NcStudio V10 Milling CNC System

Users' Manual

2nd Edition

(For four & five axes)

Weihong Electronic Technology Co., Ltd.

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

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

Second Edition, February, 2016

XX Printing, XX, XXXX

88 pages

Preface

About This Manual

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

With 6 chapters, this manual can be divided into 5 parts, as follows:

1) Part 1: preface, introducing the precautions about transportation and storage, installation, wiring, debugging, usage, and so on. You need to read them carefully beforehand to ensure safe operations.
2) Part 2: Chapter 1, introducing software setup and manual update of the driver, as well as installations and connections of hardware. It helps users to properly install the software and start it for the first time.
3) Part 3: Chapter 2~3, introducing the operational interfaces, operation procedures and functionalities of the software. It helps users to have a good knowledge of the software, its operational commands, and how-to-use.
4) Part 4: Chapter 5, introducing possible problems and situation in real practice, and treatment to different alarms. It helps users to effectively react to the possible trouble and take proper measures to fix it instantly.
5) Part 5: Chapter 6, introducing basic concepts of NcStudio, shortcut keys and the software license agreement.

Applicable Product Models

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

Product Model	Remarks
NcStudio V10 Milling CNC System (For four & five axes)	1) Used together with PM85A/95A communication cards and Lambda 4S/5S controllers. Multiple axes software of the NcStudio V10 is used to control engraving and milling machines with four & five axes, which can be used in complex molding, advertising, decorating and wood cylindrical engraving industries. 2) Lambda 4S/5S controllers can be abbreviated as Lambda controller or the controller.

Contact Us

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

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

Revision History

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

Date	Edition	Revision
2016.02	R2	Contact information updated.
2015.10	R1	This edition is released for the first time.

Precautions

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

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

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

WARNING

1) Precautions Related to Storage and Transportation

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

2) Precautions Related to Installation

- Only when this equipment installed in the qualified electricity cabinet can it be used. The construction of the cabinet must reach IP54 grade of protection;
- Paste sealing strips on the joint of the cabinet to seal all the cracks;
- Cable entry should be sealed while easy-to-open on the spot;
- A fan or heat exchanger should be adopted for the heat dissipation and air convection of the cabinet;
- If a fan is adopted, air strainer is a must in air inlet or air outlet;
- Dust or cutting fluids may have access to the CNC device via the tiny cracks and tuyere. Therefore it is necessary to pay attention to the surroundings and air flow direction of the air vent to make sure that the outflow gas is towards pollution source;
- 100 mm space should be preserved between the back of the CNC device and the cabinet wall for plugging cable connected with the device and the ventilation & heat dissipation in the cabinet;
- Space between this device and other equipment should also be preserved according to the requirements;
- The product should be installed firmly and without vibration. During installing, casting, knocking, striking, or loading on the product is forbidden;
- To reduce electromagnetic interference, power-supply components used should be above AC or DC 50V and the space between cable and CNC device should be preserved above 100mm;
- It will be better if CNC device is installed at a position facilitating debugging and maintenance.

3) Precautions Related to Wiring

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

TO BE CONTINUED

WARNING

CONTINUE

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

4) Precautions Related to Running & Debugging

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

5) Precautions in Use

- Before power-on, please make sure that the switch is on blackout to avoid occasional start-up;
- Please check the electromagnetic compatibility during electrical design in order to avoid or reduce electromagnetic interference to the CNC device. A low pass filter should be employed to reduce electromagnetic interference if there are other electrical devices nearby;
- It is not allowed to frequently power on and power off. It is recommended to power up the machine again at least one (1) minute later after power failure or blackout.

CAUTION

1) Precautions Related to Product and Manual

- Matters related to restrictions and functions available stipulated in the manuals issued by the machine manufacturer are prior to those in this manual;
- This manual assumes all the optional functions are available, which you must confirm through manuals issued by the machine manufacturer;
- Please refer to manuals issued by the machine manufacturer for the instructions of machine tools;
- Functions, and software interfaces vary with the system and the version of software. Before using the system, you must confirm the specifications.

2) Precautions When Opening the Package

- Please make sure that the products are what you have ordered;
- Check if the products are damaged in transit;
- Check if the components and accessories are damaged or missing in terms of the detailed list;
- Please contact us promptly if product discrepancy, accessory missing or transit damage occurs.

Contents

1. System Installation and Connection

1.1. Host Computer Requirements

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

1.2. System Installation and Connection

If an old version of NcStudio has been installed in the computer, please delete it first before new system setup. Please refer to Chapter 1.3 for corresponding operations.

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

1.2.1. Software Setup

Please install the software as following steps:

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

Fig. 1-1 Language selection

3) Select [ENGLISH] to begin the installation. To avoid interference of old version software to the installation of the new one, the system prompts that the setup will delete all files of the old version software, see Fig. 1-2.

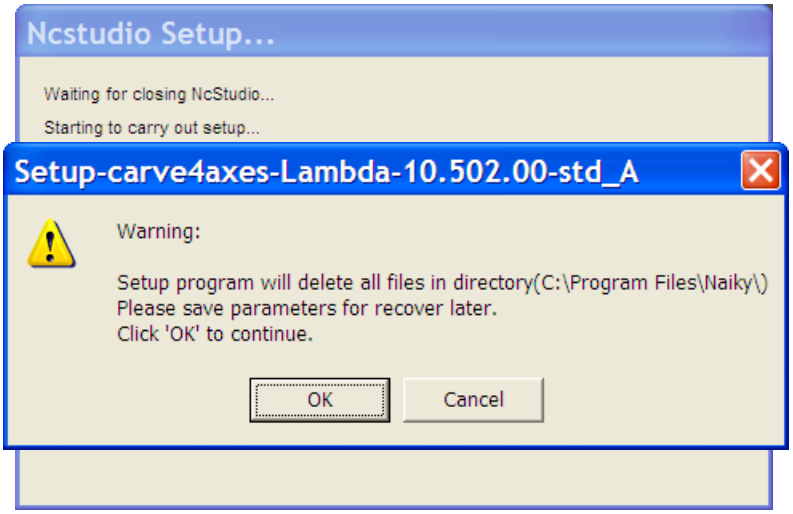


Fig. 1-2 Warning to delete previous files

4) Click [OK] for confirmation. If any old version software has been installed in this PC, the system will prompt to save its parameter settings. If you save the parameter settings here, you can apply all the settings to the current software, needless to set the parameters again afterwards. You can make your own decision and choose [Yes] or [No] to go ahead, see Fig. 1-3. If you install the software of the type for the first time, this step would be absent. Please jump to “Step 5)” to move on.

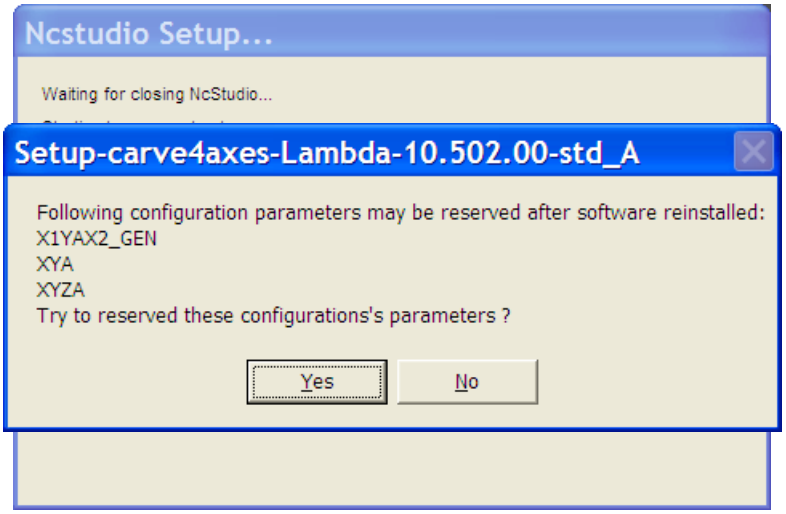


Fig. 1-3 Prompt to reserve parameter settings

5) Installation begins. The NcStudio system will be installed in the directory *C:\Program Files\Naiky* by default. Progressing picture is as shown in Fig. 1-4. With that, a dialog will appear to prompt the user to decide whether to put the generated files and the executive files together or separately, see Fig. 1-5. [No] is recommended here. The next dialog to appear is prompting shutdown of the

computer for convenience of communication card installation, and [Yes] is recommended, see Fig. 1-6.

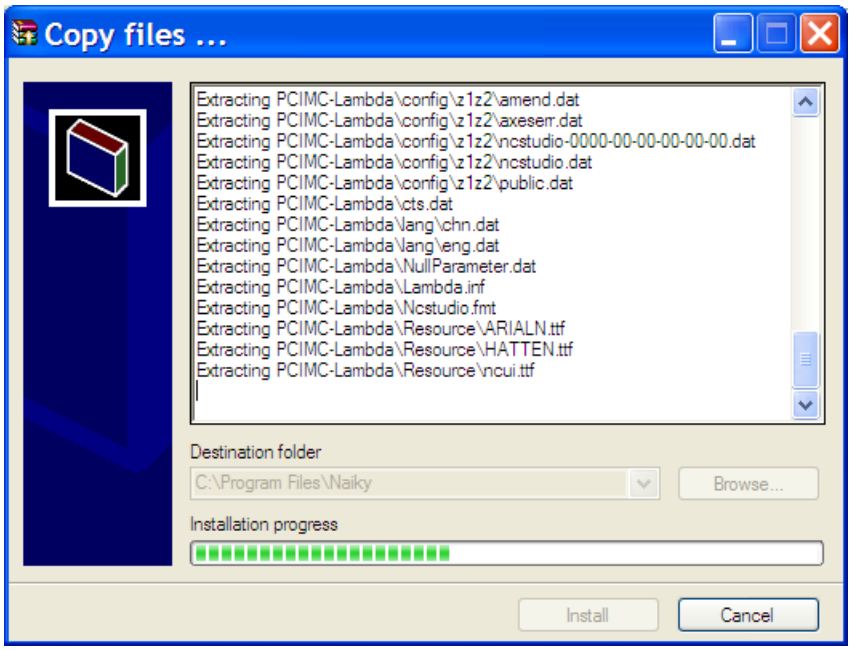


Fig. 1-4 Installing

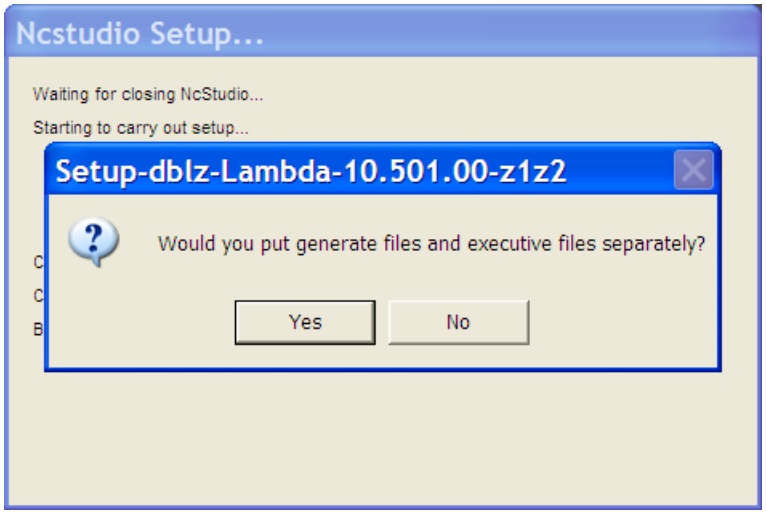


Fig. 1-5 Prompt for file location ([No] is recommended)

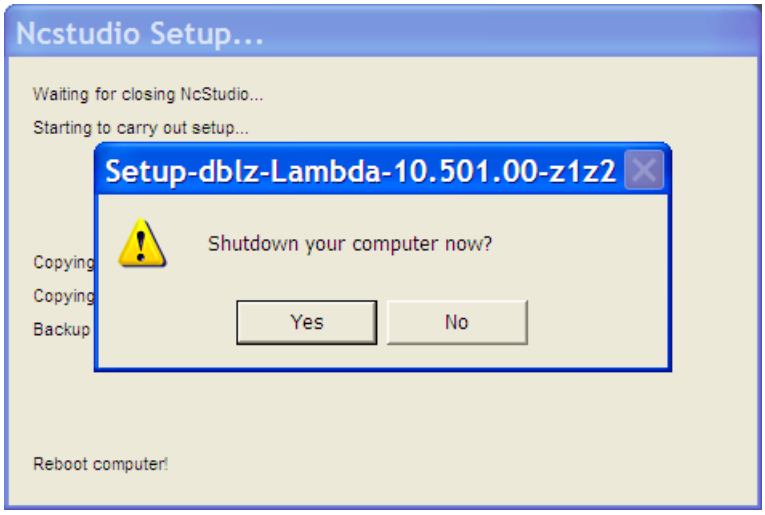


Fig. 1-6 Prompt to shut down the computer ([Yes] is recommended)

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

1.2.2. Hardware Installation & Connection

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

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

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

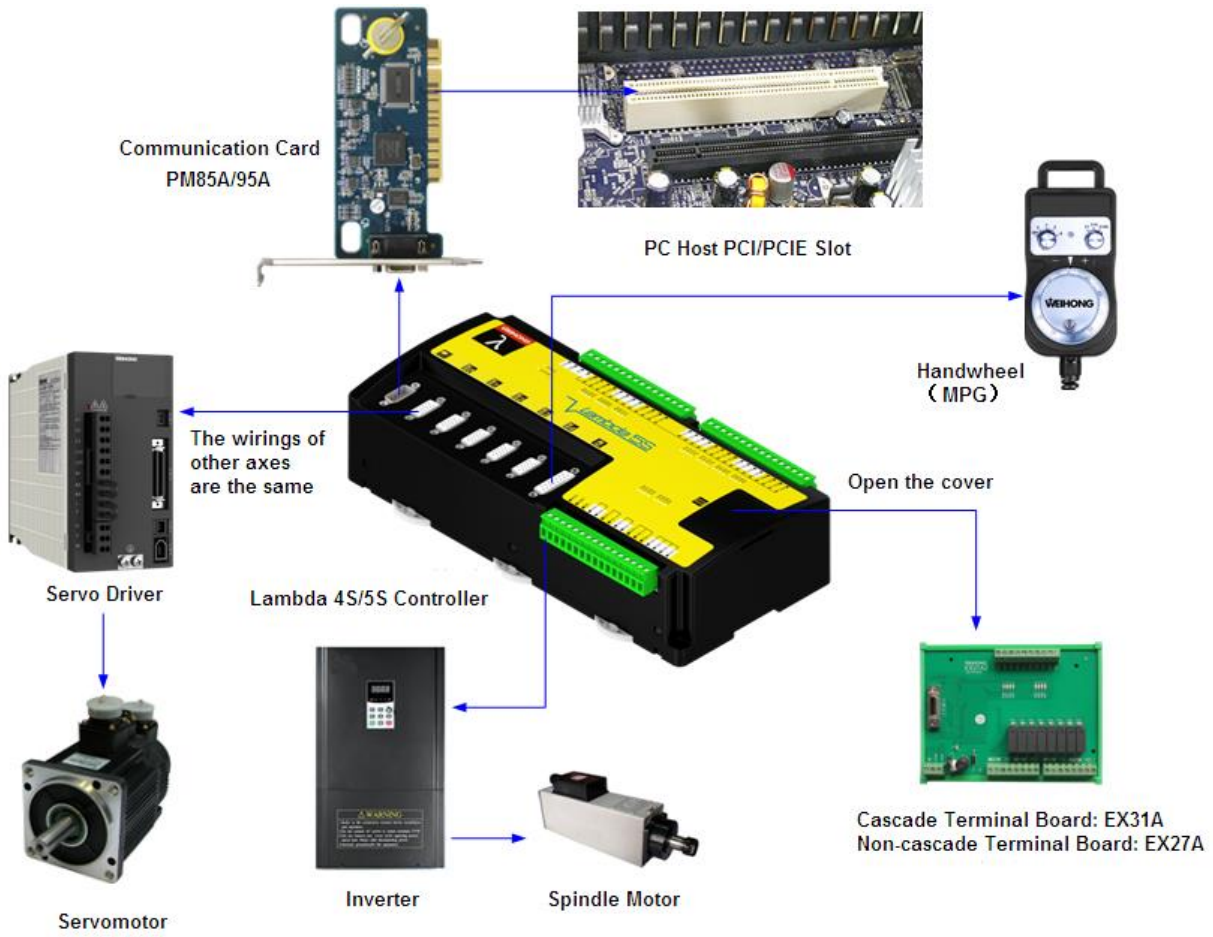


Fig. 1-7 Overall wiring diagram

CAUTION

Please connect according to the actual situations. The above picture is provided for reference only.

1.2.3. Manually Update Hardware Driver

After installations of the software and communication card and other hardware devices, the user needs to manually update the hardware driver; otherwise, the software cannot be launched normally. The specific operation steps are as follows:

1) Right click *My Computer*, select “Properties”, and then click “Device Manager”. Choose the “CNC Adaptor” item, and right click on it and select “Update Driver Software…”
2) A dialog box as Fig. 1-8 will pop up. Select “Install from a list of specific location (Advanced)”, and click [Next] to continue.

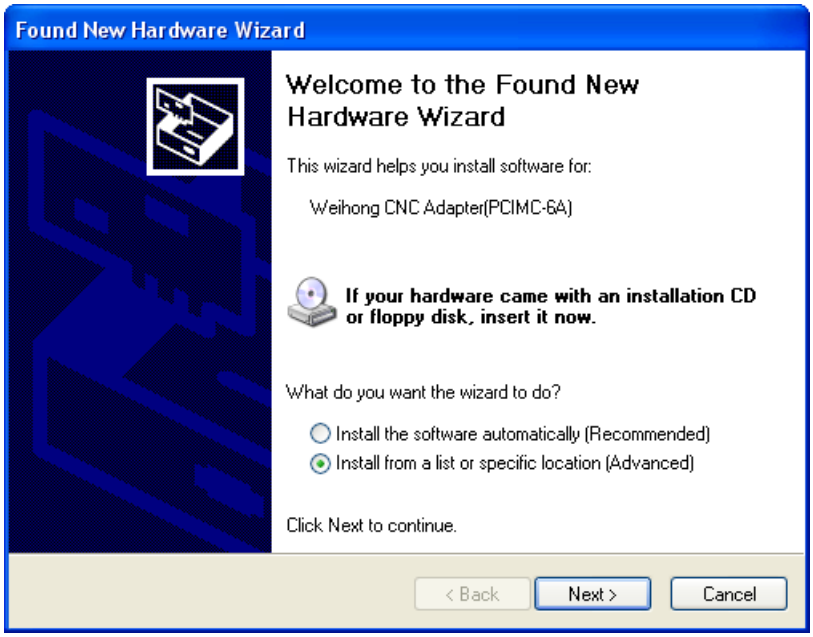


Fig. 1-8 Hardware update interface

3) A dialog box as Fig. 1-9 will pop up. Select “Don’t search, I will choose the driver to install”, and click [Next] to continue.

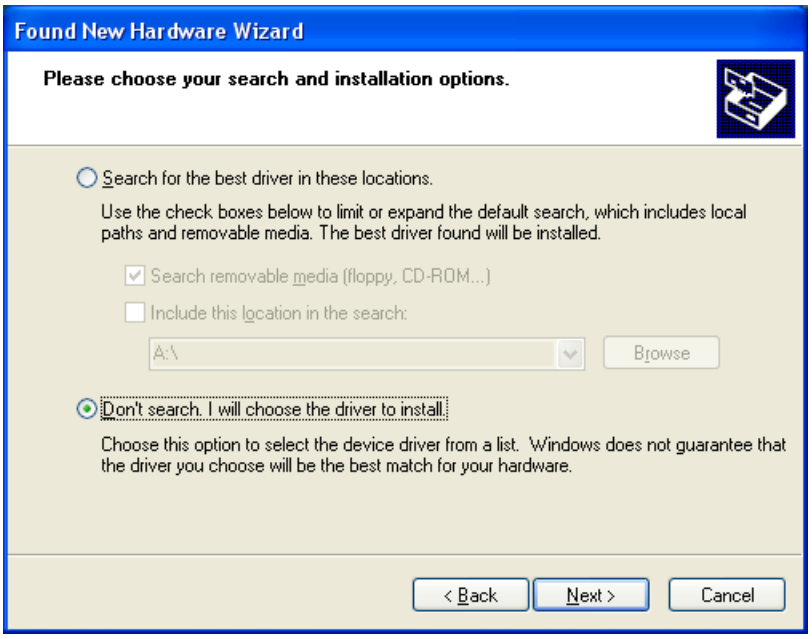


Fig. 1-9 Choose search and installation option

4) A dialog box containing compatible hardware will pop up, as shown in Fig. 1-10, click [Have Disk...] button to open the next dialog box.

