NK300BX Integrated CNC System

Manufacturers' Manual

4th Edition

Weihong Electronic Technology Co., Ltd.

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

Preface

About This manual

This manual is intended for manufacturers. If you use Weihong CNC system for the first time, it is suggested to read through this manual. If not, however, you can search for the desired information via the contents.

With 7chapters, this manual can be divided into 5 parts, as follows:

1) Preface part, introducing the precautions about transportation and storage, installation, wiring, debugging, usage and so on. You need to read them first carefully to ensure safe operations.
2) Hardware part, including chapter 1, 2 and 5. Chapter 1 and 2 gives general description of NK300BX CNC system from the perspective of system configuration, hardware information as well as wiring diagrams of ports. Chapter 5 introduces the parameter setting and wiring diagrams of drives of various brands.
3) Software operation part. Chapter 3 introduces software interfaces and operation corresponding to each function and lists related parameters and setting notes. After reading through this part, you can learn the operation menus and commands.
4) Maintenance part, including chapter 4, which contains maintenance information. You can take corresponding actions in case of breakdown according to this chapter.
5) Appendix part, including chapter 6 and chapter 7. Chapter 6 lists all the parameters while chapter 7 contains the software license agreement.

Applicable Product Model

This manual is applicable to NK300BX Integrated CNC System. Refer to the table below for details.

Product Model	Remarks
NK300BX Integrated CNC System	Abbreviated as NK300BX.

Contact Us

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

Company Name: Weihong Electronic Technology Co., Ltd.

Headquarters Address: No.1590, Huhang Rd., Fengxian, Shanghai, PRC 201400

Tel: +86-21-33587550

Fax: +86-21-33587519

Website: http://en.weihong.com.cn

Revision History

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

Date	Edition	Revision
2016.02	R4	1. Contact information updated; 2. Screenshots of the software updated; 3. Document style updated. 4. The wiring diagram in section 5.2.1 and 5.2.6 updated; 5. Chapter 1, section 2.2, section 2.3, chapter 3, section 4.1, section 4.2, and chapter 6 updated.

Precautions

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

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

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

WARNING

1) Precautions Related to Storage and Transportation

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

2) Precautions Related to Installation

- Only when this equipment installed in the qualified electricity cabinet can it be used. The construction of the cabinet must reach IP54 grade of protection;
- Paste sealing strips on the joint of the cabinet to seal all the cracks;

WARNING

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

3) Precautions Related to Wiring

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

4) Precautions Related to Running & Debugging

WARNING

- 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.

Table of Contents

1 Overview

1.1 System Configuration

◆ **NK300BX-H integrated CNC system consists of the following components:**

1) One NK300BX host
2) One WH106A3 operation panel and one KB1A1 keyboard panel
3) One Lambda 4S(BX) controller
4) Two DB9M/F cables (40cm)
5) One DB9M/F cable with optional length
6) Extended terminal board EX27A with connecting line HS20-HS20 (optional)

◆ **NK300BX-V integrated CNC system consists of the following components:**

1) One NK300BX host
2) One WH108A1 operation panel
3) One Lambda 4S(BX) controller
4) One DB9M/F cable (40cm)
5) One DB9M/F cable with optional length
6) Extended terminal board EX27A with connecting line HS20-HS20 (optional)

1.2 An Introduction to NK300BX

NK300BX is divided into NK300BX-H and NK300BX-V according to the layout of the monitor and the keyboard. The machine picture and dimensional drawing of these two types are listed below respectively.

1.2.1 Structure Specification

◆ **Product Pictures of NK300BX**

Fig. 1-1 A product picture of NK300BX-H

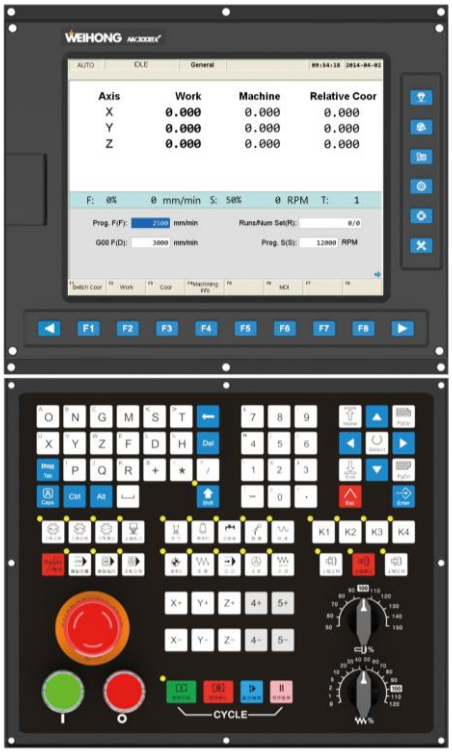
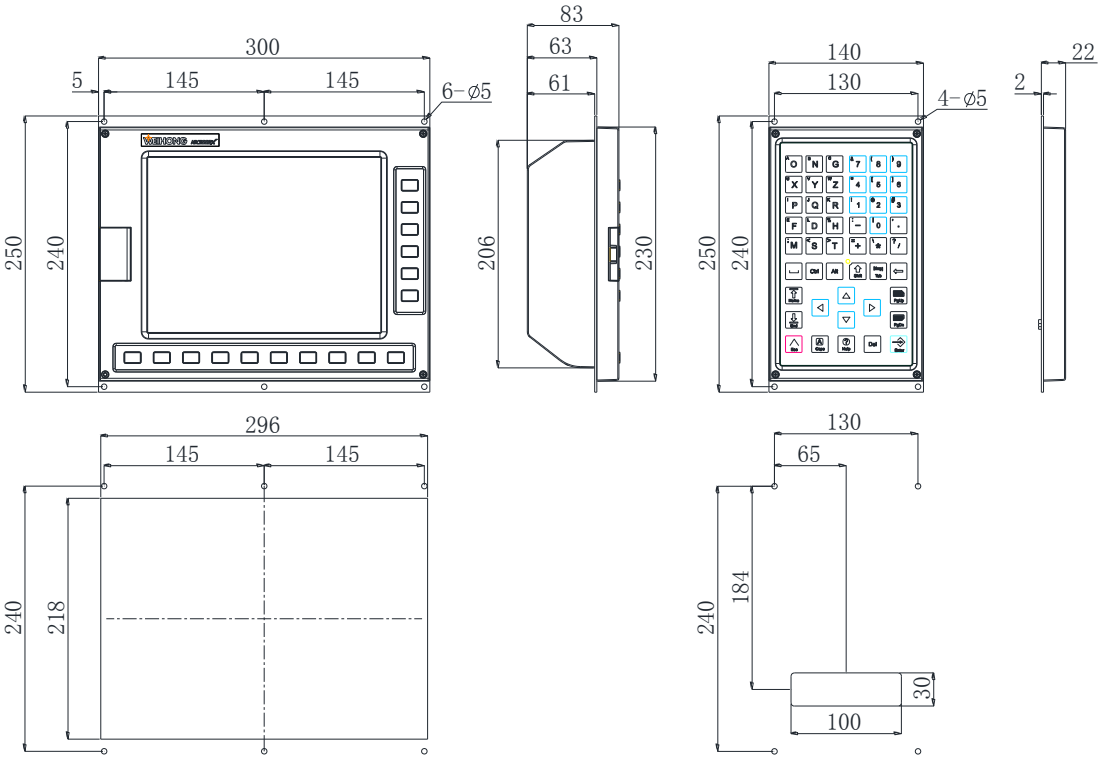


Fig. 1-2 A product picture of NK300BX-V

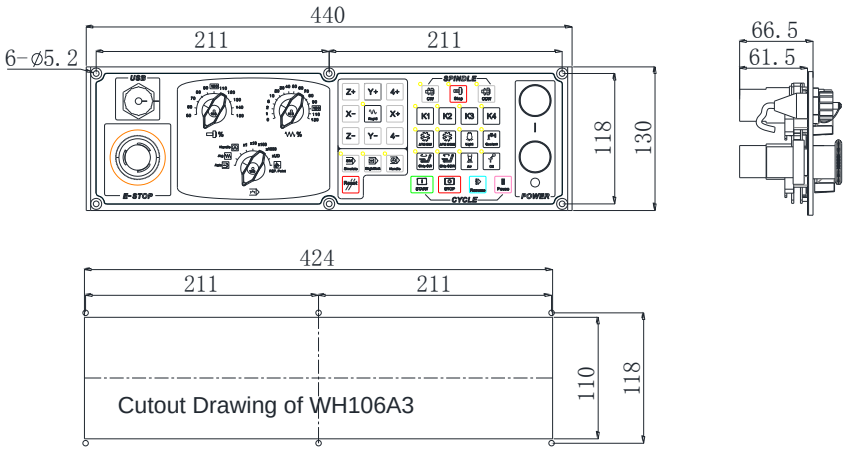
◆ **Mounting Dimension**

After NK300BX is installed on the machine, 100 mm space should be preserved in its surrounding for wiring convenience and ventilation. And the dimensional drawing of NK300BX-H is as shown in Fig. 1-3:



Dimension and Cutout Drawing of Host

Dimension and Cutout Drawing of Keyboard



Dimension and Cutout Drawing of Operation Panel

Fig. 1-3 Dimensional drawing of NK300BX-H

The dimensional drawing of NK300BX-V is as shown in Fig. 1-4.

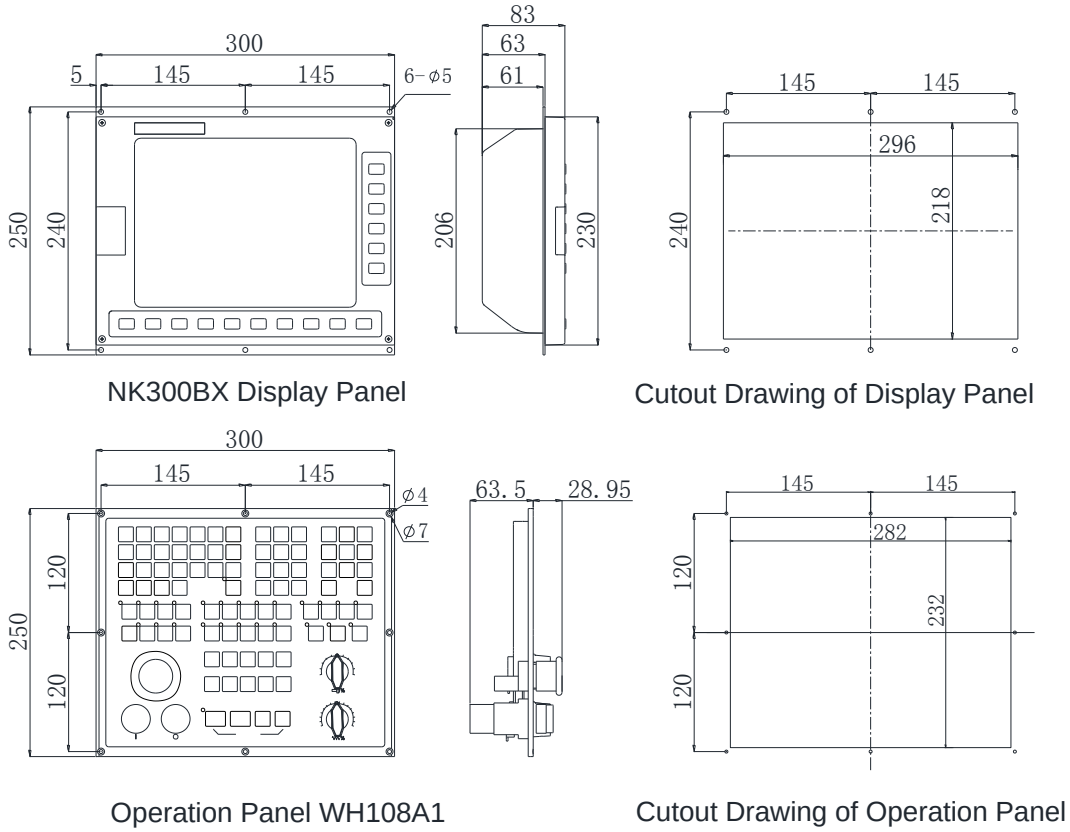


Fig. 1-4 Dimensional drawing of NK300BX-V

◆ **Host NK300BX**

The front view of host NK300BX is shown in Fig. 1-5.

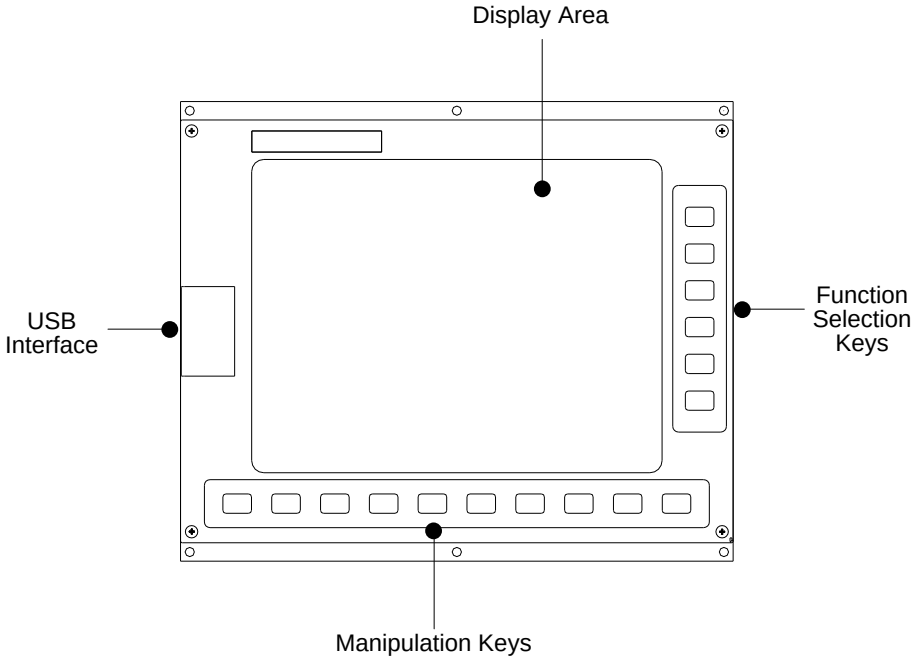


Fig. 1-5 Front view of Host NK300BX

- User interface. It is user-friendly and easy to operate.
- USB interface. It is used for USB removable flash disk connection, protected by a cover.
- Function selection keys. There are altogether 6 functional section, including machining, advanced, program, system, parameter and diagnosis. You can access any functional sections by pressing the corresponding keys directly.
- Manipulation keys. Including F1~F8, which are used to activate the functions indicated by the soft keys.

The rear view of Host HK300BX is shown in Fig. 1-6.

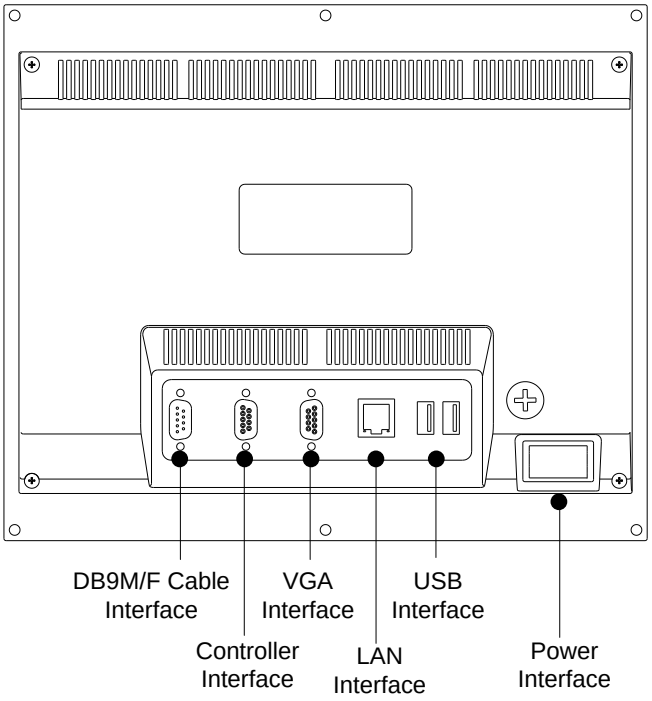


Fig. 1-6 Rear view of Host NK300BX

- DB9M/F cable interface. For NK300BX-H, the interface is used to connect with keypad panel; while for NK300BX-V, it is used to connect with operation panel.
- Controller interface. It is used to connect with Lambda 4S controller.
- VGA interface. It is used to connect with the monitor or display.
- LAN interface. It is used to connect with network, with transmission rate of 100Mbps.
- USB interface. Two USB interfaces, which are used to connect with removable flash disk.
- Power interface. It is used to connect with 220V power supply.

◆ **Operation Panel**

The operation panel for host NK300BX-H is WH106A3 and keypad panel KB1A1, while that for host 300BX-V is WH108A1.

1) Operation Panel WH106A3

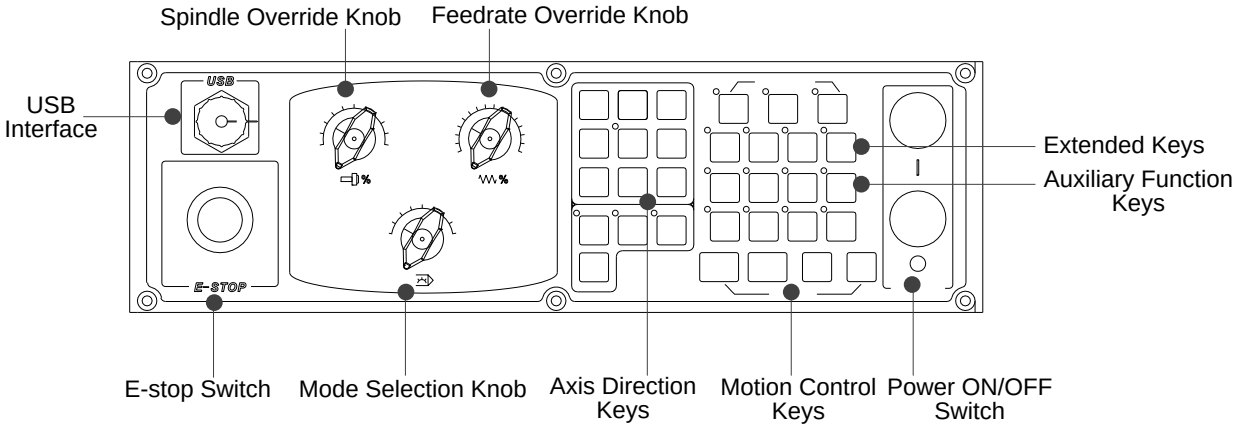


Fig. 1-7 Front view of operation panel WH106A3

- USB interface (with a protection cap), used for connection with removable flash disk.
- Emergency stop switch. Anytime there is possible danger, operator can press E-stop switch to stop the machine to protect safety of both human and machine, and when danger is cleared, turn the switch in clockwise direction to remove the alarm.
- Spindle override knob, which is used for spindle speed override adjustment. Refer to section 3.7.1 for details.
- Mode selection knob, which is used for mode selection, as shown in Fig. 1-8.

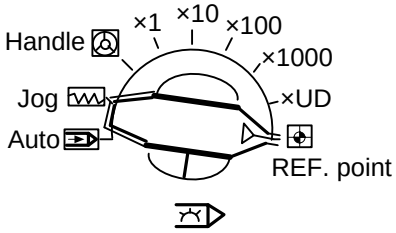


Fig. 1-8 Mode selection knob

- Feedrate override knob, which is used for feedrate override adjustment. Refer to section 3.11.1 for details.
- Axis direction keys, used for manual control of each axis movement in jog mode or jiggle mode. See Fig. 1-9 for concrete keys. How to use [Rapid] key? In manual mode, when any axis direction key and [Rapid] key are together pressed, the axis moves at manual high speed, or called rapid jog speed; when any axis direction key is pressed alone, the axis moves at manual speed, or called jog speed.

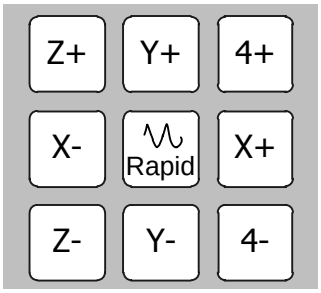


Fig. 1-9 Axis direction keys

- Motion control keys. In auto mode, you can press [Cycle Start], [Pause] and [Cycle Stop] keys to conduct corresponding functions. The moment power interruption or emergency stop occurs, you can press [Resume] key to resume machining from the interrupted point to save time on condition that the workpiece origin is accurate for sure.
- Extended keys area. [K1], [K2], [K3] and [K4] keys are included, used for user-defined functions.
- Auxiliary function keys area. See Fig. 1-10 for details.

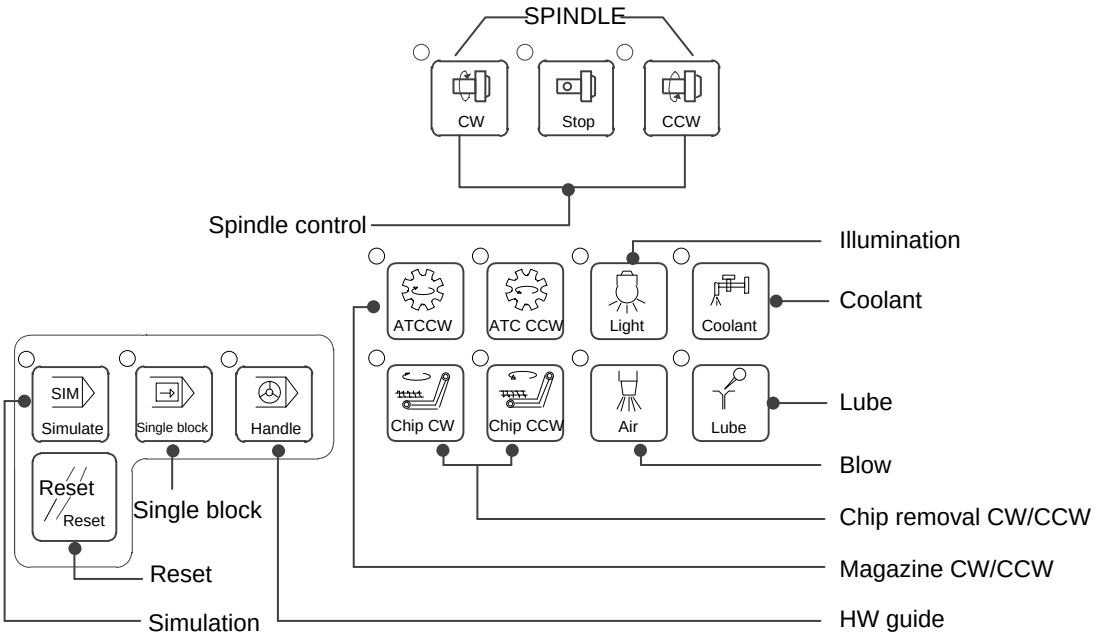


Fig. 1-10 Auxiliary function keys

- Power ON/OFF switch, used to turn ON/OFF the power supply.

