WEIHONG® ENGRAVING MACHINE'S MOTION CONTROL SYSTEM Ncstudio™-V8

User's Manual

（For PCIMC-63A/53B/53C Control Card）

Thank you for choosing our products.

This manual helps you acquaint the product of our company and understand the information of system constitution and configuration. It particularly introduces the process of system installing and various functions. Before using this system, please read this manual in particular. It is beneficial for you to use our software.

For the hardware and software update continuously, it is possible that the software and the hardware you have received differ from the statement in this manual, for which we apologize to you.

Here we list company address, telephone number and Web site. Any questions, please feel free to contact with us. You will be always welcome.

Company Name: Shanghai Weihong Electronic Technology Co., Ltd.

Linkman: Xiulong Song, Dongjing Zhao

Address: Building 29#, Lane 2338, Duhui Road, Shanghai, China

Zip Code: 201108

Phone : 021-33587550

Fax: 021-33587550-880/ 021-33587519

Web site: http://www.weihong.com.cn

E-mail: sales_weihong@188.com   support_weihong@188.com

Pay attention to the followings:

1）Don't plug or pull out the cable connected with the computer when power on.

2）Don't plug or pull out the cable of Manipulation boxes when power on.

3）In order to ensure safety and avoid disturbance, the shell of computer and engraving machine must be grounded.

4）If machine is not in use, please shut off the power supply.

5）Please unplug the power supply, if machine is not used for a period of time.

6）Life-span of the spindle's motor shaft is inversely proportional to its rotation speed.

7）The engraving cutter is very sharp. When the machining is running,in order to avoid hurt to people or damage to the equipment, do not use hand, or handkerchief, or silk kerchief to touch it.

Contents

Tel: (00)86-021-33587550   Fax: (00)86-021-33587519   Website: www.weihong.com.cn   3

E-mail: sales_weihong@188.com   Address: Building 29, Lane 2338, Duhui Road, Shanghai, China. (201108)

1 Summarization

The software of NcStudio™, engraving machine motion control system, is independently designed by Shanghai Weihong Electronic Technology CO., Ltd. It can directly support G code, DXF format, PLT code and ENG code of JDPaint processing file that are created by CAD/CAM applications such as UG, MasterCAM, CASMate, ArtCAM, AutoCAD, CorelDraw, and so on.

NcStudio™ is based on the operation system of Microsoft Windows and takes the great advantage of 32-bit computing and multi-tasks. At the same time, the standard style of Windows interface is simple and dependable, and easy to learn.

Besides functions of manual mode, automatic mode, stepping mode, and backing to the reference point, this numeric control system can also do simulation, display tracking dynamically, calibrate automatically for Z-axis, resume breakpoints (program jump execution), do machining with Y-axis, and so on.

That system can be used with all kinds of three-dimensional engraving machine and three-dimensional milling machines etc; it can be applied to all machining fields, such as mould machining, advertisement and upholster, and incision.

1.1   Software characteristics

The software has the following functions:

1) Basic configurations are four axes; can be further extended in switch value input points, switch value output points, and analog quantity; above configurations are optional to manufacturer.

2) Support NC Rotary Table.

3) Process automatically. Completely support G code of ISO Standard, HP PLT format DXF format and JDPaint ENG format created by CAD/CAM software such as UG, Pro/E, MasterCAM, Cimatron, CASMate, ArtCAM, and so on.

4) Support Manual mode, such as series, step mode, handwheel, not only by handheld device but also by input equipment of computer, such as keyboard and mouse.

5) Support array machining function. By appointing the numbers of rows and lines of the array, it can execute the procedure repeatedly.

6) Support rotating mirror-imagine function, with which user can do mirror-imagine machining with the center of workpiece origin.

7) Support stepping function. It is convenient for user to set accurate feed length, and step-size can be agilely adjusted.

8) Support manual data input (MDI). Customer can input and immediately execute G-instruction online.

9) Advanced machining instruction. By input several simple parameters, user can complete functions of bottom milling, outlining machining, and so on.

10) Support single block mode. User can set the program being executed one block by one block as [Single Block], which provides very good support for diagnose and fault resume.

11) Support advanced automatic functions, such breakpoint resume, and program jump execution.

12) Support function that feeding axis can back to the machine origin accurately.

13) Support auto tool calibration, including fixed calibration, floating calibration, and auto center.

14) Save/resume workpiece condition function; we consider the problem of sudden power off in southern China, develop the function of avoiding file damage, function of breakpoint resume and delicate back to reference point, which can greatly restore the machining condition after computer restarted.

15) Support feedrate percent on-line adjustment. During the machining process, user can adjust the feedrate percent at any time. The least value is o that indicates the processing is suspended, and the maximum value is 120%.

16) Add new function for high-smooth speed connection algorithm. In a usual NC system, connection speed between two G codes commonly is a fixed value (e.g., it may be 0 or a certain small value). However, in new version of NC system, it adopts a particular machining speed adaptive prediction algorithm. In accordance with speed value, direction, maximum acceleration, and the function of forward prediction, the algorithm adaptively decides the connection speed between the current instruction and the next instruction. It has greatly increased the processing efficiency (efficiency enhancement range from 30% to 300%), improved the processing capability, and eliminated the speed chatter marks left on the processing surface.

17) Support 3D simulation display. Through several simple operations, you can directly observe the 3D processing result.

18) Supports simulation function. This function can simulate machining process and will finish in a short time. Meanwhile, it can check if there is any mistake with the program and if the processing result can meet user's satisfaction. Moreover, it can count precisely the actual processing time.

19) The strong and agile keyboard supports. The new version gives very strong support to the keyboard operation. It satisfies the customer's demand in operation in processing.

20) Support log function. The system has a strong Log function which is very helpful for user to view the detailed processing information and system diagnoses.

21) Add an enclosed processing file manager. What the user needs to do is to save the processing file into a specified directory; NcStudio™ will administrate these files in an enclosed file manager.

22) Add an enclosed processing file editor. User can introduce processing file into file editor at any time to edit and modify it.

23) Support to display file processing information. After simulation or actual processing, [Processing Information] window can help user record some important information, such as processing time, processing range, workpiece count, etc.

24) Use CNC card of PCI data bus.

25) Support [Parameters Backup] function. User can backup parameter' setting information and resume it when needed.

1.2 Improvements and new functions

The following improvements and new functions are applied in vision 8.5 or above.

➢ New function: count the number of machining workpieces. Each time finishing a processing assignment, system will count the number of workpieces automatically.

➢ New function: when finishing a processing assignment, the buzzer will emit a reminding sound to prompt user, at the same time, the red lamp will glitter one time if user sets it.

➢ New instructions: If current coordinates system is not G54 system when machining, the CNC State Bar will show in light green to prompt user.

➢ Improvement: the catalog browsing dialog box of the file management can remember the file path selected last time.

➢ New instructions: G923, to set tool offset. For details please turn to program manual.

➢ Enhanced instruction: G906, it is used to check whether the appointed port is overtime. For detailed information please refer to the programming manual.

➢ New function: to name subprogramme, Such as: O"SubProgName". User can use any letter or integer to name subprogramme. For detailed infromation please turn to programming manual.

➢ New instruction: M903, used to change current tool number. For detailed information please turn to programming manual.

➢ New function ot tool change can be used for round cutter head and linear tool magazine.

➢ New function: there will be an option dialog flipped out when pausing; user can decide whether to back to a fixed workpiece coordinate.

➢ New function: when change the polarity of the ports, there will be records in log automatically.

➢ New alarm function for lubrication oil level and spindle fault.

➢ Improved function: the interface to modify current port polarity become more friendly.

➢ New function to control the lighting of the machine.

2 Installation

2.1　Basic Configures of NcStudio™

Host computer

CPU:　　　　　　　　　single-core processor, main frequency 1G or above.

EMS memory:　　　　512 M or above.

Hard disk:　　　　　　20 G or above

Display adaptor:　　　support 1024*768 at least, enhanced color mode

Display:　　　　　　　14" VGA or above

CD-ROM:　　　　　　4X or higher

Mainboard Expansion Slot:　More than two slots of PCI

Operation system

Microsoft Windows 2000 Professional or

Microsoft Windows XP Professional

2.2　NcStudio™ Installation

Before install the new version of NcStudio, please delete the old version of NcStudio. Please consult the section of uninstall procedure. (Section 2.3)

NcStudio includes two parts: the software and motion control card. So, the setup of the system also is divided into two stages: The software setup and the motion control card installation.

Please setup the software after installs the motion control card Say in brief; it can divide into the following steps:

1)　Shutdown the computer, install the motion control card.

2)　Restart the computer, wait for a moment after enter the Windows operate system, put into the CD for setup, choose auto update package, the installation will begin after double click auto update package, and finally restart the computer automatically.

3)　Run NcStudio.

The key steps are introduced detailed as follows.

Install NcStudio™ motion control card

Close the host PC's power supply, open the cover, and insert the motion control card into an empty PCI slot.

While installing the motion control card, lightly press two side of the motion control card by hand, insure the motion control card is firm insert in the slot, and get in touch with computer main board dependably without fluttering, then tighten the locknut of the card and finally lid it. The installation of motion control card complete.

Install the NcStudio Software

Please install the software according to the nether step:

1) Open the computer's power supply, start the computer, the system enter into the Windows operation system automatically. Please setup the operation system at first if you have not done so. (8.3 editions temporarily support the WINXP/2000 only)

2) After Windows operation system startup, please close other applications that are running.

3) Put the setup CD-ROM of NcStudio system into the CD-ROM driver.

4) Double click "My computer" icon, then double click the CD-ROM icon. Find out the SETUP.EXE file and double click the icon . The first dialog to appear is shown below.

Fig. 2-1 Software Installation Picture1
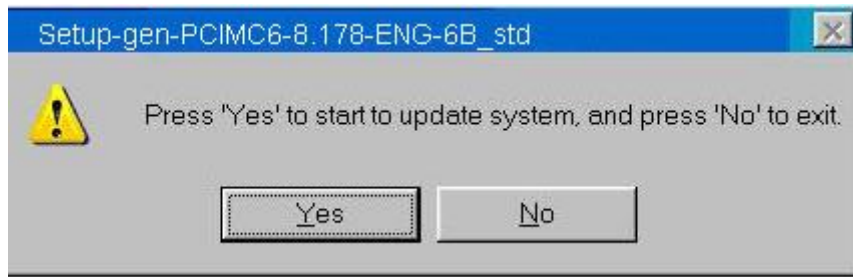
Choose 【Chinese】, go to next step.

Fig. 2-2 Software Installation Picture 2
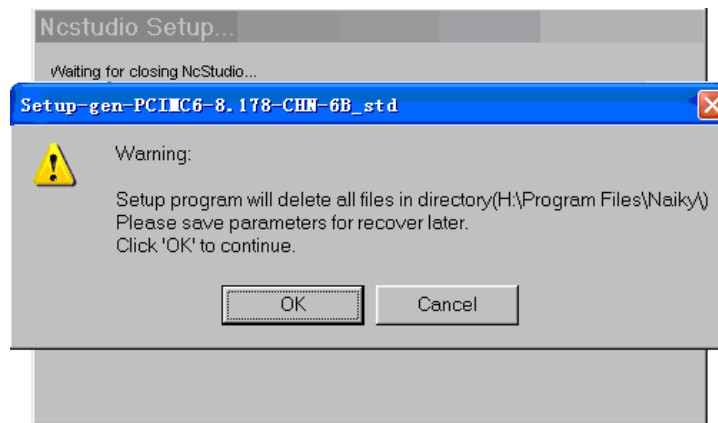
Click 【Yes】to go futher.

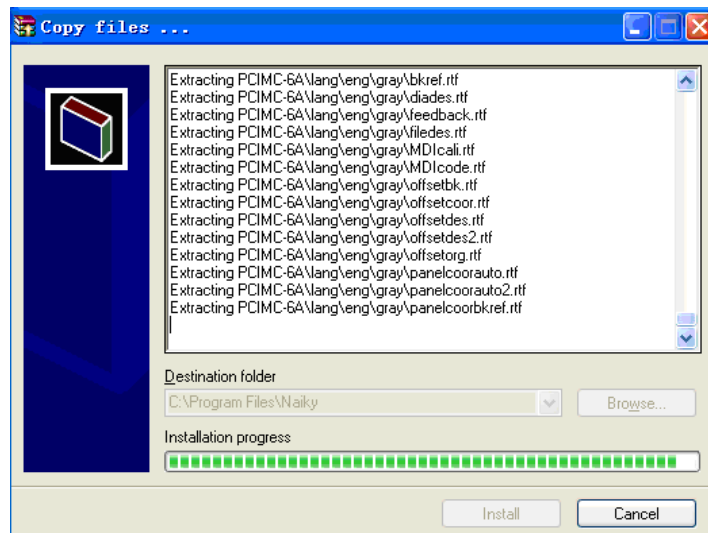Fig. 2-3 Software Installation Picture 3

Click [Yes], go to next step.

Fig. 2-4 Software Installation Picture4
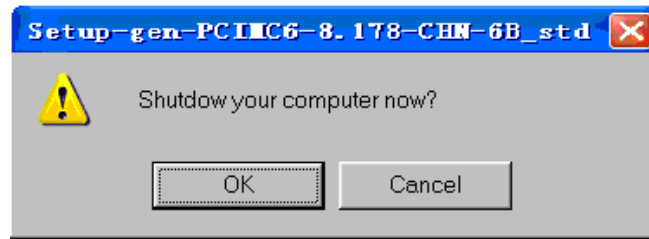
Fig. 2-5 Software Installation Picture 5

5) Click [Yes] to shut down the computer.

6) Restart the computer, you will see the following dialog:

Fig. 2-6 Hardware Installation Picture 1

7) Click [Atuo Install the Software] and click [Next], system will guide to install driver application for the motion control hard:
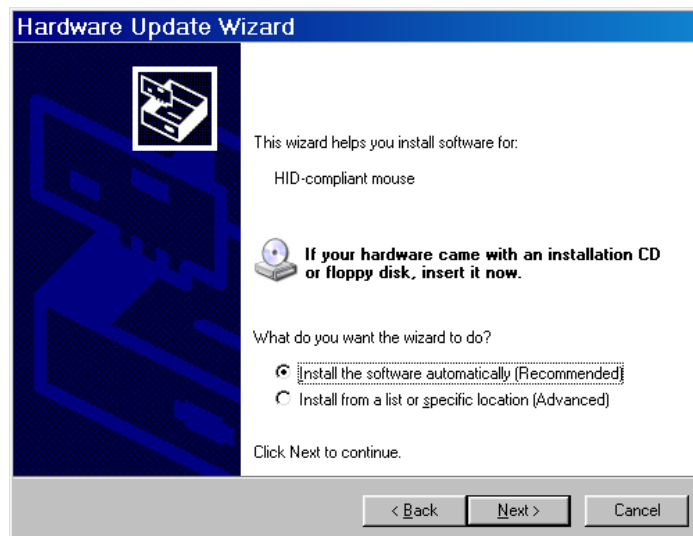

Fig. 2-7 Hardware Installation Picture 2

8) Select "Install the software automically (recommended).

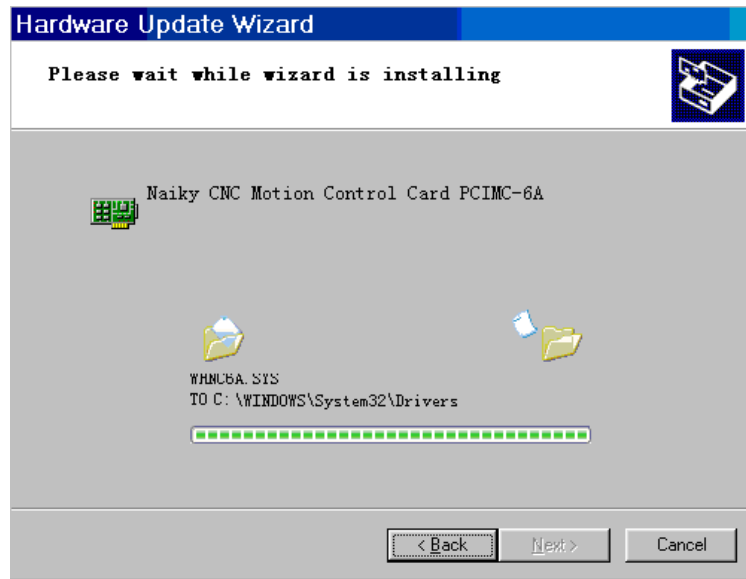9) Click [Next], system begin to install the application, the default route is C:\window\system\drivers.

Fig. 2-8 Hardware Installation Picture 3

10) Click [Next], sytem will prompt software installation finished.

Fig. 2-9 Hardware Installation Picture 4

11) Click [Finish].

12) Whole system installation has been completed, double click the icom [icon] on your desktop to open CNC sytem.

Install customized keyboard

NcStudio Version 5.4.53 and above support customized small keyboard，which is used to operate the machine tool conveniently。

Before Install the NcStudio™ customized small keyboard，please exit the objective computer's operation system in running (Windows usually) if the PC is power on, then turn off the computer's power supply. Pull out the keyboard's attachment plug from the computer, insert it into the small keyboard's receptacle with one wing of electrical cord, then insert the small keyboard's attachment plug with two wings of electrical cord into the female receptacle of the computer's keyboard. Pay attention to keep the attachment plugs touching the female receptacles well. After finished installing the keyboard, you can startup the computer, run Windows, make sure the keyboard and the mouse work well, execute NcStudio™ if necessary.

> Attention:
>
> The installing interfaces above may be different in different versions, what we give here is
>
> only for your reference.

2.3   Uninstall NcStudio™ System

NcStudio™ is green software, and has the advantage of easy installation and no need to register. Morever, to uninstall NcStudio™ system, user only need to delete its document folder: C:\Program Files\Naiky, delete [Start] menu→ [Applications] →NcStudio , and also the shortcut on the desktop. There will be no residue left on userr hard disk.

When you install new version Ncstudio™ system, it will cover and update the old version system automacitally.

2.4   Connection between Ncstudio™ Control Card and Its Driver System.

Ncstudio™ mechanical motion control signal use the Motion Control Card fixed on the computer slot to realize the communication between software Ncstudio™ and feed motor driving system fixed on the machine electrical box.

Before connecting Ncstudio™ mtion control card with feed motor driving system, user firstly needs to install electrical box on the machine. Use special cable joins the jack of Motion Control Card with the corresponding jack on the electrical box. Then we have finished connection between the Motion Control Card and feed motor driving system.

Note:

Refering the specific step to connect the card with the electrical system, please refer to the introduction of specific type card.

Different type card may have differnet working mode, and the funciton may be different either.

3 Basic concept of NcStudio™

## 3.1 Operation Mode and Operation State

Operation mode

Regarding machining operation, there are basic three modes as below.

> **Auto Mode**

Under automatic operation mode, machine tool generates motions in accordance with the prepared procedure. So the processing procedure must be loaded in advance.

> **Manual Mode**

Manual mode can be subdivided into series and stepping modes.

Under [Series] mode, user can control the machine with computer keyboard, hand operation box or manual pulse generator. When user uses this operation equipments to give out motion signals, such as press button , the machine will keep in motion until the button is released.

Under [Stepping] mode, user can control the machine with computer keyboard, hand operation box or manual pulse generator. One time operation of pressing button and releasing it, the machine will move a specified distance. Therefore, user can accurately control the displacement of the machine tool under [Stepping] mode.

Operation state

According to the machine movement, each operation mode can be divided into the following types of operation states; operation mode and operation state form together the state of a machine tool.

> **IDLE State**

Idle state is the most common state. Under this state, the machine has no motion to output, and is always prepared to accept any new task.

> **ESTOP State**

This is an abnormal state. When there is a hardware fault or when user presses down the [Reset] button, system will enter ESTOP state and implement the predetermined protection actions, such as turning off spindle motor and cool pump. In this state the machine tool is locked and cannot carry out any new motion. After hardware fault or ESTOP state is obviated, system will automatically implement [Reset] action and make the machine tool return to the IDLE state.

> **RUNNING State**

When the machine is implementing any action, system enters into Running State.

> **PAUSE State**

When a machine tool is running, if user performs [Pause] command, or presses down [Pause] button, or system parses a M01 command (wait instruction), system will enter into PAUSE state and wait for user's further instruction. To suspend the current action and let the system enter into IDLE state, several methods can be effective. For example: Perform 'Start' command, or press [Start] button, or perform 'Stop' command, or press [Stop] button, or press [Auto mode] or [Reset].

> **LOCK State**

Lock state is an internal state which occurs when switching states. Normally, user may not meet with this state.

3.2　Machine's Coordinate System

Coordinate system is a terminology that is used to describe the motion of the machine tool. For the sake of unification, standard coordinate system adopts the right-hand rule. See Machine's Coordinate System Fig. 3-1 Demonstration for Right Hand Principle.
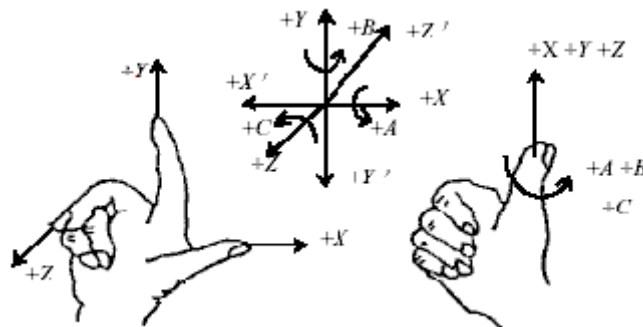

Fig. 3-1 Demonstration for Right Hand Principle

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

——Z-axis coincides with the spindle. The direction of the cutter moving away from workpiece is the positive direction of Z-axis (Z+).

—— X-axis is perpendicular to Z-axis and parallel to the clamped surface of workpiece. For

the single column milling machine, if user face the spindle of cutter and look in the column direction, right moving direction is the positive direction of X-axis (+ X).

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

> **Machine Coordinate System**

Machine coordinate system is a set of fixed right-hand coordinate system. Its coordinate origin is a fixed position that corresponds to the machine tool. Therefore, at any time, a certain point in space can be exclusively fixed by machine coordinate system.

To complete support of the machine coordinate system demands the capability of returning to reference point. Otherwise, the concept of the machine coordinate system will be incarnated in software.

> **Workpiece Coordinate System**

It is more often to adopt workpiece coordinate system at machining various kinds of workpieces. Generally speaking, the machining position that we mention is a certain point relative to the workpiece, while the position where the workpiece is clamped is always variable corresponding to the machine origin, so it is necessary to introduce a set of more convenient coordinate system for workpiece machining. That is workpiece coordinate system. Workpiece coordinate system adheres to the right-hand rule as well. Its origin is fixed corresponding to a certain point on the workpiece, and is possibly floating corresponding to the reference point.

4 Interface of NcStudio

After installing the software of NcStudio, double click the hotkey on the desktop. Operation interface of NcStudio™ are constituted by seven major sections. They are title bar, menu bar, numerical control state bar, information display bar, function window, status bar, and operation bar.
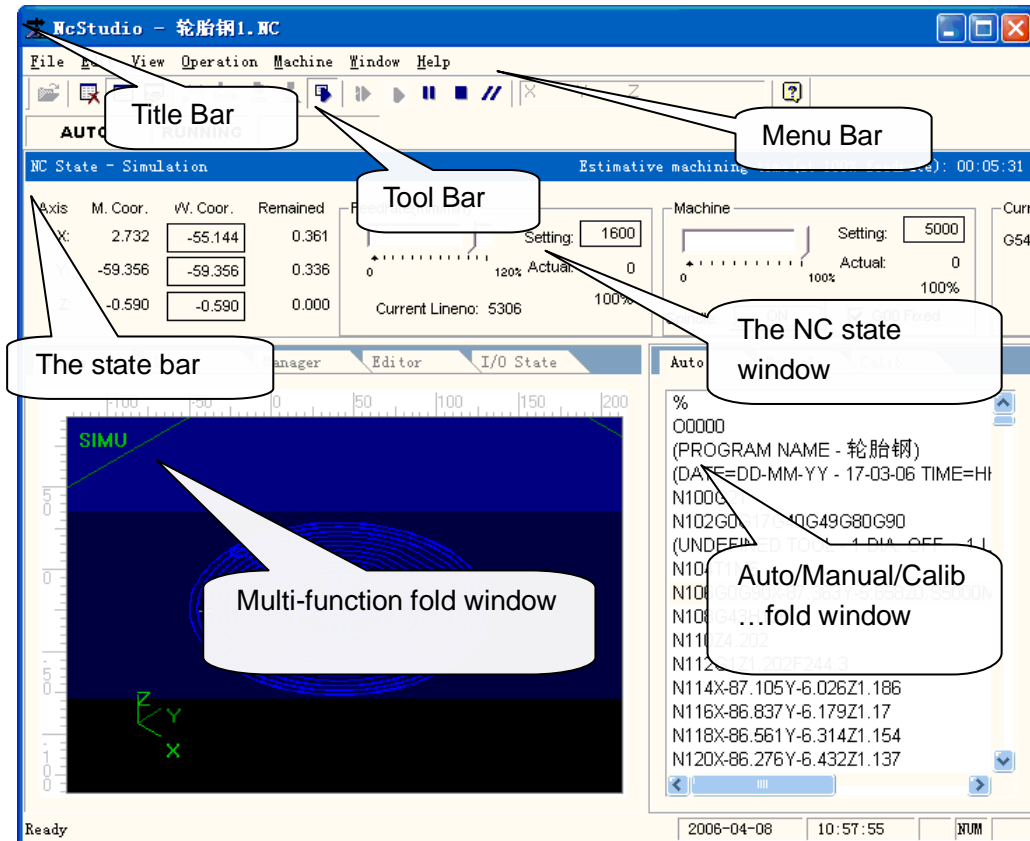

Fig. 4-1 Interface of NcStudio

The function windows are divided into three areas, and ESC key can be used to switch between them.