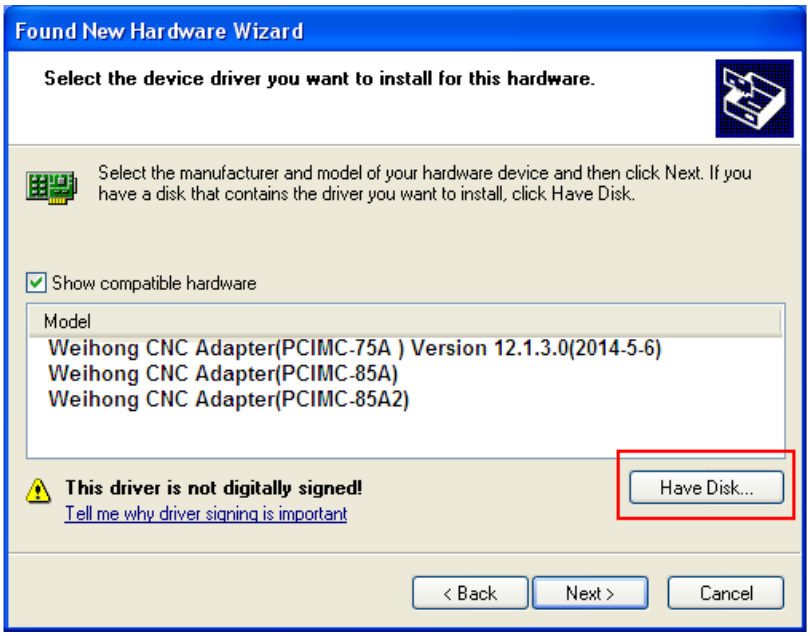


Fig. 1-10 Select the driver manually

5) Wait for several seconds, and a dialog box named “Install From Disk” will pop up, see Fig. 1-11. Click [Browse…] button to open the target file of driver.

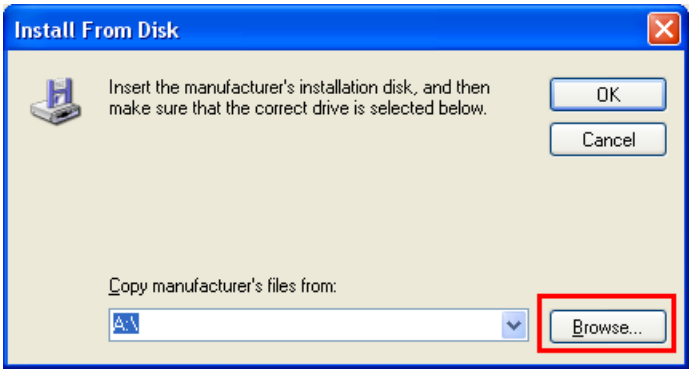


Fig. 1-11 Select the target file of driver

6) In “Locate File” dialog box as Fig. 1-12, select the target hardware drive in the list, Choose the target file “Lambda.inf” under directory “C:\Program Files\Naiky\PCIMC-Lambda”, taking PM85A as an example.

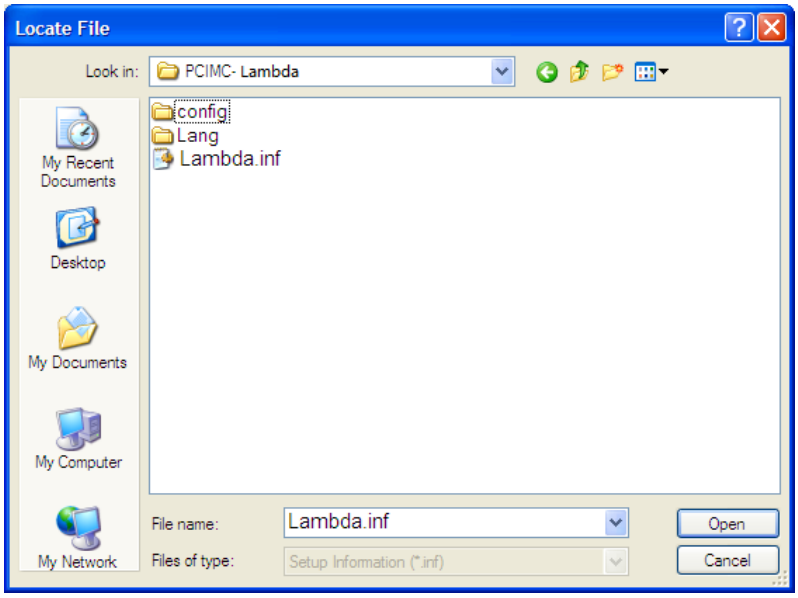


Fig. 1-12 Select the hardware driver

7) After hardware driver being correctly chosen, it jumps to the previous dialog box where the target file directory will be displayed under item “Copy manufacturer’s files from:” as shown in Fig. 1-13.

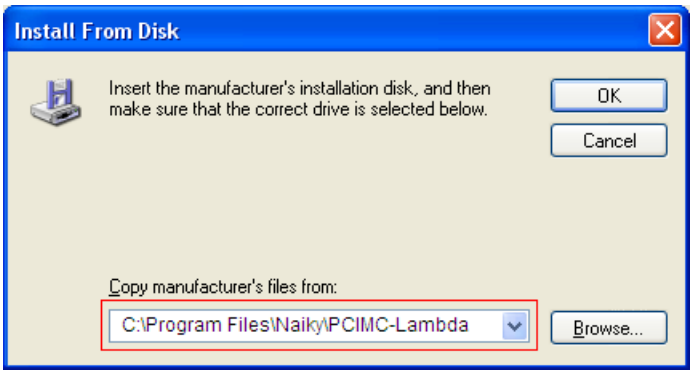


Fig. 1-13 Target file directory confirmation

8) Click [OK] to go back, and then click [Next] to start updating the driver software. The progressing picture is shown as Fig. 1-14.

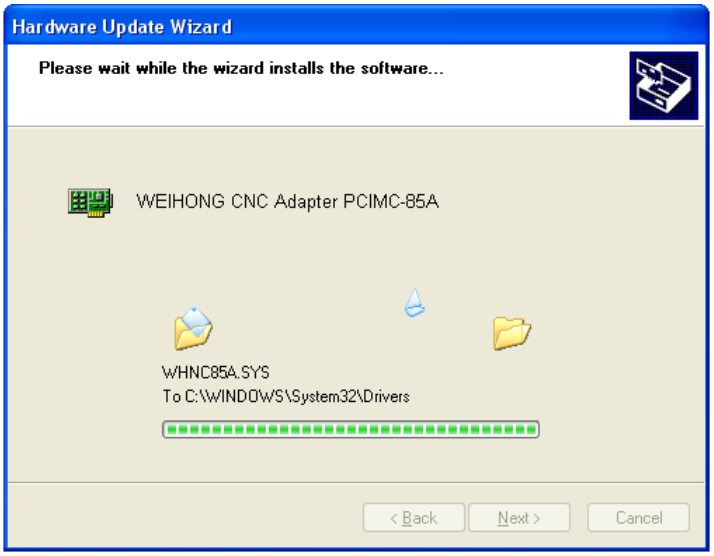


Fig. 1-14 Updating the driver

9) When the updating is finished, a dialog as shown in Fig. 1-15 will pop up. Click [Finish] to complete the update of the hardware driver. Double click the icon  on the desktop or click the icon on the menu *"Start→All Programs"* can launch the software successfully.

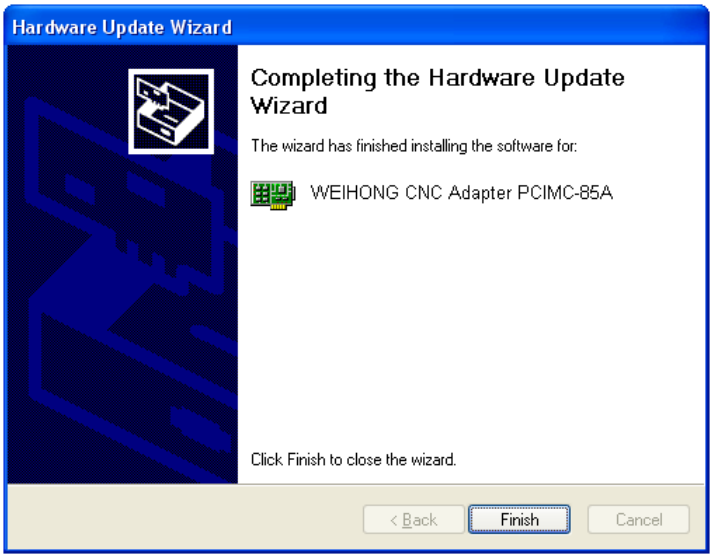


Fig. 1-15 Driver update completed

1.3. Uninstall NcStudio

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

under directory *C:\Program Files* and delete the NcStudio shortcut icon on the desktop as well as the *Start* menu.

Fig. 1-16 Shortcut icon on the *Start* menu

2. Software Operation

This Chapter focuses on the introduction to software operations. All pictures are provided for reference, taking four axes software as an example, if without special notification.

2.1. Know the Software

2.1.1. Holistic Interface

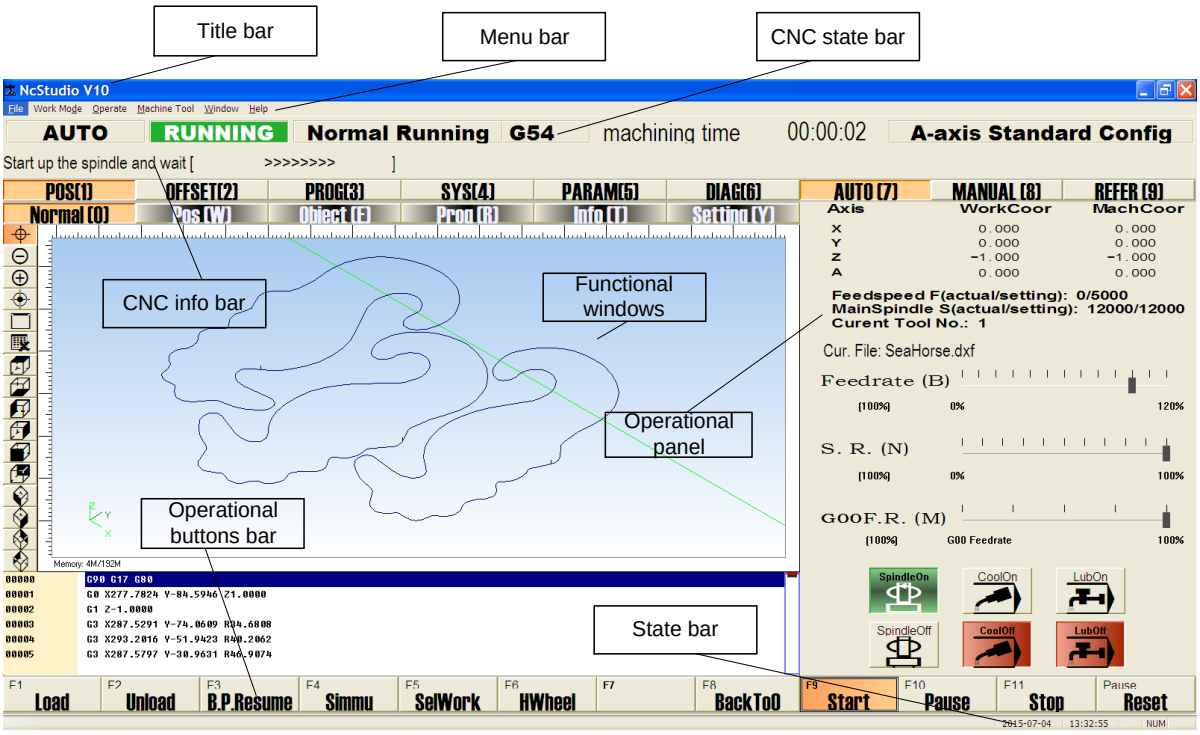


Fig. 2-1 Holistic interface of NcStudio V10 software

As see above, the holistic interface can be divided into 8 sections, including title bar, menu bar, NC status bar, info prompt bar, functional windows, operational panel, operational buttons bar and status bar.

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

Menu bar: it contains six main menus with several pull-down sub-menu items, which corresponds to a certain function, action or setting of the program state. Execution of a sub-menu item can make the machine execute related function or action or change the state. Menu selection and enabling can be done by mouse or shortcut key. With the help of menu items, the user can conveniently process the program, choose a machining mode, and switch over among windows, complete operations related to in-process machining and the machine, and get help information. For overall menu items of the system, please turn to Chapter 2.1.2.

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

CNC info bar: three different kinds of prompt information will be displayed, namely, normal prompt, warning prompt and erroneous prompt.

Functional windows: a main part of the software interface, showing various functional interfaces. Six functional buttons are included in these windows, and each contains several secondary functional selection buttons, which may vary because of different system states.

Operational panel: there are three types of panel, corresponding to *Auto* /*Manual* /*Reference point* modes respectively, please notice that. However, on three types of panel, the current workpiece coordinates, machine coordinates, current feedrate, tool number and currently loaded program file name will be displayed on the panels. In addition, the user can turn on/off the spindle, coolant and lubrication in this area.

State bar: a bar displaying current date and time, keyboard indicator as well as current status of NUM lock and Scroll lock.

Prompt information types can be told from the color of CNC info bar: when background color of the bar is the color of current interface, prompt is normal information; when it is yellow, prompt is warning information; when it is red, prompt is error information.

2.1.2. Overall Menu List

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

CAUTION

Sub-menu lists may change with the system mode, the above list is the one in Auto mode, for reference only.

Fig. 2-2 Overall menu list

2.2. Operation Step

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

THE FLOWCHART OF OPERATION

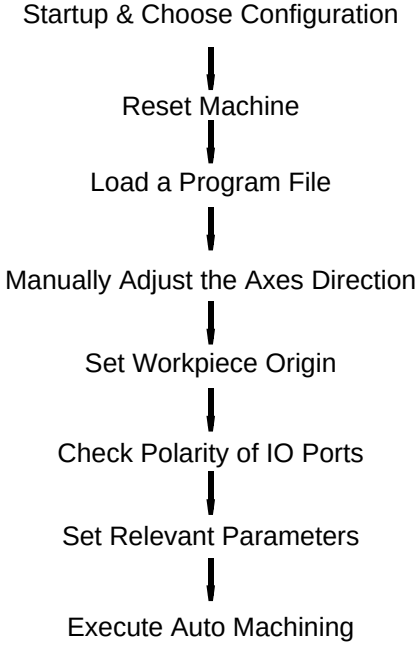


Fig. 2-3 Basic operation steps

2.2.1. Start-up and Choose Configuration

Before start-up, please make sure that the machine and computer are normally and properly connected. Turn on the power of the machine and computer, double click shortcut icon on the desktop or click it on the “Start” menu list to launch the software NcStudio. If the software is launched for the first time, the user needs to firstly select a configuration according to the machine structure.

There are three ways to access configuration management interface:

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

CAUTION

1) Numbers next to the six functional buttons are their shortcut keys, and pressing each shortcut key can access corresponding functional area directly. Likewise, pressing alphabet letters next to secondary functional buttons can access corresponding interface directly. Please note that number keys on the mini NUM keyboard are ineffective whatever their activation status is.

2) Generally, configuration is decided by machine tool builders, and the end-users cannot change it.

Configuration management window is illustrated as below:

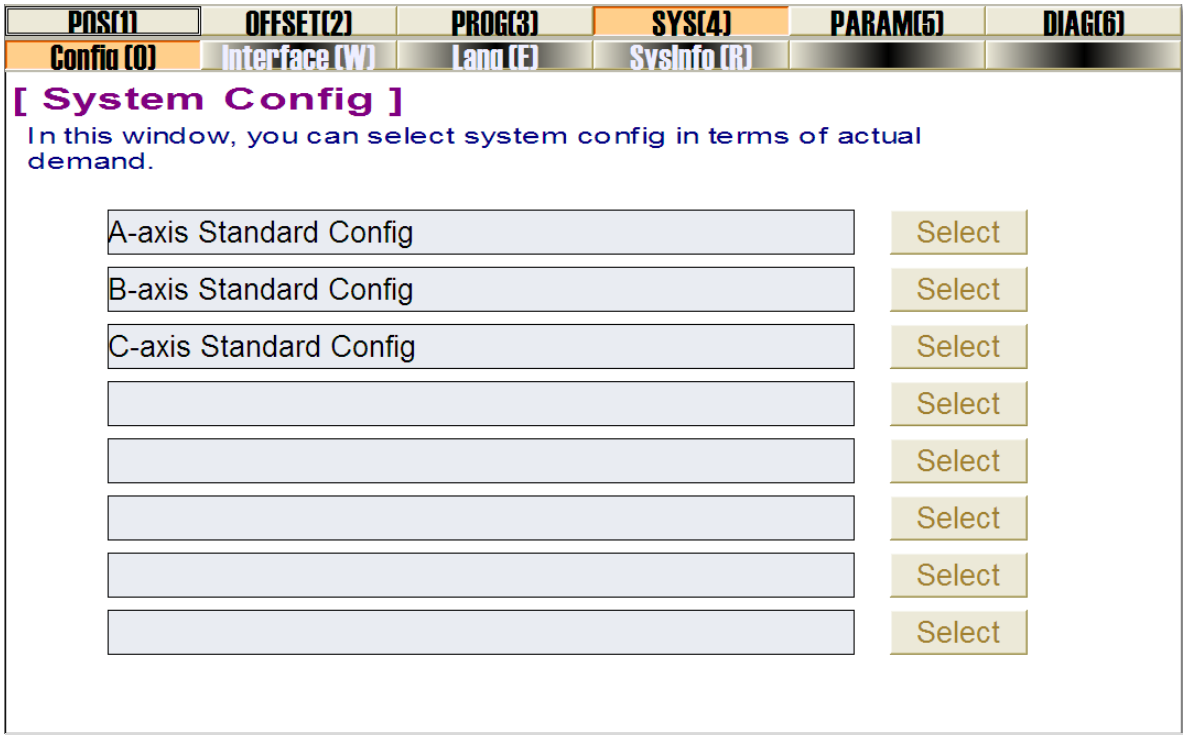


Fig. 2-4 Configuration management window

2.2.2. Reset Machine

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

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

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

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

CAUTION

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

2.2.3. Load Program File

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

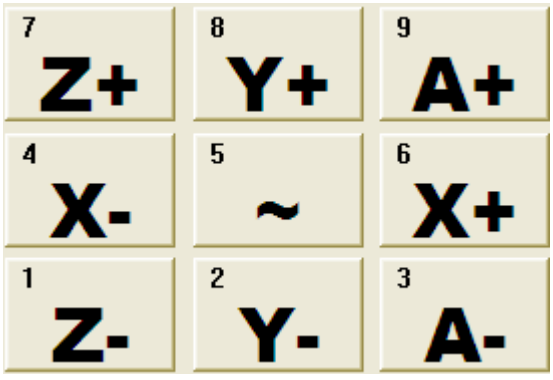
Select “Open & Load” from “File” to open a dialog box where you can choose a program file to be processed.

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

2.2.4. Manual Operation

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

As shown in the picture on the right, they are axis numeric direction keys, with the number on the left upper side being their shortcut keys. Key “~” is the rapid motion key with shortcut key being 5, and it should be used together with other axis numeric direction key.



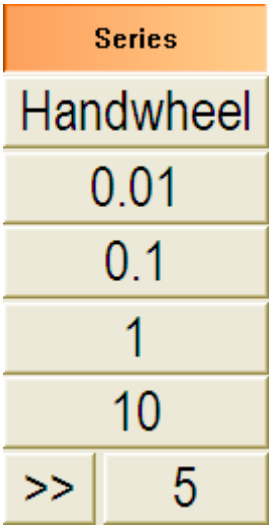
For example, clicking key “X-” or pressing number 4 on the mini keyboard directly will make the X axis

move towards negative direction at jog speed; while pressing number keys 4 and 5 on the mini keyboard will make the X axis move towards negative direction at rapid jog speed.

Handwheel mode contains three secondary modes, namely, jog, handwheel and stepping.

Key “Handwheel” on the right side is the handwheel selection key.

For more details about handwheel, please refer to Chapter 6.1.1.



CAUTION

On *Manual* operation panel, whatever the activation of numeric keys on the mini keyboard is, they can be used to manually move the axis. Please note that numeric keys on the main keyboard are used to switch over among functional windows and three modes.

2.2.5. Set Workpiece Origin

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

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

CAUTION

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

2.2.6. Check Polarity of IO Ports

In [IOPorts] window under [DIAG] functional area, all input and output ports information are displayed, which is very helpful for system monitoring and troubleshooting.