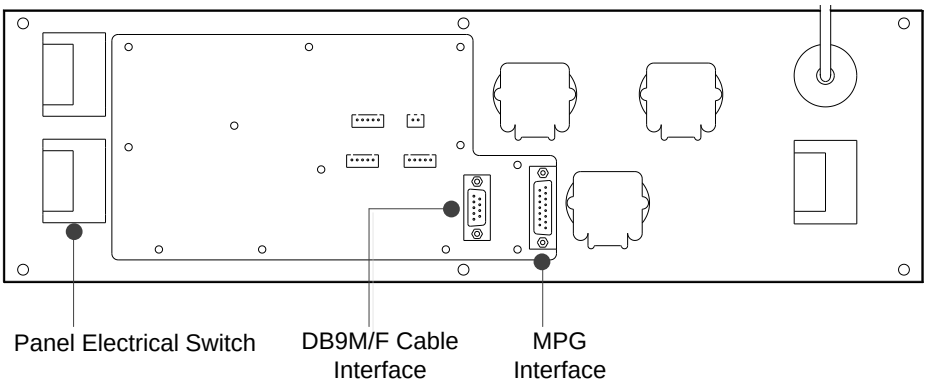


Fig. 1-11 Rear view of WH106A3 operation panel

- Panel electrical switch. It is used to control the power supply of operation panel. Refer to section 2.2 for details.
- DB9M/F cable interface. Connect WH106C operation panel with keypad panel via DB9M/F cable (40cm).
- MPG interface. It is used for connection with MPG (also called handwheel or handle).

2) Keypad Panel KB1A1

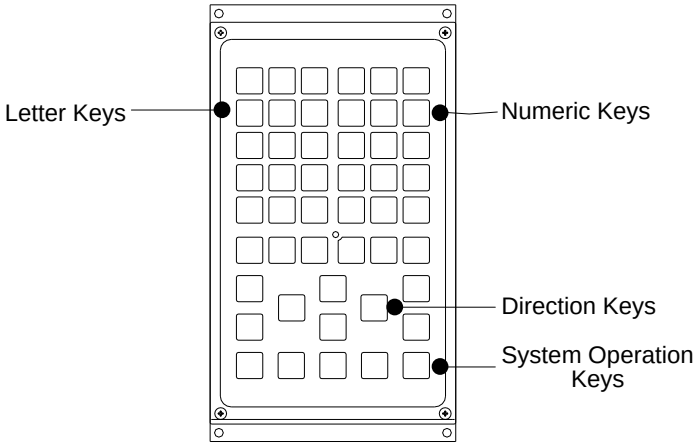


Fig. 1-12 Front view of keypad panel KB1A1

- Letter keys area, where you can enter 26 alphabetic letters. Together pressing [Shift] key and alphabet key or double pressing the alphabet key can input the letter on the upper-left of the key.
- Numeric keys area, where you can enter number or sign. Together pressing [Shift] key and number key can input the sign on the upper-left of the key.
- Direction keys area, also called arrow keys, including Up (↑), Down (↓), Left (←) and Right (→) keys. Besides, you can locate the cursor to the beginning or end by pressing [Home] or [End] key directly.
- System operation keys area. There are 7 keys, including [PgUp], [PgDn], [Enter], [Del], [Select], [Caps], and [Esc], used for jumping to the previous page, jumping to the next page, confirmation, delete, selection, input of letters in upper case and exit respectively.

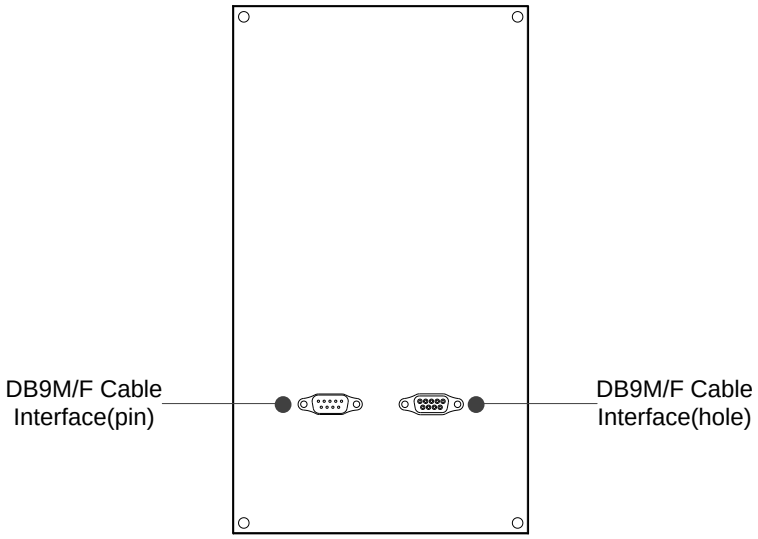


Fig. 1-13 Rear view of keypad panel KB1A1

- DB9M/F cable interface (pin), used to connect with the operation panel.
- DB9M/F cable interface (hole), used to connect with the host.

3) Operation Panel WH108A1

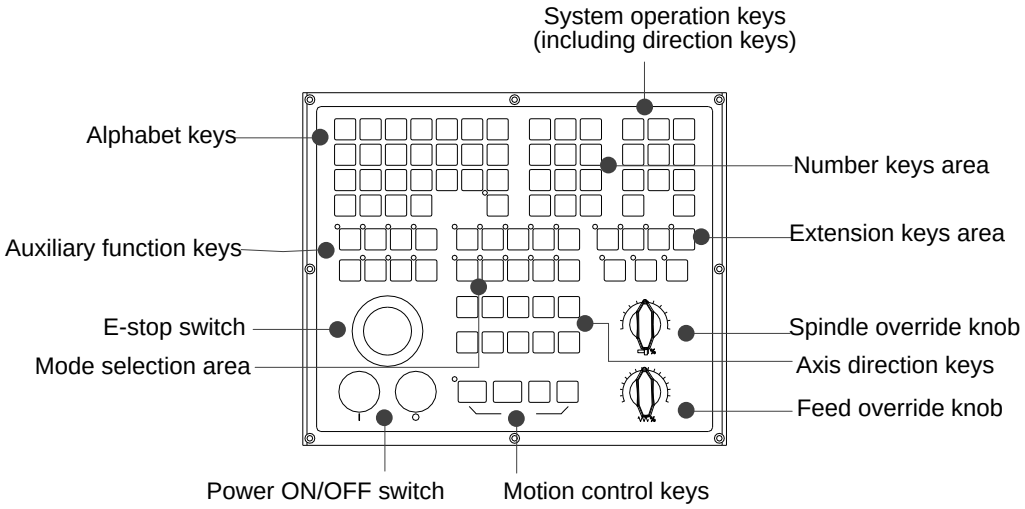


Fig. 1-14 Front view of operation panel WH108A1

Most keys on WH108A1 and WH106A3 are the same, except for their layout as well as minor difference of auxiliary function keys and mode selection keys. For example, mode selection keys on WH108A1 correspond to the mode selection knob on WH106A3 operation panel.

Refer to Fig. 1-15 for concrete keys of auxiliary functional area and mode selection area.

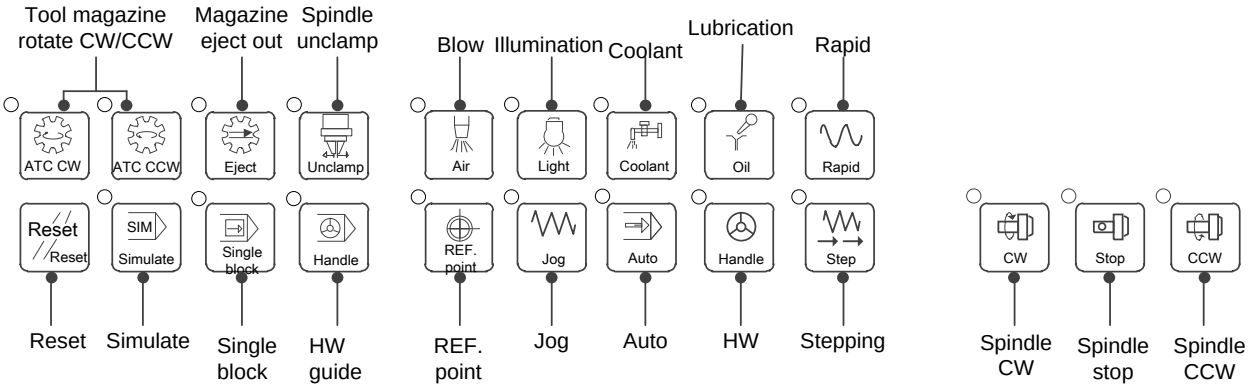


Fig. 1-15 Auxiliary functional keys and mode selection keys

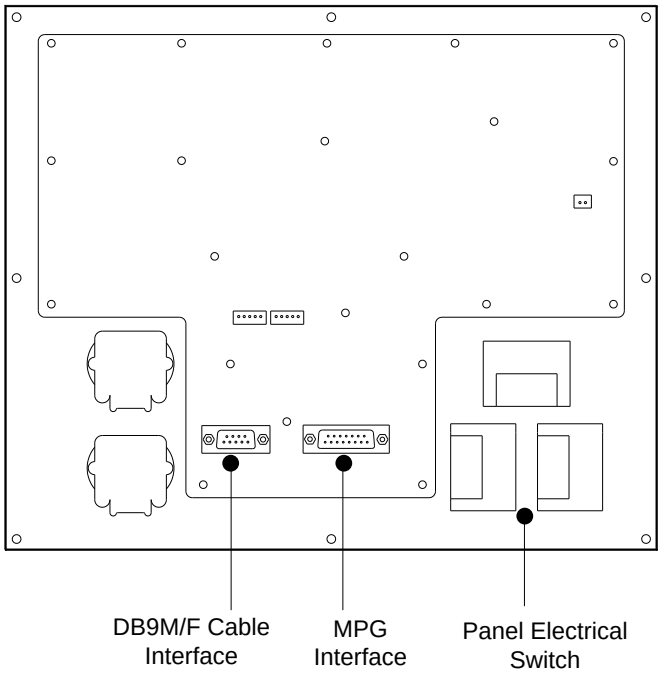


Fig. 1-16 Rear view of operation panel WH108A1

- DB9 cable interface, used for connection with the host.
- MPG interface, used for connection with MPG, handle, or handwheel.
- Panel electrical switch, used for controlling power supply for the operation panel. See section 2.2 for details.

CAUTION

A light on indicator on the upper-left side of a key represents for activation of the function indicated by the key.

1.2.2 Overall Connection Diagram

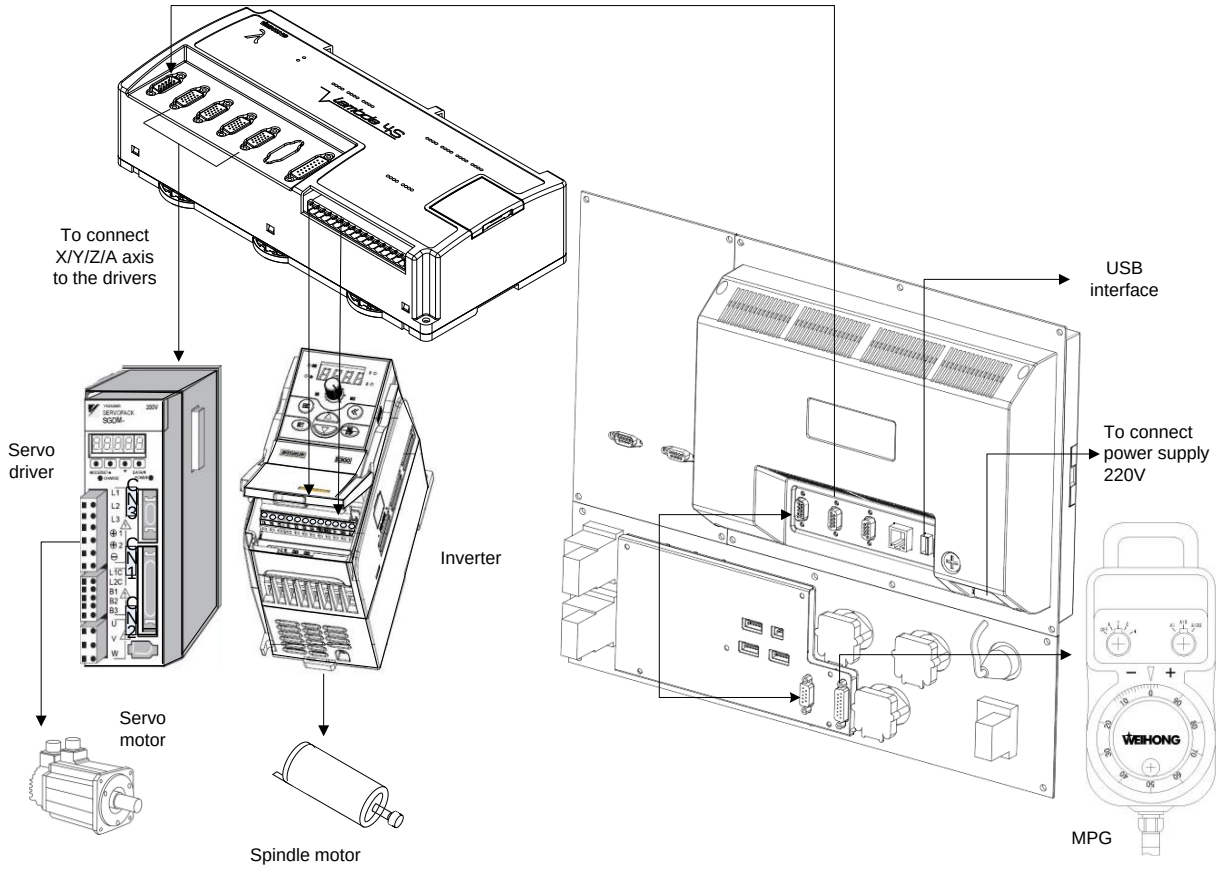


Fig. 1-17 Overall connection diagram of NK300BX-H

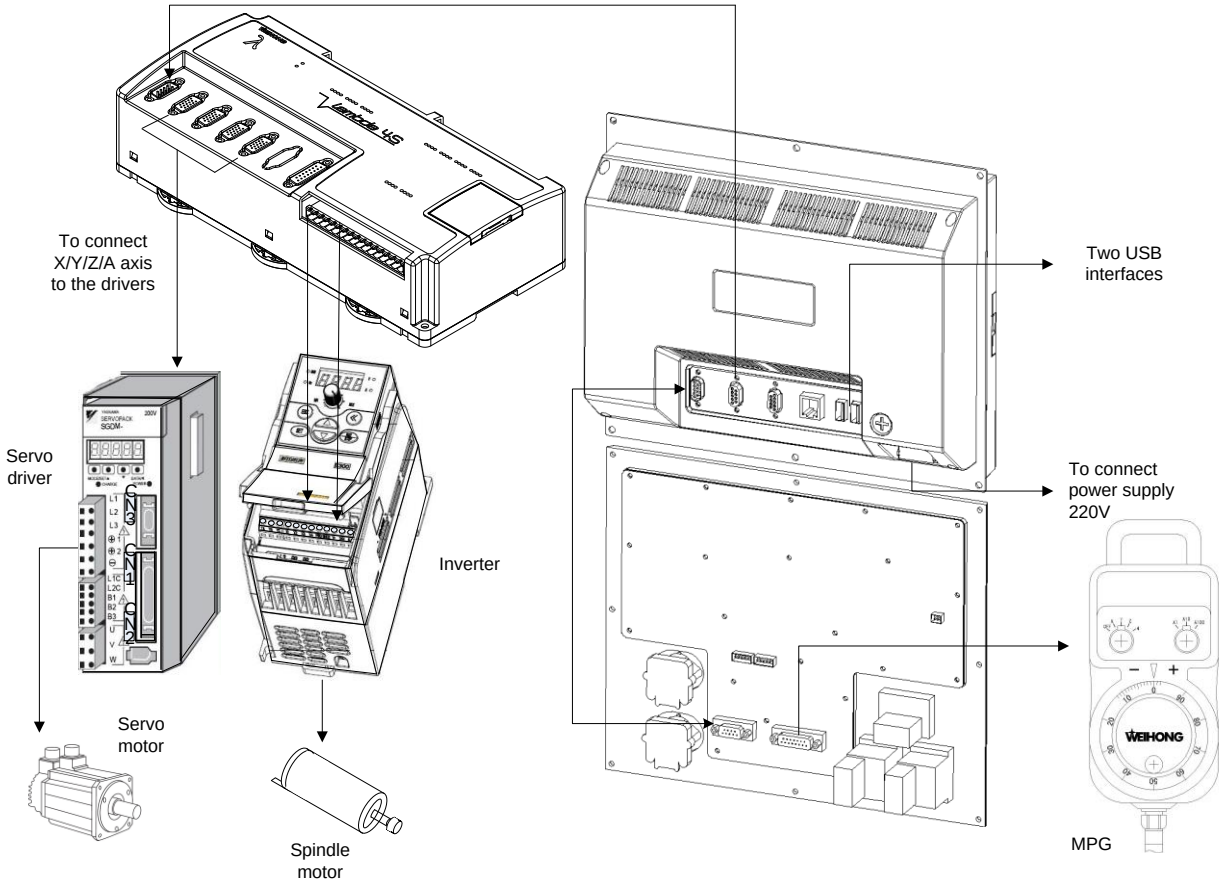


Fig. 1-18 Overall connection diagram of NK300BX-V

2 Wiring

2.1 Basic Concepts of Signal

2.1.1 Signal Types

The signal types of NK300BX system can be divided into the following 4 types: binary input signal, relay output signal and differential output signal and analog signal.

◆ **Binary Input Signal**

Binary input signal is active low/high. Conducting to GND (i.e. grounding signal) in NO connection means signal detected, while disconnecting with GND in NC connection means signal detected.

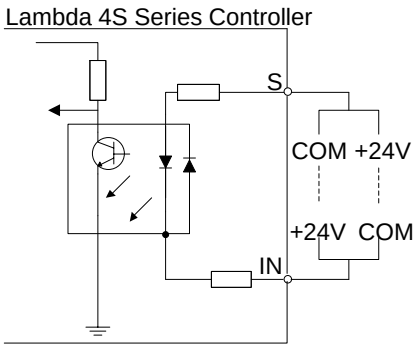


Fig. 2-1 Connection of binary input and mechanical switch

CAUTION

NK300BX system supports inputs active high/low. When the common port S on the Lambda 4S controller is connected to COM, inputs are active high after they are connected to +24V; when connected to +24V, inputs are active low after they are connected to COM.

◆ **Relay Output Signal**

The outputs on the Lambda 4S controller are relay outputs, and the relay output contact points have load capacity—7A/250VAC and 7A/30VDC, to control 220V AC load of low power. If high power load is needed, a contactor can be used. See Fig. 2-2.

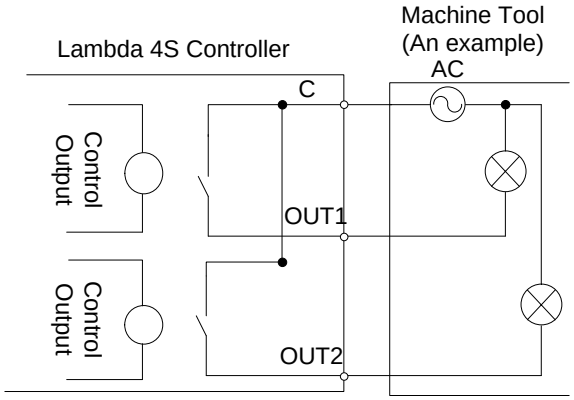


Fig. 2-2 Connection of relay output and contactor

◆ **Differential Output Signal**

Pulse command format to control driver motion is pulse + direction, negative logic. The max. pulse frequency is 1MHz. See Fig. 2-3 for pulse mode.

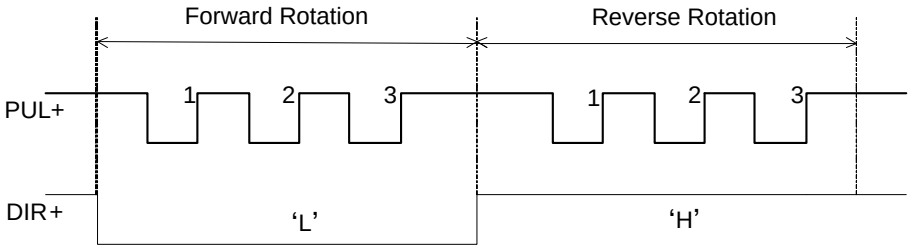


Fig. 2-3 Pulse command output mode

See Fig. 2-4 for differential signal output mode.

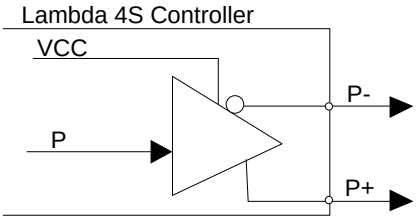


Fig. 2-4 Pulse Command Output Circuit

◆ **Analog Output Signal**

SVC port, externally connected with the inverter analog voltage frequency command input port, can output voltage controlled from 0V to 10V. And it can control inverter frequency by voltage change in order to master spindle speed.

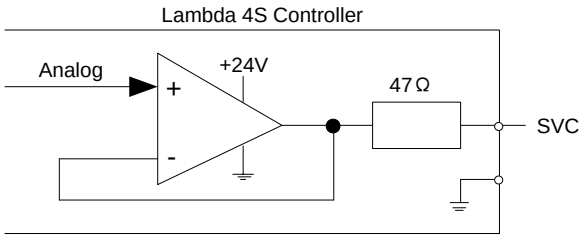


Fig. 2-5 Analog output signal circuit

2.2 Electrical Switch Wiring Diagram of Control Panel

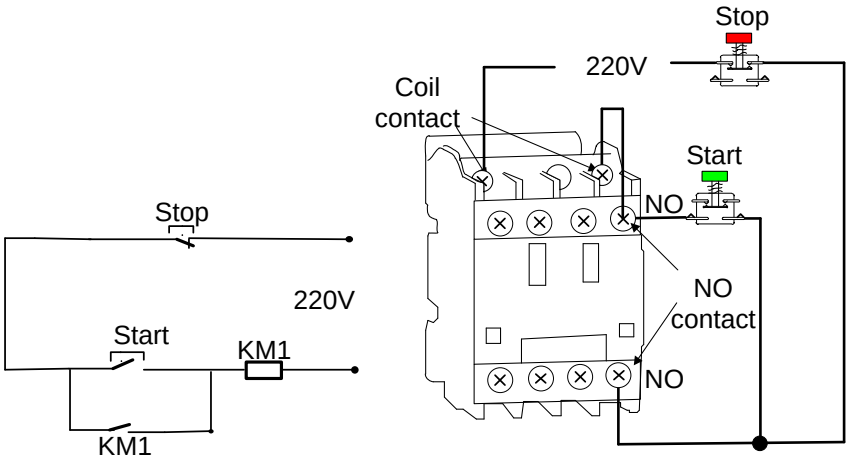


Fig. 2-6 Wiring diagram of electrical switch on 106 panel

2.3 Terminal Wiring Specification

NK300BX system is equipped with Lambda 4S controller as standard, with optional terminal board—EX31A1 for cascade connection and EX27A3 for non-cascade connection, to expand inputs and outputs. Lambda 4S controller and extended terminal board EX31A1 and EX27A3 both support active high/low.

