns and arrays pre-defined.
6) It supports rotating & mirroring processing. This function can mirror and rotate a program with the workpiece origin as the center.

7) It supports stepping function or incremental feeding: the user can set the precise feeding distance and adjust the step length value.
8) It supports user data input function: the user can input G code online and execute it right away.
9) It supports advanced processing instructions. Inputting a few parameters will suffice to perform bottom milling and frame milling.
10) 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.
11) 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.
12) High precision during motion axes returning to the machine origin (also known as the reference point).
13) It supports automatic tool measurement function, including fixed calibration, mobile calibration and auto centering, etc.
14) 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.
15) 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%.
16) 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.
17) 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.
18) 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.

19) It supports powerful and agile keyboard operation. The new system strongly supports keyboard operation and thus can fulfill the user's needs in operation.
20) It supports log function. The powerful log function can help the user check the detailed processing information and system diagnosis.
21) 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.
22) Built-in file editor: With which, the user can load a machining file into the editor for editing and modifying.
23) 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.
24) It supports auto parameters backup function. The user can back up the parameter settings and recover them when necessary.

2. System Installation and Connection

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. Installation of NcStudio

If there is already an old version of NcStudio on the computer, please delete it first before installing a new version. Please turn to Chapter 2.2.4 and follow the instructions to delete the old version. You can install a new version of NcStudio directly instead, however, please note that such operation will overwrite all data of the old version.

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

Please install the motion control card before installing the software, to save the trouble of separate installation of driver for the motion control card. Following description excludes installation and wiring of related electrical devices, which you can refer to specified manufacturers' manual.

2.2.1. Software Setup

Please install the software as following steps:

1) Power up and start the computer (hereinafter referred to as PC), the system will run the operating system of PC automatically. If your PC has not installed any OS, please install one first. After PC is started, please shut down unrelated running applications.

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.

3) The first dialog box during installation is as below, see Fig. 2-1:

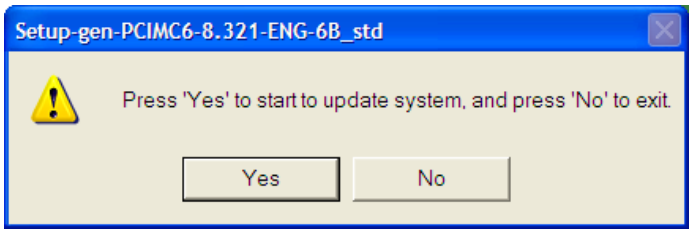


Fig. 2-1 Update prompt dialog

4) Click [Yes] to continue. 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 old version software, see Fig. 2-2.

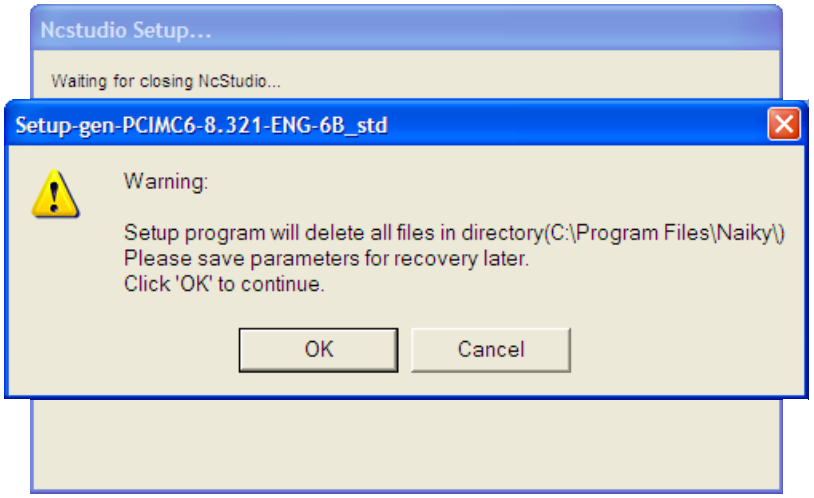


Fig. 2-2 Warning to delete previous files

5) Click [OK] for confirmation. If 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 6)” to move on.

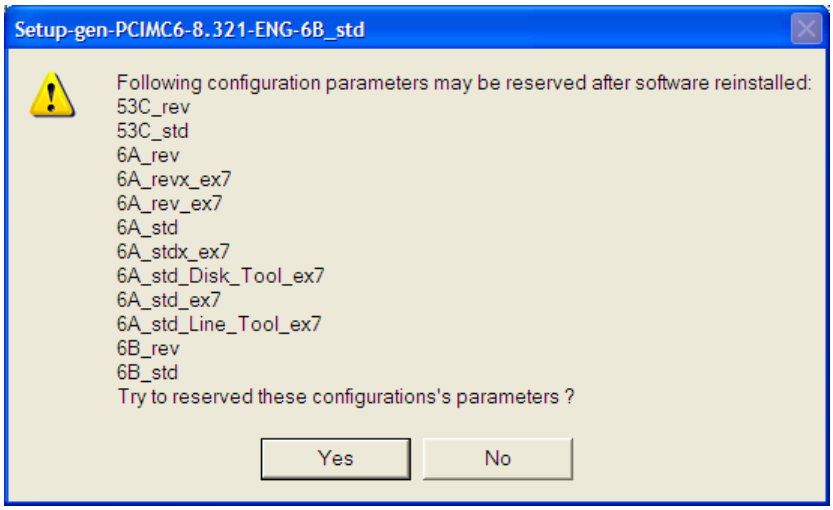


Fig. 2-3 Prompt to reserve parameter settings

6) 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 motion control card installation ([Yes] is recommended), see Fig. 2-6.

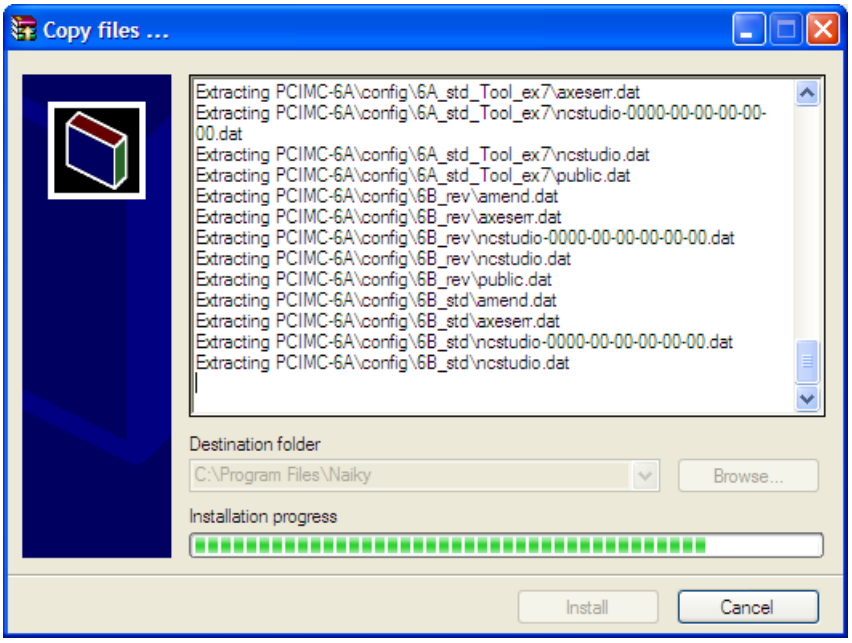


Fig. 2-4 Installing

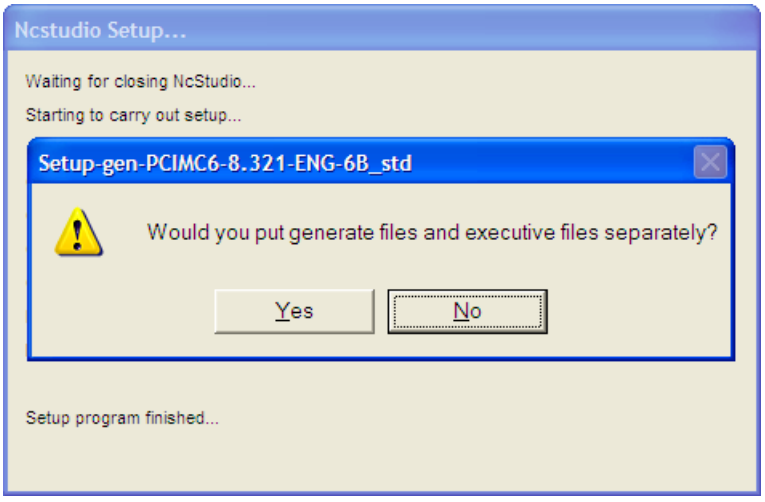


Fig. 2-5 Prompt for files location ([No] is recommended)

7) After the computer is restarted, software installation is completed.

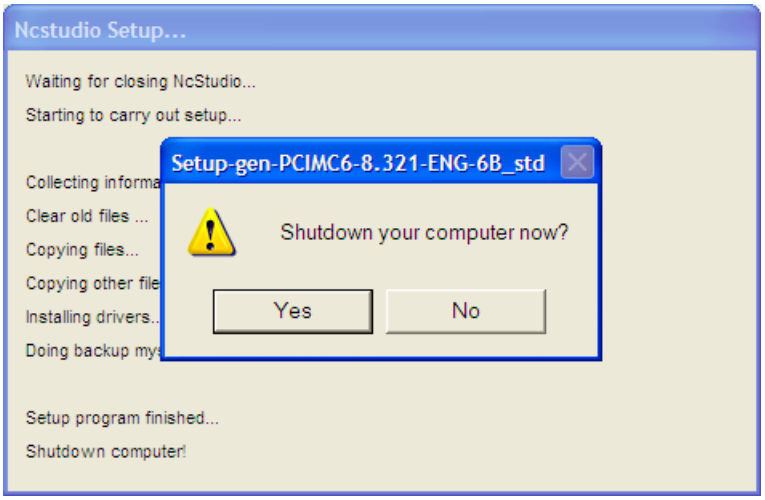


Fig. 2-6 Prompt to shut down the computer ([Yes] is recommended)

CAUTION

The installation package is separate for different languages; please verify the language version first before you install it.

2.2.2. Installation of NcStudio Motion Control Card and Other Periphery Equipment

Please install the NcStudio motion control 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. When installing the motion control card, 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 steps, 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. Update the Hardware Driver Manually

After you install the motion control card and the software, you need 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 CNC Adaptor, right click “Weihong CNC Adaptor (PCIMC-6A)” and select “Update Driver Software…”
2) A dialog as Fig. 2-7 will pop up. Select “Install from a list of specific location (Advanced)”, click [Next] to continue.

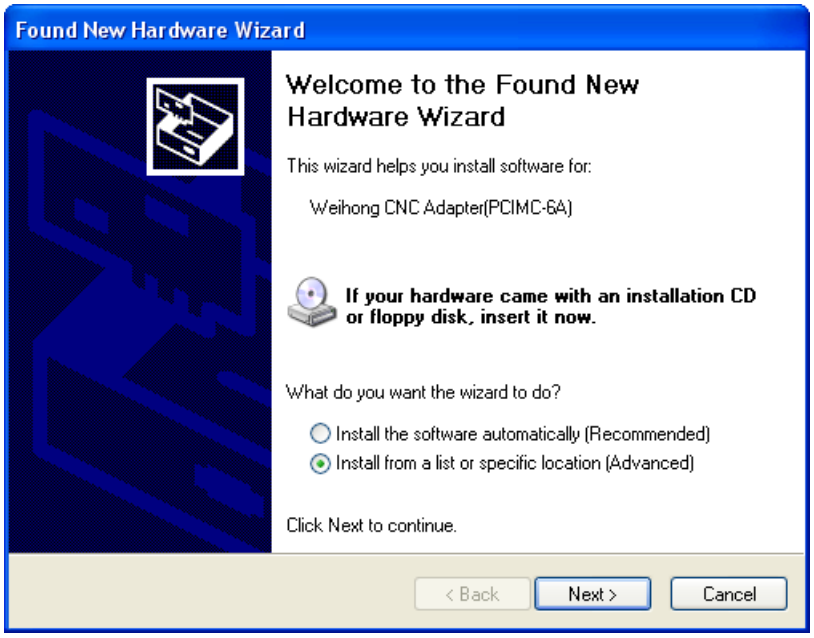


Fig. 2-7 Found new hardware wizard

3) Select “Don’t search”, I will choose the driver to install’, click [Next] to go ahead. See Fig. 2-8.

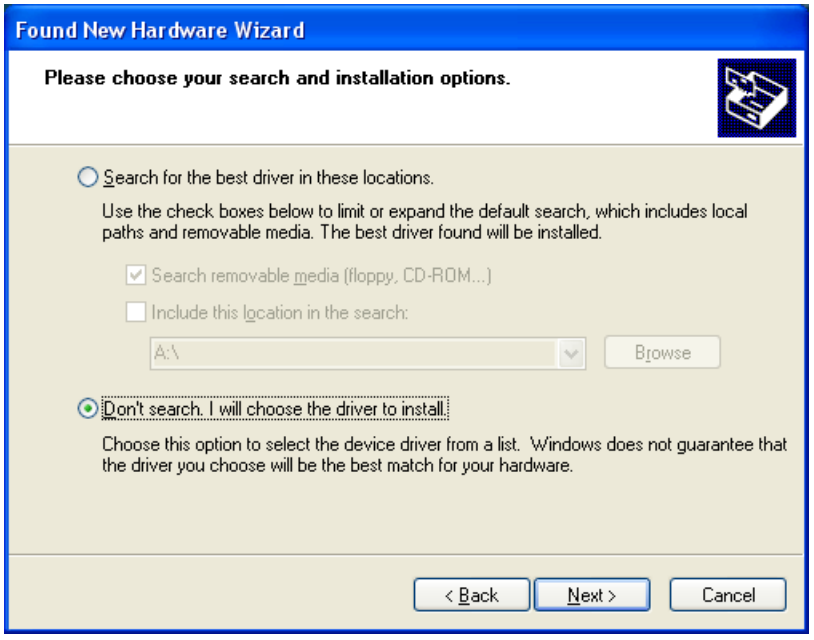


Fig. 2-8 Choose the driver manually

4) Click [Have Disk...] button in the following dialog, as shown Fig. 2-9.

5) Click [Browse] button in the pop-up dialog titled “Install From Disk”, see Fig. 2-10.

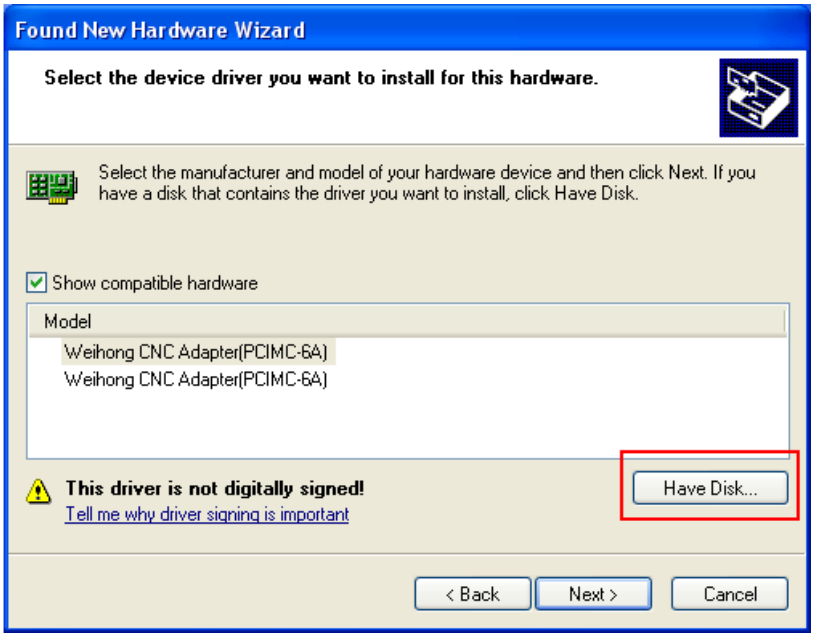


Fig. 2-9 Select the driver manually

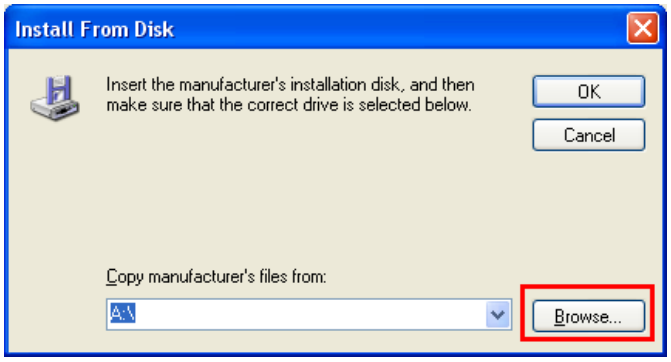


Fig. 2-10 Select the target file of driver

6) In the pop up dialog box, select the target hardware drive in the list, Choose the target file "NcadptPci(PCIMC-6A).inf" under directory "C:\Program Files\Naiky\PCIMC-6A".

7) Click [OK] to go back, and then click [Next] to start updating the driver software. When the updating is finished, a dialog as Fig. 2-11 below will pop up.

8) 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.

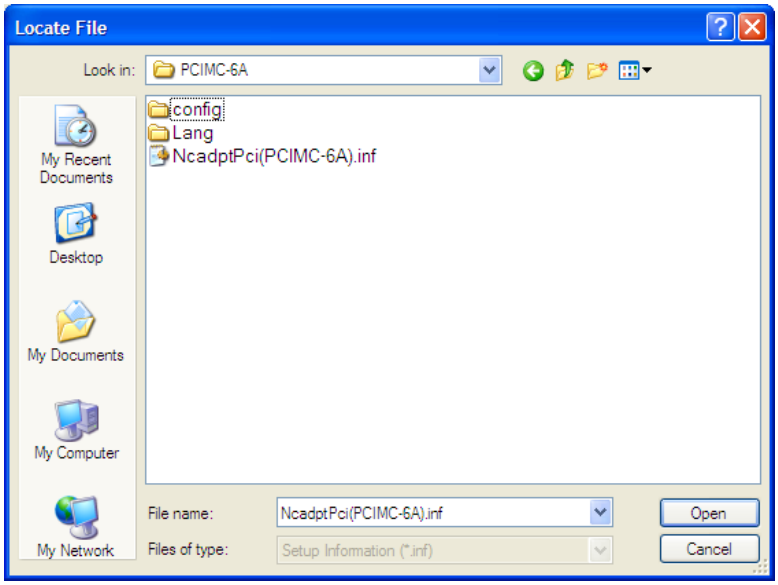


Fig. 2-11 Select the hardware drive

Fig. 2-12 Driver update completed

2.2.4.Uninstall of 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. Basic Concepts of NcStudio

As a set of versatile system, NcStudio involves various concepts, such as workpiece coordinate system, machine coordinate system, operation mode and operation status, etc. Therefore, the user needs to be well informed of these concepts before using NcStudio.

3.1. Operation Mode and State

3.1.1. Operation Mode

Under any circumstance, 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 are three ways to move an axis, to put it in other words, there are three feeding ways: jog, stepping and handwheel. Stepping is also known as incremental feed, and handwheel also known as MPG, manual pulse generator.

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  in 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.

Select “Handwheel” in the software interface to activate the handwheel mode. With enabled, the connected handwheel or the MPG can be used to control the motion of the machine tool.

3.1.2. 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] and restore the machine tool to IDLE state.

◆ **RUNNING**

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

◆ **PAUSE**

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

◆ **LOCK**

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

3.2. 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. 3-1:

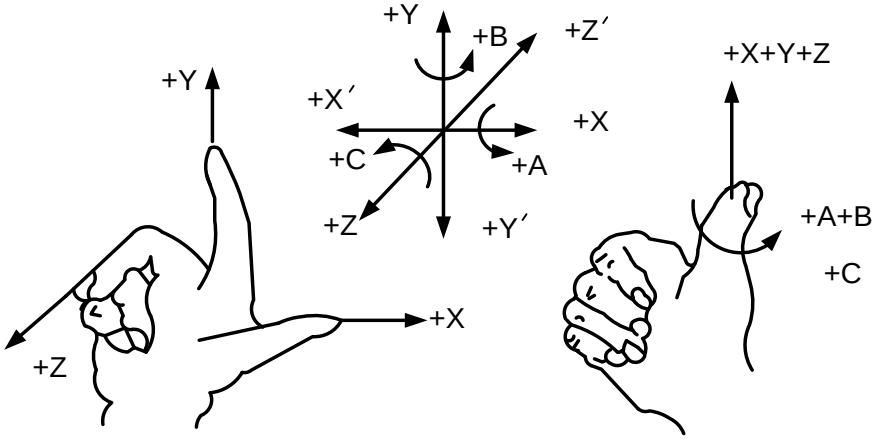


Fig. 3-1 Coordinate system following right-hand rule

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.

3.2.1.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.

3.2.2.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.

4. Operation Interface of NcStudio

The interface of NcStudio consists of the title bar, menu bar, toolbar, status bar and several functional windows. The holistic interface is shown as Fig. 4-1.

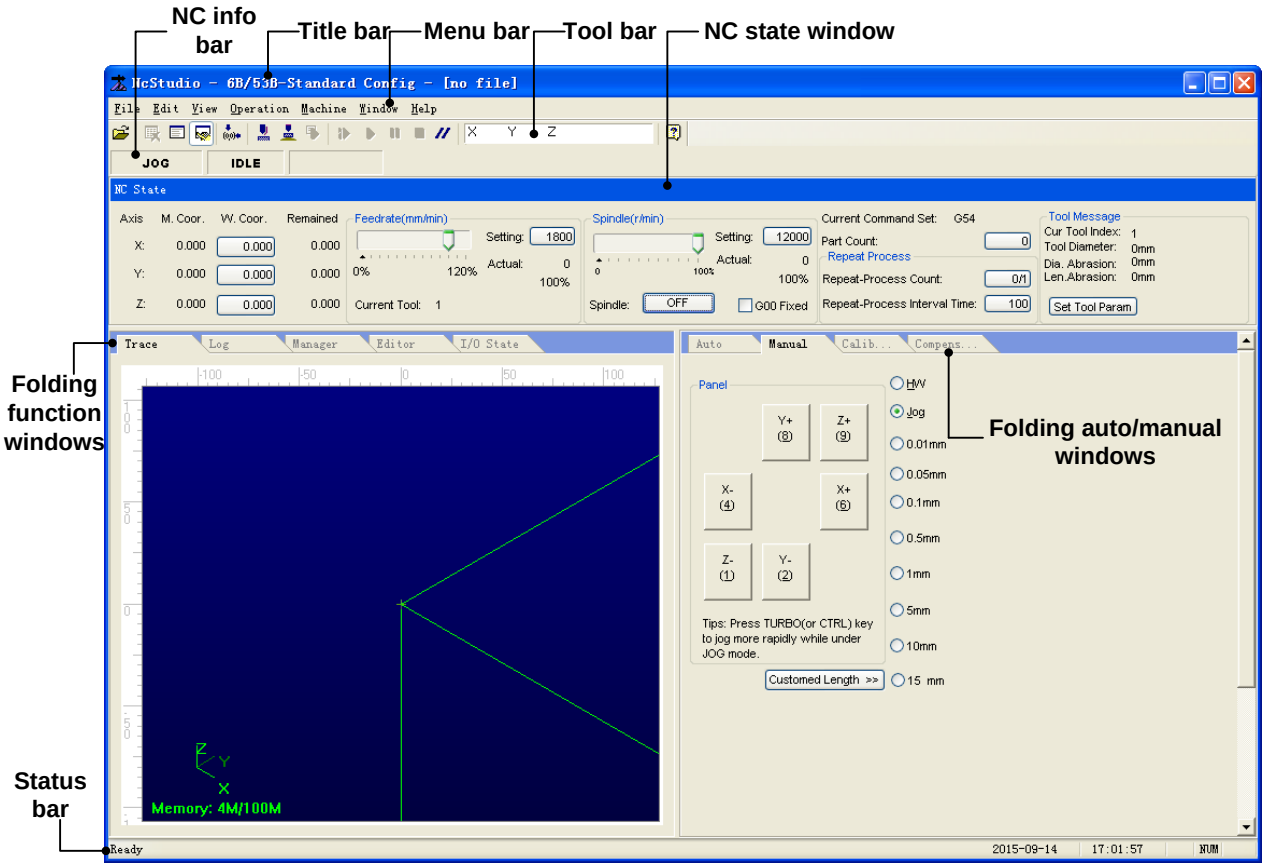


Fig. 4-1 Operation interface of NcStudio

The functional windows are divided into three sections, which can be switched by pressing *Esc* key, they are:

The first section: NC state window.

The second section: Trace window, Log window, Manager window, Editor window and I/O State window.

The third section: Auto window, Manual window, Calibration window and Compensation window.

4.1. 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.

Fig. 4-2 Title bar

CAUTION

In the PC-based system, active and inactive window are two important concepts. An active window means the window can accept keyboard input at present. At any moment, there is only one active window with all the other windows being inactive.

Color of the title bar tells its activation status. A window with blue title bar is active while one with grey title bar is inactive.

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. 4-3.

This menu can control the position and the size of the window, such as restore, move, close, maximize and minimize, etc. On the right side of the title bar, there are three control buttons. They are restoring button, maximizing button and minimizing button respectively. These buttons are used to quickly set the size of the window.

Besides, each sub-window has the corresponding title bar. An active sub-window can be distinguished from an inactive sub-window by the color of title bar. Please refer to the following chapters for details.

Fig. 4-3 System menu

4.2. Menu Bar

Below the title bar is the menu bar, which includes many normally hidden pull-down menus, as shown below. 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.

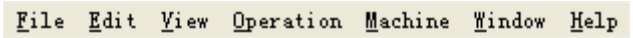


Fig. 4-4 Menu bar

◆ **Mouse operation**

Firstly, left-click the main menu on the menu bar, and then left-click one of the pull-down menu items to enable the operation.

◆ **Keyboard operation**

To choose the target menu, the user can press 'Alt' key and the hot key of the target menu simultaneously (the hot key is the underlined letter, for example, the user can select "File" by pressing the combination key "ALT+F"). Then the corresponding pull-down sub-menu will pop up.

◆ **Shortcut key operation**

In the pull-down submenu, some menu items are listed with their shortcut keys on the right side. For example, the shortcut key for [Start] in [Operation] menu is F9, which indicates that by pressing F9, instead of accessing multiple menus one by one, the user can get the menu order executed directly.

Some menu items are followed by three dots (for example, [Open and Load…] in [File] menu), which indicates that when the user chooses this item, a dialog box will pop up automatically. If some menu items on a pull-down submenu are shown as grey, this indicates that these items are unavailable under current condition.

Besides, right-click the mouse on different parts of the screen will open a context shortcut menu. From this shortcut menu, the user can execute the order which has been executed repeatedly and bears the closest relevance to the current position.

4.3. Tool Bar

Under the menu bar is the tool bar, which is composed of some operation buttons corresponding to specific menu orders or options. You can directly click one of these buttons to realize the specified function.

Fig. 4-5 Tool bar

These buttons greatly simplify and visualize operation process.

Open a program file and load it into the system. Key combination: Ctrl+O.

Clear the trace in Trace window. Key combination: Ctrl+Del.

Activate the Auto window. Key combination: Ctrl+1.

Activate the Manual window. Key combination: Ctrl+2 or ScrLk.

Return to workpiece origin. Shortcut key: F7.

Fixed calibrate, used to adjust the difference value of tools before and after tool change. Key combination: Shift+F7.

Measure workpiece surface, setting the origin in Z axis with the help of tool sensor. Key combination: Ctrl+F7.

Simulate, to start or terminate the simulated running of program. Shortcut key: F8.

Breakpoint resume, to resume machining from the interrupted point. Key combination: Shift+F9.

Start, to start or continue the program running. Shortcut key: F9.

Pause. Shortcut key: F10 or Pause key on the keyboard.

Stop normally. Shortcut key: F11.

Reset. To reset the system or restore from E-stop or erroneous status. Shortcut key: F2.

Direct positioning input box, where you can locate an axis to a desired position by inputting its coordinate value. With coordinate entered and *Enter* button pressed, the system will move the tool to the position at rapid speed.
The former means the value input is workpiece coordinate; while the latter with an asterisk mark "*" before means the value input is machine coordinate.

Get info about the NcStudio system and registration.

4.4. CNC Info Bar

Under the toolbar is CNC info bar, showing the current CNC state and some other info, including the current operation mode, operation status, running state of machine as well as some possible cautions and alarm events, as shown below. To name but a few, the following two pictures are only provides as examples.

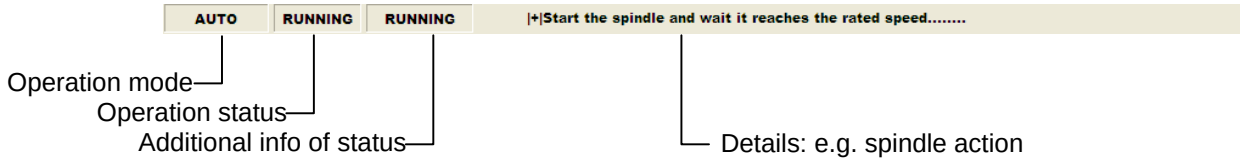


Fig. 4-6 NC information bar-1

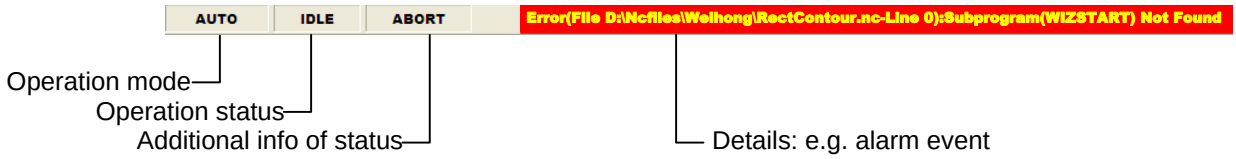


Fig. 4-7 NC information bar-2

4.5. CNC Status Bar

At the bottom of the interface is status bar, as illustrated in Fig. 4-8.

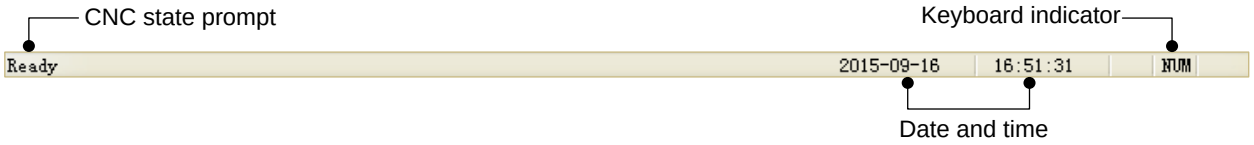


Fig. 4-8 Status bar

◆ **CNC State prompt**

It shows the prompt info of current operation or of the selected order.

◆ **Date & Time**

It shows the current date and time.

◆ **Keyboard Indicator**

It shows the current state of the keyboard, including Caps Lock, Num Lock or ScrLK.

4.6. CNC State Window

Under the CNC Info Bar is the CNC State window, which can be divided into 6 areas according to functions, as illustrated below:

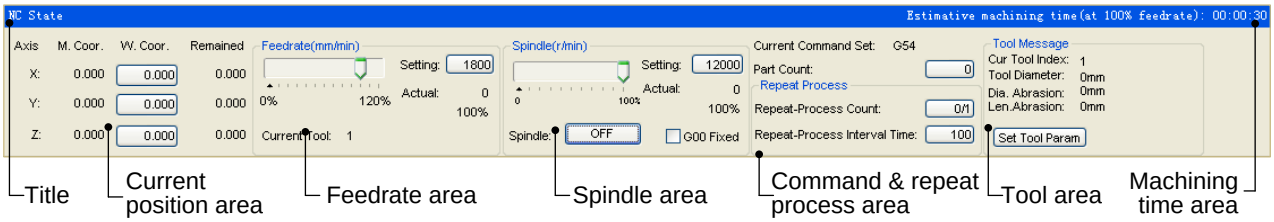


Fig. 4-9 CNC state window

4.6.1. 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. 4-10, 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.