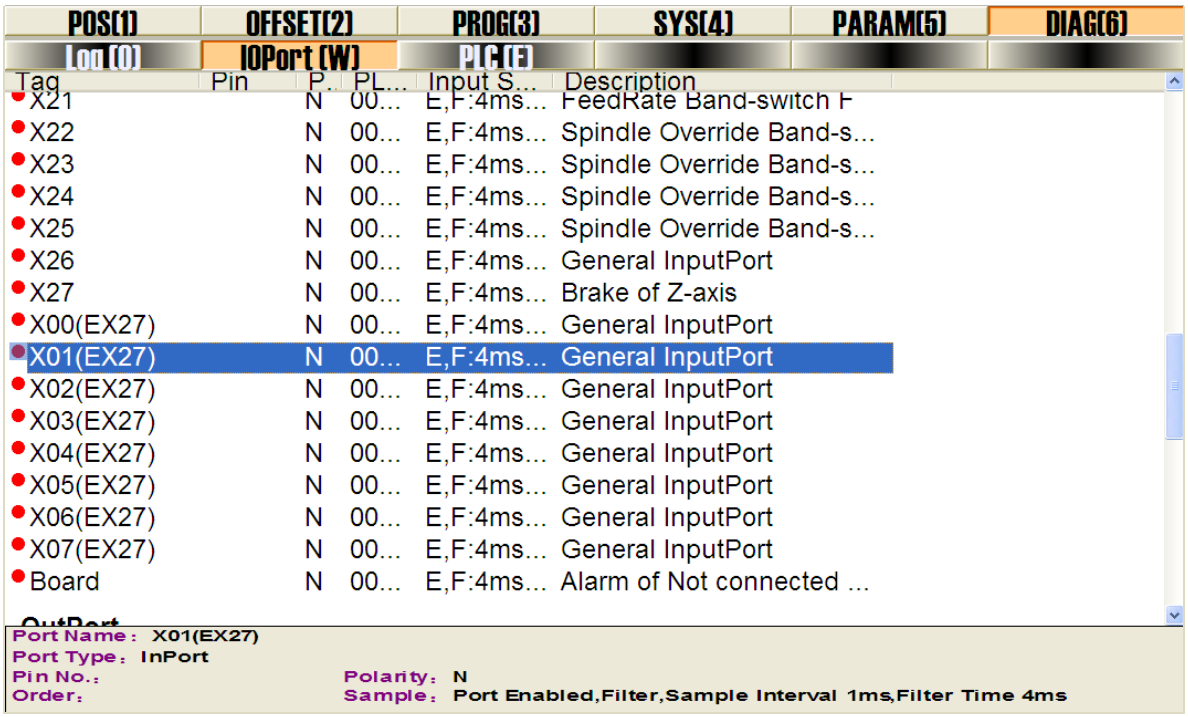


Fig. 2-5 IO ports of hardware

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

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

CAUTION

For an input port, port attribute can be set. In [IOPort] window, click F7 “PortAttr” to open a dialog box named “Inport Sampling Setting”, where sampling interval, filter function, port enabled, port name and description can be set.

2.2.7. Set Speed-related Parameters

According to the real situation of the machine, set operator’s parameters N64020 “RAPID_TRAVEL_FEEDRATE”, N64021 “DEFAULT_FEEDRATE”, and N64030 “REV_RAPID_TRAVEL_FEEDRATE”.

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

RAPID_TRAVEL_FEEDRATE: it refers to G00 feedrate, which is the feedrate when axis is locating or positioning. Unit for G00 of a linear axis is *mm/min*, while of a rotary axis is *rpm* (revolutions per minute).

2.2.8. Execute Auto Machining

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

Start Auto Machining

1) There are three ways to execute auto machining:
2) Select “Start” item under “Operate” menu list.
3) In auto mode, click button F9 “Start”.
4) In auto mode, press the shortcut key F9 on keyboard.

Stop

During auto machining, there are three ways to stop machining and turn the system into “IDLE” state:

1) Select “Stop” item under “Operate” menu list.
2) In auto mode, click button F11 “Stop”.
3) In auto mode, press the shortcut key F11 on keyboard.

Pause

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

1) Select “Pause” item under “Operate” menu list.
2) In auto mode, click button F10 “Pause”.
3) In auto mode, press the shortcut key F10 on keyboard.

3. Software Functions

3.1. Axis Control

For four axes system, up to four axes can be supported, and it can be used to control X/Y/Z/A, X/Y/Z/B and X/Y/Z/C type machine. For five axes system, as implied by the name, up to five axes can be supported, and it can be used to control X/Y/Z/A/B, X/Y/Z/A/C and X/Y/Z/B/C type machine. Please refer to Chapter 6.1.3 for detailed definitions of each axis. In [Config] window under [System] functional area, configuration can be switched. However, please note that configuration selection and switchover should be done by the manufacturer.

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

For five axes system, the software interfaces and operations are similar with that of four axes system, except for an additional axis. The following chapters introduce software functions taking four axes system as an example, please notice that.

◆ **Relevant parameters (of MFR's access)**

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

3.2. Tool Measurement

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

3.2.1. Fixed Calibration

Fixed calibration refers to measurement operation at a certain fixed position on the machine. The tool sensor position is decided by parameter setting of N75210 “FIXED_CALI_POS”.

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

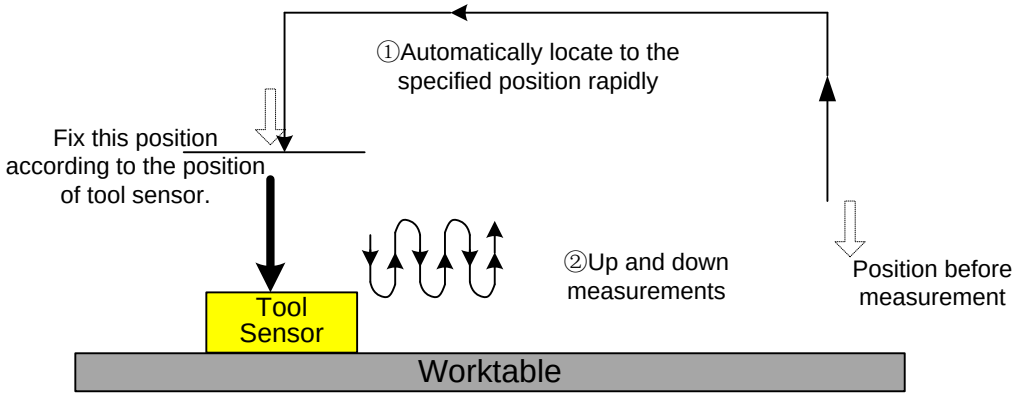


Fig. 3-1 Fixed calibration process

During fixed calibration, the machine coordinate will be recorded when tool nose touching the surface of tool sensor. After tool calibration, the system will automatically make the coordinate subtract from the sensor thickness and set it to the tool offset. At the same time, calibration result (the machine coordinate of the tool nose when it touches the sensor surface) will appear in CNC info bar, as illustrated in Fig. 3-2:

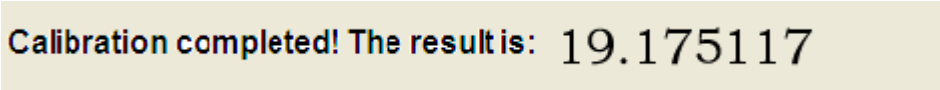


Fig. 3-2 Prompt info after fixed calibration

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

Tool Offset=Fixed Calibration Result-Tool Sensor Thickness

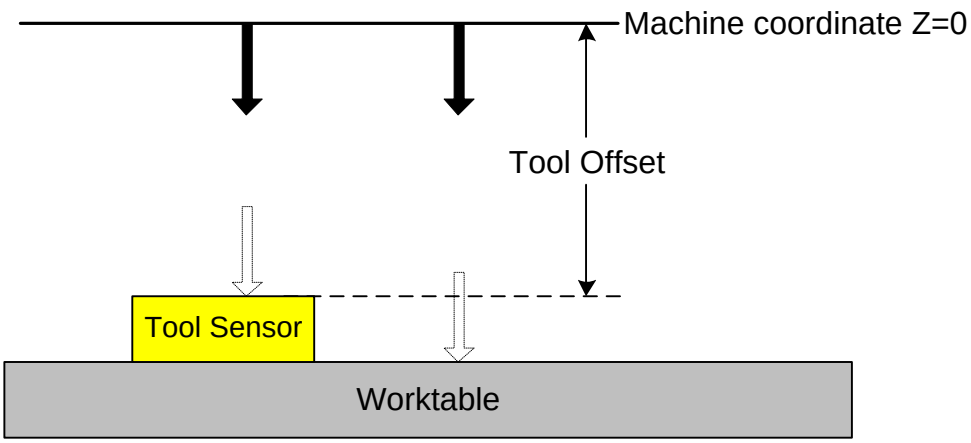


Fig. 3-3 Sketch map of tool offset

In *Manual* mode, enter into [POS] functional area, and click button F6 "Fix Cali" to conduct fixed calibration operation.

Specific calibration steps are as below:

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

CAUTION

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

◆ **Relevant parameters (N75210 is of operator ’ access and the rest of MFR’s access)**

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

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

Parameters N75001/N75002/N75020/N75025/75220/75230/N75240 will possibly be used in fixed calibration, please refer to Chapter 4.2 for details.

3.2.2. Mobile Calibration

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

Workpiece Offset=Mobile Calibration Result-Tool Sensor Thickness-Public Offset-Tool Offset

Generally, the factory setting value of public offset and tool offset is 0.

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

The sketch map of mobile calibration is illustrated as below:

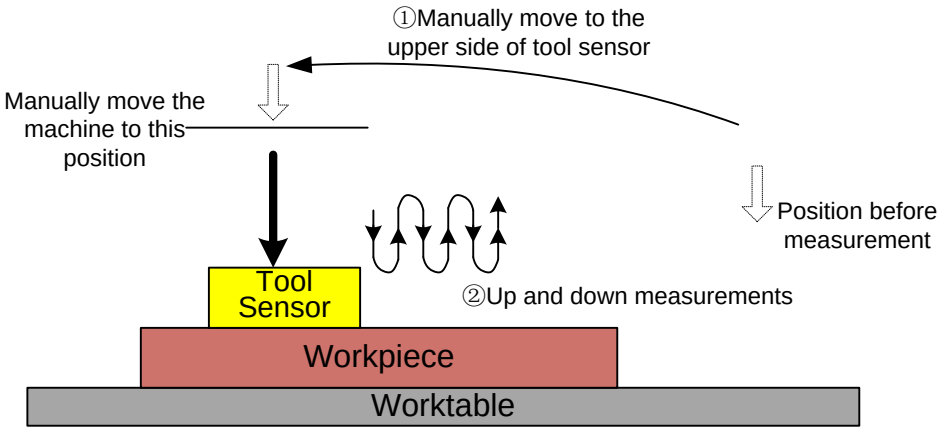


Fig. 3-4 Sketch map of mobile calibration process

Place tool sensor onto workpiece, move Z axis to the upper side of the tool sensor in *Manual* mode, and click button F7 “MobCali” in [POS] functional area. The system will automatically finish the calibration process and set the workpiece origin in Z axis.

◆ **Relevant parameters (all are of MFR's access)**

Parameter		Description	Setting Range
N75001	CALI_SPEED	The speed of tool when it approaches tool sensor surface in calibration.	-
N75002	CALI_TIMES	The times of accurate positioning when tool approaches tool sensor surface in calibration.	-
N75020	CALI_TORLERANCE	The maximum tolerance of tool calibration.	0~10
Value of parameter N75020 is the maximum tolerance of tool calibration, which is compared with average value of calibration results. When average value is smaller than this value, calibration succeeded, otherwise, calibration failed.			
N75025	CALI_PROTECT _ENABLED	Whether to protect the tool from over-travel in calibration.	No: invalid Yes: valid
N75100	MOB_CALI _THICKNESS	In mobile calibration, the height from tool sensor surface to workpiece origin in Z axis.	-1000~1000
Setting method of parameter N75100: 1) Manually move Z axis to a point above the workbench surface, and slowly move the tool nose downward until it touches the workbench surface. At this time, Z coordinate will be recorded as Z1; 2) Lift Z axis, move it to the upper side of tool sensor on workpiece, slowly move Z axis downwards until the tool nose touches tool sensor and obtain tool sensor signal. At this time, Z coordinate will be recorded as Z2; 3) Z2-Z1, and the difference is the tool sensor thickness. Set the value to parameter N75100.			
N75220	CaliWorkBenchUpper Limit	The machine coordinates of the upper limit of workbench in calibration.	Lower Limit~99999
N75230	CaliWorkBenchLower Limit	The machine coordinates of the lower limit of workbench in calibration.	-99999~Upper Limit
N75240	CaliAreaIsValid?	Whether the calibration range is effective or not.	No: invalid Yes: valid

3.2.3. Special Measurement

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

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

With manual mode activated, turn to [Measure] window under [POS] function area, as shown below, where you can conduct special measurement.

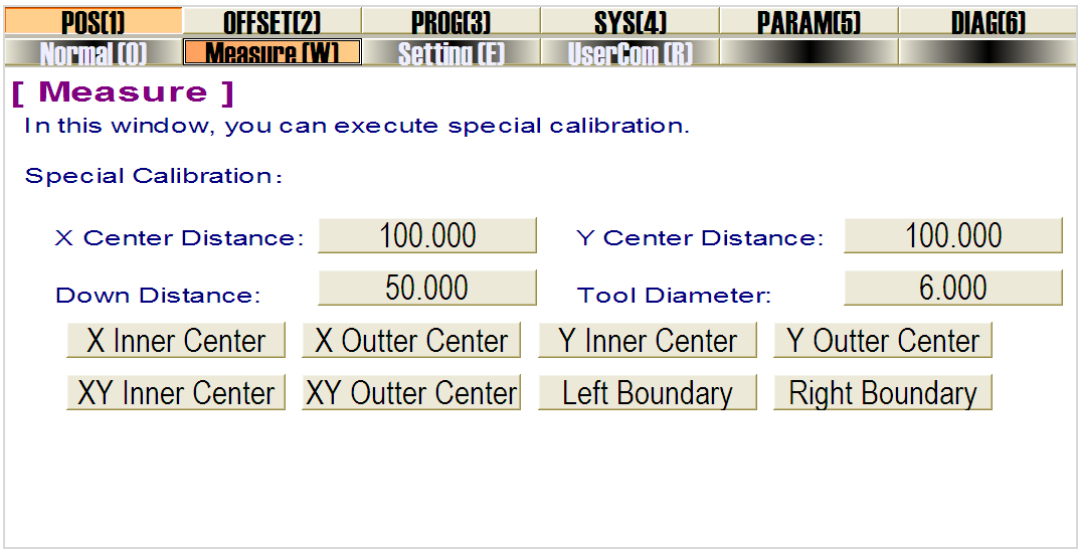


Fig. 3-5 Special calibration

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

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

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

Tool Diameter: the actual diameter of tool.

◆ **Centering**

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

Taking the process of [X Inner Center] as an example: place a conducting workpiece (copper, iron, aluminum) on the insulated workbench, and connect it to the port CUT on the terminal board, while the cutter to COM port. Before automatic centering, put the cutter on the predicated center point position, and then click [X Inner Center] to make the cutter fall [Down] distance, and translate towards the workpiece a short distance until it reaches the conducting workpiece. At this time, the circuit is conducted and the signal is sent to the system which automatically records the current axial coordinate X1. Then the cutter will move up [Down] distance, move horizontally two [X Offset], move down [Down] distance, and translate towards the workpiece a short distance until it reaches the conducting workpiece. As a result, the circuit is conducted and the signal is sent to the system which automatically records the current axial coordinate X2. The system will then calculate the X coordinate of workpiece center point and move the cutter to this center point. The sketch map is as shown below.

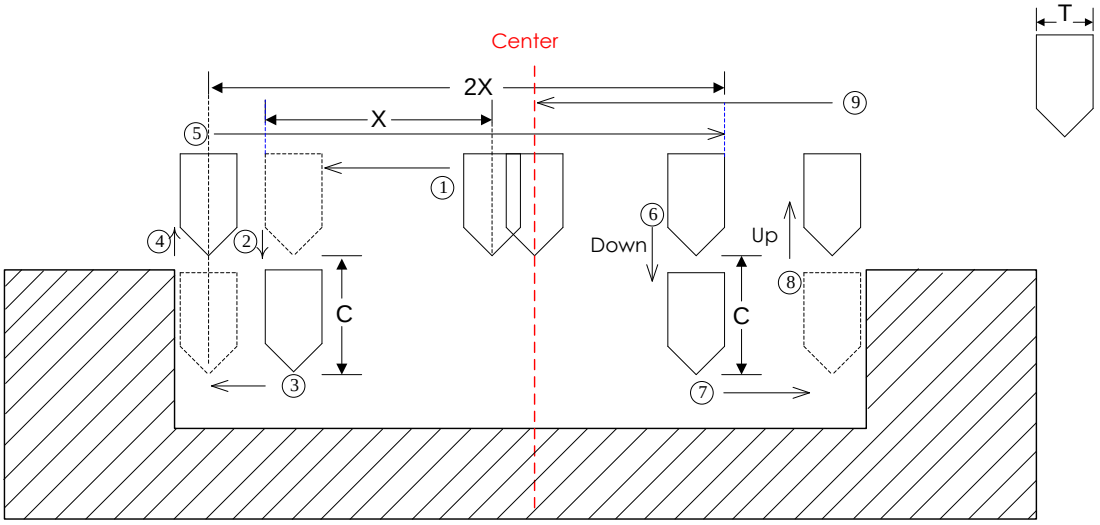


Fig. 3-6 Measurement process of [X Inner Center]

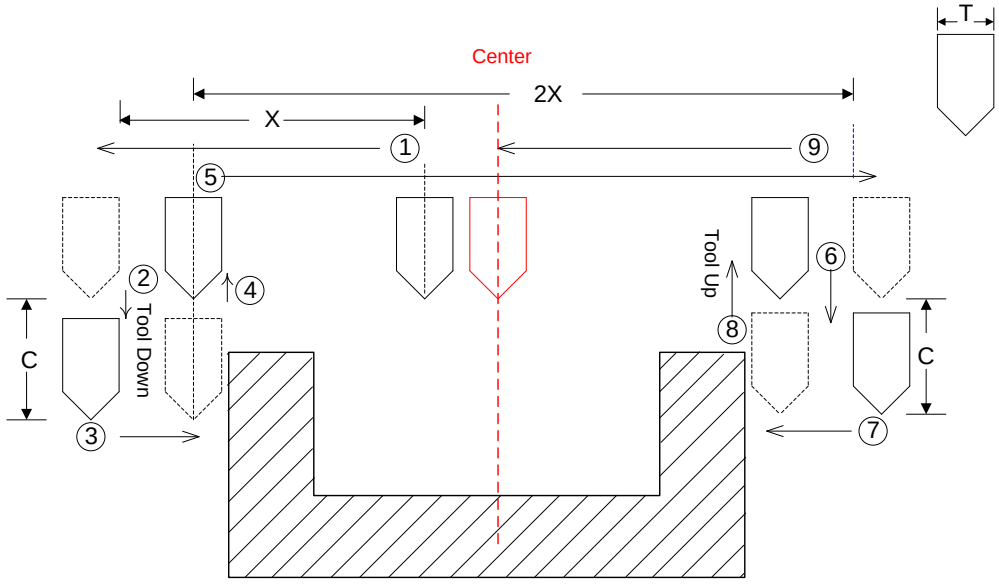


Fig. 3-7 Measurement process of [X Outer Center]

CAUTION

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

◆ **Boundary Measurement**

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

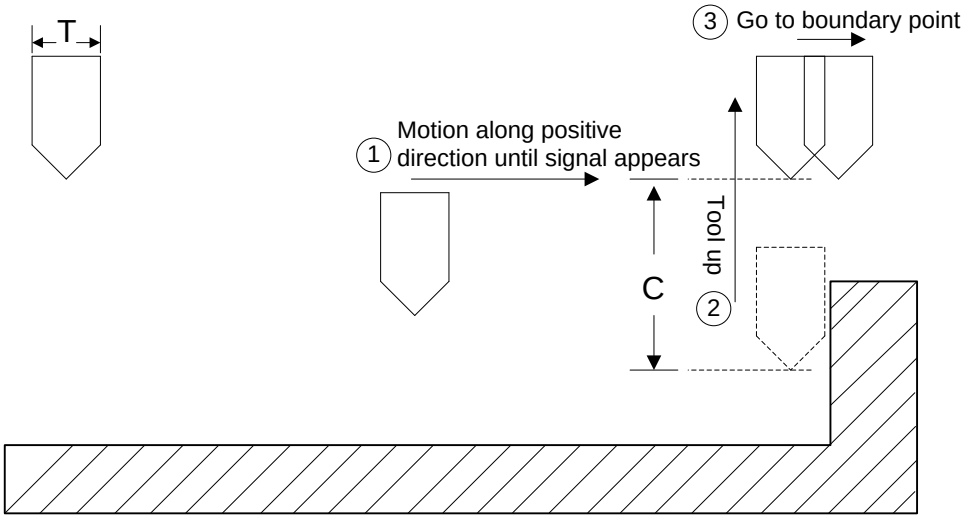


Fig. 3-8 Measurement process of +X boundary

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

3.3. Offset Management

3.3.1. Workpiece Coordinate System (WCS)

In programming, programmers select a certain given point on workpiece as origin (also called programming origin) to establish a new coordinate system (i.e. workpiece coordinate system), also a set of right-hand coordinate system. The origin of WCS, i.e. workpiece origin, is fixed relative to a certain point on workpiece and floating relative to the machine origin. The selection of origin of WCS should meet the conditions of simple programming, simple dimensional conversion, and small machining error.