> ➢ **The first areas: state window**

> ➢ **The second areas: track window, log window, procedure manager window, editor window and I/O state window**

> ➢ **The third areas: auto window, manual window and calibrator window**

4.1　Title bar

The first column of NcStudio operation interface is title bar shown in Fig.4-2. It is used to show the name of the application and the name of the loaded processing file. The color of title bar is used to indicate whether the window is activated or not.

Fig. 4-2 Title bar

Note:

In Windows system, the concepts of active window and inactive window are very important. An active window refers to the window which can accept the input of keyboard at present. Anytime there is only one active window, and all the other windows are inactive.

Please pay attention to the color difference of the title bar between active window and inactive window. The color of an active title bar is blue, while the color of an inactive title bar is grey.

The icon in the left of the title bar is a system menu box. It is used to open the window control menu. Click the icon or press "Alt + spacebar", and a system menu will pop up.

This menu can control the position and the size of the window, such as restore, move, and close, maximize, minimize, etc. On the right side of the title bar there are three control buttons, including button for restoring, button for maximizing and button for minimizing. These buttons are used for quickly setting the size of the window. Referring to the detailed operation manner, please turn to the on-line help in MS-Windows.

In addition, each sub-window also has a relative title bar. The active sub-window and the inactive sub-window can be distinguished from the color of the title bar. Please refer to the describing of chapters below.
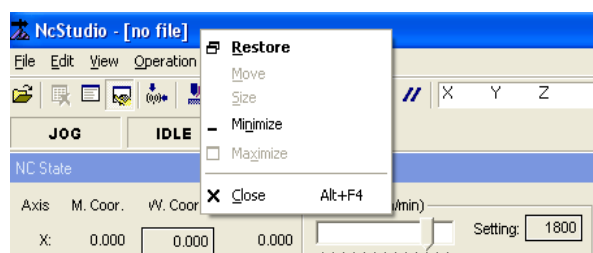

Fig. 4-3 System Menu

4.2 Menu bar

Under the title bar is the menu bar; it includes many concealed menus, shown as below.
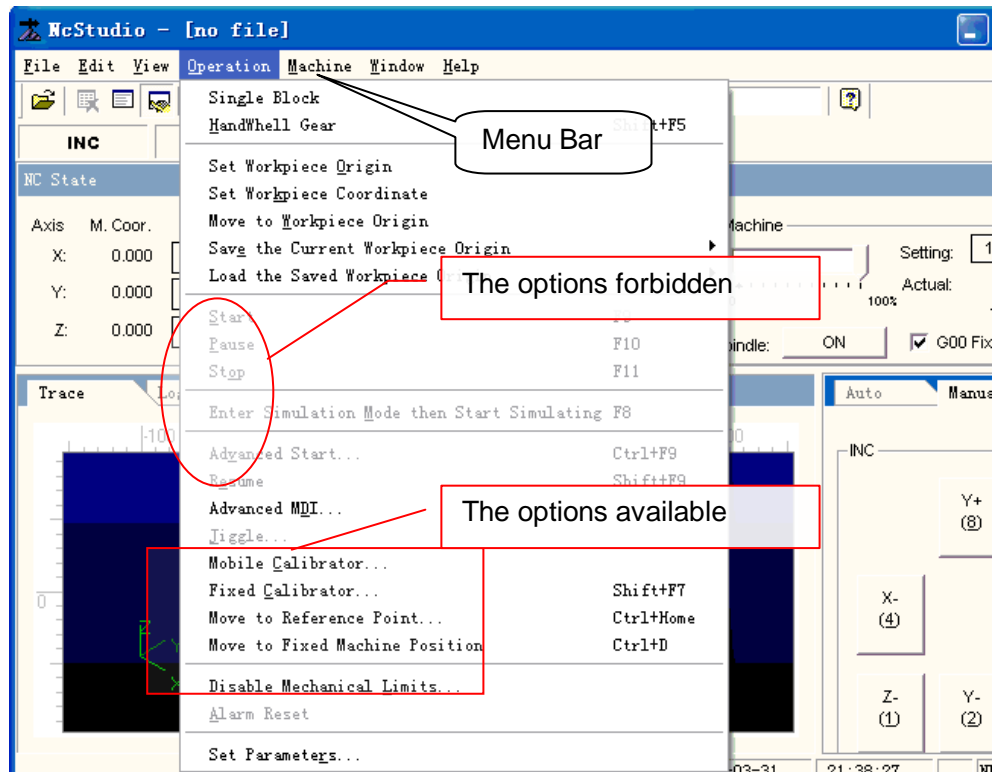
Fig. 4-4 Menu Bar

The menu bar includes several concealed pull-down menus; each pull-down menu is composed of several menu items; each menu item can realize a special function, action, or state that is relative to a certain procedure. To implement a function, an action, or to change the state setting, user only needs to select the relative menu item. Menu selection can be realized by mouse or by keyboard.

➢ **Mouse operation**

Click the menu on the menu bar, and a pull-down menu will pop up. Click the target item.

➢ **Keyboard operation**

Press down simultaneously 'Alt' key and hot letter key. For example, there is a line under 'F' in the "File" menu, so the combination key is "ALT+F". After that, a relative pull-down submenu will pop up. Regarding the pull-down submenu, user can also use keyboard to choose target item. For instance, [File] → [Open and Load]. When the pull-down menu of [File] appears, directly press the letter 'O' can select the item.

➢ **Shortcut key operation**

In pull-down submenu, some items have shortcut keys in their right. For example, [File] → [Open and Load], its shortcut key is 'Ctrl + O'. That is to say, user can directly select the target item by direct shortcut key, which can avoid the trouble to enter into multilayer menus.

In the drop down menu there is a relative hot key at the right of some menu options, for example: F9 is the hot key of "beginning …" in the menu of" operation", that is to say the hot key will implement the order directly.

Some menu option has three dots behind. For example, [File] → [Open and Load…], it indicates that a dialog will automatically pop up. If the item of the submenu shows in grey, it indicates the item is unavailable under current state.

In addition, right click on any position, a corresponding shortcut menu will pop up, then choose the relative item.

4.3　Tool bar

Underneath the menu bar is a toolbar. Tool bar is composed of operation buttons, which correspond to the functions of menu items or options. Use mouse to click the button to realize the function.
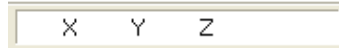

Fig. 4-5 Tool bar

Tool bar has greatly simplified user's operation process, and makes the whole operation process visual instead of the tedious order list.

> **Direct Location**

Under the idle state, customer can directly enter Workpiece Coordinate into " direct location edit frame", after press "Enter" key, system will quickly move tool to the appointed position.

 :input workpiece coordinate;

 : with symbol "✳" in front, input machine coordinate.

On the tool bar, each button corresponds to a menu item. Click a button, a prompt dialog will flip out. Referring to the details, we will introduce in following chapters.

4.4　NC State bar

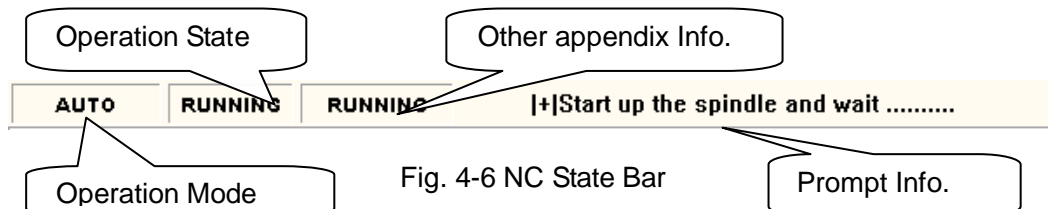Underneath the tool bar is NC state bar. NC state bar displays current NC states and some alarming information.

Fig. 4-6 NC State Bar

4.5   Status bar

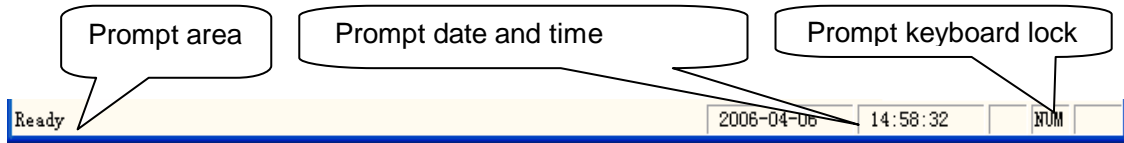The bottom of the screen is a status bar, shown as diagram:

Fig. 4-7 NC Status bar

➢ **Prompt area**

Prompt information about current operation or chosen order.

➢ **Prompt date and time:**

Show information of current date and time.

➢ **Prompt keyboard lock:**

Show current state of the keyboard capital lock, number lock and scroll lock.

4.6   NC State window

Underneath the state bar is NC state window. It can be divided into four areas according to their functions, including "current position" area, "feed speed" area, "rotation speed of spindle" area, "current interpolation command display" area.
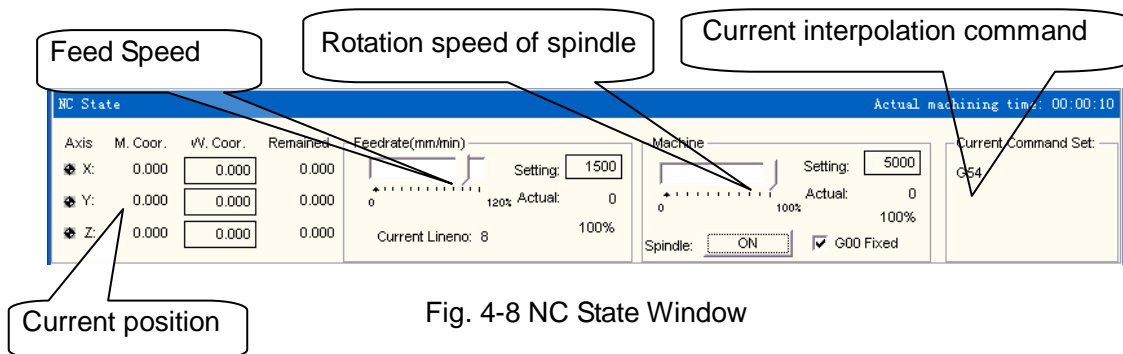

Fig. 4-8 NC State Window

➢ **Current position**

Show current position of the spindle tool, including workpiece coordinate, machine coordinate and residual distance, and user can set current position as workpiece origin.

➢ **Residual/Remaining distance**

It is the absolute distance between the position where current command asks the tool to get and current instantaneous position of the tool.

➢ **Feed speed**

Set feed speed, adjust feedrate percentage, and display actual value of the percentage and

feed speed.

➢ **Spindle speed**

Set spindle speed, adjust spindle speed percentage, display the actual value of the percentage and spindle speed, and also can control the rotation of the spindle.

➢ **Current interpolation command**

Show the state of current command, such as: G54, mode/modeless, G01, G17, G18, G19 and etc.
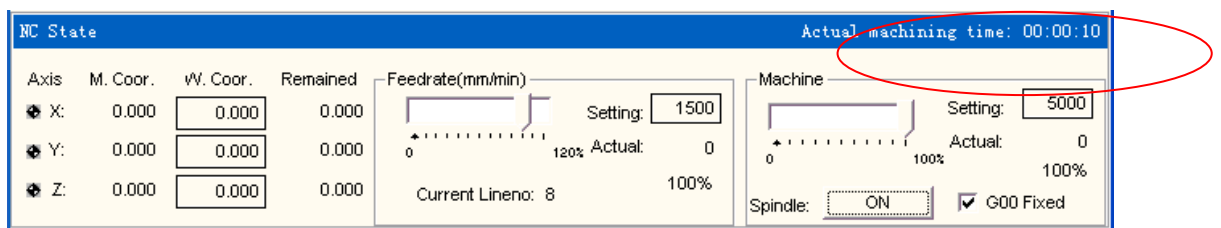
➢ **Time information**

Fig. 4-9 Time Information

On the right side of the title bar, under simulation mode, it shows the predetermined processing time (100% feedrate percentage). While actual machining, it shows the actual machining time.

Current position

In order to describe all kinds of position expediently, NcStudio shows two sets of coordinate systems at the same time: machine coordinate system and Workpiece coordinate system. NcStudio provides many functions to support the two sets of coordinate systems; user can neatly set the relative offset between them.

After finishing backing to the reference point, in front of the each axis there will be a symbol ⊕, which indicates the machine coordinate is valid.
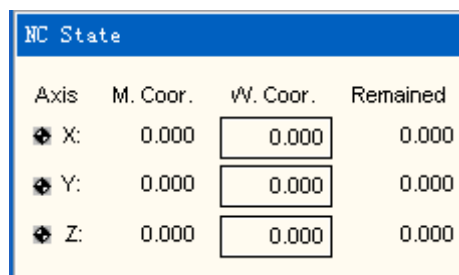

Fig. 4-10 Current Position

System provides a convenient method to set and modify the workpiece origin. To set current point as workpiece origin, move cursor to the coordinate axis zone, click the workpiece coordinate, you will see the following dialog.
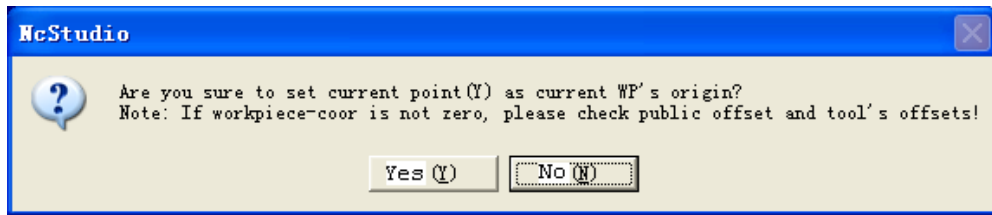
Fig. 4-11 Set Workpiece Origin

Click [Yes], you will see the workpiece coordinate change into 0. To set all 3 axes of current positions into 0, user only need to click each workpiece coordiante one by one.

> Prompt:
>
> If the workpiece coordinate can't be set as 0, please check the value of public offset

Feed speed

In feed speed zone, it displays setting speed, actual speed, and feedrate percentage. User can also modify values of setting speed and the feedrate percentage.
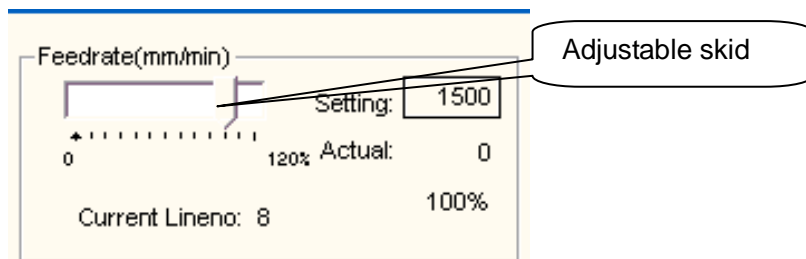

Fig. 4-12 Feed Speed Zone

➤ **Adjustable skid of feed speed**

Pull the adjustable skid to adjust current feedrate within scope of 0~120%. Feedrate ratio shows in percentage style.

➤ **Setting value:**

Under auto mode , click value of , a dialog will flip out.
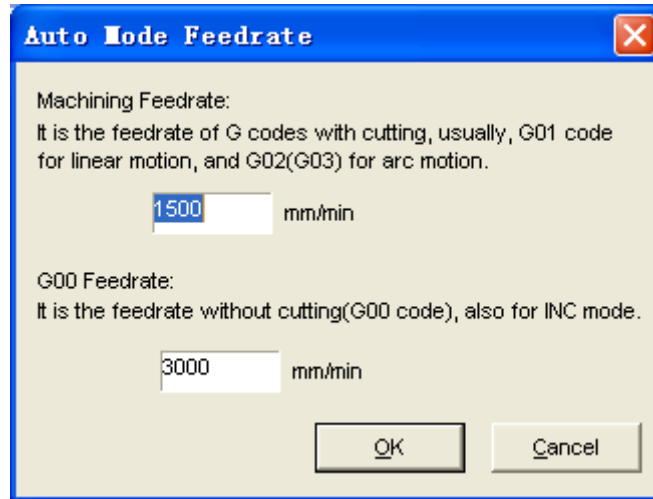
Fig. 4-13 Set Auto Speed

Under manual mode, click the value of
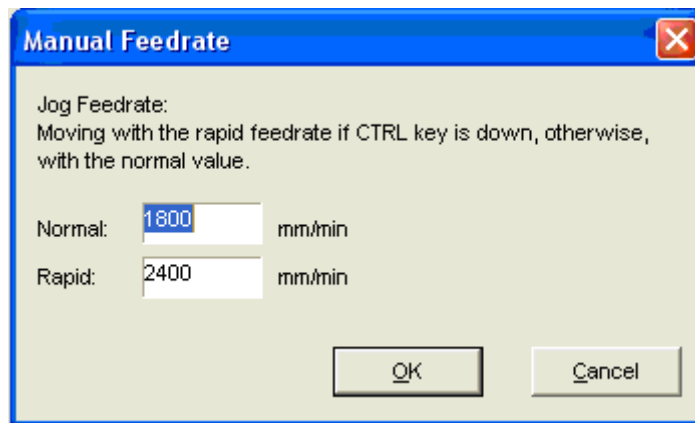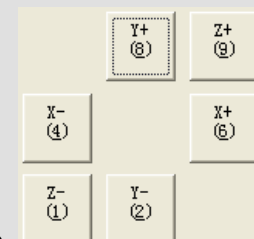
 , a dialog will flip out.

Fig. 4-14 Set Manual Speed

Manual high speed and manual low speed can be adjustable.

Prompt:

Switch between manual high speed and manual low speed:

Press 0 and direction key of the number key (number on picture) to set manual high speed.

Directly press number key to set manual low speed.

The setting operation here is the same as the setting operation in [Operate Parameter] in [Parameter] window.

➢ **Actual value**

It is the instantaneous value of the feed speed. The value will vary with setting value, current state of deceleration or acceleration. K

➢ **Count workpiece number**

It shows the finished workpiece number. Click the number can set as 0.

Spindle speed

In spindle speed zone, it displays setting speed, actual speed, and spindle ratio. User can also modify values of setting speed and spindle ratio.
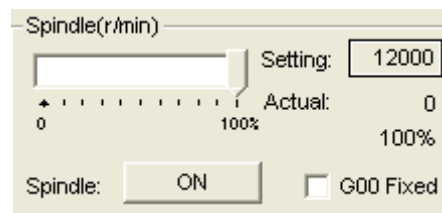
Fig. 4-15 Spindle Speed Zone

In The main spindle speed area is similar to the feed rate area, it display setting speed, actual value, feed rate and etc. It can also modify the setting speed and the value of feeding rate.

➢ **Adjustable skid of spindle zone**

Pull the adjustable skid to adjust current speed within scope of 0~100%. Spindle ratio shows in percentage style. Actual value = setting value✕ current spindle speed ratio.

➢ **Spindle On button**

This button is used to control the spindle rotation on or off.

➢ **Setting of spindle revolution speed (rotation speed)**

Clicking [setting] value button, a dialog of setting spindle speed will flip out, user can set the spindle here. See the following Figure.
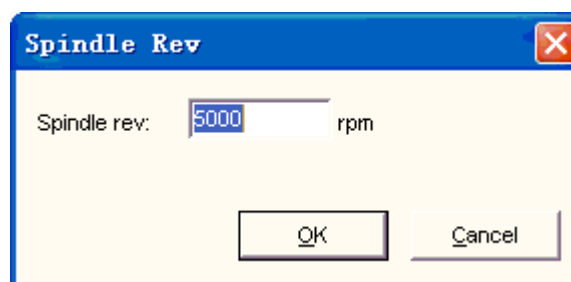
Fig. 4-16 Spindle Rev Dialog

➢ **G00 speed**

Select the item, while dry run, running speed is fixed at 100% setting speed of dry run, and will not be influenced by the feedrate ratio.

Current interpolation command and current tool no.

Fig. 4-17

It shows the state of current command, such as: G54, mode/modeless, G01, G17, G18, G19 and etc.

Current axis number is the axis that can be currently used, it can be single axis or double axes, 1 indicates Z1.

4.7   Automatic Operation Window

It shows the current opened processing file. NcStudio supports processing procedure of ISO standard, including G code, HP PLT code, DXF code, ENG code, and our company's NCE code. With this window, user can look up and edited the current processing file.

Fig. 4-18 Automatic Operation Window

Right click the mouse in this window, a context menu will flip out.

```
Show File Line Number
Trace Current Line

Edit Loaded File

Open and Load...
Unload
```
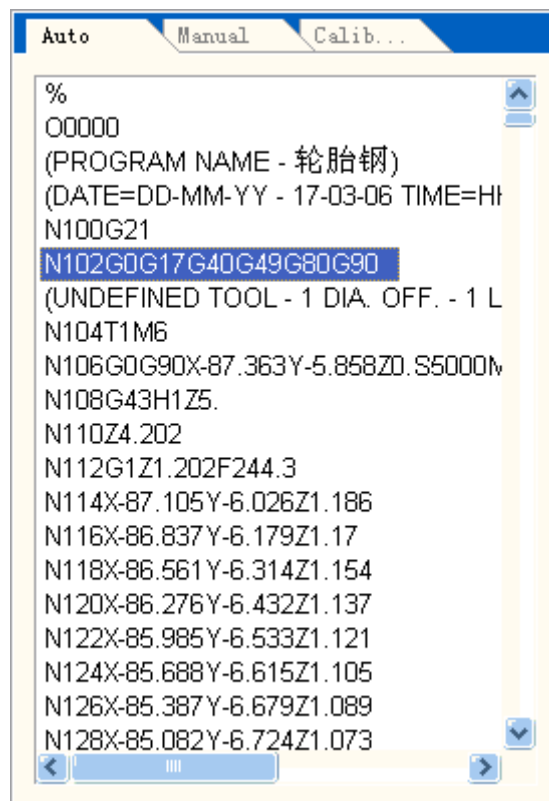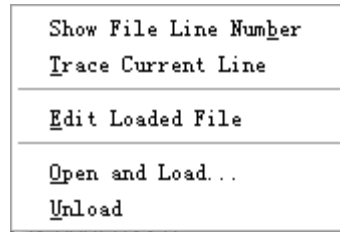
Fig. 4-19 Context Menu of Automatic Operation Window

These items can also be found in standard menu. Items "Show File Line Number" and "Trace Current Line" can be found in "View" menu; Three residual items can be found in "File" menu. For detailed information please refer to chapter 5.1 and 5.3.

Windows of auto mode, manual mode and calibration mode can switch from one to another.

Methods to activate "Auto" window

Menu mothod: Choose" window| Show Auto Window ";

Hotkey method: click key "Ctrl+1" and activate that window;

Mouse method: directly click the window title.

Prompt:

The processing procedure shown in the window can only be looked into, but can't be edited and modified. If you need to edit the file, please click "file| edit loaded file"; or right click in this window and select "edit current processing file" in shortcut menu, then modify it in file "Editor" window.

4.8   Manual Operation Window

Manual window is an interactive operation environment
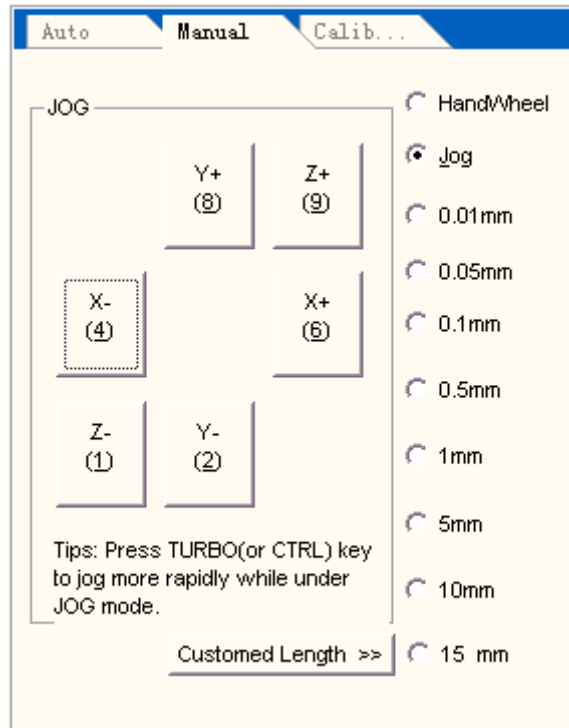
Fig. 4-20 Manual operation window

Methods to activate manual window

Menu method: Choose" window | Show Manu Window";

Hotkey method: Click button "Ctrl+2" to activate that window;

Mouse method: directly click the title of the window.

Direction button

 , : corresponding to positive and negative direction of X-axis.

 , : corresponding to positive and negative direction of Y-axis.

 、 : corresponding to positive and negative direction of Z-axis.

Feed mode selection

Under manual mode, there are two kinds of feed mode, series and stepping. We will introduce in details below.

> **Series mode**

Select to select series mode,

Press the direction number key of the number keyboard, the machine will move serially until you release the key.

Select ⊙ HW , sway handwheel, and the machine will move serially until you stop swaying the handwheel.

> Note:
>
> When activating manual window, number lock is not taken into consideration.

➢ **Stepping mode**

Select any of the following items, they are all stepping mode.
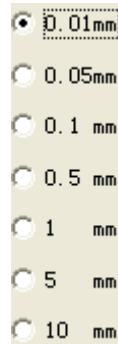
Fig. 4-21

When performing stepping mode, in "Trace" window machining track will show in the color of G01.

User can implement stepping operation by mouse, keyboard and operate panel. Each time to trigger the button, the corresponding axis will move the appointed step-length.

➢ **Customized step length**

It is another manual operation mode that is similar to manual operation. But what different from continuous point-motion operation mode is that it can control the stepping distance of machine tool's motion-axis accurately.