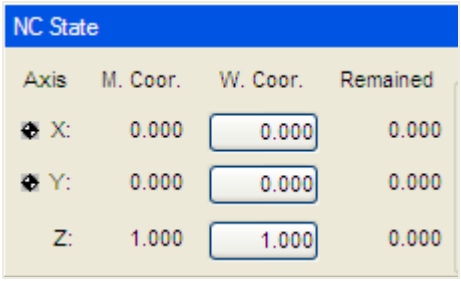


Fig. 4-10 Current position area

It is easy to set and adjust the workpiece origin. To set the current point as the workpiece origin, the user only needs to move the cursor on the workpiece coordinate of the target axis and then click its button. Taking X axis as an example, manually move the X axis to the desired origin position, and click the button of it. A dialog box will pop up, as illustrated below:

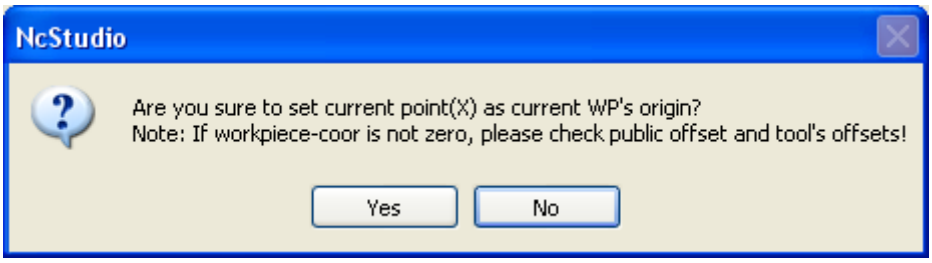


Fig. 4-11 Setting workpiece origin

Click [Yes] to finish this operation. It is the same operation for other axes.

Remaining distance, also known as distance to go, is the remaining distance the tool should move under the current command, with its sign indicating the moving direction.

CAUTION

Please check the public offset value and tool offset value if the value of “W. Coor” .is not zero after setting.

4.6.2.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.

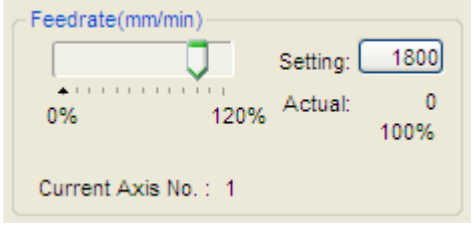
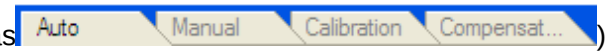


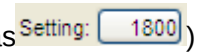
Fig. 4-12 Feedrate area

◆ **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.

◆ **Setting Value**

In auto mode (as 

), when the user clicks the figure after "Setting" (as 

), a dialog box will pop up for setting machining speed and G00 speed.

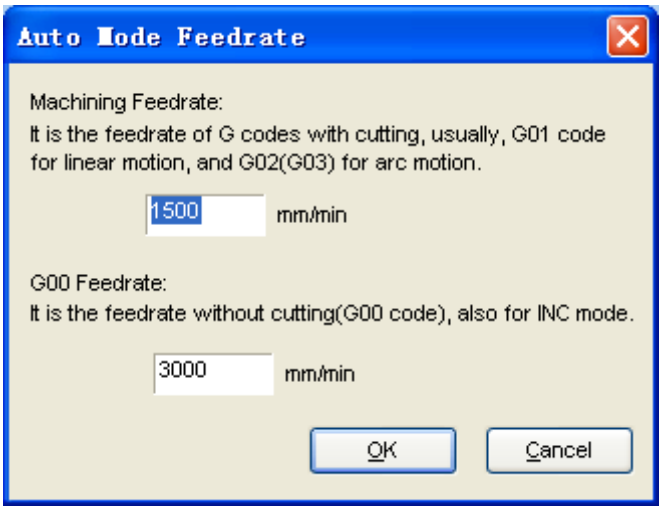
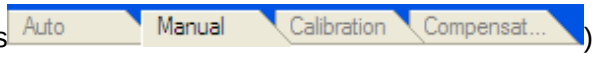


Fig. 4-13 Setting auto speed dialog box

In manual mode (as 

), 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) and rapid jog speed (also known as manual/jog high speed).

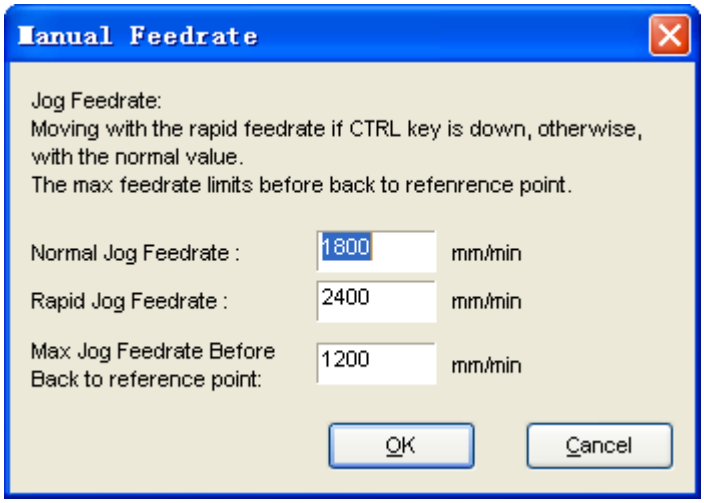
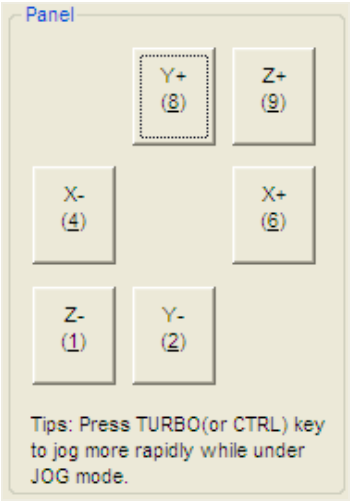


Fig. 4-14 Setting manual speed dialog box

To toggle between the jog speed and rapid jog speed:

Pressing “Ctrl+ a Num direction key” together makes the system run in rapid jog mode; while pressing a Num direction key alone makes the system run in normal jog mode.

e.g.: as shown in picture on the right side, pressing “Ctrl+6” makes the X axis moving at rapid jog speed; while pressing “6” makes it moving at normal jog speed.



Please note that the setting operation in this area shares the same effect with the setting operation in [Operation| Set Parameters…|Opera(P)].

◆ **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. The relationship between setting feedrate and actual one is as follows:

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

◆ **Current Tool**

It shows the number of the currently used tool, namely the number of spindle tool.

4.6.3.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. 4-15.

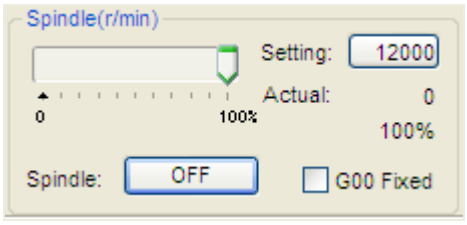


Fig. 4-15 Spindle speed area

◆ **Spindle Override Sliding Block**

By dragging the sliding block, the user can regulate the current spindle speed within a scope of 0~100%. 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 Startup Button**

This button is used to turn on/off the spindle.

◆ **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. 4-16.

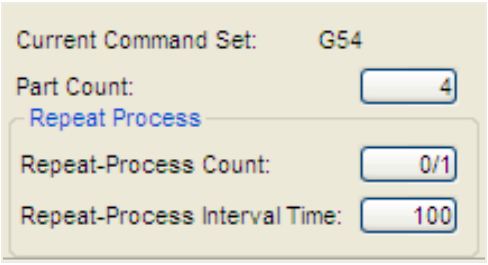
Fig. 4-16 Setting spindle speed dialog box

◆ **G00 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.

4.6.4.Current Command Set, Repeat Process

This area displays the state of currently executed command, e.g.: G54, modal/non-modal state, G01, G17, G18, G19, etc, as shown in the picture on the right side.



◆ **Part Count**

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

◆ **Repeat-Process Count**

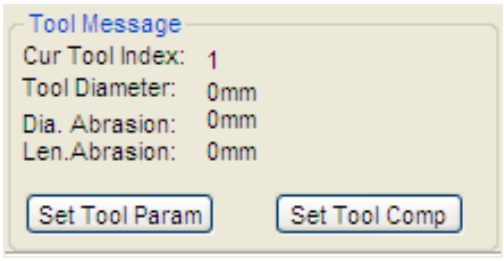
The processing times that has been cycled and the total processing times set. Clicking the figure button next to it can set the total processing times of a program file.

◆ **Repeat-Process Interval Time**

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.

4.6.5.Too Info Area

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



◆ **Set Tool Parameter**

Clicking [Set Tool Param] button will open the parameter dialog box where you can directly set the relevant parameters, as shown below. 255 tools are supported at most.

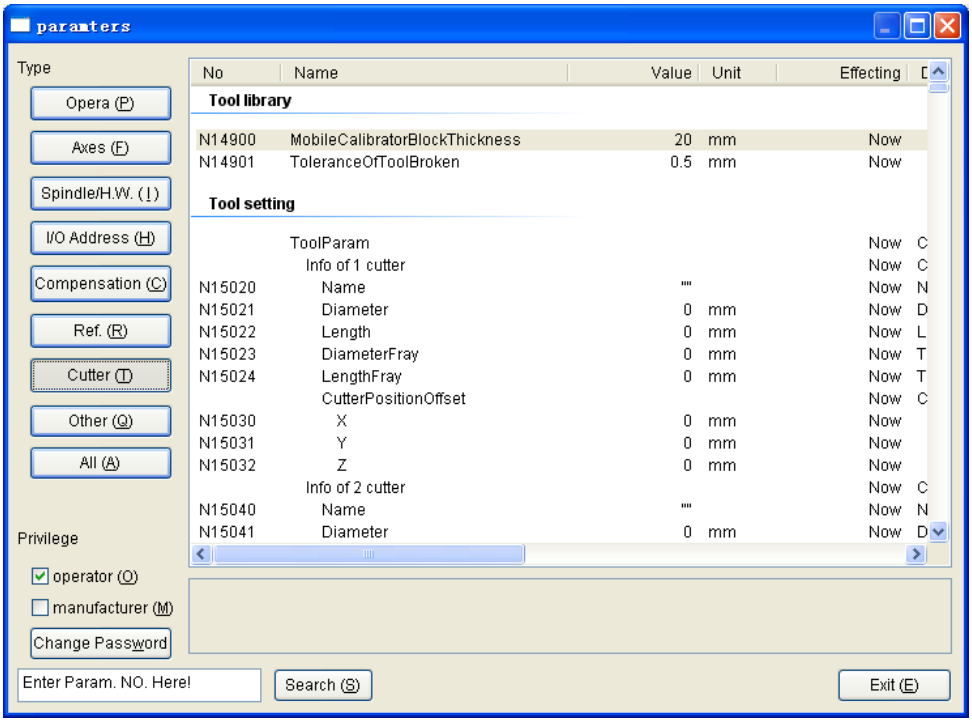


Fig. 4-17 Tool parameters

4.6.6.Time Info

On the right side of the title bar of NC State Window, you can obtain the machining time information of machine tool. In simulation, the time displayed is the estimated time at 100% feedrate override, while in actual machining, the time is the actual running time.

4.7. Auto Operation Window

The auto operation window displays program lines of the currently loaded machining 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 our company. The user can view and edit the current machining file in this window.

Right clicking on this window will open a shortcut menu, as shown in Fig. 4-18.

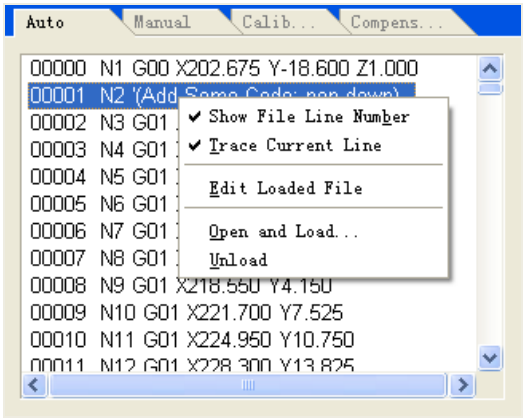


Fig. 4-18 Auto operation window

All of those menu items can also be found in standard menus, among which, “Show File Line Number” and “Trace Current Line” can be found in “View” menu, while the other 3 items can be found in “File” menu. Refer to chapter 5.1 and 5.3 for details.

The user can switch among the 4 mode windows: Manual, Auto, Calibration and Compensation. You have three ways to do it, taking Auto window as the object as an example:

Menu method: choose "Window | Show Auto Window".

Shortcut key method: press “Ctrl+1" key to activate Auto window.

Mouse method: click the window title “Auto” directly.

CAUTION

In this window, the user can only view the loaded file instead of editing or modifying it. To edit this file, the user needs to execute [File | Edit Loaded File] or click the right mouse button in this area and then choose [Edit Loaded File] before editing the file in [Editor] window.

4.8. Manual Operation Window

Manual window provides the user with an interactive machine tool operation environment, as shown in Fig. 4-19.

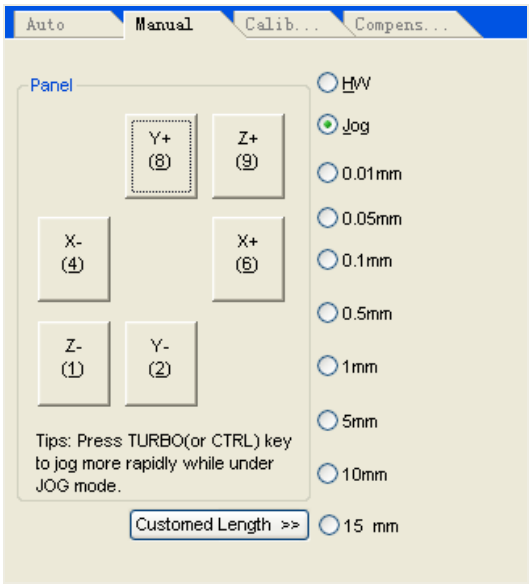


Fig. 4-19 Manual operation window

To switch to Manual window:

Menu method: choose "Window | Show Manual Window".

Shortcut key method: press “Ctrl+2" key to activate Manual window.

Mouse method: click the window title “Manual” directly.

◆ **Numeric Direction Buttons:**

X+ (6) and X- (4) correspond to the positive and negative direction of X axis respectively;

Y+ (8) and Y- (2) correspond to the positive and negative direction of Y axis respectively;

Z+ (9) and Z- (1) correspond to the positive and negative direction of Z axis respectively.

◆ **Selection of Feed Mode**

There are three feed modes: handwheel mode, jog mode and stepping mode, as illustrated below.

1) Handwheel mode

Model of a handwheel is on the right side, as you can see.

1st , check  to enable the MPG.

2nd, turn the “Axis Select Knob” to activate an axis to move.

3rd, turn the “Magnification Gear” to decide the magnification ratio for the moving distance of a tool.

4th, turn the “Turning Wheel” to manipulate the movement of a tool, with turning direction corresponding to the moving direction of tool.

2) Jog Mode

1st, check  to choose the jog mode.

2nd, 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.

CAUTION

When Manual window is active, NUM LOCK will not be taken into consideration.

3) Stepping Mode

Checking any item in Fig. 4-20 will set the system into stepping mode.

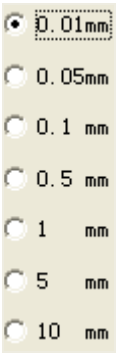


Fig. 4-20 Step-size in Stepping mode

In stepping mode, the machining track in [Trace] 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 [Customed Length] button to open a dialog box where you can set a step length, as illustrated below.

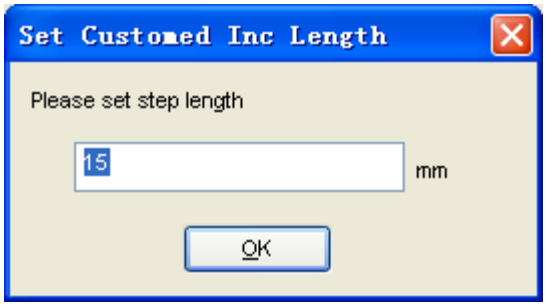


Fig. 4-21 User-defined step length

Enter the proper step length and click “OK” for confirmation.

CAUTION

The customized step length should not be set too large in case of equipment damage caused by mal-operation.

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.”

4.9. Calibration Operation Window

In this window, the user can conduct tool measurement, as shown in Fig. 4-22.

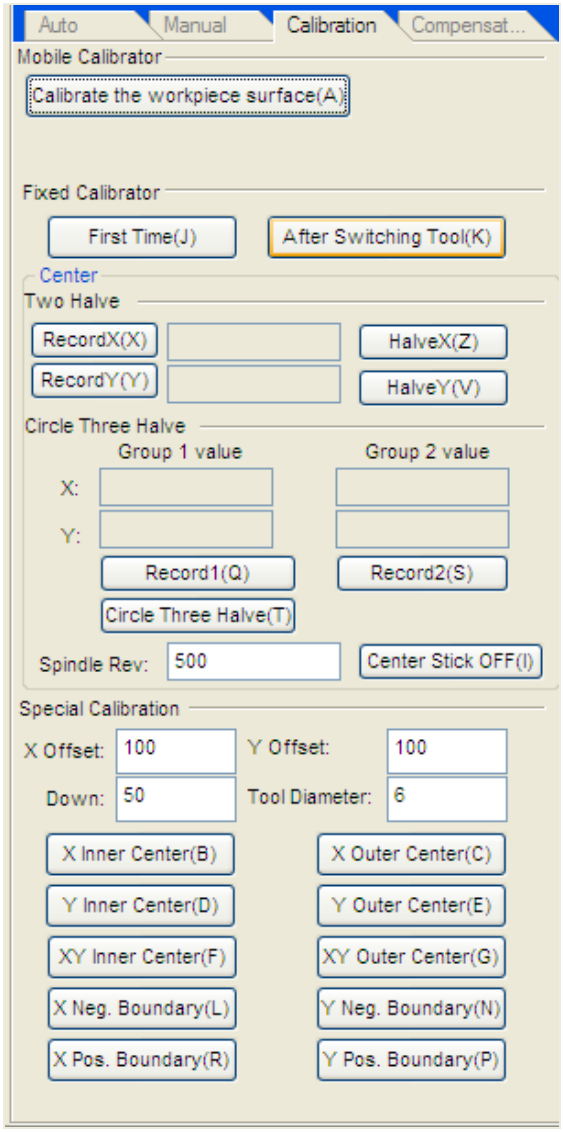


Fig. 4-22 Tool measurement window

4.9.1. Calibrate the Workpiece Surface

Specially designed for Weihong system and used to set the Z axis workpiece origin automatically, this function can help the user to confirm the height of workpiece surface, equaling to mobile calibration when the thickness of tool sensor is 0. The machine tool worktable must be insulated.

The method of mobile calibration is to place the tool sensor on the workpiece surface and then operate Z axis to make the tool nose touch the tool sensor. After the measurement signal (also called calibrating signal) is obtained and calibration stops, the system records the position of tool nose. Thus, the Z axis coordinate of workpiece origin is obtained after the thickness of tool sensor is subtracted from tool nose position.

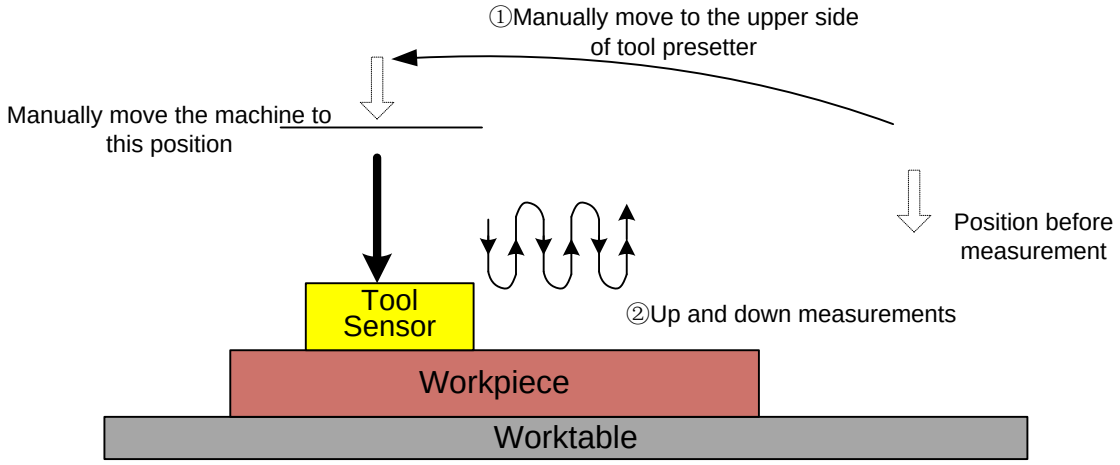


Fig. 4-23 Sketch map of mobile calibration

4.9.2.Fixed Calibration

As the name implies, with an aim to adjust the workpiece origin, fixed calibration refers to the measurement performed via a tool sensor at a fixed position on the machine tool. Weihong company divides it into two types in this system, namely “First time calibration” and “Calibration after tool change”, which is exclusively owned by Weihong company. The interface is shown as below:

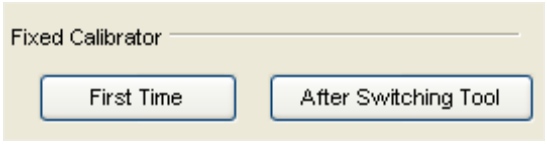
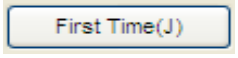


Fig. 4-24 Fixed calibration

To begin, you should decide the baseline of workpiece surface firstly, namely, manually move tool onto the surface and set the coordinate as the workpiece origin or work zero of Z axis. Next, before processing a program file, perform first time calibration. In machining, anytime calibration is required, perform calibration after tool change. Detailed steps are as followed:

1st, manually move the Z axis onto the workpiece surface, and set the origin through manual clear operation or mobile calibration.

2nd, click 

button to perform the first time calibration. The system automatically records the current machine coordinate of Z axis. The process is illustrated as Fig. 4-25.

3rd, the first time calibration is finished, ready for workpiece machining.

4th, during machining, when calibration is required in case of tool change or tool breakage, click 

button to perform calibration after tool change. The system will restore the workpiece origin of Z axis before tool change or breakage automatically. The process is illustrated in Fig. 4-26.

5th, when the calibration is finished, continue machining.

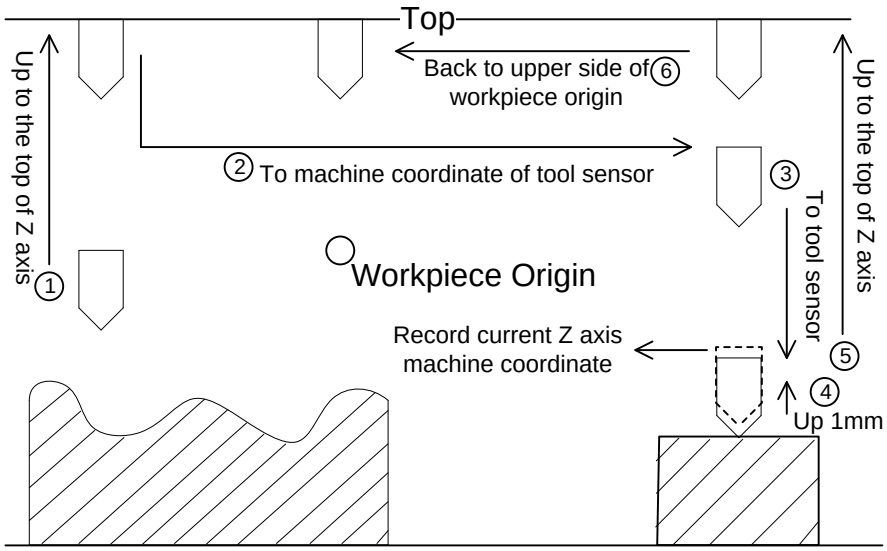


Fig. 4-25 First time calibration

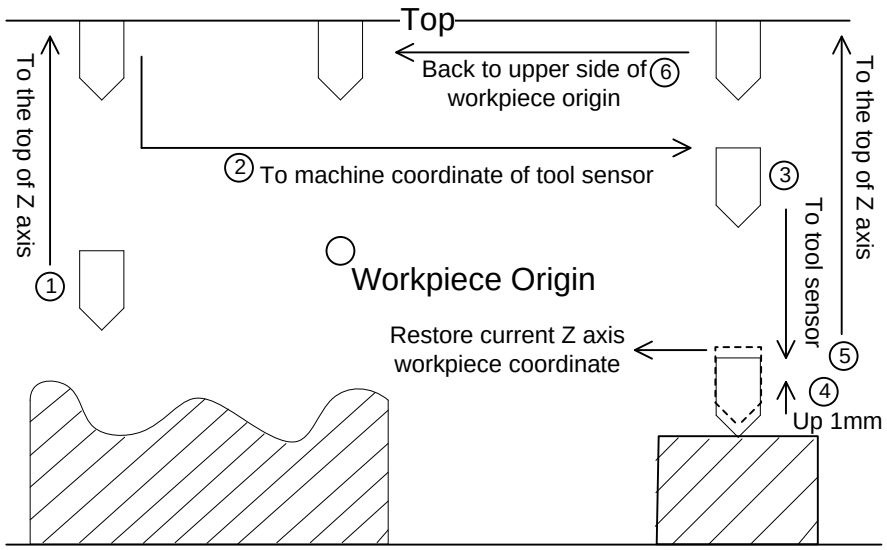


Fig. 4-26 Tool calibration after tool change

CAUTION

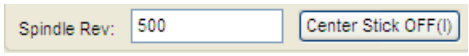
The above measurement results will be ineffective instantly after NcStudio being shut down. Therefore, you need to re-perform the calibration when you start NcStudio again.

"First-time calibration/calibration after tool change" are the most frequently used tool measurement methods and mainly used in simply mode with single WCS or tool. It is different from the fixed calibration used in multiple WCS or multiple tools modes, which is not listed here.

4.9.3.Centering

Centering is used to get the center point of a regular workpiece and set it as the workpiece origin. NcStudio supports two ways of centering, namely two-point centering and circle centering.

In order to get more precise results, a "Center Stick" (i.e. edge finder or touch point sensor) is needed in centering. You can set the spindle speed for the stick and the activation of it in



. Enter a desired value for the centering stick, and click the button to turn on/off the stick.

WARNING

The spindle should be turned off before the centering stick being turned on. Otherwise, there may be dangerous as a result of the fast spindle speed.

◆ Two-point Centering

Two-point centering, also known as line centering, refers to finding the midpoint of a line between any two points in X/Y direction. It can be used to locate the center point of a workpiece, and set it as the starting point of a machining program, namely, the workpiece origin.

Fig. 4-27 Two-point centering (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 X] button. The software will get and record the machine coordinate.

2nd, move the axis to the other side of the workpiece, click [Halve 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.

CAUTION

When one axis is being centered, the other should be motionless.

◆ 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.

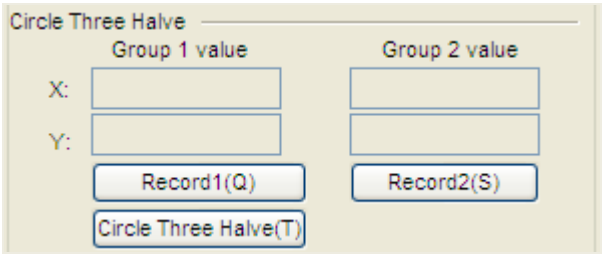


Fig. 4-28 Circle centering

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 1]. 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 2]. 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 [Circle Three Halve]. 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.

4.9.4.Special Measurement

Special measurement includes auto measurement of workpiece origin and workpiece boundary. As measurement signal is used, they are called special measurement, exclusively used in Weihong system. Special measurement is used to calibrate X and Y center of workpiece in order to facilitate machining and generating a machining file, under the precondition that the Z axis workpiece coordinate is confirmed and the worktable of machine tool is insulated.

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

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

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

Tool Diameter: the actual diameter of tool.

◆ **Centering**

Centering is used to confirm the center point of workpiece blank. It includes X inner/outer centering, Y inner / outer centering, and XY inner / outer centering.

Taking the process of [X Inner Center] as an example: place a conducting workpiece (copper, iron, aluminum) on the insulated workbench, and connect it to the port CUT on the terminal board, while the cutter to COM port. Before automatic centering, put the cutter on the predicated center point position, and then click [X Inner Center] to make the cutter fall [Down] distance, and translate towards the workpiece a short distance until it reaches the conducting workpiece. At this time, the circuit is conducted and the signal is sent to the system which automatically records the current axial coordinate X1. Then the cutter will move up [Down] distance, move horizontally two [X Offset], move down [Down] distance,

and translate towards the workpiece a short distance until it reaches the conducting workpiece. As a result, the circuit is conducted and the signal is sent to the system which automatically records the current axial coordinate X2. The system will then calculate the X coordinate of workpiece center point and move the cutter to this center point. The sketch map is as shown below.