A red LED indicator next to each input on the Lambda 4S controller is used to tell whether the input port receives the input signal or not. The concrete method: taking a NO switch as an example, press down the switch to send the signal to the corresponding port; if the LED near the port is on, it indicates the signal is successfully sent to the input port; if not, check whether the input is wrongly connected.

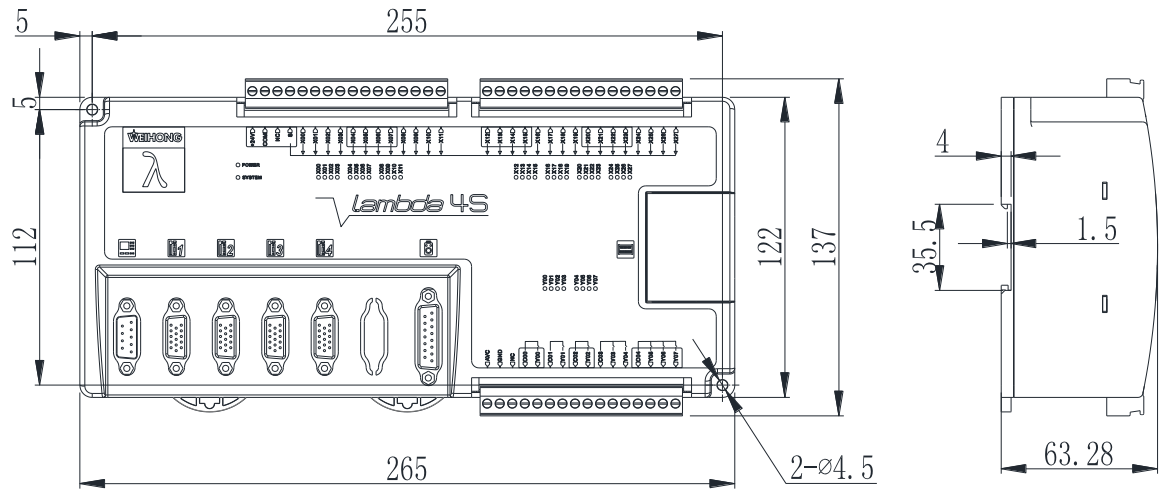
In addition, the flash frequency of the LED indicator for system (LED next to the SYSTEM port) can be used to tell the state of system communication. Here are several situations, for reference only.

- When physical connection is interrupted, the LED indicator flash on and off in turn: ON 0.5s and OFF 0.5s, and etc.
- If physical connection is normal while data connection is interrupted, with no upload block of logical

data or data, the indicator flash on and off in turn at a lower frequency: ON 1.5s and OFF 1.5 s, etc.

- If physical connection is normal, namely data connection and transmission is normal, the indicator flash on and off in turn: ON 0.25s and OFF 0.25s, etc.
- When system abnormal occurs, such as crc confirmation error, full pulses, hardware encryption error and the like, the indicator flash on and off in turn: ON 0.05s and OFF 0.05s.
- When hardware failure occurs, such as under voltage, hardware damage, false weld and short circuit and the like, the indicator will be completely OFF or completely ON.
- For Lambda 4S/5S serial controller, when PC logical data or physical connection is interrupted, the I/O ports state of both the controller and extended terminal boards will remain unchanged.

2.3.1 Mounting Dimension of Lambda 4S Controller



Mounting Dimension of Lambda 4S

Fig. 2-7 Mounting Dimension of Lambda 4S

2.3.2 Wiring Diagram of Lambda 4S Controller

To secure a normal communication of electrical circuit, a protective circuit is added to terminals Y00/C00, Y01/C01 (marked with ★ in the wiring diagrams below) on Lambda 4S controller, which is used for brake function. Therefore, the two groups of terminals should only be connected with load of voltage lower than or equal to 24V, otherwise, they will be damaged and cannot work due to burn-out of the piezoresistor.

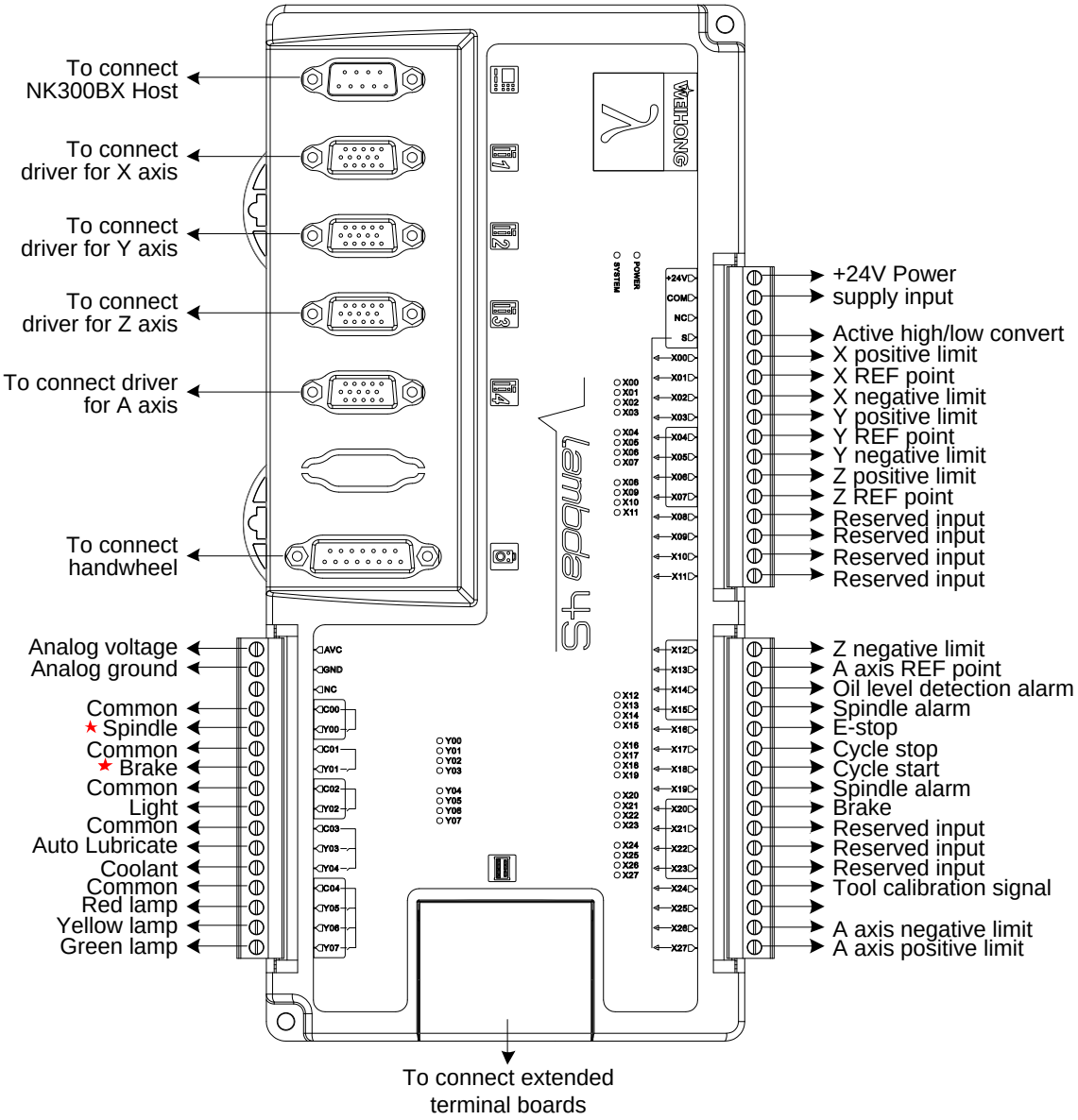


Fig. 2-8 Wiring diagram of Lambda 4S controller

CAUTION

1) The "Brake" output port should be connected with 24V voltage.

2) To secure normal communication of electrical circuit, protective circuit is added for Y00/C00 and Y01/C01 ports on Lambda 4S controller. Note that Y00/C00 and Y01/C01 ports can only be supplied by power lower than 24V; otherwise, piezoresistor may be burn out and result in circuit failure.

3) Lambda controllers used is varied because of different types of extended terminal boards. For cascade extended terminal EX31A1, Lambda 4S (LD5S) controller is used, while for non-cascade extended terminal board EX27A3, Lambda 4S controller is used.

4) For rotary Y configuration of four axes, the rotary Y axis is outputted through the fourth port by default; while for double Y axes configuration of three axes, the Y2 axis is outputted through the fourth axis port by default.

2.3.3 IO Specification of Lambda 4S Controller

Table 2-1 IO specification of Lambda 4S controller

Classification	Pin No.	Description	Remark
External power	+24V	DC 24V power	Powered by external power supply.
	COM		
Common	S	S port	When the common S is connected with COM port, it is active high; while connected with +24V, it is active low.
Origin signal	X01	Reference point of X-axis	Binary input, connects to home switch of X-axis
	X04	Reference point of Y-axis	Binary input, connects to home switch of Y-axis
	X07	Reference point of Z-axis	Binary input, connects to home switch of Z-axis
	X13	Reference point of 4th-axis	Binary input, connects to home switch of 4th-axis
Limit signal	X00	Positive limit of X-axis	Binary input, connects to positive limit switch of X-axis
	X02	Negative limit of X-axis	Binary input, connects to negative limit switch of X-axis
	X03	Positive limit of Y-axis	Binary input, connects to positive limit switch of Y-axis
	X05	Negative limit of Y-axis	Binary input, connects to negative limit switch of Y-axis
	X06	Positive limit of Z-axis	Binary input, connects to positive limit switch of Z-axis
	X12	Negative limit of Z-axis	Binary input, connects to negative limit switch of Z-axis
	X26	Negative limit of 4th-axis	Binary input, connects to negative limit switch of 4th-axis
	X27	Positive limit of 4th-axis	Binary input, connects to positive limit switch of 4th-axis
Commonly used inputs	X14	Oil level detection alarm	Binary input signal, connects to oil level sensor
	X15	Spindle alarm	Binary input signal, connects to spindle alarm switch
	X16	E-stop	Binary input signal, connects to E-stop button on the operation panel

Classification	Pin No.	Description	Remark
	X17	Stop	For external connection with a reset switch
	X18	Start	For external connection with a reset switch
	X19	Spindle alarm	Binary input signal, connects to spindle alarm switch
	X20	Brake	Connects to brake coil of servo driver
	X24	Tool presetter signal	Binary input signal, connects to tool presetter
Reserved inputs	X08~X11	Reserved	Reserved for user-defined inputs
	X21~X23	Reserved	Reserved for user-defined inputs
Outputs	SVC	Spindle rotational speed control	$\frac{\text{Actual Spindle Rotational Speed}}{\text{Max. Spindle Rotational Speed}} = \frac{\text{SVC}}{10\text{V}}$
	GND	Analog ground	
	Y00	SPIN	Controls spindle ON and OFF
	Y01	Brake	Connects to brake coil of servo driver
	Y02	Light	For external connection with an illuminating lamp
	Y03	Auto lubrication	Controls auto lubrication; relay contact output, LED on during lubrication and off when lubrication stops
	Y04	Coolant	Relay contact output, two terminals equaling to a switch; connects to workpiece cooling switch
	Y05	Red	Red light on when machining ends regularly or during E-stop
	Y06	Yellow	Yellow light on during idle state after machining ends or during wait state
	Y07	Green	Light on during normal working state of machine

2.3.4 Mounting Dimension of Terminal Board EX31A1

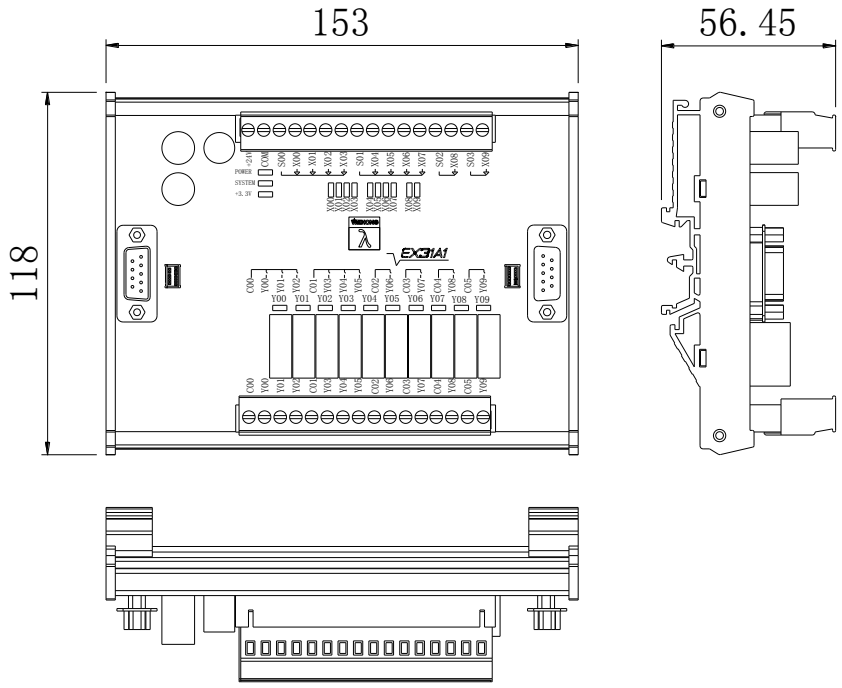


Fig. 2-9 Mounting Dimension of Terminal Board EX31A1

2.3.5 Wiring Diagram of Terminal Board EX31A1

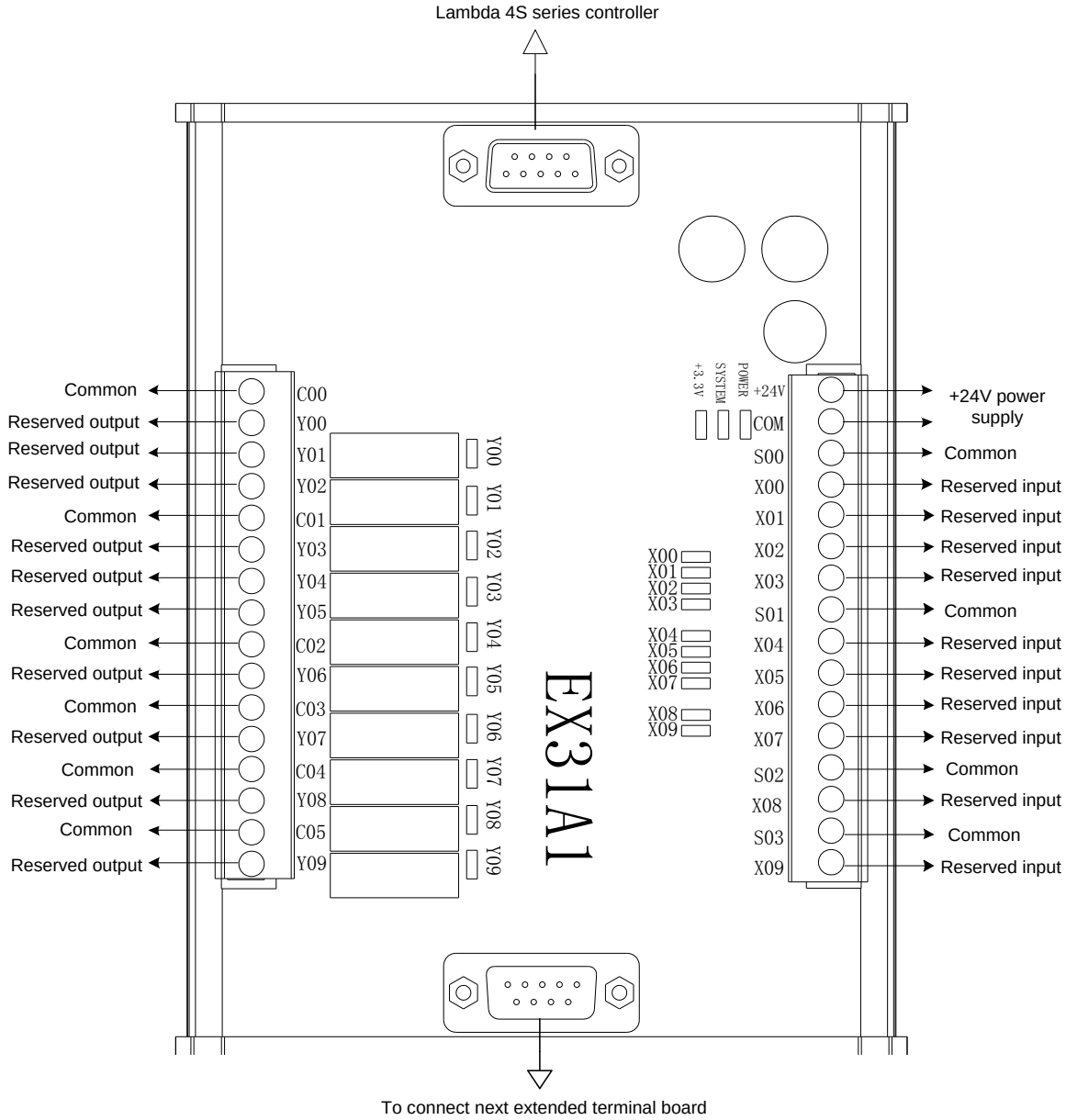


Fig. 2-10 Wiring diagram of terminal board EX31A1

CAUTION

1) When common port S is connected with +24 V power supply, X00 ~ X09 are active low; while common port S is connected with COM, X00~X09 are active high.

2) EX31A1 is used together with Lambda 4S (LD5S) controller.

2.3.6 Mounting Dimension of EX27A3

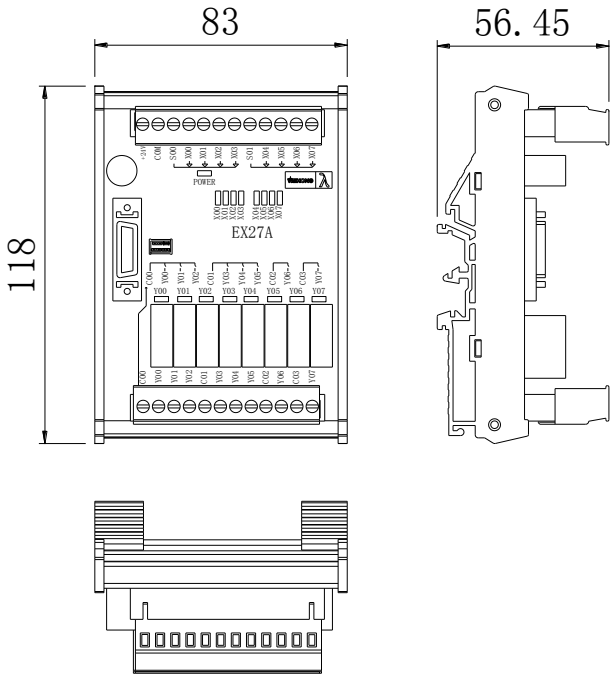


Fig. 2-11 Mounting Dimension of Terminal Board EX27A3

2.3.7 Wiring Diagram of Terminal Board EX27A3

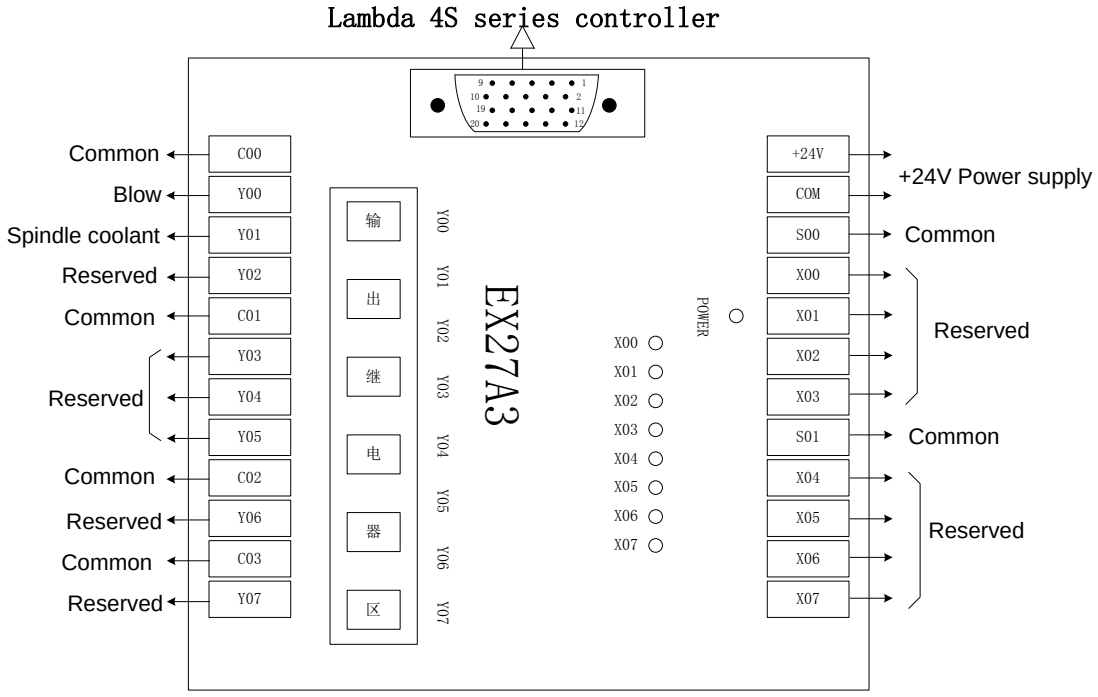


Fig. 2-12 Wiring diagram of terminal board EX27A3

CAUTION

1) X00~X07 correspond to PLC addresses of port 00216~00223 in the software.
2) When common port S is connected with +24V, X00~X07 are active low; while connected with COM, they are active high.
3) EX27A3 is used together with Lambda 4S controller.

2.4 Port Definition and Wiring Specification

2.4.1 Driver Interface Definition

NK300BX system provides 4 pulse feed driver interfaces. The type of the 4 interfaces is 15-pin D-type socket (DB15 pins). The pin definition is as shown in Fig. 2-13.

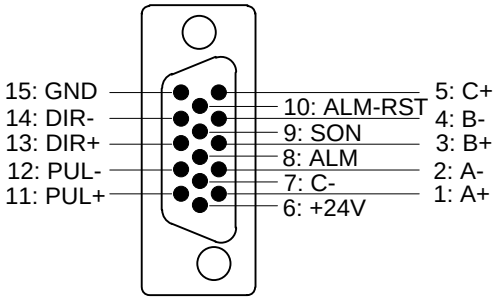


Fig. 2-13 Driver interface definition

Table 2-2 Driver Interface Description

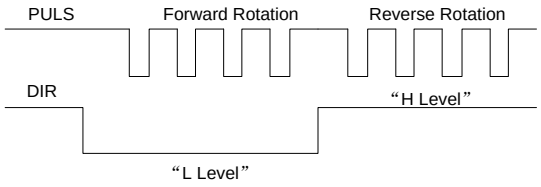
Name	Definition	Input /Output	Description
A+, A-	Feedback signal of encoder phase A	Input, differential signal transmission mode	Receive the differential output from encoder signal (phase A, B, C) of driver frequency divider (equaling to RS422).
B+, B-	Feedback signal of encoder phase B	Input, differential signal transmission mode	
C+, C-	Feedback signal of encoder phase C	Input, differential signal transmission mode	
ALM	Driver alarm signal	Input	When breakdown occurs in driver, the output (transistor) will be closed or disconnected.
SON	Servo ON signal	Output	This signal is used for opening (power on) and closing (power off) servo motor. When this signal is connected to COM-, dynamic brake will be released and thus the driver is allowed to work (servo enabled).
ALM-RST	Driver alarm clear	Output	This signal is used for alarm/warning

Name	Definition	Input /Output	Description
	signal		status clear, and can only remove the alarms that can be removed.
PUL+, PUL-	Pulse output	Output, differential signal transmission mode	
DIR+, DIR-	Direction output	Output, differential signal transmission mode	
+24V	DC 24V power	Output	Connected to driver

CAUTION

SON signal will be effective in 2 seconds after connecting of power supply. Don't try to drive the motor through the external servo ON or servo OFF drive signal at any time, since the software will control the power-up state of the servo motor.