The corresponding coordinate systems of workpiece offset are G55, G56, G57, G58, G59 and G54 (the default coordinate system after the system is opened), as well as G59P1 ~ G59P256. And the relationship of workpiece offset and machine coordinate system is shown in Fig. 3-9.

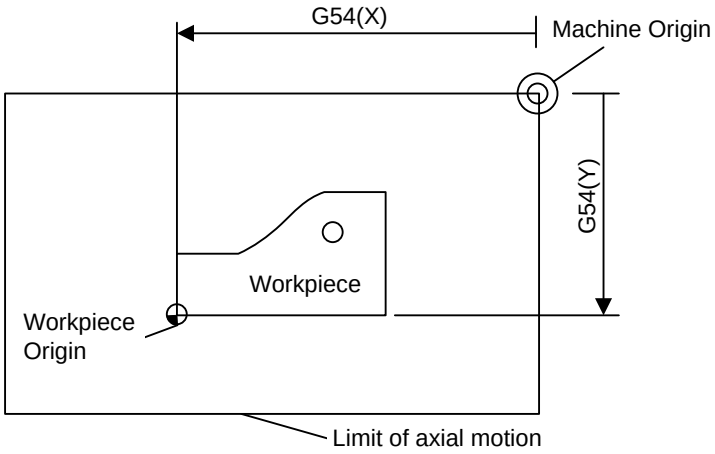


Fig. 3-9 The relationship between workpiece offset and machine coordinate system

One, two or multiple workpiece offsets can be used in machining program. As shown in Fig. 3-10, if three work pieces are installed on the workbench, then each workpiece holds a workpiece origin relative to G code of WCS. The programming example is as follows: drill one hole on each of the three work pieces, with calculation depth as Z-0.14.

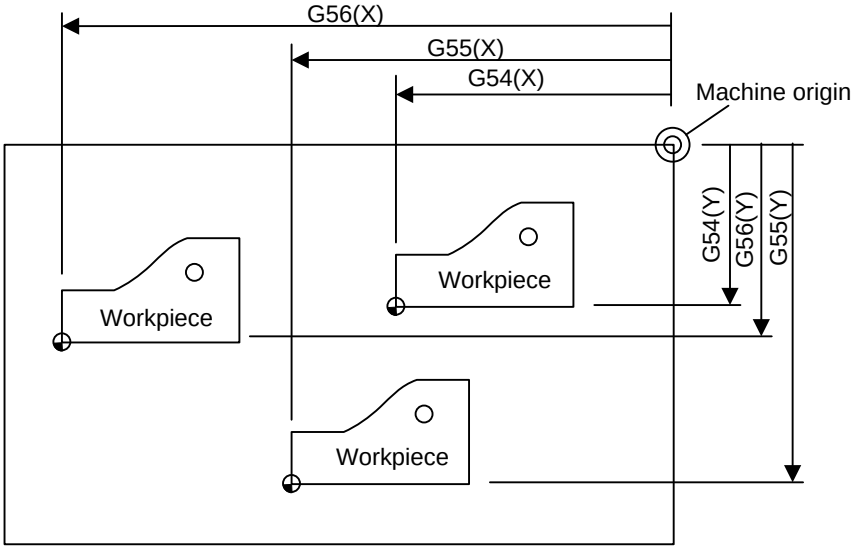


Fig. 3-10 Sketch map

```
O1801
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X5.5 Y3.1 S1000 M03          （Select G54）
N4 G43 Z0.1 H01 M08
N5 G99 G82 R0.1 Z-0.14 P100 F8.0
```

```
N6 G55 X5.5 Y3.1                    （Switch to G55）
N7 G56 X5.5 Y3.1                    （Switch to G56）
N8 G80 Z1.0 M09
N9 G91 G54 G28 Z0 M05               （Switch to G54）
N10 M01
…
```

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

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

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

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

3.3.2. Extended WCSs

Also known as additional workpiece coordinate system. Up to 256 extended WCSs can be supported, whose number is determined by setup value of parameter N61111 “Size of Extra Workcoors”. When parameter N61110 “Support Extra Workcoors” is set to “Yes”, extended WCSs are available in programming. Note that extended WCSs are extension for G59 (from G59P1 to G59P256)

◆ **Related Parameters (all are of MFR’s access)**

Parameter		Description	Setting Range
N61110	Support Extra Workcoors	Whether the extended WCSs can be supported.	0;1
N61111	Size of Extra Workcoors	The number of extra WCSs can be supported.	1~256
N61110 is set to “No” by default. When more than 6 parts need to be setup on worktable at the same time, set N61110 to “Yes” to activate the extended WCSs, and further set N61111 to a desired value.			

3.3.3. Offset Setting

[Offset] window under [OFFSET] function area shows the current public offset and workpiece offset of each axis. As shown in below, click “Advanced Setting” button to open “Offset Setting” dialog box, as Fig. 3-12. Following two figures change according to setup value of parameter N61100

"Support Extra Workcoors". When the parameter is set to "1" (default), namely, to simple offset mode, [Offset] window and "Offset Setting" dialog box are displayed as Fig. 3-11 and Fig. 3-12 separately.

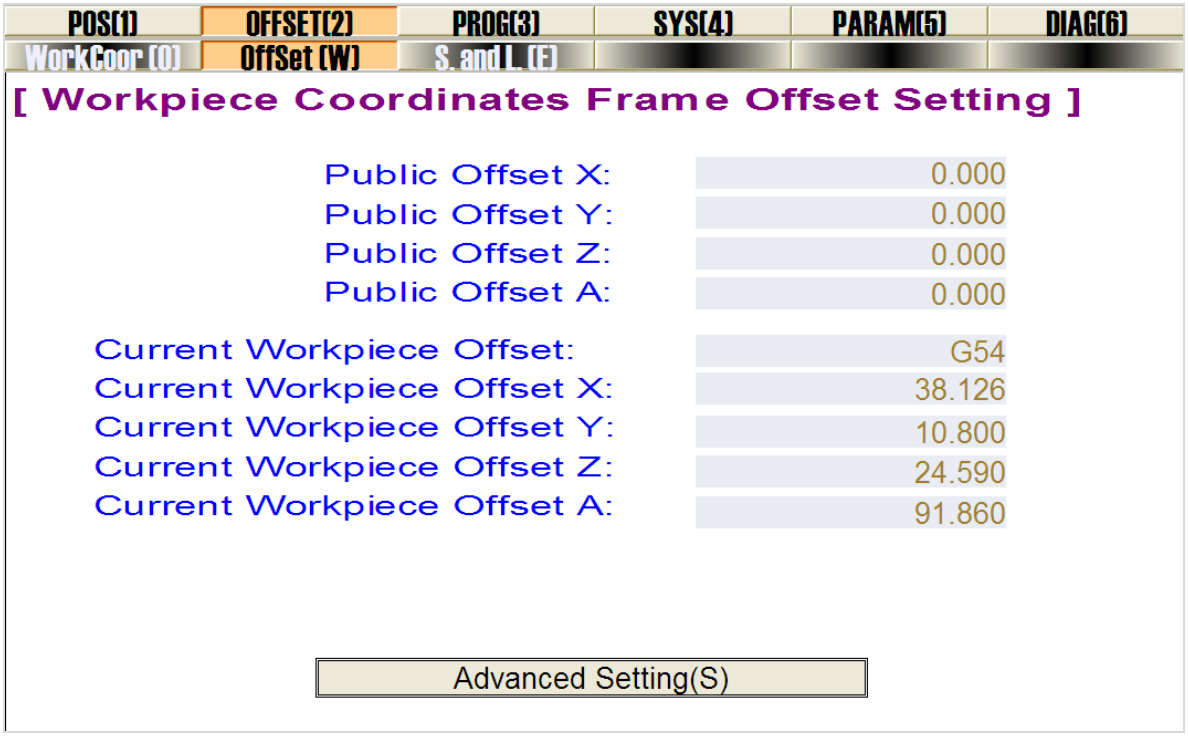


Fig. 3-11 Offset

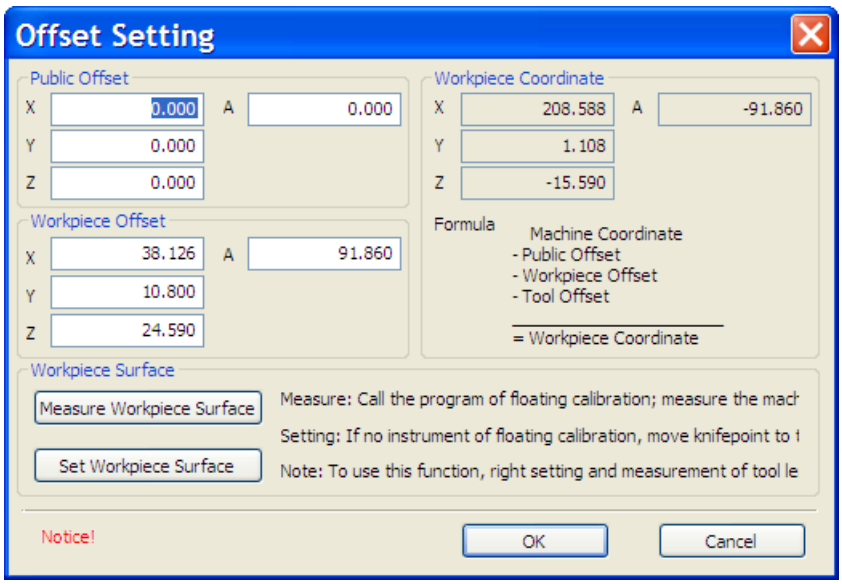
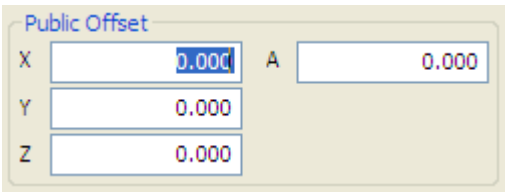


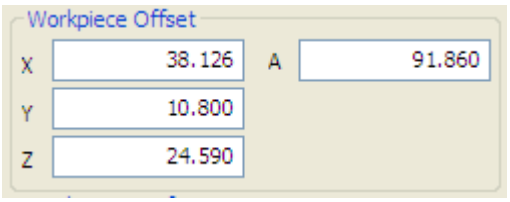
Fig. 3-12 Offset setting dialog box

Public offset



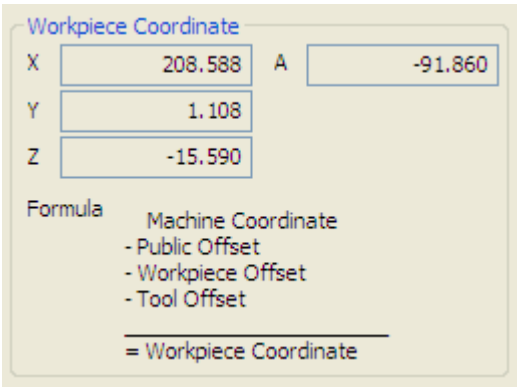
Also called external offset, it is used to record temporary adjustment value of the workpiece origin. Note that this value can only be modified manually.

Workpiece offset



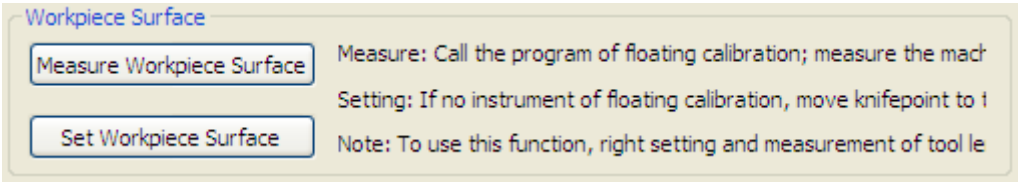
It shows the distance of the workpiece origin relative to the machine origin, that is, the machine coordinate of the workpiece origin. You can manually input number to define the machine coordinate of the workpiece origin (it is NOT recommended).

Workpiece coordinate



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

Workpiece surface



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

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

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

CAUTION

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

workpiece origin. Up to 10 groups of origins can be saved. After a workpiece origin is saved, it can be called and re-used through item “Read workpiece origin” under menu “Operate” afterwards.

3) Workpiece origin saving and reading can also be done on [Register] window under [OFFSET] function area, which shares same effect with above step.

4) If parameter N61100 is set to other values other than the default one, you can make offset setting according to tips in “Offset Setting” dialog box.

◆ **Related Parameter (of developer’s access)**

Parameter		Description	Setting Range
N61100	COORDINATEOFFSETSCHEMA	Coordinate offset mode. 1: SWST; 2: MWST; 3: SWMT; 4: MWMT.	1;2;3;4
	Four modes of coordinate offset are offered, including SWST (simple mode); MWST (multiple parts mode); SWMT (multiple tools mode); MWMT (multiple parts and tools mode). The default value is 1. Note that the setup value of this parameter determines contents presented on “Offset Setting” dialog box in Fig. 3-12.		

CAUTION

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

3.4. Program Management

3.4.1. Program Wizard

The software offers four basic processing program wizards: circular pocket, circular frame, rectangular pocket and rectangular frame. You just need to input some simple parameters to complete the milling operation of circular frame and rectangular frame, etc.

Turn to [Process Wizard] window under [PROG] function area, as shown in Fig. 3-13. Press button F1~F4 to switch to corresponding wizard windows.

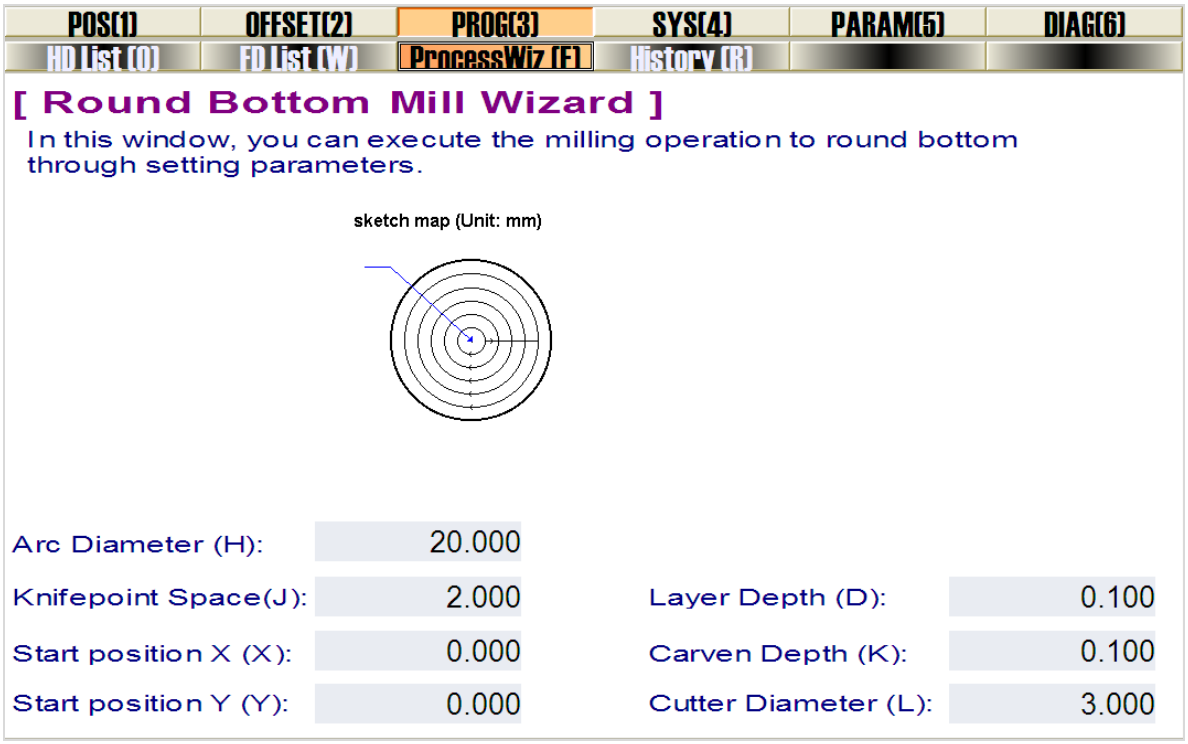


Fig. 3-13 Program wizard---Round bottom mill

After all parameters are well set, press F5 to load current wizard program into the system, and press F6 to save the program.

3.4.2. Program Files

After [PROG] functional area is active, press shortcut key Q to access [Hard disk list] window, and press W to access [Floppy disk list] window.

[Hard disk list] window is as shown in Fig. 3-14, which shows all program files and folders under directory D:\NCFILES. You can click buttons on operational bar at the lower part of window to load, edit, delete, create as well as rename the program. For file folder, double click to enter it, after the program file is selected, F1~F8 buttons can be used to enable functions as button names implies.

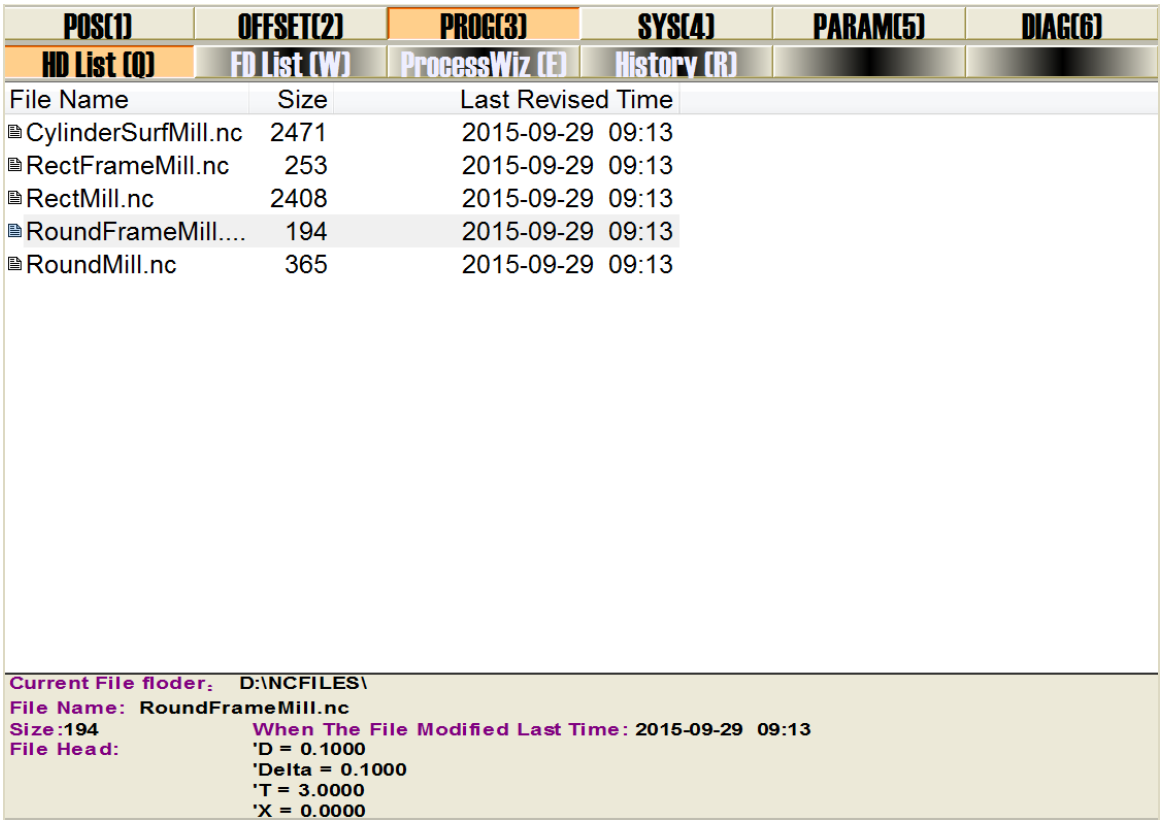


Fig. 3-14 Program files list in hard disk

CAUTION

1) Users can save and store program files under directory *D:\NCFILES*, thus, operations to those programs can be directly enabled under this directory.

2) Apart from editing on [Hard disk/Floppy disk list] window, users can edit the program by clicking item "Open and edit" or "Edit current program" under "File" menu. Note that "Edit current program" item validates only when G code program file is active.

[Floppy disk list] window is as shown in Fig. 3-15, where displays program files in removable disk. If several removable disks exist, press F3"Select disk" to make a choice.