Click [Customized length] button, a dialog will pop up.
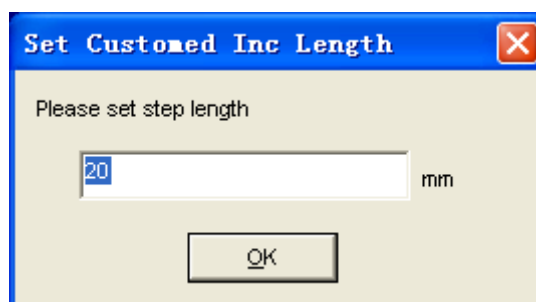
Fig. 4-22 Customized Step Length

Input suitable step lenth, click [OK].

> Note:
>
> To avoid misoperation cause damage to the machine, the customized step length can't

be set too big.

Because system needs time to perform each time click operation, so the click operation can't be too frequent. Or a fault prompt information will remind you that "Unable to perform the action because last operation is not finished yet"

4.9 Calibration Window

Calibration window is to execute tool calibration.
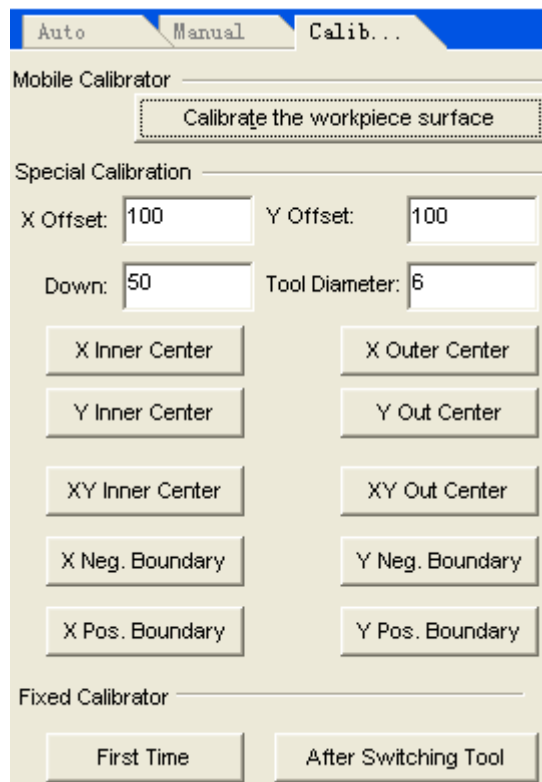
Fig. 4-23 Calibration Window

Measure workpiece surface

To measure workpiece surface can help user to confirm the height of workpiece. Measurement of workpiece surface is specially used in Naiky or Weihong system. It is used to set Z-axis origin automatically, and has the same function with floating calibration (when the thickness of mobile calibrator is 0). To measure the workpiece surface requires the machine worktable must be insulated.

Floating calibration requires putting the calibrator on workpiece surface, manipulate the tool tip of Z-axis to touch the calibrator, when calibrating stops, system can get the position of the tool tip, then subtract the thickness of calibrator, system can confirm Z coordinate of workpiece origin.
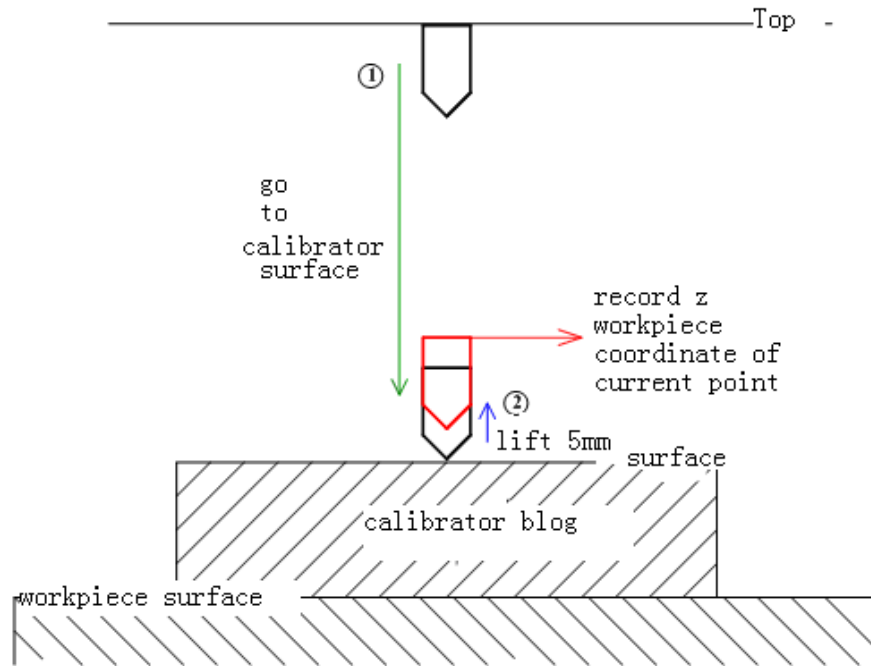
Fig. 4-24 Floating Calibration

Special Calibration

Special calibration, including auto calibration of workpiece origin and workpiece boundary, because these two auto calibration also use calibrating signal, so they all called special calibration. All these calibration are specially used in Naiky/Weihong system.

Special calibration is used for calibrating X and Y center of workpiece, which is to machining and make a processing file conveniently. But the precondition is you must make sure z coordinate of workpiece, and the machine must be insulated.

X center distance: when executing center calibration, the preestimated distance from tool tip to X boundary.

Y center distance: when executing center calibration, the preestimated distance from tool tip to Y boundary.

Tool falling distance: when executing center calibration, the tool's falling distance when one side of tool goes out of boundary. Let the tool touch the workpiece in its moving back process, and get the signal.

Tool diameter: tool's diameter.

Center calibration

Select this order, enter function menu of center calibration. Center calibration is used to calibrating workpiece center. It includes center calibration of X inner center, X outer center, Y inner center, Y outer center, XY inner center, XY outer center. Inner center calibration is user for calibrating center of cavity workpiece.

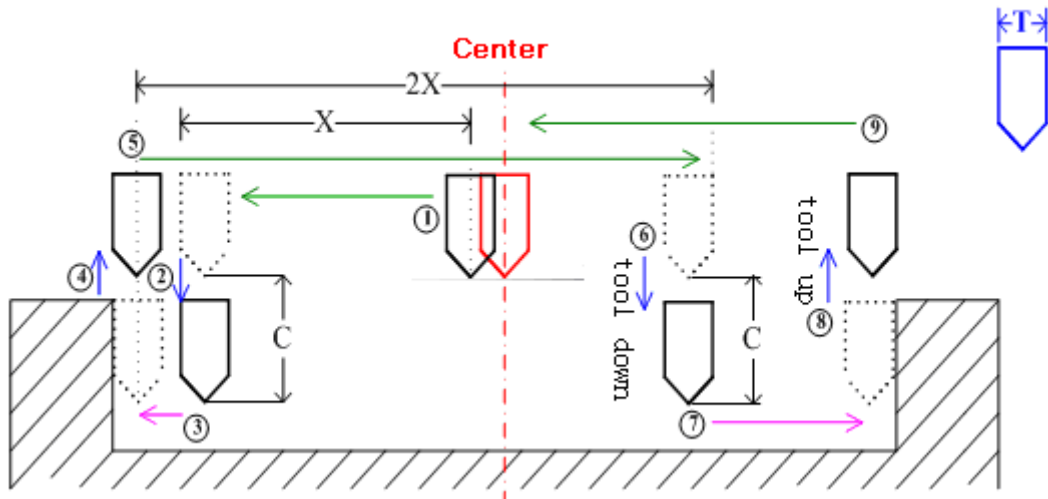Fig. 4-25 Calibration Process of X Inner Center
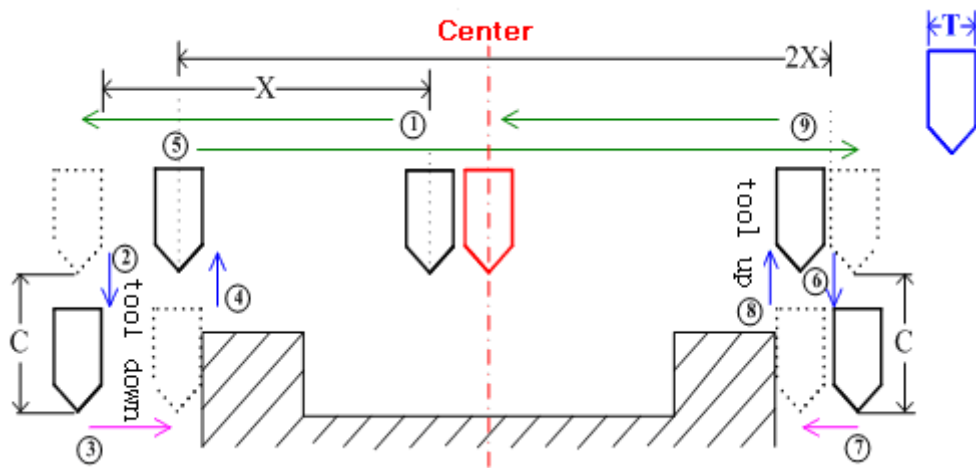
Fig. 4-26 Calibration Process of X Outer Center
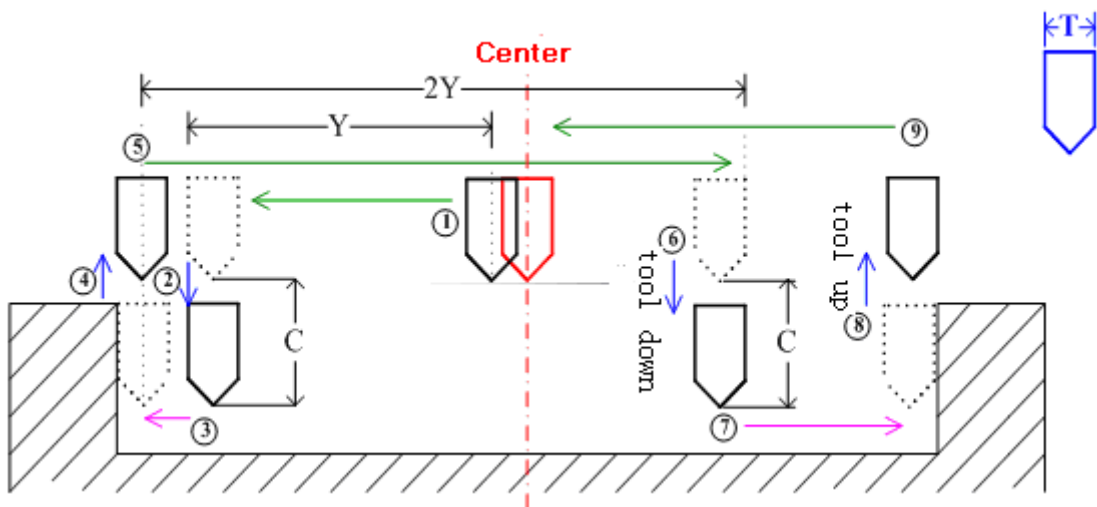
Fig. 4-27 Calibration Process of Y Inner Center
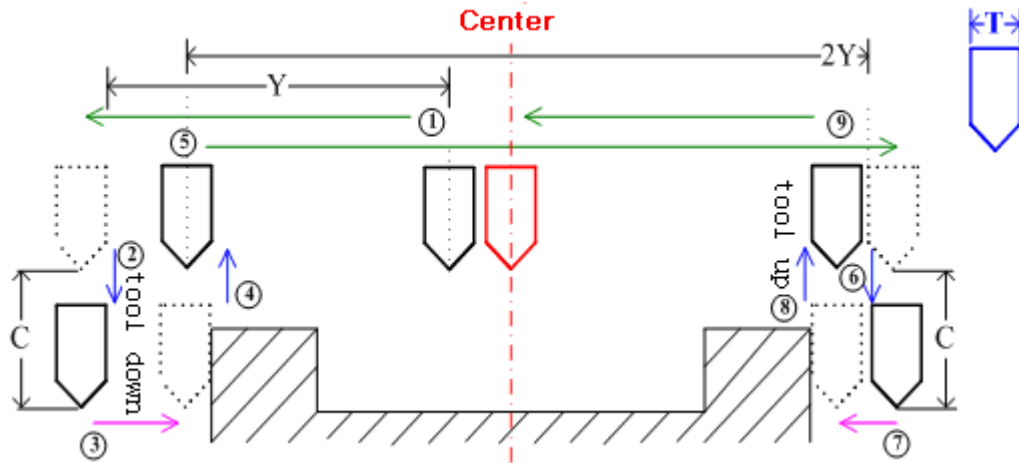
Fig. 4-28 Calibration Process of Y Outer Center

Before calibrating, please use "X" command to set X center distance, use "Y" command to set Y center distance, use "C" command to set tool falling distance, and use "T" command to input tool diameter.

Before using tool calibration, tool tip must be located at the preestimated center position; when center calibrating, tool's falling or lifting distance are all called tool falling distance.

Center distance is the estimated distance form center to both sides; when executing inner center calibration, the distance must be smaller than actual distance; when outer executing center calibration, the distance must be bigger than actual distance.

For inner center calibration, the falling distance must be smaller than the actual distance from tool to workpiece surface. For outer center calibration, the falling distance must be bigger than the actual distance from tool to workpiece surface.

Boundary calibration

Boundary calibration includes calibrating + X boundary, - X boundary, + Y boundary and -Y boundary,
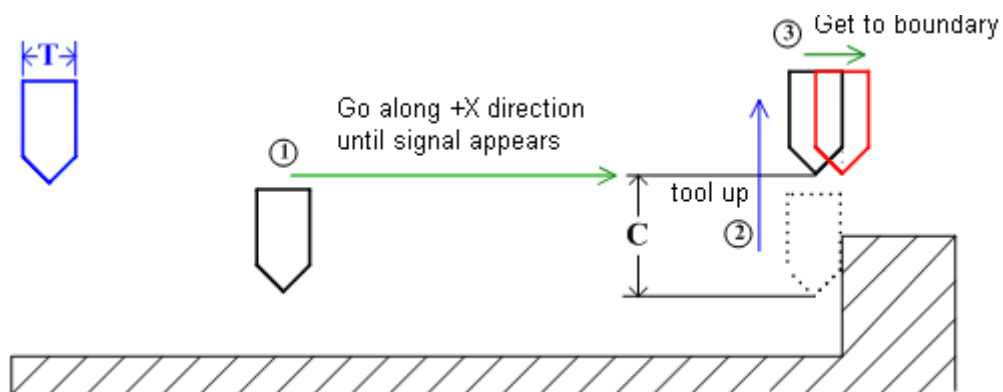
Fig. 4-29 Calibrating Process of +X Boundary

Calibration process of calibrating - X boundary, + Y boundary and -Y boundary are the same as calibrating + X boundary.

Fixed calibration
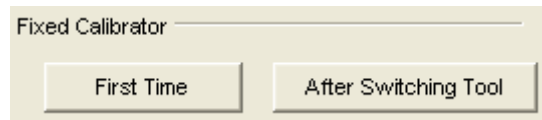
Fig. 4-30 Fixed Calibration

Fixed calibration is used to revise z coordinate of workpiece origin after tool change.

When calibrating, at first, ensure workpiece surface artificially, set workpiece surface as Z coordinate of workpiece origin, then click button "First Time", system will automatically record current Z workpiece coordinate. If meeting tool change or tool break, click button "After Switching Tool", system will resume previous z coordinate of workpiece origin before tool change or tool break.

Attention:

Fixed calibration will be invalid after NcStudio is closed, each time to start NcStudio system, user needs to reset fixed calibration.

Operate as following prompt:

First calibration
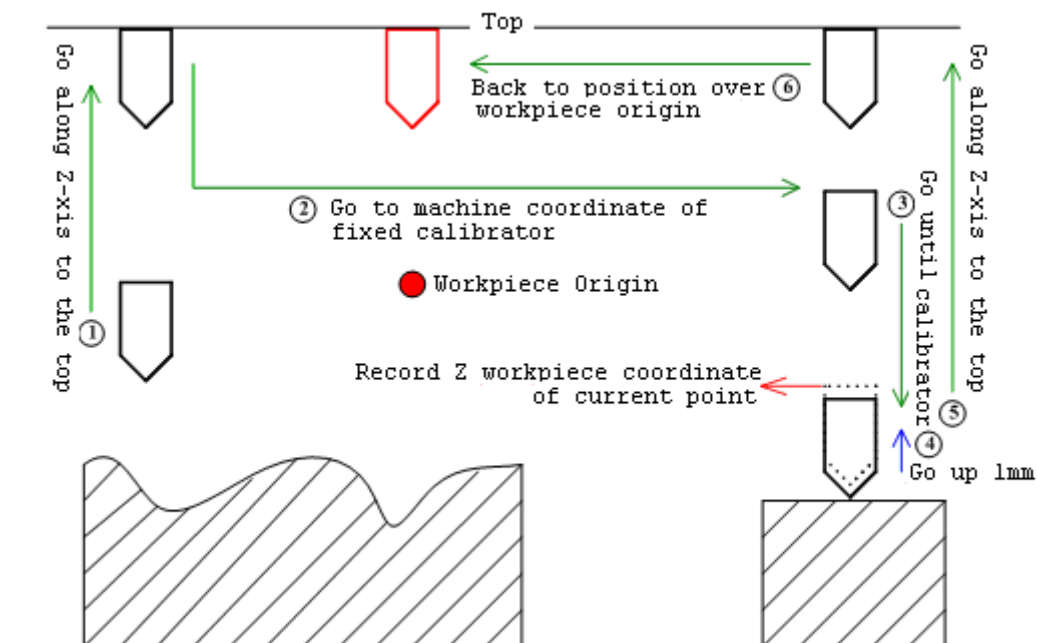
Fig. 4-31 First Calibration

Calibration after tool change
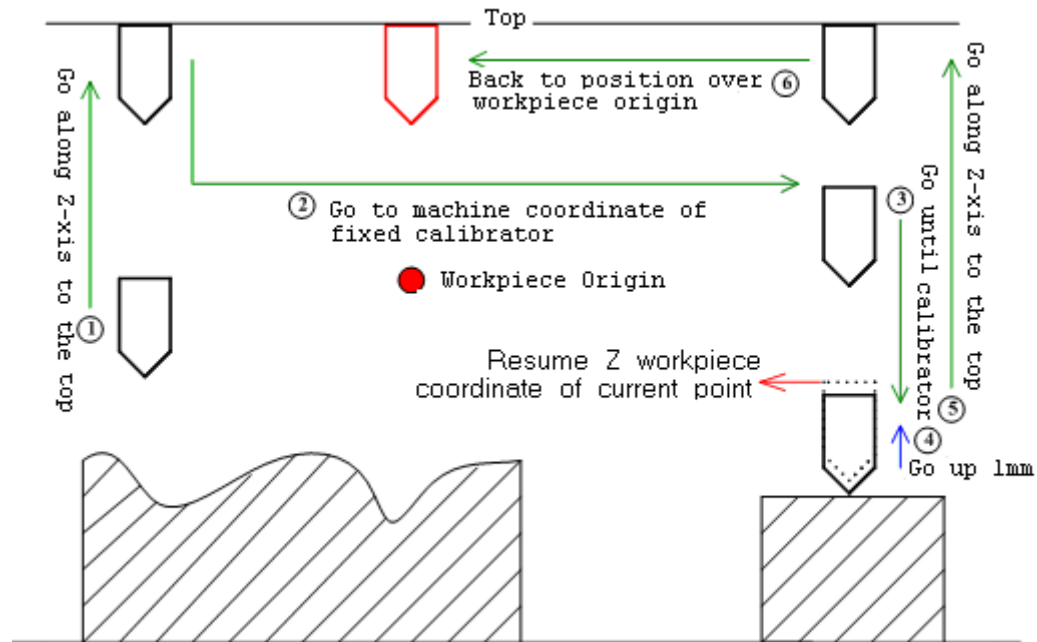
Fig. 4-32 Calibration after Tool Change

4.10   Machining Track Window

When the machine tool implementing processing procedure or simulating, the Machining trace window can follow the cutter-processing track real-time. The ability of three-dimensional real-time showing of following processing track can check the cutter's track more visually to ensure the processing correct.

The trace window adopts three-dimensional view mode. You choose the menu of "View/Customize…" to set it characteristically. You can refer to forward section "Customize parameters".

This window provides user abundant operation to examine the sketch from different angles and scaling.
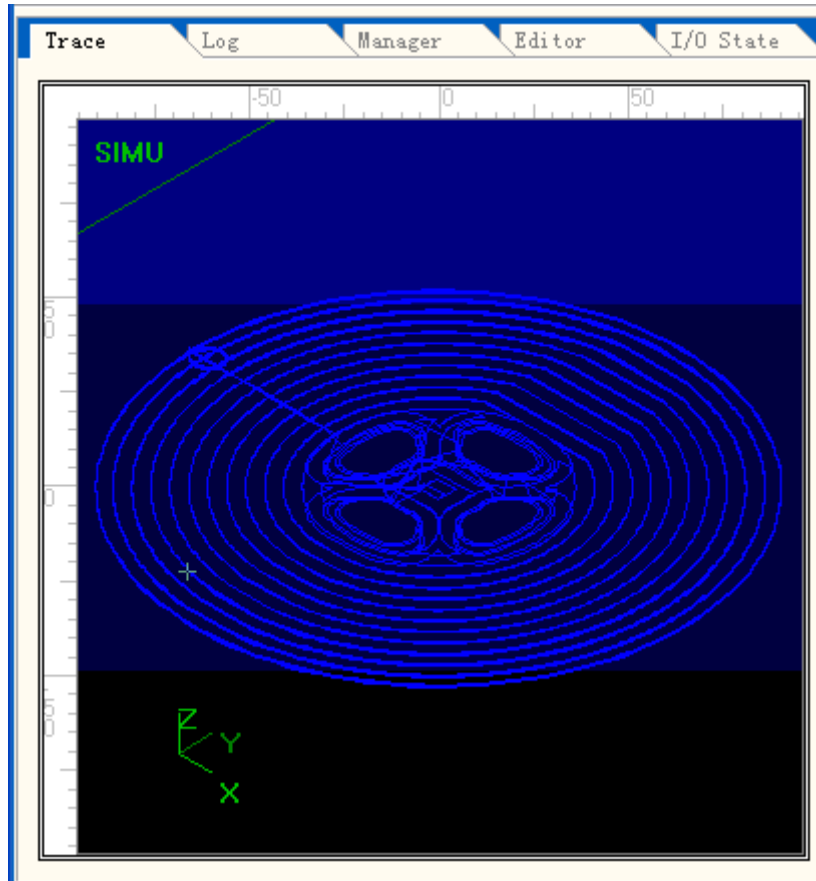
Fig. 4-33 Machining Track Window

Right click in the window, a shortcut window will pop up.

Fig. 4-34 Shortcut Menu of Machining Track Window

Clear function

After a long time of processing, the imitating figure becomes very complex and the temporary folder used to record processing track becomes more and more bulkiness. It will spend a lot of time in redrawing, moving, or revolving the imitating figure. That will need user to clean the tracking window.

There are many methods to clear the tracking window. You can complete it by menu, toolbar button, hot key, and the keyboard.

Menu or toolbar method: choose the menu of "Edit| Clear View", or choose the relative button on the toolbar. The corresponding mark of this button on the tool column shows as:

Hot key method: Press down the key of "CTRL+ DEL" to realize the clearance of tracking window at anytime.

Mouse method: Moving the mouse into the manifestation area, when the mouse shows or , clicking the right key of the mouse, then it will flip out a context menu, chose the option of "Clear".

Keyboard method: Press the key of DEL while the trace window is current active window.

Moving function

Mouse method:

Click the right key of the mouse in the window, then it will flip out a context menu, choose the option of "Move". The mouse shows . Clicking the left key of the mouse, the mouse shows . Press down the left key of the mouse and drag the mouse slowly, the track will move with the mouse.

Keyboard method:

When the track window is activated, use the four directions key in keyboard to move the track.

Zoom function

You can use three methods like the menu, keyboard, and the mouse added keyboard to realize the zoom of the following track figure.

Menu method: Choose" View| Zoom Ratio … "to set the fit scaling.

Mouse plus keyboard method: right click in the window, select "zoom in" or "zoom out" in the shortcut menu, you can switch between and .

> Prompt:
>
> When using mouse to zoom the window, part of the window can be zoomed in or zoomed

out, user only needs to use mouse to select rectangle zone.

When the mouse is enlarged to the limit, it will turn into "zoom out" function.

When the mouse is deflated to the limit, it will turn into "zoom in" function.

Keyboard method: activate this window first, then choose the key ("+" "−") on small keyboard to realize zoom.

Note:

On the main keyboard the key of "+" and "-" are invalid.

Center view function

Show current machining scope in the center of the window.

Menu method: Choose "View︱ Center View" to perform.

Mouse method: right click in the window, a context menu will flip out, choose" in the center" option.

Keyboard method: first, activate this window, press "Home" key on the keyboard.

Adjust to the window size function

This window can show all track, and customer can see the entire content of the track without rolling over the scroll bar.

Menu method: Choose "View︱Fit to window".

Mouse method: right click in the window, a context menu will flip out, choose" Fit to window" option

Keyboard method: first, activate this window, press "*" key on the keyboard.

Show current point function

The function will show current machining point in the center of track window.

Menu method: Choose "View︱Show current point " to perform.

Mouse method: right click in the window, a context menu will flip out, choose" Show current point" option.

Keyboard method: first, activate this window, choose "End" key on the keyboard.

Customize parameters.

To activated the function, use can choose "View︱Show current point " to perform, or right click in the window, a context menu will flip out, choose" Customize…" option.