Table 2-3 Technical Specifications

Technical parameters	Description
Max. pulse frequency	1M
Encoder power	+5V, 150mA
Encoder signal	RS422 level
Signal output	Differential signal output, both pulse and direction signals adopt differential signal transmission mode.
Pulse format	Pulse + direction, negative logic. The "pulse + direction" output wave form of NK300BX is shown as follows: 

2.4.2 Handwheel Interface Definition

NK300BX can be externally connected to a manual pulse generator (MPG, or called handwheel). The interface consists of DB15-pins dual-in-line holes, and the pins definition is as shown in Fig. 2-14.

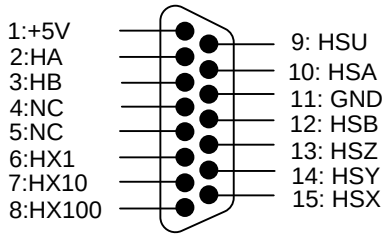


Fig. 2-14 Handwheel interface definition

Table 2-4 Handwheel interface description

Pin No.	Definition	Description
1	+5V	Power on handwheel
2	HA	Encoder phase A signal
3	HB	Encoder phase B signal
4	NC	
5	NC	
6	HX1	Selection of X1 override
7	HX10	Selection of X10 override
8	HX100	Selection of X100 override
9	HSU	Selection of the 4th axis
10	NC	
11	GND	Digital ground
12	NC	
13	HSZ	Selection of Z-axis
14	HSY	Selection of Y-axis
15	HSX	Selection of X-axis

2.4.3 USB Interface Definition

There are two USB interfaces at the back of NK300BX host, another one on the front for external connection of an USB device (E.g. U disk).

3 Operation

3.1 Debugging Steps

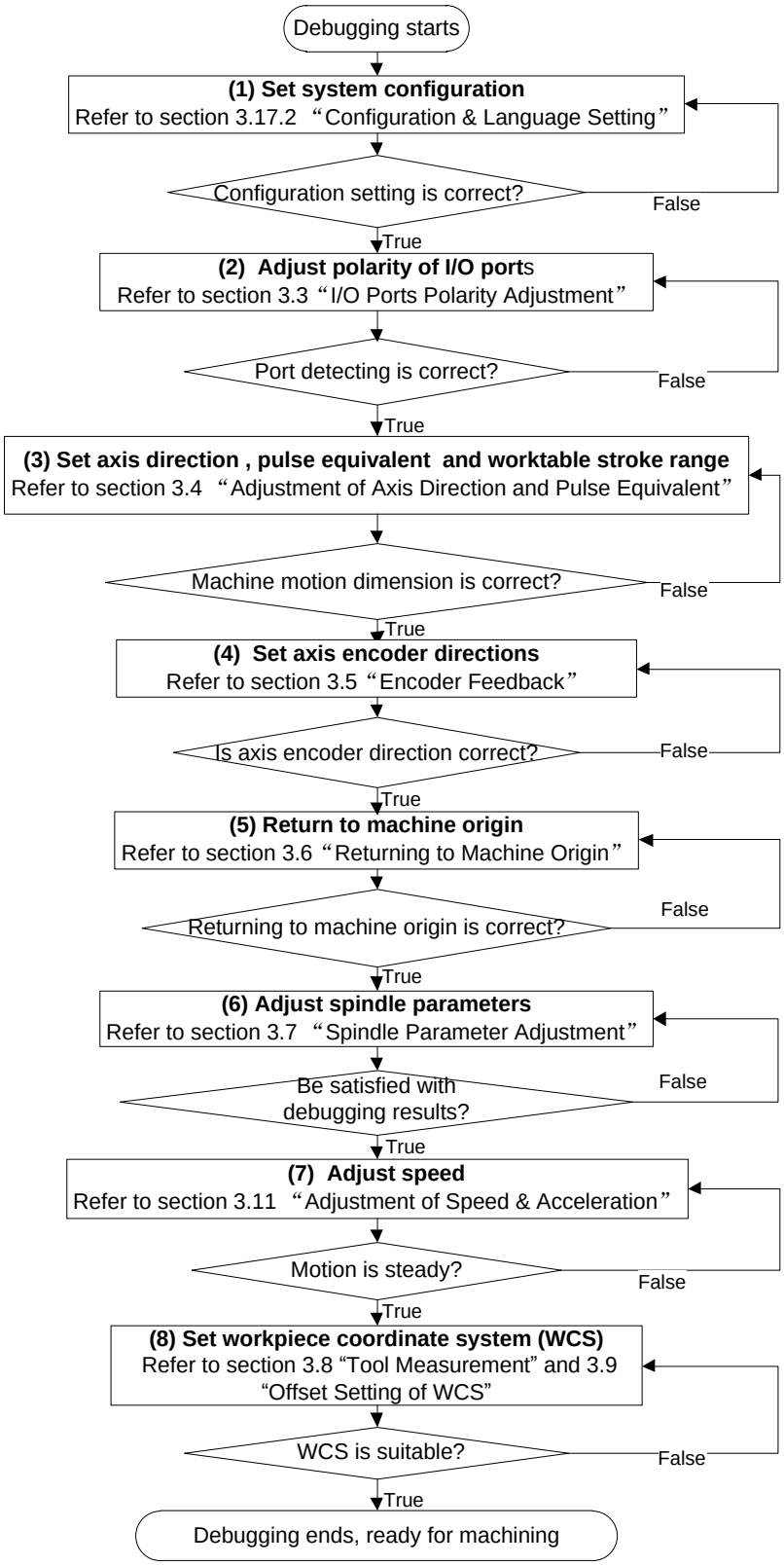


Fig. 3-1 Debugging steps

3.2 Operational Interfaces

Catering to operational habits of different users, the NK300BX software offers two types of interfaces. You can choose a main interface style by setting parameter N80030 “Use Old Mainpage”.

The classic main interface is shown in Fig. 3-2.

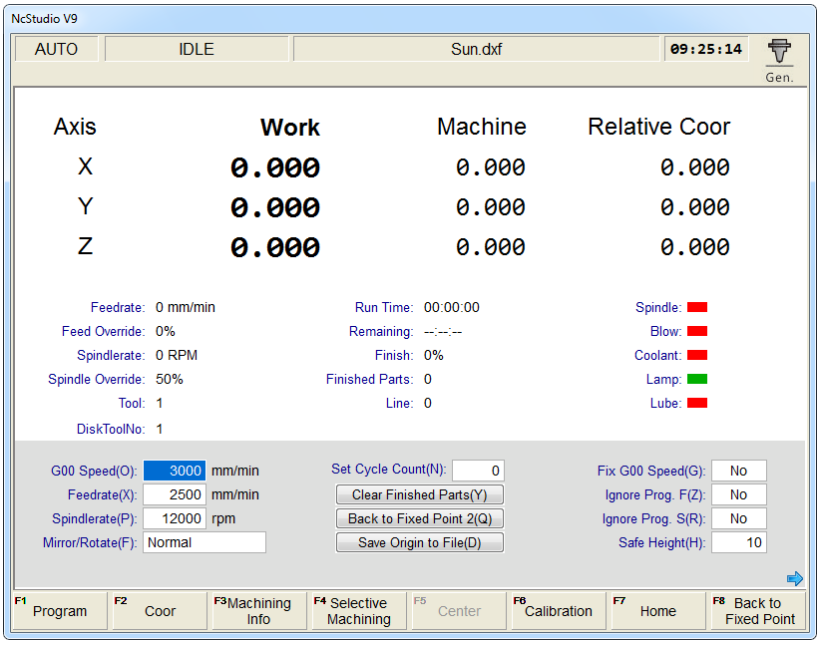


Fig. 3-2 Classic main page of NK300BX software

The new main page of the software is as shown in Fig. 3-3

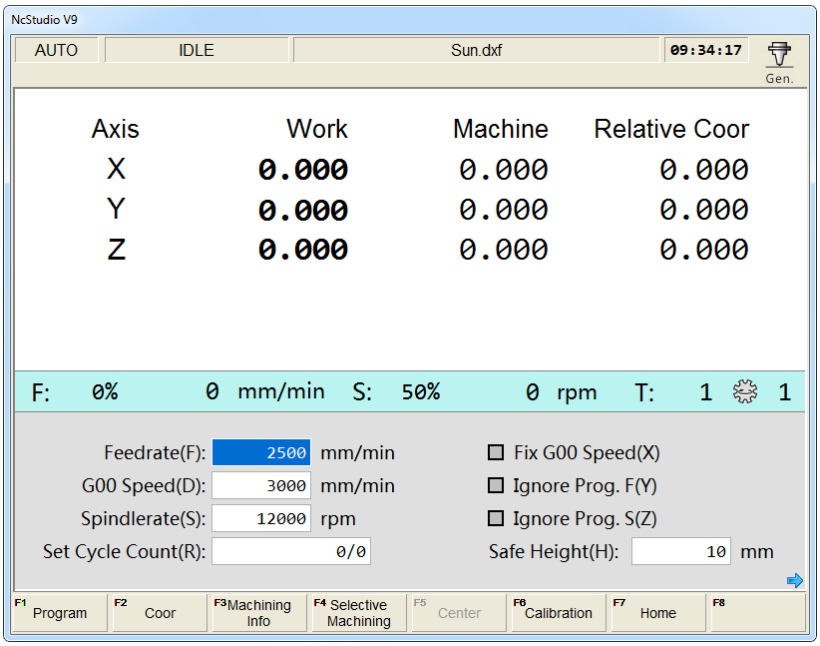


Fig. 3-3 New main page of NK300BX software

Because most operations and methods are same for the classic and new interfaces, the following chapters will introduce operations of the NK300BX system taking the classic interface as an example.

Explanations will be made for the difference, please note that.

To toggle between the classic and new interfaces, press function button  to access the parameters area, press next button  under the monitor to open the next manipulation buttons bar, and then press F1 button to open [Software Option] tab. With the help of arrow key " ↑ " or " ↓ ", move the cursor onto the parameter N80030, and press Enter key to open a dialog box for modifying parameter value, as shown in Fig. 3-4. Choose "Yes" to activate the classic interfaces while choose "No" to activate the new interfaces. The modification takes effect after the software is restarted.

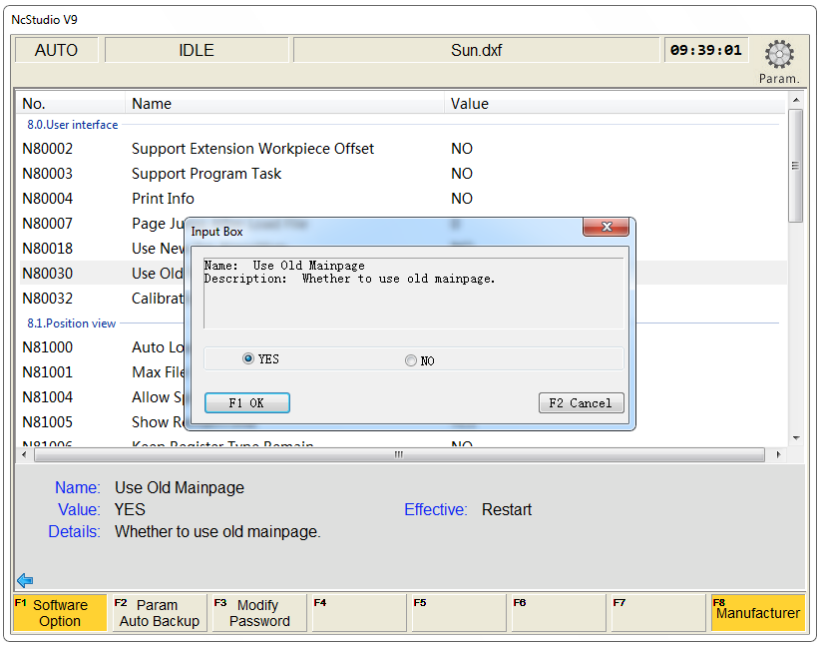


Fig. 3-4 Parameter setting for interface switchover

◆ **Related Parameters**

Parameter		Definition	Setting Range
N80030	Use Old Mainpage	Whether to use old main page.	Yes: use; No: do not use

3.3 I/O Ports Polarity Adjustment

The polarities of input/ output ports in the software are specified in terms of the switch type: the polarity of normally closed switches should be "NC"; the polarity of normally open switches should be "NO". In the software interface, the ports with preceding filled dot  are input ports, while the ones with hollow point  are output ports.

After the connection of a machine tool and power on, the dots should be in red in front of reference point,

E-stop, cycle start, cycle stop and tool presetter signal indicating these signals are invalid, or it is necessary to check whether the connection is correct. If there is no problem with the connection, the polarity of the corresponding port should be changed.

The method of modifying polarity: press the diagnosis function selection key  to access [Diagnosis] interface after opening NcStudio → press F3 to enter [I/O Port] window → select the target I/O port for modification by pressing the Up and Down keys→ press F4 [Invert Polarity]. Restart to validate the modification to port polarity.

[I/O Port] screen is as shown in Fig. 3-5, and some function screens need password before operation, such as [Test On], [Test Off], [Cancel All], [Invert Polarity], [Change Property] [Display All] and [Watch Port].

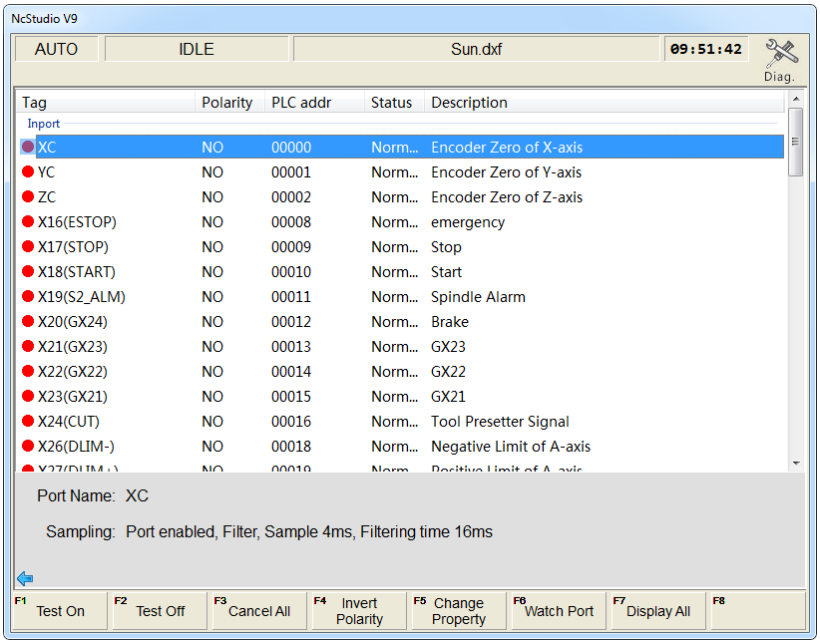


Fig. 3-5 I/O port screen

◆ **[Test On], [Test Off]**

The shortcut keys are F1 and F2 respectively, which are only available under [I/O Port] screen.

Pressing down F1 or F2 will make the indicator light before the port selected shift between green and red. Green light means there is signal in the port; red light means there is no signal in the port.

This group of buttons is mainly used for simulating hardware signal, which is for simulation test.

CAUTION

The indicator lights before ports are slightly different in test mode and in practice:

Green light in test mode: Red light in test mode:

Green light in practice: Red light in practice:

Output can be detected by testing the signals of ports.

◆ **[Cancel All]**

The shortcut key is F3, which is only available under [I/O Port] screen.

Pressing F3 will cancel simulation test and signals to replace analog signals with real hardware signals.

◆ **[Invert Polarity]**

The shortcut key is F4, which is only available under [I/O Port] screen.

Pressing F4 will change port polarity between NO and NC.

The polarities of feedrate override, spindle override, mode switch, handwheel and encoder zero should be “NO”.

Except for particularly defined ones, the polarities of output ports are generally “NO”.

◆ **[Change Property]**

The shortcut key is F5, which is only available under [I/O Port] screen.

After F5 is pressed, a new manipulation dialog will appear, as shown in Fig. 3-6.

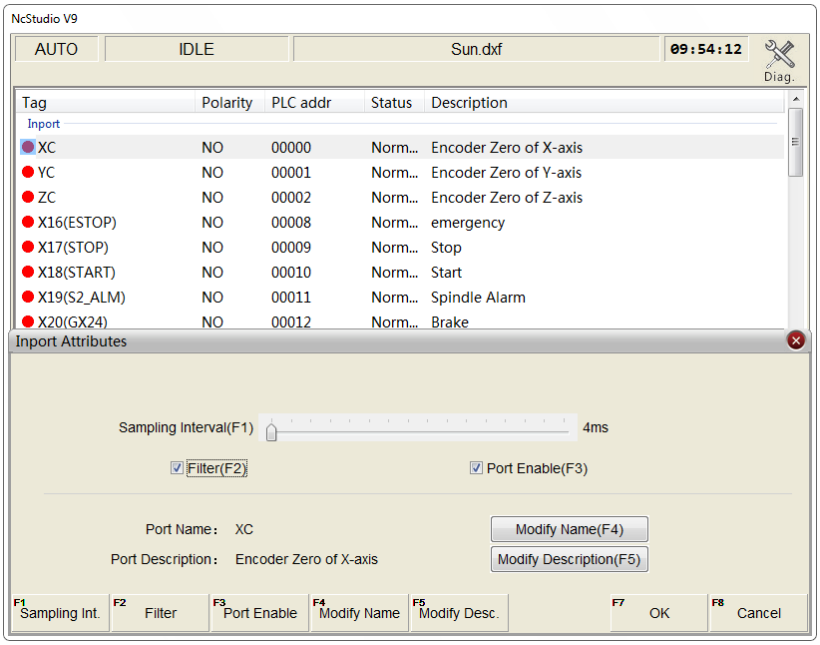


Fig. 3-6 Setting window of port property

In this port attribute dialog, you can set sampling interval (Press “F1” to activate interval adjustment, and press PageUp/ PageDown keys to adjust interval time), filter function and port enable function, and modify port name and port description.

◆ **[Watch Port]**

After selecting one port, press the shortcut key F6 to enable the monitor function of this port, refer to section 3.14.2 [Diagnosis] window for detail.

◆ **[Display All]**

Pressing F7 will display all the input and output ports (including the undefined ports concealed in ex-factory setting). And pressing F7 again will hide all the unused input and output ports.

3.4 Adjustment of Axis Direction and Pulse Equivalent

3.4.1 Axis Direction Adjustment

The first thing to do in machine debugging is to confirm the positive direction of each axis. The coordinate system of right-hand rule is as shown in Fig. 3-7.

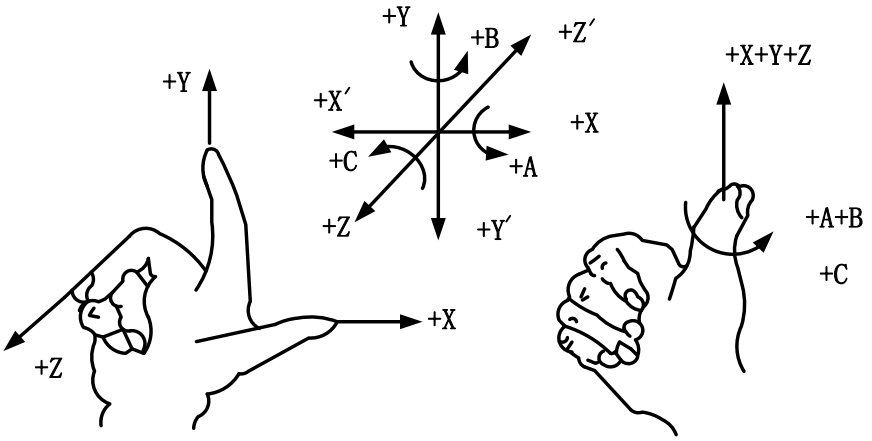


Fig. 3-7 Standard coordinate system of right-hand rule

The axis directions of a machine are decided by both the type of the machine tool and the layout of each component. The basic coordinate axes of engraving & milling machines/ routers are X-, Y-, and Z-axis:

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

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

——The positive direction of Y-axis is the cutter moving away from the operator (+Y).

◆ **Related Parameters (Under "Axis Parameter")**

Parameter		Definition	Setting Range
N10000	Axis Direction (X/Y/Z)	It specifies the motion direction of each axis.	"1" and "-1" represent the two motion directions of each axis.
Fix the positive direction of each axis following the right-hand rule, and then manually operate			

Parameter	Definition	Setting Range
the machine to check if the axis moves in the correct direction. If the direction is opposite, modify the value of N10000. Take X-axis as an example, manually move X-axis, just to find it moves oppositely, just change the X value of N10000 from "-1" ("1") to "1" ("-1").		

3.4.2 Pulse Equivalent Adjustment

Pulse equivalent (p): the moving distance of workbench or rotation degree of rotary axis per pulse sent by the CNC device, the minimum available distance controlled by the CNC system as well. The smaller the pulse equivalent is, the higher the machining precision and surface quality will be. The large, the faster feedrate will be. Therefore, lower pulse equivalent should be set under condition of meeting the demand of feedrate. The relationship between maximum feedrate and pulse equivalent is as following:

$$\text{Max. Feedrate} = \text{Pulse Equivalent} \times 60 \times \text{Frequency}$$

For example, the hardware frequency of NK300BX is 1MHz, and provided the pulse equivalent is 0.001mm/p, then:

$$\text{Max. Feedrate} = 0.001 \times 60 \times 1000000 = 60\text{m/min}$$

Mechanical deceleration ratio (m/n): the ratio of reducer input speed to output speed, equal to the ratio of the teeth number of driven wheel to that of driving wheel. When applied in CNC machines, it specifies the ratio of motor speed to screw speed.

$$\text{Mechanical Deceleration Ratio} = \frac{\text{Reducer Input Speed}}{\text{Reducer Output Speed}} = \frac{\text{Teeth No. of Driven Wheel}}{\text{Teeth No. of Driving Wheel}} = \frac{\text{Motor Rotational Speed}}{\text{Screw Roational Speed}}$$

Pitch (d): The axial distance between the corresponding points of two adjacent teeth on the threads.

The calculation of pulse equivalent varies with different motor systems.