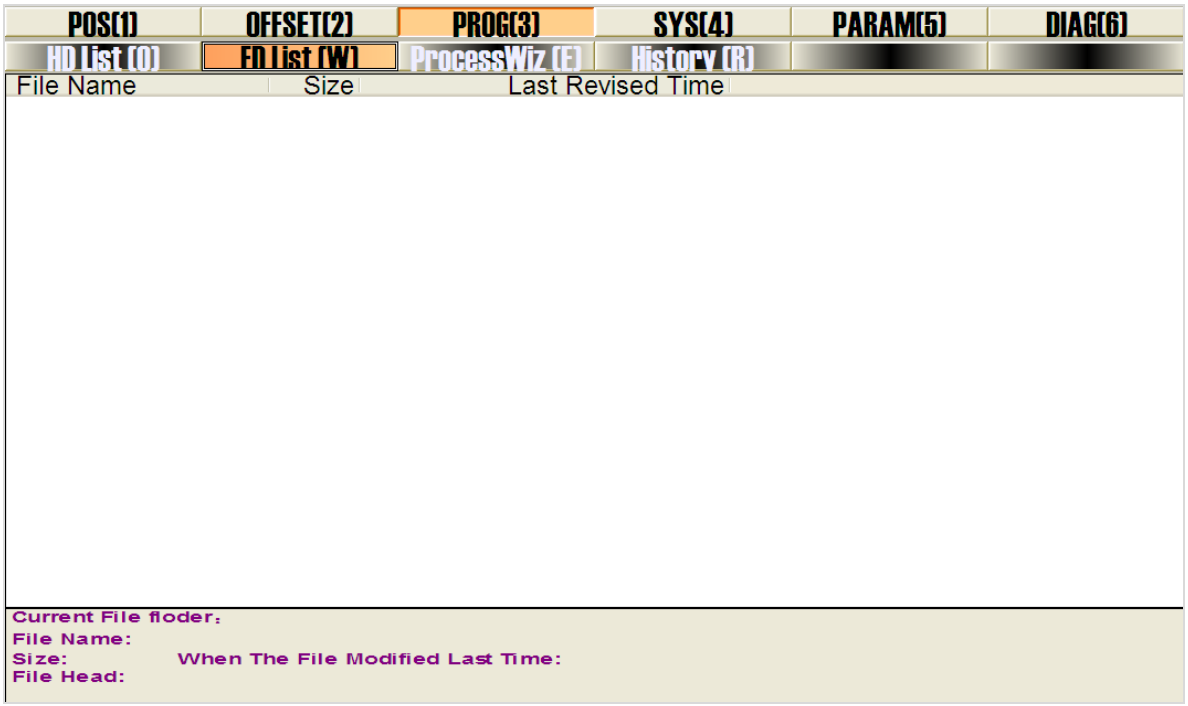


Fig. 3-15 Program files list in floppy disk

3.5. Speed Management

3.5.1. Spindle Speed

Spindle speed refers to spindle rotational speed, with unit of “rpm (revolution per minute)”.

In auto mode, turn to [Setting] window under [POS] functional area, as shown in Fig. 3-16. Click figure button next to each parameter to open input box, where you can enter desired spindle speed. Modification takes effect immediately.

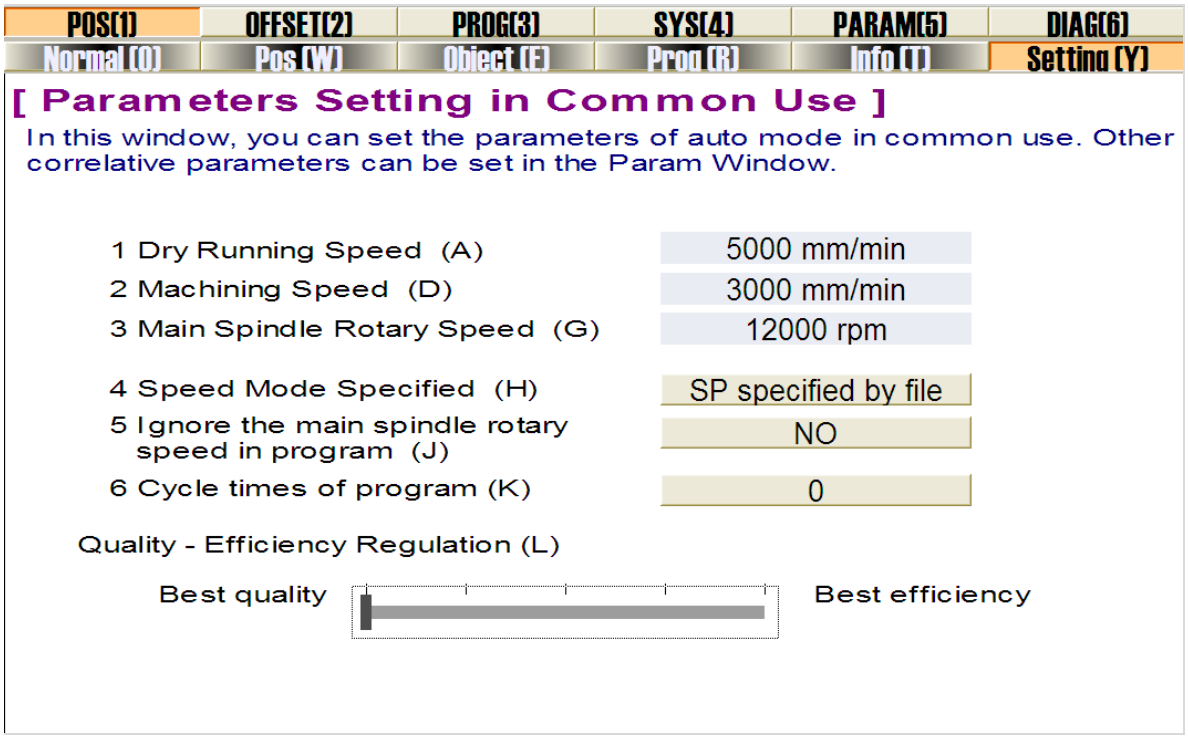


Fig. 3-16 Spindle speed setting in auto mode

CAUTION

As shown in Fig. 3-16, if parameter “Ignore the main spindle rotary speed in program” (corresponding to N72002 “IGNORE_PROG_SPINDLE_REV”) is set to “No”, the spindle speed specified by S command in the program file will be enabled if S command exists; otherwise, spindle speed specified by “Main spindle rotary speed” will be enabled in machining. While if parameter “Ignore the main spindle rotary speed in program” is set to “Yes”, spindle speed specified by “Main spindle rotary speed” will be enabled regardless of S command in program file.

3.5.2. Feedrate

It refers to feeding speed or Gxx speed, the running speed in machining specifically. Following are two methods to set feedrate.

1) In auto mode, turn to [Setting] window under [POS] functional area, as shown in Fig. 3-16. Click figure button next to “Machining speed” to open input box, where you can enter a desired number. Modification takes effect immediately.
2) Turn to [Parameter] window under [PARAM] functional area, locate to parameter N64021 “DEFAULT_FEEDRATE”, double click it and input in pop-up box. Modification takes effect immediately.

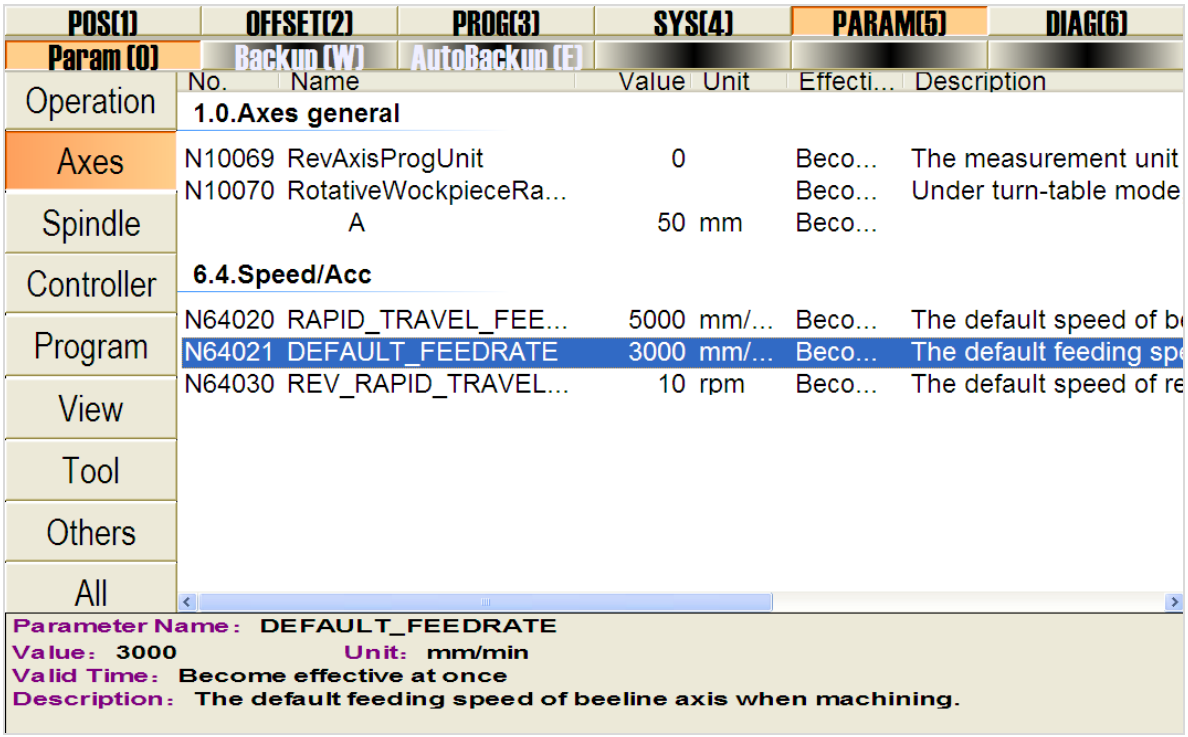


Fig. 3-17 Feedrate setting parameter

CAUTION

1) The machining speed (Gxx speed) should be smaller than the dry running speed (G00 speed).

2) If item "Speed Mode Specified" in Fig. 3-16 is set to "SP specified by the file" (corresponding to parameter N72001 "SPEED_ASSIGN_TYPE" is set to "0"), feedrate specified by F command will be enabled if F command exists in the program file; otherwise, feedrate specified by "Machining speed" (corresponding to setup value of parameter N64021) will be enabled in machining. While if the item is set to "Default SP" (corresponding to parameter N72001 is set to "1"), the "Machining speed" will always be enabled regardless of existence of F command in the program file.

3.5.3. G00 Speed

G00 speed refers to the axial speed when machine is positioning. Unit for linear axis is *mm/min*, while for rotary axis is *rpm* (revolution per minute). Like setting of feedrate and spindle rate, there are also two methods to set G00 speed: modification in [Setting] window or in [Parameter] window. In the parameter list, find parameters N64020 "RAPID_TRAVEL_FEEDRATE" and N64030 "REV_RAPID_TRAVEL_FEEDRATE" and set them to desired values.

CAUTION

Dry running speed should be larger than machining speed, namely, G00 speed should be larger than Gxx speed.

3.5.4. Jog Feedrate/ Rapid Jog Feedrate

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

1) Turn the machine to manual mode, and access [Setting] window under [POS] functional area, as shown in Fig. 3-18. Press number button next to each parameter to open an input box, where you can set appropriate value and confirm it.
2) Turn to [Parameter] window under [PARAM] functional area, find parameters N71000 “JOG_VOL” and N71001 “RAPID_JOG_VOL” and modify setup values.

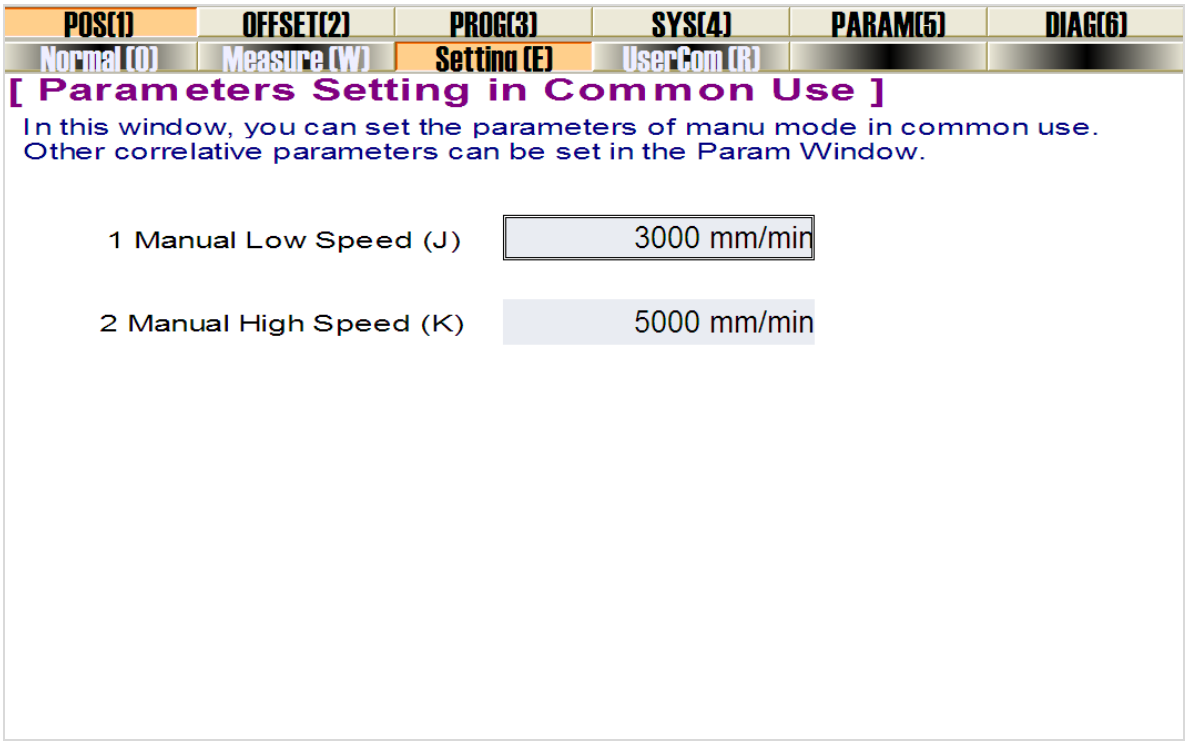


Fig. 3-18 Feedrate setting in manual mode

CAUTION

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

3.5.5. Acceleration

Acceleration is a physical concept describing velocity changing rate of a moving object, equaling to ratio of velocity changing value against time used. It includes angular acceleration of rotary axis, machining acceleration, free run acceleration, acceleration at corner as well as jerk (acceleration of acceleration). See details in table below.

Parameter		Description	Setting Range
N64080	ANGLE_ACCELERATION	It specifies angular acceleration of rotary axis.	0.001 ~ 100000
N64100	AX_LINEAR_MACH_ACC	It specifies the max. acceleration of each linear axis in machining.	0.001 ~ 10000000
They are used to describe the acceleration/ deceleration capability of each feed axis, with unit “mm/s^2”, depending on the physical feature of machine, such as the quality of motion part, torque, cutting load and resistance of the feed motor. The larger the value of the parameter is, the less time the machine will spend in acceleration/ deceleration during motion process, the higher the efficiency is. Generally, for servo motor system, it should be within “400 ~ 1200”. Set a smaller value at first, and then repeatedly execute typical motion for a period of time. If there is no abnormal situation, gradually increase the value. If abnormal condition occurs, reduce the value, with “50% ~ 100%” insurance allowance.			
N64101	AX_LINEAR_POST_ACC	It specifies the max. acceleration of each linear axis when machine is positioning.	0.001 ~ 10000000
N64120	AX_CON_ACC	The max. resultant feed acceleration of adjacent two axes. 1~2 times of the single axis acceleration is recommended.	0.001 ~ 100000
They are used to describe the acceleration/ deceleration capability of multiple linkage axes, with unit “mm/s^2”. The larger the value of the parameter is, the higher the allowable velocity is in arc motion. Generally, for servo motor system, it should be within “1000 ~ 5000”. While for heavy machine, it should be a smaller value. Set a smaller value at first, and then repeatedly execute typical motion for a period of time.			

Parameter		Description	Setting Range
If there is no abnormal situation, gradually increase the value. If abnormal condition occurs, reduce the value, with "50% ~ 100%" insurance allowance.			
N64150	AX_ACC_ACC	The acceleration of the linear acceleration under S-type velocity curve.	0.001 ~ 100000

CAUTION

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

3.5.6. Reference Circle Speed

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

◆ **Related parameter (all parameters are of MFR's access)**

Parameter		Description	Setting Range
N64208	REF_CIRCLE_MAX_VELO	The max. velocity of circle with 10mm-diameter. The value is used to calculate the maximal velocity of circle motions.	0.001 ~ 100000
After installation of machine, users can make the machine process an arc, in which vibration will occur due to centrifugal force. The higher the speed is, the stronger the vibration will be. To minimize the vibration, the software makes limitation on machining speed when process an arc. Taking default setting as an example, diameter of reference circle is 10mm, and maximum linear velocity is 1800mm/min, then according to formula for centripetal acceleration: $$a = \frac{V^2}{R}$$ Where: $r = (10/2)mm$; $v = 1800mm/\min$; Centripetal acceleration a can be figured out, which will be the maximum allowable centripetal acceleration when machining other arc or circle. If the arc speed is too big, resulting in a larger acceleration value than the formula result, limitation to the speed will be enabled.			

Parameter		Description	Setting Range
N64209	CIRCLE_MIN_VELO	The minimal velocity of circle motions.	0.001 ~ 100000
According to formula above, the result of a is not the sole factor to decide the activation of speed limitation. According to the formula, it can be concluded that time will be wasted when a circle with a relatively small diameter is being machined. For this reason, the parameter is developed, with which no matter how small the circle radius or diameter is, the actual machining speed will not be smaller than the minimum speed specified by this parameter.			

3.6. Simulation and Track

3.6.1. Simulation

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

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

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

1) By clicking "*Operate ==> Simulation mode"*, simulating machining will be enabled, starting from the first program line. At this time, sub-menus under "*Operate"* menu change, with "*Simulation mode"* changing to "*Stop simulation and withdraw the simulation mode"*, "*Start"* changing to "*Simulation start"*, "*Pause"* changing to "*Simulation pause"*, and "*Stop"* changing to "*Simulation stop"*. If you want to disable simulation mode, click menu item "*Stop simulation and withdraw the simulation mode" to exit* when simulation finishes, or firstly click "*Simulation stop"* to stop simulation and then click "*Stop simulation and withdraw the simulation mode"* to exit.
2) In auto mode, turn to [POS] functional area, click button F4 [Simulate] first and F9 [Start] next to start simulation machining from the very beginning. Click F11 [Stop] to stop simulation, and F10 [Pause] to pause. If you want to exit from simulation mode, firstly click F11 [Stop] and sub menu "*Stop simulation and withdraw the simulation mode"* under "*Operate"*.

Below is [Normal] window under [POS] functional area in simulation mode.

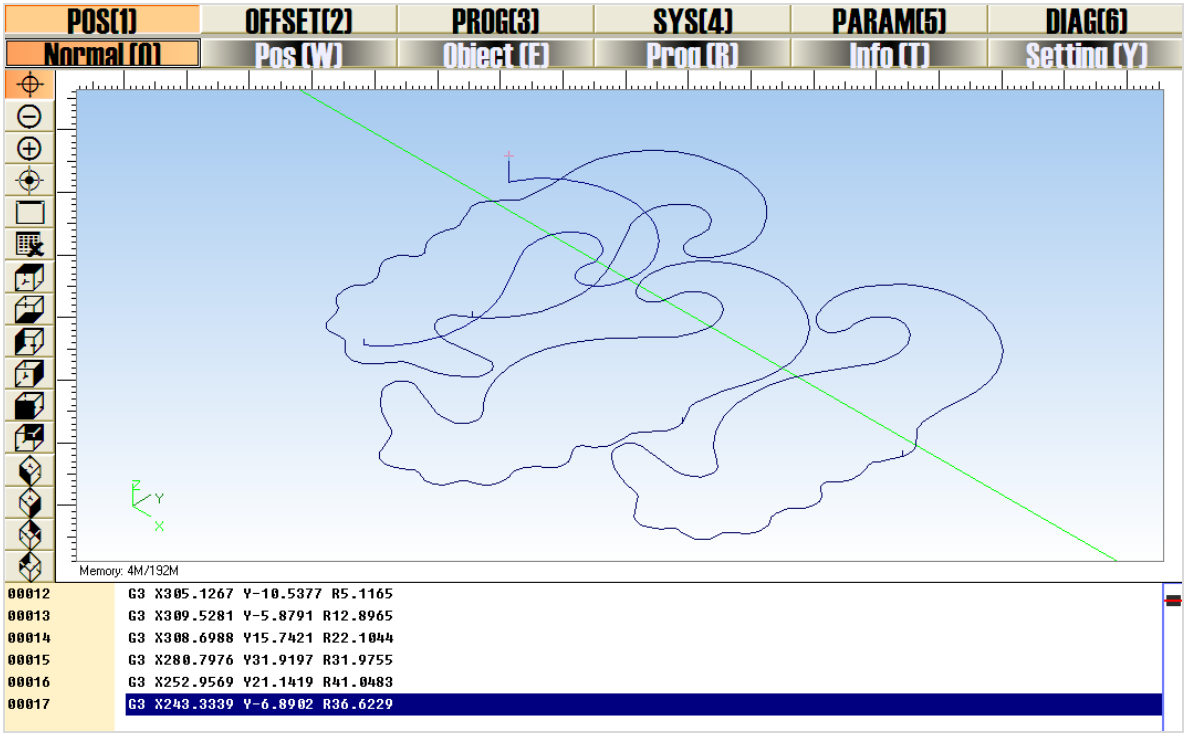


Fig. 3-19 Simulation interface

3.6.2. Track

In [Normal] and [Object] windows under [POS] functional area, different tool path tracks can be displayed, including simulation track, pre-loaded machining track as well as actual machining track.