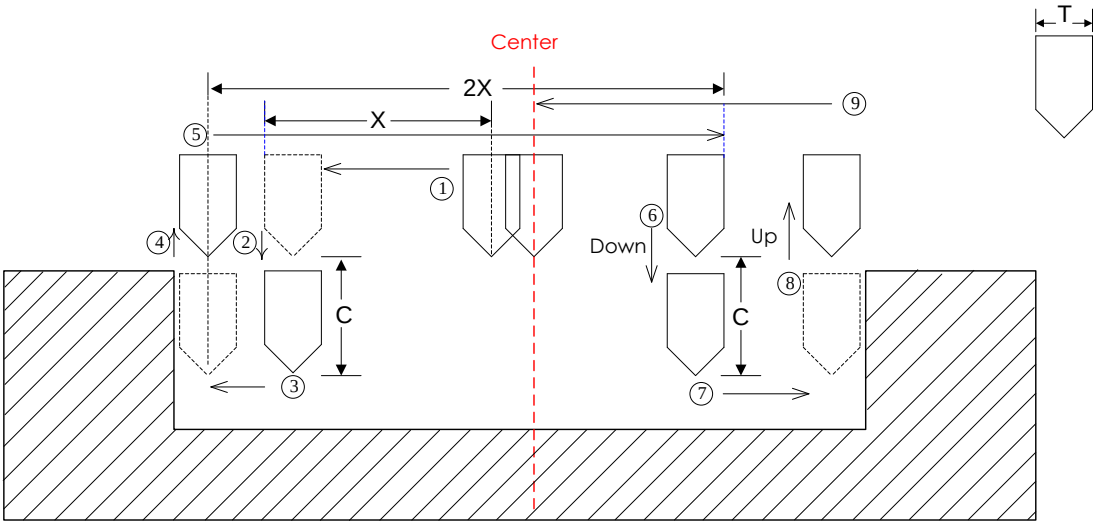


Fig. 4-29 Measurement process of [X Inner Center]

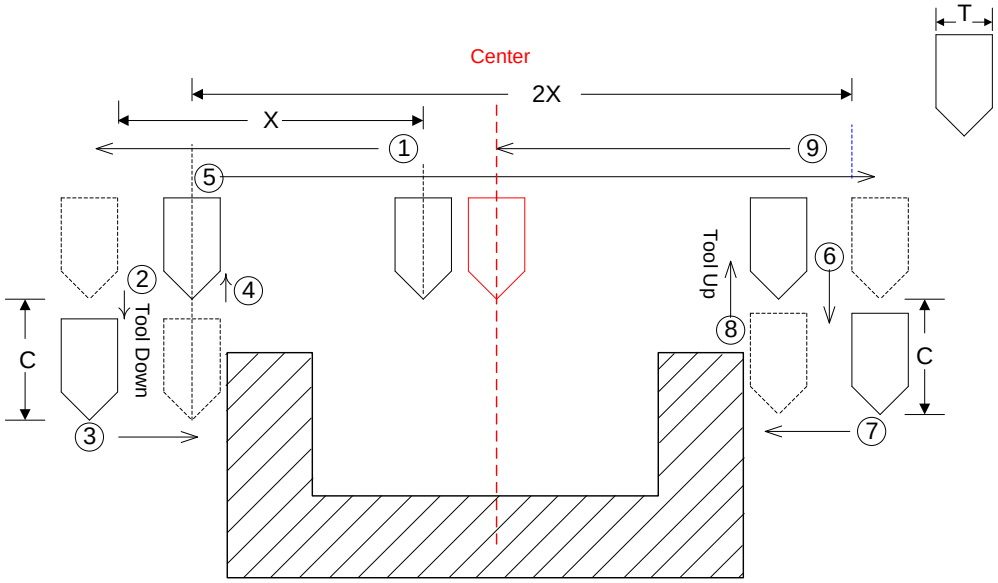


Fig. 4-30 Measurement process of [X Outer Center]

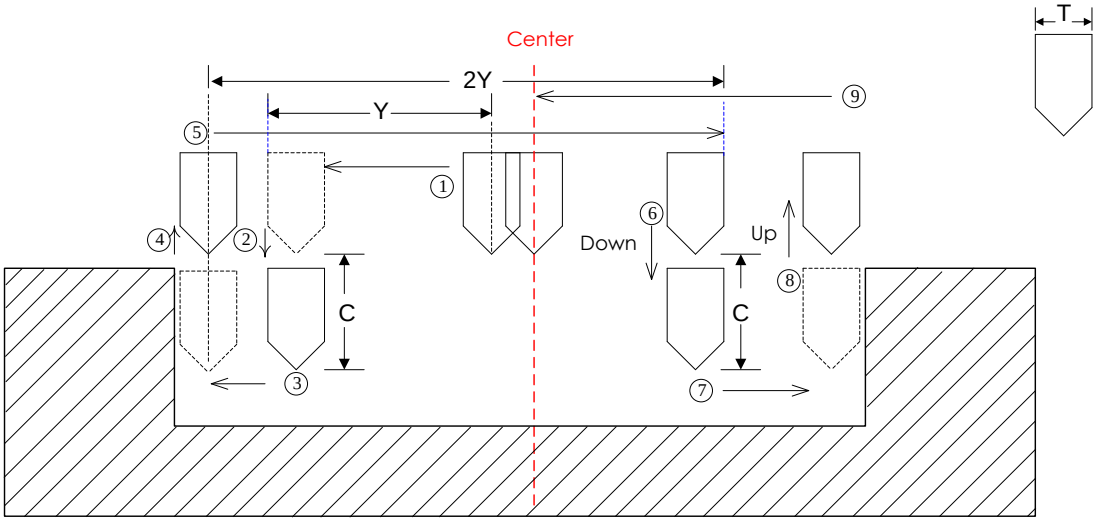


Fig. 4-31 Measurement process of [Y Inner Center]

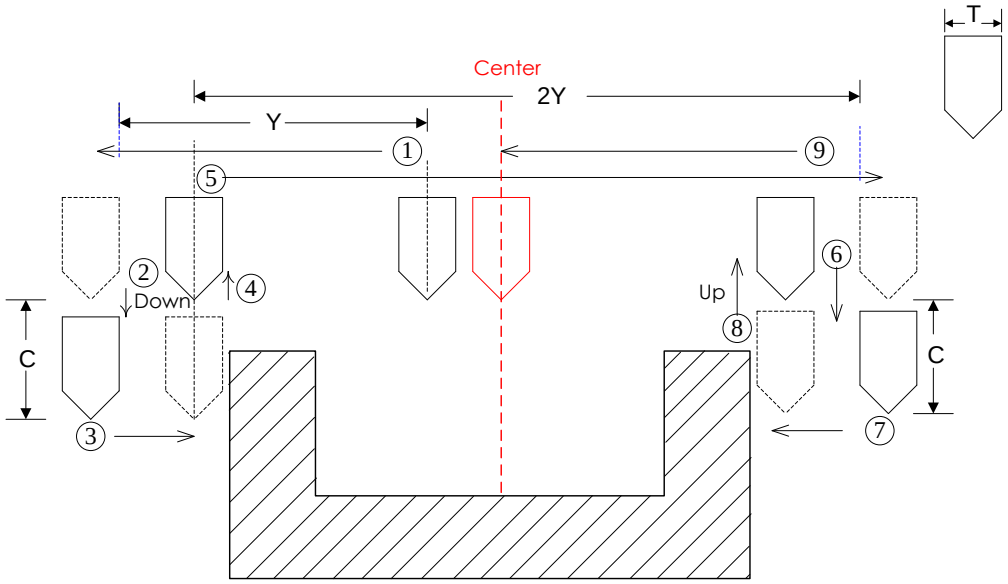


Fig. 4-32 Measurement process of [Y Outer Center]

CAUTION

Before centering, the user must place the tool nose over the pre-estimated center point, as well as set the value of "X\Y Offset", "Down" and "Tool Diameter".
In inner centering, the "Down" distance should be smaller than the distance between tool nose and workpiece surface, while larger in outer centering.

◆ Boundary Measurement

Boundary measurement includes calibrating +X boundary, -X boundary, +Y boundary and -Y boundary, mainly used for setting the boundary point as the workpiece origin.

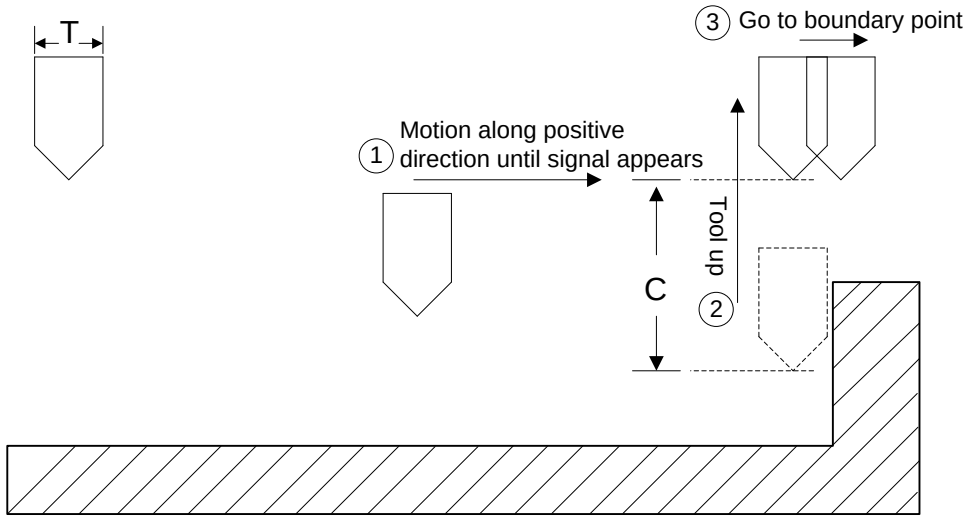


Fig. 4-33 Measurement process of +X boundary

The measurement processes of -X boundary, +Y boundary and -Y boundary are the same as that of +X boundary.

CAUTION

Special measurement function is not available for double Z axis series software, please note that.

In double Z axis series software, “Calibrate the workpiece surface” function is absent; instead, the counterpart “Coordinate manager and calibrate” function is available. The two may differ in minor ways which should be paid attention to. You can turn to chapter 11.2 for details.

4.10. Compensation Operation Window

The system offers two kinds of workpiece compensation, single compensation and row-column compensation. When one of them is selected, the corresponding button will turn to grey. In Fig. 4-34, single compensation is selected. Single compensation will be made to compensate each workpiece separately, i.e. the compensation for each file can be different, while row-column compensation is to compensate machine files in a row or in a column. For example, X01Y01 in row-column compensation mode can compensate the first row and column of all files, and so on.

To enable workpiece compensation, firstly select “Comp enabled” as “Yes”. Otherwise, workpiece compensation does not work.

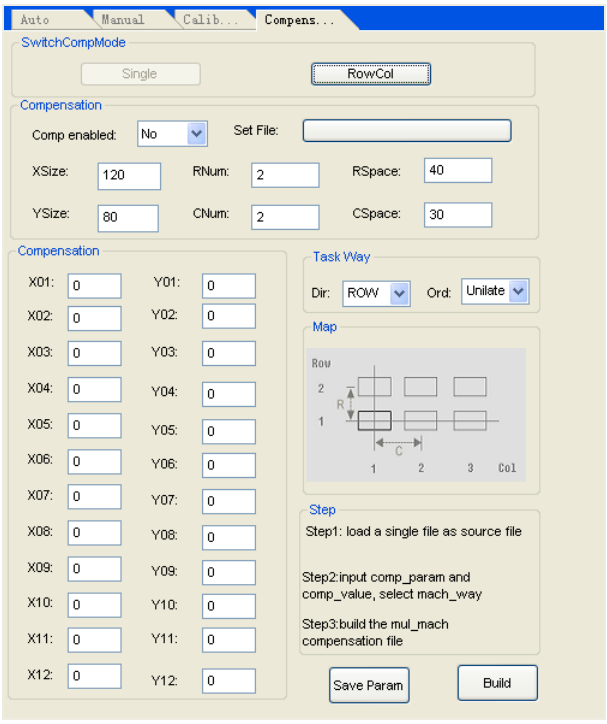


Fig. 4-34 Compensation window

Set File: used to load the single workpiece file to be processed. Before clicking this button, load the file into the system and then click this button to load it into this function, or the system will give a prompt that the source file does not exist, as shown below.

Fig. 4-35 Source file error prompt

After entering the value of X\Y size, row\column number and row\column space, first select direction and order under “Task Way” tab, then enter the relative compensation value of each workpiece. Next click “Build” to generate a multi workpiece file. Designated to the save path, the new generated files will be loaded into the system automatically.

CAUTION

After the file is loaded into this compensation function, the source file in the system will be unloaded automatically, because the final machining file is the newly generated compensation file.

After parameter setting, remember to click “Save Param”. Otherwise, the setting of parameters will be lost after a new file is loaded.

Some codes are not supported in scale and array functions, like G28, G29, G65, G92, M30 and M2, neither are the subroutines in the tool path. If there are above-mentioned codes, the system will prompt the user to delete them manually or automatically.

4.11. Trace Window

When machining or simulation is being performed, the trace window will follow the machining track of tool in real time. In this window, the user then can view the tool path intuitionally so as to ensure the proper implementation of machining file.

Trace window adopts 3D view mode and can be personalized via the option [Customize Trace View…] under [View] menu. Refer to the following chapters for details.

In 3D tracking mode, the various functions this system provides enable the user to zoom in/out and view the object from different orientations and at proper scaling.

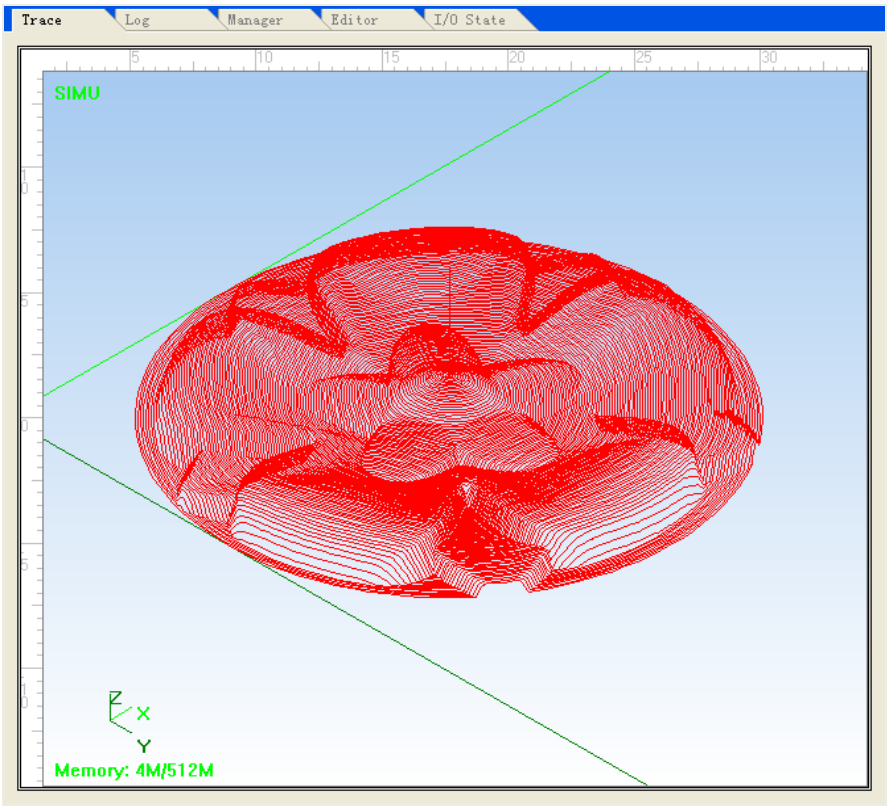


Fig. 4-36 Trace window

Right clicking on the window will open a shortcut menu, as shown in Fig. 4-37. All of these options can also be found under [View] menu.

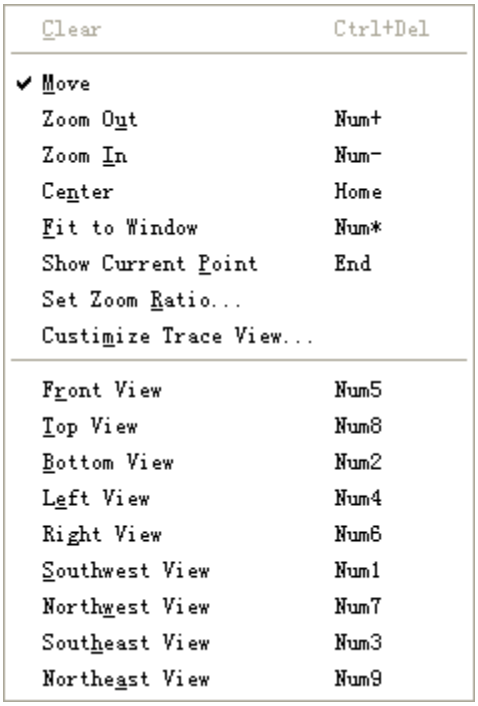


Fig. 4-37 Shortcut menu of trace window

◆ **Clear**

After a long period of machining, the simulation graph will become very complicated and the temporary file recording the machining track will also become excessively big. As it is time-consuming to re-draw, move or rotate the simulative graph, the user needs to clear the machining track regularly.

To clear machining track, several options are available: menu, toolbar, shortcut key or keyboard.

Menu or Toolbar: choose menu "Edit | Clear View", or choose  on the toolbar.

Shortcut Key: at any moment, pressing “Ctrl+ Del” keys will clear the machining track.

Mouse Method: move the mouse into Trace window, right click to eject a shortcut menu, and then choose “Clear”.

Keyboard Method: when the Trace window is activated, press “Delete” key to clear the track.

◆ **Move**

Mouse Method: after the mouse is into Trace window and right clicked, a shortcut menu will pop up. Select “Move”, and you will see the icon changing into  . Hold down the left mouse, and you will see the icon changing into  . At this time, the machining track moves along the mouse.

Keyboard Method: when the Trace window is active, the machining track can be moved by pressing the four direction keys on the keyboard.

◆ **Zoom in/out**

The user can zoom in/out the machining track via mouse, keyboard or menu.

Menu Method: choose “View | [Zoom in] or [Zoom out]”.

Mouse Method: move the mouse into the Trace window and click the right mouse button. Select “Zoom in” or “Zoom out” in the pop-up menu to switch between and .

Keyboard Method: when the Trace window is active, the user can zoom in/out the track by pressing “＋” or “－”on the mini-keyboard. Note that “+” or “-“ on the main keyboard are invalid.

CAUTION

The user can use mouse to pick a part or an area of the track in the Trace window to zoom in/out.

When the track is magnified to the maximum size, the system will automatically switch to “Zoom out” mode.

When the track is reduced to the minimum size, the system will automatically switch to “Zoom in” mode.

◆ **Center**

It is used to show the center of current machining scope in the central area of Trace window.

Menu Method: select "View｜Center View".

Mouse Method: move the mouse into Trace window and click the right mouse button. Then select "Center" in the pop-up shortcut menu.

Keyboard Method: press “Home” key on the keyboard when the Trace window is active.

◆ **Fit to Window**

This function enables the entire machining track displayed on the Trace window so that the user can view the whole machining track without scrolling.

Menu Method: select "View | Fit to Window" on the menu bar.

Mouse Method: move the mouse into the Trace window and click the right mouse button. Select "Fit to Window" in the pop-up shortcut menu.

Keyboard Method: press “ * ” on the keyboard when the Trace window is active.

◆ **Show Current Point**

The current machining position will be displayed in the center of the Trace window.

Menu Method: choose "View| Show Current Point".

Mouse Method: move the mouse into the Trace window and click the right mouse button. Select "Show Current Point".

Keyboard Method: press "End” on the keyboard when the Trace window is active.

◆ **Set Zoom Ratio**

To show the machining file track in the Trace window at an appropriate scale.

The easiest way to set the ratio is adjusting by rolling the mouse wheel. Move the cursor into the Trace window, and roll the mouse wheel when the cursor turns to . Rolling up equals to zoom out, while rolling down to zoom in.

This function (Ratio/Set Zoom Ratio) can also be found under “View” menu or right-click menu when Trace window is active. After “Set Zoom Ratio” is selected, a dialog as shown in Fig. 4-38 will pop up. The user can drag the block or use the direction keys to adjust the display scale.

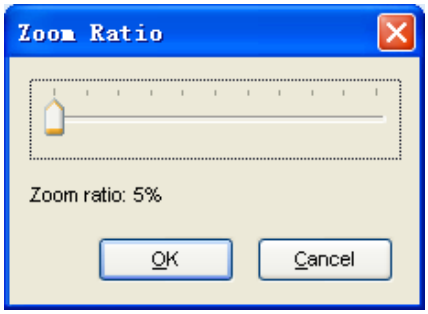


Fig. 4-38 Zoom ratio

◆ **Customize Trace View**

Choose “View | Customize” on the menu bar, or when the trace window is active, right click the mouse button to choose “Customize trace view…” from the pop-up shortcut menu. A “Customize” dialog box will pop up as below, where the user can customize the trace window and design the tracking mode and tracking color.

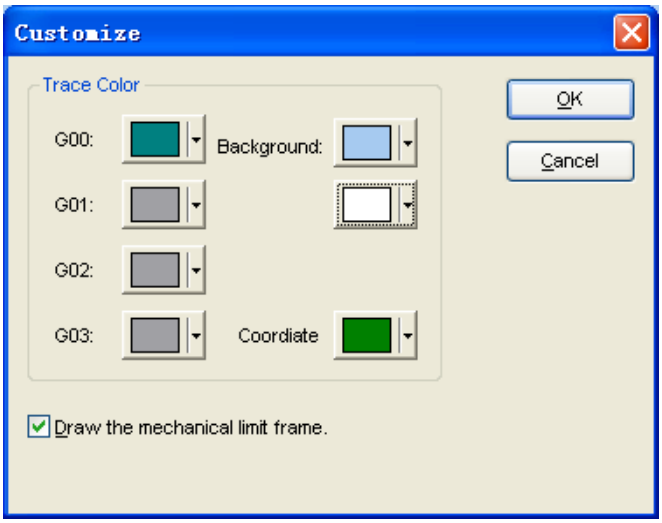


Fig. 4-39 Customize dialog

With a click on a color button, a color selection box will pop up, as illustrated below.

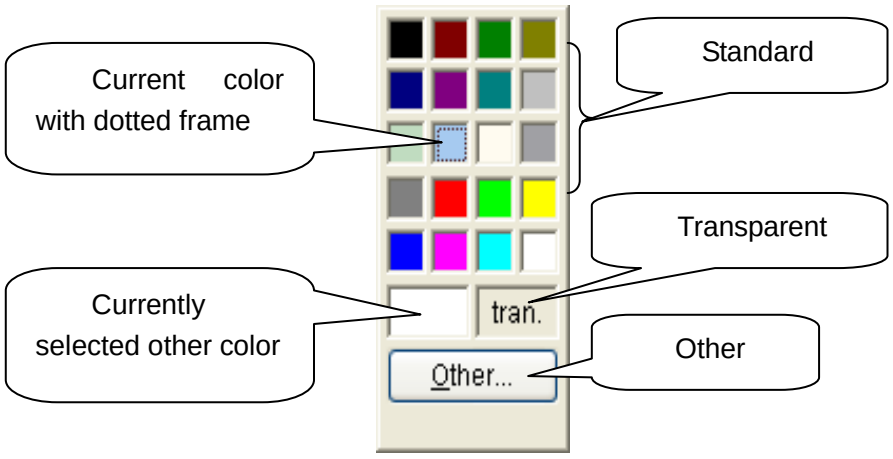


Fig. 4-40 Color selection box

1) Trace Color

In the Trace window, the user can set different colors for different codes via “Customize Trace View”.

G00 color: this color indicates the track color of G00.

G01 color: this color indicates the track color of G01.

G02 color: this color indicates the track color of G02.

G03 color: this color indicates the track color of G03.

The track color in manual mode is the same as that of G01.

2) Background Color

It is the background color of Trace window. The user can choose two different colors to realize color gradient.

3) Coordinate Color

This color is used to highlight the worktable borders and the coordinate system.

CAUTION

If the trace color of a certain code is set as transparent, the track of that code will be hidden. If the background color is set as transparent, the window can’t be refreshed properly. Therefore, the user needs to be cautious when selecting “transparent”.

◆ **View Function**

The system provides 9 types of commonly-used views to help the user switch among these views by pressing the Num keys on the small keyboard.

Southwest equiaxial lateral view	Num key 1
Bottom view	Num key 2
Southeast equiaxial lateral view	Num key 3

Left view	Num key 4
Front view	Num key 5
Right view	Num key 6
Northwest equiaxial lateral view	Num key 7
Top view	Num key 8
Northeast equiaxial lateral view	Num key 9

4.12. Log Window

The system log window records all the critical operations and events. The user can not only review all the existing log information recorded since system starts, but can review the historical log information.

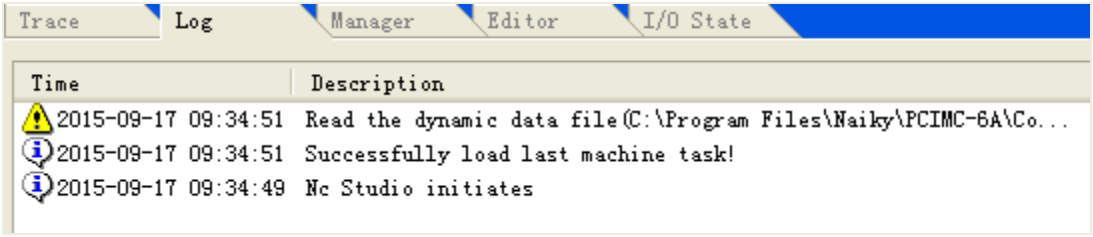


Fig. 4-41 System log window

The log information includes:

1) System on/off;
2) Auto machining on/off;
3) Workpiece coordinates modification;
4) System alarms;
5) Other system information.

A shortcut menu will pop up by right-clicking the mouse button in the system log window, as illustrated below. A check mark "√" before the item means it is selected, while none means it is deselected.

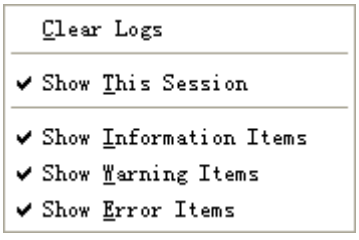


Fig. 4-42 Shortcut menu of system Log window

◆ **Clear Logs**

It will clear the current log information in the Log window.

Menu method: choose "Edit| Clear Log".

Mouse method: move the mouse into the Log window and right click the button to choose "Clear Logs" in the pop-up shortcut menu.

CAUTION

Please clear the system log regularly, or the system performance and responding time will be affected due to the overlarge log file

◆ **Show This Session**

Right click on “Log” window and then choose “Show This Session” from the pop-up shortcut menu, or choose “Show This Session’s Logs” from the menu “View”.

With this item selected, the window will only show the log information of operation this time. If this item is not selected, the user can view the log information of both operation this time and history operation.

◆ **Show Information Items**

Right click on “Log” window and then choose “Show Information Items” from the pop-up shortcut menu, or choose “Show Information Items” from the menu “View”.

Each information log is marked with the icon .

With this item selected, log information like system on/off will be shown in the window. If it is not selected, the log information will be concealed.

The system log item is marked with the icon .

◆ **Show Warning Items**

Right click on “Log” window and then choose “Show Warning Items” from the pop-up shortcut menu, or choose “Show Warning Items” from menu “View”.

Each warning log is marked with the icon.

This function can show warning log. If the function is not selected, the warning information will be concealed.

◆ **Show Error Items**

Right click on “Log” window and then choose “Show Error Items” from the pop-up shortcut menu, or choose “Show Error Items” from menu “View”.

Each error log is marked with the icon.

This function can show error log. If the function is not selected, the error info will be concealed.

4.13. Manager Window

Manager window is an area where the machining files are managed. NcStudio can manage the machining files saved in the designated directory via a built-in manager. In this window, the user can conduct functions like new, edit, delete, rename and load, etc.

After a right click in the Manager window, a shortcut menu will pop up, as illustrated below.

Fig. 4-43 Shortcut menu in Manager window

◆ **Create a new program file**

There are three ways to create a new machining file instantaneously:

One: select “File| New” (shortcut key: Ctrl+N);

Two: right click in the Manager window and select [New] from the pop-up shortcut menu;

Three: click 

on the lower part of Manager window.

Then the system will automatically generate a new machining file “Untitle1.nc” and the user can decide the save location of the new file.

Fig. 4-44 Save location

Click  to choose the target location or directory.

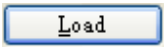
The user can also edit, delete, rename or load the newly created file. Refer to the subsequent chapters for details.

◆ **Open an existing program file**

File list box in the Manager window shows the machining files under current folder.

The user can not only enter the designated path into “Current file folder path” or select another path by clicking  . “File list” displays all machining files under current path.

“File extension” box is used to display the extensions of files under current path. The user can delete or add one. The extension of file is also displayed in the “File list” box.

Double clicking one of the files in the “File list” box will load it automatically. The user can also click 

at the lower part to load the selected file. In addition, “File| Open and Load” can also be used.

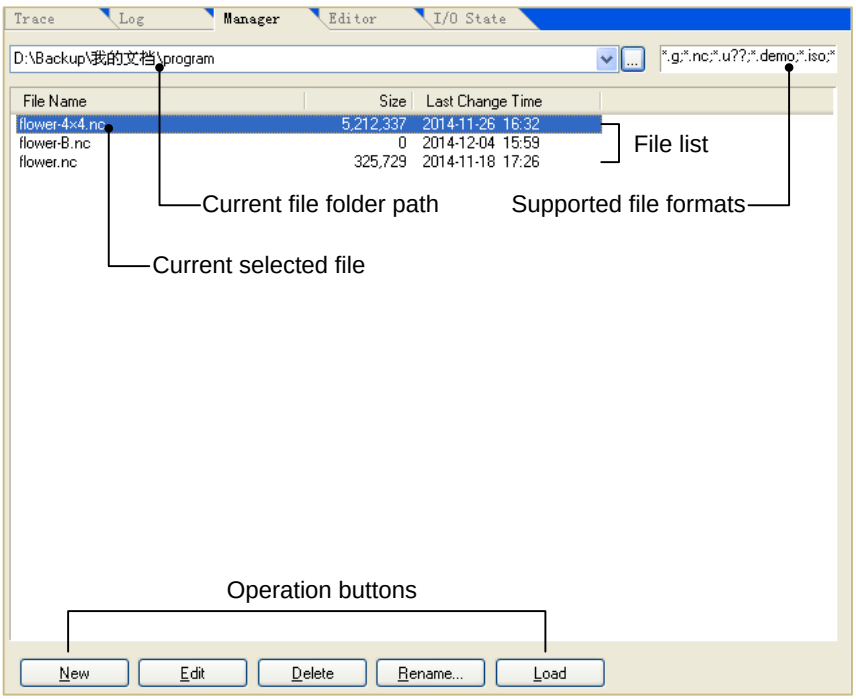


Fig. 4-45 Manager window

◆ **Edit**

There are three ways to edit the selected machining file:

One: select “Edit| Edit the Selected File”.

Two: right click in the Manager window and then select “Edit” from the pop-up shortcut menu.

Three: click 

on the lower part of Manager window.

And then the system will switch to the Editor window automatically. The user then can do editing in this window. For details, refer to Editor Window.

For a loaded program file, apart from the above three ways, you can also edit it in the auto mode window. Of course, you need to load the program file into the system first, right click the mouse button in the Auto window where program lines of the loaded file will be shown, and select “Edit Loaded File” item from the pop-up menu.

◆ **Delete**

There are three ways to delete the selected machining file:

One: select “Edit| Delete the Selected File”.

Two: right click in the Manager window and select “Delete” from the pop-up shortcut menu.

Three: click 

on the lower part of Manager window.

A dialog box will pop up for confirmation. And you need to confirm that the selected machining file to be deleted is not being edited or called at present. Click [Yes] to delete it.

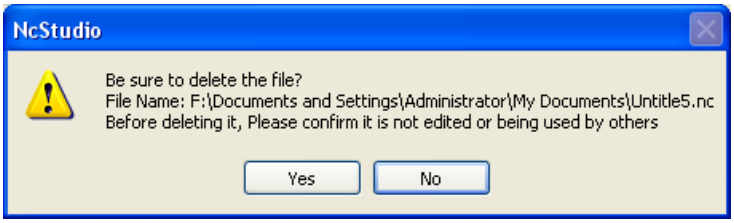


Fig. 4-46 Delete a machining file

CAUTION The selected file in the file list box is highlighted (in blue).

◆ **Rename**

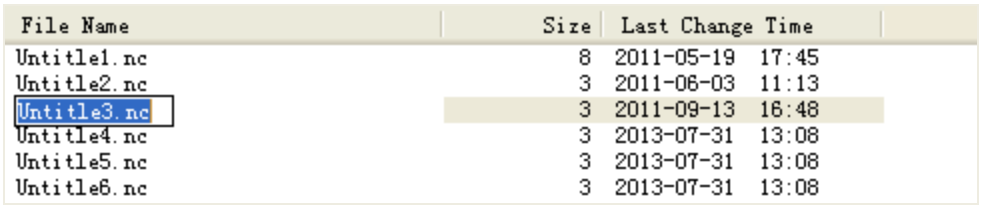
There are three ways to rename the selected machining file:

One: select “Edit| Rename the Selected File”.