◆ **Stepping Motor**

In general, firstly set the subdivision and then calculate the pulse equivalent. You can also set the pulse equivalent before calculating subdivision. Their relationship can be shown as:

$$\frac{d}{p} = \frac{360}{\theta} \times x \times \frac{m}{n}$$

Hereinto, p stands for pulse equivalent, x represents subdivision of stepping motor while θ refers to stepping angle. Therefore,

$$\text{Pulse Equivalent} = \frac{\text{Screw Pitch}}{\frac{360}{\text{Stepping Angle}} \times \text{Subdivision} \times \text{Mechanical Deceleration Ratio}}$$

For instance, the selected screw lead of X-axis for a certain type of machine tool is 5mm, the stepping angle of stepping motor is 1.8 degree, with "10" subdivision and motor directly connected with screw by coupling. Thus, the pulse equivalent of X-axis is:

$$\text{Pulse Equivalent} = \frac{5\text{mm}}{\frac{360}{1.8} \times 10 \times 1} = 0.0025\text{mm/p}$$

◆ **Servo Motor**

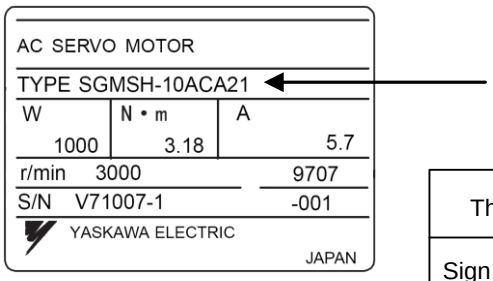
In general, set the default value of pulse equivalent (p) as 0.001mm/p and calculate electronic gear ratio (B/A). Their relationship can be shown as:

$$\text{Electronic Gear Ratio } \frac{B}{A} = \frac{\text{Encoder Resolution}}{\frac{\text{Screw Pitch}}{\text{Pulse Equivalent}}} \times \text{Mechanical Deceleration Ratio}$$

Namely, $\frac{B}{A} = \frac{F \times p}{d} \times \frac{m}{n}$

Electronic gear ratio: if servo motor makes one circle per every 5000 pulse commands sent by the system, setting electronic gear ratio of servo motor can make servo rotate twice with the same amount of pulse commands (please refer to parameters setting of the specific servo).

Please see the servo motor label plate compared to the corresponding manual to confirm its encoder resolution. A label plate of YASKAWA SGMSH type servo is as shown below, and the 4th character in motor type is the serial encoder specification, with resolution of 2^{17}, i.e. 131072.



Motor Type：
TYPE SGMSH-1 0 A C A 2 1
(The 4th Character)

The 4th Character: Serial Encoder Spec.		
Sign	Spec.	Remark
2	17-bit absolute	Standard
C	17-bit incremental	Standard

Fig. 3-8 Servo motor brand-encoder resolution

For instance: (an example of YASKAWA servo) screw pitch of a certain type of machine is 5mm, with 17 bit encoder resolution, “0.0001mm/p” pulse equivalent and “1:1” deceleration ratio.

$$\text{Electronic Gear Ratio} = \frac{PN202}{PN203} = \frac{2^{17}}{5/0.0001} \times 1 = \frac{8192}{3125}$$

◆ **Rotary Axis**

The pulse equivalent of rotary axis refers to the rotation degree of the axis clamping the workpiece corresponding to each pulse. The rotated degree of workpiece per revolution of motor equals to screw pitch.

➢ For Stepping Motor

$$\text{Pulse Equivalent} = \frac{360}{\frac{360}{\text{Stepping Angle}} \times \text{Subdivision} \times \text{Mechanical Deceleration Ratio}}$$

➢ For Servo Motor

$$\text{Electronic Gear Ratio } \frac{B}{A} = \frac{\text{Encoder Resolution} \times \text{ Pulse Equivalent}}{360} \times \text{Mechanical Deceleration Ratio}$$

◆ **Related Parameters (Under "Axis Parameter")**

Parameter		Definition	Setting Range
N10010	Pulse Equivalent (X/Y/Z)	It refers to the displacement or angle generated on the relative feed axis per control pulse.	-0.0000009 ~ 999

CAUTION

The setting of pulse equivalent must be matching with the electronic gear ratio of servo driver or subdivision of stepping driver.

3.4.3 Upper & Lower Limit Setting of Worktable Stroke

Worktable stroke refers to the valid machining stroke range of a machine tool in the X, Y, and Z directions, and the system will carry out soft limit in terms of this range in order to protect the machine.

◆ **Related Parameters (Under "Axis Parameter")**

Parameter		Definition	Setting Range
N10020	Travel Limits-Negative(X/Y/Z)	It sets the machine coordinate of the allowable lower limit of worktable when the parameter N10040 is valid.	-99999 ~99999
N10030	Travel Limits-Positive(X/Y/Z)	It sets the machine coordinate of the allowable upper limit of worktable when the parameter N10040 is valid.	-99999 ~99999
N10040	Enable Travel Limits (X/Y/Z)	It sets whether to check the stroke range of worktable.	YES: enabled; NO: disabled
N67000/ N67001/ N67002	Negative Change Tool Travel Limits (X/Y/Z)	It sets the machine coordinate of the allowable lower limit of travel in tool change.	-99999 ~99999
N67010/ N67011/ N67012	Positive Change Tool Travel Limits (X/Y/Z)	It sets the machine coordinate of the allowable upper limit of travel in tool change.	-99999 ~99999

CAUTION

In the first setting of the upper & lower limit of worktable stroke, please verify the actually valid range of machine motion in case of accident.

3.5 Encoder Feedback

3.5.1 Direction Setting for Axis Encoder

You can find the parameter N11110 “Axis Encoder Dir” following operations: [Parameter] function area → F8 [Axis Parameter] → 1.1 Encoders.

There are two methods to decide and set the axis encoder direction, namely setting via operation and setting via reasoning.

◆ **Setting via Operation**

Taking X axis as an example, manually move X axis towards positive direction, and during the process, press E-stop button. If the coordinate value after adjustment is larger than the value before adjustment, it tells that the current [Axis Encoder Dir] is correct, otherwise, incorrect.

Likewise, manually move X axis towards negative direction, and during the process, press E-stop button. If the coordinate value after adjustment is smaller than the value before adjustment, it tells that the current [Axis Encoder Dir] is correct, otherwise, incorrect.

It is the same operation with other axes.

◆ **Setting via Reasoning**

On condition that the axis direction and pulse equivalent settings are correct:

$$\text{Axis Encoder Direction} = \text{Axis Direction} \times \text{Logical Direction}$$

For example, setting X axis direction to -1, and pulse to “Pulse + Direction, Negative Logic”. The value of parameter N11110 will be 1= [(-1)*(-1)].

CAUTION

Please refer to section 3.4.1 for axis direction setting, and refer to section 3.4.2 for pulse direction setting.

◆ **PG Frequency Division (×4) Setting**

Parameter N11160 “Frequency Division Pulse of PG (×4)” refers to the feedback pulses number when motor rotates one revolution without any acceleration or deceleration settings, or the feedback pulses the linear axis moves one-pitch-distance. During debugging in the field, please refer to the manuals for drivers of varied brands.

3.5.2 Parameter Specification

Parameter		Definition	Setting Range
N11304	Encoder Feedback	Whether to enable encoder feedback function or not.	Yes: Enable; No: Disabled

Parameter		Definition	Setting Range
N80004	Print Info	Whether to show debugging info or not.	Yes: Show; No: Not show
N11110	Axis Encoder Dir	It specifies the direction of encoder.	1: positive -1: negative
N11130	Check Encoder Error	Whether to check the encoder error between feedback value and output value or not.	Yes: check; No: not check
N11140	Static Tolerance	When the axis is steady, if the difference between the feedback value and output value is bigger than this value, alarm will occur.	1~999999
N11150	Dynamic Tolerance	When the axis is dynamic, if the difference between the feedback value and output value is bigger than this value, alarm will occur.	1~99999
N11160	Frequency Division Pulses of PG (×4)	It specifies the encoder feedback pulse number via frequency division of servo per revolution of motor.	1~99999

Encoder feedback function is used to detect and feedback the angular or linear displacement of servo motor. When the parameter is set to “No”, please refer to section 3.6.1 for the principle and process of returning to the reference point; when it is set to “Yes”, please refer to the principle and process of returning to the reference point in section 3.6.4.

Only when parameter N11304 is set to “Yes”, the parameter N80004 can be set to “Yes”.

Encoder error refers to the absolute difference value of pulse number sent and fed back. (E.g. value of |Un00C – Un00D| in YASKAWA servo) when the detected value is larger than the allowable value set by the parameter, the system will stop emergently and report “(X/Y/Z) Axis dynamic / static error alarm”.

The dynamic encoder error refers to the error in running. Dynamic error = Motion speed / Position loop gain. (Assuming that the feedrate of X axis is 6000mm/min, or 100mm/s, position loop gain of servo driver is $100s^{-1}$, the X axis dynamic error will be 100mm/s ÷$100s^{-1}$=1mm. If the pulse equivalent is 0.001mm/p, the dynamic of X axis at 6000mm/min will be 1000p. At this time, if the parameter setting value is lower than 1000p, and X axis has already returned to the REF point, the system will alarm and stop. In other words, the system will prompt X axis dynamic error exceeding setting value and the X axis will make relative adjustment.)

Static error refers to the encoder error when the system is in idle (with idle time longer than 8s). Default setting is 500.

3.6 Returning to Machine Origin

Origin of Machine Coordinate System (inherent coordinate system of a machine tool), also called mechanical origin, and home, is a fixed point assigned by design, manufacturing and debugging before

the machine tool leaving factory. After startup of the CNC system, it is necessary to back to machine origin (home all axes) manually or automatically.

CAUTION

The necessity of returning to machine origin: these below functions will not be activated until backing to machine origin completed: soft limit, setting fixed point and tool change.

◆ **The Process of Returning to Machine Origin**

The processes of returning to machine origin of X, Y, Z axis are included and identical, as shown in Fig. 3-9 (an example of X-axis).

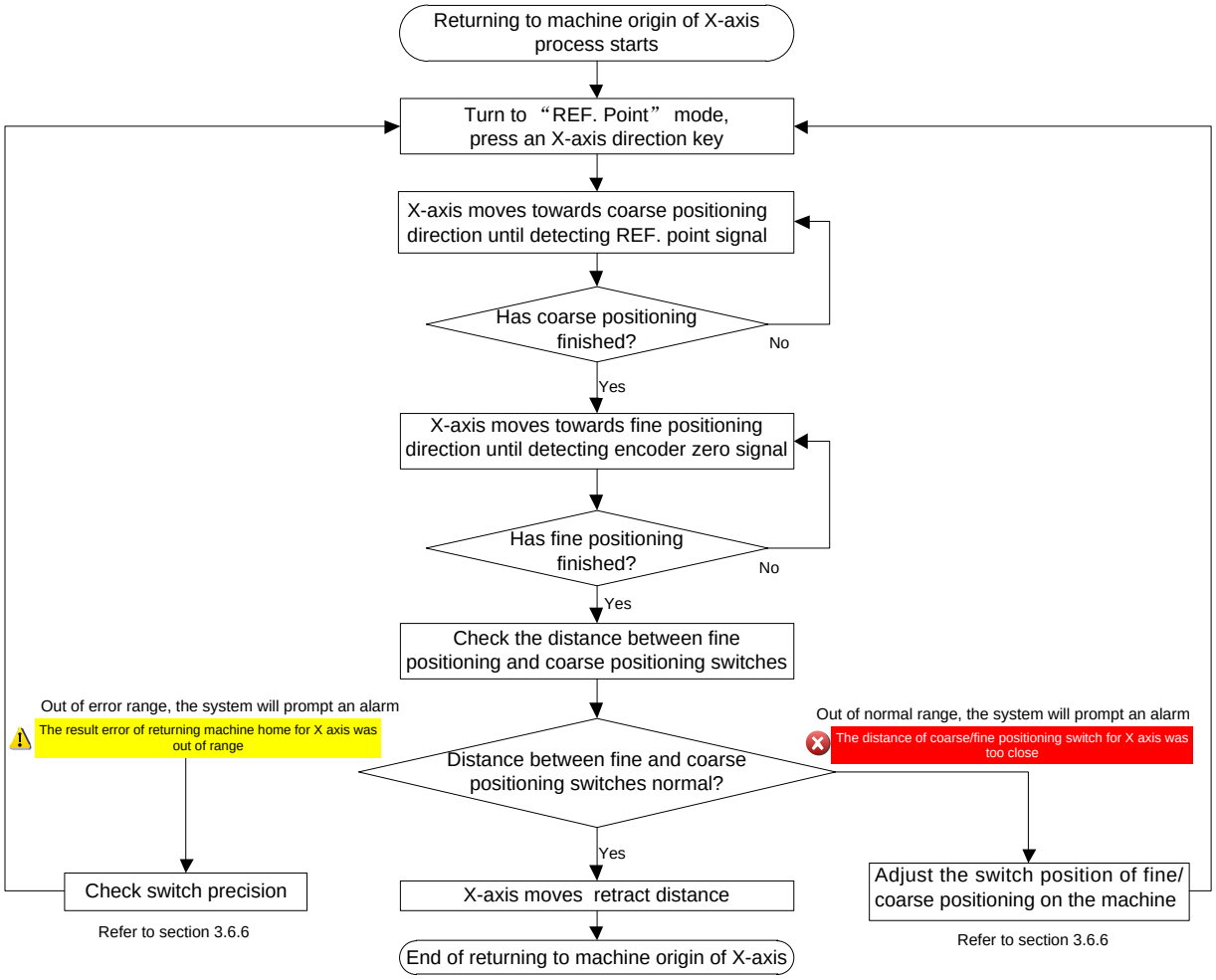


Fig. 3-9 The process of returning to machine origin (X-axis)

3.6.1 Returning to Machine Origin Operations under Three Axes Configuration

When the three axes configuration is active, turn the system into REF point mode, press function button

 to enter the following interfaces, one for classic interface and one for new interface.

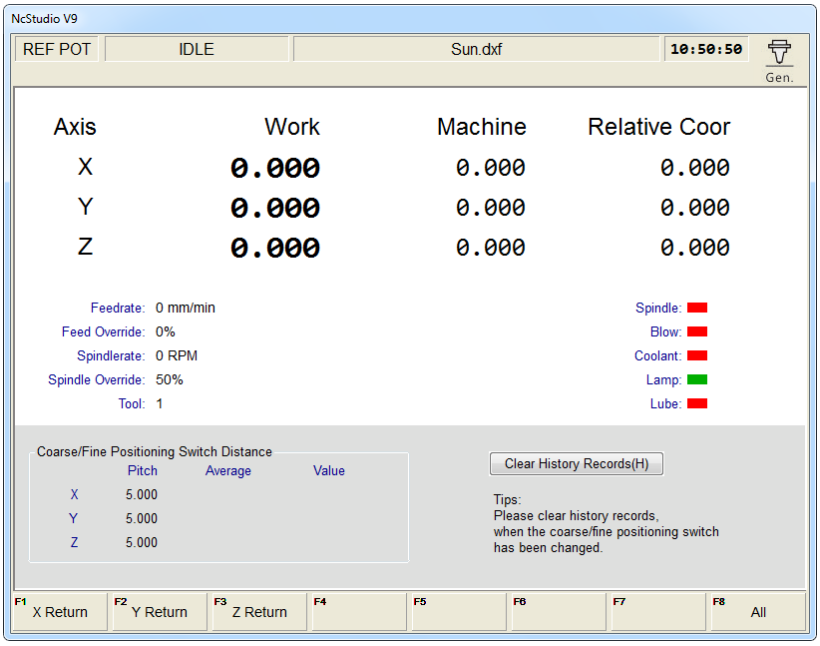


Fig. 3-10 Classic interface in REF mode---Three axes configuration

◆ **All Axes Returning to the Machine Origin**

Press F8 [All] to execute the operation. For safety, Z axis will return to REF point before other axes returning.

◆ **Single Axis Returning to the Machine Origin**

Press shortcut keys F1/F2/F3 to return a single axis to the REF point at a time.

The system entitles Z axis the highest priority in returning home by default. If Z axis is returned first, a prompt box will pop up, as shown below. Select “No” to exit the operation and “Yes” to make the selected axis return to the REF point.

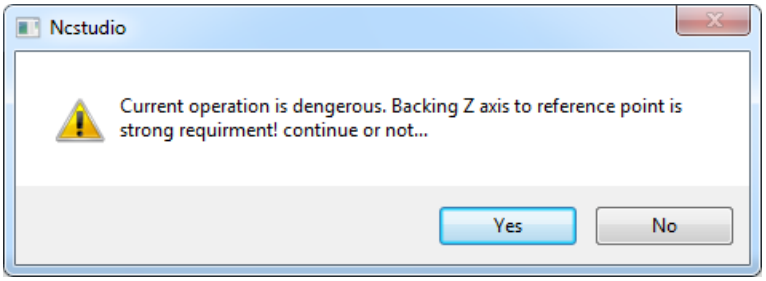


Fig. 3-11 Dangerous operation prompt

3.6.2 Returning to Machine Origin Operations under Four Axes Configuration

When four axes configuration is active, turn the system into REF point mode and access the operation

interface, as shown in Fig. 3-12.

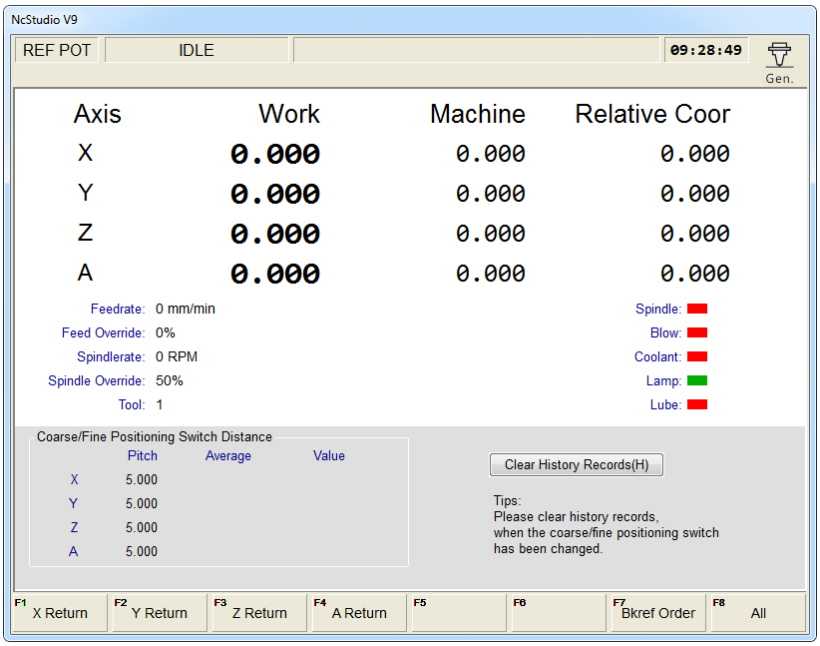


Fig. 3-12 Classic interface in REF mode---Four axes configuration

Compared with that under three axes configuration, users can freely define the returning order of axes. Press F7 [Bkref Order] to open setting dialog box, as shown in Fig. 3-13.

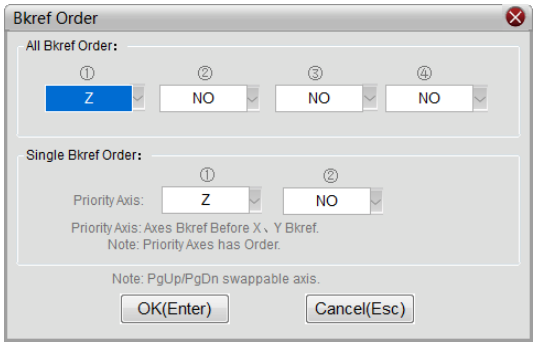


Fig. 3-13 Returning order setting dialog box

◆ All Axes Returning to Machine Origin

According to demand in the field, you can set returning order in the dialog box named “Bkref Order”, where you can press arrow keys to move the cursor, and press “PageUp” and “PageDown” keys to switch among options for each order position. See Fig. 3-14. When settings are done, press Enter for confirmation and back to the main interface in REF mode. Press F8 [All] to make all axes returning to the REF point according to the order you have set.

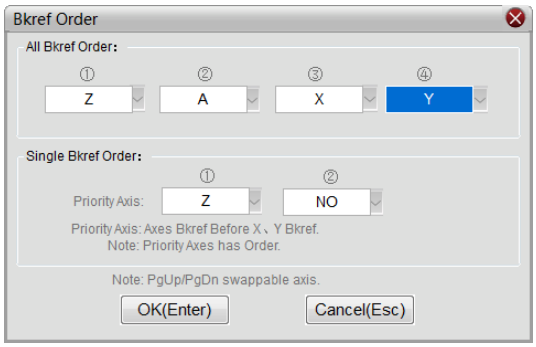


Fig. 3-14 All axes returning to the REF point

◆ Single Axis Returning to Machine Origin

Before returning single X/Y axis to the REF point, you can define one axis with priority, to avoid possible machine damage caused by mal-operations.

Likewise, users can make a prior axis returning before X/Y axis returning. Press arrow keys to locate the cursor and press PgUp and PgDn keys to switch among options for each order.

For example, users want an axis returning before X/Y returning, press PgUp and PgDn keys to select an axis for the first order. See below.

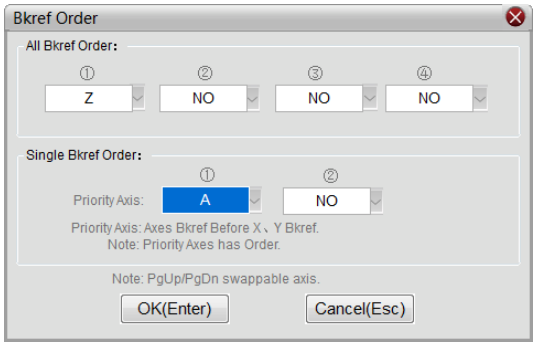


Fig. 3-15 Single axis returning order

After setting, press Enter for confirmation and turn to the main interface as Fig. 3-12, select single axis to return. The system will pop up prompt dialog box as shown in Fig. 3-16.

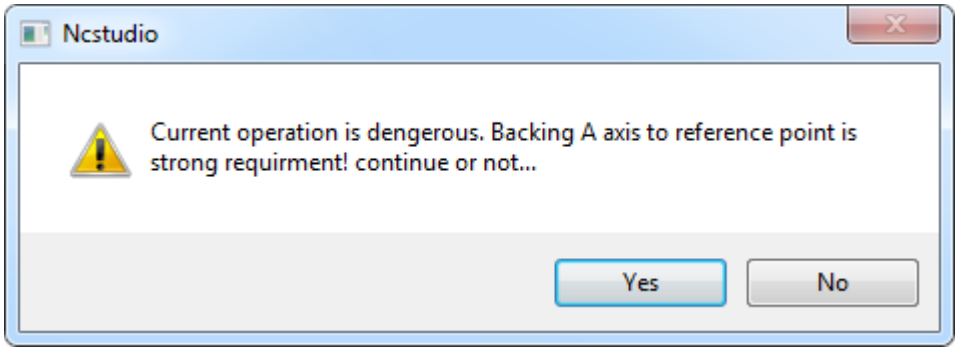


Fig. 3-16 Dangerous prompt

CAUTION

The system assumes Z axis returning as the priority by default. That is, if no modification has been made, the system will prompt Z axis returning first.