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

Apart from the absence of file information display area, [Object] window shares the same with [Normal] window. Here is introduction to operation on track, taking [Normal] as an example.

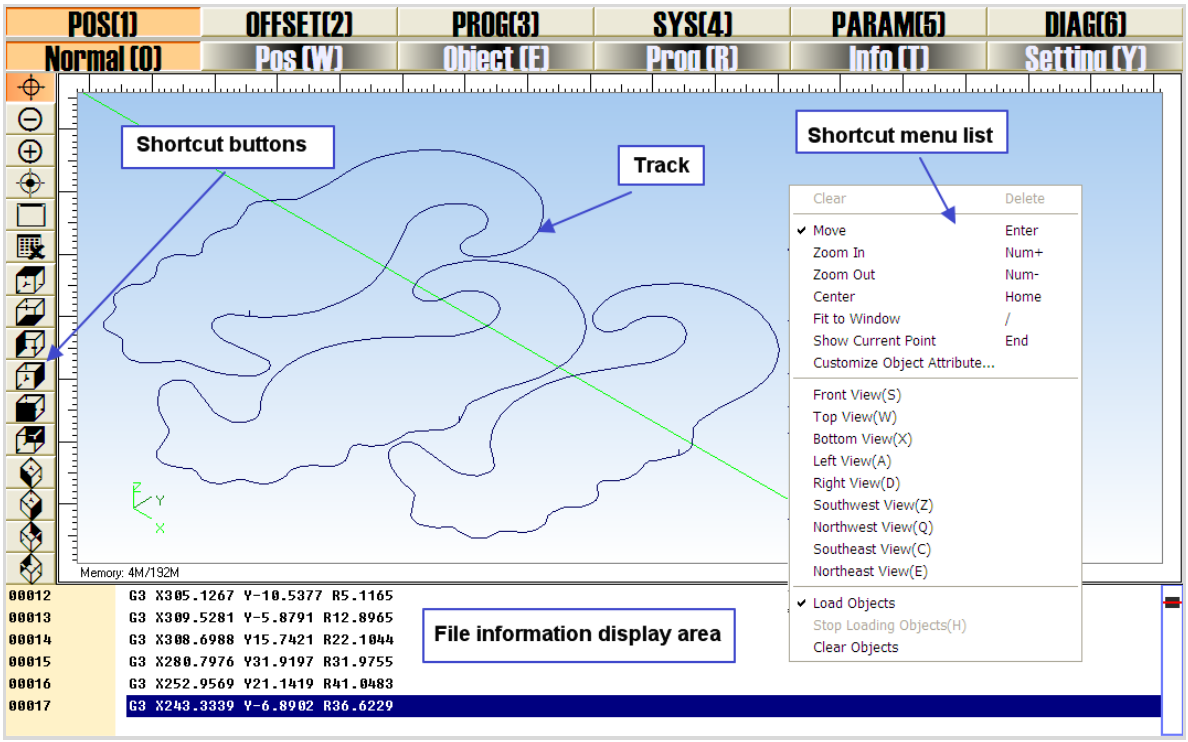


Fig. 3-20 Normal view

As shown in Fig. 3-20, right-click the mouse, and a shortcut menu list will pop up, most of which share the same effect with shortcut buttons on the left side.

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

Move: it is used to move the track. Shortcut keys: Enter + direction keys.

Zoom in/out: they are used to scale the track, with shortcut keys “+/-”. In addition, the scroll of the mouse can be used to zoom in/out the track.

CAUTION

1) When scaling the tool path track, the user can scale a part of the track by selecting the region by mouse.
2) When zooming in the track to the largest extent, it will be switch to zoom out automatically; likewise, when zooming out the track to the largest extent, it will be switched to zoom in automatically.

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

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

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

Customize object attribute: it is used to make individual setting of track mode and trajectory color. With this item selected, a dialog box will pop up, as shown in Fig. 3-21.

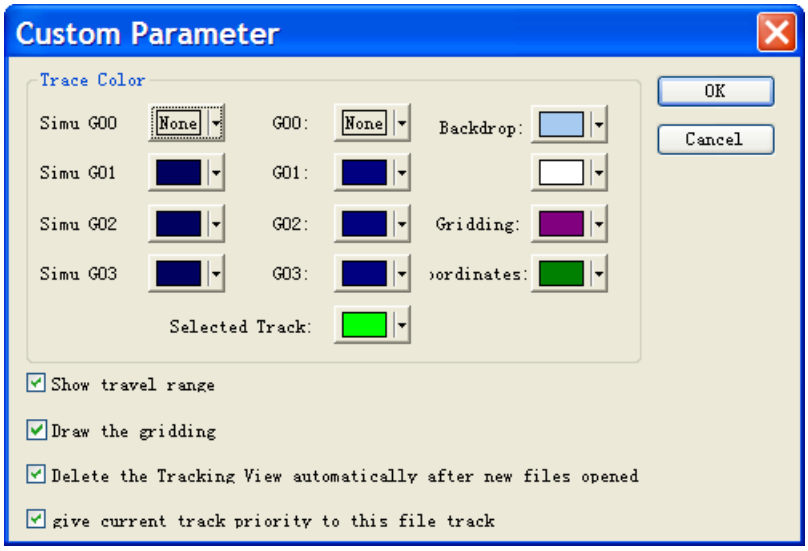


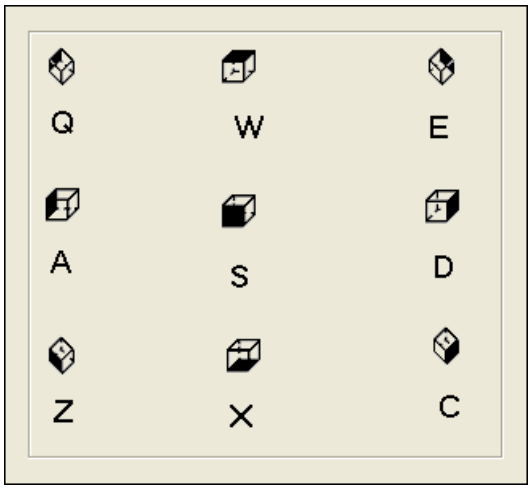
Fig. 3-21 Custom setting dialog box

Click color box next to each item to select desired color for it. You can customize a color as well.

CAUTION

1) If the trace color of a certain code is set as transparent, the track of that code will be hidden. If the background color is set as transparent, the window can’t be refreshed properly. Therefore, the user needs to be cautious when selecting “transparent”.
2) The background color can be combination or color gradient of two self-defined colors.
3) Tacking color for motion in manual mode or stepping mode is the color for G01, while pre-analyzed track color is the color for simulation codes.

View function: 9 types of frequently used observance views are offered. Press shortcut key “S” in [Normal] window to open shortcut keys reminder box, as shown on the right side. You can switch to different views by corresponding shortcut keys.



Load object/Stop loading object: it is used to load the pre-analyzed track into the system before actual machining begins. If the file is too large, select “Stop loading object” to terminate it.

Clear object: it is used to remove the pre-analyzed track.

3.7. Handwheel Operation

3.7.1. Handwheel Mode

The system supports three operation modes: auto mode, reference point mode and manual mode, and the manual mode is further subdivided into jog, stepping and handwheel. Users can select handwheel mode by pressing 

button under [Manual] functional area, or by selecting “Handwheel” item under “Work Mode” menu.

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

When handwheel mode is activated, you can control the machine movement as follows:

1st, turn the “Axis selection button” to activate an axis to move.

2nd, turn the “Gear selection button” to decide the displacement of moving component (linear axis or rotary axis) against each gear of the handwheel. The displacement is determined by setup value of parameter N52003~N52005.

3rd, turn the “Handwheel control rotation disk” to manipulate the movement of a tool.

CAUTION

Handwheel is option.

3.7.2. Handwheel Guide

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

There are two methods to enable the function:

1) With auto mode is activated, turn to [POS] functional area, and click F6 

.
2) Select “Handwheel Guide” item under “Operate” menu.

3.8. Log

In [Log] window under [DIAG] functional area, important operation and system events can be recorded and displayed. As shown in following figure, you can browse the log info since this time start-up as well as history log info.

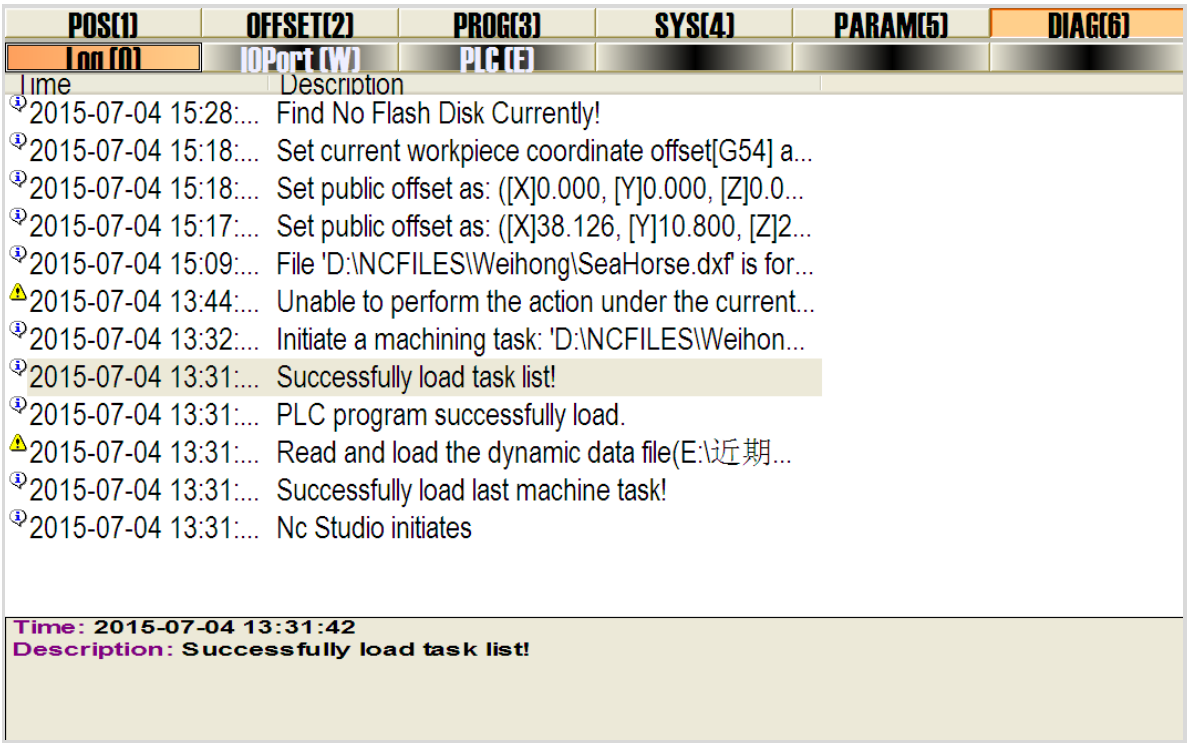


Fig. 3-22 Log list

As shown above, the log list contains info like:

1) Initialization and shutdown of the system;
2) Starting and end of auto machining;
3) Changes of workpiece coordinate;
4) System alarms;
5) Other information.

The manipulation button bar at the lower part of the window is illustrated below:

Fig. 3-23 Operation buttons below log list

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

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

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

CAUTION

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

, all logs instead of logs since this time start-up will be shown in the list.
3) Please clear logs regularly. When the record file becomes too large in size, it may lag down the system perfomance and responsive time.

3.9. Register

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

Click “About NcStudio” item under “Help” menu to open a dialog box named “About NcStudio”, as shown in Fig. 3-24, where you can obtain information such as software version, the developer, system information, as well remaining usage time. If the remaining time runs out, please send the “Card No.” to manufacturer to get a registration code. After you receive the registration code, click “Register”

button in Fig. 3-24, and a dialog box named “Register” will pop up. Input the registration code and confirm to register usage time.

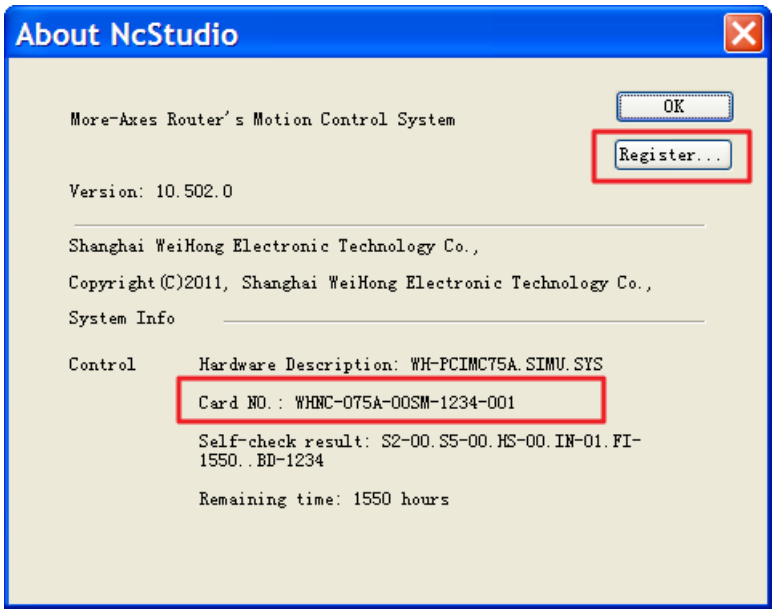


Fig. 3-24 About NcStudio dialog box

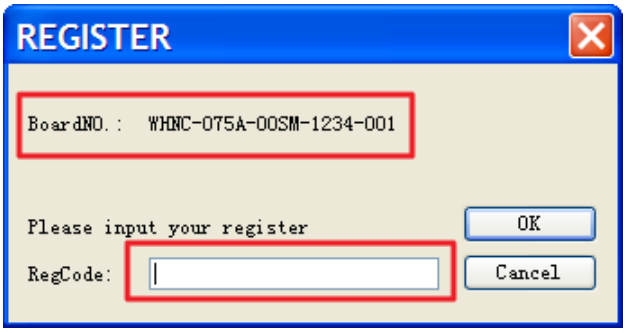


Fig. 3-25 Register dialog box

3.10. System Language and Theme

3.10.1. Language Switch-over

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

Turn to [Lang] window under [SYS] functional area, click “Select” button to select target language, as shown below:

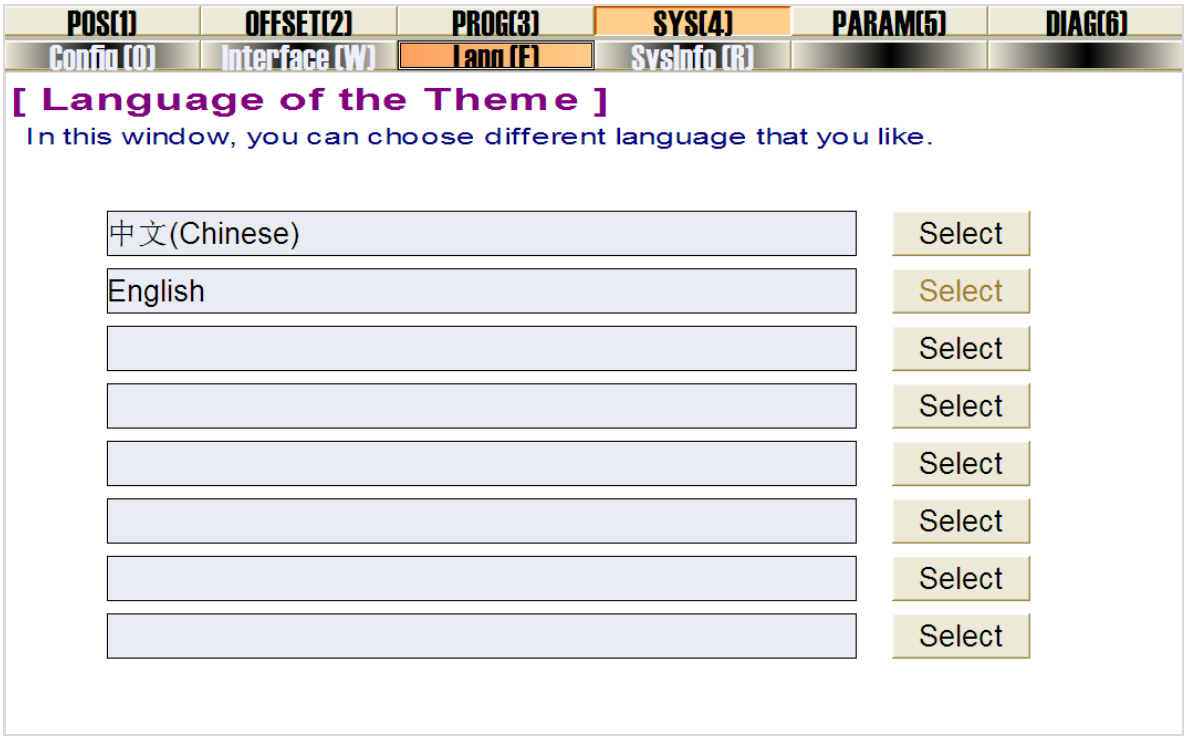


Fig. 3-26 Language selection

3.10.2. Theme Selection

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

Turn to [Interface] window under [SYS] functional area, and click “Select” button to select the desired theme, as shown in Fig. 3-27.

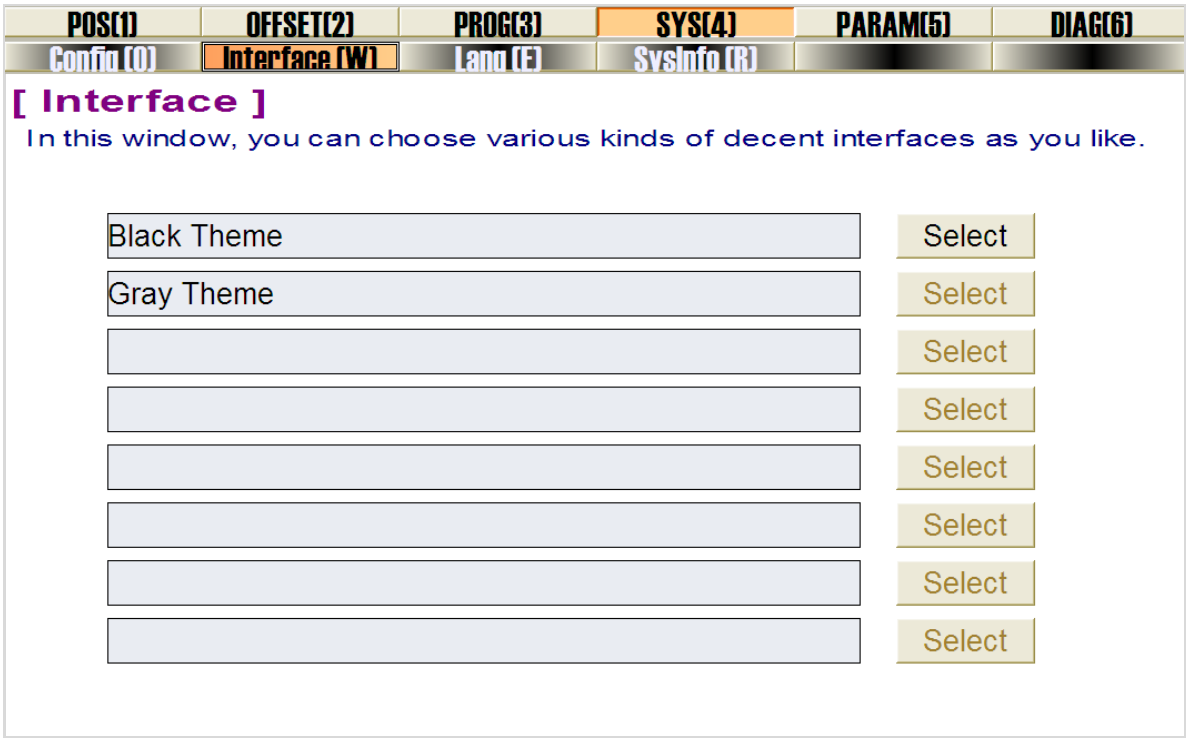


Fig. 3-27 Theme selection

3.11. System Maintenance

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

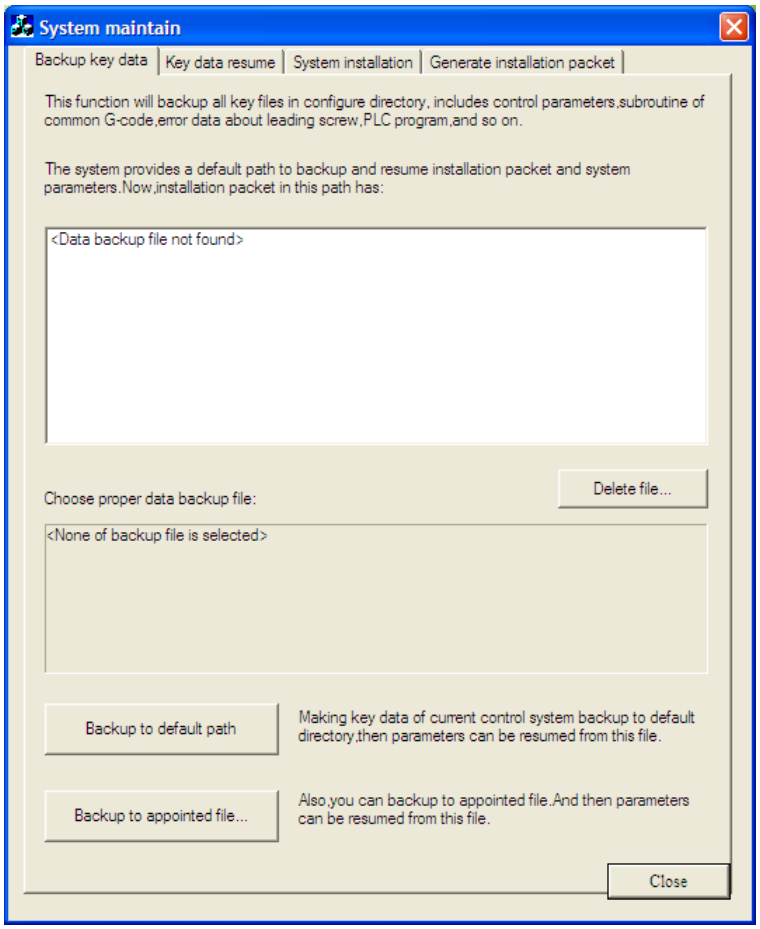


Fig. 3-28 System maintenance dialog box

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

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

When the tab “System installation” is active, the user can reinstall NcStudio system. But it should be paid special attention that after the re-installation, all the former files under the current installation directory will be deleted and the new version won’t keep any data of the old version. So it’s

recommended to do key data backup before the reinstallation. If there have been installation packages under the default directory, they will be displayed in the top white panel. Clicking one of the packages, the user can check its property in the indented panel below and execute [Delete] and [Start installation] function. The user can also use the direction keys to select other packages. Clicking the [Start installation] button which is at the bottom of the tab control will start installing.