Two: right click in the Manager window and then select “Rename” from the pop-up shortcut menu.

Three: click Rename... on the lower part of Manager window.

Then the file name will be in edit state, as illustrated in Fig. 4-47:



File Name	Size	Last Change Time
Untitle1.nc	8	2011-05-19 17:45
Untitle2.nc	3	2011-06-03 11:13
Untitle3.nc	3	2011-09-13 16:48
Untitle4.nc	3	2013-07-31 13:08
Untitle5.nc	3	2013-07-31 13:08
Untitle6.nc	3	2013-07-31 13:08

Fig. 4-47 Rename dialog box

After entering the file name, press “Enter” key or click on a blank area to activate it.

◆ **Load**

There are five ways to load the selected machining file:

One: select “File| Open and Load”.

Two: right click in the Manager window and then select “Load” from the pop-up shortcut menu.

Three: click Load on the lower part of Manager window.

Four: click on the toolbar.

Five: right click in the Auto window and then select “Open and Load” from the pop-up shortcut menu.

4.14. Editor Window

In the Editor window, on the top of this window is the name and path of the file being edited, as shown in Fig. 4-48. As you can see, it is a standard PC-style editing window.

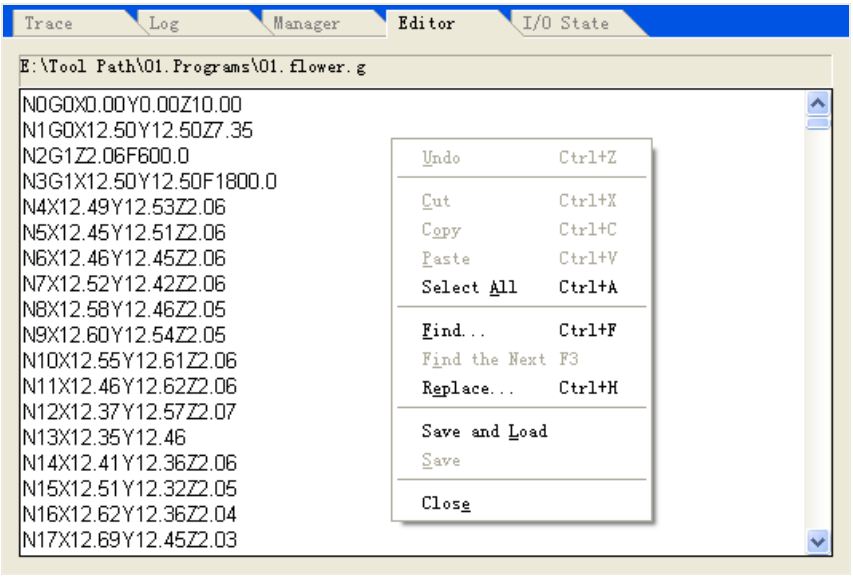


Fig. 4-48 Editor window

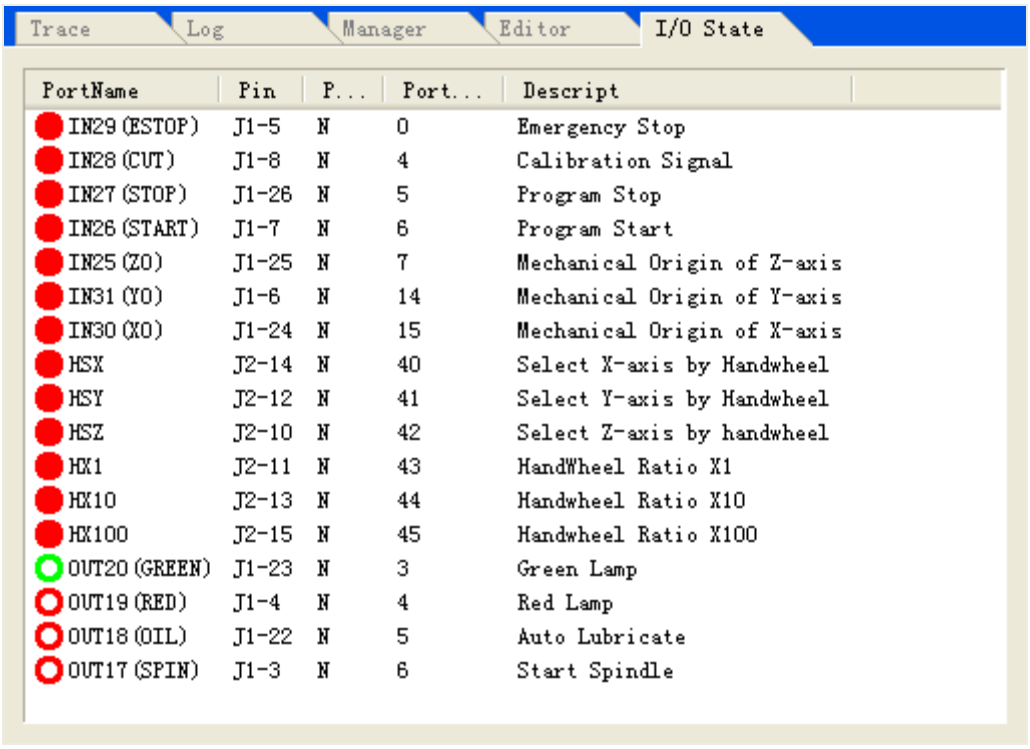
It can edit a machining file with a size bigger than 1000M byte. In other words, it can meet the editing requirements of any complex machining file.

Right clicking in the Editor window will open a shortcut menu for your choice.

CAUTION Releasing time values represents the ones with DC-cutoff using a varistor.

4.15. I/O State Window

The I/O State window displays the current state of system I/O ports, which is very helpful for system monitoring and troubleshooting.



PortName	Pin	P...	Port...	Descript
IN29 (ESTOP)	J1-5	N	0	Emergency Stop
IN28 (CUT)	J1-8	N	4	Calibration Signal
IN27 (STOP)	J1-26	N	5	Program Stop
IN26 (START)	J1-7	N	6	Program Start
IN25 (ZO)	J1-25	N	7	Mechanical Origin of Z-axis
IN31 (YO)	J1-6	N	14	Mechanical Origin of Y-axis
IN30 (XO)	J1-24	N	15	Mechanical Origin of X-axis
HSX	J2-14	N	40	Select X-axis by Handwheel
HSY	J2-12	N	41	Select Y-axis by Handwheel
HSZ	J2-10	N	42	Select Z-axis by handwheel
HX1	J2-11	N	43	HandWheel Ratio X1
HX10	J2-13	N	44	Handwheel Ratio X10
HX100	J2-15	N	45	Handwheel Ratio X100
OUT20 (GREEN)	J1-23	N	3	Green Lamp
OUT19 (RED)	J1-4	N	4	Red Lamp
OUT18 (OIL)	J1-22	N	5	Auto Lubricate
OUT17 (SPIN)	J1-3	N	6	Start Spindle

Fig. 4-49 I/O State window

As shown in the above picture, the icons in the front of the ports indicate:

- Green filled dot means that there is signal input in this port;
- Red filled dot means that there is no signal input in this port;
- Green hollow dot means that there is signal output in this port;
- Red hollow dot means that there is no signal output in this port.

CAUTION

Please note that the content displayed in this window may vary because of the configuration of different motion control cards and various user-defined requirements. The information provided here is for reference only.

5. NcStudio Menu System

5.1. "File" Menu

The optional items in File menu is as following:

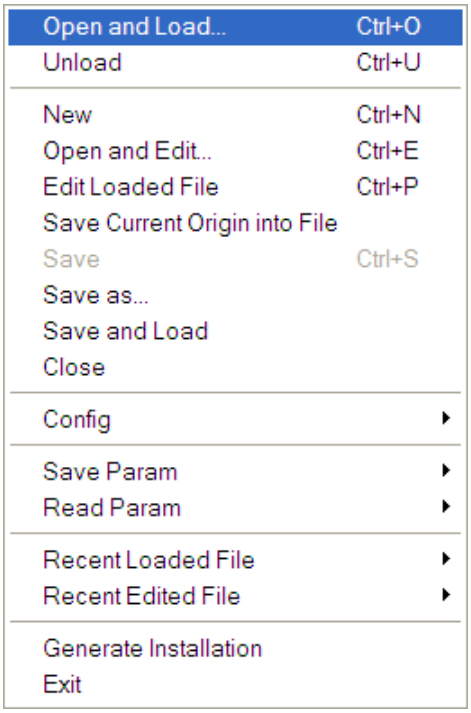


Fig. 5-1 File menu

◆ **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.

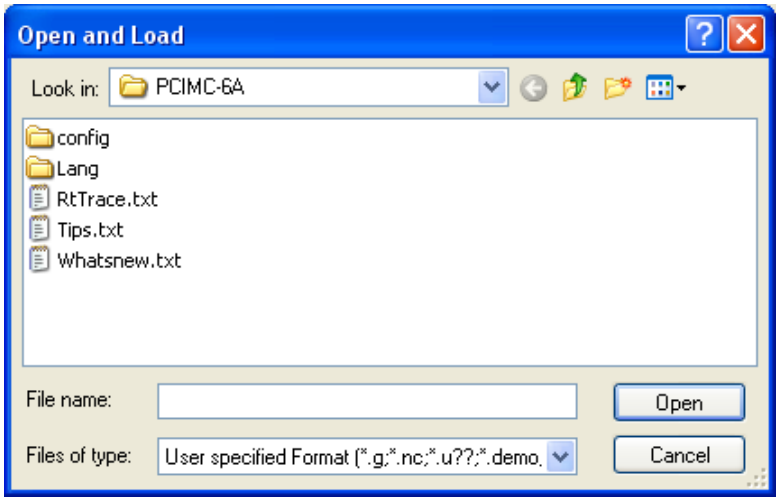


Fig. 5-2 Open and load dialog box

Clicking the pull-down button in 

can call the processing files under other routes. After selecting the file, click “Open”, the name of the file displayed on the title bar.

◆ **Unload**

It is used to unload the current machining file.

◆ **New**

It is used to create a new machining file.

With this item selected, “Editor” window will be activated for file editing. After editing, right click and select “Save” from the pop-up shortcut menu.

◆ **Open and Edit**

It is used to open an existing machining file and switch to the Editor window to edit it.

◆ **Edit Loaded File**

It is used to edit the currently loaded machining file.

◆ **Save the Current Coordinate Origin**

It is used to save the current coordinate origin into its machining file.

CAUTION Releasing time values represents the ones with DC-cutoff using a varistor.

◆ **Save**

It is used to save the machining file after editing.

◆ **Save as**

It is used to resave the current machining file in the Editor window in another name.

◆ **Save and Load**

It is used to save and load the current machining file in the Editor window as the currently loaded machining file.

◆ **Close**

It is used to close the machining file being edited.

◆ **Config**

You can check the current configuration and change it through this item. Configuration option with mark “√” before is the active configuration.

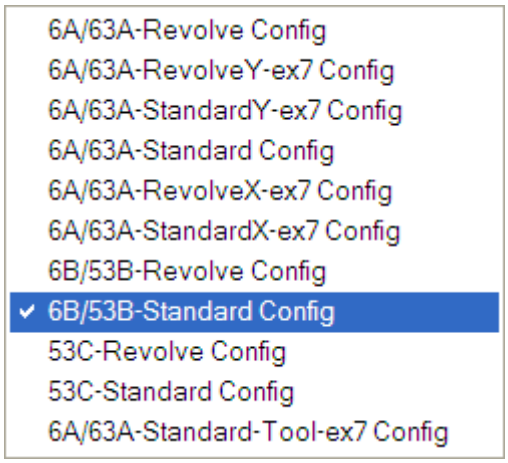


Fig. 5-3 Configuration options

When changing the configuration, a prompt dialog will pop up before a new one activated. You need to confirm your new selection and restart the software as required.

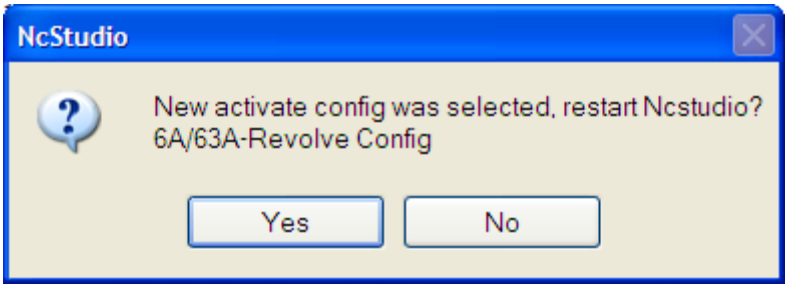


Fig. 5-4 Prompt before configuration change

CAUTION

The configuration is set by the machine tool builder, and the user is not entitled to change it. Otherwise, the machine tool may not work properly.
The above examples are configuration options in general three axes software, please turn to chapter 11.3.1 for details about those in double Z axes software.

◆ **Save/Read Parameter**

To save or read the parameter setting made in the current configuration. As many as 10 groups of parameter settings can be backed up.

◆ **Recent Loaded File**

This menu item includes a new submenu, displaying the recently loaded machining files. The user can re-load them quickly with this function.

◆ **Recent Edited File**

This menu item includes a new submenu, displaying the recently edited machining files. The user can re-edit them quickly with this function.

◆ **Generate Installation**

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:

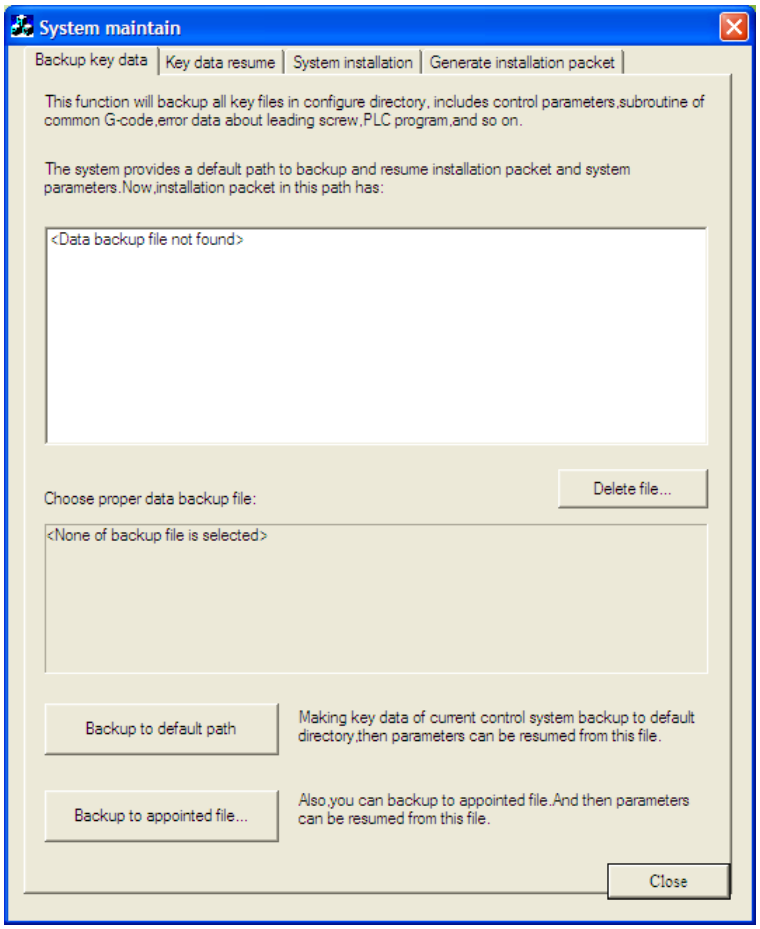


Fig. 5-5 System maintenance dialog

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.

◆ **Exit**

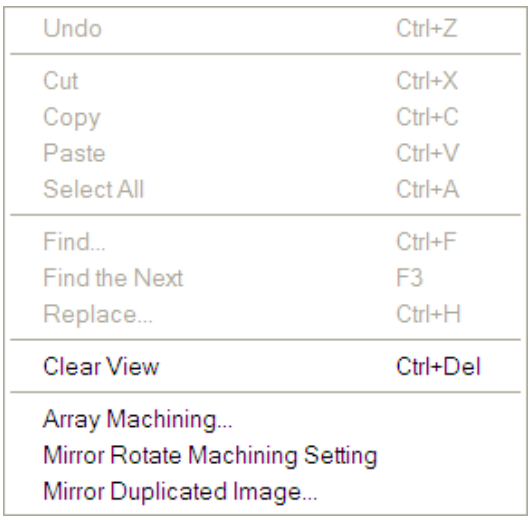
It is used to shut down and exit the NcStudio system.

Menu items “Save the current coordinate origin”, “Save parameter”, “Read parameter” as well as “Generate installation” are absent in double Z axes software, please note that. You can turn to chapter 11.3.1 for more details.

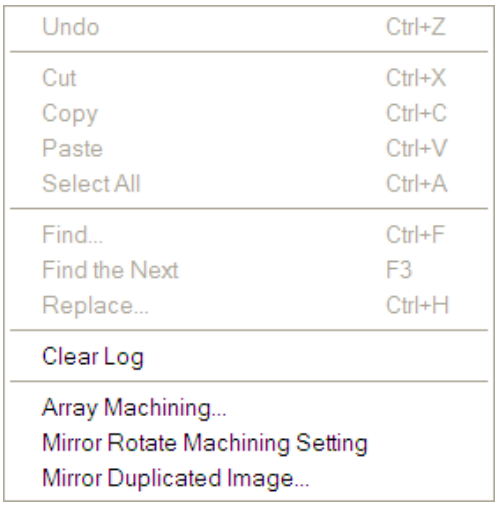
5.2. "Edit" Menu

The items in “Edit” menu change with the currently active window in the second window area (including Trace window, Log window, Manager window, Editor window and I/O State window).

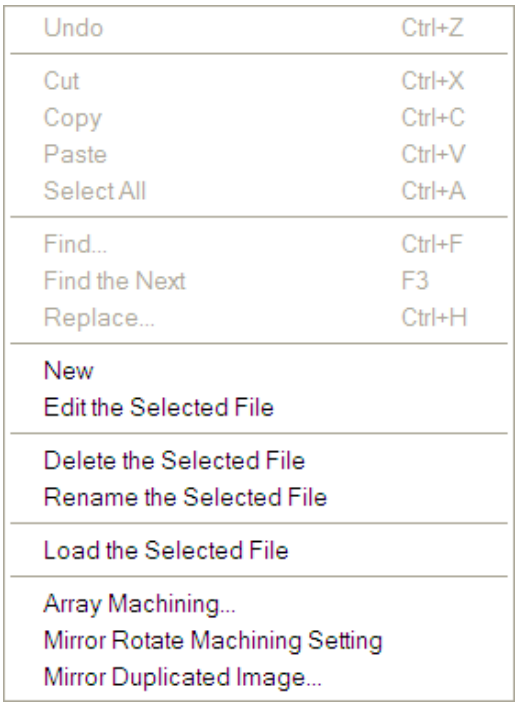
With Trace window active



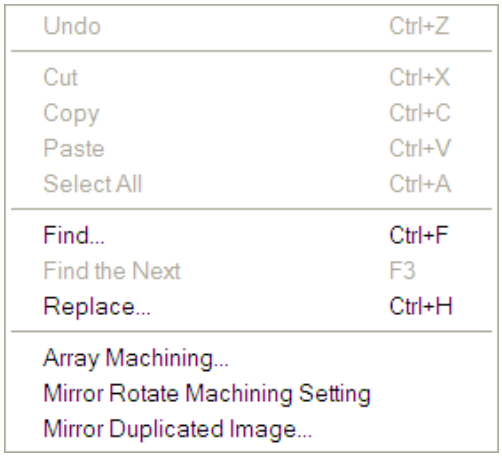
With Log window active



With Manager window active



With Editor/IO State window active



◆ **Clear view**

Clear the tracking image of the program in the Trace window. Please turn to chapter 4.11 for details.

◆ **Clear log**

Clear all log information in Log window. Please turn to chapter 4.11 for details.

◆ Array Machining

This function allows executing array machining on one machining file. Clicking this item will open a dialog box, as illustrated below:

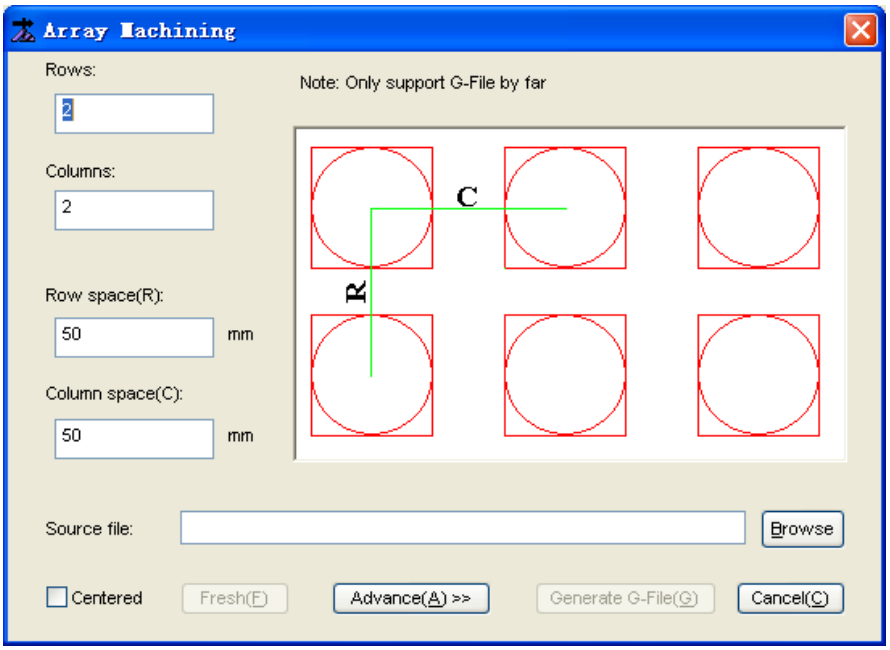
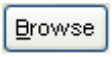


Fig. 5-6 Array machining dialog

Click  and select the machining file for array operation.

Set the correct column number, row number, row space (distance between two workpiece origins, as R length mentioned in the above figure) and column space (distance between two workpiece origins, as C length mentioned in the above figure) and then click [Generate G-File]. The newly generated processing file after array will be loaded into the NcStudio automatically.

G codes like G65 and G92 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.

To set different row spaces and column spaces, click [Advance]. Then a dialog box will pop up shown as below.

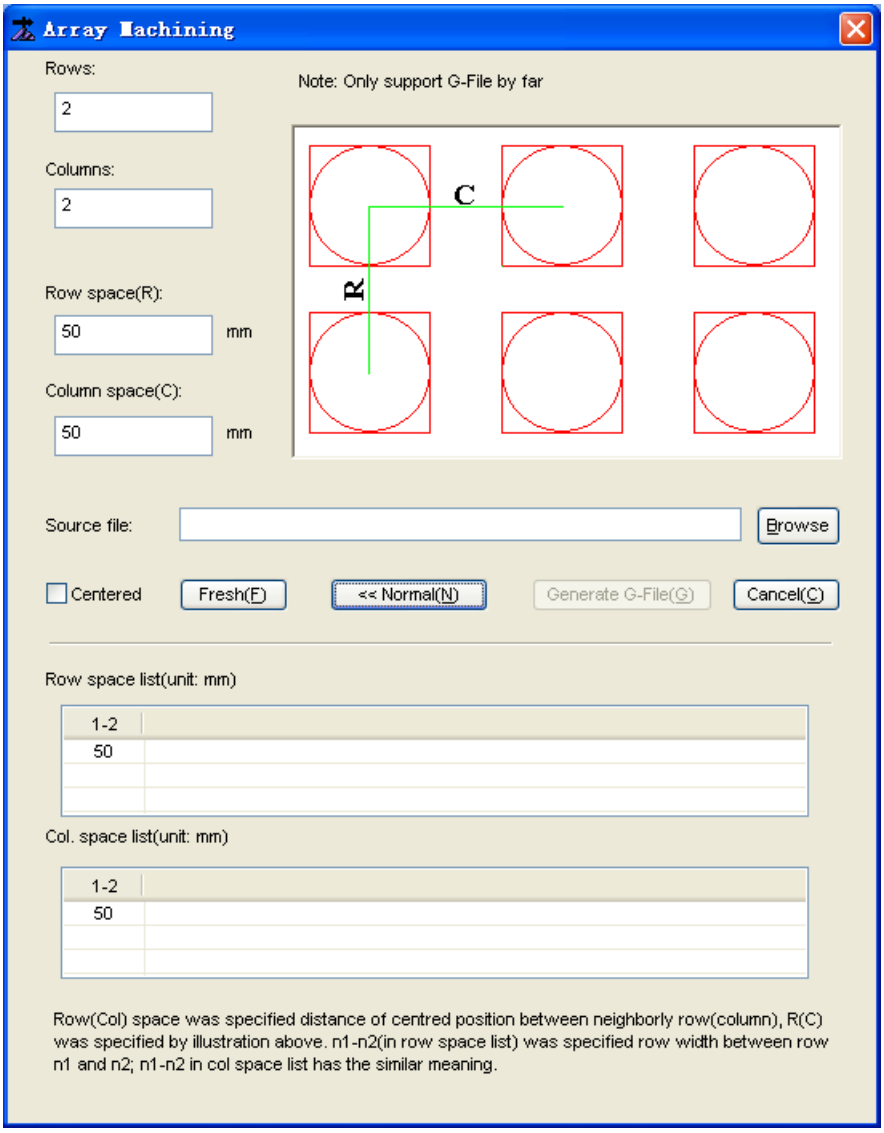


Fig. 5-7 Array machining in advanced mode

In [Row space list] and [Column space list], the user can separately set the row/column space between any two rows or columns. Click “Fresh” button to update the value entered.

◆ **Mirror Rotate Machining Setting**

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. 5-8.

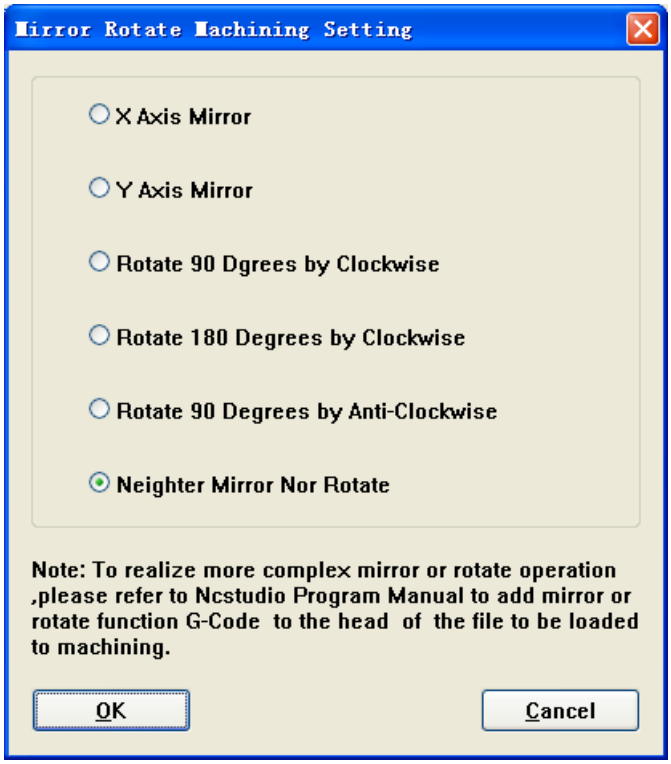
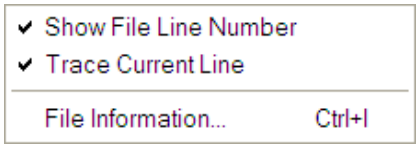


Fig. 5-8 Mirror & rotation machining

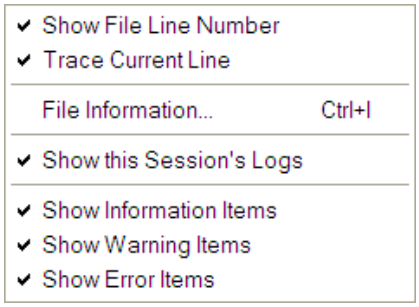
After the user selects one of the above options and clicks [OK], the newly-generated machining file will be loaded into NcStudio automatically.

5.3. "View" Menu

The items in “View” menu change with the currently active window in the second window area (including Trace window, Log window, Manager window, Editor window and I/O State window).

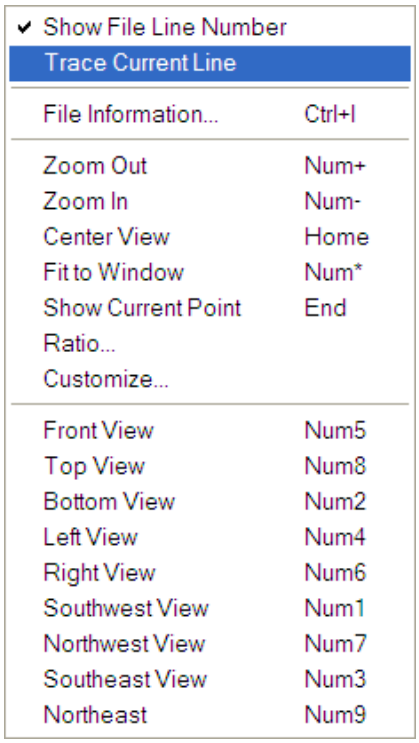


The “View” menu with Manager/Editor/IO State windows active



The “View” menu with Log window active.

Please turn to chapter 4.12 for details.



The “View” menu with Trace window active.

For menu items related with tracking image views adjustment, please turn to chapter 4.11 for details.

◆ **Show File Line Number**

This function is used to display or conceal the program line number of machining file, only available when the Auto window is activated.

◆ **Trace Current Line**

The function is used to trace the program line number of current instruction in the Auto window during machining.

◆ **File Information**

Click this item to open a dialog box titled “File Information”, as illustrated below.

The dialog box displays the statistic information of machining file during auto processing, such as total time, machining range, etc. Combining with simulation function, the user can learn various information of machining file quickly and rapidly.

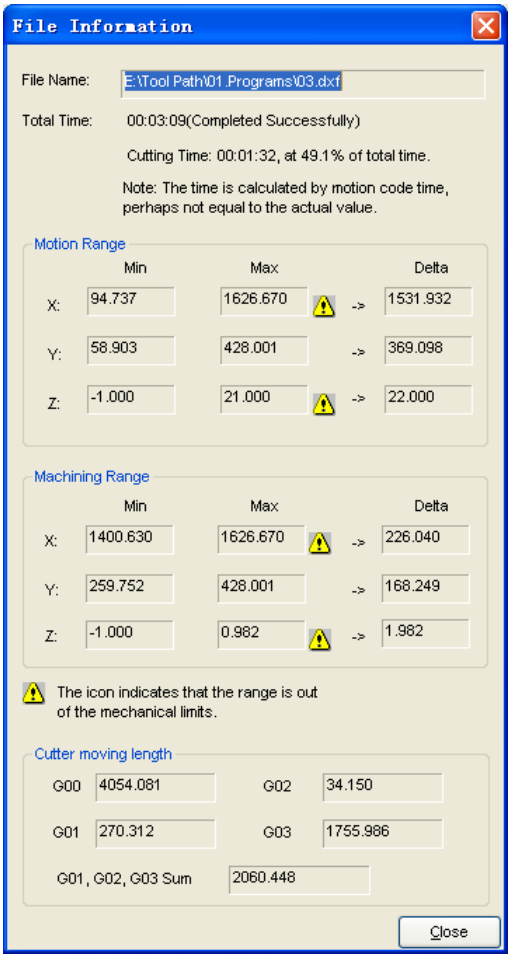


Fig. 5-9 Statistics of machining file

◆ **Total Time**

It displays the total time of machining and cutting time.