3.6.3 Principle of Returning to Machine Origin (without Encoder Feedback)

The encoder feedback function is involved in the system, specified by parameter N11304. The sketch map of returning to machine origin with servo motor is as below (without encoder feedback):

◆ **Coarse Positioning Stage**

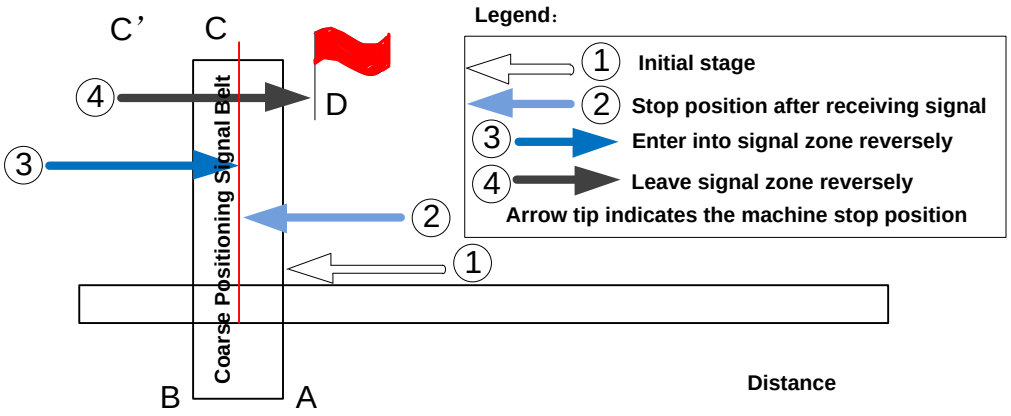


Fig. 3-17 Sketch map of coarse positioning (stopping within the signal belt after receiving coarse positioning signal)

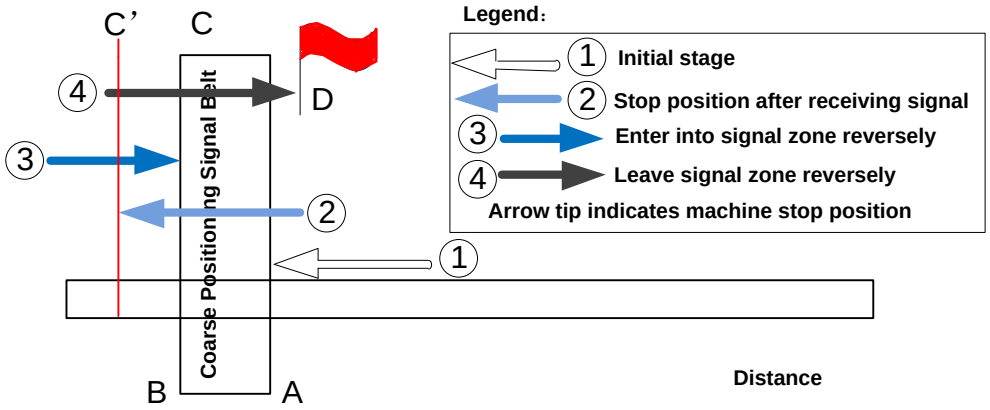


Fig. 3-18 Sketch map of coarse positioning (stopping out of the signal belt after receiving coarse positioning signal)

1) When the machine keeps moving until receiving REF. point signal at place A, it should stop immediately, but it may stop at place C or C' due to time lag and inertia.

2) The machine keeps moving reversely at one third of coarse positioning speed until receiving REF. point signal (if the machine has stayed in the signal belt in the above step 1, it will make no motion in this step).

3) The machine keeps moving reversely at one-tenth of coarse positioning speed until the REF. point signal disappears (across the signal belt).

4) The machine halts at the red flag place D after the end of this stage.

◆ Fine positioning Stage

The process of fine positioning stage is identical with that of coarse positioning stage.

After coarse positioning, the machine will move to encoder zero rapidly, executing slow positioning several times.

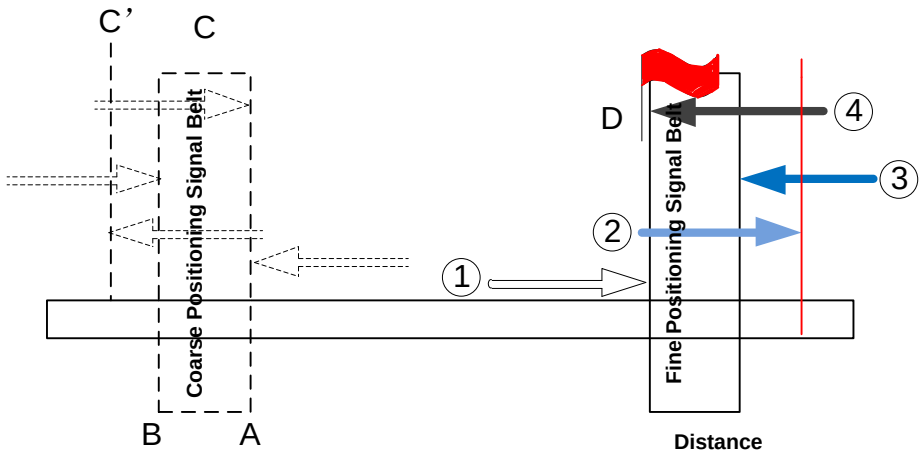


Fig. 3-19 The process of fine positioning

◆ Retracting Stage

After finishing the fine positioning stage, the system will execute retracting motion once with recommended retract distance as half of the screw pitch. The sketch map is shown in Fig. 3-20.

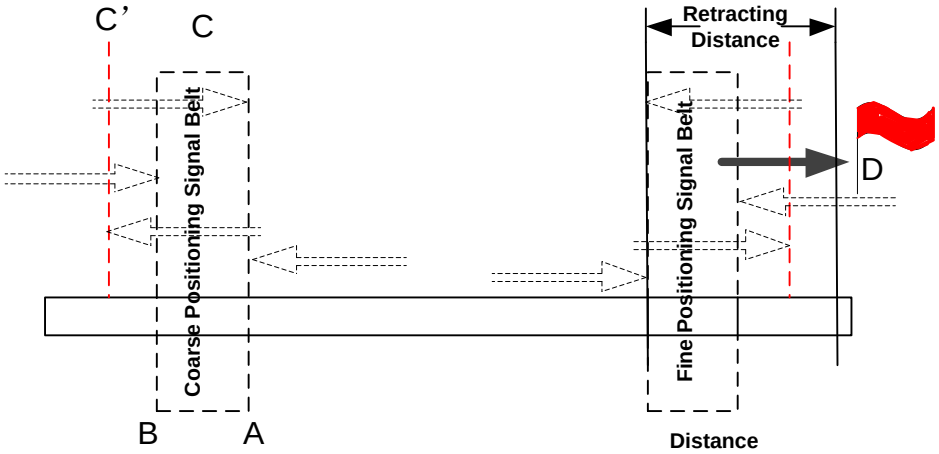


Fig. 3-20 Retracting stage

3.6.4 Principle of Returning to Machine Origin (with Encoder Feedback)

With encoder feedback function, the system will execute coarse positioning and fine positioning only once in returning to machine origin. The retracting distance after fine positioning is the actual retracting distance adjusted in terms of actual situation. And the concrete process is as below:

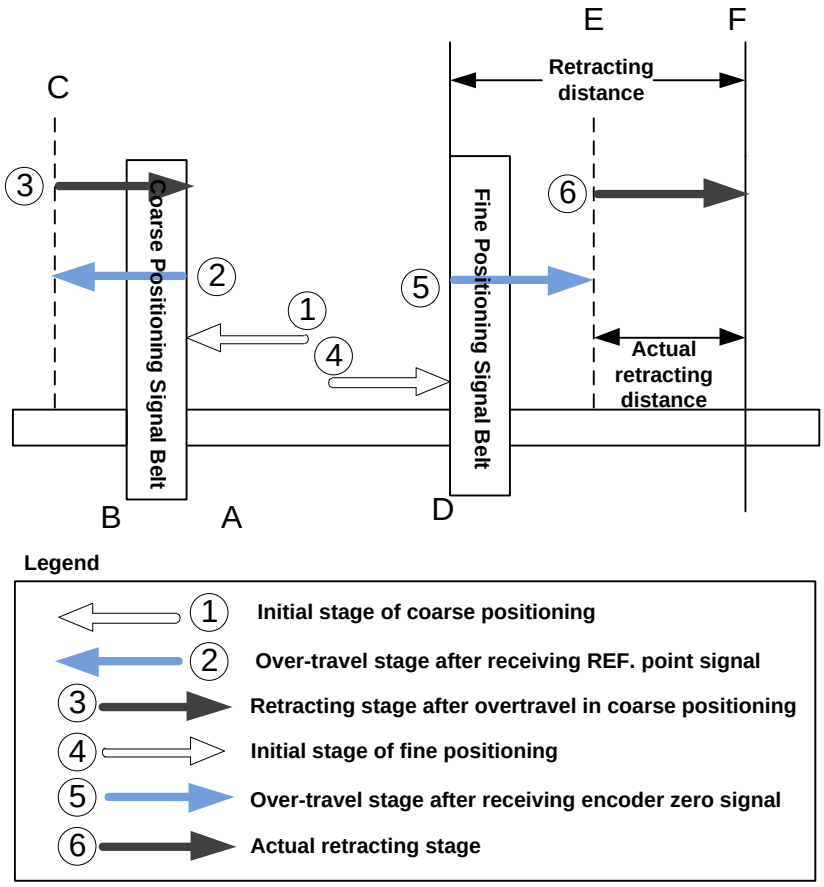


Fig. 3-21 The process of returning to machine origin

1) In coarse positioning stage, the machine tool should stop immediately at place “A” when receiving REF. point signal, but it may stop at place “C” due to over-travel caused by inertia and time-lag.
2) The machine executes retracting stage of coarse positioning.
3) In fine positioning stage, the machine tool moves reversely and should stop at place “D” immediately when receiving encoder zero signal; at the meantime, the feedback data of encoder will be latched, but the machine will generate over-travel and stop at place E due to inertia and time-lag. And the displacement between encoder zero signal and stop position (i.e. DE) is the across distance of signal deceleration.
4) In terms of the retracting distance and across distance calculated, calculate the actual retracting distance, and use this actual distance to make the machine move and stop at place F, keeping it free from the signal source.

3.6.5 Parameters Specifications

◆ **Related Parameters of Safe Operations**

Parameter		Definition	Setting Range
N74000	Cancel REF Sign when Reset	Once reset operation is executed in machining, whether to remove the mark of backing to machine origin.	YES : Cleared NO: Not cleared
N74001	Back to REF Required	Whether backing to machine origin before machining is required or not.	YES: Required NO: Not required
N74002	Cancel REF Sign when Estop	Whether the mark of backing to machine origin will be cleared or not once E-stop occurs.	YES: Cleared NO: Not cleared
Returning to machine origin before machining can avoid machining offset to ensure position precision. Setting "YES" for N74002 and N74000 is recommended so that once E-stop or reset operation occurs in the process of machining, the mark will be cleared, and the system will remind to home all the axes again. When N74001 is set to "YES", if there is no backing to machine origin mark "⊕" before each axis, the machine is not allowed to move until returning to REF. point is completed. N74001 can be set to "NO" when failure to return to machine origin is caused by home switch fault. When encoder feedback function is enabled, it is suggested to set N74002 to "NO", since the system will correct the position automatically after E-stop by synchronizing the data in the system with the actual position of the machine tool, but N74000 is still suggested to set to "YES".			

◆ **Related Parameters in the Process of Backing to Machine Origin (N74090 under "Operation", others under "Axis Parameter")**

Parameter		Definition	Setting Range
N74090	Home Latch Count	Times of fine positioning in returning to machine origin, with default setting of "1"	1~100
N74010	Home Offset	Machine coordinate of machine origin, with default setting of "0"	0 ~ Upper limit of workbench stroke
N74020	Home Search Dir	The moving direction of machine at any point towards home switch	-1: Negative direction 1: Positive direction
N74030	Home Search Velocity	Moving speed of machine towards home switch (coarse positioning speed)	0.001 ~ 10000
N74050	Home Latch Dir	The moving direction of machine at any point towards encoder zero	1: Positive direction -1: Negative direction
N74060	Home Latch Velocity	Moving speed of machine towards encoder zero (fine positioning speed)	0.001 ~ 10000
N74080	Back Off Distance	The additional moving distance after the end of fine positioning in returning to	-1000 ~ 1000

Parameter		Definition	Setting Range
		machine origin, i.e. retract distance to move away from signal sensitive zone.	

When parameter N11304 “Encoder Feedback” is set as “NO”, parameter N74090 is valid while parameters N11110 and N11160 are invalid.
When parameter N11304 “Encoder Feedback” is set as “YES”, parameter N74090 is invalid while parameters N11110 and N11160 are valid.
In order to establish a machine coordinate system (MCS) correctly for machining, at machine start-up, generally returning to reference point will be executed automatically or manually, i.e. the machine tool will return to its measuring beginning (X, Y, Z=0) to establish the machine coordinate system. Machine reference point can be coincident with machine origin (in default system setting), or not, and the distance between reference point and machine origin can be specified by parameter N74010.
When home switches work normally, if the spindle moves away from home switch direction in the process of returning to machine origin (homing), the value of N74020 (coarse positioning direction), opposite to that in fine positioning stage, should be modified, please refer to question No. 2 in section 3.6.6 when the moving direction of machine is incorrect during backing to machine origin. If the speed of returning to machine origin is too low, properly adjust the value of N74030 (coarse positioning speed). “Back Off Distance” refers to a certain moving distance away from REF. point to leave the signal sensitive zone of home switches after backing to machine origin completed.

◆ **Related Parameters to Detect Distance between Coarse and Fine Positioning Switches (N74120 under “Operation”, others under “Axis Parameter”)**

Parameter		Definition	Setting Range
N74100	Leadscrew Pitch	For analysis of switch distance of fine and coarse positioning in backing to machine origin	0 ~ 360
N74110	Coarse/ Fine Switches Min Dist	To detect whether the switches of fine/coarse positioning are too close in backing to machine origin	0 ~ One half of screw pitch
N74120	Coarse/Fine Pos Distance Tolerance	The allowable error range by comparison of current result of backing to machine origin with history average record	0 ~ 100

Too close distance between home switch and encoder zero switch may lead to deviation of one screw pitch in REF. point positioning during returning to machine origin, and the system will check if this offset is reasonable or not via parameter N74110 with setting range of “0 ~ one half of screw pitch” (unit: mm). The normal range of switch distance between fine and coarse positioning is (0+ value of N74110, screw pitch- value of N74110), and the switch distance will be automatically measured in returning to origin. An alarm of “ The distance of coarse/fine positioning switch for Z axis was too close ” will occur if the distance is out of the above range. To remove this error, adjust the home switch position or check if the setting of parameter N74110 is reasonable or not.
With comparison between current measured value and history average value, the percentage of

Parameter	Definition	Setting Range
"(current measured value – history average value) / history average value" should be within the setting value of N74120; if not, this measured value is invalid. And the system will prompt alarm of "⚠ The result error of returning machine home for Z axis was out of range". Click the shortcut key N "Clear History Records(N)" to clear the measured history record after changing the home switch. Related to the specific machine tool, N74100 should be set after measured in actual operation.		

3.6.6 FAQ & Troubleshooting

1) REF. point signal cannot be detected in the process of returning to machine origin.

It is generally caused by home switch fault. The adjusting & debugging steps are as shown in Fig. 3-22.

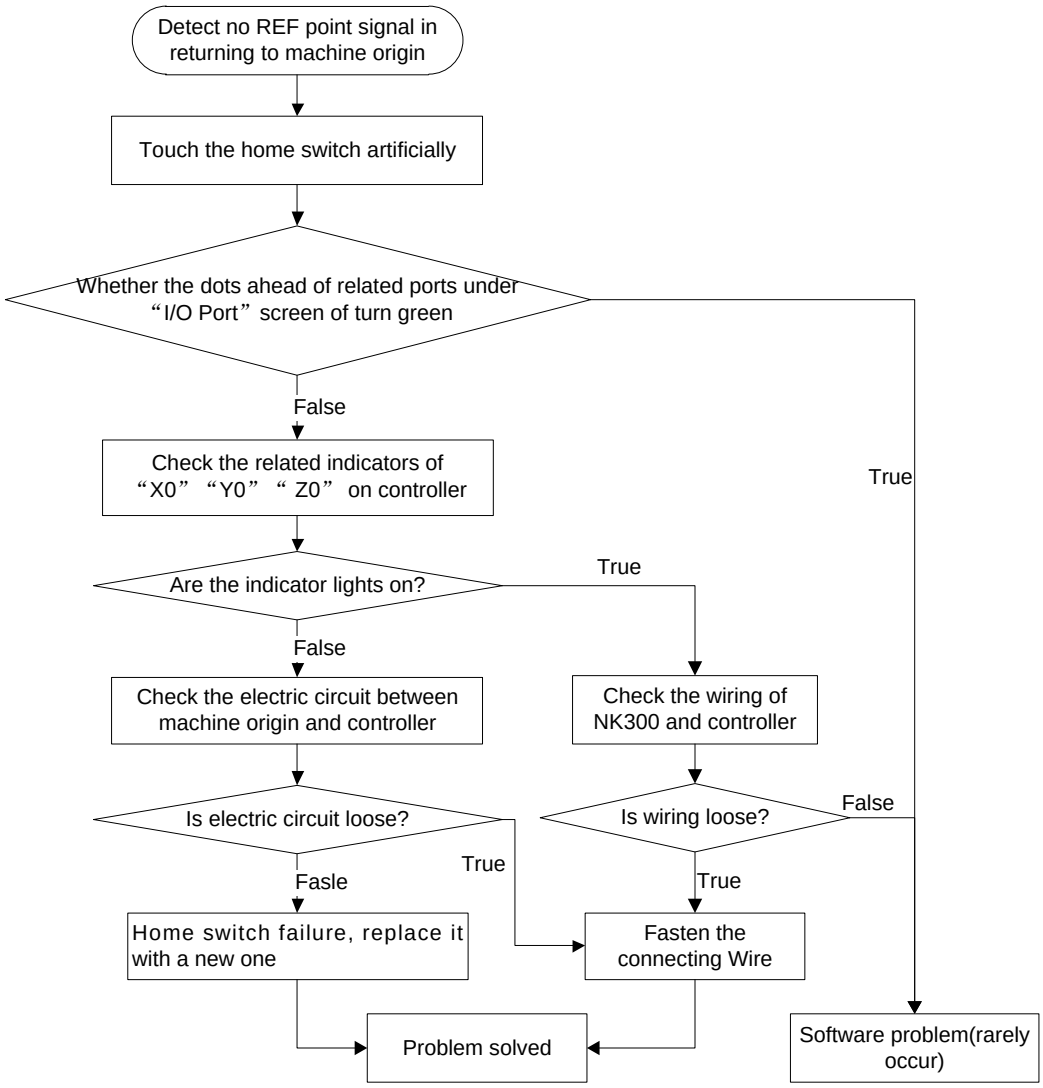


Fig. 3-22 Debugging steps

2) Incorrect motion direction of machine in returning to machine origin may be caused by the following reasons:

- Incorrect polarity of REF. point signal: when the home switch is normally open, the polarity is “NO”; when normally closed, the polarity should be “NC”.
- Incorrect parameter setting: check the parameter N74020 “Home Search Dir”, and adjust the related parameters.

3) Too slow coarse positioning speed in returning to machine origin may be caused by the below reasons:

- The setting value of N74030 “Home Search Velocity” is too small.
- The polarity setting of REF. point signal in the software is mismatching with the home switch type. If a NC-type home switch is adopted and the polarity of REF. point signal is NO, the REF. point signal is valid at beginning of backing to machine origin, so the machine will slowly move away from machine origin at the speed of fine positioning.

4) The distance between fine and coarse positioning is out of normal range, the system prompting an alarm “ The distance of coarse/fine positioning switch for Z axis was tool close ”, which may be caused by too close switch distance between fine and coarse positioning, so the actual position of home switch and encoder zero should be readjusted to make the distance within the range of (0+ value of N74110, screw pitch- value of N74110).

5) The distance between fine and coarse positioning is out of the allowable error range, the system prompting an alarm “ The result error of returning machine home for Z axis was out of range ”, with possible causes as below:

- The accuracy error of home switch: check home switch precision.
- The accuracy error of encoder zero: check whether encoder zero signal in the system is correct or not.
- After a home switch is reinstalled, the detecting environment changes in returning to machine origin: press the shortcut key N to clear the history record of measurement.

3.7 Spindle Parameter Adjustment

3.7.1 Spindle Speed Setting

In auto mode, press  to enter the interface as shown in Fig. 3-23, in which spindle speed can be set directly.

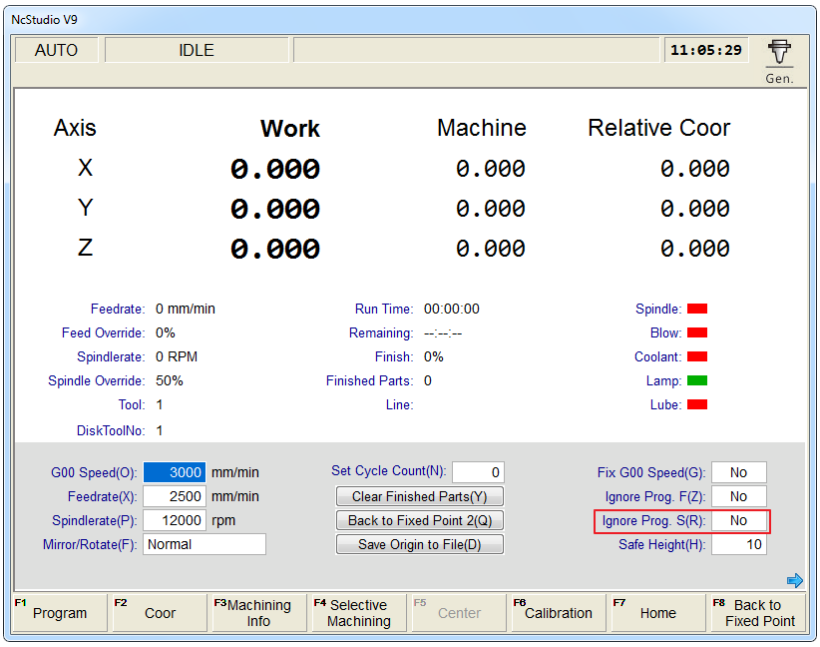


Fig. 3-23 Spindle speed setting on classic interface

Spindle speed can be directly set in the parameter setting area above the manipulation button bar. When parameter N72002 “Ignore Programmed Spindlerate” is set to “YES”, spindle speed in auto machining will adopt the system setting value, i.e. the value of “Prog.S”; when set to “NO”, spindle speed in auto machining will adopt the specified spindle speed in the machining file.

There are two ways for changing parameters under [Coordinate-auto screen]:

1) Press “↑”, “↓”, “→” or “←” to move the cursor onto the desired parameter, and then press “Enter” to eject an input box.