A dialog will flip out. This dialog is used for customized setting of the track window and to realize self-define of trace mode and color.
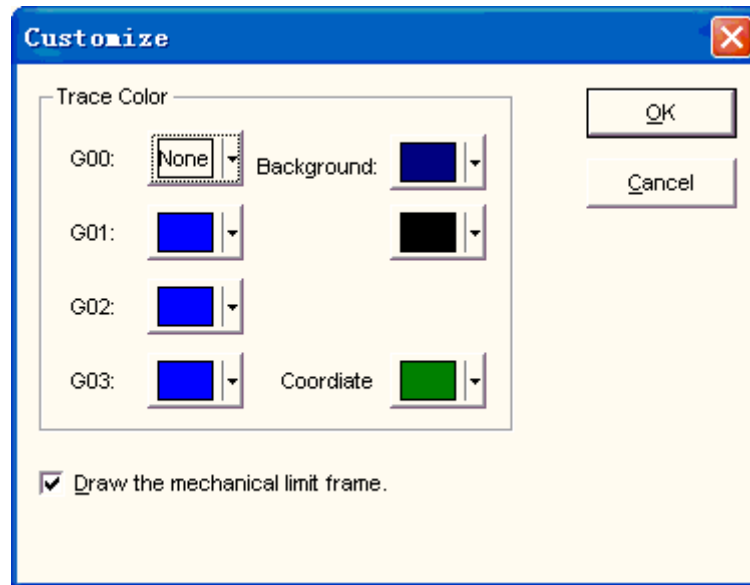
Fig. 4-35 Customize Dialogue

Click each color button, you will see the following color selection box.
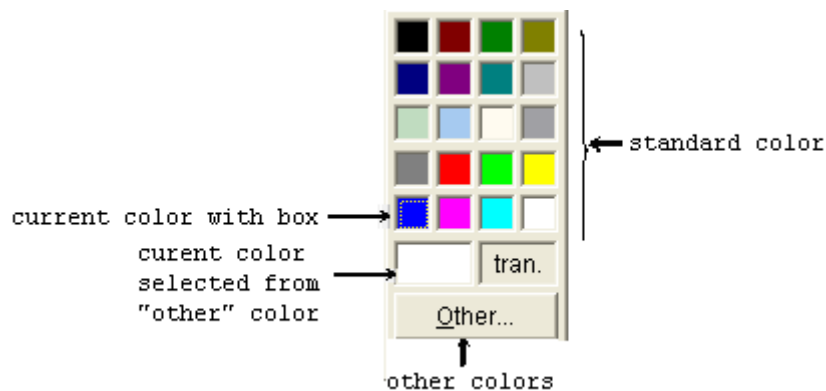
Fig. 4-36 Color Selection Dialog

> **Tracking color**

In the track window, you can allocate each different display elements respectively. Include:

1) Color of Ｇ00 instruction: This color indicates the color of G00 instruction's track.

2) Color of Ｇ01 instruction: This color indicates the color of G01 instruction's track.

3) Color of Ｇ02 instruction: This color indicates the color of G02 instruction's track

4) Color of Ｇ03 instruction: This color indicates the color of G03 instruction's track

Another, feeding by manual mode is shown in color of G01.

> **Background color**

Background color of track window has two kinds of colors, user can collocate two kinds of colors and realize two kinds of colors changing gradually.

> **Coordinate color**

It is the color of the coordinate and worktable boundary shown in the Track window.

> Prompt:
>
> If the color of the command you selects is transparent, the command will be concealled. If
>
> the background color is set as transparent, the track window will not be rightly shown, so
>
> user must be serious to select "transrarent" color.

View funciton

System provides 9 kindsof view function; you can switch it quickly by the number key on the small keyboard.

>	**Front View**	**number key 5**
>	**Top View**	**number key 8**
>	**Bottom View**	**number key 2**
>	**Left View**	**number key 4**
>	**Right View**	**number key 6**
>	**Southwest View**	**number key 1**
>	**Northwest View**	**number key 7**
>	**Southeast View**	**number key 3**
>	**Northeast View**	**number key 9**

Revolving function

It can be only completed by keyboard combination. (Alt + direction keyboard). At the same time of pressing the key of ALT, press down the direction key.

4.11   System log window

This window records the important operation and happened events, from it not only can you browse the log information that is happened from this start, but also you can review the historical records.
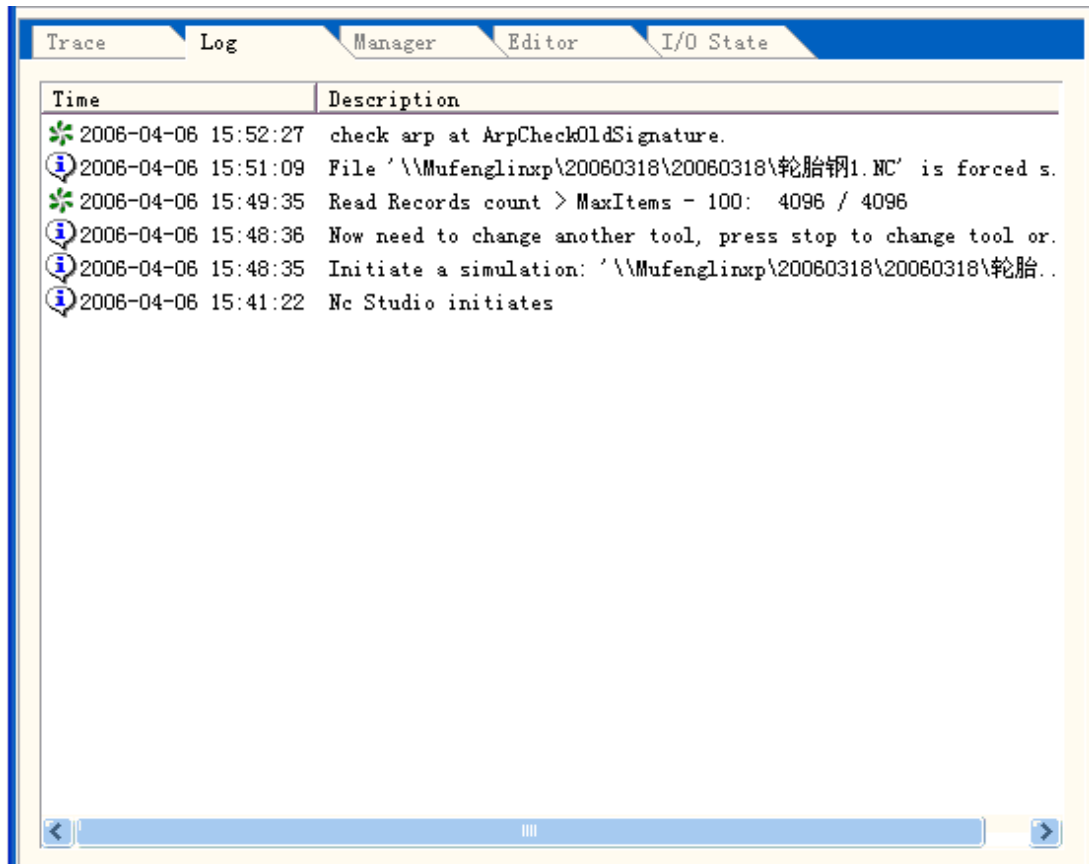
Fig. 4-37 System Log Window

The log information of current record in system includes:

1) Start and close.

2) Information of the beginning automatically and ending in processing automatically.

3) Change of the working coordinates.

4) Alarm information of the system.

5) Other system information.

Click the window with the right key of mouse, the context menu will be flipped out.
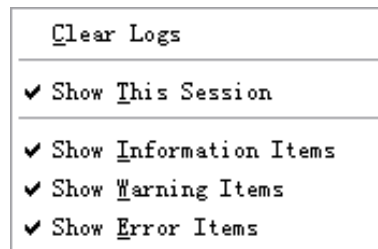
Fig. 4-38 Context Menu of System Log Window

➢ **Log clear function**

Delete current log record in the system log window.

Menu method: Choose" Edit| Clear log ".

Mouse method: right click in system log window, a context menu will flip out, choose "Clear log" option.

> Note:
>
> Remember to clear the system log record periodically, otherwise the file of the system log record will be too big and may influence performance and response time of the system.

If the following option is selected, a symbol " √ " will appear in the front of the option; select the item once again, symbol " √ " will disappear, which indicates the option is unselected.

➢ **Show this session**

Right click in the log window, a context menu will flip out, choose "Show this session" option, and also can choose menu "View |Show this session's logs" to activate the function.

Activate the function, customer can look into log records of this session only; If not choose this function, user can look into not only log records of this session but also previous log records.

➢ **Show information items**

Right click in the log window, a context menu will flip out, choose "Show information items" option, and also can choose menu "View |Show information items" to activate the function.

Information log has symbol in front.

Activate the function, customer will see logs about system start, system exit, and so on. If not choose this function, information log items will be concealed.

➢ **Show warning item**

Right click in the log window, a context menu will flip out, choose "Show warning items" option, and also can choose menu "View |Show warning items" to activate the function.

Warning log has symbol in front.

Activate the function, customer will see warning log items. If not choose this function, warning log items will be concealed.

➢ **Show error item**

Right click in the log window, a context menu will flip out, choose "Show error items" option, and also can choose menu "View |Show error items" to activate the function.

Error log has symbol in front.

Activate the function, customer will see error log items. If not choose this function, error log items will be concealed.

> Prompt:

System log has mark �֍ in front.

4.12　Procedure Manager Window

Procedure manager window is mainly used for the management of processing file. Customer only needs to save procedure files to a fixed catalogue; then NcStudio can manage the files in the procedure window. In this window, user can implement creating, calling, editing, delete, renaming, loading procedure file and etc. It is extremely convenient to user.
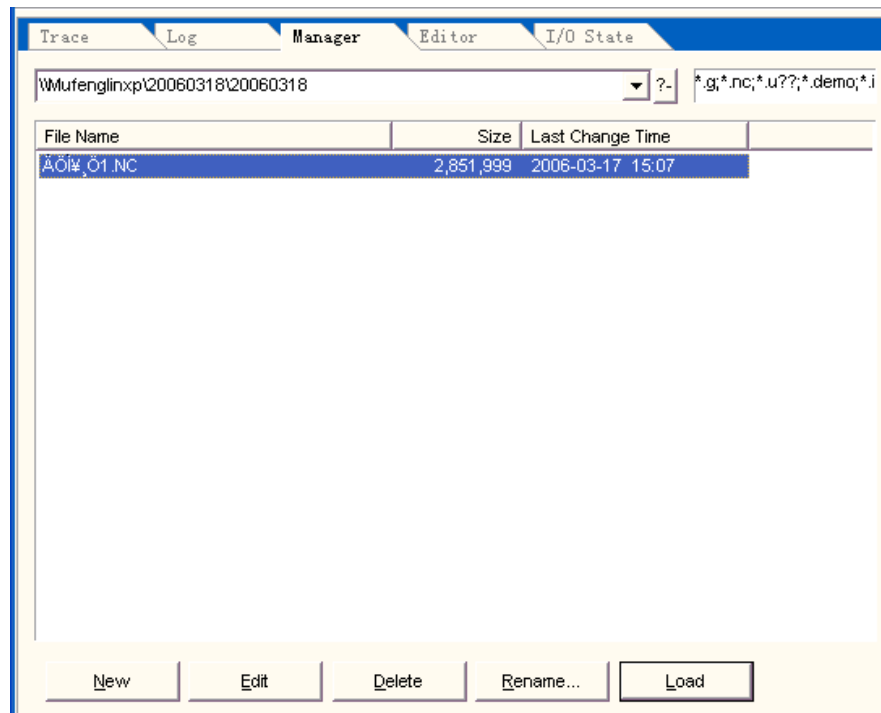
Fig. 4-39 Procedure Manager Window

Right click in procedure manager, a context menu will flip out.

Fig. 4-40 Context Menu of Procedure Manager Window

Create new procedure file

Three methods can be used to activate function of creat new procedure file.

Select menu file "file (F)｜New" (shortcut key Ctrl+N);

Right click in procedure manager window, and select "New" option in the context menu;

Click button "New" at the lower part of procedure manager window.

System will creat a new procedure file with name "Untitle1.nc", user can modify the saving position.

D:\NcFiles

Fig. 4-41

Click button ▣ to select new saving path for the file.

Moreover, customer can edit, delete, rename, load the new file, etc. For detailed information please refer to following chapter.

Open a procedure file existed

The procedure files listed in procedure manager window are files in current folder.

User can directly input specified path in box D:\NcFiles, or click button ▣ to select other path, file list box will show all procedure file in current folder.

The extension names of files in current folder are shown in "Extension Name box", user can delete or add extension name for the files. The corresponding files with specified extension name will be displayed in "file list box".

When the aim file is listed in "File List Box", double click it to select it as current file.
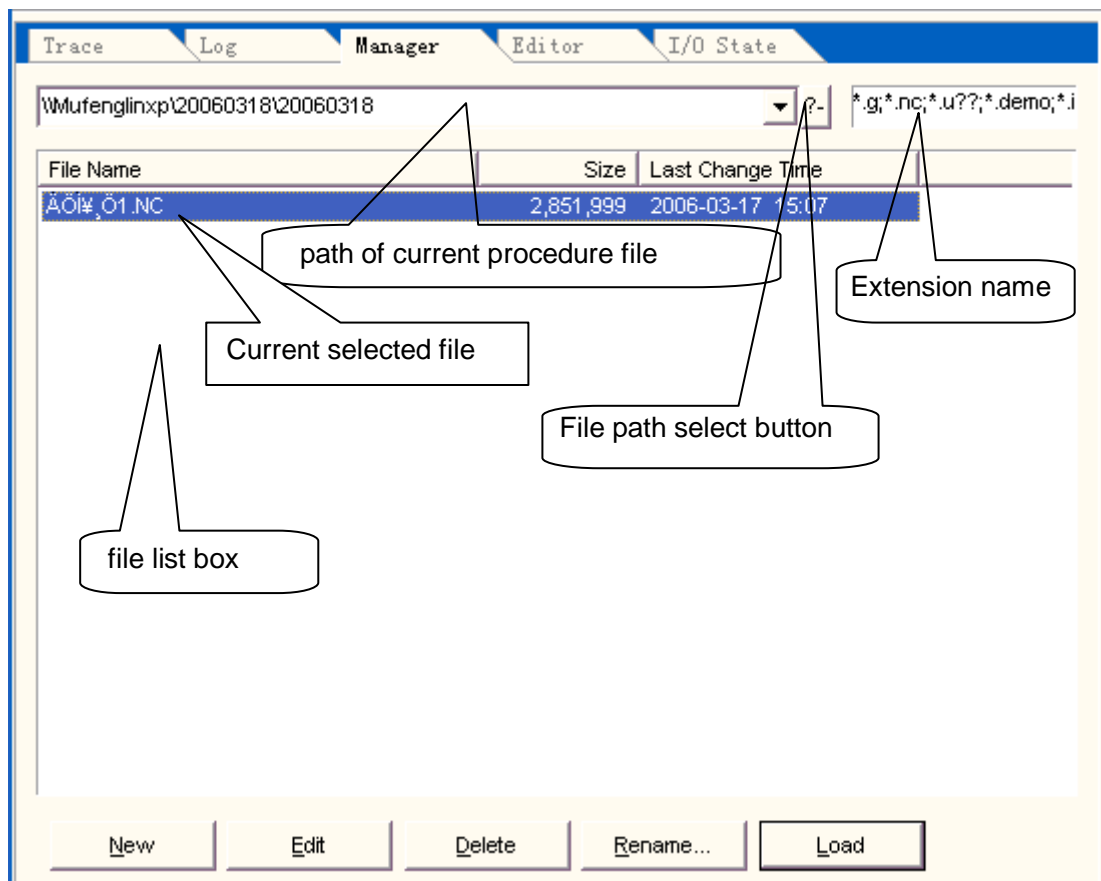
Fig. 4-42 Procedure Manager Window

What's more, user can use menu "File (F) ｜ Open and Load …" to open a procedure file.

Edit a procedure file

Three method can be used to edit a selected procedure file.

Method one: select menu "Edit｜Edit the selected file"；

Method two: right click in procedure window, and select "Edit" in the context menu;

Method three: click "Edit" button in the lower part of procedure window.

Then system will jump into "File Edit Window". For detailed information please refer to chapter 4.13.

Delete a procedure file

Three method can be used to delete a selected procedure file.

Method one: select menu "Edit｜delete the selected file"；

Method two: right click in procedure window, and select "Delete" in the context menu;

Method three: click "Delete" button in the lower part of procedure window.

Then a dialog will pop up, and ask if you confirm to delete the file and the file can't be edited or used by others.
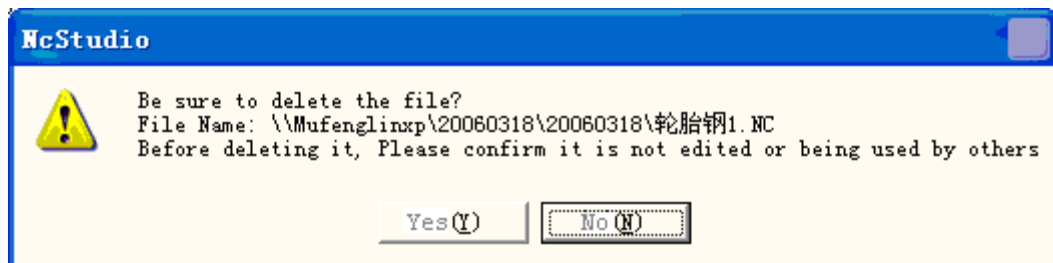
Fig. 4-43 Prompt Dialog of Delete Procedure File

Click "Yes" to delete the selected file.

> Prompt:
>
> The selected file in "file list box" is shown in deep blue.

Rename a procedure file

Three method can be used to rename a procedure file.

Method one: select menu "Edit｜rename the selected file"；

Method two: right click in procedure window, and select "Rename" in the context menu;

Method three: click "Rename" button in the lower part of procedure window.

Then the name of selected file can be renamed.

File Name	Size	Last Change Time
Untitle1.nc	3	2009-05-25 21:00
Untitle2.nc	3	2009-05-25 21:00
Untitle3.nc	3	2009-05-25 21:00
Untitle4.nc	3	2009-05-25 21:00

Fig. 4-44 Rename the Selected File

Enter a new file name, press "Enter" key or use mouse to click other blank position.

Load a procedure file

Three method can be used to rename a procedure file.

Method one: select menu "Edit︱open and Load …" ;

Method two: right click in procedure window, and select "Load" from the context menu;

Method three: click "Load" button in the lower part of procedure window.

Method four: click "open" button 📂 on the tool bar.

Method five: right click in auto operation window, and select "Open and Load" option from the shortcut menu.

4.13   Procedure Edit Window (Editor)

At the upper part of the editor window, there is the file name being edited. Above the edit window is the file name being edited. Underside is a standard editor window.

| Trace | Log | Manager | **Editor** | I/O State |

\\Mufenglinxp\20060318\20060318\轮胎钢1.NC

```
%
O0000
(PROGRAM NAME - 轮胎钢)
(DATE=DD-MM-YY - 17-03-06 TIME=HH:MM - 11:51)
N100G21
N102G0G17G40G49G80G90
(UNDEFINED TOOL - 1 DIA. OFF. - 1 LEN. - 1 DIA. - 10.)
N104T1M6
N106G0G90X-87.363Y-5.858Z0.S5000M3
N108G43H1Z5.
N110Z4.202
N112G1Z1.202F244.3
N114X-87.105Y-6.026Z1.186
N116X-86.837Y-6.179Z1.17
N118X-86.561Y-6.314Z1.154
N120X-86.276Y-6.432Z1.137
N122X-85.985Y-6.533Z1.121
N124X-85.688Y-6.615Z1.105
N126X-85.387Y-6.679Z1.089
N128X-85.082Y-6.724Z1.073
N130X-84.776Y-6.751Z1.057
N132X-84.468Y-6.758Z1.041
```

Fig. 4-45 Editor Window

This window has a powerful editing ability. In this window, you can edit any complex procedure as big as 1000M (1G).

Clicks the right key of mouse in the editor window, system will flip out a context menu, the options in the menu is the most in common use, customer can choose the option according to own demand, so we will not give unnecessary details.

Undo	Ctrl+Z
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Select All	Ctrl+A
Find...	Ctrl+F
Find the Next	F3
Repeat...	Ctrl+H
Save and Load	
Save	
Close	

4.14   Input / Output (I/O) State Window

Input/output state window displays current states of I/O, which is very helpful for system monitoring and malfunction checking.

	PortName	Pin	P...	PortAddr	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)	J1-25	N	6	Program Start
●	IN25 (Z0)	J1-25	N	7	Reference Point of Z-axis
●	IN31 (Y0)	J1-6	N	14	Reference Point of Y-axis
●	IN30 (X0)	J1-24	N	15	Reference Point 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. 4-46 I/O Sate Window

Attention:

Content of the window may vary with different software versions. What we provide here is

only for your reference.

Symbol before each port means:

🟢: Green solid point means that there is signal input at this interface.

🔴: Red solid point means that there is no signal input at this interface.

🟢: Green hollow point means that there is signal output at this interface.

🔴: Red hollow point means that there is no signal output at this interface.

5 NcStudio™ Menu System

5.1 "File" menu

This menu includes order options used to operate file.

```
Open and Load...      Ctrl+O
Unload                Ctrl+U

New                   Ctrl+N
Open and Edit...      Ctrl+E
Edit Loaded File      Ctrl+P
Save                  Ctrl+S

Save as...
Save and Load
Close
Config                      ▶

Recent Loaded File          ▶
Recent Edited File          ▶

Exit
```

Fig. 5-1   "File" Menu

Open and load

The 'open and load' menu item whose shortcut key is "CTRL+O", is used to open procedure file existed in disk and to popup the dialog box of "Open and Load" as shown in the following figure:
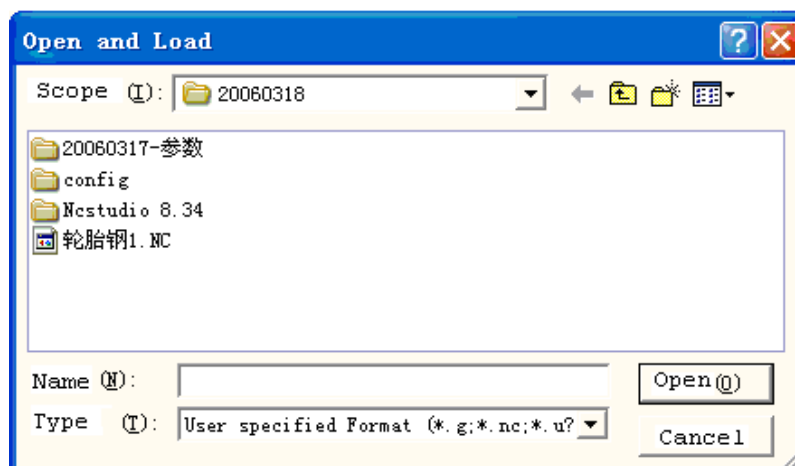
Fig. 5-2 Open and Load Dialogue

Click the rear part of button **Scope (I): 📁 20060318 ▼** to review processing file under other path; select the file and click button "Open", and then the toolbar of system main window will show the file names for current processing procedures.

Unload

Unload the current processing procedure.

New

It is used to create a new processing procedure.

Select this option and the procedure edit window will be active to allow user edit the procedure at this window; click with the right key of mouse to popup a context menu, and then select "save" to complete.

Open and Edit

This menu item is used to open procedure file that saved in disk and to load it into edit window where can edit a file.

Edit Loaded File

The function is used to edit processing procedure loaded currently.

Save current workpiece origin to file

Save current workpiece origin to the procedure being used.

Save

The function is used to save processing procedure edited.

Save As

This function can save the current procedure files in the editor window into disk with another name.
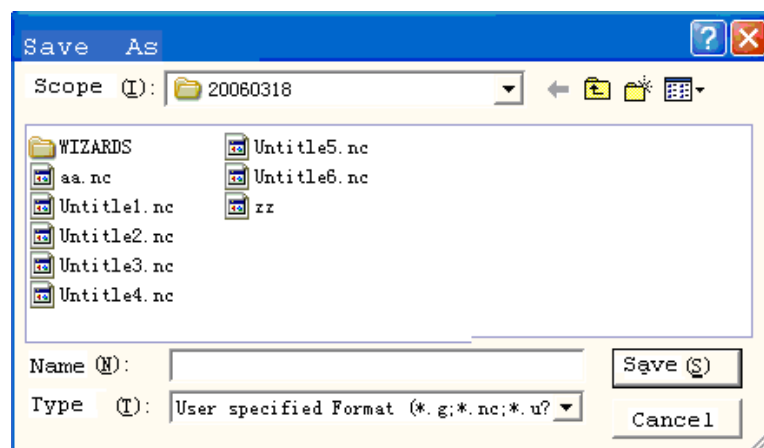
Fig. 5-3 'Save As' Dialog

Save and load

The function is used to save and load the procedure file edited currently into current processing procedure.

Close

The function is used to close the current procedure files when being edited.

Configuration
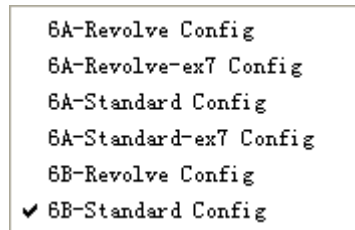
Fig. 5-4 Configuration Options

The user can select corresponding configuration as required. For detailed configuration, please refer to Manufacturer Manual.

Recent Loaded Processing Procedure

This menu pops out a submenu which shows the name of processed files which were loaded recently. If user wants to load any file again, clicking the name of the document directly will load it quickly.

Recent Edited Processing Procedure

This menu pops out a submenu which shows the name of processed procedure which were edited recently. If user wants to edit again, clicking the name of the document directly will load it quickly.

Exit

It is used to close Ncstudio^{TM} system.