◆ **Motion Range**

It specifies the maximum and minimum workpiece coordinates of machine tool movement in machining.

◆ **Machining Range**

It specifies the maximum and minimum workpiece coordinates of machine tool in actual cutting. As shown in Fig. 5-9, coordinate with a warning icon ⚠ means the axis is travelling out of the limit.

5.4. "Operation" Menu

Here are the menu items in “Operation” menu:

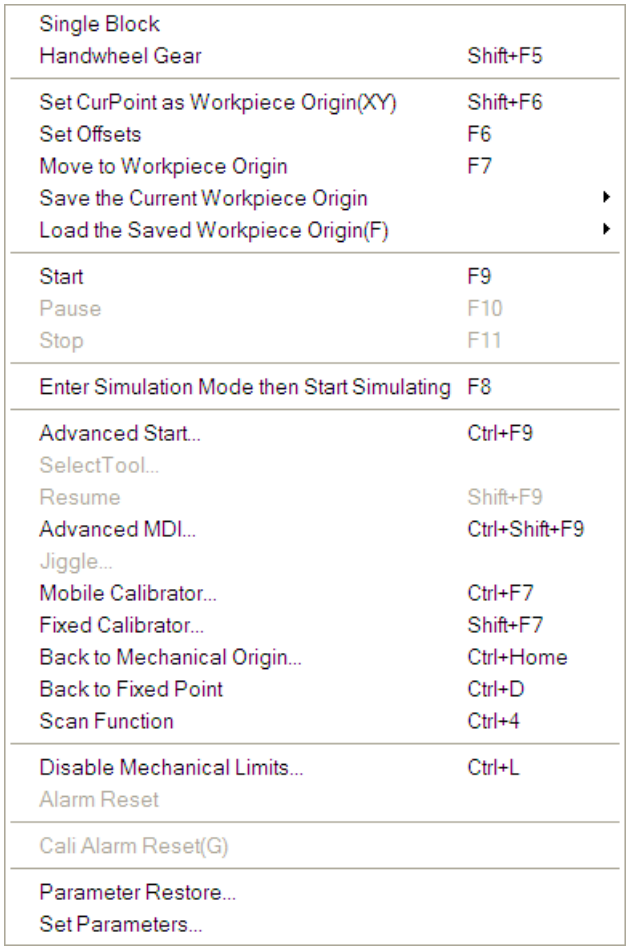


Fig. 5-10 Operation menu

◆ **Single Block**

With the function activated, every time you click (Start), the system runs a program block and pauses; when you click (Start) again, it will run the next program block.

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.

◆ **Hand wheel Gear**

In Auto mode, with the item activated, the system will implement the machining file with the turning of handwheel when 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 Current Coordinate as Workpiece Origin

Sets current coordinates of X axis and Y axis as the workpiece origin.

◆ Set Offsets

Right clicking in any part of NC State Widow can also call this function, Its dialog shown in Fig. 5-11.

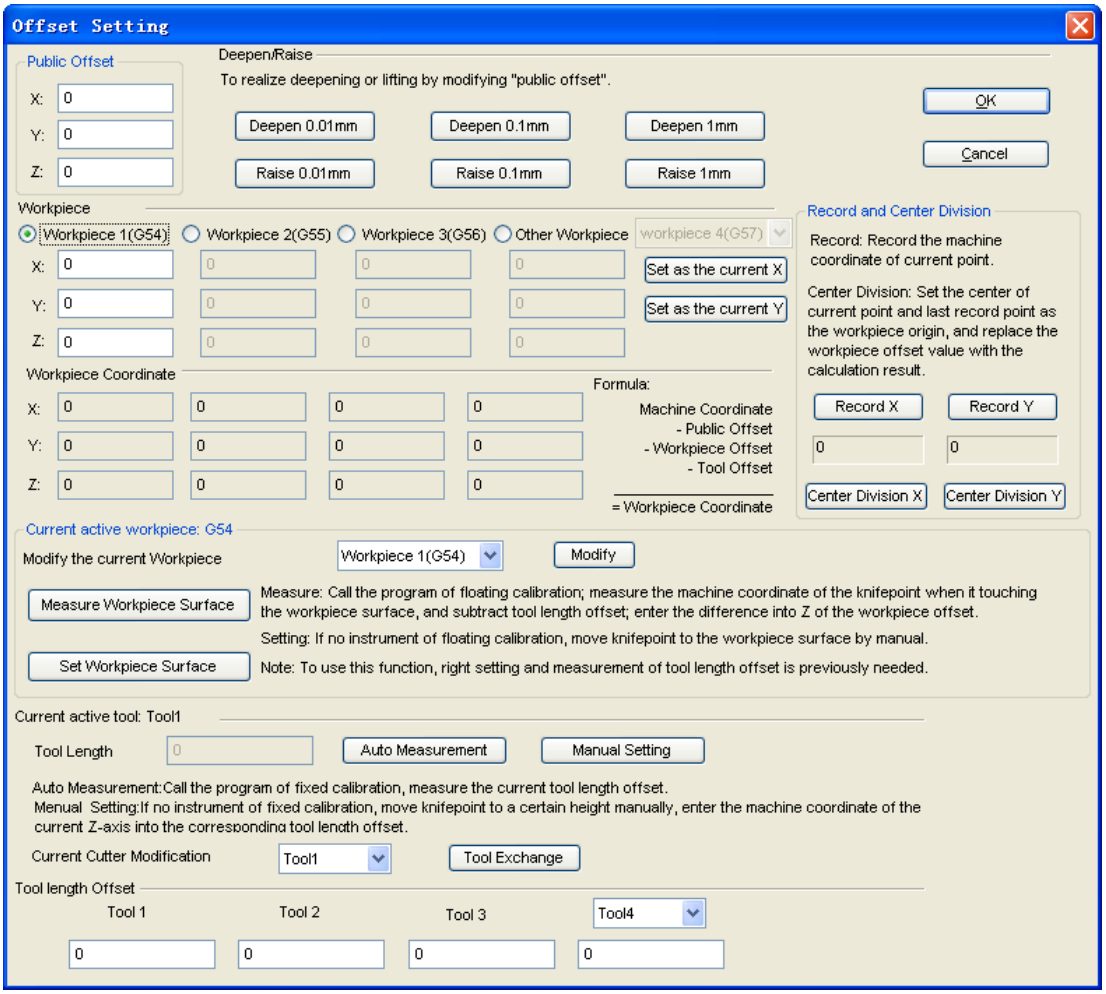


Fig. 5-11 Set offset

A. Public Offset

Also called external offset, it is used to record the temporary adjustment value of workpiece origin. As it can only be adjusted manually and will not be changed during the execution of any auto function, the value will not be changed during fixed calibration and mobile calibration. For example, if the value is not 0 before measurement, the workpiece coordinate will not be 0 either when measurement completes. The following buttons can be used to modify the value of Z public offset.

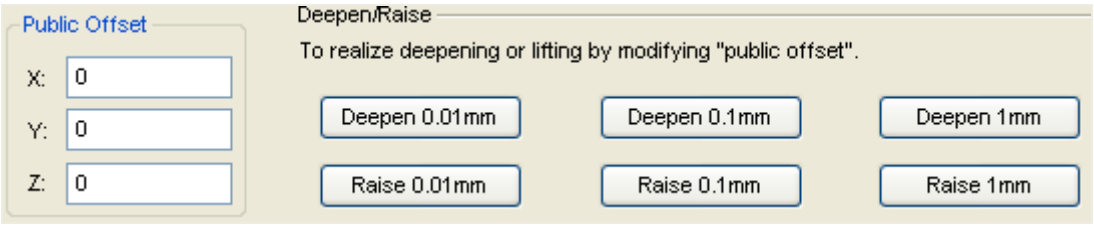


Fig. 5-12 Buttons to modify Z public offset

One of the above buttons clicked, the Z workpiece origin will move up or down a specified distance to form a new workpiece coordinate system; this distance will accumulate with the button clicked repeatedly.

B. Workpiece

It specifies workpiece origin. See Fig. 5-13.

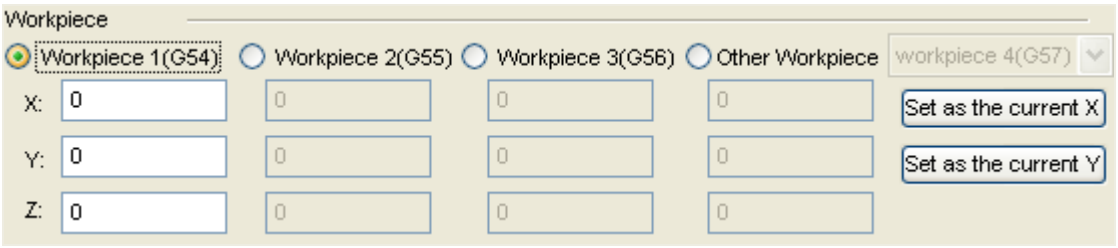


Fig. 5-13 Workpiece offset

It displays the machine coordinates of workpiece origin. Manually entering values can be used to set the machine coordinates of workpiece origin, though not recommended.

C. Workpiece Coordinate

It displays the workpiece coordinates of current cutting point. See below.

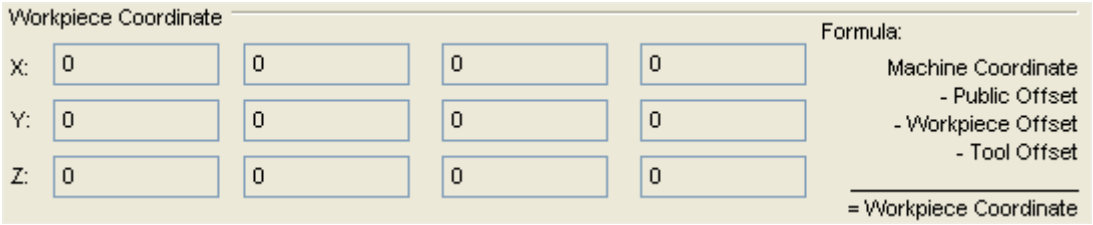


Fig. 5-14 Workpiece coordinate

The equation is as below:

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

D. Record and Center Division

The following buttons are used to get the intermediate point of two points on a regular workpiece, with aim to obtain the origin. The method is: firstly, move the tool to the first target point and then click [Record X], the system recording the X machine coordinate of this point. Secondly, move the tool to the second target point and then click [Center Division X], the system automatically working out the X

machine coordinate of the intermediate point and make it as the X origin. This method goes for Y coordinate too. For more details, see chapter 4.9.3.

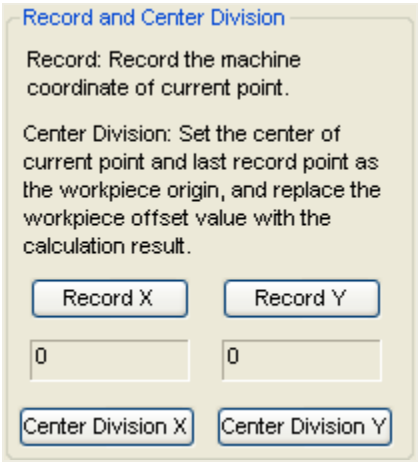


Fig. 5-15 Record and center division

◆ **Move to Workpiece Origin**

With the item selected, if the tool nose is below the safe height, the Z axis will move up to the safe height first and then the X axis and Y axis will move to the workpiece origin together; if the tool nose is above the safe height, the X axis and Y axis will move to the workpiece origin together first and then the Z axis will move down to the safe height.

The user can also execute this function by clicking  on the toolbar.

◆ **Save the Current 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. See Fig. 5-16.

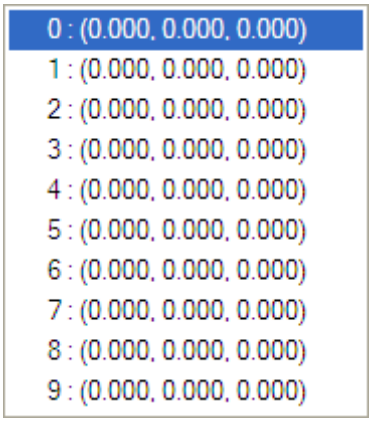


Fig. 5-16 Save the current workpiece origin

Before you save the origin, a dialog box will pop up asking for confirmation.

Click [Yes] to confirm and save the origin, click [No] to cancel it.

◆ **Load the Saved Workpiece Origin**

It is used to load the coordinate value of the saved workpiece origin.

◆ **Start**

With [Start] clicked, the system will enter into auto processing mode. If simulation is activated, the system will run the program in simulation mode.

The user can also execute this function by clicking  on the toolbar.

◆ **Pause**

During auto processing, with [Pause] selected, the system will suspend processing and raise the cutter to enter into “Auto| Pause” state. To resume processing, click [Start].

When simulating, [Pause] chosen, the system will suspend simulation and enter into “Auto| Pause” state. To resume simulation, choose [Start].

The user can also execute this function by clicking  on the toolbar.

◆ **Stop**

During auto processing, [Stop] chosen, the machine tool will cease processing and raise the cutter to enter into “Auto| Idle” state. This is the normal way to stop when processing.

When simulating, [Stop] chosen, the system will suspend simulation and enter into “Auto| Idle” state. To start or resume simulation, choose [Start], [Advanced Start] or [Breakpoint Resume], etc.

The user can also execute this function by clicking  on the toolbar.

◆ **Enter Simulation Mode then Start Simulating**

With the item selected, the machine tool will conduct high-speed simulation from the beginning of the machining file rapidly and vividly.

In simulation mode, differing from actual processing, the system just displays the tool path in the Trace 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.

Once simulation starts, this menu item will change into” Stop Simulating then Leave Simulation Mode”. It will make the system deactivate the simulation mode immediately.

The user can also execute this function by clicking  on the toolbar.

◆ **Advanced Start**

With the item selected, a dialog box titled “Execute (Advanced Options)” will pop up, as in Fig. 5-17.

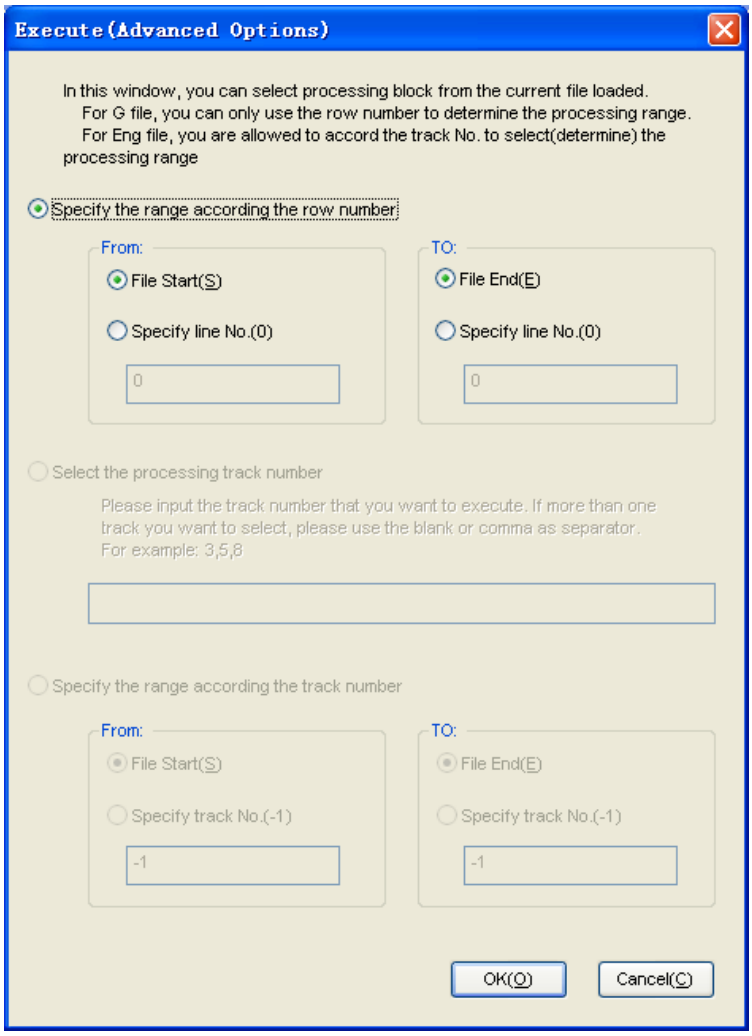


Fig. 5-17 Advanced start

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.

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

If a G code file is loaded, only "Specify the range according to the row number" is available, while the other two unavailable.

If an ENG file is loaded, the three options are all available.

◆ Select Tool

It is a function specified for ENG file or G file. In addition, to enable this function, the parameter "N4093 MachiningEngFileByToolNumber" or "N4129 MachiningGFileByToolNumber" should be set as "true" correspondingly first.

◆ **Resume**

It is also called breakpoint resume or resume from interrupted point. With the item selected, the system will automatically resume processing from the stop line number (breakpoint) of last machining.

In case of sudden power failure, E-stop, etc, this function (breakpoint resume) can be executed to make the machine tool quickly move to the breakpoint and resume processing, which will save considerable machining time.

The user can also execute this function by clicking on the toolbar.

◆ **Advanced MDI**

With the item selected, an “Advanced Functions” dialog box will pop up, as shown in Fig. 5-18, including 5 function windows: [Rectangle Mill], [Round Mill], [Rectangle Frame Mill], [Round Frame Mill] and [MDI]. 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).

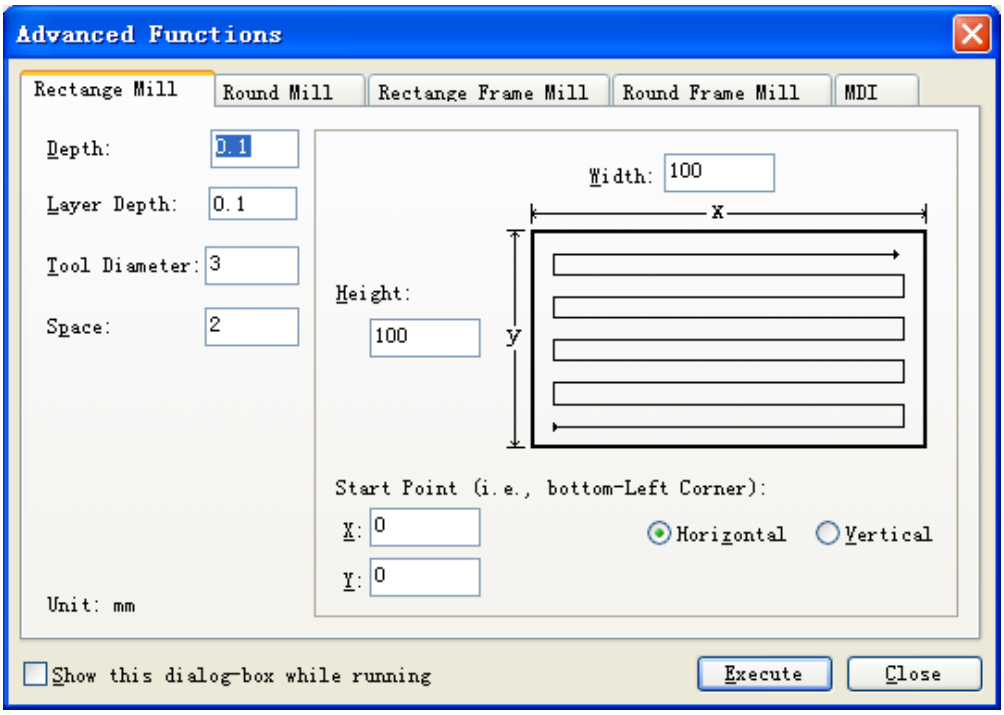


Fig. 5-18 Rectangle bottom milling window

See Fig. 5-19. After entering standard instructions (like G codes, T codes and M Codes, etc) into the edit box of “MDI” window, click “Execute”, the system will execute the entered instructions instantly. In addition, instructions entered previously are recorded in the window for inquiry.

When you want to enter more than one instruction at a time, please input semicolon “;” to break or separate them. When the instructions are not entered properly, NcStudio will give out a prompt.

If “Show this dialog-box while running” at the bottom is checked, this dialog can still be seen when executing the instructions, for the convenience of rapidly setting relevant parameters of bottom milling and frame milling or entering standard instructions.

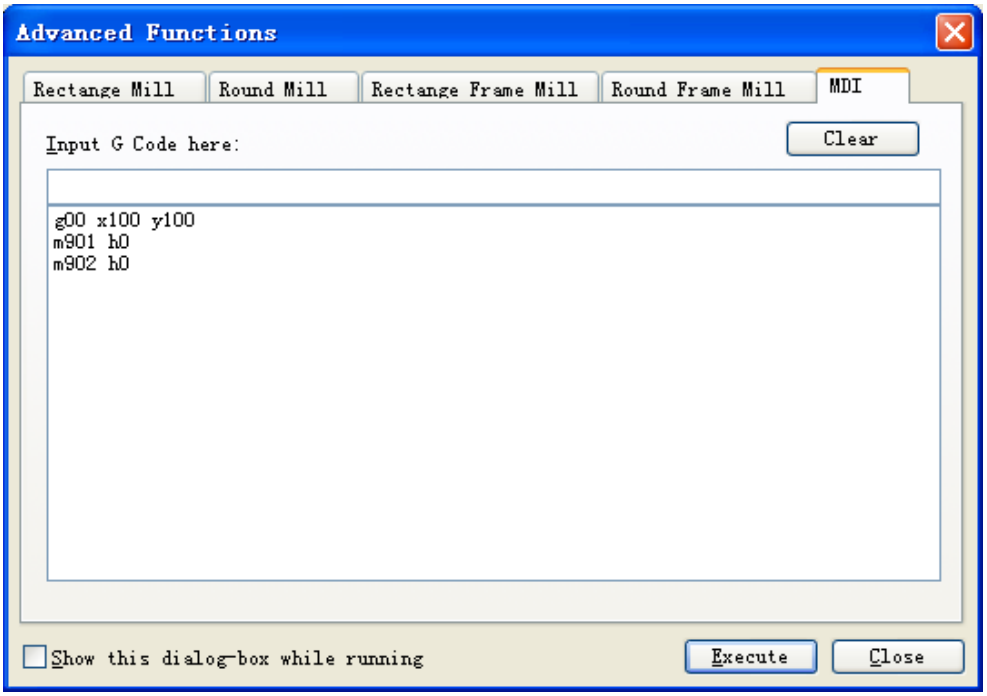


Fig. 5-19 MDI window

◆ **Jiggle**

This function is for fine tuning without stopping machining, available in the PAUSE state and in machining state. See Fig. 5-20.

The result of jiggle will only have an effect on the current machining task. It will become invalid when [Start] or [Resume] is executed after [Stop].

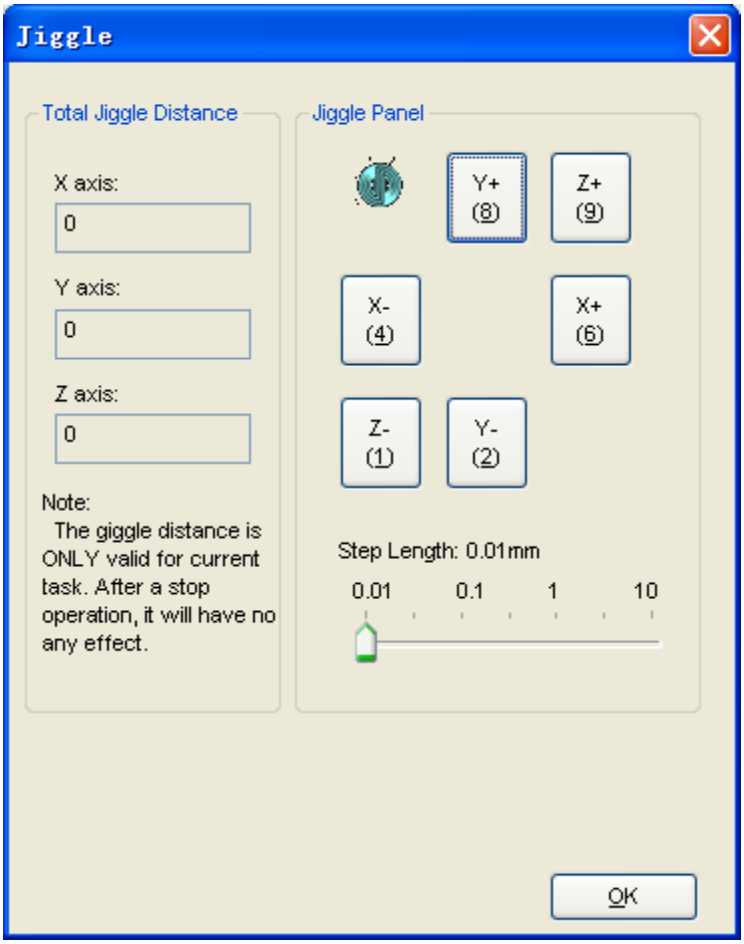


Fig. 5-20 Jiggle dialog box

◆ **Mobile Calibrator**

Choosing this menu item will open a dialog box, as shown below.

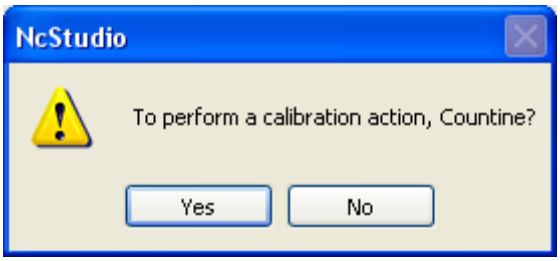


Fig. 5-21 Tool measurement

Choosing “Yes” will execute tool measurement. Refer to chapter 4.9.1 Calibrate the Workpiece Surface for detail.

Choosing “No” will cancel tool measurement.

◆ **Fixed Calibrator**

Choosing this menu item will open a dialog box, as illustrated below.

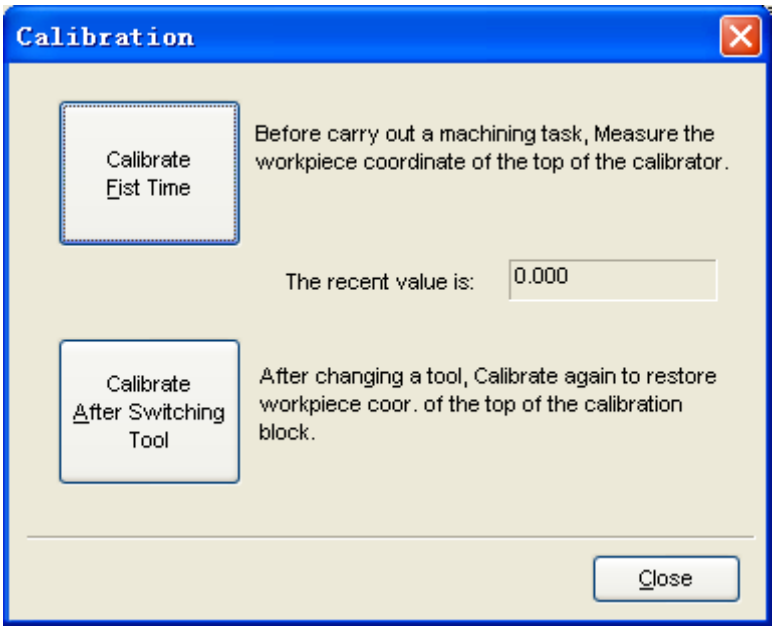


Fig. 5-22 Fixed measurement

Refer to chapter 4.9.2 for details. “The recent value” in the above picture is entered automatically by NcStudio.

CAUTION

Please note that items “Mobile Calibrator” and ”Fixed Calibrator” are absent in item list of “Operation” menu in double Z axes series software. You can turn to chapter 11.3.2 for details.

◆ **Back to Mechanical Origin**

Also called go home, back to machine origin, back to reference point or homing. Selecting this menu will open a dialog box, as shown in Fig. 5-23.

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

[Setting 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], [Y Axis] and [Z Axis]: homing the corresponding axis alone.

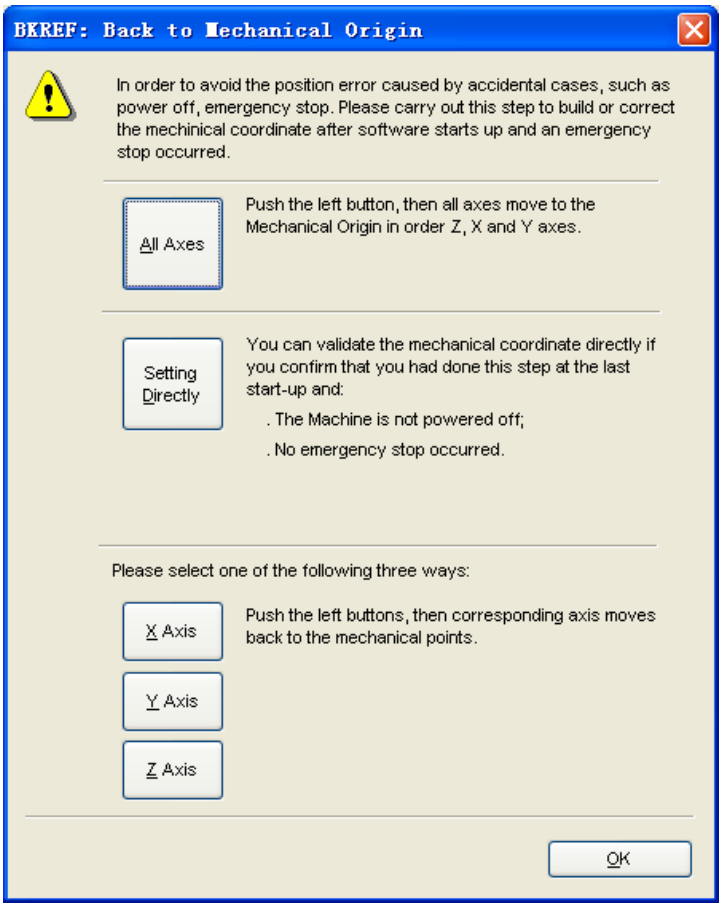


Fig. 5-23 Back to machine origin

Axis	M. Coor.	W. Coor.	Remained
X:	0.000	0.000	0.000
Y:	0.000	0.000	0.000
Z:	0.000	0.000	0.000

Fig. 5-24 NC state window

After all the axes have returned to machine origin, a mark "✥" will appear before each axis in the NC state window. There are three ways to open the dialog box:

One: upon startup of the software;

Two: selecting the item "Back to Mechanical Origin" folded in the "Operation" menu;

Three: pressing shortcut keys "Ctrl+Home".

◆ **Back to Fixed Point**

The machine tool will move to the fixed point (machine coordinates) automatically when this menu item is selected. The machine coordinates of fixed point should be set in a position facilitating fast tool change and workpiece replacement.

The machine coordinates of fixed point are determined by parameters [N4210], [N4211] and [N4212].

◆ **Disable Mechanical Limits**

See Fig. 5-25. In case of hard limit alarm, namely the machine limit being triggered, execute this function. The system will disable limit function and remove the alarm. At this time, move the machine tool away from the limit switch to a normal position via the manual buttons in the Manual window.