2) Press the corresponding shortcut key to eject an input box. For instance, for “Spindlerate(P)”, pressing the letter key “P” on the operation panel will eject an input box for entering a value.

CAUTION

Concrete setting methods on the new operation interface are the same with that on classic interface; however, shortcut letter next to each parameter may be different on two interfaces. The counterpart on the new interface is as shown in Fig. 3-24.

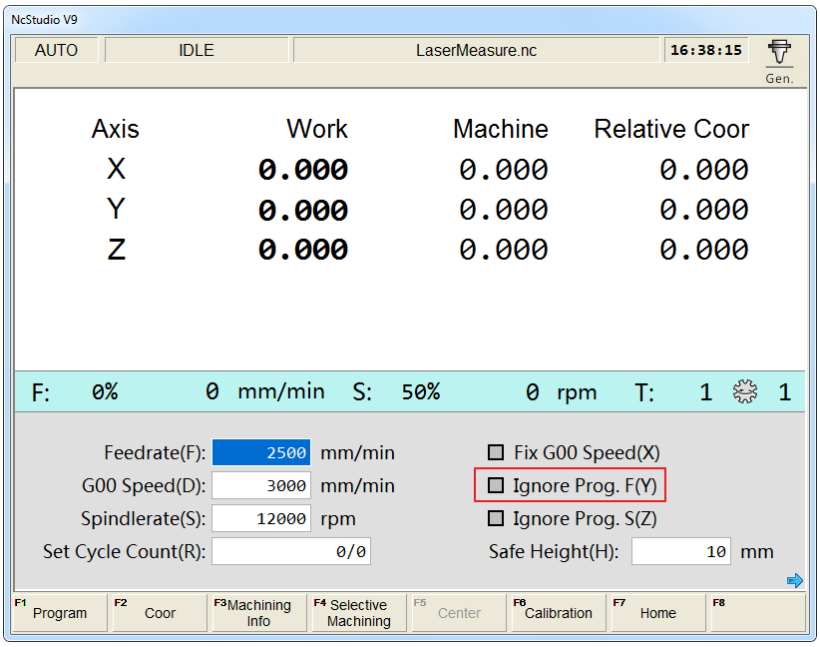


Fig. 3-24 Spindle speed setting on the new interface

Spindle speed can be controlled by adjusting spindle override. Their relationship is as following:

$$\text{Current Spindle Speed} = \text{Spindle Speed} \times \text{Current Spindle Override}$$

Spindle override knob is on the operation panel, as shown in Fig. 3-25.

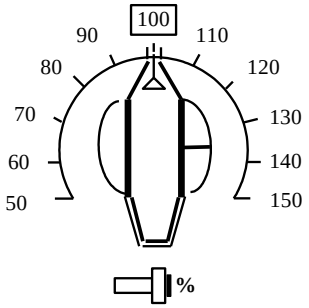


Fig. 3-25 Spindle override knob

The least unit of ruler of spindle override is 10% (10% for each scale), with setting range of spindle override “50% ~ 150%”.

◆ **Related Parameters**

Parameter		Definition	Setting Range
N20001	Max Spindle Speed	The max. allowable rotation speed of spindle (matched with the inverter setting)	0~ 999999
N20005	SpindleCool Off Delay Time	Delay time of closing spindle cooling pump after spindle stop	0~ 600
N20010	Spindle On Delay Time	Delay time after spindle receiving “start” or “stop” command	0~ 60
N20011	Spindle Off Delay	Delay time after spindle receiving “start” or	0~ 60

Parameter		Definition	Setting Range
	Time	"stop" command	
The value of "Spindlerate" under [Coordinate-auto] must be less than that of N20001; the max. setting value of N20001 is corresponding to analog SVC 10V; when the inverter reaches the max. voltage 10V, the corresponding rotary speed of inverter is the max. spindle speed, i.e. the value of N20001. $\text{Real-time Voltage of Analog SVC} = \frac{\text{Current Spindle Speed}}{\text{N20001}} \times 10\text{V}$ Parameter N200010 and N20011 set the delay time of spindle on/ off, due to a certain time is needed before spindle reaches rated rotary speed since start-up or stops until reaching zero speed; if machining begins before the machine reaching rated rotary speed or other operation is executed before spindle stops completely, it's possible to damage the tool or produce a scrap.			

◆ **Related Parameters**

Parameter		Definition	Setting Range
N72004	Spindle Off when Cycle Stop	Whether spindle will automatically stop when machining stops regularly	YES: Stop NO: Not stop
N72008	Spindle On when Cycle Start	Whether spindle will automatically rotate when machining begins	YES: Start NO: Not start
N73005	Stop Spindle on Pause	Whether spindle will automatically stop when machining pauses	YES: Stop NO: Not stop
This group of parameters sets the spindle action when commands of machining stop/ start/ pause are executed.			

3.8 Tool Measurement

The process of tool measurement refers to the process of establishing the concrete position of workpiece coordinate system (WCS) in the machine coordinate system (MCS).

When the parameter N11304 "Encoder Feedback" is set to "Yes", tool measurement with encoder feedback function will be used; while it is set to "No", tool measurement without encoder feedback (or the traditional one) will be used.

With the help of a tool presetter, tool measurement is realized. As shown in Fig. 3-26, there are ports on the controller corresponding to CUT and COM on the tool presetter. If necessary, such port as "Over-travel Protection" can be added on the controller according to customers' needs. According to the different installation positions of a tool presetter, tool measurement is divided into mobile calibration and fixed calibration, first calibration and calibration after tool change.

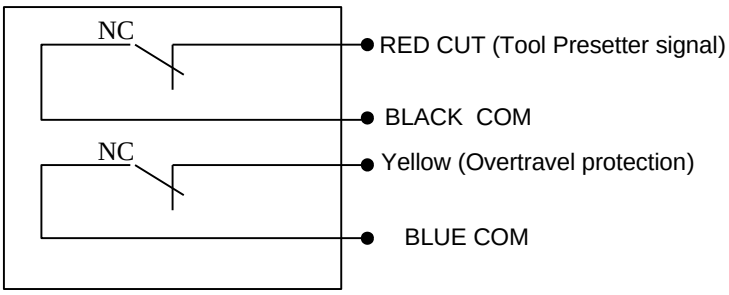


Fig. 3-26 Electrical wiring diagram of a WEIHONG tool presetter

Fig. 3-27 is the sketch map of tool calibration using of a tool presetter.

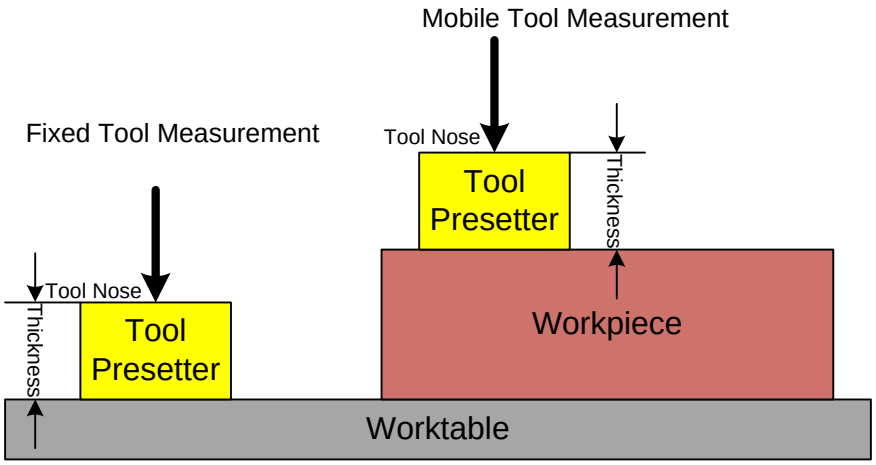


Fig. 3-27 Sketch map of using a tool presetter

3.8.1 Software Interface

Press  to access the [State] function area, and then press F6 “Calibration” to enter the tool measurement interface, as shown in Fig. 3-28. Pressing a shortcut key will select the corresponding measurement type under this interface.

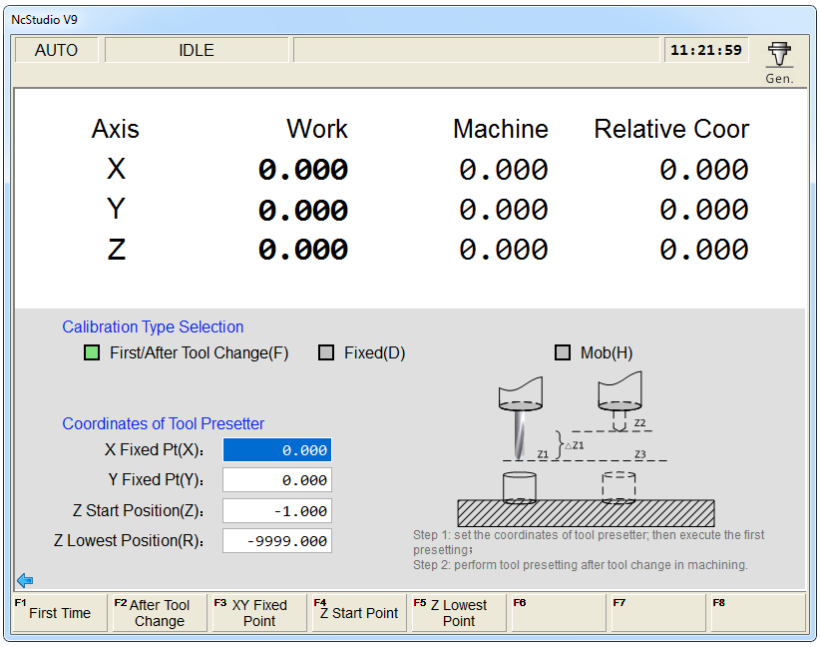


Fig. 3-28 Sub-screen of tool measurement

3.8.2 First/After Tool Change

Press  ==> F6 “Calibration” to access the tool measurement interface, and press letter key F to activate the first calibration/calibration after tool change. See Fig. 3-29.

Before calibration begins, manually move the spindle and make the tool nose onto the tool presetter, press F3 “XY Fixed Point”. The system will automatically record the current coordinate of X and Y axis to the fixed point in X and Y directions. According to actual situation, move Z axis onto a reasonable height above the presetter, and press F4 “Z Start Point” to set the beginning coordinate of fixed calibration.

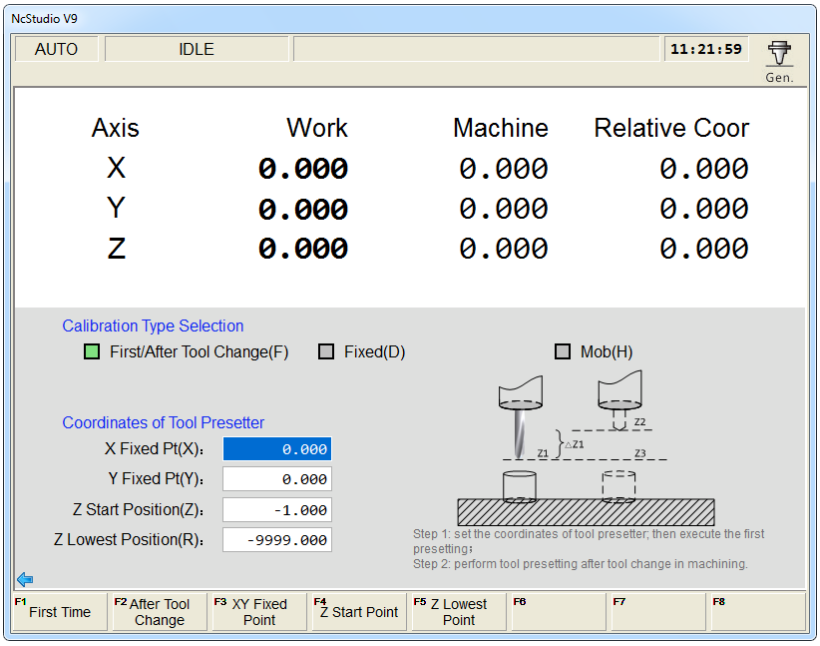


Fig. 3-29 First calibration/calibration after tool change

The operation steps are as below:

1) Manually move Z axis to workpiece surface, and then confirm workpiece origin by mobile calibration or manual clear (the method for manual clear: pressing  ==> F2 [Coor] ==> F2 “Clear” ==> F3 “Clear Z”).

2) Pressing  ==> F6 [Calibration] ==> letter key F to activate this calibration method. Press F1 “First Time”, and the system will automatically record the current machine coordinate of Z axis. The process is as shown below, and automatically completed by the system.

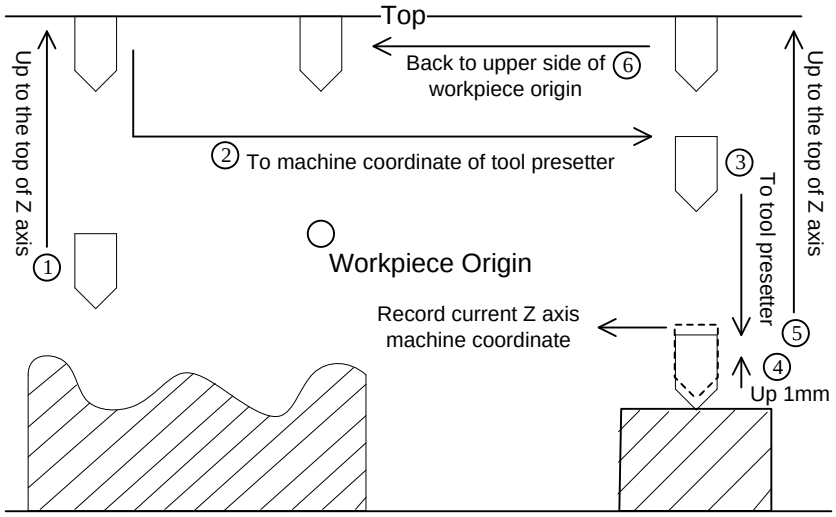


Fig. 3-30 First tool measurement

3) Start machining after first tool calibration completed.

4) After tool change or tool break, press F2 “After Tool Change” in Fig. 3-31 to execute calibration after tool change. The process is as shown below, and automatically completed by the system.

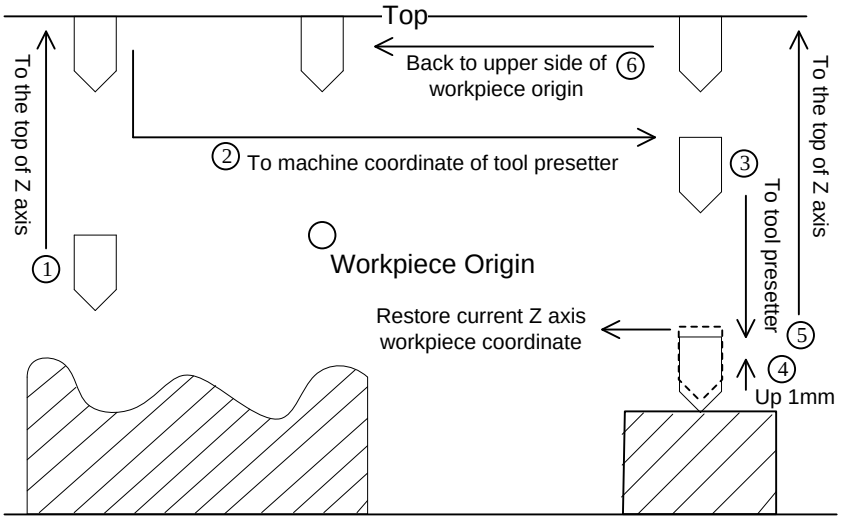


Fig. 3-31 Measurement after tool change

5) Start machining after calibration after tool change completed.

3.8.3 Fixed Calibration

Fixed calibration refers to the measurement operation on a certain fixed position of a machine tool due to tool damage or other causes, frequently used in multi-tool mode. The length of a tool and the clamping position may vary, thus tool offset should be reconfirmed by fixed tool calibration. See Fig. 3-32 and Fig. 3-33 for the sketch map of fixed tool calibration.

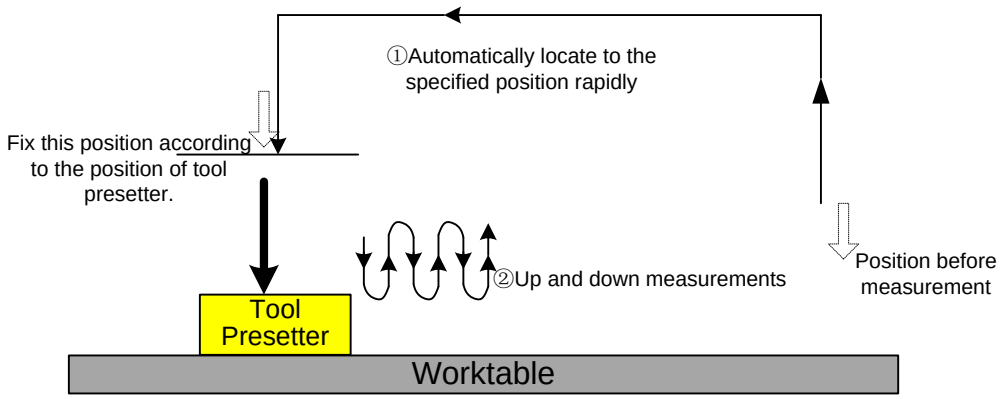


Fig. 3-32 The process of fixed calibration without encoder feedback function

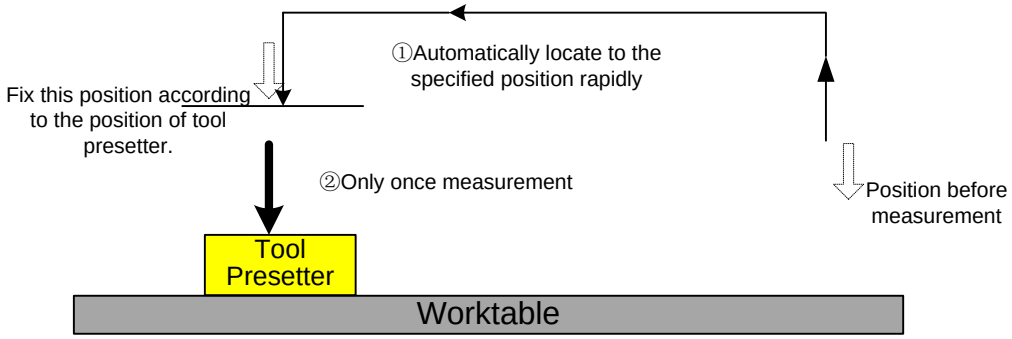


Fig. 3-33 The process of fixed measurement with encoder feedback function

The process of fixed calibration records the machine coordinate when the tool nose touches the surface of the tool presetter. Tool offset is the thickness of the tool presetter subtracted from the recorded machine coordinate.

Tool Offset= Machine Coordinate – Thickness of Tool Presetter

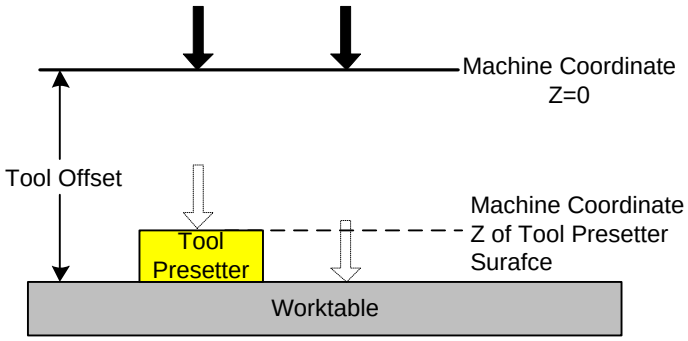


Fig. 3-34 The sketch map of tool offset

◆ **How to Set the Position of Fixed Tool Presetter**

Press  ==> F6 “Calibration” to access the tool measurement interface, and press letter key D to activate the fixed calibration.

Before calibration begins, manually move the spindle and make the tool nose onto the tool presetter, press F5 “XY Fixed Point”. The system will automatically record the current coordinate of X and Y axis to the fixed point in X and Y directions. According to actual situation, move Z axis onto a reasonable height above the presetter, and press F6 “Z Start Point” to set the beginning coordinate of fixed calibration.

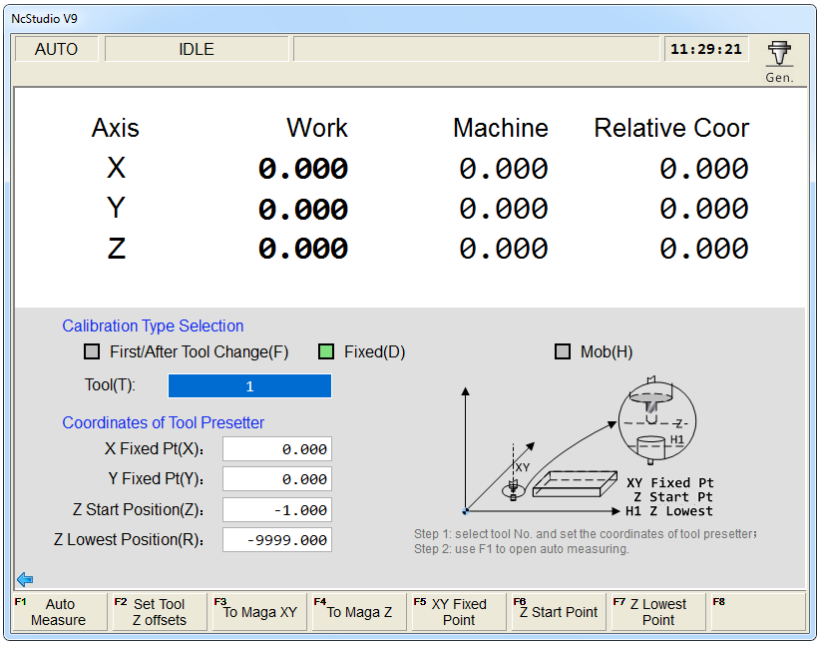


Fig. 3-35 Fixed calibration sub-screen

The steps of fixed calibration are as below:

1) Select a tool according to tool No.;
2) Execute fixed calibration to the selected tool and record the tool offset. When tool presetter is available, press F1 “Auto Measure” to enable auto calibration; while tool presetter is absent in this step, you can press F2 “Set Tool Z Offsets” to manually set the tool offset in Z axis.
3) Record tool offset values.
4) Execute step 1 and 2 to each tool;
5) Select any tool to move to workpiece surface for clearing.

CAUTION

1) Only when linear tool magazine is used, pressing F3 “To Maga XY” in Fig. 3 36 will make the current X and Y machine coordinates the X and Y machine coordinates of the spindle tool in linear magazine. And pressing F4 “To Maga Z” will make the current Z machine coordinate the Z machine coordinate of the spindle tool in linear magazine. Please note that it is only available for linear magazine.

2) Before pressing F3 “To Maga XY” or F4 “To Maga Z”, please make sure that the tool number in the spindle is the same as that tool number setting.