If the processing procedure is edited by user and is not saved, a dialog box shall popup for saving or not. If you want to save, click "Yes", otherwise, click "No", and if you want to exit the function, click "Cancel". If the user is carrying out automatic processing, the system will prompt the user to end the current processing task and then exit the system.

5.2 "Edit" Menu

Edit Menu When Processing Trace Window Activated

Fig. 5-5 Edit Menu 1

Clear View

Refer to "Clear View" in chapter 4.10 Processing Trace Window.

Array Machining

The function is used to have array machining for the same processing procedure. The following dialog box will popup by selecting the option:
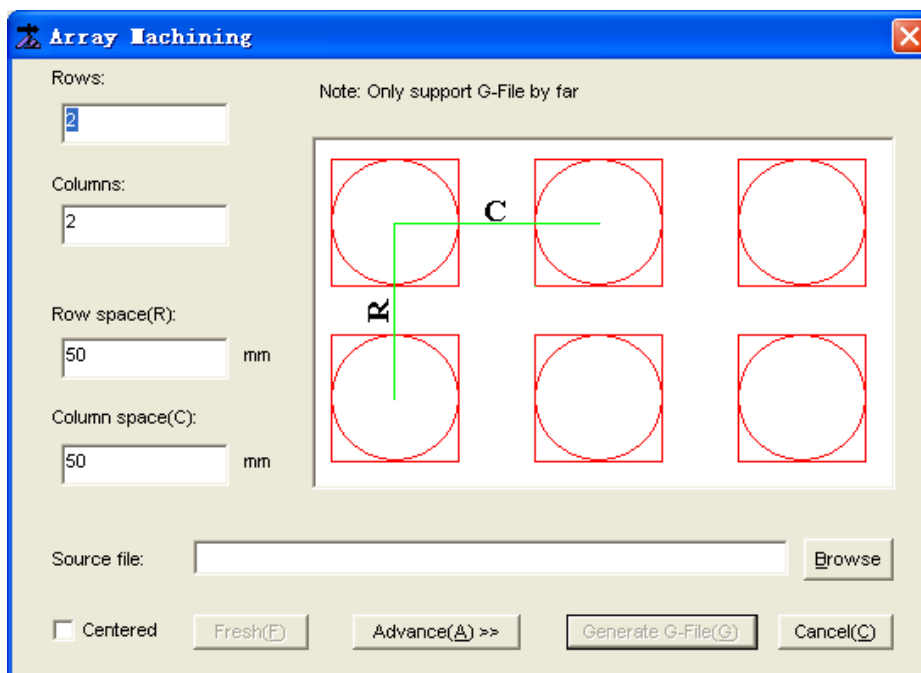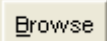
Fig. 5-6 Array Machine Dialog

Click browse button [Browse] to select array machining procedure as required.

Set column and row number, row space (the spacing between two origins of workpieces as the R length in the above mentioned figure), column space (the spacing between two origins of workpieces as the C length in the above mentioned figure), and click the button "Generate Processing File" to complete arraying; the new generated processing file will be load into NcStudio system automatically.

Click the button "Advanced" to set different row space and column space and the corresponding dialog box is shown as below:
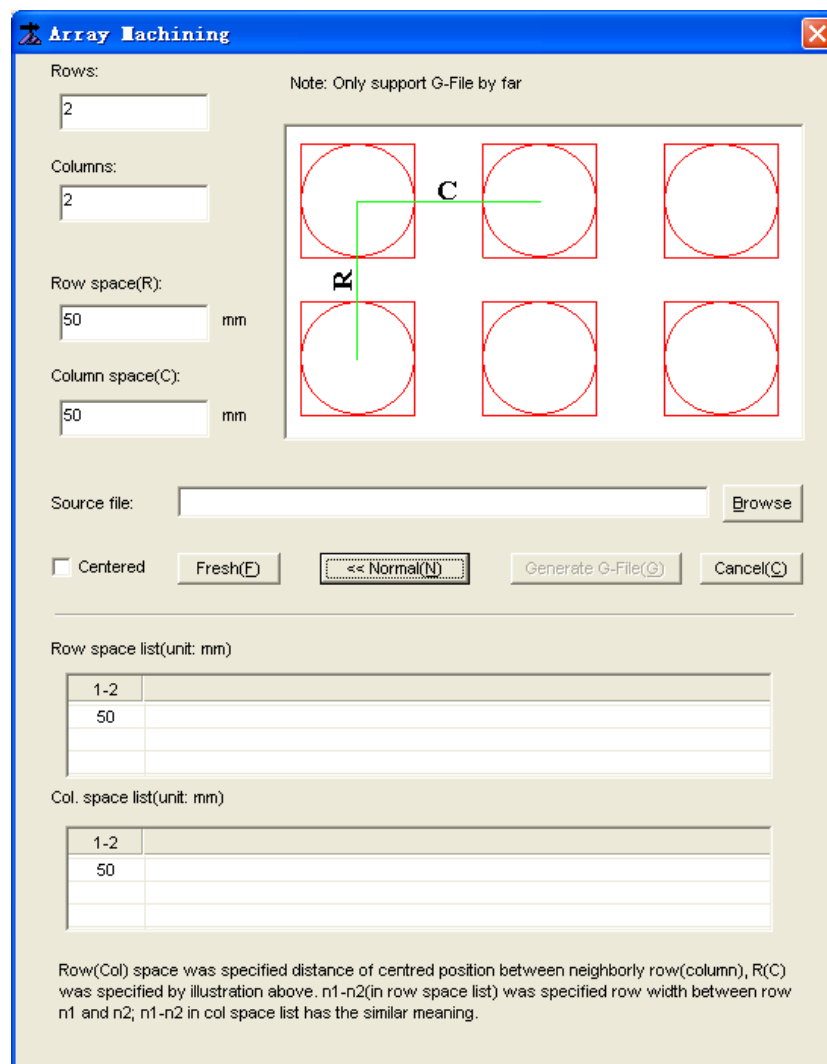
Fig. 5-7 Array Machine

"Row Space Table" and "Column Space Table" is used to set the space for row and column respectively.

Edit Menu When System Log Window Activated

```
Undo                              Ctrl+Z

Cut                               Ctrl+X
Copy                              Ctrl+C
Paste                             Ctrl+V
Select All                        Ctrl+A

Find...                           Ctrl+F
Find the Next                     F3
Repeat...                         Ctrl+H

Clear Log

Array Machining...
Mirror Rotate Machining Setting
```

Fig. 5-8 Edit Menu 2

Clear Log

Refer to "Clear Log" in article 4.11 System Log Window.

Array Machine

Refer to "Array Machine" in "Edit Menu When Processing trace Window Activated".

Edit Menu When Procedure Management Window Activated

```
Undo                              Ctrl+Z

Cut                               Ctrl+X
Copy                              Ctrl+C
Paste                             Ctrl+V
Select All                        Ctrl+A

Find...                           Ctrl+F
Find the Next                     F3
Repeat...                         Ctrl+H

New
Edit the Selected File

Delete the Selected File
Rename the Selected File

Load the Selected File

Array Machining...
Mirror Rotate Machining Setting
```

Fig. 5-9 Edit Menu 3

Please refer to the introduction of Procedure Management Window in 4.12.

Edit Menu When Procedure Edit Window and I/O Status Window Activated

Fig. 5-10 Edit Menu 4

Array Machine

Refer to "Array Machine" in chapter "Edit Menu When Processing Trace Window activated".

5.3 "View" menu

The items in "View" menu will be changed along with the current active window of the second window (including processing track, system log, procedure management, procedure edit and I/O state window).

"View" Menu When Procedure Management, Procedure Edit and I/O Status Window Activated

Fig. 5-11 "View" Menu 1

Show File Line Number

The function is used to show or hide the processing procedure row number in automatic window. The function is affective only when automatic window is activated.

Trace Current line number

The function is used to trace the line number of current procedure in automatic window

during processing.

File Information

Click "File Information" to popup the following dialog box:
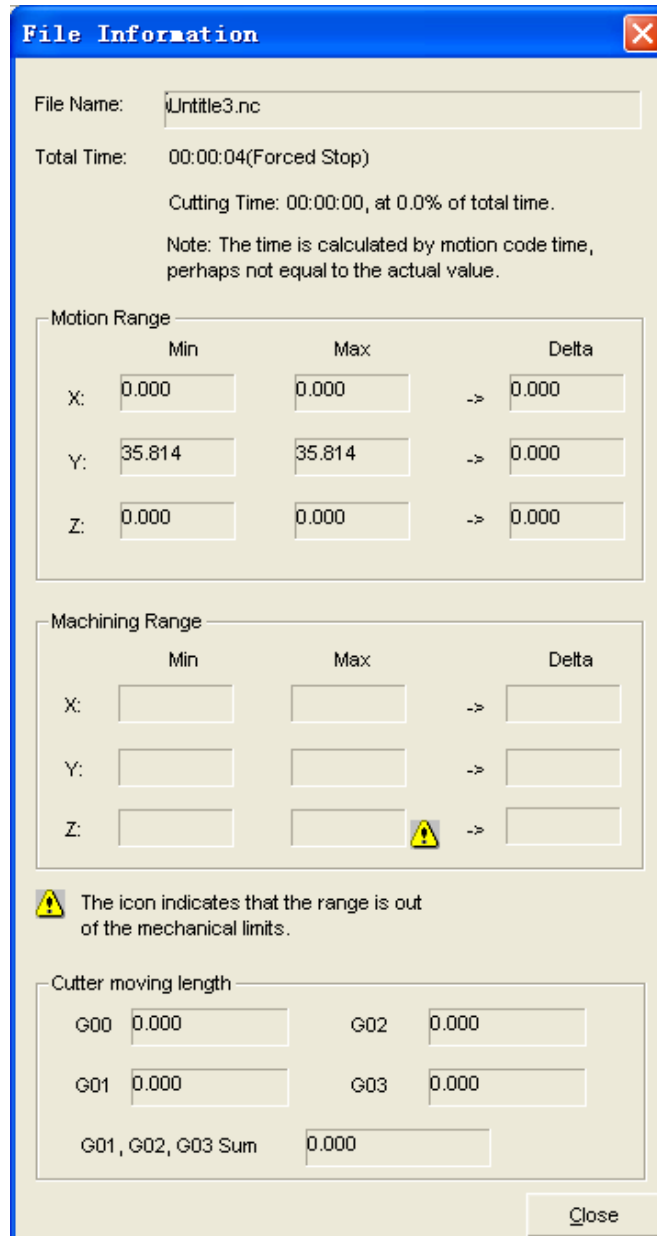
Fig. 5-12 Processing Procedure Statistical Information

The dialog box shows the statistic information in automatic processing, such as the processing time, processing range etc. It would be know exactly and quickly when combining with the function of simulation.

Total Time

Statistic for total machine tool processing time and for machine tool cutting time

Motion Range

The motion range means the max and min workpiece coordinate values of machine tool in processing cycle.

Machining Range

The Machining scope means the max and min workpiece coordinate values of machine in actual cutting cycle.

"View" Menu When Processing Track Window Activated

```
✔ Tool Bar
✔ Status Bar
─────────────────────────
✔ Show File Line Number
✔ Trace Current Line
─────────────────────────
  File Information...
─────────────────────────
  Zoom Out             Num+
  Zoom In              Num-
  Center View          Home
  Fit to Window        Num*
  Show Current Point   End
  Ratio...
  Customize...
─────────────────────────
  Front View           Num5
  Top View             Num8
  Bottom View          Num2
  Left View            Num4
  Right View           Num6
  Southwest View       Num1
  Northwest View       Num7
  Southeast View       Num3
  Northeast            Num9
```

Fig. 5-13 Menu 2

Zoom out, zoom in, center view, fit to window size, show current point, show ratio, customize, front view, top view, bottom view, left view, right view, southwest view, northwest view, southeast view, northeast view.

For above mentioned items, please refer to the introduction of processing track window in section 4.10.

"View" Menu When System Log Window Activated
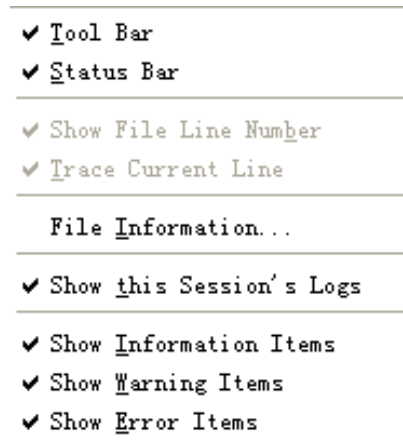
Fig. 5-14 Menu 3

Show this session's logs, show information items, show warning items, and show error items

Please refer to the introduction of log information window in section 4.11.

5.4 "Operation" menu

"Operation" menu consists of the following items:

Fig. 5-15 "Operation" Menu

Single Block

While single stepping mode was selected, operating one order each time would be to the Pause mode. User has to click "start" button ▶ to execute the next order to continue. When next order have finished, the system will be to the Pause mode again.

User can set the processing assignment as a single step mode in order to provide a fine support for diagnosis mistake and malfunction restoring.

MPG Guide/Handwheel Gear

When it is operated with handwheel guide under auto mode, user shall click button ▶ for machining, then the system will execute the processing procedure as the rotation of handwheel. Therefore the procedure processing will stop when the handwheel is stopped and the processing speed varies along with the speed of handwheel.

User can select the function to help the user to judge that processing procedure is correct prior to carrying out processing.

Set Workpiece Coordinate Offset

Select the option to popup the following dialog box:
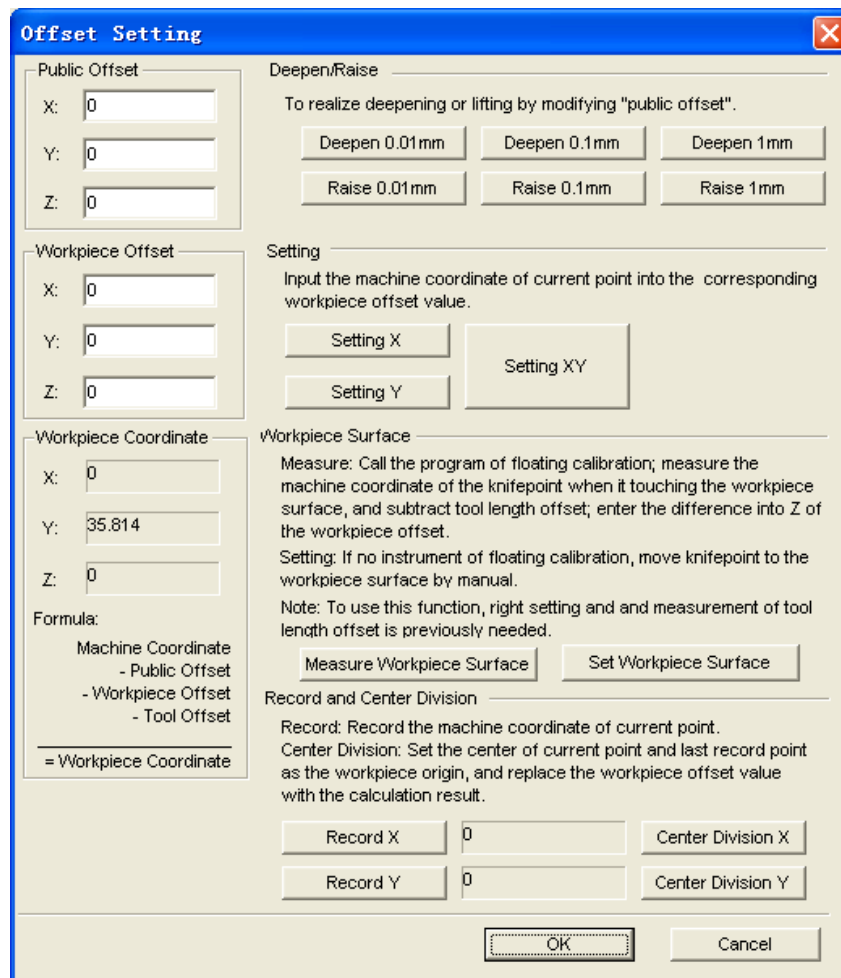
Fig. 5-16 Set Workpiece Coordinate Offset

Public offset: Public Offset: it's used to record the temporary adjustment value to the workpiece origin. The value can only be revised artificially, not automatically. Whether Public offset exists or not can't cause any difference to fixed calibration or floating calibration. That's to say, if public offset exists, after floating calibration finished, the coordinate of workpiece surface are not zero.

For example: if outer offset is set as 10 before calibration, then the outer offset is still be 10 after calibration, system workpiece coordinate is -10.
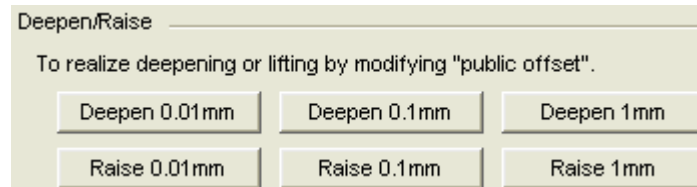
Fig. 5-17

Click one of the buttons to revise the public offset of Z-axis, and the workpiece origin of Z-axis will deepen or rise a specified distance, and form a new workpiece coordinate system; if click a button several times, the deepening or raising distance is the accumulated value of several time value.

Workpiece offset: that is the machine coordinate of workpiece origin.
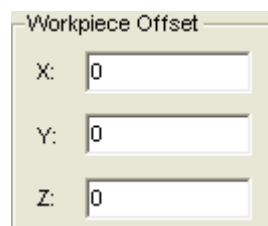
Fig. 5-18

Workpiece offset displays the machine coordinate od workpiece origin. You can also input number to set machine coordinate of workpiece origin. (we don't recommand user to revise the value manually)
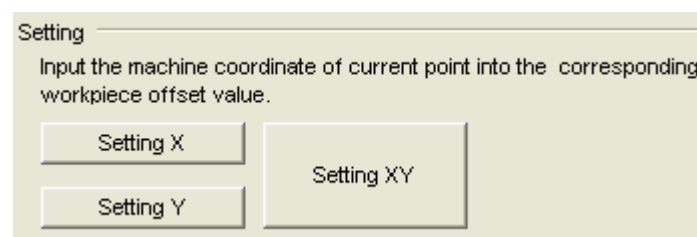
Fig. 5-19

Buttons in Fig.5-19 are used to set current point as workpiece origin.
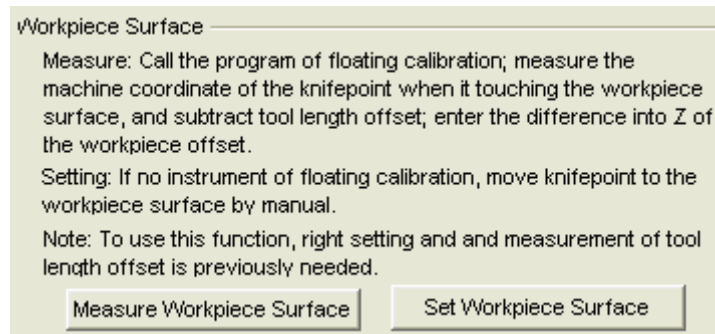
Fig. 5-20

Measure workpiece surface is the same as in chapter 4.9.

Set workpiece surface: set current z coordinate as workpiece origin.
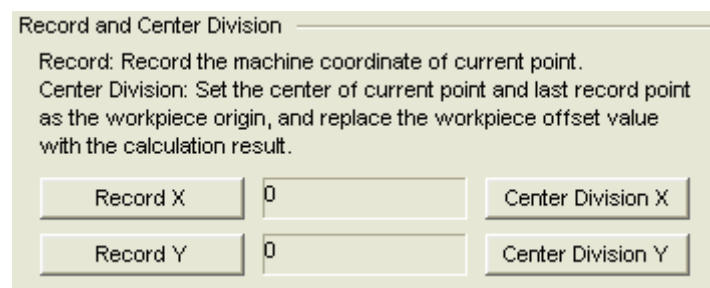
Fig. 5-21

Record and center division is used to get the center of two points. When tool goes to the first point, click button "Record X", system will record the X machine coordinate of the point; when tool goes to the second point, click button "Center Division X", system will calculate the X machine coordinate of the two point center.

"Record Y" and "Center Division Y" are the same with "Record X" and "Center Division X".

Right click in the NC state bar, the "offset setting" dialog will also appear.

Back to Workpiece Origin

Select "Set Workpiece Origin" and Z axis will rise up to safety height when the tool point is lower than safety height and then the X and Y axis will be returned to workpiece origin together firstly and then Z axis will be lower to safety height.

Users also can select the button which located in the Tool Bar to realize this function.

Save the Current Workpiece Origin

This function is used to save the current workpiece origin into the file of processing system. User can save the workpiece origin as well as save procedure name together to avoid procedure confusion after several times of workpiece origin saving and to help the operator to find the saved workpiece origins. The function can save ten coordinate data totally.
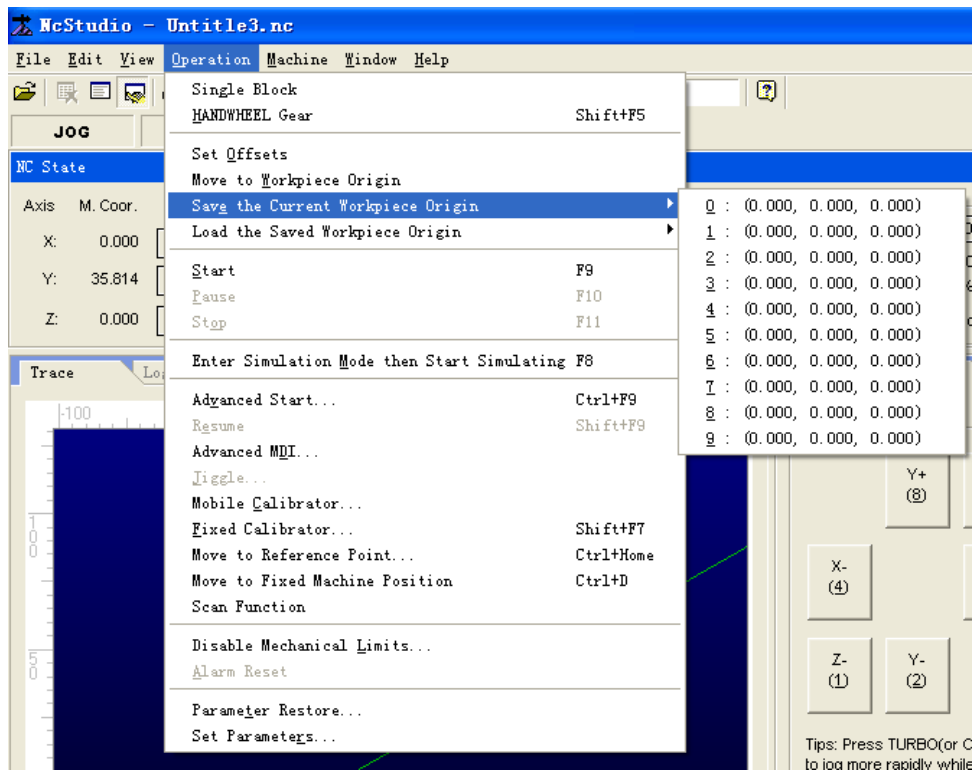
Fig. 5-22 (Save the Current Workpiece Origin)

Load the Saved Workpiece Origin

The function is used to read coordinate value of the saved workpiece origin.

Start

Click Start option and the system will enter into automatic processing mode. If the system is under simulation status, it will execute the processing procedure with simulation mode.

User also can choose the button ▶ on the tool bar to operate this function.

Pause

In automatic processing, selecting the menu item of "pause", the machine tool processing will pause and raise the cutter then turn into the state of "Auto | Pause". If you want to keep on processing, you can select "Start".

If the system is in the state of simulation it will be paused by selecting Pause. And if you want to keep on simulating, you can select Start.

User also can select the button ‖ on the tool bar to operate this function.

Stop

In automatic processing, selecting this menu, the machine tool processing will stop and raise the cutter, and then the system will terminate the whole processing task, and enter

into the state of "Auto | Idle". This method is to break off the processing procedure normally in processing.

If the system is under simulation status, the system will be paused and enter to the status of "Auto | Idle" after selecting the menu of "stop". If the user wants to re-simulate, you can continue by selecting the options "start", "Advanced start", "continue from the breakpoint" and so on.

User also can select the button ■ located on the tool bar to operate this function.

Enter Simulation Mode and Start Simulating

Select this menu, machine tool will execute simulating in high speed automatically from the first row which the function similar to the display function of CNC but priority to.

Running under the mode of simulating, the system will not drive the machine tool to do the relative actions but to show the processing trace of the cutter in high speed in the trace window. By simulating, user can foresee that the moving form the machine tool will do to avoid damaging the machine tool when program processing procedure. By simulating you can also know some additional information.

Once the simulation started, the menu was replaced by "Stop Simulating and Leave Simulation Mode". If you execute this function, simulating will be terminated immediately.

User also can select the button located on the tool bar to operate this function.