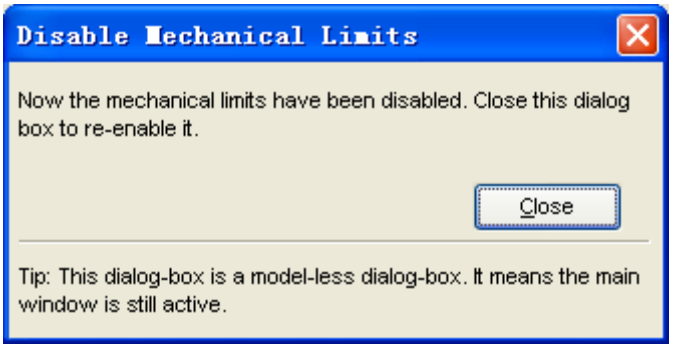


Fig. 5-25 Disable mechanical limits

CAUTION

Please pay close attention to the direction when moving the machine tool away from the limit switch to a normal position, or the machine tool may be damaged!

◆ **Alarm Reset**

When an alarm occurs, executing this function will restore the system to the “IDLE” state.

◆ **Scan Function**

With the menu selected, a dialog will pop up, as shown in Fig. 5-26.

This function is mainly for punching. Manually move X and Y axes to the target position, then click “Note” to record the workpiece coordinates, then to the next target position, then record…When finishing recording, click “”Save As” to generate a machining file. Since the generated file cannot be loaded into the system automatically, the user needs to load it into NcStudio before machining.

[Position of REFER Plane]: a plane where the punching speed is activated; the rapid traverse rate is active above this plane.

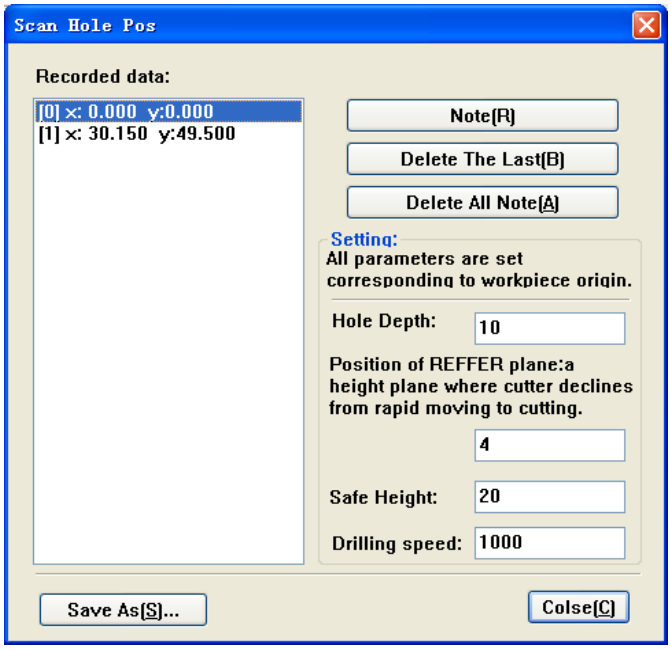


Fig. 5-26 Scan hole position

CAUTION

Please note that item "Scan Function" is absent in "Operation" menu in double Z axes software. You can turn to chapter 11.3.2 for details.

◆ Parameter Restore

NcStudio can backup parameters automatically. With this option chosen, a dialog box will pop up, as illustrated below.

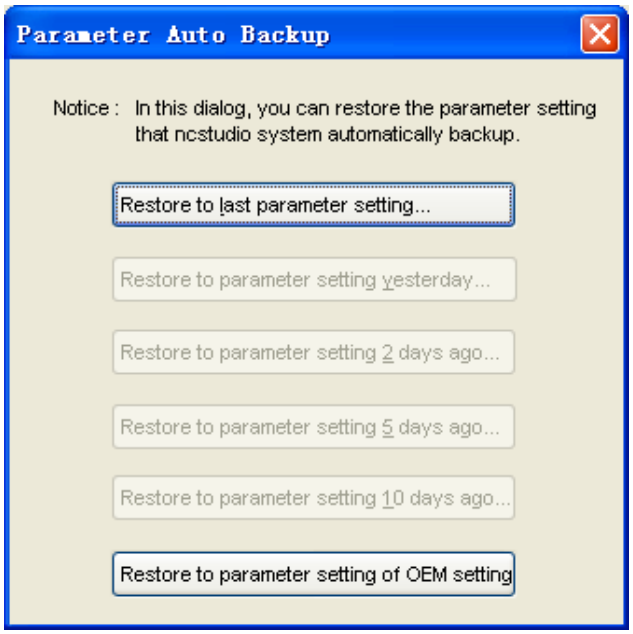


Fig. 5-27 Parameters auto backup

The system provides six kinds of parameter backups: "last parameter setting", "parameter setting yesterday", "parameter setting 2 days ago", "parameter setting 5 days ago", "parameter setting 10 days

ago" and "parameter setting of OEM setting". The user can select one of them according to actual situations.

After one of them is selected, NcStudio will exit automatically. At this time, the user needs to re-open it manually.

◆ **Set Parameters**

It is used to open the parameter window to set parameters. Refer to Chapter 6.3 for details.

5.5. "Machine" Menu

"Machine" menu includes items like "Turn on Spindle", "Turn on Coolant", "Turn on Light" and "Modify Tool No.", etc, as following.

◆ **Turn on Spindle**

It is used to turn on the spindle.

◆ **Turn on Coolant**

It is used to turn on the coolant pump.

◆ **Turn on Light**

It is used to turn on/off the working lamp on the machine tool.

◆ **Modify Tool No.**

Mainly used during tool changing, this function can guarantee the number of current spindle tool is the same as that of next tool (namely the tool to be changed to), so that the system can place the current tool to the correct position or call the right tool from the correct position in the tool magazine.

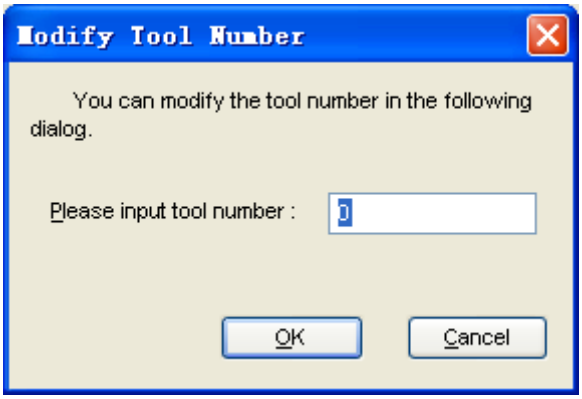


Fig. 5-28 Modify tool number

◆ **Feedrate**

The feedrate can be 0%, 10%, 20%, 50%, 90%, 100%, and 120%.

By choosing these items, the user can adjust the feedrate override to the optional percent, equaling to adjusting the feedrate slide block in NC state window. For details, refer to Sliding Block of Feedrate Override in chapter 4.6.2.

◆ **Efficiency/Quality Adjustment**

The option selected, a dialog box will pop up, as illustrated below:

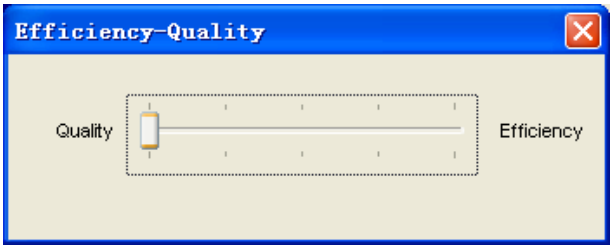


Fig. 5-29 Efficiency – Quality adjustment

Giving priority to quality will lead to high processing quality while to speed will result in high processing efficiency. Please balance the quality and efficiency as required.

◆ **Modify Menu**

Clicking this item will open the dialog below.

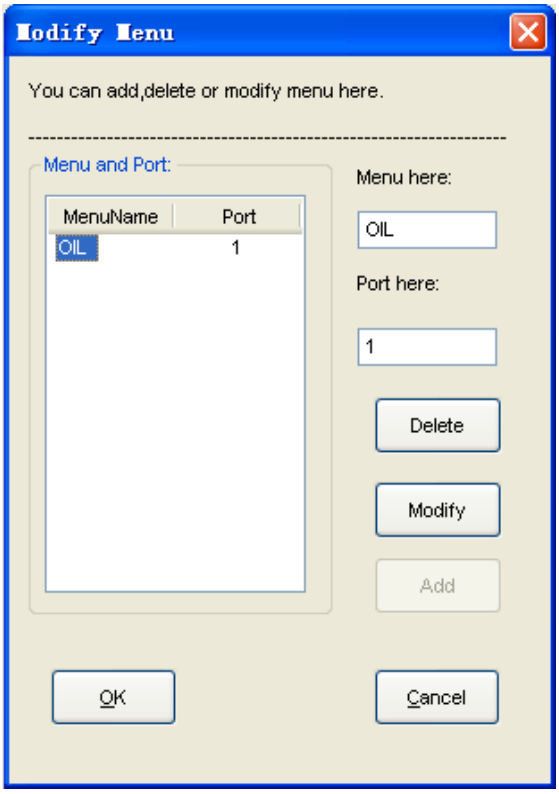


Fig. 5-30 Modify menu window

In this dialog, menu can be added, deleted and modified. Take adding menu as an example. First enter the menu to be added in “Menu here”, and then enter a reserved input port of terminal board in “Port here”. Please note that “Add” should be clicked before “OK”, or the port adding fails.

You can check the “Machine” menu to confirm the addition of the port. The following shows a successful operation.

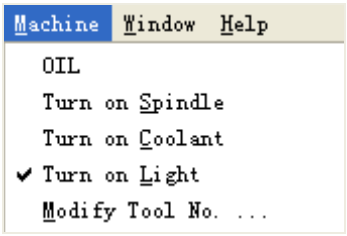


Fig. 5-31 Adding menu successfully

CAUTION

Menu items in “Machine” menu in double Z axes software differ slightly. You can turn to chapter 11.3.3 for details.

◆ **Port Setting of Origin/Limit Switches**

This submenu clicked, a dialog for password pops up. After the input, the setting dialog is entered, shown as Fig. 5-32.

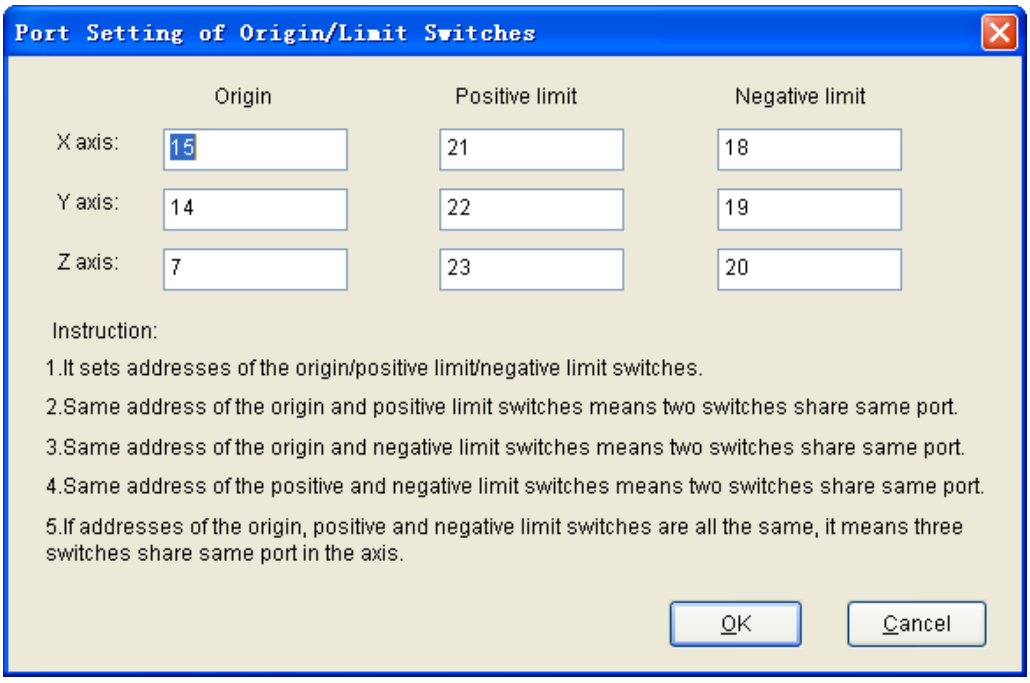


Fig. 5-32 Port setting of origin/limit switches dialog box

The following lists the operation instructions:

1) In this dialog box, you can set the addresses of the origin and the positive/negative limit switches;
2) When the addresses of the origin and positive limit are set the same in certain axis, origin and positive limit switches of this axis share one port;
3) When the addresses of the origin and negative limit are set the same in certain axis, origin and negative limit switches of this axis share one port;

4) When the addresses of the origin, positive limit and negative limit are set the same in certain axis, origin switch, positive switch and negative switch of this axis share one port;

5) When the addresses of positive limit and negative limit are set the same in certain axis, positive and negative switches of this axis share one port.

Port information can be modified in the software, as Fig. 5-33. Click [Modify Port Info] to eject the dialog box as Fig. 5-34, then press [Modify Des] to pop up the input box as Fig. 5-35. After input, click [F1 OK], port information modified successfully.

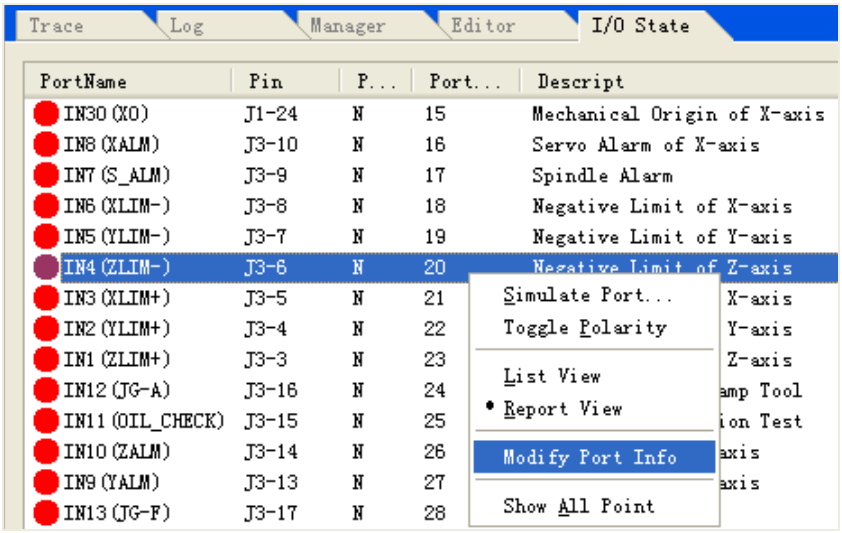


Fig. 5-33 Modify port information

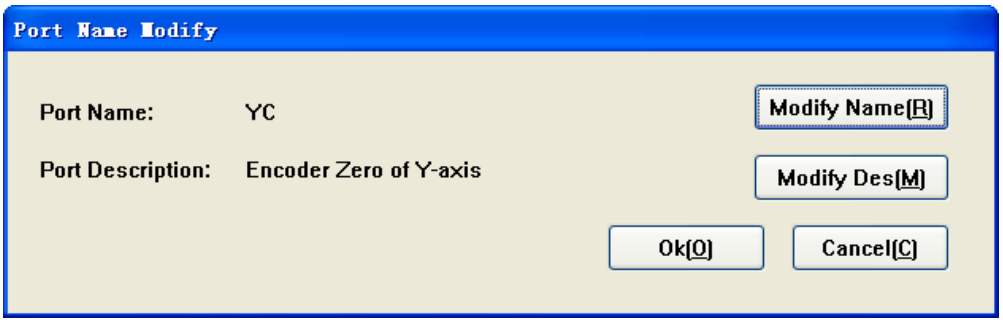


Fig. 5-34 Port name modify dialog box

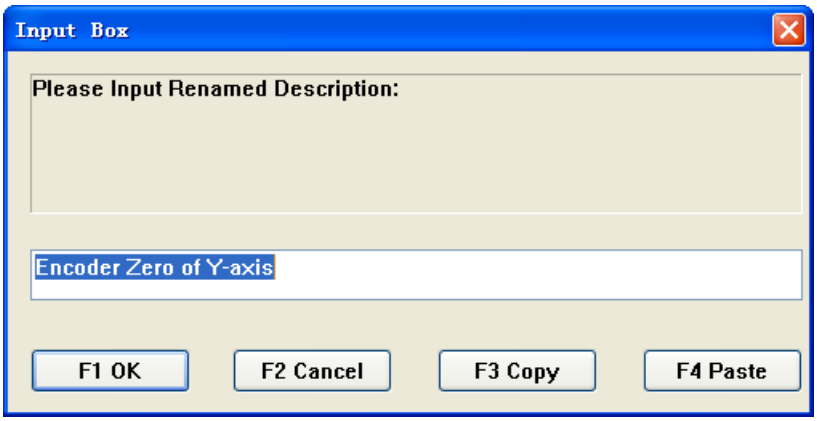


Fig. 5-35 Input box of renamed description

5.6. "Window" Menu

“Window” menu consists of multiple optional items, such as “Show Auto Window”, “Show Manu Window”, “Show Calibration Window” and “Show Trace Window”, and so on.

Choose items in this menu to switch among functional windows.

Program Lock: used to lock operation interface. No operation is allowed after the operation interface locked.

“Ctrl+Alt+L” is the shortcut key to lock the NcStudio operation interface, while “Ctrl+Alt+K” to unlock it.

5.7. "Help" Menu

“Help” menu includes the following menu items.

◆ **Tip of the Day**

With the item selected, a dialog box will pop up, as illustrated below, showing information and operation of NcStudio.

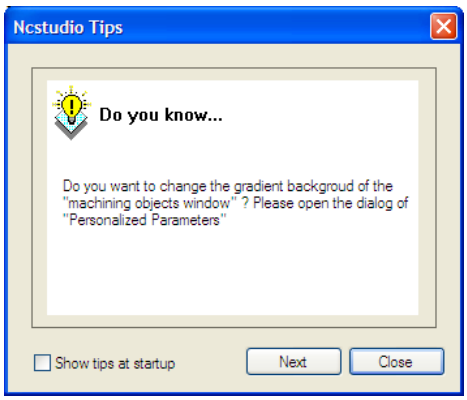


Fig. 5-36 Daily tip

◆ **Shortcut Keys List**

With the item chosen, a dialog box will pop up, displaying shortcut key information of NcStudio.

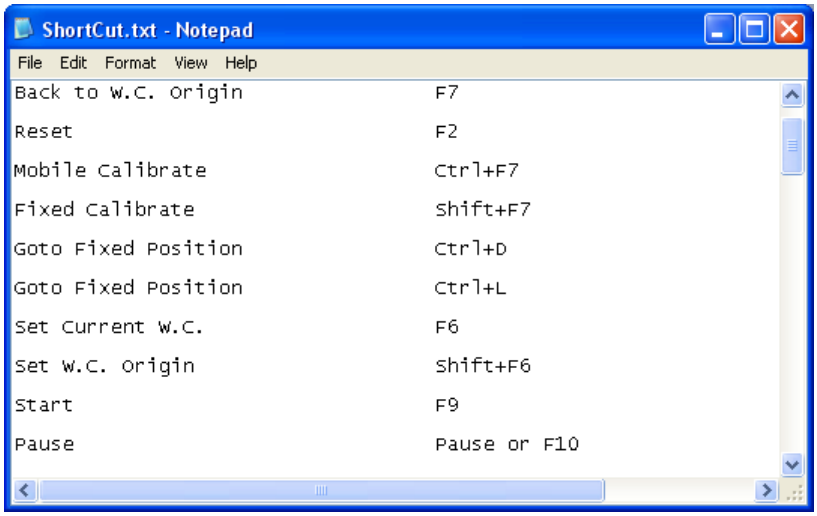


Fig. 5-37 Shortcut key list window

◆ About NcStudio

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

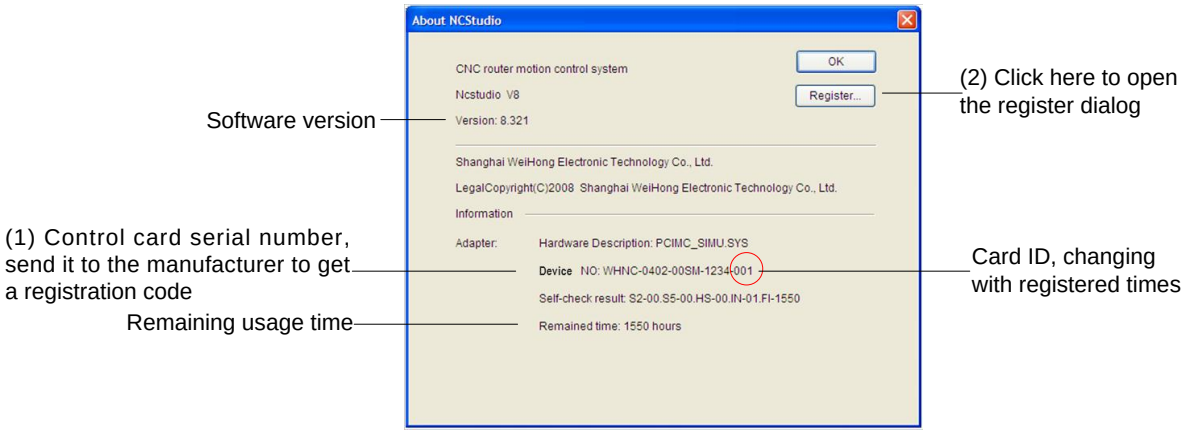


Fig. 5-38 "About NcStudio"

In the above dialog, the user can view such info as version, control 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. 5-39, input the received registration code, and click "OK" to finish registration.

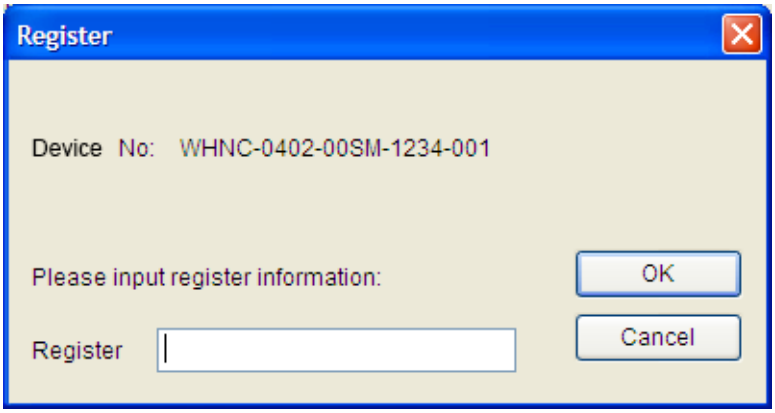


Fig. 5-39 Input registry code

CAUTION

The card ID changes with registration times, which can be told from the last three numbers of Control Card No. For instance, when registered times is 0, the last three numbers are 000; when the registration times is 1, the last three numbers will be 001, and so on.

◆ Visiting NcStudio Homepage

With this item selected, the user can visit the homepage of Weihong Electronic Technology Co., Ltd. to get informed of the latest products and relevant information about the company.

6. Parameter Setup

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/hand wheel. parameters, I/O address parameters, compensation parameters, reference point parameters, cutter parameters, other parameters and parameters overview.

6.1. Parameter Modification Permission

Parameters displayed are different under different permissions. Password "ncstudio" is required to access the manufacture's parameters.

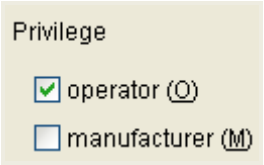


Fig. 6-1 Parameter permission

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

Fig. 6-2 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”.

Parameters cannot be modified in machining.

6.3. Operator’s Parameters

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

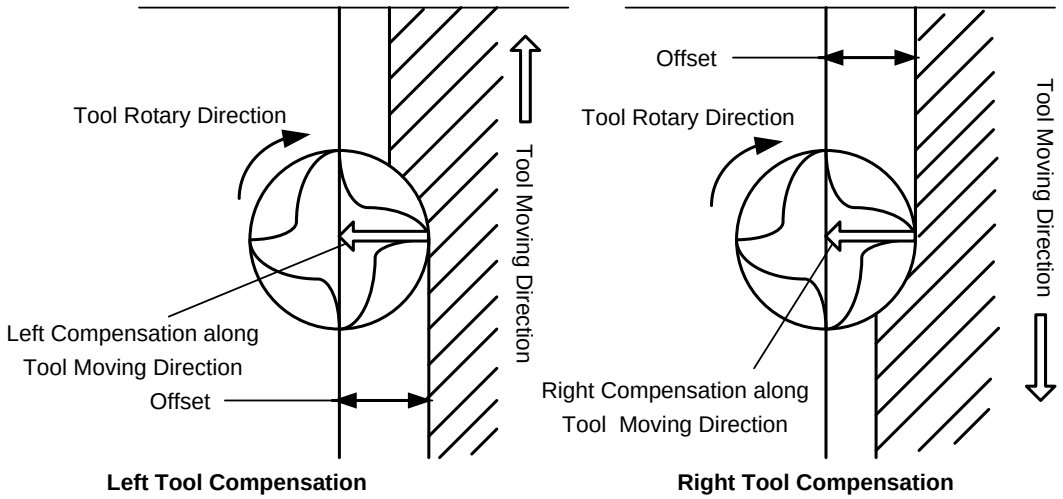
No.	Name	Setting Range	Default	Effective
Opera parameters				
N4025	NormalJogFeedrate	0～100000（mm/min）	1800	Immediately
	The moving speed of machine tool in manual mode when the feedrate override is 100%. The manual moving speed varies with the feedrate override.			
N4026	RapidJogFeedrate	0～100000（mm/min）	2400	Immediately
	The speed of machine tool when one of the Num direction keys and Ctrl are pressed simultaneously at 100% feedrate override.			
N4260	MaxJogFeedrateBefore-BKREF	0～100000（mm/min）	1200	Immediately
	The max. speed in jog mode before returning to the reference point.			
N4031	RapidTravelFeedrate	0～100000（mm/min）	3000	Immediately
	The default positioning speed (instead of machining speed).			
N4032	DefaultFeedrate	0～100000（mm/min）	1500	Immediately
	The default machining speed (instead of positioning speed).			
N4034	UseDefaultFeedrate	0: Not use 1: Use	0	Immediately
	If it is set as “1”, the speed specified in the machining file will become invalid.			
N4035	UseDefaultSpindleRev	0: Not use 1: Use	0	Immediately
	If it is set as “1”, the spindle speed specified in the machining file will become invalid.			
N4027	PauseDownSpeed	0～100000（mm/min）	600	Immediately
	Plunge rate of Z axis (backing to the cutting point before pause) in continuous machining after pause.			
N4028	PauseUpSpeed	0～100000（mm/min）	600	Immediately
	Tool lifting speed of Z axis at pause.			
N4049	ToolRasingHWorkCoorOn-Pause	0～9999（mm）	100	Immediately

No.	Name	Setting Range	Default	Effective
	The pause position of Z axis in machining.			
N4050	ToolRasingHeightOnPause	0～500（mm）	10	Immediately
	The lifting height of Z axis at pause with respect to cutting point.			
N4200 ～ N4202	FixedCalibratorBlockPosition (X/Y/Z)	-99999～99999（mm）	X: 0 Y: 0 Z: -1	Immediately
	X/Y: machine coordinates of X and Y when the tool nose reaching the range of tool measurement (the nearer to central point, the better). Z: machine coordinate of Z axis when the tool nose arriving at a certain height above the surface of tool sensor (running at G00 speed above this height while at measurement speed below this height).			
N4006	G73_G83SafeHeight	-99999～99999（mm）	0	Immediately
	Specifying the retract distance after each feed in the G73_G83 cycle drilling command.			
N4007	DirectionWhileFixedDrillStop	0: +X; 1: -X; 2: +Y; 3: -Y	0	Immediately
	Only valid in X-Y plane (G17).			
N4063	IJKIncrementModeValid	0: Invalid 1: Valid	1	Immediately
	0: the coordinates of circle center are with respect to workpiece origin. 1: the coordinates of circle center are with respect to the start point of machining arc.			
N1002	NeedConfirmClearWC	0: Invalid 1: Valid	1	Immediately
	Whether to give a prompt when clearing current workpiece coordinates, i.e. setting current point as workpiece origin, in order to avoid misoperation.			
N4005	ActionAfterProgramming	0: Keep still; 1: Back to fixed point; 2: Back to workpiece origin	0	Immediately
	0: Keep still; after machining ends normally, the spindle remains in the stop position. 1: Back to fixed point; after machining ends normally, the machine tool will go to the position set by parameters “N4210”, “N4211” and “N4212” (machine coordinates) automatically. 2: Back to workpiece origin; after machining ends normally, the machine tool will return to the current workpiece origin automatically.			
N4029	JiggleFeedrate	0～100000（mm/min）	60	Immediately
	The speed in jiggle.			
N4030	JiggleStepLength	0.01～0.5（mm）	0.01	Immediately
	The moving distance of machine tool in each jiggle.			
N4044	ZDownFeedrateOption	0～3	0	Immediately

No.	Name	Setting Range	Default	Effective
	0: No special treatment; parameter “N4045” is invalid. 1: N4045 enabled when only Z axis moves downward; feeding speed is decided by parameter N4045. 2: N4045 enabled when Z-axis downward movement included; when the Z axis moves downward (no matter whether X axis and Y axis move or not), feeding speed is decided by N4045. 3: Slowly regulate feed speed: from initial feedrate override to default feedrate override over the set time.			
N4045	Z_DownFeedrate	0～100000（mm/min）	480	Immediately
	Enabled when parameter N4044 set as “1” or “2”.			
N4051	SafeHeight	0.001～1000（mm）	10	Immediately
	Calculated with respect to workpiece origin; the system regards it is safe to move horizontally at this height; mainly used in “back to workpiece origin” and “breakpoint resume”.			
N4068	PauseAndPromptWhile-ChangeTools	0: false 1: true	false	Immediately
	Whether to pause and prompt the operator when the machine tool is changing tool.			
N4210 ～ N4212	FixedPointPosition (X/Y/Z)	-99999～99999（mm）	0	Immediately
	The machine coordinates of fixed point; used together with parameter N4005.			
N4070	SafeHeightAtG00Feedrate	0～99999（mm/min）	1	After re-loading
	The tool lifting height of Z axis in rapid traverse when machining a PLT file.			
N4071	PLTUnit	0.001 ～ 99999（mm/plu）	10	After re-loading
N4072	PLTToolsDistanceWhileProcessArea	0.001～99999（mm）	0.025	After re-loading
	The ToolsDistance (tool space) should be smaller than tool diameter when machining a PLT file.			
N4073	TwoDimensionalFileDepth	-99999～0（mm）	-1	After re-loading
	The machining depth of 2D PLT file.			
N4080	SafeHeightAtG00Feedrate	0～99999（mm/min）	1	After re-loading
	The tool lifting height of Z axis in rapid traverse when machining a DXF file.			
N4081	TwoDimensionalFileDepth	-99999～0（mm）	-1	After re-loading
	The machining depth of 2D DXF file.			
N4082	TheSourceOfMachineDepth	0, 1	0	After re-loading
	0: From parameter; 1: From DXF file			
N4083	DepthDelta	-99999～0（mm）	-1	After

No.	Name	Setting Range	Default	Effective
				re-loading
	The machining depth of 2D file each time in layer machining.			
N4084	UseFirstPointAsOriginInDXFFiles	0: Invalid 1: Valid	1	After re-loading
	0: set the origin in the DXF file as workpiece origin 1: set a user-defined point in the DXF file as workpiece origin. For instance, when drawing a picture in CAD, draw a point (better near or inside the picture). The system will treat this point as workpiece origin and not process it. If there are several points in the DXF file, the first drawn point will be set as workpiece origin.			
N4085	EnableMachineDividually	0: Invalid 1: Valid	0	After re-loading
	If set as “1”, the system will not process the next shape until finishing current one.			
N4086	MethodForDrillingDXFFiles	0, 1	0	After re-loading
	Hole machining mode in machining a DXF file. 0: Once done; 1: Reciprocating chip removal			
N4087	EachDepthForPolyDrill	0～99999（mm）	0.5	After re-loading
	Each drilling depth in reciprocating chip removal mode.			
N4089	SafeHeightAtG00Feedrate	0～99999（mm/min）	1	After re-loading
	The tool lifting height of Z axis in rapid traverse when machining an ENG file.			
N4090	PauseAndPromptWhileChangeTools	0, 1	1	After re-loading
	Whether to pause and prompt tool change when it is time for tool change when machining an ENG file. 0: Pause but not prompt; 1: Pause and prompt			
N4092	MachiningEngFileByToolNumber	0: Invalid 1: Valid	0	After re-loading
	When set as “1”, machining will be executed in terms of the specified tool number, and only the machining file corresponding to this tool will be processed.			
N4093	PauseTimeAfterEachCycle	0～99999（ms）	0	After re-loading
	The dwell time after each cycle in ENG file machining.			
N4094	DeepHoleMachiningWay	0, 1	0	After re-loading
	Deep hole machining mode. 0: Reciprocating chip removal; 1: High-speed reciprocating chip removal			
N4095	RETRACT_VALUE	0～99999（mm）	1	After re-loading
	Retract distance after each feed in high-speed reciprocating chip removal mode.			
N4096	MachiningEngFileModifyToolNumb	0: Invalid	0	After

No.	Name	Setting Range	Default	Effective
	er	1: Valid		re-loading
	With the function, tool number can be modified during machining.			
N4097	DepthToolSpeedWay	0: Machining speed way 1: Fast across move speed way	0	After re-loading
	In deep-hole drilling, the plunging speed of Z axis, 0 for machining federate, 1 for rapid traverse speed.			
N4098	ZUpTypeAfterDrill	0: Up to R-plane 1:Up to specified position	0	Immediately
	Lifting type of Z axis after single drilling. Please note that it is only effective for ENG file with version 5.5 or above.			
N4099	ZPosAfterDrill	-1000~1000 mm	10	Immediately
	When parameter N4098 set as 1, the workpiece coordinates of the fixed position predefined. Please note that the version of ENG file should be 5.5 or above.			
N4129	MachiningGFileByTool-Number	0: Invalid 1: Valid	0	After re-loading
	Whether to execute G-code file machining according to the specified tool number.			
N8020	Z_Axis_Stop_Options	0, 1, 2	0	Immediately
	The position of Z axis at pause. 0: To raise a given distance; 1: To specified workpiece coordinate; 2: To specified machine coordinate			
N8021	Z_Axis_Stop_Pos_in_MCS	-350～0	0	Immediately
	The machine coordinate of Z axis at pause, available when parameter N8020 is set as “2”.			
Axes parameters				
N1150 ～ N1152	CheckWorkCoordinateRange (X/Y/Z)	0: Invalid 1: Valid	0	Immediately
	If set as “1”, the system will check whether machining is within the workpiece coordinate range (upper limit and lower limit).			
N1160 ～ N1162	WorkCoordinateLowerLimit (X/Y/Z)	-99999～99999（mm）	-10000	Immediately
	The lower limit of workpiece coordinate in each axis.			
N1170 ～ N1172	WorkCoordinateUpperLimit (X/Y/Z)	-99999～99999（mm）	10000	Immediately
	The upper limit of workpiece coordinate in each axis.			
Spindle/H.W. parameters				
N0004	StopSpindleWhenFinish	0: Invalid 1: Valid	1	Immediately
	Whether the spindle stops rotating when machining stops.			