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

You can enter into NcStudio again when “System Maintenance” dialog box is closed.

3.12. Auxiliary Function

3.12.1. Single Block

The user can enable single block function in machining task, which serves as a good support for error diagnosis and troubleshooting, where once axial resultant velocity is 0, program stops.

To enable the function, you can click sub-menu item “Single Block” under “Operate” menu. When the item is check, namely with mark  in front of the item, each time the “Start” button is clicked, the program goes on for one block and pause; click “Start” button once more, program goes on for next blcok, and the like. Whole program blocks will be run in this way.

3.12.2. Breakpoint Resume

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

To enable the function, you can click sub-menu item “Breakpoint Resume” under “Operate” menu, or turn to [POS] function area in auto mode, click F3 

button. Once the function is enabled, the system will resume machining from the position where last time machining stops.

3.12.3. Select Processing Block

With this function, the user can select any program block in a file to process. Note that the function is only effective in auto mode.

Select sub-menu item “Select Processing Block” under “Operate” menu, or turn to [POS] function area and click F5 

button to open a dialog box, as shown below. As shown in Fig. 3-29, according to tips, define the starting and end blocks by line No., and click “OK” to confirm it. And then click “Start” button to initiate the machining, or click sub-menu item “Simulation Mode ---> Simulation Start” to initiate simulative machining on the selected program blocks.

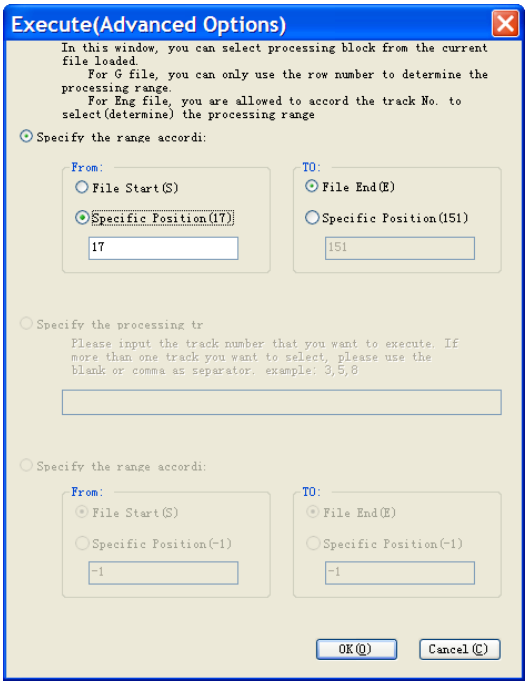


Fig. 3-29 Select processing block

CAUTION

1) If the currently loaded program file is G-code file, such as files formatted with *.nc/.g/.nce* file extensions, selective machining function by line No. can only be enabled, while selective machining according to track No. is unavailable.
2) If the currently loaded program file is *Eng* file, both types of selective machining are available.

3.12.4. Mirror and Rotate

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

Click sub-menu item “Circumrotate Mirror Processing” under “Operate” menu, and a dialog box as shown below will pop up. In the dialog box, you can choose an option and enable mirror and rotating machining.

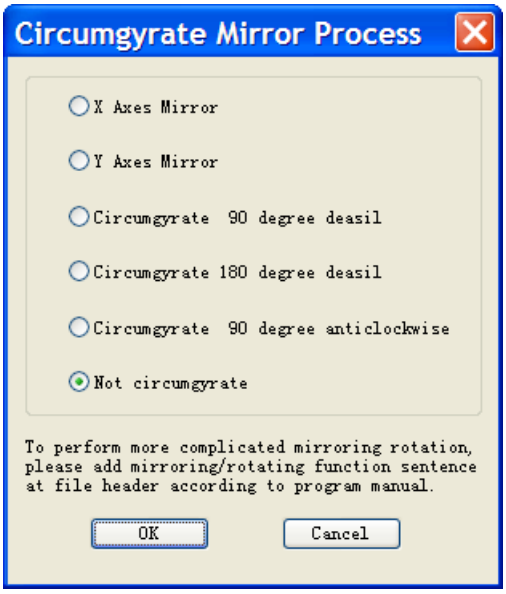


Fig. 3-30 Mirror and rotate

3.12.5. Parameter Backup and Auto Backup

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

Parameter backup

In [Backup] window under [PARAM], the user can back up the parameter settings and restore it. As shown in Fig. 3-31, after all parameters have been well set, click “Save” button to save the parameter setting. A dialog box where the user can input name for it will pop up. Up to 8 groups of parameter settings can be saved. When any one of the parameter settings group is needed to be restored, the user can click “Load” button to read from the settings and load it into the system.

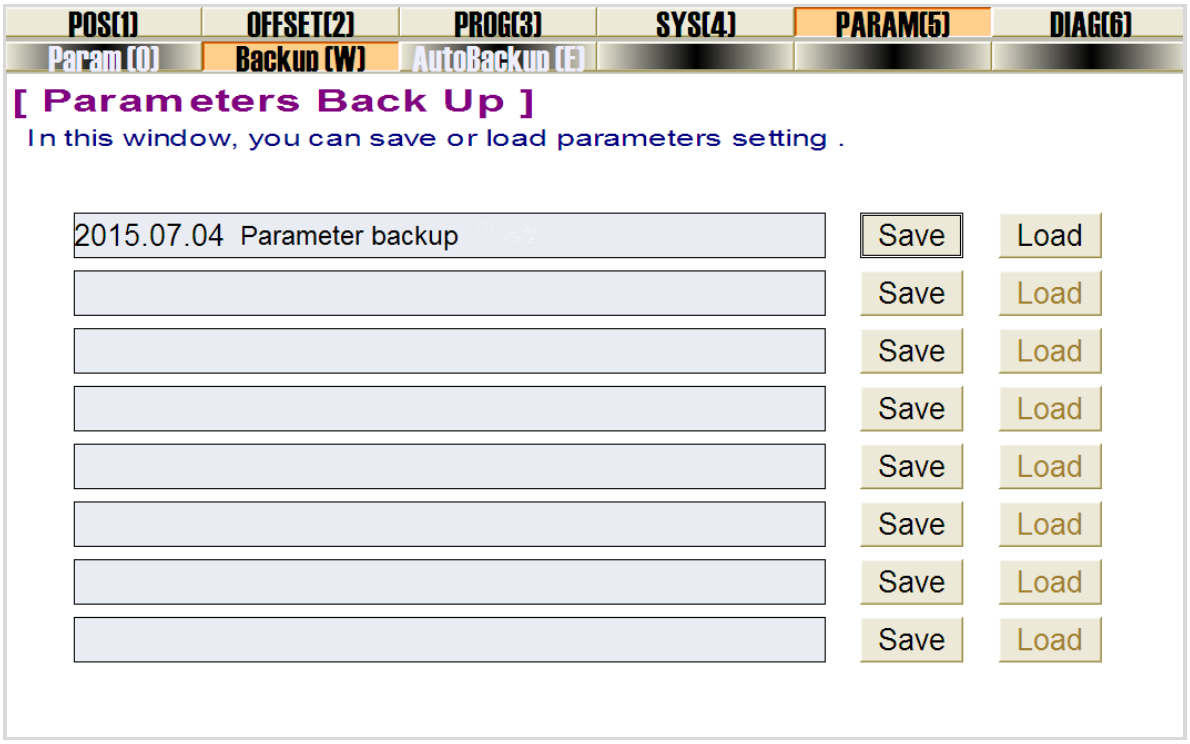


Fig. 3-31 Parameter backup

Automatic backup of parameter

In [AutoBackup] window under [PARAM], the user can restore parameter settings which have been saved automatically. Several options are offered, in which the user can select and load it into the system. As shown in Fig. 3-32.

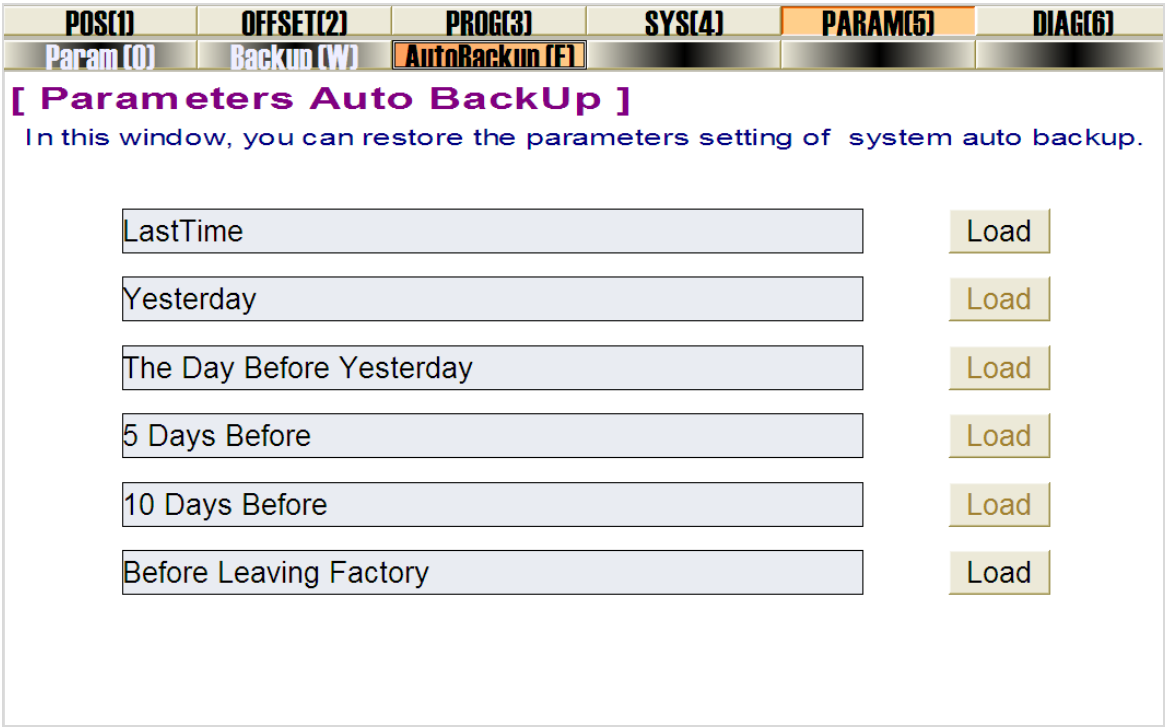


Fig. 3-32 Parameter auto backup

3.12.6. Graduation Machining

Aiming at rotary axis, this function can realize array machining of equivalent angle (known as the graduation) when processing circular arc with a certain degree of angle. Note that the tool path program can only contain program blocks for X, Y and Z axes. As illustrated in Fig. 3-33, in order to enable “Graduate processing” function, click button “Browse” to select a source program file (or called original program file), and then click button “Generate file” to generate target file. The system will load the target file automatically after it has been saved.

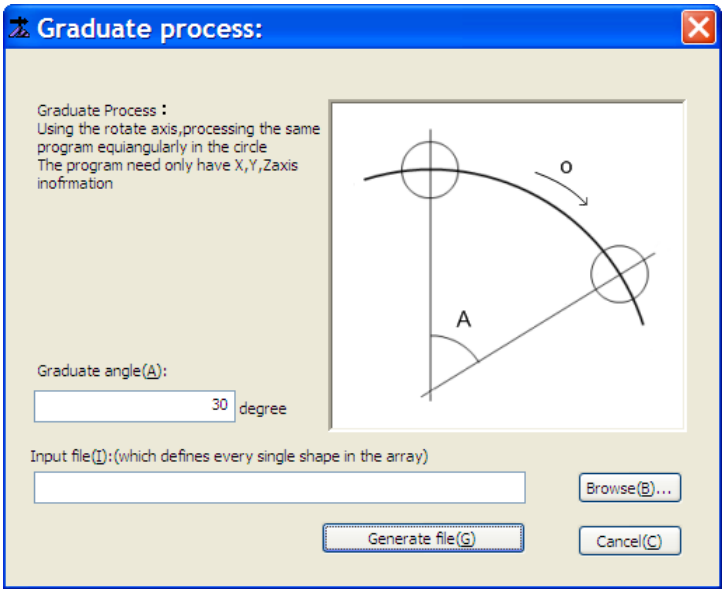


Fig. 3-33 Graduate processing dialog box

CAUTION

Note that this function is absent in five axes software.

3.12.7. PLC

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

You can turn to [PLC] window under [DIAG] functional area to run the PLC. Refer to *Programming Manual of NcStudio* for details.

[PLC] window is shown as below:

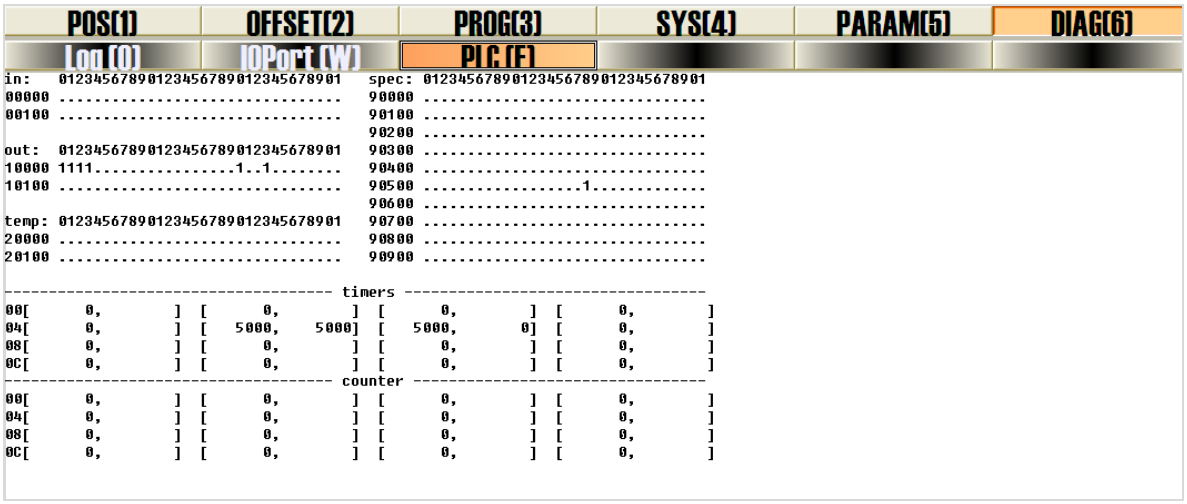


Fig. 3-34 PLC

Buttons on manipulation bar are shown as Fig. 3-35. When buttons are clicked, corresponding dialog boxes will pop out, where the user can input the channel No., timer No. and counter No. which need to be displayed. After it is confirmed, the ports information will be displayed in [PLC] window.

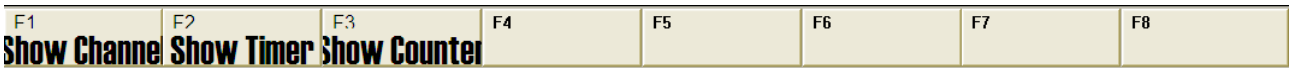


Fig. 3-35 Operation buttons below PLC window

4. Parameter Setting

NcStudio system boasts of abundant machining parameters to deal with various machining task of different requirements. Parameters can be divided into parameters of operator's access, of manufacturer's access and of developer's access according to limits of authority.

This chapter focuses on parameters of operator's access. Parameters of manufacturer's access and of developer's access need corresponding passwords.

4.1. Parameter Modification Method

Turn to [Param] window under [PARAM] functional area, double click on the parameter to open an input box, and modify the value.

CAUTION

All parameters should be modified in IDLE state.

4.2. Parameter List of Operator's Access

Table below lists all parameters of operator's access in ascending order of parameter No.

No.	Name	Setting Range	Default	Effective
1.0. Spindle general				
N10069	RevAxisProgUnit	0～60 (mm/min)	0	Immediately
	The measurement unit of the data in machining files under turn-table mode. 0: Angle (unit: degree); 1: Surface distance of the rotary workpiece (unit: mm).			
N10070	RotativeWorkpieceRadius	0～99999 (mm)	50	Immediately
	Under turn-table mode, the diameter of the workpiece to be machined.			
3.2. Tool setting				
N32020	NAME	-	-	Immediately
	Name of cutter. Maximal length is 127 letters.			
N32021	DIAMETER	-	0	Immediately
	Diameter of cutter.			
N32022	LENGTH	-	0	Immediately
	Length of cutter.			
N32023	DIAMETER WEAR	-	0	Immediately
	The amount that is lost by wear in cutter diameter.			
N32024	LENGTH WEAR	-	0	Immediately
	The amount that is lost by wear in cutter length.			

No.	Name	Setting Range	Default	Effective
N32030	OFFSET X	0: Not use 1: Use	0	Immediately
N32031	OFFSET Y	0～100000（mm/min）	0	Immediately
N32032	OFFSET Z1	0～100000（mm/min）	0	Immediately
N32033	OFFSET Z2	0～9999（mm）	0	Immediately
Note: up to 8 tools can be supported in the software, only parameters related to T1 have been listed above, and parameters related to other tools are the same with that of T1.				
4.1. Lube				
N41003	MachTaskEndInformType	0; 1; 2	0	Immediately
	How to inform operators after the end of machining task. 0: Red lamp is out; 1: Red lamp lights for 3 seconds; 2: Red lamp lights until any input from mouse or keyboard.			
6.2. G code options				
N62020	IJK_INC	0～500（mm）	Yes	Immediately
	The value of the IJK addresses corresponding to the circle center in a circle programming.			
N62022	Tool Selection for G Code	-99999～99999（mm）	No	After reloading
	When processing G code (or files can be translated to G code except Eng and NCE), tool can be selected if there are many tools.			
N62730	G73_G83_SAFE_HEIGHT	-99999～99999（mm）	0	Immediately
	Specifying the retract distance after each feed in the G73_G83 cycle drilling command.			
N62760	DRILL_STOP_DIR	-	0	Immediately
	0/1: (G17:+X/-X) 2/3: (G17:+Y/-Y)			
6.4. Speed/Acc				
N64020	RAPID_TRAVEL_FEEDRATE	0~100000 mm/min	3000	Immediately
	The default speed of beeline axis when positioning.			
N64021	DEFAULT_FEEDRATE	0~100000mm/min	1500	Immediately
	The default speed of beeline axis when machining.			
N64030	REV_RAPID_TRAVEL_FEEDRATE	0～100000rpm	10	Immediately
	The default speed of rotary axis when positioning.			
6.5. File translation				
PLT_FORMAT_CONVERT_PARAM				
N65000	TOOL_UP_HEIGHT	0~99999 (mm)	1	Reload program
	Upward height of the tool while dry run.			
N65001	PLT_UNIT	0.001~99999	40	Reload program
	Length in millimeter (or inch) per PLT unit.			
N65002	TOOL_DISTANCE_AT_AREA	0.0001~99999 (mm)	0.025	Reload program
	Distance between tools while machining in PLT-area.			