Advanced Start

Selecting this function, the system will pop out a dialog box of "Start (with advanced options)", as showed in below.
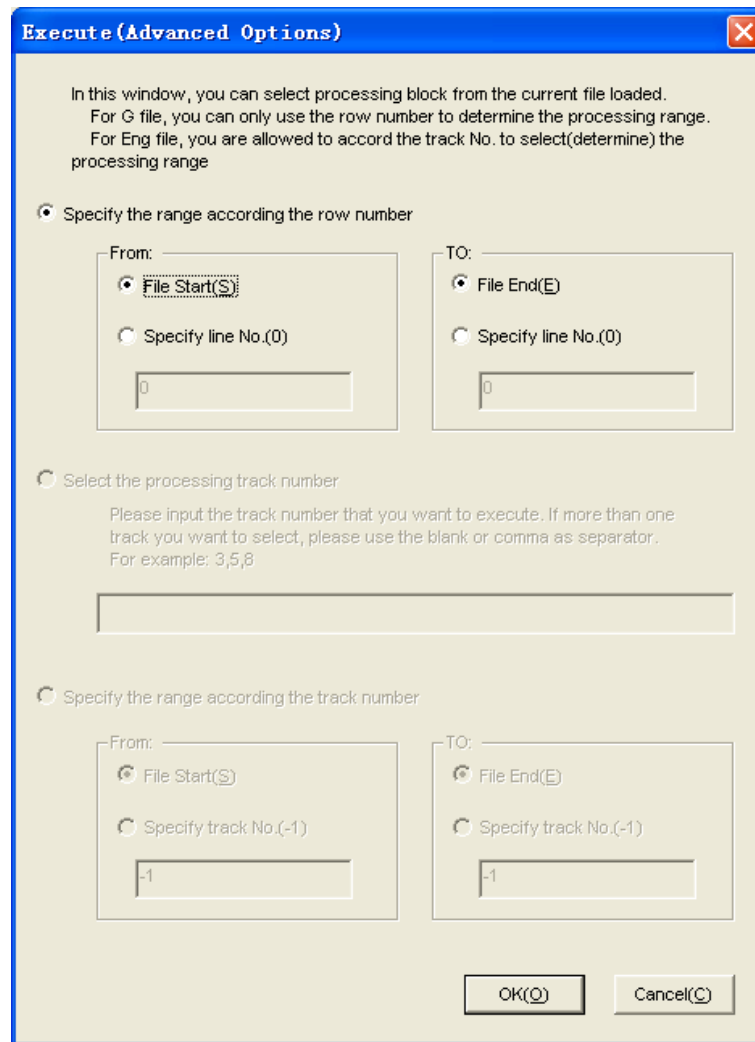
Fig. 5-23 Advanced Start

The function is used to execute any selected produre block. User can select processing range in the dialog.

Specify Range by Row Number
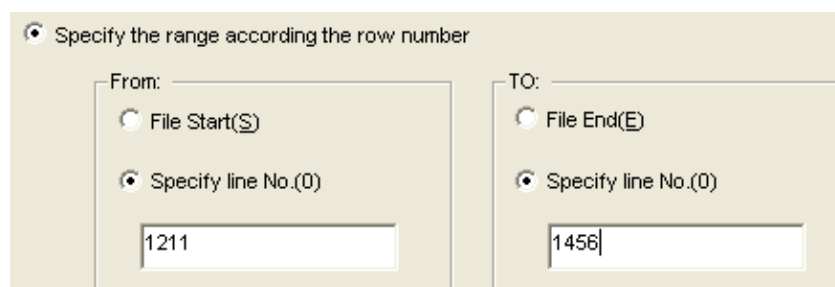
User can select any section for machining in the program according to row number.

Select Track Number
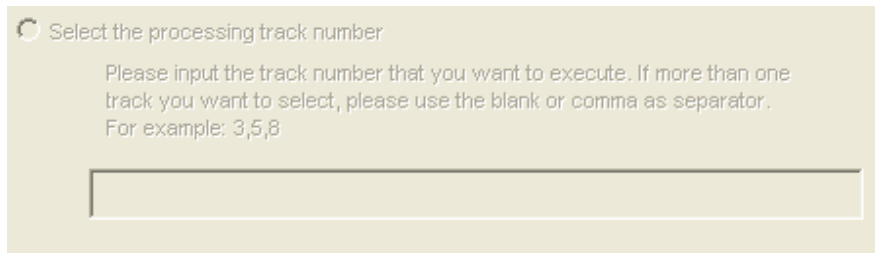
User can select any section for machining in the program according to processing path number.

Specify the range according to the track number
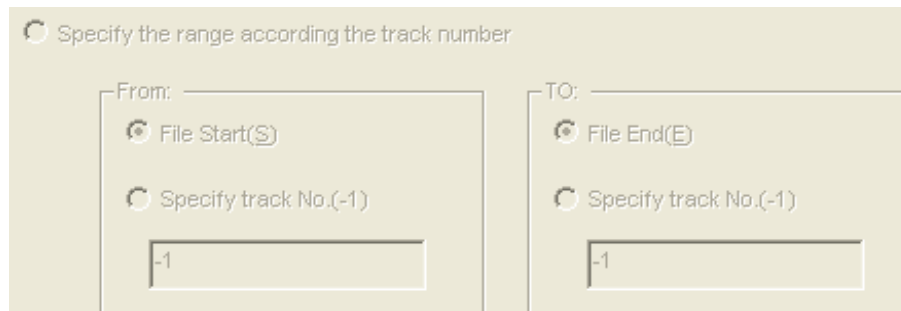
User can select any section for machining in the program by assigning processing path range.

Breakpoint Resume

When you execute this function, system will execute from the breakpoint that is broken off last time.

User can select the function after unexpected power off or emergency stop and then the machine tool will move to breakpoint rapidly for processing again which will save user's processing time.

User also can select the button ![button] located on the tool bar to operate this function

Execute Processing Instruction

The system will pop out the "Advanced MDI" dialogue box when the option is selected. The dialogue includes "Rectangle Mill", "Rectangle Frame Mill", "Round Mill", "Round Frame Mill" and "MDI". As long as customer input some parameters in the previous four windows, user can complete the function of milling or drawing.

Rectangle Mill Window
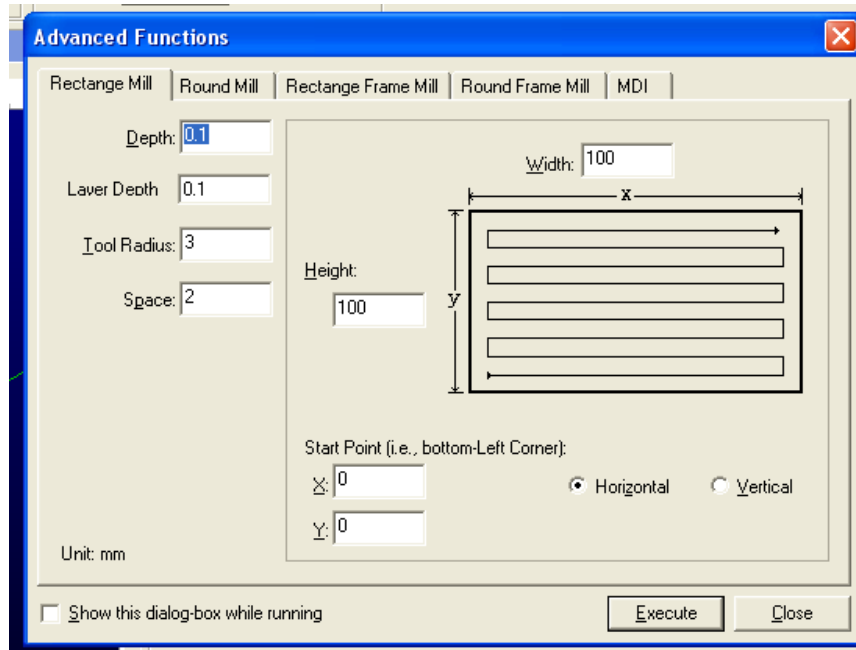
Fig. 5-24 Rectangle Mill Window

Round Mill Window
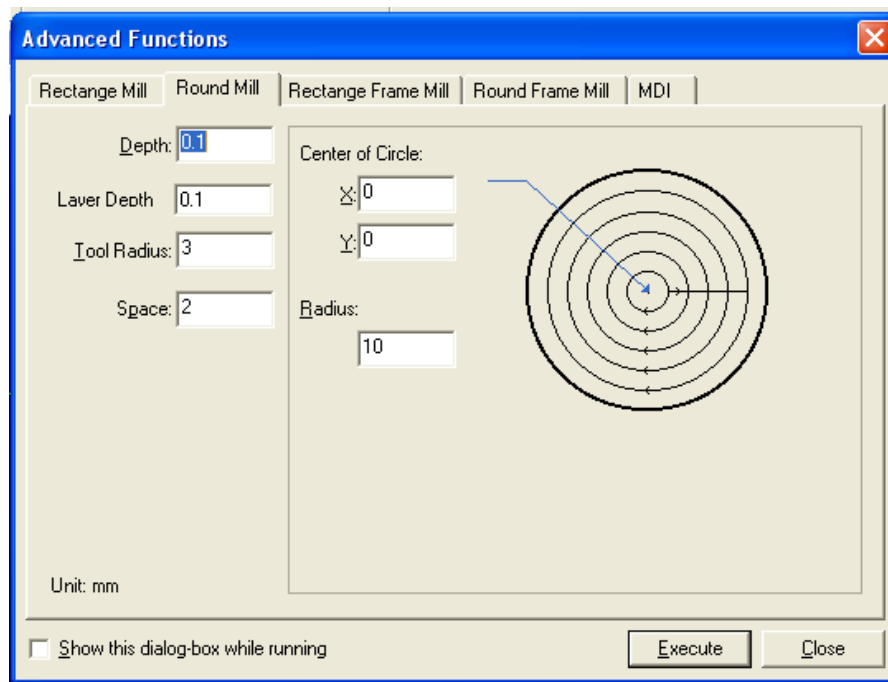
Fig. 5-25: Round Mill Window

Rectangle Frame Mill Window
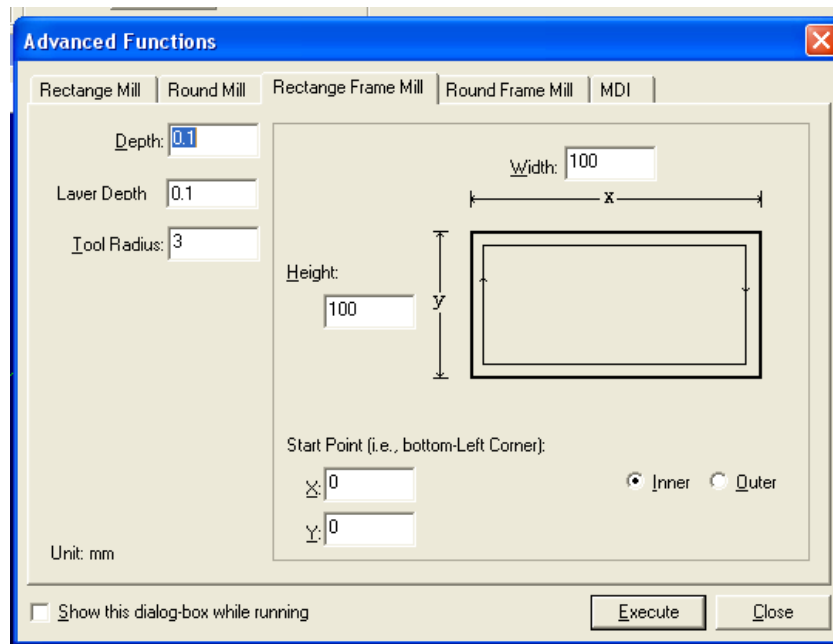
Fig. 5-26 Rectangle Frame Mill Window

Round Frame Mill Window
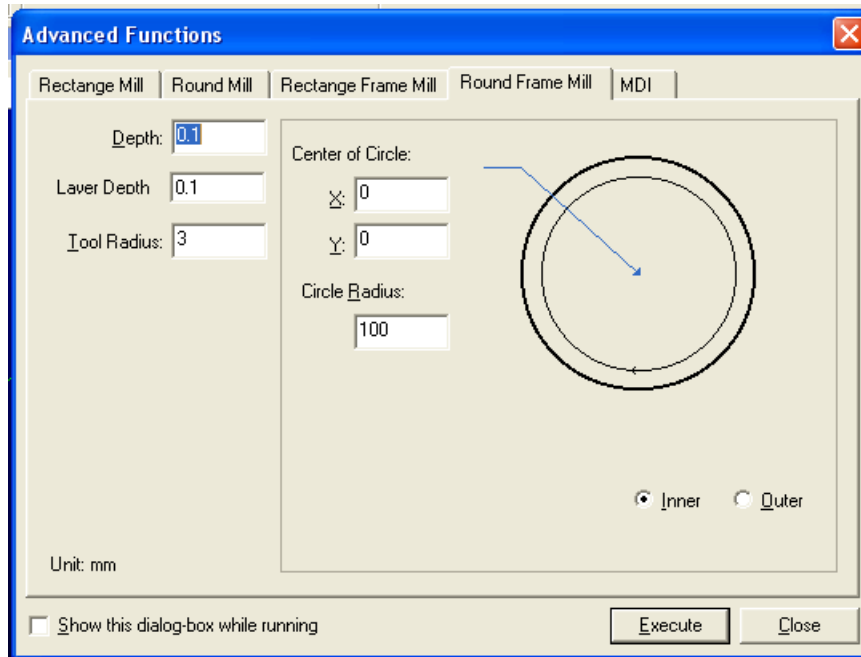
Fig. 5-27 Round Frame Mill Window

MDI window (Manual data input)
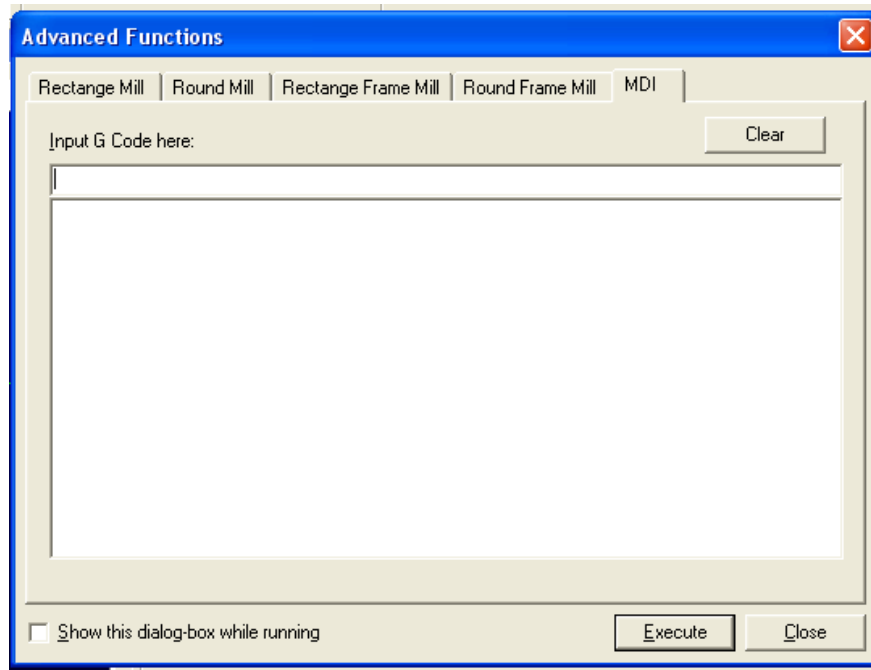
Fig. 5-28 MDI Window

The system will execute the instruction entered immediately by inputting standard instruction (G instruction, T instruction, M instruction) into the editor box of MDI window, and pressing "Enter" key. Meanwhile, it would record history input instructions in the List Box for user's recall.

The semicolon ";" should be used to separate each instruction while there are many instruction entered. When incorrect instruction is inputted, the system will sent out prompt information.

Selected this option "Show this dialog-box while running", the system can display the MDI window when executing commands. It is convenient for user to modify, call the parameter of default processing procedure file and input G command immediately.

Jiggle

This function is only valid under the state of pause in the course of automatic process. This function is used to realize the tiny regulation of the depth without breaking off processing cycle.

The result of jiggle operation is only effective in the current processing task. Therefore the jiggle operation will be invalid after "Stop" function executed and "Start" or "Breakpoint Resume" function used again.
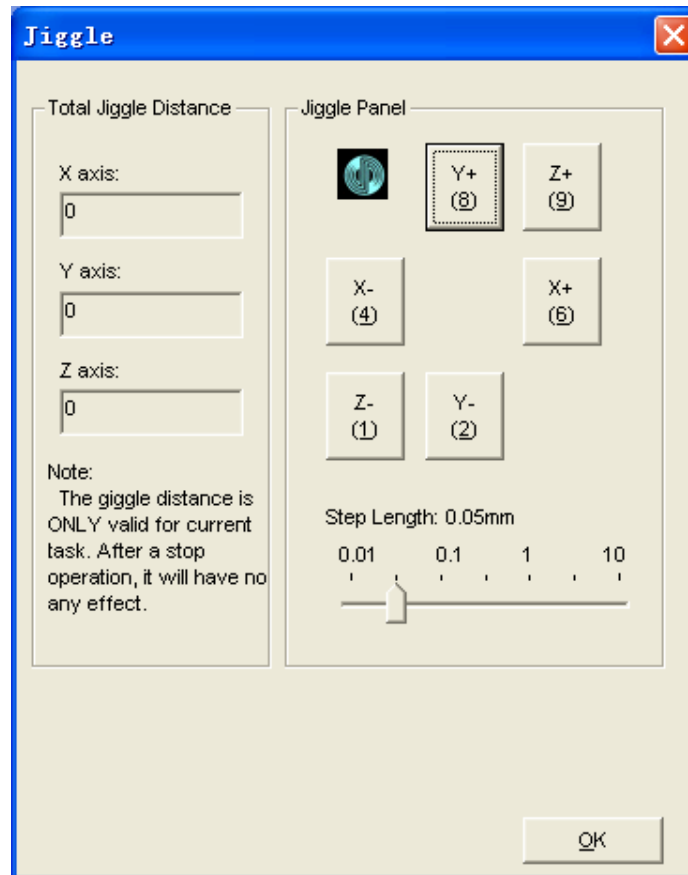
Fig. 5-29 Jiggle Dialog

Mobile Calibration

Making use of the floated (mobile) calibration, user can make sure the height of workpiece surface conveniently. Floated calibration process are: put the calibrator on workpiece surface, run the machine to let the tool tip touch the calibrator along Z-axis, calibration stops, system will get the position of tool tip, and then subtract the thickness of calibrator, and finally get Z-axis machine coordinate of workpiece origin.

Select the menu, you will see a dialog to ask if to do calibration, user can select according to the real situation.
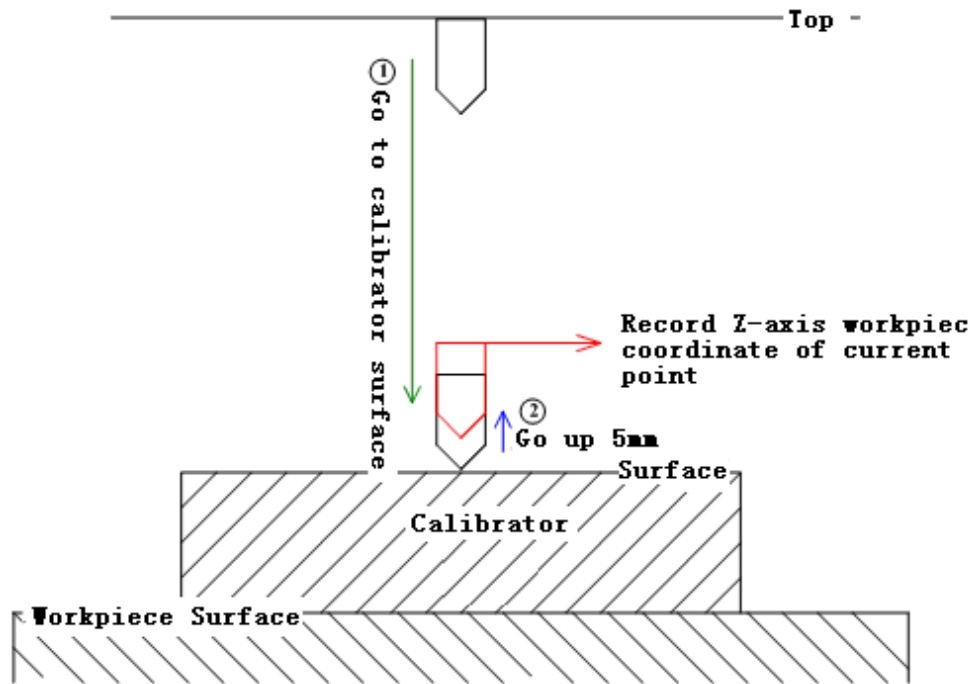
Fig. 5-30 Demonstration for Floated Calibration

User can also use button on the tool bar to realize the floated function.

Fixed Calibration

With the fixed calibration, user can expediently ensure reasonable Z-axis machine coordinate of workpiece origin and also readjust Z-axis machine coordinate after tool change.

Select the menu, you will see the following dialog.
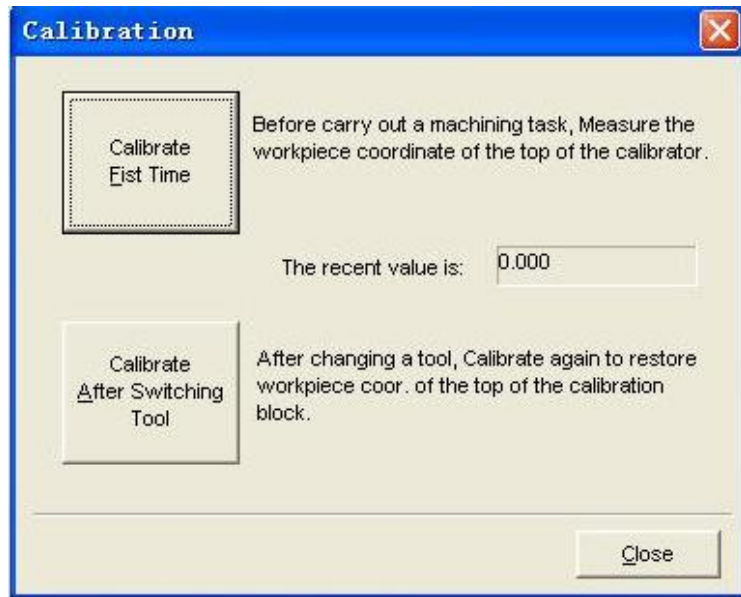
Fig. 5-31 Fixed Calibration

During calibration, check workpiece surface firstly, and set workpiece surface as Z-axis workpiece origin. Then execute the first calibration, and each time afer tool change, execute the calibration once again.

Operate ccording to the following demonstration.

First Calibration:
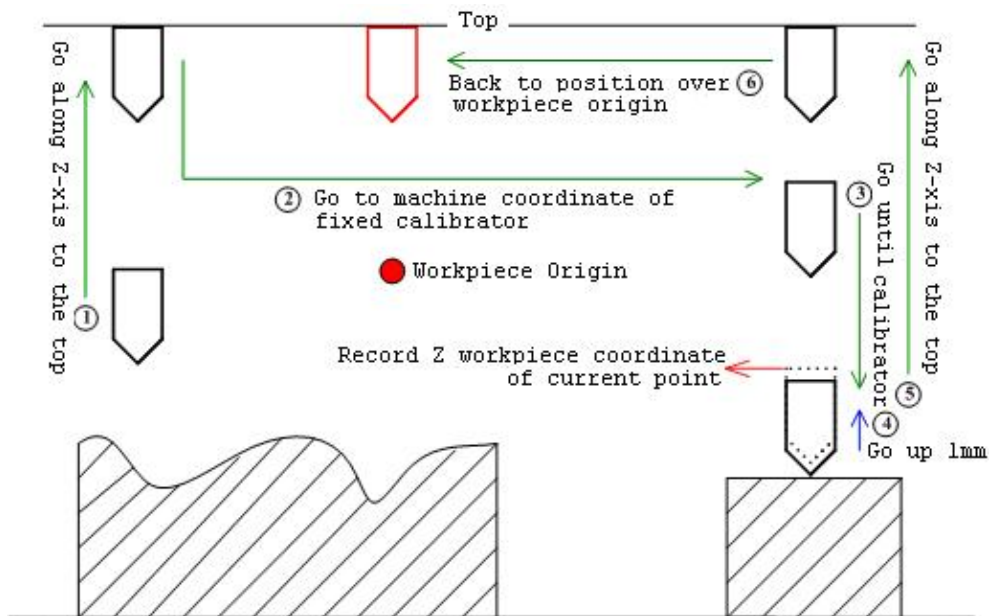
Fig. 5-32 First Calibration Diagram

Calibration after cutter replaced:
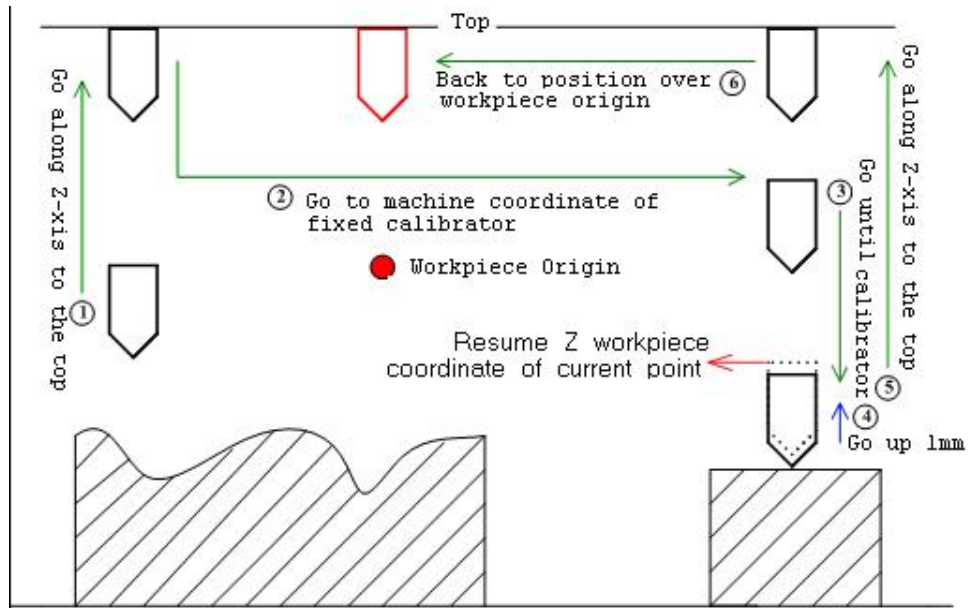
Fig. 5-33 Calibration After Cutter Replaced

User can also use button ![icon] on the tool bar to realize the fixed calibration.