No.	Name	Setting Range	Default	Effective
N0005	StopSpindleWhilePauseStop	0: Invalid 1: Valid	1	Immediately
	Whether spindle stops rotating when machining pauses.			
Compensation parameters				
N3004	CutterCompensationValid	0: Invalid 1: Valid	0	Immediately
	Whether to enable tool compensation.			
N3005	CUTTER_COMPENSATION_DIRECTION	0, 1, 2	0	Immediately
	Specifying the direction of tool compensation. 0: No tool compensation; 1: Left tool compensation; 2: Right tool compensation  **Left Tool Compensation** **Right Tool Compensation**			
Ref. parameters				
N2001	MoveToMechanicalPointBeforeMachining	0: Invalid 1: Valid	1	Immediately
	When set as “1”, homing will be prompted and must be executed before each machining. When “0”, the system will not give a prompt and can execute machining directly.			
Cutter parameters				
N14900	MobileCalibratorBlockThickness	0～10000（mm）	20	Immediately
	The max. tool length.			
N14901	ToleranceOfToolBroken	0～100（mm）	0.5	Immediately
	The allowable tolerance when checking broken tools.			
N15020	Name	Max 127 letters.	-	Immediately
N15021	Diameter	（mm）	0	Immediately
	Tool diameter.			
N15022	Length	（mm）	0	Immediately
	Tool length.			
N15023	DiameterFray	（mm）	0	Immediately
	Wear loss of tool diameter.			

No.	Name	Setting Range	Default	Effective
N15024	LengthFray	（mm）	0	Immediately
	Wear loss of tool length.			
N15030 ～ N15032	CutterPositionOffset (X/Y/Z)	（mm）	0	Immediately
	The offset between current tool and the first tool.			
The system supports 255 tools at most. The above parameters are information of the first tool and only for reference.				
Other parameters				
N0020	WhetherUseWorkCoorShow-Trace	0: Invalid 1: Valid	0	Immediately
	Whether to show the machining trace by workpiece coordinates in Trace window.			
N7003	AutoBackToZero	0: Invalid 1: Valid	0	After restart
	Whether to go home automatically after starting the system each time.			
N7016	MachTaskPauseInformType	0: No red light on; 1: Red light on for 3 seconds; 2: Red light on until any input from mouse or keyboard to make yellow light on	0	Immediately
N7017	MachTaskEndInformType	0, 1, 2	0	Immediately
	0: Red light off; 1: Red light on for 3s; 2: Red light on until there is mouse or keyboard input to make yellow light on			
N7302	NotToCoolantWhenStartMachining	0: Invalid 1: Valid	1	Immediately
	When machining task begins, whether to turn on the coolant automatically.			
N7303	NotToClooantWhenFinish-Machining	0: Invalid 1: Valid	1	Immediately
	When machining task ends, whether to turn off the coolant automatically.			
N7312	DelayTimeForCloseAirValve	0～100000（ms）	1000	Immediately
	The delay time for tool change.			
The above parameters are ones shared by both general three axes software and double Z axes software.				
Exclusive Parameters in Double Z Axes Software				
N7123	StartSpindleAfterChange-Tools	0: Invalid 1: Valid	0	Immediately
	Whether to turn on the spindle after tool change. It is only effective in alternative configuration. The spindle cannot be started until tool change is finished.			
N7124	SwitchToZ1AfterTaskEnd	0: Invalid	0	Immediately

No.	Name	Setting Range	Default	Effective
		1: Valid		
	Whether to switch to Z1 axis after task ends, only valid in alternative configuration.			
N7125	StopSpindleWhenChange-Tools	0: Invalid 1: Valid	1	Immediately
	Whether to turn off the spindle during tool change, only valid in alternative configuration.			
N7126	MoveToOriginPositionAfter-ChangeTools	0: Invalid 1: Valid	1	Immediately
	Whether the tool moves to the previous position (workpiece coordinate before tool change started) after tool change completed, only valid in alternative configuration.			
N7127	TwoSpindleUseDifference-WorkpieceOrigin	0: Invalid 1: Valid	0	Immediately
	Whether the two spindles share the same workpiece origin, only valid in alternative configuration.			

7. Operation Steps

After the motion control 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

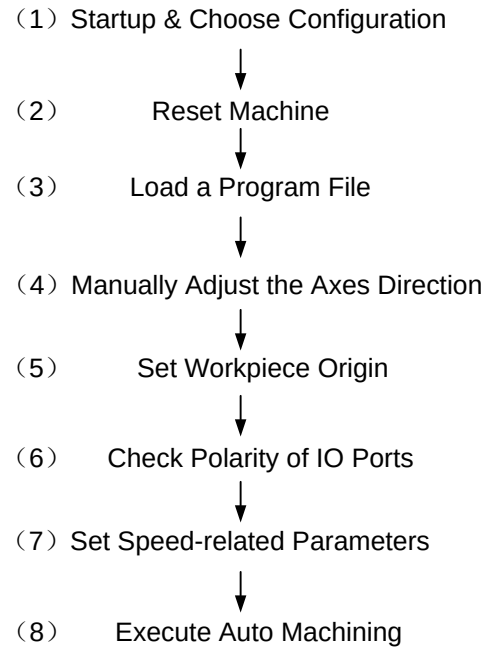


Fig. 7-1 Basic operation flowchart

7.1. Start-up

1) Before booting up the computer, first make sure that the machine tool has been well connected to the computer properly. Then power on the machine tool and the computer.

2) Double-click the shortcut icon 

on the desktop to open the software, or you can open it by clicking the icon in the list of *Start-All Programs* of the computer. If the software runs for the first time, a dialog box will pop up, asking you to choose a configuration before opening the software, as shown below. You need to configure the software according to the actual condition of your machine tool.

Fig. 7-2 Select a configuration

3) Select a desired configuration among the pull-down configuration list, and click [Yes] to activate and go ahead; click [No] to cancel your choice and back to the configuration list and re-select one.

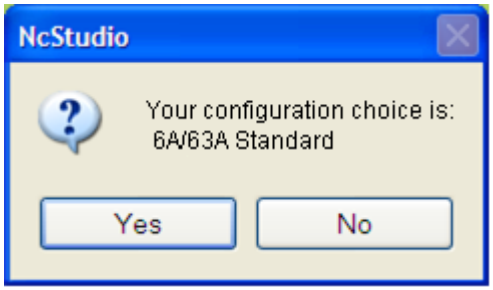


Fig. 7-3 Confirm your choice

4) A prompt dialog box will pop up, showing that “The operation is successful”. Click [OK] to finish. The software will be started automatically. Please note that configuration selection only appear during the first startup, to put it in other words, you can open the software directly by double clicking the shortcut icon on the desktop any time after the first startup.

7.2. Machine Reset

This section is provided for machine tools with the function of backing to machine origin.

For machine tools supporting “back to machine origin” function, choosing “back to mechanical origin” menu item will return the machine tool to the mechanical origin automatically and reset 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..

Additionally, you can skip this operation if you are definitely secure that there is no problem with the current position.

7.3. 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.

7.4. 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 Z axis

1------Negative direction of Z axis

Pressing any one of the above keys and the number key 0 or Ctrl simultaneously will make the machine tool move at rapid jog speed.

7.5. Set Workpiece Origin

The workpiece origin is defined as the coordinate origin of X, Y and Z in the machining file. Before machining starts, the workpiece origin should be fixed first. 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 “*W.Coor*.” buttons (X, Y) in the “NC State” 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 Z axis measurement (calibration) function.

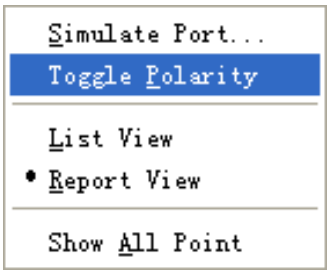
After all these operations, the workpiece origin is set.

7.6. 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 4.15. The modification of port polarity takes effect after the software being re-started.

To invert the polarity of a port:

Firstly locate and select the target port, then right click the mouse with keys “Ctrl+Alt+Shift” pressed at the same time, a dialog box as shown on the right side to pop up, and then select “Toggle Polarity”.



7.7. Set Speed Parameters

Apart from feederate (G01 speed) and rapid traverse speed (G00 speed), the system provides speed-related parameters like axial linear acceleration, connection acceleration at corners, the maximum speed for reference circle as well as the minimum speed for reference circle. The default setting is only for general situation yet cannot secure the performance and outcome. Therefore, the user needs to adjust settings of those parameters according to the conditions of machine tool as well as the requirement, to yield the best and satisfactory result.

7.7.1. LinearAcceleration (N4053)

It is used to describe the acceleration / deceleration ability of a single axis, with unit mm/s^2. The value is determined by the physical characteristic of machine tool, such as quality of movement part, torque, resistance, cutting load of feed-motor, and so on. The larger the value is, the less time spent in the process of acceleration / deceleration will be, and the higher the efficiency will be. Generally, for servo motor systems, the value is between 400 and 1200. Set a small value at the beginning; make the machine tool perform various typical movements for a period of time, and carefully observe it; if there is no abnormal situation, increase the value gradually; otherwise, decrease the value and reserve 50% ~ 100% insurance allowance.

7.7.2. ConnectionAcceleration (N4054)

It is used to describe the acceleration/deceleration ability in synchronized motion of multiple feeding axes, with unit mm/s^2. The value limits the maximum speed of machine tool in circular movement. The larger this value is, the higher the maximum allowable speed on circular movement of machine tool will be. Generally, for servo motor systems, the value is between 1000 and 5000; for heavy machine tools,

the value should be smaller. Set a small value at the beginning; make the machine tool perform various typical movements for a period of time, and carefully observe it; if there is no abnormal situation, increase the value gradually; otherwise, decrease the value and reserve 50% ~ 100% insurance allowance.

7.7.3.ReferenceCircleMaxSpeed (N4058)

Maximum speed of reference circle (Reference Circle Max Speed) corresponds to arc speed limit function. When a machine tool processes an arc, it will vibrate due to centripetal force. To reduce this kind of vibration, the software limits machining speed during machining an arc in terms of centripetal acceleration. Take default setting as an example, the maximum line velocity of the reference circle (Diameter: 10mm) is 1800mm/min.

The formula to calculate centripetal acceleration is as following:

$$a = \frac{v^2}{r}$$

Among the formula, $r = (10/2)mm$; $v = 1800mm/\min$;

Thus centripetal acceleration a can be calculated; when other arcs are processed, this centripetal acceleration is the maximum allowable centripetal acceleration. Arc speed will be limited, if it is too large causing centripetal acceleration larger than a calculated in this formula.

7.7.4.MinSpeedInACircularMotion (N4059)

From the above formula, it can be seen the centripetal acceleration a is not the only factor to determine arc speed limit.

$$a = \frac{v^2}{r}$$

According to the formula, in processing an arc with small radius, the line speed will be limited, so will the processing speed. To improve machining efficiency, the software provides this parameter to ensure the machining speed will be larger than the value of this parameter regardless of the radius.

Usually, given the drive ability of servo motor, frication of machine assembly, and endurance capacity of mechanical components, the maximum speed of the three axes in actual machining can be limited by modifying the manufacturer's parameter "Axis Max Feedrate" (N4250~N4252).

End-users can also adjust machining speed in operator's parameters according to actual conditions. For details, please refer to the chapter 6.3.

7.8. Execute Auto Machining

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

◆ **Start Auto Machining**

Select the menu item [Operation| Start] or click  on the toolbar 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 [Operation| Stop]; clicking  on the toolbar; 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 [Operation| Pause]; clicking  on the toolbar; clicking shortcut key F10. To continue machining, choose [Operation| Start] or click  on the tool bar or press F9.

◆ **Advanced Start**

Also known as program block skip execution. With the menu item [Advanced Start] selected, a dialog box will pop up. Select the starting and end program line No. to define the program block to be executed. Refer to “Advanced Start” in chapter 5.4 “Operation” Menu for details.

8. Precautions in Operation

8.1. 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.

8.2. 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) As the limit (machine origin) signal is still on, port alarm may occur;

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

3) 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.

9. Operation of Software with Multi-tool

For complex machining, multi-tool is always in need and tool library is added to the machine tools. Accordingly, NcStudio XYZ type provides the line-tool configuration and disk-tool configuration, which will be introduced in details as below.

9.1. Line-tool Config

9.1.1. Related Parameters Setting

After the [6A/63A-Standard-Line-Tool-ex7 Config] is chosen under [File] menu, you can choose [ToolLibParameterSet] under [Operation] menu. Its interface is shown as Fig. 9-1.

Through this dialog box, you can set line-tool related parameters, including outport address, inport address, library installed position and so on.

1) Outport/inport address setting: you can set related parameters of outport/inport address. When the value of one of the parameters is set -1, its corresponding port will be disabled.
2) Tool library out back, library installed position and speed setting: you should set them based on actual situation.
3) Ahead point setting: the following operation assumes the tool library parallel to X axis. When ahead point is valid, you need to set the machine origin of ahead point. Before tool change, the spindle is lifted to the up position at the X-axis coordinate of the current tool position and Y-axis coordinate of the ahead position. When ahead point is invalid, however, the spindle rises to the up position and directly moves to the current tool position to execute tool change.
4) Up down position setting: it is set invalid by default and the input of up position suffices. Otherwise, you need to set the down position as well.
5) Tool position setting: you should first choose a tool No. and then click [Set current tool pos] to obtain current position for the selected tool.

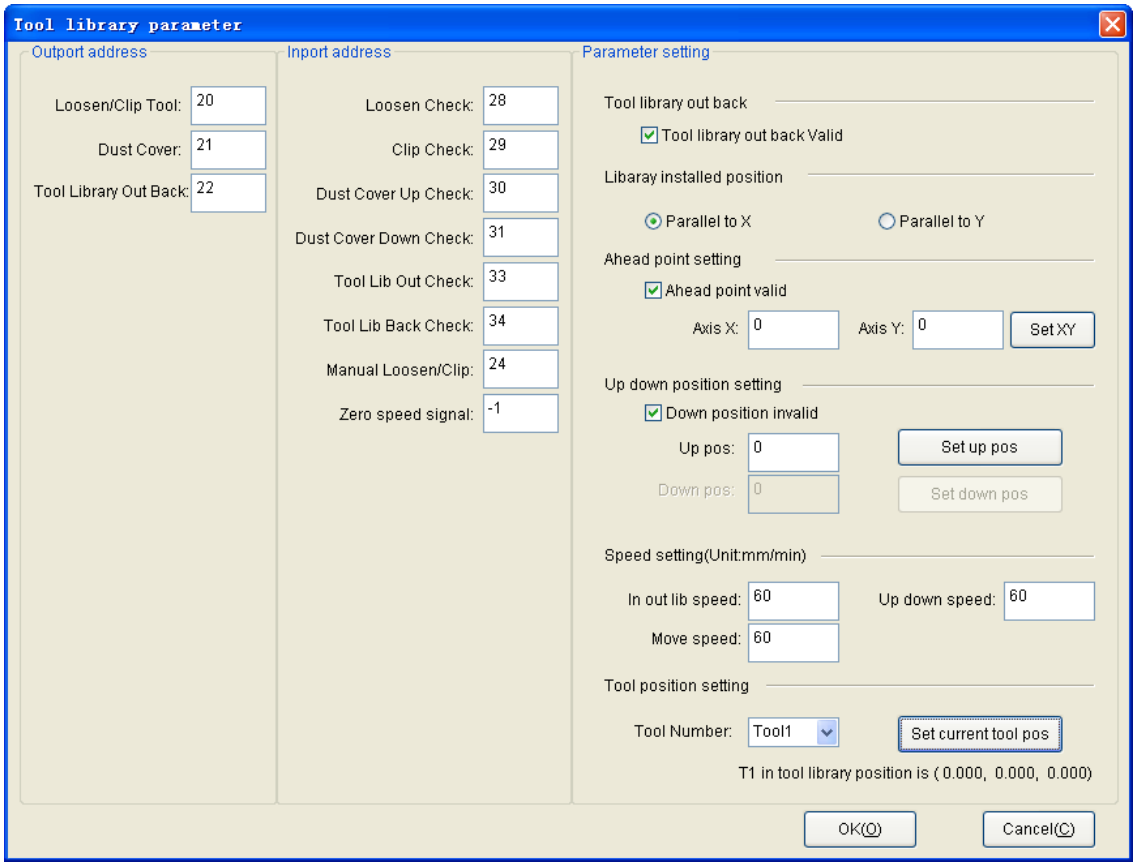


Fig. 9-1 Tool library parameters of line-tool

9.1.2.Function Introduction

Under the [6A/63A-Standard-Line-Tool-ex7 Config], you can operate the tool library in the manual window, as Fig. 9-2:

◆ **Tool length offset setting**

After modifying active tool, you can click [change tool] to execute this function;

1) Single measurement: it specifies the measurement on the current tool.
2) Multi-measurement: it specifies the measurement on multi-tools. When [Multi-Meas] is clicked, the multi-tool measure manage dialog box pops up, shown as Fig. 9-3. You can select all the tools you want to measure and click [start measure]. Measurement sequence: it is suggested to measure the current toll before the next one, sticking to the order of small to large successively.
3) Manual set: it specifies setting the current tool length directly.

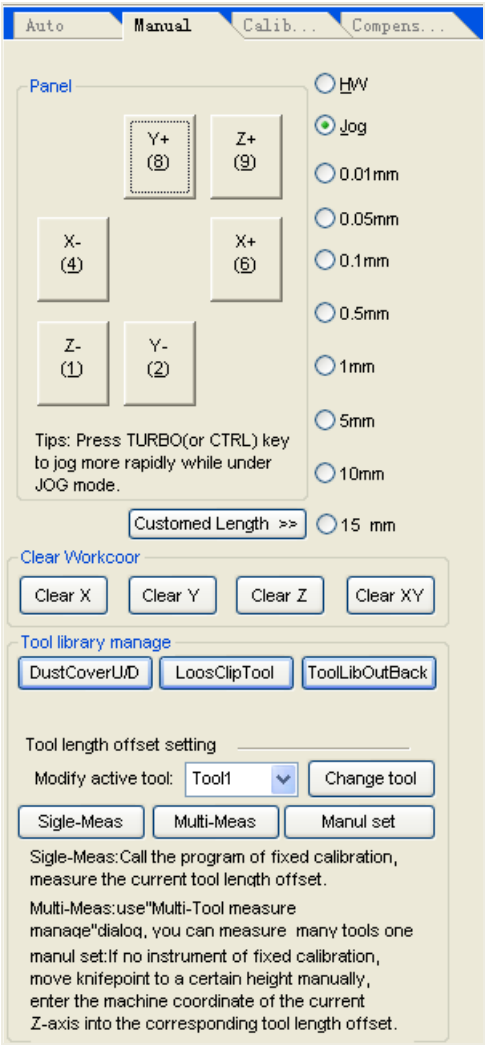


Fig. 9-2 Manual function window of line-tool

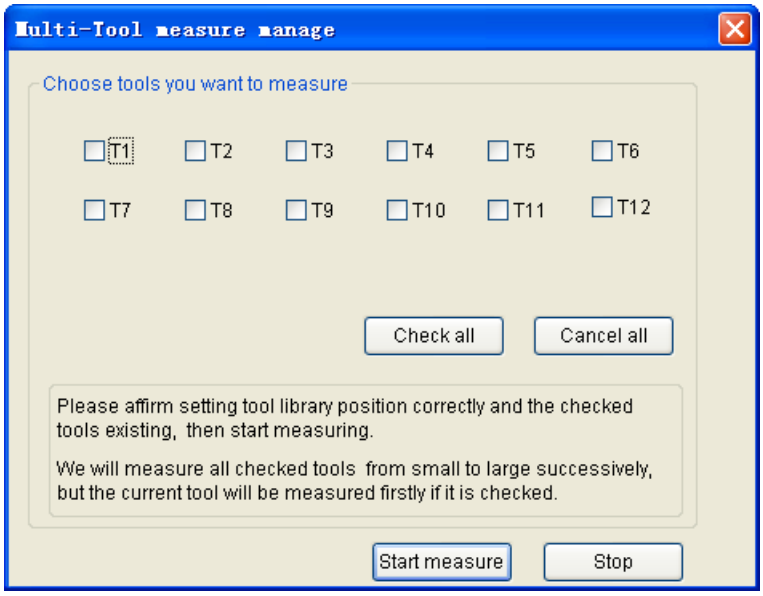


Fig. 9-3 Multi-tool measure manage of line-tool

9.2. Disk-tool Config

9.2.1. Related Parameters Setting

After the [6A/63A-Standard-Disk-Tool-ex7 Config] is chosen under [File] menu, you can choose [ToolLibParameterSet] under [Operation] menu. Its interface is shown as Fig. 9-4.

Through this dialog box, you can set line-tool related parameters, including outport address, inport address, library installed position and so on.

1) Outport/inport address setting: you can set related parameters of outport/inport address. When the value of the parameters is set -1, corresponding port will be disabled.
2) Library installed position and speed setting: you should set them based on actual conditions of tool library.
3) Up down position setting: up position and down position should be set together. Taking tool placement operation as the example, the spindle first moves to the down position, and after tool disk is ejected and tool is unclamped, it then rises to the up position.

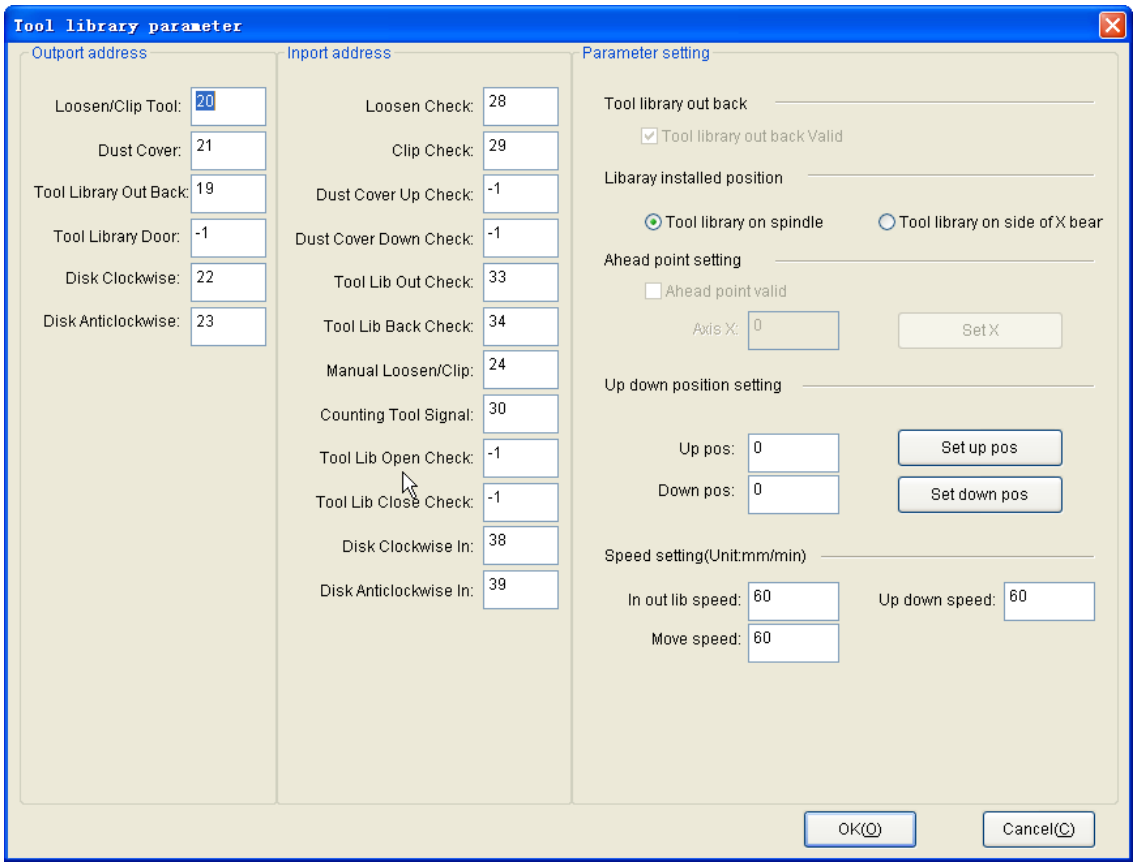


Fig. 9-4 Tool library parameters of disk-tool

9.2.2.Function Introduction

Under the [6A/63A-Standard-Line-Tool-ex7 Config], you can operate the tool library in the manual window, as Fig. 9-5:

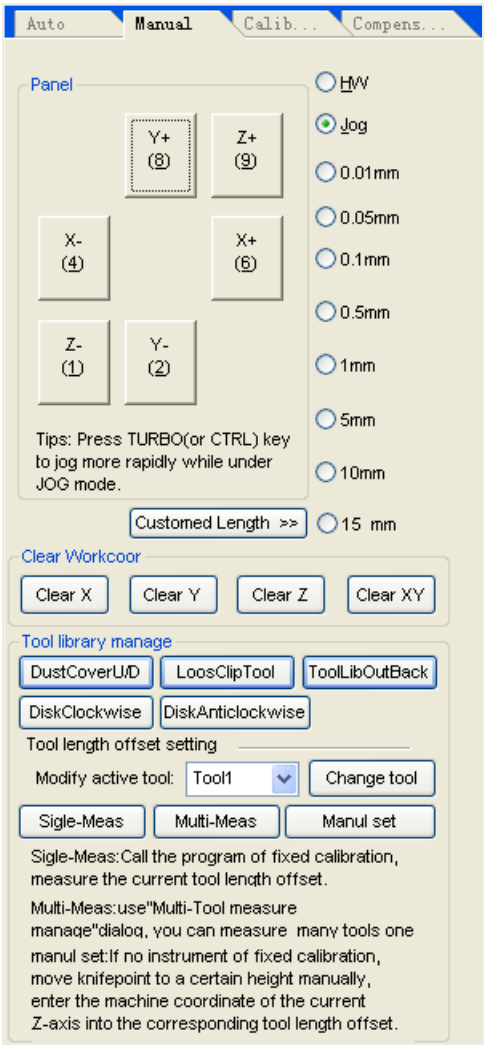


Fig. 9-5 Manual function window of disk-tool

◆ **Tool length offset setting**

After modifying active tool, you can click [change tool] to execute this function;

1) Single measurement: it specifies the measurement on the current tool.
2) Multi-measurement: it specifies the measurement on multi-tools. When [Multi-Meas] is clicked, the multi-tool measure manage dialog box pops up, shown as Fig. 9-6. You can select all the tools you want to measure and click [start measure]. Measurement sequence: it is suggested to measure the current toll before the next one, sticking to the order of small to large successively.
3) Manual set: it specifies setting the current tool length directly.

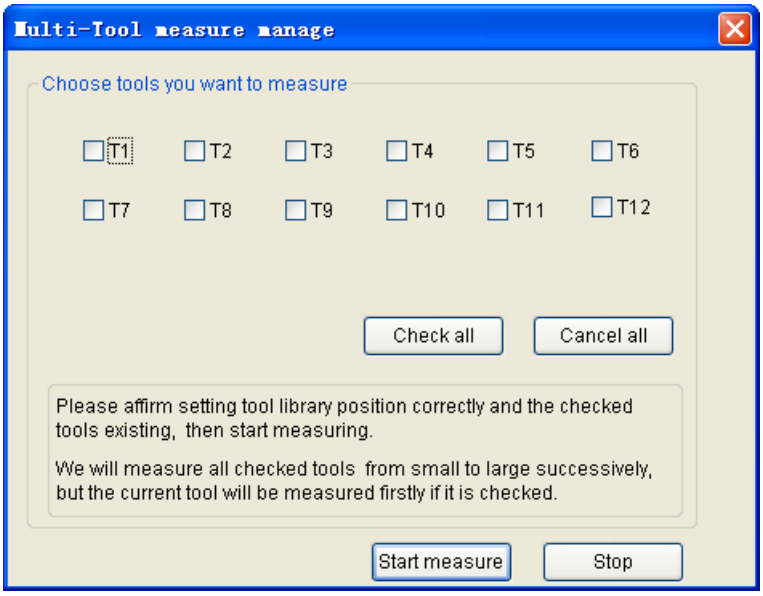


Fig. 9-6 Multi-tool measure manage of disk-tool

10. Operation of Software with Multi-cylinder

According to the number of cylinder and inverter, the multi-cylinder software can be divided into configurations of 2-cylinder-1-inverter, 2-cylinder-2-inverter, 3-cylinder-1-inverter, 3-cylinder-3-inverter, 4-cylinder-1-inverter and 4-cylinder-4-inverter. This chapter is the introduction to the software with multi-cylinder; the user needs to pay attention to the difference between it and general three axes software.