No.	Name	Setting Range	Default	Effective
N65003	CUTTING_DEPTH	-99999~0 (mm)	-1	Reload program
	Machining depth of 2D files.			
DXF_FORMAT_CONVERT_PARAM				
N65100	TOOL_UP_HEIGHT	0~99999 (mm)	1	Reload program
	Upward height of the tool while dry run.			
N65101	CUTTING DEPTH	-99999~0 (mm)	-1	Reload program
	Machining depth of 2D files.			
N65102	DEPTH_LAYER	-99999~0 (mm)	0	Reload program
	Machining depth of each tool layer in 2D files.			
N65103	FIRST_POINT_AS_ORIGIN	-	-	Reload program
N65104	ENABLE_MACHINE_INDIVIDUALLY	0: No 1: Yes	No	Reload program
	Machining only one shape each time and go to next shape only after last one finished.			
N65105	ENABLE_MACHINE_BOTTOM	0: No 1: Yes	No	Reload program
	[3D Cutting] Valve operation is performed only when cutter arrives at workpiece surface.			
ENG_FORMAT_CONVERT_PARAM				
N65200	TOOL_UP_HEIGHT	0~99999 (mm)	1	Reload program
	Upward height of the tool while dry run.			
N65201	ENABLE_CHANGE_TOOL_PROMPT	0: No 1: Yes	Yes	Reload program
	When it meets tool exchange in Eng file, pause and remind to change tool.			
N65202	RecycleMachiningTimes	1~9999	1	Reload program
	Machining times that needs to be cycled while machining Eng file.			
N65203	MachiningEngFileByToolNumber	0: No 1: Yes	No	Reload program
	With this function, Eng file machining can be executed by tool selection according to the number specified.			
N65207	Modify By Tool Number	0: No 1: Yes	No	Reload program
	With this function, Eng file machining can be executed by specified tool number.			
N65208	Z Up Type After Drill	0;1	0	Immediately
	Z Up Type After Drill, 0: Up to R plane; 1: Up to specified workpiece coordinate position, exclusively allowed in ENG file.			
N65209	Z Pos After Drill	-1000~1000 (mm)	10	Immediately
	Z Up Type After Drill is one, and then lifts up to this workpiece coordinate position.			
6.6. Change tool				
N66000	PROMPT_CT_CMD	0: No 1: Yes	No	Immediately
	Whether to suspend the system and to prompt users while meeting the change tool instructions, only using in union mode.			

No.	Name	Setting Range	Default	Effective
N66011	CHANGETOOLAHEADPOS	-99999~100000 (mm)	-	Immediately
	Low speed position before get into the tool slot.			
N66019	Change Tool Speed	(mm/min)	3000	Immediately
	Spindle speed when changing tool.			
N66020	Change Tool Speed	(mm/min)	1800	Immediately
	The default speed when Z axis moves from the upper position to lower position in changing a tool.			
N66042	Pause And Prompt While Change Tools	0: No 1: Yes	No	Immediately
	Pause and prompt while change tools.			
N66067 ~ N66267	ToolYDirSpaceBetween (X\Y\Z)	-	-	Immediately
	There are altogether 21 groups.			
7.1. Manu				
N71000	JOG_VOL	0.001～100000	2400	Immediately
	The default velocity under jog mode.			
N71001	RAPID_JOG_VOL	0.001～100000	2400	Immediately
	The velocity under rapid jog mode.			
N71002	MaxJogFeedrateBeforeBKREF	0～100000（mm/min）	1200	Immediately
	The maximum speed under jog mode before returning to the reference point.			
N71003	REV_RAPID_HOG_VOL	0～100000（rpm）	10	Immediately
	The velocity of rotary axis under rapid jog mode.			
7.2. Auto				
N72001	SPEED_ASSIGN_TYPE	0: false 1: true	0	Immediately
	Specify what feeding speed will be selected. 0: Use file speed; 1: Use default speed; 2: Specify speed proportionally.			
N72002	IGNORE_PROG_SPINDLE_REV	-99999～99999（mm）	No	Immediately
	If true, the system will ignore the rotational speed specified by the file and use the default speed specified by SPIND_VELO_DEFAULT.			
N72004	STOP_SPIND_AT_END	0～99999（mm/min）	Yes	Immediately
	Whether to stop the spindle after a task. Note: if parameter “AutoStopSpindleWhenPause” is set to “Yes”, the spindle will be stopped after a task stops regardless of the parameter value.			
N72005	MACH_DEPTH	-99999~0 (mm)	-1	Immediately
	Machining depth of 2D file.			
N72006	HEIGHT_Z_RAISE	0.001～99999 (mm)	1	Immediately
	The lift height of Z axis during dry run.			

No.	Name	Setting Range	Default	Effective
7.3. Pause				
N73000	PAUSE_DOWN_VOL	0~100000 (mm/min)	600	Immediately
	The downward speed of Z axis at the previous cutting point after a pause.			
N73001	PAUSE_UP_VOL	0~100000 (mm/min)	600	Immediately
	The lift speed of Z axis while entering a pause.			
N73002	PAUSE_OPTION	0;1	0	Immediately
	The lifting options of Z axis action when pause is specified. Values: 0: lift up a specified value; 1: go to a specified position in WCS.			
N73003	Z_WPCOOR_ON_PAUSE	0~9999 (mm)	10	Immediately
	The value is used to specify the workpiece coordinate of the stoppage position of Z axis when a pause is specified. The value is valid only when PAUSE_OPTION is 1.			
N73004	Z_OFFSET_ON_PAUSE	0~500 (mm)	10	Immediately
	When a pause is specified, the value is used to specify the lift height of Z axis corresponding to the previous height before the pause. The value is valid only when PAUSE_OPTION is 0.			
N73005	STOP_SPIND_AT_PAUSE	0: No 1: Yes	Yes	Immediately
	Whether the spindle can automatically stop after a pause.			
N73006	Z_MACHCOOR_ON_PAUSE	-99999~99999 (mm)	0	Immediately
	The value is used to specify the machine coordinate of the stoppage position of Z axis when a pause is specified. The value is valid only when PAUSE_OPTION is 2.			
7.4. Bkref				
N74001	NEED_REFPT_BEFORE_MACHINING	0: No 1: Yes	No	Immediately
	Whether to back to the machine origin before machining.			
N74101	AutoBackToZero	0: No 1: Yes	No	Reboot
	Whether to back to zero automatically.			
7.5. Measure				
N75210	FIXED_CALI_POS (X\Y\Z)	-99999~99999 (mm)	0\0\-1	Immediately
	The machine coordinates of the fixed tool calibrator.			
7.9. Operation others				
N79000	Z_DOWN_VELO_OPTION	0;1;2	0	Immediately
	The options of Z down speed, including 0: not disposed; 1: direct-Z-motion only; 2: general Z-down motion.			
N79001	Z_DOWN_VELO	0~100000(mm/min)	480	Immediately
	Z down speed for machining.			
N79003	SAFE_HEIGHT	0~1000 (mm)	10	Immediately
	The safe height to avoid collisions in WCS, which is used after returning to the machine origin or breakpoint resume.			
N79100	IS_FIXED_VALID	0: No	Yes	Immediately

No.	Name	Setting Range	Default	Effective
		1: Yes		
	Whether to back to the fixed point after a program.			
N79110	FIXPT_POS X/Y/Z/A	-99999~99999mm	0	Immediately
	The position of the fixed point.			
N79200	DisplaySimuOutRange	0: No 1: Yes	Yes	Immediately
	Whether to prompt the user if simulation is out of travel range.			
8.1. Position view				
N81000	AUTO_LOAD_TRACK	0: No 1: Yes	Yes	Immediately
	Whether to parse the file automatically after load. .			
N81001	AUTO_LOAD_TRACK_LIMIT	0~100000 KB	1000	Immediately
	The file size limit. A machining file can be automatically loaded only when the file size is less than this value.			
8.3. Door Open Parameters				
N83014	Back to origin mode	0;1	1	Immediately
	Back to origin mode. Available value: 0: Three axes back to origin; 1: All axes back to origin.			
N83015	Z Axis Position when back to origin	0,1	0	Immediately
	Z axis position when return to the reference point. Available options: 0: Back to safe height; 1:Back to position #AREAMAX.Z-1.			

5. Precautions and Troubleshooting

5.1. Common Troubleshooting

5.1.1. What should users do if the spindle does not rotate?

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

5.1.2. What should users do if an axis does not move?

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

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

1) Check if there is signal for Z axis brake input. If not, check whether the servo driver is enabled or not, or parameter related to brake output are correctly set or not.
2) If the signal exists, remove wiring between brake output terminals (Y00-C00), and then start the system and power up machine (ruling out system alarm signal) and test the conduction

between the terminals with a multimeter. If not, check the host system; if yes, brake output normally works.

3) Power off the machine tool, re-connect the two wires which have been removed above (connect 24V power supply). Check if the voltage across brake wire of the motor with a multimeter is 24V or not. If yes, the motor is damaged.

4) If the problem remains unsolved, please change another Lambda controller.

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

◆ **Limit alarm or servo driver alarm occurs during returning to the machine origin.**

1) Observe the port "× machine origin" in [IOPort] window under [DIAG] functional area, and secure that the polarity is consistent with the home switch signal type (N for normally open / P for normally closed).

2) Check if the software can receive the REF. point signal of this axis. The method is: trigger the home switch, and then see if the color of the dot before the "× machine origin" changes from red to green in [IOPort] window under [DIAG] functional area. If there is no color change, it indicates the software can't receive the REF. point signal, needing to check if there is an error in the home switch or in the wiring of home switch. To see if the system failure occurs, make the REF. point signal on the controller and COM port into conduction directly with a conducting wire, and then check whether the color of the dot before "× machine origin" changes in [IOPort] window.

3) Check whether settings of parameter “N74020 COARSE_LOCATING_DIR” and “N74080 BACK_DISTANCE” are correct or not, whose settings should be opposite to each other.

4) Check whether the position of home switch is appropriate to avoid the following three situations: the distance between home switch and limit switch is too small; the home switch is installed behind the limit switch; or the position of home switch is out of the mechanical stroke of machine tool.

◆ **When backing to the machine origin, the machine tool motions towards a certain direction at a relatively low speed (ten percent of the speed of coarse positioning) until limit is triggered.**

See if the polarity of “× machine origin” input port is correct in [IOPort] window under [DIAG] functional area. When this home switch is triggered, i.e. there is signal input, the color of the dot should be green. Otherwise, it is red.

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

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

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

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

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

5.2. Alarm Information

Type	Warning Content	Cause	Solution
⚠ Warning message	Simulation results show that program range exceeds the machine travel limit.	Tool path of the program file exceeds the upper/lower limit of workbench travel, which are decided by settings of parameter N10020 and N10030 separately.	Check if the WCS zero is reasonable. Check the tool path program file. Modify parameter settings of N10020 and N10030 to enlarge the workbench travel limit. (see chapter 3.1)
	The system has not returned to the	The system has not returned to machine	Use this function after returning to the machine

Type	Warning Content	Cause	Solution
	machine origin, failed to execute the operation!	origin. Whether the system has to return to the machine origin is decided by parameter N74001.	origin.
	The system is busy, this operation can't be executed.	Some illegal operations are performed under machining state.	Stop machining, and execute some operations under idle state.
Limit alarm	Positive (negative) limit of X (Y\Z) axis	The polarity of X axis positive limit port is wrong.	Enter [IOPort] window under [DIAG] functional area, and modify the port polarity. (see chapter 2.2.6)
		X axis runs into limit switch directly during motion.	Manually move X axis away from limit switch.
		There is an error in limit switch itself.	Check if limit switch works normally.
Servo alarm	Servo alarm of X (Y\Z) axis	The polarity of X axis servo alarm port is wrong.	Enter [IOPort] window under [DIAG] functional area, and modify the port polarity. (see chapter 2.2.6)
		There is an error in X axis servo driver itself.	Check if X axis servo driver works normally.
E-stop alarm	E-stop button is pressed.	The polarity of E-stop port is wrong.	Enter [IOPort] window under [DIAG] functional area, and modify the port polarity. (see chapter 2.2.6)
		The E-stop button is pressed.	Turn the E-stop button clockwise to make it bounced.
Spindle alarm	Spindle alarm	The polarity of spindle alarm port is wrong.	Enter [IOPort] window under [DIAG] functional area, and modify the port polarity (see chapter 2.2.6)

Type	Warning Content	Cause	Solution
		There is an error in inverter.	Find the reason based on the alarm type of inverter.
File error alarm	No NC file. Please load a file into the code interpreter first.	Start file machining with no file loaded in the system.	Load a machining file before start machining.
Terminal board not connected error alarm	Terminal board not connected.	Cable has not been firmly connected or there is error in Lambda controller.	Re-plug the cable and restart the software to observe the occurrence of the error; Polarity of port is wrong. Invert the port polarity and restart the software; Analyze possible causes according to the status of indicator “SYSTEM” on Lambda controller; Change another Lambda controller.

CAUTION

Some alarms are customized by users according to special mechanical structures, while may not listed in above table. Please contact with manufacturer in case of any question.

6. Appendix

6.1. Basic Concepts of NcStudio

6.1.1. Operation Mode

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

AUTO Mode

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

Manual Mode

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

1) In jog mode, the user can directly control the motion of the machine tool via manual operation equipment, such as computer keyboard, handheld box, and MPG. When the user sends out the motion signal with the help of one of the equipment, for example, clicking the button 

and 

in synchrony on the software interface will make the machine tool move consecutively until the buttons are released.
2) In stepping mode, the user also uses manual operation equipment, such as computer keyboard, handheld box and MPG to control the machine tool. However, different from the jog mode, when the user clicks a button once (from clicking the button to releasing it), the machine tool only moves a specific distance (known as the step-size as well). In this way, the user can control the displacement of the machine tool precisely.
3) Select “Handwheel” in the software interface to activate the handwheel mode. With enabled, the connected handwheel or the MPG can be used to control the motion of the machine tool.

Reference Point Mode (Homing)

Returning to the machine origin, also called homing, is the process to synchronize the internal coordinate and machine actual coordinate in CNC system. For this reason, homing is a required step during startup. And reference point mode is the default operational mode after system startup, in convenience of related operation execution.

CAUTION

The user can freely define the step length for stepping mode by clicking button >> and input the desired value in the pop-up box.

6.1.2. Operation State

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

IDLE

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

E-STOP

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

RUNNING

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

PAUSE

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

LOCK

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

6.1.3. Coordinate System

Coordinate system is a terminology describing the motion of a machine tool. For the sake of unification, standard coordinate system adopts right-hand rule, as illustrated in Fig. 6-1:

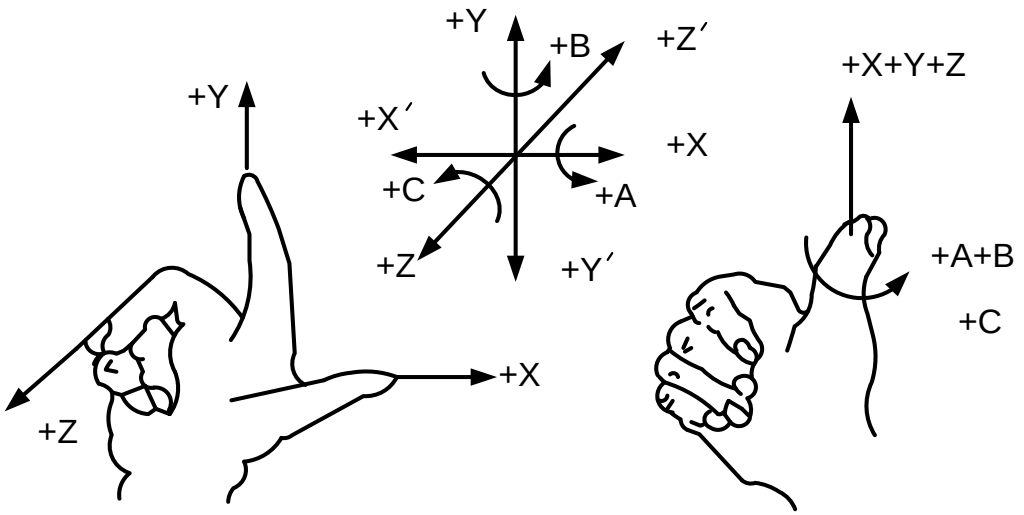


Fig. 6-1 Coordinate system following right-hand rule

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

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

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

——A-axis rotates with X-axis as the centerline, B-axis rotates with Y-axis as the centerline and C-axis with Z-axis as the centerline. Counter-clockwise direction is referred to as the positive direction for A/B/C axes.

Machine Coordinate System (MCS)

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

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

Workpiece Coordinate System (WCS)

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

6.2. Shortcut Keys List

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

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

6.3. Software License Agreement

Important—Read Carefully before Using This Product:

The term “Software Product” includes all copies of the licensed software and its documentation. This license agreement is a legal agreement between You (either an individual, a legal entity or any affiliated companies or other entities) and Weihong Electronic Technology Co., Ltd. (hereinafter referred to as Weihong Company). By installing, copying, or otherwise using the Software Product, you agree to be bound by the terms and conditions of this license. Unless otherwise stated in this agreement, you shall not use, copy, revise, rent, or transfer the Software product or any part of the Software Product for any other purposes.

Description of Further Rights and Restrictions:

1. You may install for use one copy of the Software Product on a single machine;
2. You may make a copy of the Software Product for archive or backup purposes and the copy is to be used on this machine only;
3. You may transfer the Software Product and the license agreement to a third party, provided that the third party accepts the terms and conditions stated in this agreement, with prior express permission from Weihong Company;
4. When transfer confirmed, you shall transfer all the copies of the original documents and the supplementary documents to the third party or destroy all the copies un-transferred.
5. You can use the Software Product on a network server or intranet server only if it is stipulated in explicit terms that you are allowed to use the Software Product on a network server or intranet server, or you have purchased license for each node and terminal using the Software Product;
6. You may NOT sublicense, assign or transfer the license agreement;
7. You may NOT or direct any third party to reverse engineer, decompile or disassemble the Software Product;
8. You may NOT copy or transfer the Software Product or any part of the Software Product unless otherwise expressly stated in this agreement;
9. The license agreement shall be terminated automatically upon you transfer the Software Product or copies of all or part of the Software Product to a third party.

Intellectual Property Rights Notice

The Software Product and all intellectual property rights therein (including but not limited to any all copyrights, patents, trademarks, and publicity rights) are owned by Weihong Company. The Software Product is protected for Weihong Company on the basis of copyright law and international treaty

provisions as well as on the basis of other laws and agreements regarding intellectual property. You are not allowed to remove the copyright statement made in the Software Product, and guarantee that you shall copy the copyright statement in all copies of the Software Product or of any part of the Software Product. You are obliged to stop any form of illegal copying of the Software Product and accompanying materials.

After-sales Guarantee:

Weihong Company guarantees that for 90 days from the date of shipment the software carrier will be free from defects in materials and workmanship. When such a defect has been confirmed, our only responsibility is to replace the software carrier. This remedy is your exclusive remedy. This after-sales guarantee is invalid for any carrier defect caused by accidents, abuses or mal-operation. The replaced software carrier enjoys the remaining guarantee time of the original software carrier or of a 30-day guarantee time, whichever is longer.

Except as the after-sales guarantee made above, the Software Product does not enjoy any other form of after-sale guarantee.

Limitation of Liability:

The above guarantee, whether made explicitly or by implication, constitutes the entire contents of the guarantee, including guarantee for the commerciality and applicability of special application aim. Whether you follow other terms in this agreement or not, Weihong Company, as well as its agents and sales staff, will not be responsible for any profits loss, availability loss, business break-off or any forms of indirect, special, accidental or inevitable damage or claim made by any third party, generated from the using of the Software Product, even if Weihong Company has been informed of the possible occurrence of such events in advance.

Termination:

This license may be terminated by Weihong Company at any time once you violate any terms or conditions made in this agreement. Once the license is terminated, you are obliged to destroy all the copies of the Software Product or return them to Weihong Company.

Applicable Law:

Copyright Law, Regulations for the Protection of Computer Software, Patent Law and other relevant laws and regulations.

Now, we affirm that you have already read through this agreement and understood it thoroughly and agreed to comply with all of the terms and conditions of this agreement strictly.

Weihong Electronic Technology Co., Ltd.