Back to Machine Reference Point

Select the menu to popup the following dialog box:
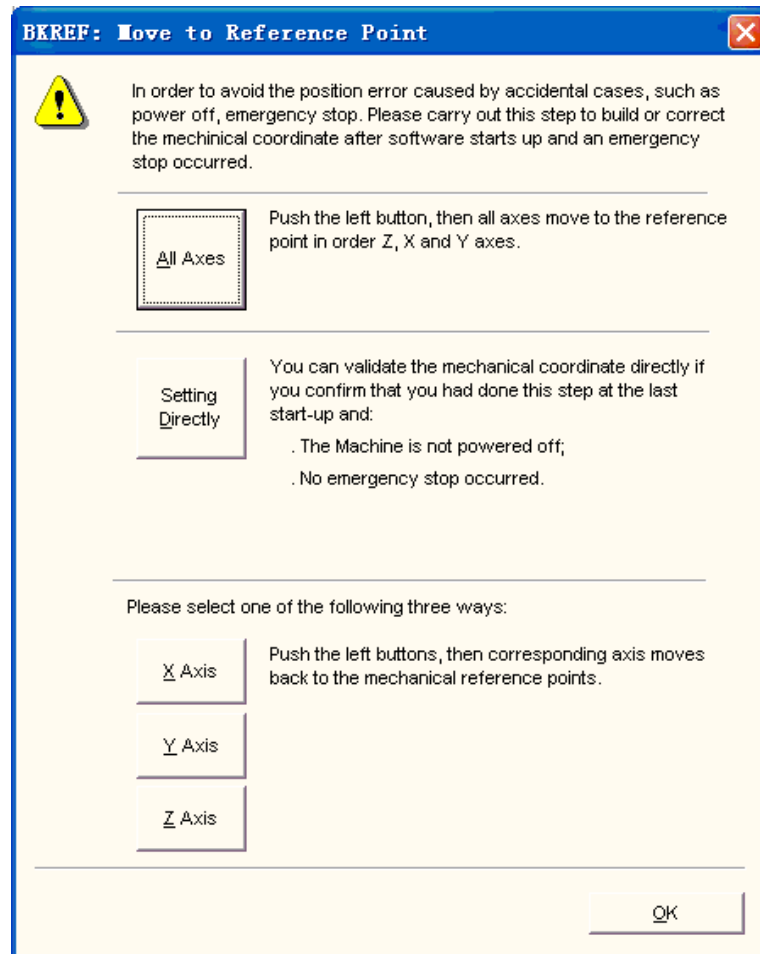
Fig. 5-34 Back to Machine Reference Point Dialogue

: Push the left button, all axes will move to the machine reference point in order (Z axes, XY axes, the order of Z1 and Z2 can be set by manufacturer)

: set the current X, Y coordinate to be mechanical coordinate and confirmed that current position is coincided with mechanical coordinate. If the machine tool was closed or stopped urgently, you are recommended not to execute the operation. The thing that should pay attention to is Z axis can not be set directly.

: return back each axis to corresponding mechanical origin.

Fig. 5-35

When all axis returned back to mechanical origin, "❖" mark will be presented at the front of each axis in NC bar.

There are several means to activate the interface of "Back to Machine Reference Point" function:

1) When the software is rigid Starting up.

2) Menu selection: "Operation" －－＞ "Back to Machine Reference Point".

3) Shortcut key "Ctrl+Home".

Move to Fixed Machine Position

The machine tool will move to fixed machine position when the function is selected. Setting a proper fixed machine position value, the user can replace cutter or workpiece conveniently.

For fixed machine coordinate, please refer to parameter "N4210", "N4211", "N4212".

Scan Function
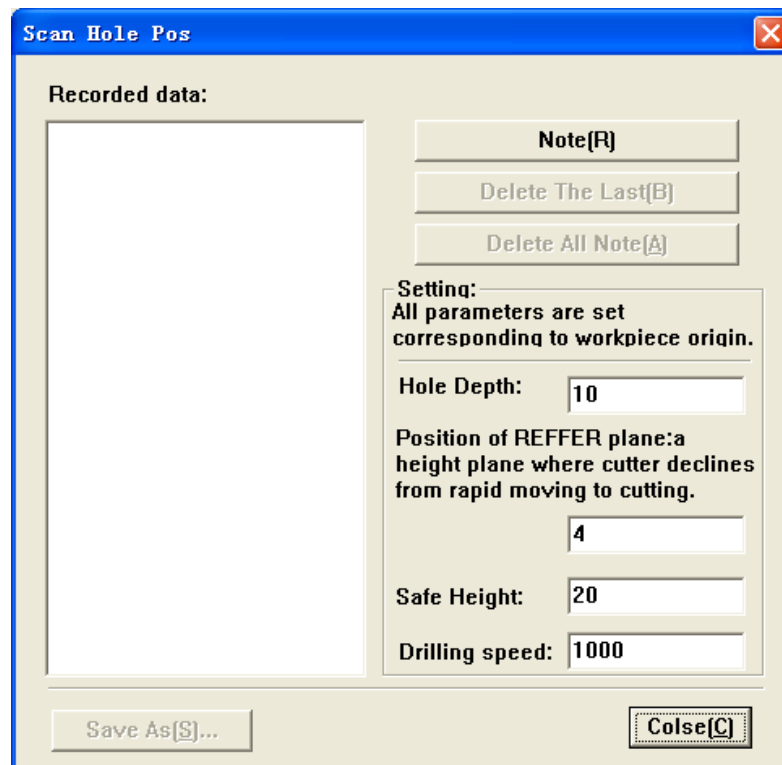
Fig. 5-36 Scan Function Dialog

Disable Mechanical Limits

The system will mask the limit bit temporarily when encountered a hard limitation. The system will mask the function to avoid warning. The user can recover its normal position by moving the machine tool through manual window to disengage limit switch, as shown in below：
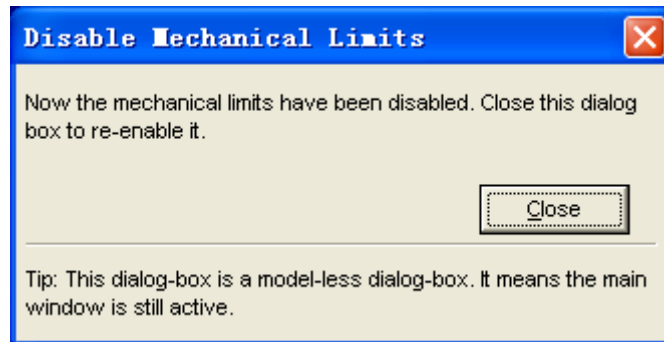
Fig. 5-37 Disable Mechanical Limits

Caution:

Please note your hand moving direction when disable mechanical limits, or the machine tool may be damaged!

Alarm Reset

The system will alarm when detected that the workbench's travel path off the soft limitation. To choose this option can let the workbench back to IDLE state.

Parameters Restore

This function is used to backup parameter automatically. Select the option, a dialog box will popup:
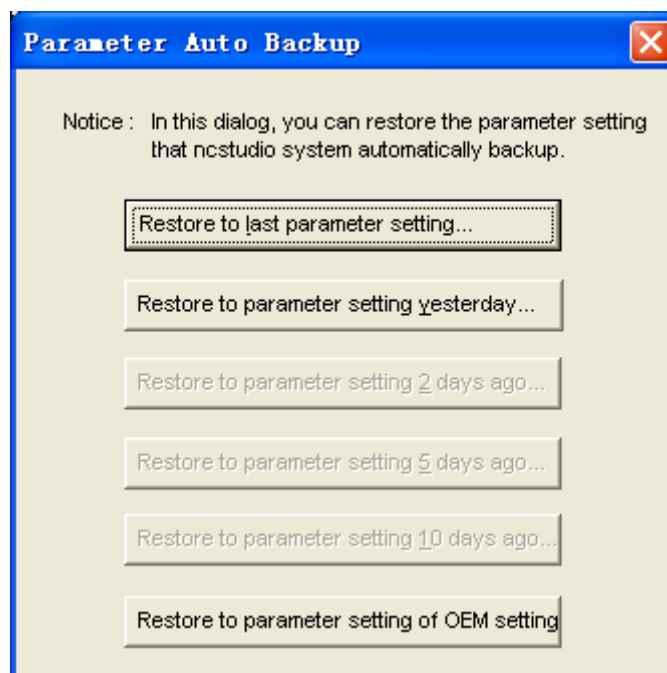
Fig. 5-38 Auto Backup Parameter

User can recover all automatic backup parameter as required. The system provide user with six kinds of parameter backup consisting of "Last Time", "Yesterday", "The Day Before Yesterday", "Five Days Ago", "Ten Days Ago", "Factory Parameter".

Set Parameters

The function is used to open parameter window for parameter setting, for details please refer to Chapter 6.

5.5 "Machine"menu
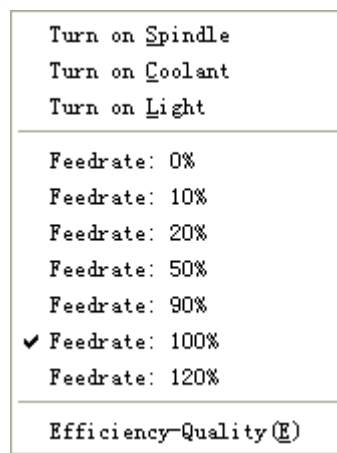
Fig. 5-39 "Machine" menu

Spindle Starting

The function is used to control the starting of spindle.

Coolant On

The function is used to control the starting of coolant pump.

Lighting

The function is used to turn on the lamp on machine tool.

Feedrate

Feedrate: 0%, 10%, 20%, 50%, 90%, 100%, 120%

The function is used to adjust the feedrate to 0%, 10%, 20%, 50%, 90%, 100%, 120%. The function is equivalent to the feedrate adjusting button located on NC information bar.
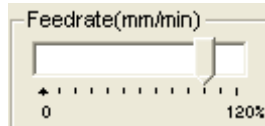
Fig. 5-40 Feedrate Adjusting Button

Efficiency - Quality Adjustment

Click the option to popup the following dialog box:
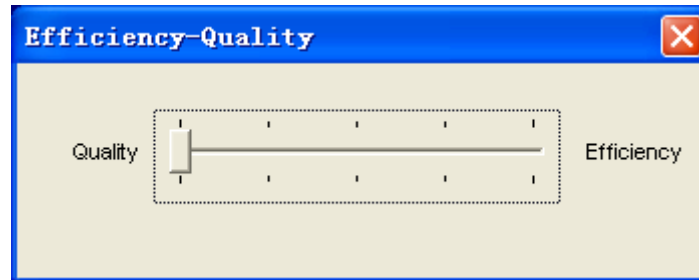
Fig. 5-41 Efficiency – Quality Adjustment Button

The quality is related to efficiency: when the quality takes the priority, the quality of processing is higher; when speed takes the priority, the efficiency of processing is higher.

5.6 "Window" menu

Fig. 5-42 "Window" menu

That menu is used for switching between each window.

5.7 "Help" menu
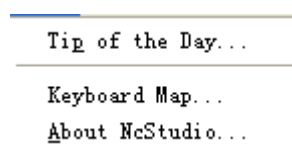
Fig. 5-43 "Help" menu

"Daily Tip"

When user selects this option, a dialogue will pop out and tell some related information and operating methods about NcStudio^{TM}. For example：
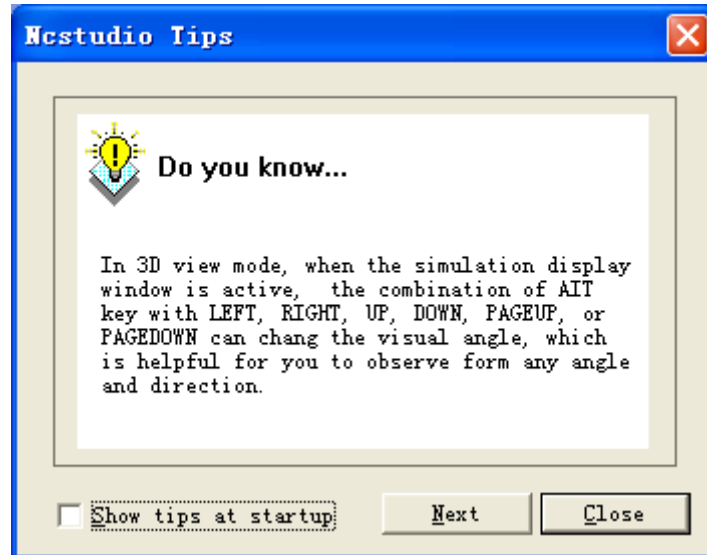
Fig. 5-44

Description of Shortcut Key
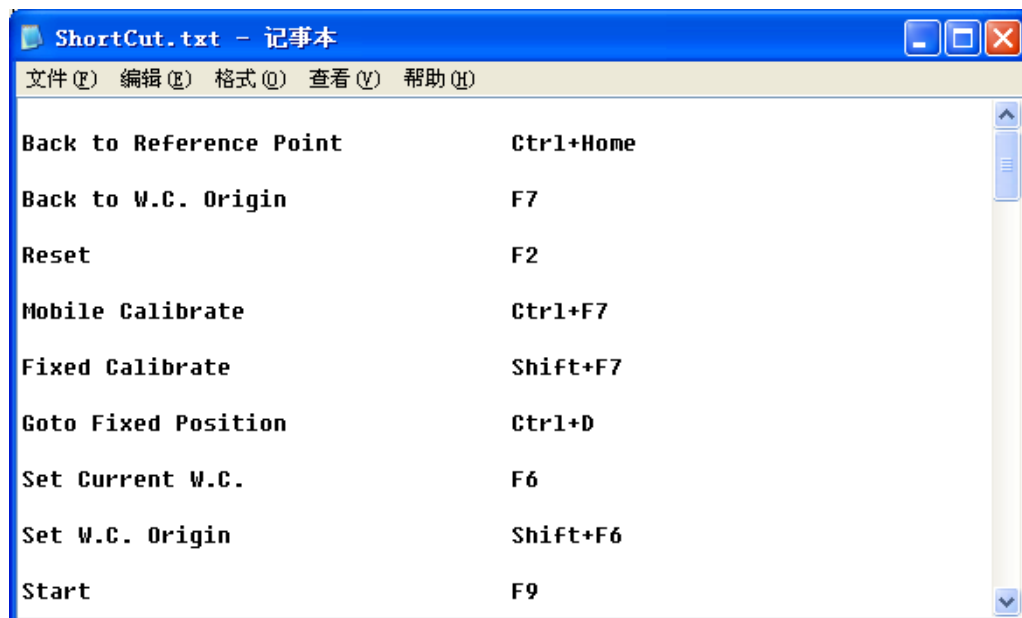
Fig. 5-45 Shortcut Key List Window

About Ncstudio

This item is used to prompt user the information about system software and hardware. When this item is selected, a dialog will pop up and tell user some information such as the Version No., Type of t he Motion Control Card, and Register Information of the NcStudio.
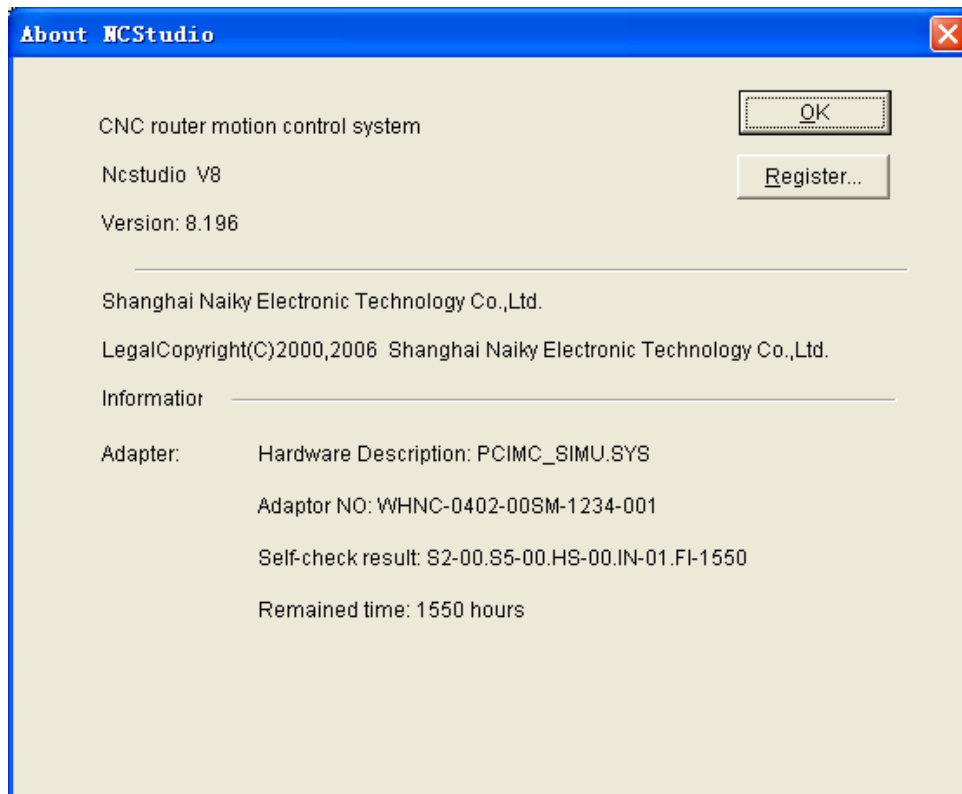
Fig. 5-46 System Information

Visit NcStuidio

User can use this item to visit NcStudio homepage of our website and to know our new products and other relative information.

6 Parameter Configuration

There are abundant processing parameters in Ncstudio, which can make the system more competent for each processing mission. This chapter will introduce operation authorization parameter only. For the authorization parameter for manufacturer which is used for inside communication for developers, please refer to Manufacturer's Manual.

Ncstudio parameters consist of operation parameter, feed axis parameter, spindle parameter, handwheel parameter, compensation parameter, origin parameter, tool parameter, other parameter and parameter overview.

6.1   Right of Parameter Modification

The parameter shown varies with different authorization parameter. There are passwords for manufacturer parameter and developer parameter.

Level	The parameter that can look into and modify
Visitor	none
Operator	processing parameters
Manufacturer	processing parameters,manufacturer parameters,
Developer	processing parameters, manufacturer parameters, impolder parameters

6.2   Parameter modification method

The method of the parameter modification is to select the parameters which need to modify by arrow key on the keyboard. Input the value in parameter inputting area after pressed the "Enter" key or double clicked the line where the parameters are.

For the parameter whose value is "true" or "false", inputting "1" means "true", inputting "0" means "false", user also can input "true" or "false" to modify the parameter.

Prompt:

All parameters can't be modified during processing.

6.3　Processing parameters

Operation Parameter

【N4005】　SpindleActionsWhenFinished

Type	Int
Unit	None
Range	0: do not move; 1: Back to fixed point; 2: Back to workpiece origin.
Default	0
Valid time	Be valid immediately. Need not to restart.

Remark：

0: do not move. It means that after the normal completion of processing procedure, the spindle axis will stop at the coordinate value when processing finished.

1: Back to fixed point. It means that after the normal completion of processing procedure, the machine will return back to the fixed mechanical coordinate value set by "N4210", "N4211", and "N4212".

2: Back to workpiece origin. It means that after the normal completion of processing procedure, the machine will return back to the workpiece coordinate origin of current procedure.

【N4006】G73_G83CutterRetractRate

Type	Float
Unit	Mm. (millimeter)
Range	-99999~99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Remark：It means the retract rate after each feed operation by G73_G83 "high speed" reciprocating chip convey drilling for deep hole.

【N4007】G76_G87 DirectionWhileFixedDrillStop

Type	Int
Unit	None
Range	0：+X , 1：-X,2：+Y , 3：-Y
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Remark: the above mentioned direction is only effective when it is under X-Y(G17).]

【N4025】ManualLowSpeed

Type	Float
Unit	mm/min(millimeter/minute)
Range	0~ manual high speed
Default	1000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The value means the motion speed of machine driven manually with 100% of feedrate (the manual speed varies along with the adjustment of feedrate).

【N4026】ManualHighSpeed

Type	Float
Unit	mm/min(millimeter/minute)
Range	Manual high speed ~Max speed supported by the hardware
Default	2400
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The setting value means the machine motion speed when feedrate is 100% and manual direction key and "high-speed" key ("0" key on small keyboard) are pressed down simultaneously.

【N4027】Tool Falling Speed On Pause And Continue

Type	Float
Unit	Mm/Min. (millimeter/minute)
Range	0~ Max speed supported by the hardware
Default	600
Valid time	Be valid immediately. Need not to restart the system.
Instruction	It means the tool falling (back to the cut point before pausing) speed of Z axis when it is paused and continued for processing.

【N4028】Tool Raising Speed On Pause

Type	Float
Unit	mm/min. (millimeter/minute)
Range	0~ Max speed supported by the hardware
Default	600
Valid time	Be valid immediately. Need not to restart the system.
Instruction	It means the tool raising speed on pause.

【N4029】Jiggle Feedrate

Type	Float
Unit	mm/min. (millimeter/minute)
Range	0~ Max speed supported by the hardware
Default	60
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The speed when jiggling.

【N4030】JiggleStepLength

Type	Float
Unit	Mm. (millimeter)
Range	0.01~0.5
Default	0.01
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The jiggling length corresponding to the button knocked down.

【N4031】RapidTravelFeedrate

Type	Float
Unit	Mm/Min. (millimeter/minute)
Range	0.001 ~ Max Speed
Default	3000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The motion speed of G00 order, namely the IDLE stroke speed

【N4032】DefaultFeedrate

Type	Float
Unit	Mm/Min. (millimeter/minute)
Range	0.001 ~ RapidTravelFeedrate
Default	1500.000000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The interpolation speed of G01, G02 and G03 order, namely processing speed.

【N4034】UseDefaultFeedrate

Type	Bool
Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

Select "false" for using the speed specified in processing file; Select "true" for using the default feedrate set with parameter "N4032".

For some processing files (DXF, PLT format etc.) without specified processing speed, no mater you select "false" or "true", the system will operate with the default feedrate speed set with parameter "N4032".

【N4035】UseDefaultSpindleRev

Type	Bool
Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid after restart the system

Remark:

Select "false" for using the spindle rev specified in processing file; Select "true" for using the default rev. set with parameter "N0002".

For some processing files (DXF, PLT format etc.) without specified spindle rev., no mater you select "false" or "true", the system will operate with the default rev. set with parameter "N0002".

【N4044】SpeedConfirmationMethodWhenZ-DownFeeds

Type	Int

Unit	None
Range	0: without special treatment; 1: the downward speed of Z cutter is effective when Z axis moves in negative position alone; 2: the downward speed of Z cutter is effective when Z axis moves in negative position; 3: Adjust feedrate slowly.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

0: do not have special treatment; there will be no Z direction tool falling speed set by "N4045', when Z axis is processing downward.

1: the downward speed of Z cutter is effective when Z axis moves in negative position alone; the system will adopt the Z down feedrate set by "N4045" when Z axis is processing downward alone.

2: the down forward speed of Z cutter is effective when Z axis moves in negative position; the system will adopt the Z down feedrate set by "N4045" for processing when Z axis (regardless the movement of X and Y) moves in negative direction.

3: Adjust feedrate gradually. Adjust the initial feedrate set by "N7018" to be default feedrate through the time set by "N7019".

【N4045】Z_DownFeedrate

Type	Float
Unit	Mm/Min. (millimeter/minute)
Range	0.001~9999
Default	500.000000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The parameter is valid when "N4044' parameter is selected to be "1" or "2".

【N4047】OptimizingZ-ToolRaisingSpeed

Type	Bool

Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	It is invalid to located instrument G00 regardless that the tool raising speed is optimized when Z axis moves upward vertically during processing.

【N4048】RaisingZAxisToolToSpecifiedWorkpieceCoordinate_ ValidOnPause.

Type	Bool
Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

1. When parameter 【4048】is selected with "false" for pausing, the parameter " 'N4050' Z-Tool Raising Rate On Pause" is valid, meanwhile, the parameter "'4049' Z-Axis Position On Pause" is invalid.

2.When parameter【4048】is selected with "true" for pausing, the parameter " 'N4049' Z-Axis Position On Pause" is valid, meanwhile, the parameter " 'N4050' Z-Tool Raising Rate On Pause" is invalid.

【N4049】Z-AxisPositionOnPause

Type	Float
Unit	Mm (millimeter)
Range	0 ~500

Default	100
Valid time	Be valid immediately. Need not to restart the system.

【N4050】Z-ToolRaisingRateOnPause

Type	Float
Unit	Mm (millimeter)
Range	0 ~500
Default	10
Valid time	Be valid immediately. Need not to restart the system.

【N4051】SafetyHeight

Type	Float
Unit	Mm. (millimeter)
Range	0.001 ~ 1000.0
Default	10.000000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The height is corresponding to workpiece coordinate origin, as the system regards it as safety height for horizontal movement. The parameter is used to execute the operation of "Back Workpiece Origin" and Breakpoint Resume.

【N4063】ArcIJKIncrementModeValid

Type	Bool
Unit	None
Range	0(false): invalid 1(true): valid

Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

1: when "true" is selected, the center coordinate is relative to the starting point for processing arc.

0: when "false" is selected, the center coordinate is relative to the coordinate of workpiece origin.