Cylinder settings can be done in Manual window, as shown below:

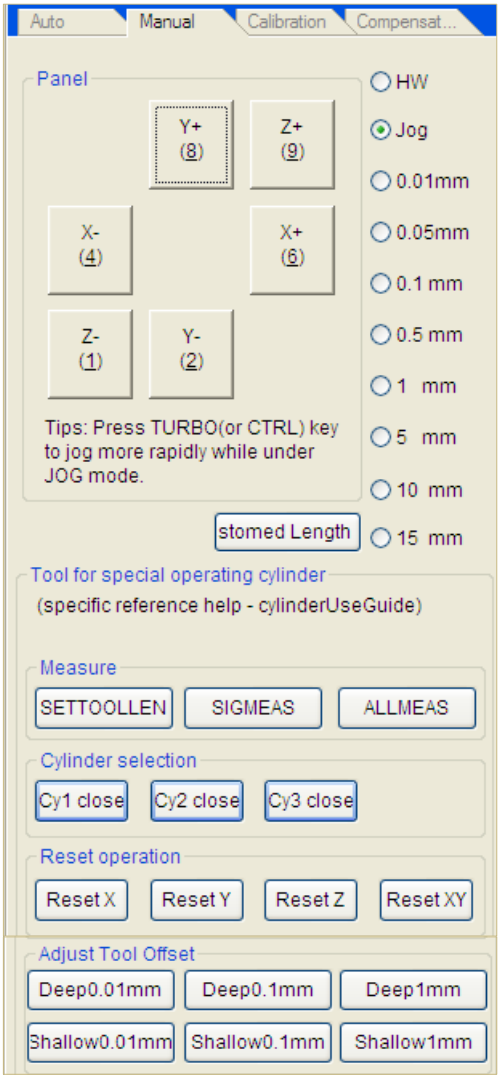


Fig. 10-1 Manual window

10.1. Measurement

◆ **Set Tool Length**

Manually move the tool to a fixed position, click SETTOOLLEN button to set the machine coordinate of Z axis into the tool offset in Z direction.

◆ **Single Measure**

To measure tool offset for one tool at one time. Namely, SIGMEAS button clicked once, measurement of one tool will be conducted. With this function activated, subroutines for fixed calibration will be called, and the calibration result will be set into the tool offset in Z direction.

During calibration, the user does not need to manually move the tool to the fixed position of tool sensor. The system automatically moves the Z axis downward and performs calibration according to the number of active tool, and set the result into the tool offset at the end of calibration. To finish the process, the Z axis retracts 10 mm relative to the ending position and the cylinder will be pulled back.

◆ **Measure All**

To measure the offsets for tools one after one. Namely, ALLMEAS button clicked once, measurement of tools will be conducted one after one till all tools are calibrated. Similar to single measurement, subroutines for fixed calibration will be called during measurement, and the result will be set into the tool offset in Z direction. The calibrating orders vary due to different configurations and different active tool number.

- For configuration with two cylinders:

The spindle tool being T1, the calibrating sequence is 1-2; the spindle tool being T2, the sequence is 2-1.

- For configuration with three cylinders:

The spindle tool being 1, the calibrating sequence is 1-2-3; the spindle tool being 2, the sequence is 2-3-1; the spindle tool being 3, the sequence will be 3-2-1.

- For configuration with four cylinders:

No matter what the tool number of the spindle tool is, the calibrating sequence remains the same, namely, 1-2-3-4.

◆ **Process when Measuring All (Taking 3-cylinder-1-inverter configuration as an example)**

1) Assuming the spindle tool is T1, after ALLMEAS button clicked, cylinder 1 opens, and the Z axis moves to the pre-defined fixed position (decided by parameter “FixedCalibratorBlockPosition”) at

rapid traverse speed, then heads towards the tool sensor surface at presetting speed (decided by parameter “CALI_SPEED”) to find tool calibrating signal, with calibrating times being 1.

2) After calibrating, the result is entered into the parameter of tool position offset automatically. Z axis moves to the upper position (decided by parameter “CTUP”); cylinder 1 turns off while cylinder 2 turns on.

3) The X and Y axes move by corresponding offset (XY machine coordinates=the current machine coordinates + tool position offset of T2). The Z axis moves to the fixed position at rapid traverse speed.

4) The Z axis moves at presetting speed to find calibrating signal, with calibrating times being 1; at the end of it, the result is recorded into the parameter of tool position offset automatically. The Z axis moves to the upper position.

5) Cylinder 2 turns off and cylinder 3 turns on. The X and Y axes move by corresponding offset (XY machine coordinates=the current machine coordinates + tool position offset of T3). The Z axis moves to the fixed position at rapid traverse speed.

6) The Z axis moves at presetting speed to find calibrating signal, with calibrating times being 1. After calibrating, the result is recorded into the parameter of tool position offset automatically. The Z axis moves to the upper position.

7) The calibration ends, and the current cylinder does not turn off. The system prompts “All measurement of tool length complete”. Click [OK] to finish the measurement.

CAUTION

Functions “Set Tool Length”, “Single Measure” and “Measure All” are available only in IDLE state and the tool number being a specific number, namely, the tool number being 1 or 2 or 3 or 4 at one time.
In the process of “Single Measure” and “Measure All”, clicking ■ button or shortcut key F2 / F11 will terminate the calibration.

10.2. Select Cylinders

1) For 2-cylinder-1-inverter, 3-cylinder-1-invereter and 4-cylinder-1-inverter software, buttons of cylinder selection are interlocked, namely, clicking one button will open the cylinder and changing the tool number accordingly while making the rest button pop-up; clicking it again will make the cylinder closed and the button pop-up In other words, when one cylinder is opened (the button with raised appearance), the other cylinders will be closed (the buttons with sunken appearance).

2) For 2-cylinder-2-inverter, 3-cylinder-3-invereter and 4-cylinder-4-inverter software, all buttons of cylinder selection are non-interference. Both alternative and linkage controls can be realized. For

example: cylinder 1 is opened (the button being held down) and the tool number is 1, if at this time, cylinder 2 is opened (the button being held down), the cylinder 2 will be opened with cylinder 1 still being opened, while the tool number will be 12. Clicking the button of cylinder 1 again will make it pop-up, cylinder 1 pulled back and tool number changed to 2, and so on.

CAUTION

All cylinder selection buttons are available only in IDLE status. Please note that the system will lift the tool to the position "CTUP" first before switching among cylinders. Delay in tool change is effective only when encountering T command in machining.

10.3. Clear

Manually move the spindle to the desired origin position; click Reset X Reset Y Reset Z to set the position as the origin in X\Y\Z axis separately; clicking Reset XY will set the origin in XY axes together.

The Clear operation in Z axis can also be done via calibration. Manually move the spindle to the desired position, click [Measure Workpiece Surface] in Calibration window.

10.4. Fine Tune the Tool Position Offset

This function aims to fine tuning the tool position offsets. As shown below, click the button to make corresponding adjustments.

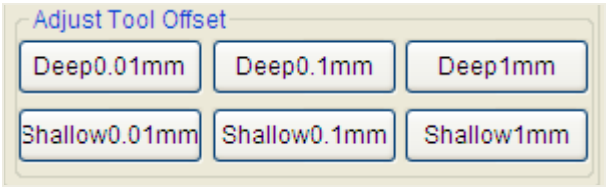


Fig. 10-2 Adjust tool offset

CAUTION

Before adjusting tool offsets, please make sure that the cylinder is open.

11. Operation of Double Z Software

Operations of double Z axes software are similar with those of general three axes software in most occasions. The most obvious difference is that double Z axes software includes four axes, namely X/Y/Z1/Z2, while general three axes software includes three axes, namely X/Y/Z axes. The following parts will illustrate the differences of double Z axes software from general three axes software.

11.1. CNC State Window

Under the NC Information Bar is the NC State window, which can be divided into five areas according to their functions. From the left to the right, they are Current position area, Feedrate area, Spindle speed area, Command set and machining cycle area as well as Tool info area.

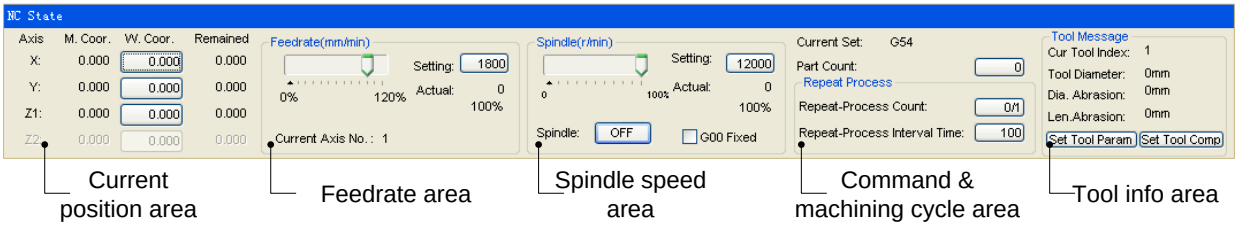


Fig. 11-1 NC State window of double Z axes software

◆ **Current Position Area**

Different from that of general three axes software, the double Z axes software involves four axes, namely X, Y, Z1, and Z2 axes. Therefore, specific position information of four axes will be displayed here.

There are items “Select Z1”, “Select Z2” and “Select Z1+Z2” in the menu list of Machine. The system chooses “Select Z1” by default. You can make your own choice and the active axes will be displayed in the current position area, while inactive axis will be displayed in grey. For example, the following two pictures correspond to the selection of “Select Z2” and “Select Z1+Z2” separately.

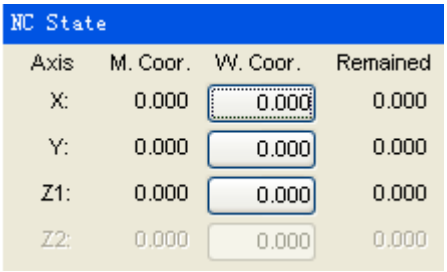


Fig. 11-2 Select Z2

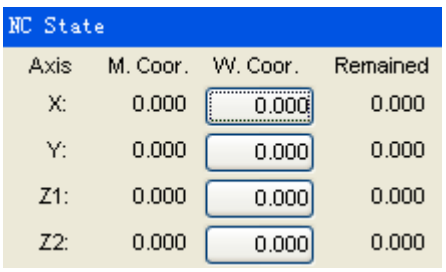


Fig. 11-3 Select Z1+Z2

11.2. NcStudio Operation Windows

The different window is “Calibration Operation Window”. This window is for tool calibration or conducting tool presetting operation. Please note that it is much different from that of general axes software. The calibration window of double Z axes software is shown as below:

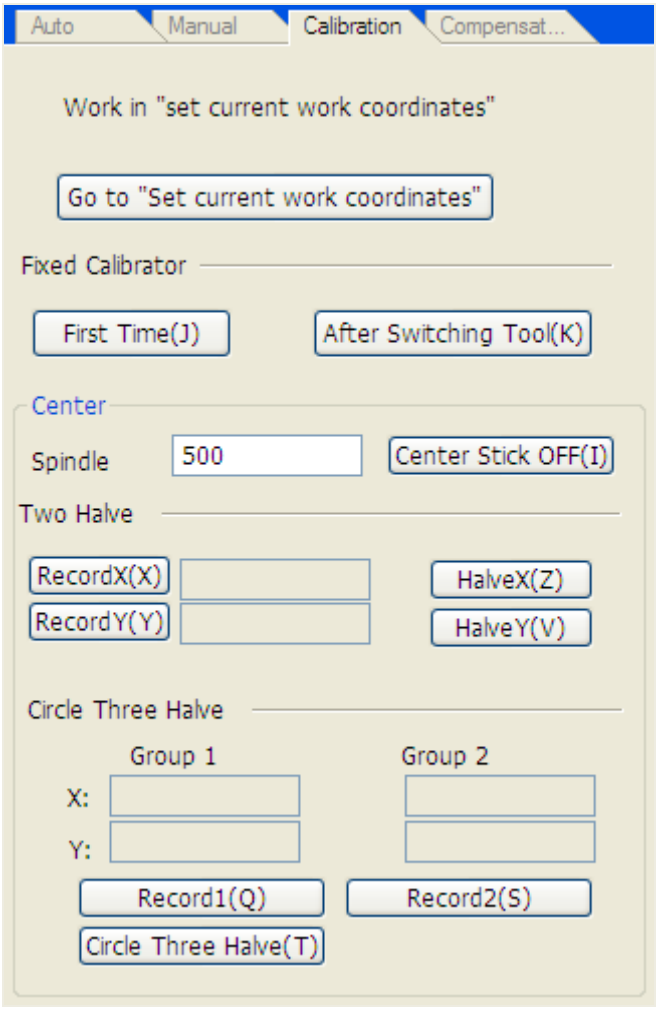


Fig. 11-4 Calibration window in double Z axes software

◆ **[Go to “Set current work coordinates”] Button**

Click this button or right-click at any position in NC State Window to open a dialog box, as illustrated below:

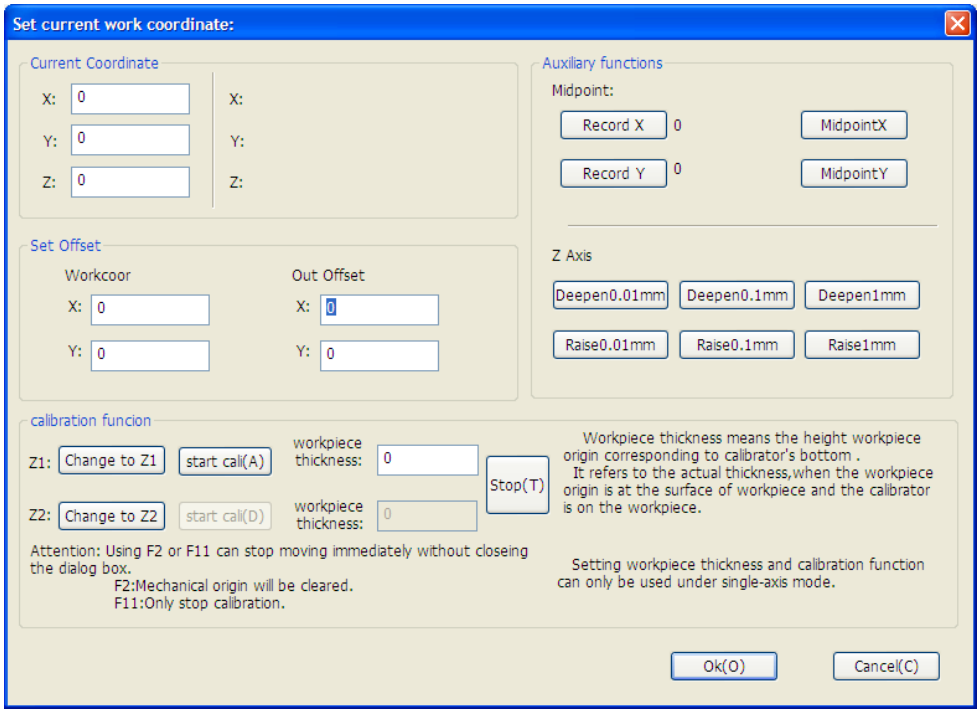


Fig. 11-5 Current workpiece coordinate setting

◆ **Current Coordinate:**

It displays the current workpiece coordinates. The user can modify the value directly here, and each modification will be followed by a prompt, as illustrated below:

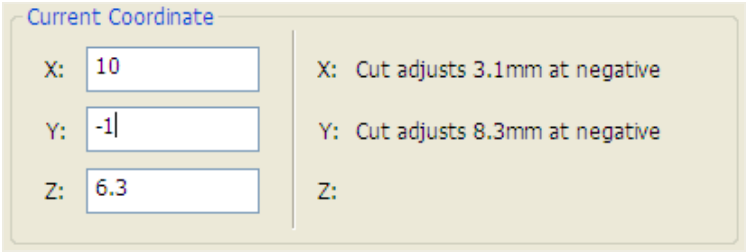


Fig. 11-6 Current coordinates setting

◆ **Workpiece offset:**

Also called part offset. Offset X/Y represents the offset of workpiece origin with respect to the machine origin. In other words, it is the machine coordinates of X/Y axis when tool nose moves to the workpiece origin. In this system, only G54 WCS is supported.

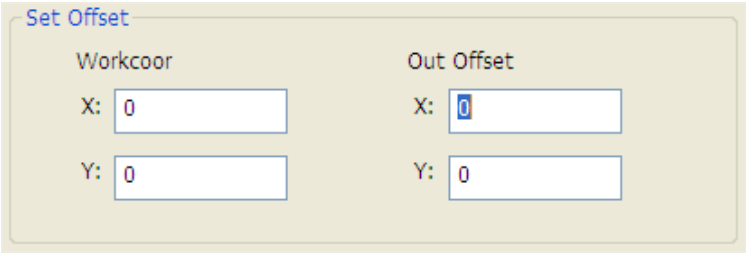


Fig. 11-7 Offset setting

External offset, also known as public offset, records the temporary adjusted value of workpiece origin. This value can only be adjusted manually and cannot be changed by any auto function. This means that the value will not be changed and maintained during fixed calibration or mobile calibration. If the external offset is not 0 before calibrating, the workpiece coordinate will not be 0 either after calibrating. For example, if the external offset is 10 before calibrating, this value will remain 10 after calibrating and the workpiece coordinate will be -10.

◆ **Z Axis**

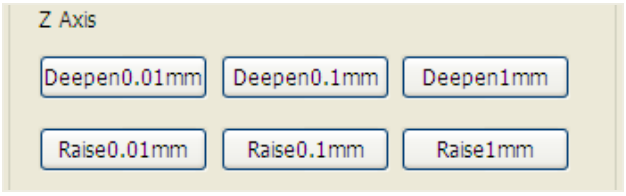


Fig. 11-8 Z axis adjustment

Clicking one of the buttons will make the Z axis workpiece origin move up or down a fixed distance to form a new WCS; clicking the button repeatedly will make the Z axis workpiece origin move the accumulated distance.

◆ **Calibration Function**

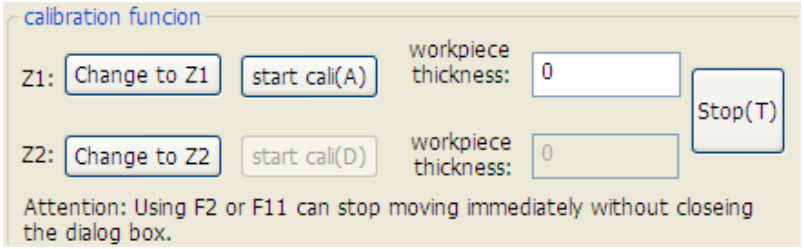
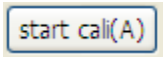


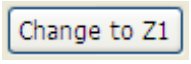
Fig. 11-9 Calibration function

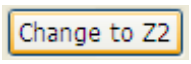
Workpiece thickness: the height from the workpiece surface (Z axis workpiece origin) to the workbench surface (normally≥0).

Input a value into the box 

and click 

button to make the system start calibrating after finding the calibration signal.

You can calibrate the tools in Z1 and Z2 axis respectively by repeating the above operation. Click button 

or 

to switch between the two Z axes.

CAUTION The above operation is mobile calibration, you can refer to 4.9.1 for more details.

11.3. NcStudio Menu System

11.3.1. “File” Menu

◆ **Configuration**

The configuration of double Z axes software is determined by the combination of rotary table and controlling pattern. Namely, it can be divided into four types, linkage configuration without rotary table, alternative configuration without rotary table, linkage configuration with rotary table and alternative configuration with rotary table, as you can see below. Choose the desired configuration according to the machine conditions. It is much different from the counterpart of general three axes software, which should be pay attention to. You can refer to the introduction in chapter 5.1.

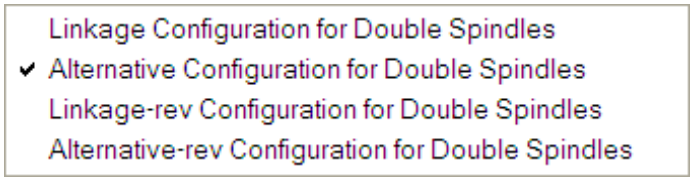


Fig. 11-10 Configuration options

◆ **Linkage Configuration for Double Spindles**

Linkage configuration without rotary table. With the configuration active, when switching between the Z1 and Z2 axes, the system will move the current axis to the safety position “ZMAX-1” (parameter N1172 “ZWorkCoordinateUpperLimit”), while keeping the target axis unmoved. For example, the current axis being Z1 and the target axis Z2, when switching from Z1 to Z2, the system will move Z1 axis to the safety position first before activating Z2 axis.

◆ **Alternative Configuration for Double Spindles**

Alternative configuration without rotary table. With the configuration active, when switching between Z1 and Z2 axes, similar to the linkage configuration, the system will move the current axis to the safety position first before activating the target axis.

◆ **Linkage-rev Configuration for Double Spindles**

Linkage configuration with rotary table. A rotary table is added based on the linkage configuration, with Y axis as the revolving axis.

◆ **Alternative-rev Configuration for Double Spindles**

Alternative configuration with rotary table. A rotary table is added based on the alternative configuration, with Y axis as the revolving axis.

CAUTION

The configuration is pre-set by the machine tool builder and beyond modification by the end-users. Otherwise, possible machine failure may occur.

11.3.2. “Operation” Menu

The item list under “Operation” menu in double Z axes software is shown as below:

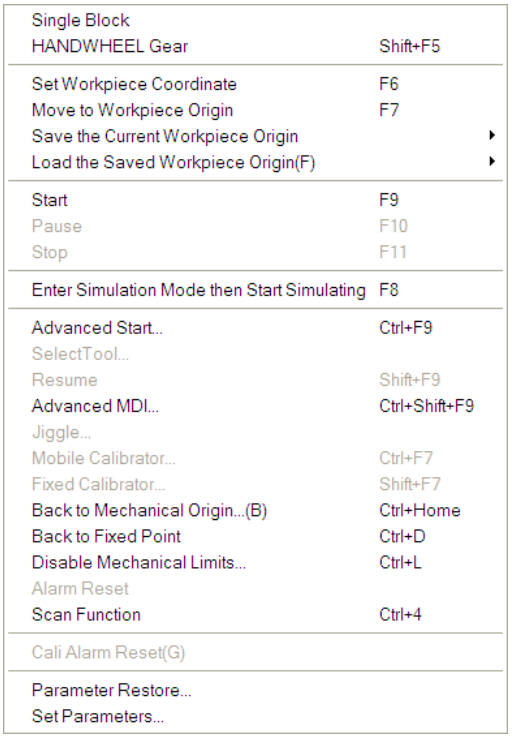


Fig. 11-11 Operation menu

◆ **Set Workpiece Coordinates**

With the item selected, a dialog box titled “Set current work coordinate” will pop up, as shown below, where you can set the current position as the workpiece origin, adjust the offset in the Z axis, and conduct centering operation, etc. please refer to chapter 11.2 for details.

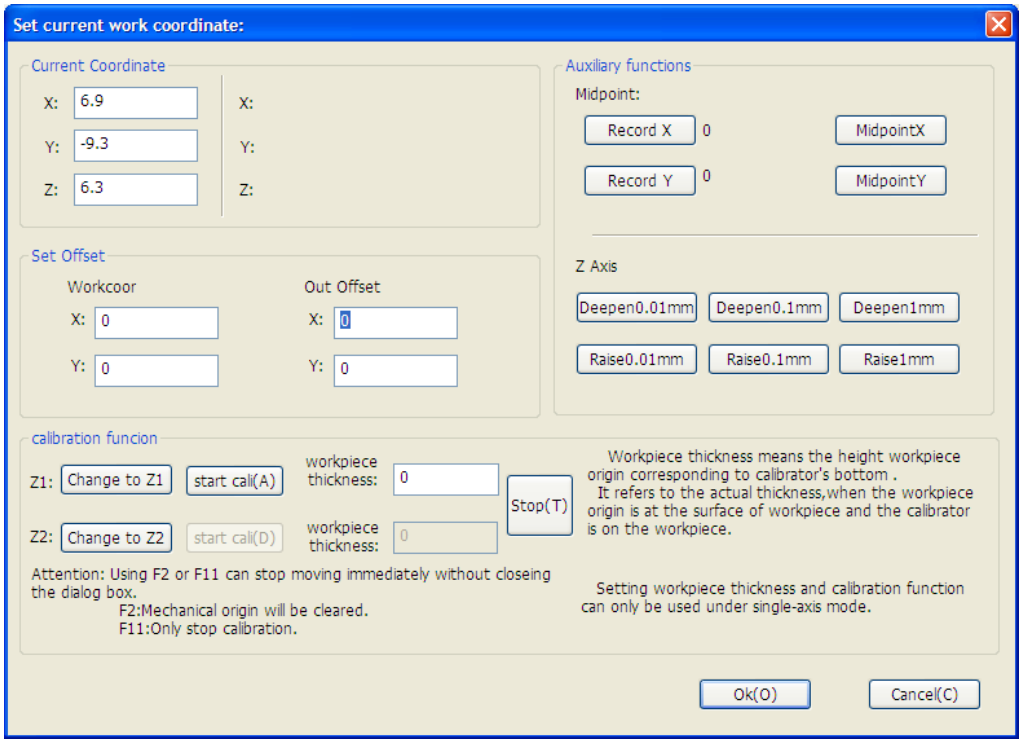


Fig. 11-12 Set work coordinates

11.3.3. “Machine” Menu

The item list under “Machine” menu is shown as below:

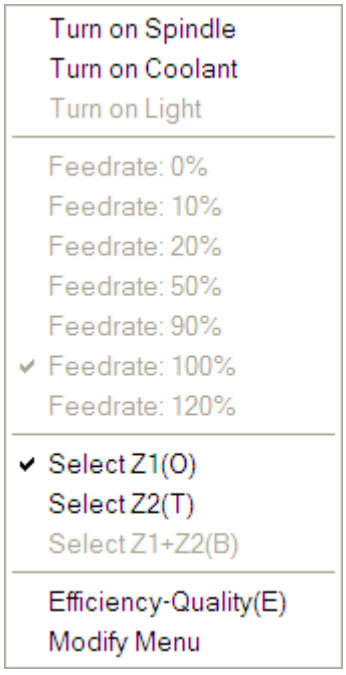


Fig. 11-13 Machine menu

◆ **Select Z1, Select Z2, Select Z1+Z2**

There are four available axes in double Z axes software, as implied by the name. There are three kinds of enabled axes, that is, enabled axes XYZ1, enabled axes XYZ2 and enabled axes XYZ1Z2. Besides,

“Select Z1+Z2” item is exclusively effective under linkage configurations. When single Z axis is active, only one and the selected Z axis can move; while when dual Z axes are active, the two axes can move simultaneously and share the same actions.

◆ **Linkage Configuration for Double Spindles**

Under this configuration, the machine tool will not move when the system is switching between [Select Z1] and [Select Z2]. That is, the Z1 and Z2 axes will maintain their respective original height.

When switched to [Select Z1+Z2], the machine tool may generate motion if the workpiece coordinates of Z1 axis and Z2 axis do not coincide. Otherwise, the machine tool will keep still. When the machine tool ceases moving, the workpiece coordinates of Z1 and Z2 axes will be the same.

◆ **Alternative Configuration for Double Spindles**

Under this configuration, item [Select Z1+Z2] is unavailable, shown in grey.

When item [Select Z1] or [Select Z1] is chosen, the current axis will return to the machine origin but the target axis will not move. For example, the current axis being Z1 and the selected axis being Z2, the Z2 axis will be not activated until the Z1 axis returns to the machine origin successfully.

Under this configuration, the machine tool will generate the following actions when executing T code (e.g. change tool from T1 to T2):

1) Stop Z1 spindle;
2) Raise Z1 to the machine origin;
3) Disable Z1 axis;
4) Select and enable Z2 axis;
5) Move the Z2 axis to the previous workpiece coordinate position of Z1 before tool change.

No matter under linkage configuration or alternative configuration, the current workpiece coordinates can be cleared only in single-Z-axis mode. When double Z axes are both active, such operation is forbidden.

12. Shortcut Keys List

Here are shortcut keys for NcStudio V8 software.

Shortcut Key	Function	Shortcut Key	Function
Global shortcut keys			
Esc	Switch among function windows	Tab	Switch among controls
Ctrl+1	Activate *Auto* window	Ctrl+2 /ScrLk	Activate *Manual* window
Ctrl+3	Activate *Calibration* window	Ctrl+4	Enable Scan Function
Ctrl+F1	Switch to Z1 axis	Ctrl+F2	Switch to Z2 axis
Ctrl+F7	Mobile calibration	Ctrl+F9	Advanced start
Ctrl+Home	Return to the reference point	Ctrl+Enter	Full screen
Ctrl+Tab	Switch among the folding windows	Ctrl+Del	Clear trace
Ctrl+Shift+F9	Execute MDI(Manual Direct Input) instructions	Ctrl+A	Select all
Ctrl+C	Copy	Ctrl+E	Open and edit
Ctrl+F	Search/Find	Ctrl+H	Replace
Ctrl+I	Show machining info	Ctrl+N	New
Ctrl+O	Open and load	Ctrl+P	Edit
Ctrl+S	Save	Ctrl+U	Unload
Ctrl+V	Paste	Ctrl+X	Cut
Ctrl+Z	Undo	ALT+1/F4	Activate *Trace* window
ALT+2	Activate *Log* window	ALT+3	Activate *Manager* window
ALT+4	Activate *Editor* window	ALT+5	Activate *I/O state* window
F3	Find next	F5	Direct positioning
F6	Set workpiece coordinates of the current point	F7	Return to the WCS Zero
F8	Enter (Exit) simulation	F9	Start
F10/Pause Break	Pause	F11	Stop
F12	Return to tool change position	Shift+F6	Set as the WCS Zero
Shift+F7	Fixed Calibration	Shift+F9	Resume machining form the interrupted point

Shortcut Key	Function	Shortcut Key	Function
Shortcut keys for *Maunal* window			
ScrLk	Activate *Manual* window	4	X- (In jog and increment mode)
6	X+ (In jog and increment mode)	2	Y- (In jog and increment mode)
8	Y+ (In jog and increment mode)	1	Z- (In jog and increment mode)
9	Z+ (In jog and increment mode)		
Shortcut keys for *Trace* window			
Home	Center	End	Show current machining point
+ (mini-keyboard)	Zoom in	- (mini-keyboard)	Zoom out
* (mini-keyboard)	Fit to window size	5 (mini-keyboard)	Front view
8 (mini-keyboard)	Top view	2 (mini-keyboard)	Bottom view
4 (mini-keyboard)	Left view	6 (mini-keyboard)	Right view
1 (mini-keyboard)	Southwest view	7 (mini-keyboard)	Northwest view
3 (mini-keyboard)	Southeast view	9 (mini-keyboard)	Northeast view
Alt+→ or Alt+←	Rotate around Z axis	Alt+↑ or Alt+↓	Rotate around X axis
Alt+PgUp/Alt+PgDn	Rotate around Y axis		

13. 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.