【N4068】 ToolReplacingPromptValid

Type	Bool
Unit	None
Range	0(false): not prompt; 1(true):prompt
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	When 0 "false" is selected, the system will not have a pause but continue to carry out processing when it is encountered with tool replacing instruction. When 1"true" is selected, the system will have a pause and prompt: there is a cutter required to be replaced, if you want to replace it, please stop the system and carry out breakpoint resume after the replacement; if you do not need to replace the cutter, please press "Start" to continue.

Translation Parameter for PLT File

【N4070】 ToolRaisingHeightWhenMovingWithIdleStroke

Type	Float
Unit	mm. (millimeter)
Range	1~500
Default	1

Valid time	Be valid after the procedure file reload.

【N4071】PLTUnit

Type	Float
Unit	Mm/PLU. (millimeter/plu)
Range	40 or 1016
Default	40
Valid time	Be valid after the procedure file reload.

【N4072】PLTToolsDistanceWhileProcessArea

Type	Float
Unit	mm. (millimeter)
Range	0.0001 ~ 99999.0
Default	0.025000
Valid time	Be valid after the procedure file reload.
Instruction	The distance of tools in PLT area when processing should be less than tool diameter.

【N4073】TwoDimensionalFileDepth

Type	Float
Unit	mm. (millimeter)
Range	-99999 ~ 0.0
Default	-1.000000
Valid time	Be valid after the procedure file reload.
Instruction	Specify the processing depth of two dimensional file.

Translation Parameter for DXF Files

【N4080】ToolRaisingHeightWhenMovingWithIdleStroke

Type	Float
Unit	mm. (millimeter)
Range	1~500
Default	1
Valid time	Be valid after the procedure file reload.

【N4081】TwoDimensionalFileDepth

Type	Float
Unit	mm. (millimeter)
Range	-99999~0
Default	-1
Valid time	Be valid after the procedure file reload.
Instruction	Specify the processing depth of two dimensional file.

【N4082】ProcessingCapacityEveryTime

Type	Float
Unit	mm. (millimeter)
Range	-99999~0
Default	-1
Valid time	Be valid after the procedure file reload.
Instruction	It is processing depth of two dimensional file when carrying out layer processing.

【N4083】UseFirstPointAsOriginInDXFFiles

Type	Bool
Unit	None
Range	0(false):Don't use first point as origin. 1(true): Use the first point as origin.
Default	1(true)
Valid time	Be valid after the procedure file reload.
Instruction	Specify whether use the first point as origin in the DXF file.

Remark：

0：False   Set the origin coordinate in DXF file to be workpiece origin.

1: True   The system will set the customized point in DXF file to be workpiece origin. For example, when drawing with CAD, we can set a point (it will not be processed) at any part in the drawing (recommended to near the figure or in the figure), and the system will default the point to be workpiece origin. If there are several points contented in DXF file, the system will regard the first point to be workpiece origin.

【N4084】ShapeUniqueProcessingValid

Type	Bool
Unit	None
Range	0(false); 1(true)
Default	0(false)
Valid time	Be valid after the procedure file reload.
Instruction	Process a shape each time; treat another one until the former shape processing is completed.

ENG File Translation Parameter

【N4090】ToolRaisingHeightWhenMovingIdleStroke

Type	Float
Unit	mm. (millimeter)
Range	1~500
Default	1
Valid time	Be valid after the procedure file reload.

【N4091】PauseAndPromptWhileChangeTools

Type	Bool
Unit	None
Range	0(false):Don't pause and prompt while change tools. 1(true): Pause and prompt while change tools.
Default	1(true)
Valid time	Be valid after the procedure file reload.
Instruction	Specify whether pause and prompt while change tools.

【N4092】ProcessingCycleTimesOfENGFile

Type	Int
Unit	None
Range	1～99999
Default	1
Valid time	Be valid after the procedure file reload.
Instruction	It is necessary to re-circulate the processing with several times when carrying out ENG file processing.

【N4093】CutterChoosingForProcessingWithENGFile

Type	Bool
Unit	None
Range	0(false): not use cutter choosing for processing; 1(true):use cutter choosing for processing
Default	0(false)
Valid time	Be valid after the procedure file reload.
Instruction	Use the function you can carry out processing with chosen cutter and treat its corresponding processing files only.

【N4094】PauseTimeAfterEachCirculationFinished

Type	Int
Unit	None
Range	0～99999
Default	0
Valid time	Be valid after the procedure file reload.
Instruction	Specify the pause time after each circulation finished when it is processing the ENG file.

【N4095】DeepProcessingMethod

Type	Int
Unit	None
Range	0~1
Default	0
Valid time	Be valid after the procedure file reload.

Instruction	Deep processing method: 0, reciprocating chip breaking; 1, high speed reciprocating chip breaking.

【N4096】ToolRestractValue

Type	Float
Unit	Mm(millimeter)
Range	1～99999
Default	1
Valid time	Be valid after the procedure file reload.
Instruction	Tool retract rate after the tool is punched into a deep hole with high speed reciprocating chip breaking.

Fix Calibrator Position

【N4200】X   【N4201】Y   【N4202】Z

Type	Float
Unit	Mm(millimeter)
Range	Work bench range lower limit(machine coordinate) ; Work bench range upper limit (machine coordinate)
Default	X：0 Y：0 Z：-1
Valid time	Be valid immediately. Need not to restart the system.

X：The value is set to be X axis machine coordinate value when the tool nose is reached into calibration range (it is better to get close to the center).

Y：The value is set to be Y axis machine coordinate value when the tool nose is reached into calibration range (it is better to get close to the center).

Z：The value is set to be Z axis machine coordinate value when the tool nose is located in a certain height range of calibrator (the tool is operated with idle stroke speed in the area above the height range, and it is operated with calibration speed in the area below the height range).

Machine Coordinate of Fixed Point

【N4210】X   【N4211】Y   【N4212】Z

Type	Float
Unit	Mm(millimeter)
Range	Work bench range lower limit(machine coordinate); Work bench range upper limit (machine coordinate)
Default	X：0 Y：0 Z：0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The matching use of machine coordinate of fixed point and parameter【N4005】.

【N7018】PercentInitialSpeedForLowSpeedFeedAtZAxisDirection

Type	Int
Unit	None
Range	1 ~ 100
Default	5
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The parameter is valid when 【N4044】is set to "Adjust feedrate slowly"

【N7019】Z-LowFeedRateRecoversToSettingMagnificationTime

Type	Float
Unit	S (seconds)
Range	0~10
Default	5

Valid time	Be valid immediately. Need not to restart the system.
Instruction	The parameter is valid when 【N4044】is set to "Adjust feedrate slowly", and in the period, the feedrate will recovers from 【N7018】setting value to initial setting feedrate.

Feedrate Parameters

【N1002】ComfirmPriorToResettingWorlpieceCoordinate

Type	Bool
Unit	None
Range	0(false): Need not to have a confirmation 1(true): Need to have a confirmation
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Confirm that whether the resetting of workpiece coordinate value (set the current point to workpiece origin) should be prompted, if there is a prompt, it may avoid incorrect operation.

The check for workpiece coordinate range is valid.
【N1150】X,【N1151】Y,【N1152】Z

Type	Bool
Unit	None
Range	0(false): invalid. 1(true): valid.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.

Instruction	True is for check the current point for locating in workpiece coordinate extents, and false is for unchecking.

Workpiece Coordinate Lower Limit

【N1160】X　　【N1161】Y　　【N1162】Z

Type	Float
Unit	mm(millimeter)
Range	-99999～99999
Default	-10000
Valid time	Be valid immediately. Need not to restart the system.

Workpiece Coordinate Upper Limit

【N1170】X　　【N1171】Y　　【N1172】Z

Type	Float
Unit	mm (millimeter)
Range	-99999～99999
Default	10000
Valid time	Be valid immediately. Need not to restart the system.

Spindle Parameter

【N0004】RevStopWhenStopped

Type	Bool
Unit	None
Range	0(false):not stop; 1(true):stop

Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether spindle is topped

【N0005】RevStopWhenPaused

Type	Bool
Unit	None
Range	0(false):not stop; 1(true):stop
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether spindle is topped

Origin Parameter

【N2001】NeedToBackToMachineReferencePointPriorToProcessing

Type	Bool
Unit	None
Range	0(false): Need ; 1(true): Need not
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	When "True" is selected, the system will have a prompt for backing to machine reference point prior to carrying out processing; when "False" is selected, the system will enter into processing state without having any prompt.

Cutter Parameter

【N6121】Diameter

Type	Float
Unit	Mm. (millimeter)
Range	0～99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The diameter of cutter selected.

【N6122】Length

Type	Float
Unit	Mm. (millimeter)
Range	0～99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The length of cutter sselected.

【N6123】DiameterWearExtent

Type	Float
Unit	mm (millimeter)
Range	0～99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Instruction	The diameter wear extents of cutter selected

【N6124】LengthWearExtent

Type	Float
Unit	mm (millimeter)
Range	0～99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The length wear extent of cutter selected.

Cutter Position Offset

【N6130】X　　【N6131】Y　【N6132】Z

Type	Float
Unit	mm (millimeter)
Range	Work bench range lower limit(machine coordinate); Work bench range upper limit (machine coordinate)
Default	X: 0 Y: 0 z: 0
Valid time	Be valid immediately. Need not to restart the system.

Remark: the three values of 【N6130】X,【N6131】Y,【N6132】Z mean the offset values of the cutter corresponding to the first cutter.

Other Parameter

【N7012】AdditionalCalibrationFunction

Type	Bool

Unit	None
Range	0(false): not use; 1(True):use
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Select "true" for using of additional calibration function; Select "false" for not using of additional calibration function.

【N7017】InformTypesForCompletionOfProcessingTask

Type	Int
Unit	None
Range	0:Red light goes out; 1: Red light comes on for 3 seconds; 2: Red light always comes on until there are mouse or keyboard input by user and then the yellow light comes on.
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Remark:

0: Red light comes on

a) When the software is under idle state, red light and green light shall go out, and the yellow light comes on;

b) When the software is under normal processing state, the red light and yellow light shall go out and the green light comes on;

c) After the completion of processing task, the red light and green light shall go out and yellow light comes on;

1: Red light comes on for 3 seconds

a) When the software is under idle state, red light and green light shall go out, and the yellow light comes on;

b) When the software is under normal processing state, the red light and yellow light shall go out and the green light comes on;

c) After the completion of processing task, the green light and yellow light shall go out and the red light comes on for 3 seconds and then the yellow light comes on.

2: Red light always comes on until there are mouse or keyboard input by user and then yellow light comes on.

a) When the software is under idle state, red light and green light shall go out, and the yellow light comes on;

b) When the software is under normal processing state, the red light and yellow light shall go out and the green light comes on;

c) After the completion of processing task, the green light and yellow light shall go out and red light always comes on until there are mouse or keyboard input by user and then yellow light comes on.

【N7100】InterfaceSignalChangeforSystemStop

Type	Int
Unit	None
Range	0～FFFFFFFFFFFFFFFF
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	After hexadecimal value is changed into binary system, the interface corresponding to the position 1 in binary system will have a signal change to stop the whole system.

【N7110】InterfaceSignalChangeForSystemPause

Type	Int
Unit	None
Range	0～FFFFFFFFFFFFFFFF
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Instruction	After hexadecimal value is changed into binary system, the interface corresponding to the position 1 in binary system will have a signal change to pause the whole system.

Double Z-Axis Parameter

【N7123】StartSpindleOrNotAfterCutterReplaced

Type	Bool
Unit	None
Range	0(false): Not start 1(true): Start
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether the current spindle should be started immediately after cutter replaced.

【N7124】SwitchToZ1AxisOrNotAfterProcessingCompleted

Type	Bool
Unit	None
Range	0(false):Not switch; 1(true): Switch
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether the current spindle should be switched to Z1 Axis after processing completed.

【N7125】StopSpindleWhenCutterReplaced

Type	Bool
Unit	None

Range	0(false): Not stop 1(true): Stop
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Select "false" for cutter replacement directly without stopping current spindle; Select "True" for cutter replacement after the current spindle stopped.

【N7126】BackToOriginalPositionAfterCutterReplaced

Type	Bool
Unit	None
Range	0 (false): Not turn back; 1(true):Turn back.
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Select "false" for not turning back to the coordinate before cutter replaced; Select "True" for turning back to the coordinate before cutter replaced.

7 Operation Instruction for Customized Small Keyboard

NcStudio™ version 5.4.53 and above support customized small keyboard.

For user's convenience of using the Ncstudio^{TM} software, a customized small keyboard which arrayed in the manner of 5×4 in the layout with 20 keys totally was equipped with especially as shown in below:

Start Pause	Stop	Break point Resume	Jog
Spindle switch	Set to be origin	Back to origin	×1
X+	Y+	Z+	×10
X-	Y-	Z-	×100
Shift	Calibrator	F-	F+

Fig. 7-1 Small Keyboard

Each key has specified function in the software, which listed as below:

1) Start/Pause: After pressed this key, the system begin to run the processing file if a NC file have been loaded and the system is idle; if the system is in running mode, pressing this key will make the system pause.

2) Stop: Stop the current operation. If the system is in the status of processing, pressed this key, processing will stop. But the mobile calibrator will be stopped after this key was pressed if the system is under mobile calibrator status.

3) Break Point Resume: The system will be resumed to machine after this key was pressed if the system is in the status of processing pause.

4) Jog: Switch the manual data input (MDI) mode to manual jog.

5) Spindle switch: Turn on the spindle after this key was pressed when the spindle was turned off. Otherwise, turn on the spindle.

6) Back to origin: The system will execute the order of "back to workspiece origin" when pressed.

7) ×1: After this key was pressed, the jog stepping length will be set as 0.01mm.

8) ×10: After this key was pressed, the jog stepping length will be set as 0.1mm.

9) ×100: After this key was pressed, the jog stepping length will be set as 1mm.

10) X+: Move the cutter along the X axis' positive direction. If the machine tool is in manual continual jog state, the cutter will be moved to the X axis' positive direction continually with manual low speed by pressing the key; the cutter will be moved to the X axis positive direction with manual high speed by pressing "Shift" and "X+" simultaneously; if the machine tool is under the state of X1,X10 or X100, the cutter will be moved to X axis positive direction with corresponding distance by pressing X+ key.

11) X-: Move the cutter towards the X axis' negative direction. The speed, the stepping length and the combination keys are the same as X+ key.

12) Y+: Move the cutter towards the Y axis' positive direction. The speed, the stepping length and the combination keys are the same as X+ key.

13) Y-: Move the cutter towards the Y axis' negative direction. The speed, the stepping length and the combination keys are the same as X+ key.

14) Z+: Move the cutter towards the Z axis' positive direction. The speed, the stepping length and the combination keys are the same as X+ key.

15) Z-: Move the cutter towards the Z axis' negative direction. The speed, the stepping length and the combination keys are the same as X+ key.

16) Shift: Nothing will happen if press this key separately. It will be valid only when you press the key combined with "X+","X-","Y+","Y-","Z+", or "Z-".

17) Calibrator: The system will mobile calibrate after this key was pressed.

18) F-: The feed rate will be decreased if this key is pressed.

19) F+: The feed rate will be increased if this key is pressed.

8 Operation steps

8.1   Power on

Before starting the machine tool, you should make sure all the connections of machine tool and computer are normal firstly. Then turn on the power of machine tool and the computer. After the system was loaded, enter the NcStudio™ digital control system.

8.2   Machine Reset (Optional)

Only the machine tool with the function of "Back to machine reference point" is involved in this section as required.

Choose "Back to Machine Reference Point" menu if machine tool support operation of it. Machine tool will back to Machine Reference Point automatically, and calibrate the coordinate system.

In some circumstance, user is not necessary to execute the operation of machine restoration when restart the system and continue previous operation if stopped normally while the current coordinate information have been saved when the Ncstudio system existed normally.

Additionally, if user confirm current position is correct, it is not necessary to execute such operation.

8.3   Loading the Processing Procedure

Before processing, the user need loading the processing procedure generally, otherwise, the corresponding automatic function is null.

When the menu of " File | open… " is selected, the system will flip out a dialog box of the standard document operation, which you can choose the driver, path, and the file name that you want to open. After clicked the button of "open", the processing procedure will be loaded. Then, to press the key of F2 to switch to the window of "processing procedure", the user can view the current processing procedure.

8.4   Manual Operation

Display the Interface of Manual Operation.

When the menu of "Window| showing Manu window " is selected, a manual operation interface will be displayed in the parameter list window, which can have manual operation to the machine tool.

Manual Moving

User can have manual moving to the machine tool by the corresponding key in the small keyboard while the NUMLOCK light is bright.

The corresponding key is:

4	negative direction of X-axis
6	positive 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

The machine tool will be moved manually with high speed by pressing down any of these keys together with 0 key located on small keyboard.

8.5   Set Workpiece Origin

When processing, the origins of the X, Y, and Z axes are the workpiece origin. Therefore, we should set the workpiece origin as below prior to processing:

Moving the X, Y axes to position of expected origin manually, choosing the menu of "Set current point as workpiece origin" or clearing the coordinate value of current position as zero, the current position will be the start point to process when execute processing file.

Workpiece origin setting of X, Y axes was completed after the steps listed above. But more exact operation are needed to set workpiece origin of Z axis, the system cooperating with workpiece hardware provide the function of Z axis calibrator.

Select the menu of "operate |mobile calibrator…"which help you to complete mobile calibrator.

After these operations, the workpiece origin is confirmed.

8.6   Execute Auto Machining

Auto machining means that system will carry out the machining program automatically.

Start Auto Machining

Select the menu of "Operation (O) |Start(S)" or click the button ▶ on the toolbar, the machine tool will execute the processing automatically from the first sentence of the procedure. Hotkey: F9.

Machine Tool Stop

During auto machining, if you want to stop the processing procedure, the machine tool will stop processing immediately and enter into "idle" state through selecting the menu

"Operate| Stop", clicking the button ■ on toolbar or use hotkey "F11", this is the recommended method for stopping the system accurately and orderly.

> Note:
>
> When the connection characteristic of super-speed smooth speed is valid, system will stop while connection speed is zero.

Machine Tool Pause

During automatically processing, if it is necessary to pause the processing, select the menu of " operate | pause" or corresponding button ❚❚ in the toolbar, the machine tool will stop after finished the current sentence of processing, which also can be performed by hotkey of "F10" .If you want keep on processing, you can choose the menu of " operate| Start".

Program Jumping Execution

Selecting the menu of "Advanced Start (A)", the system will flip out a dialog box for you to select the starting row and end row. For detailed operation, please refer to "Advanced Start" in 5.4-Operation Menu.

9 Precautions in operation

9.1 Precautions for Multi-Task Executing

Because the Windows is time-sharing operating system, general speaking, when runs the auto processing, you also can run other tasks (for example, edit the processing procedure), but please pay attention to the following two items:

For the procedure of Windows taking a lot of memory, please do not open too much windows, which confirmed by the size of computer memory.

The movement of some applications themselves may be not stable, for example, some game procedure, VCD player etc. They may take the system resources without limit in running, such as the memory, CPU time-piece etc, which can cause the computer crack down. So, in processing time, please do not start these procedures, so as not to result in accidental processing interruption.

9.2 Precautions for Backing to Machine Reference Point

During the process of backing to machine reference point, according to difference requirement of system, difference procedure would be occurred. In the system which requires high precision, ultimately, the procedure of the correcting would be slow. Please pay attention to the statue window, to back to the machine reference point after the system in status of idle, otherwise, backing to machine reference point would not complete normally but terminated manually.

The consequences of manual termination during the process of backing to machine reference point are listed as below. 1. Because the software limit signal did not disappear, port alarm may occur; 2. Inexact allocation (the function of calibrating machine reference point is artificially destroyed, so it may lead to nonstandard machine coordinate); 3. Software limit doesn't function (Because the process of backing to reference point did not complete, system considers the software limit is invalid; only when backing to the reference point is finished, software limit can effect).

10 User software permission agreement

Permission:

Shanghai WeiHong Electronic Technology Co., Ltd. (hereafter referred to as WeiHong Company) awards you the software right of use. But you must guarantee toward to the WeiHong Company: Does not use, copy, revise, rent, transfer either this system or any part of the software for other purpose beyond the articles stipulated in this agreement .

You must guarantee that:

1.   Only be used this system on one machine;

2.   Only be used on this machine. In order to backup or manage the archives, permit to copy this system by the read-only mode.

3.   Only when the third party accepts the provision and condition of this agreement, permit to authorize to transfer this system and permission agreement to the third party.

4.   When transferring, the original documents, supplementary documents and all copies must be transmitted to the other party together; or destroyed the surplus products completely;

5.   Only under the circumstance of accepting one provision of items below, permit to use this system on the multi-user environment or the networks:

This agreement definite provision permitted to use in the multi-user environment or the network system;

Usage of each pitch point and terminal of the system have all purchased the use permission.

You must guarantee not do those below:

1.   Transfers the permission any more;

2.   Reverse project, disassemble or disintegrate this system;

3.   Copy or transmit this system completely or partially except the regulation listed in this permission.

When you hand over the system or the copy to another completely or partially, your permission power terminated automatically.

Copyright and Property Rights:

This system and the documents enjoy the copyright, and are protected by the international agreement provision. You can not remove the copyright from the copyright statement of the software. You guarantee to copy duplicates of the copyright statement for this system (completely or are partial).You agreed to stop copying the system and documents by any illegal forms.

After-sale Guarantee:

The WeiHong company guarantees, in the normal use situation, from sells the date in 90 days, the software carrier has the material or the craft flaw, when really has the flaw after confirmation, the WeiHong company's complete responsibility is to exchanges a software carrier; also it is the only compensates. When the accidents, abuses or error usage resulted in paralysis of software, the after-sale guarantee is invalid. The carrier which exchanges from WeiHong company enjoy surplus time of original guarantee time, or 30 day-long guarantee; Takes its elder first.

Except the after-sale guarantee above, this system does not enjoy the after-sale guarantee of any other forms.

Responsibility limited:

The assurance declared above, regardless of listed or hided, is complete content of guarantee, including commercial character and compatible guarantee for special application goal. No matter either follows this agreement and other provisions or not, produced by using this system: The loss of profit, the usability lose, the commercial severance, or indirect, special, accidental and inevitable destruction of any forms, or the other party's any suing for damages, the WeiHong company, its agent and its seller are not responsible generally, even if the WeiHong company is informed the possibility occurrence of this kind of matter.

Permission termination:

If you disobey no matter what a provision and the condition of this agreement, the WeiHong company may terminate this permission. When the permission was terminated, you must destroy this system, documents and all copies immediately, or return them to WeiHong company.

Applicable Law:

Intellectual property rights Protection Rule, *Copyright Law, Patent Law* and so on.

Now, we affirm you already have read and understood this permission in detail, and agree to obey all of the provisions and conditions of the agreement strictly.

Shanghai WeiHong Electronic Technology Co., Ltd.

11 Appendix

11.1　General Shortcut Key

ESC	Switch between windows
TAB	Switch between control buttons
Ctrl+1	Display Auto Window
Ctrl+2 /Scroll Lock	Display Manual Window
Ctrl+F1	Switch to Z1
Ctrl+F2	Switch to Z2
Ctrl+F7	Floating Calibration
Ctrl+F9	Advanced Start
Ctrl+ Home	Back to Machine Reference Point
Ctrl+ Enter	Full Screen
Ctrl+ TAB	Switch between Collapsing Windows
Ctrl+ Del	Clear Machining Track
Ctrl+Shift+F9	Execute Machining Instruction
Ctrl+ A	Select All
Ctrl+ C	Copy
Ctrl+ E	Open and Edit
Ctrl+ F	Find
Ctrl+ H	Replace
Ctrl+ I	Information of machining Procedure
Ctrl+ N	New Procedure
Ctrl+ O	Open and Load
Ctrl+ P	Edit Current Processing Program

Ctrl+ S	Save
Ctrl+ U	Unload
Ctrl+ V	Paste
Ctrl+ X	Cut
Ctrl+ Z	Undo
ALT+1/F4	Display Processing Track Window
ALT+2	Display System Log Window
ALT+3	Display Manager Window
ALT+4	Display Editor Window
ALT+5	Show I/O State Window
F3	Find Next
F5	Direct Setting
F6	Set the Current Workpiece Coordinates
F7	Back to Workpiece Origin
F8	Enter (Exit) Simulaton Mode
F9	Start
F10/Pause Break	Pause
F11	Stop
F12	Back to Position of Tool Change
Shift+F6	Set Currrent Point as Workpiece Origin
Shift+F7	Fixed Calibration
Shift+F9	Breakpoint Resume

11.2 Shortcut Key of Manual Window

Scroll Lock	Activate Manual Window
4(Small keyboard)	Manual Direction of X- (Including Jog and increment)

6(Small keyboard)	Manual Direction of X +(Including Jog and increment)
2(Small keyboard)	Manual Direction of Y- (Including Jog and increment)
8(Small keyboard)	Manual Direction of Y+ (Including Jog and increment)
1(Small keyboard)	Manual Direction of Z- (Including Jog and increment)
9(Small keyboard)	Manual Direction of Z+ (Including Jog and increment)

11.3   Shortcut Key in Track Window

Home	Center
End	Show Current Machining Point
+ (Small Keyboard)	Zoom In
- (Small Keyboard)	Zoom Out
* (Small Keyboard)	Adjust to Window Size
5 (Small Keyboard)	Front View
8 (Small Keyboard)	Top View
2 (Small Keyboard)	Bottom View
4 (Small Keyboard)	Left View
6 (Small Keyboard)	Right View
1 (Small Keyboard)	Southwest View
7 (Small Keyboard)	Northwest View
3 (Small Keyboard)	Southeast View
9 (Small Keyboard)	Northeast View
Alt+→or Alt+←	Rotate round Z-axis
Alt+↑ or Alt+↓	Rotate round X-axis
Alt+PgUp or Alt+PgDn	Rotate round Y-axis