◆ **Related Parameters**

Parameter		Definition	Setting Range
N75201	Fixed Presetter Surface to WCS Z0	The distance between tool presetter surface and worktable surface in fixed measurement	
N75203	Fixed Preset Speed	Moving speed from the top point to the initial height in fixed measurement	
N75025	Enable Preset Overtravel Alarm	Whether to enable over-travel protection in tool measurement	YES: enabled NO: disabled
The measurement method for parameter N75201 is as below: 1) Manually move Z-axis to a certain point over workpiece surface→ shift down the tool nose until reaching the surface of workpiece→ record the current coordinate Z1 of Z-axis. 2) Uplift Z-axis→ put the fixed tool presetter on workpiece surface→ shift down Z-axis slowly until touching the presetter and getting the tool presetter signal→ record the current coordinate Z2 3) Z2- Z1, and the result equals to the thickness of the tool presetter. Manually enter this result into parameter N75201.			

For other related parameters in fixed measurement, such as N75001, N75002, N75020, N10050 and N10055, refer to section 3.6.1.

3.8.4 Mobile Tool Measurement

Press  ==> F6 “Calibration” to access the tool measurement interface, and then press the letter key “H” to select “Mob” tool measurement, namely, mobile calibration.

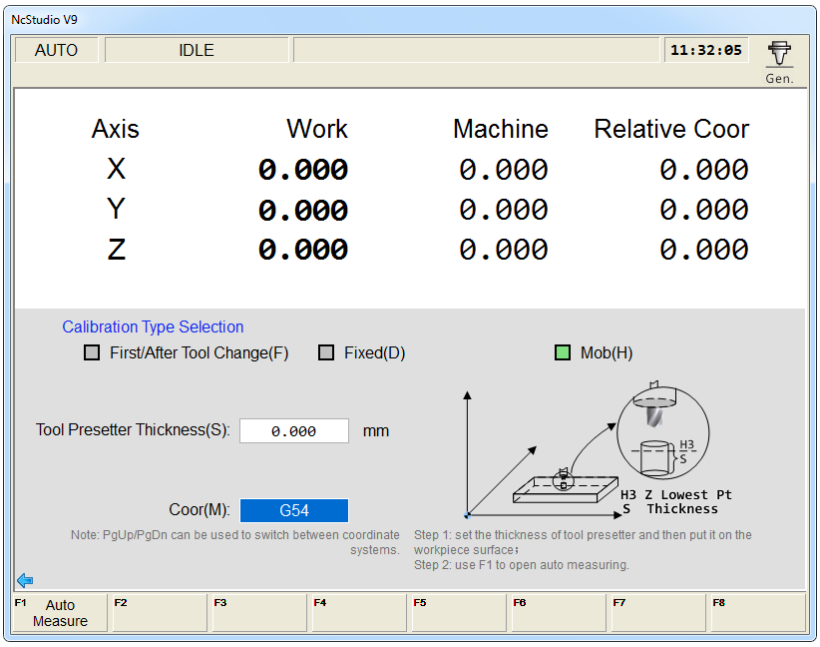


Fig. 3-36 Mobile Calibration

Mobile tool calibration can be used to set workpiece origin of Z-axis by executing calibration at the current position, the thickness of the tool presetter decided by parameter N75100. After mobile tool calibration, the system will automatically set workpiece offset.

Workpiece Offset＝Machine Coordinate -Thickness of Tool Presetter - Public Offset - Tool Offset

Generally, the default setting values of public offset and tool offset are both “0”.

See Fig. 3-37 and Fig. 3-38 for the sketch map of the process of mobile tool calibration.

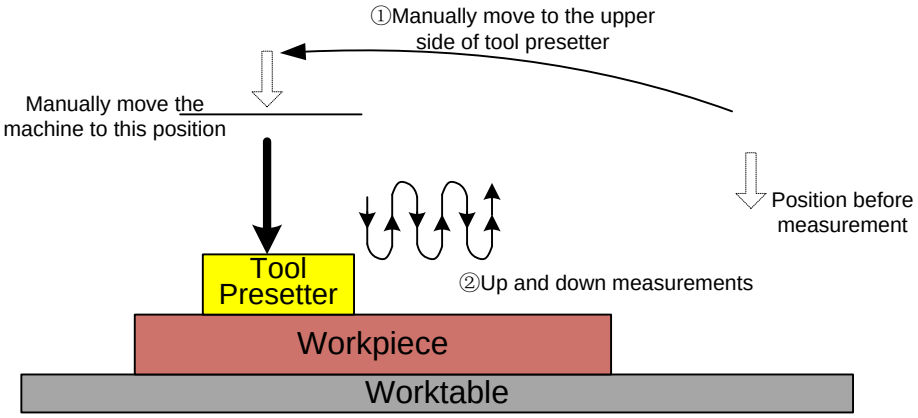


Fig. 3-37 The process of mobile calibration without encoder feedback function

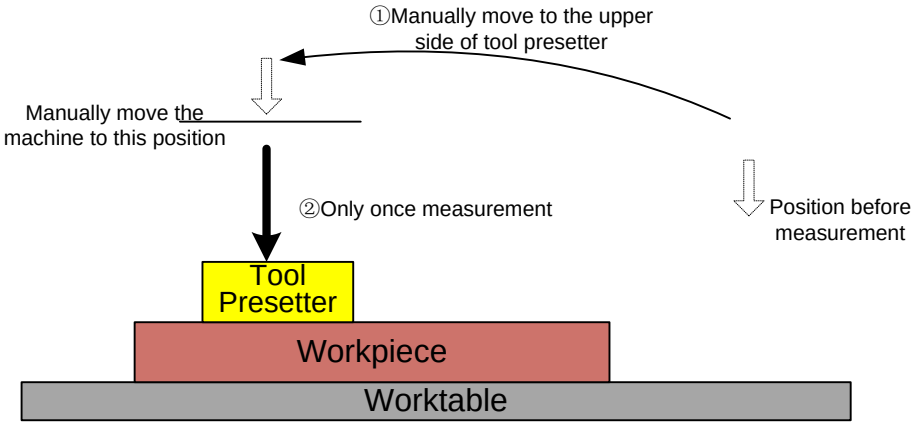


Fig. 3-38 The process of mobile calibration with encoder feedback function

◆ **Related Parameters**

Parameter		Definition	Setting Range
N75100	Mobile Surface to WCS Z0 Distance	Height difference from top surface of a tool presetter to its bottom	-1000 ~1000
The measurement method of this parameter is as follows: 1) Manually move Z-axis to a certain point over workpiece surface→ shift down the tool nose until reaching workpiece surface→ record the current coordinate of Z-axis (Z1). 2) Uplift Z-axis→ put a tool presetter on workpiece surface→ shift down Z-axis slowly until reaching the presetter and getting the tool presetter signal→ record the current coordinate of Z-axis (Z2). 3) Z2- Z1, and its result equals to the thickness of the tool presetter. Manually enter this result into parameter N75100.			

◆ **Related Parameters (N10050 and N10055 under "Axis Parameter")**

Parameter		Definition	Setting Range
N75001	Fine Preset Speed	Tool speed when approaching the presetter surface in tool measurement	-
N75002	Fine Preset Count	The times of repeated up & down measurements after receiving tool presetter signal when the tool approaches the presetter surface in tool measurement	-
N75020	Preset Result Tolerance	The max. allowable error value of tool measurement in multiple tool measurements	0 ~ 10
N10050	Change Tool Workbench Range Upper Limit	Machine coordinate of upper limit of worktable range in tool measurement	-99999~ 99999
N10055	Change Tool Workbench Range	Machine coordinate of lower limit of worktable range in tool measurement	-99999~ 99999

Parameter		Definition	Setting Range
	Lower Limit		
N10060	Enable ToolMeas. Travel limits	-	Yes: enabled; No: disabled
Parameter N75020 refers to the max. allowable error of tool measurement set in the system, relative to the average error value of repeated tool measurements in the process of measurement; if average error value is less than N75020, tool measurement succeeds, or measurement fails.			

3.9 Offset Setting of WCS

3.9.1 WCS (Workpiece Coordinate System)

In programming, programmers select a certain given point on workpiece as origin (also called program 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 mobile relative to machine origin. The selection of origin of WCS should meet the conditions of simple programming, simple dimensional conversion, and small caused machining error, etc.

The corresponding coordinate systems of work offset are G55, G56, G57, G58, G59 and G54 (the default coordinate system). And the relationship of work offset and machine coordinate system is as shown in Fig. 3-39.

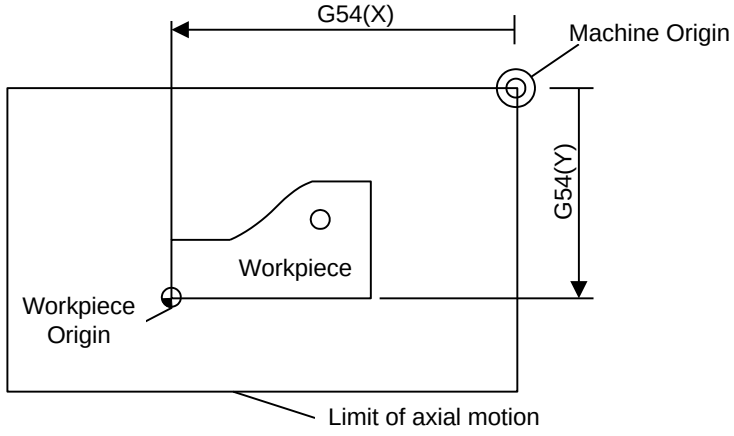


Fig. 3-39 The relationship of work offset and machine coordinate system

One, two or multi-work offset can be used in machining program. As shown in Fig. 3-40, if three workpieces are installed on the worktable, 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 workpieces, with calculation height Z-0.14.

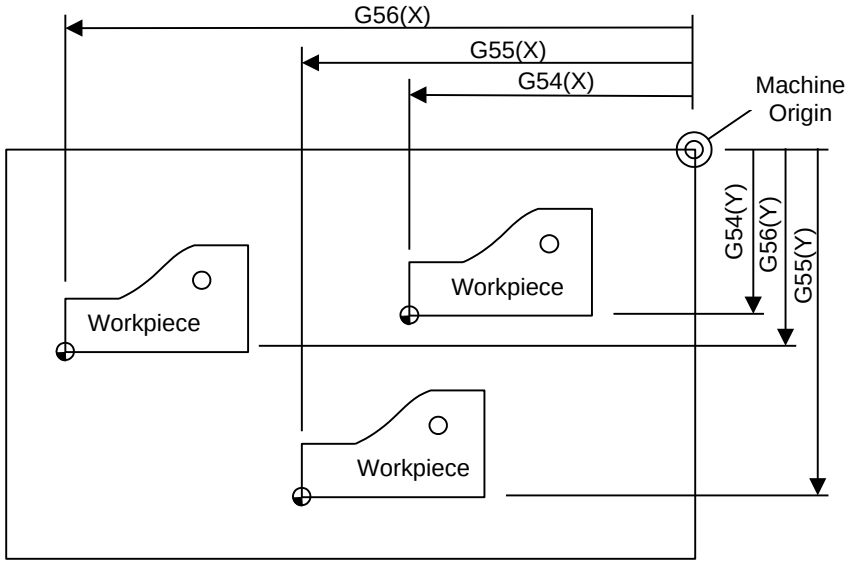


Fig. 3-40 Sketch map

```
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
…
```

Program segments N3 ~ N5, within WCS of G54, are 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 WCSs, public offset is used for adjusting workpiece origin of X-, Y-, and Z-axis, but will not change the offset value of “G54 ~G59”.

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

Workpiece Coordinate= Machine Coordinate – Work Offset - Tool Offset – Public Offset

3.9.2 Extended Coordinate System

With up to 120 extended coordinate systems (also known as additional coordinate systems) provided, the total number of WCS is 126 (6+120) in NK300BX system. 126 work offsets are optional in programming. The extended coordinate systems are the extension for G54, from G54P0 to G54P119. To view or change the setting of these systems, “PgUp” and “PgDn” are used for page turning while “Home” and “End” for page heading and page footing.

Command G54 Px: Select an extended coordinate system, and “x” here refers to a number within [0,

119].

Example:

G54 P0	Select extended coordinate system 1
G54 P1	Select extended coordinate system 2
G54 P2	Select extended coordinate system 3
G54 Px	Select extended coordinate system (x + 1)
G54 P119	Select extended coordinate system 120

◆ **Related Parameters**

Parameter		Definition	Setting Range
N80002	Support Extension Workpiece Offset	To display the 120 groups of extended WCS (G54 P0~G54 P119) on the [Coordinate Management Interface] screen or not.	YES: Support NO: Not support
The default parameter setting is “NO”. When above 6 workpieces are clamped on a worktable, set this parameter to “YES” to support the extended coordinate systems of work offset, so as to save multi-group of work offsets, which is user-friendly.			

3.9.3 Software Interface

Press  ==> F2 “Coor” to access coordinate system management interface, as shown in Fig. 3-41. This interface displays currently being edited WCS and its corresponding work offset and public offset.

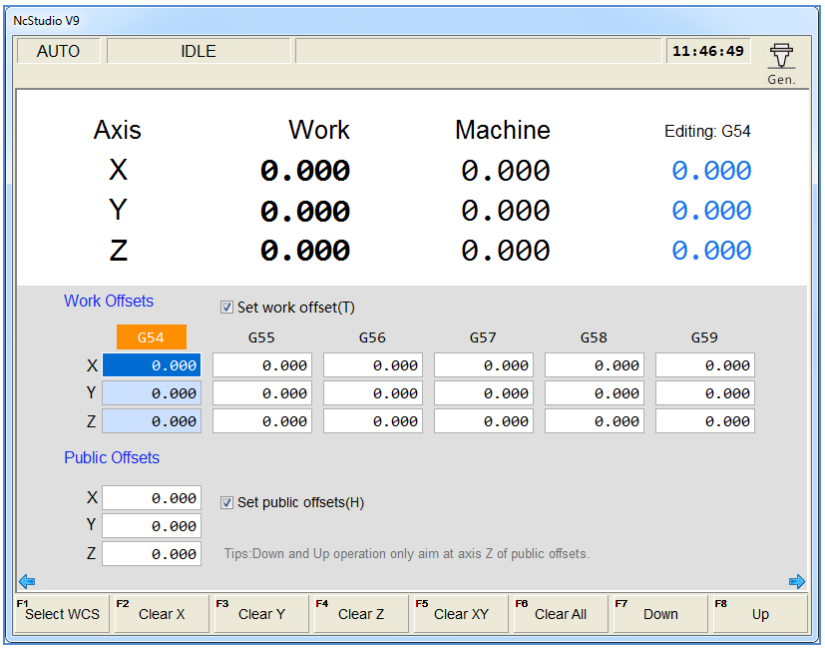


Fig. 3-41 Coordinate system management interface

Press the arrow keys to move cursor onto the work offset, or to public offset, and then press the Enter key to modify workpiece offset or public offset. Or you can press F7 “Down” or F8 “Up” to change the

public offset of Z axis.

◆ **Select WCS**

Press arrow keys "←" or "→" to move cursor to the desired WCS, and then press F1 "Select WCS" to set the currently being edited coordinate system as current WCS.

◆ **Clear X, Clear Y, Clear Z**

As shown in Fig. 3-41, Pressing F2 "Clear" to open the secondary manipulation buttons bar. Active F1~F6 buttons correspond to "Clear X/Y/Z/A/XY/All" respectively. Pressing F1~F3 will respectively set the value of current machine coordinate to X/Y/Z work offset in the current WCS, while the corresponding machine coordinate will not change.

◆ **Clear XY**

As said above, pressing shortcut key F5 in the secondary buttons bar will set the value of current machine coordinates to XY work offsets in the current WCS as, while Z work offset will remain the same.

◆ **Clear All**

Pressing shortcut key F6 in the secondary buttons bar will set XYZ work offsets in the current WCS as the value of current machine coordinates.

◆ **Down**

In Fig. 3-41, press F7 "Down" to eject an input box→ input the adjusting value of Z feed → press Enter, Z-axis workpiece origin to move down specified distance.

◆ **Up**

In Fig. 3-41, press F8 "Up" to eject an input box→ input the adjusting value of Z feed → press Enter, Z-axis workpiece origin to move up specified distance.

Both "Up" and "Down" only modify public offset of Z-axis.

◆ **Save Workpiece Origin**

Press  to turn to the next button bar. Then press F1 in the new interface as shown in Fig. 3-42.

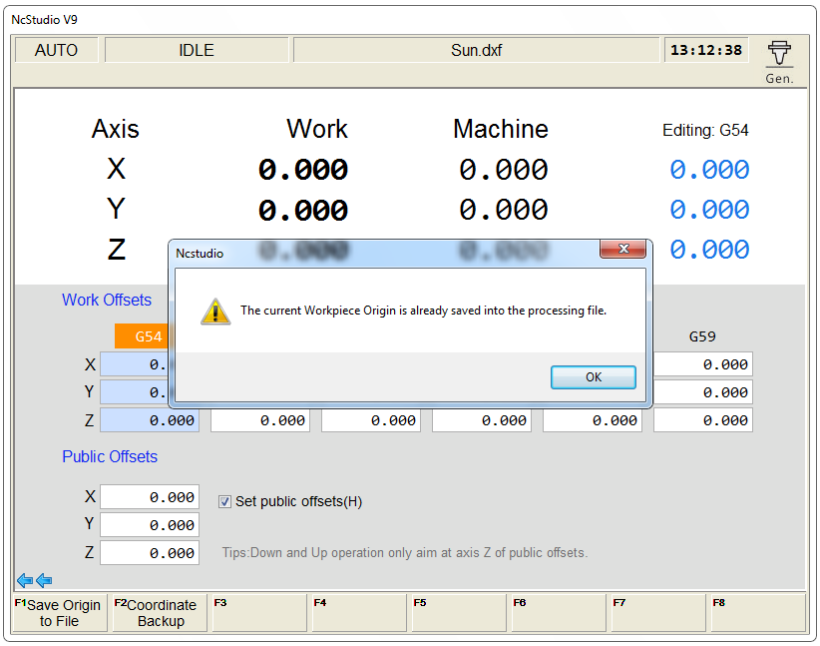


Fig. 3-42 Save Origin to File

Pressing F1 “Save Origin to File” will save the current workpiece origin to machining file, which can be called and used when next time machining file being loaded. As shown in Fig. 3-43, select “Yes” and the existing workpiece origin will be reloaded. Otherwise the origin will not be reloaded.

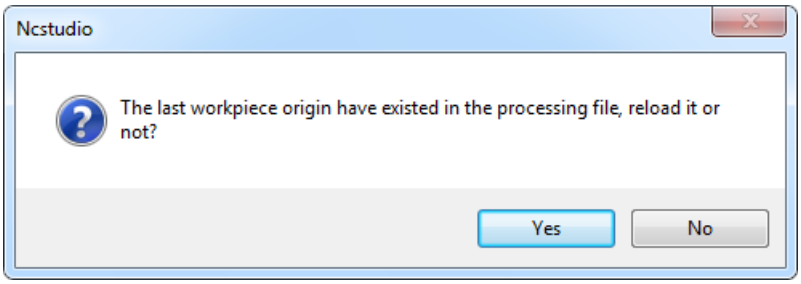


Fig. 3-43 Dialog box whether to reload existing origin

CAUTION

If you have not uninstalled the machining file after saving workzero to the file, the dialog box as shown in Fig. 3-43 will pop up when the software is restarted.

◆ Coordinate Backup

In Fig. 3-41, press  to turn to the next buttons bar. Pressing F2 “Coordinate Backup” to open coordinate backup sub-screen, as shown in Fig. 3-44. On this sub-screen, you can save the current workpiece offsets into the system.

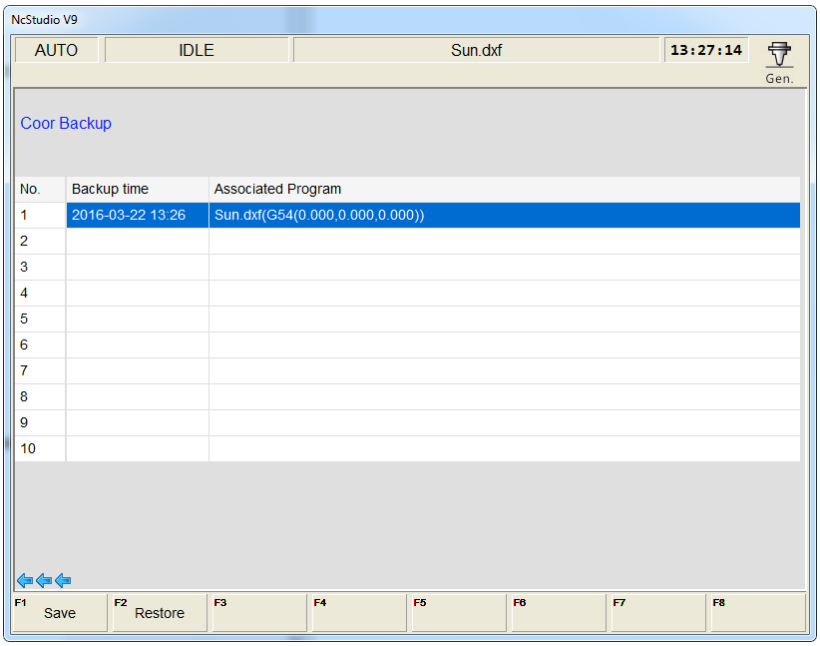


Fig. 3-44 Coordinate backup

With workpiece offsets saved before, anytime a new machining program file has been loaded, you can press arrow keys " ↑ " " ↓ " to select the desired offsets and press F2 "Restore" on the sub-screen to restore the offsets into the current WCS. Before restoring, a dialog box of prompt as below will show up.

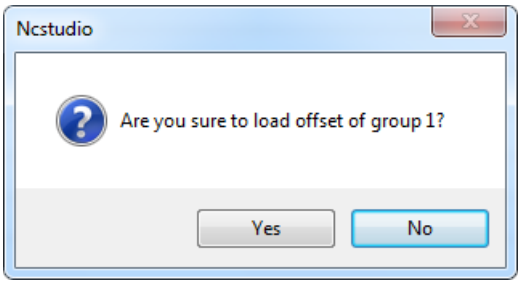


Fig. 3-45 Work offset restore prompt

Furthermore, after you choose to restore and load the offsets, a new dialog box will show up, asking whether to change Z offset too. As shown in Fig. 3-46. If "Yes", Z axis offset will be changed too, if "No", offsets of axes except Z axis will restore.

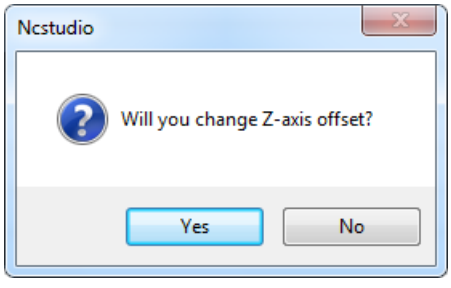


Fig. 3-46 Prompt to change Z offset or not

3.10 Centering

Centering is a way to find the center point on a part and make it the part zero (or workpiece origin). The system supports manual and auto centering. In auto centering, the spindle does not rotate. Manual centering is divided into “Manual Center (two-point centering) and “Circle Center”. An edge finder can be used for accurate centering. Centering operation is only available in manual mode or MPG mode, and on most occasions, MPG is used.

When the parameter “N81004 Allow Spindle-On when centering” is set to “YES”, before executing manual centering, press “F7 ENBL EdgeFinder” to make it turn to orange and start spindle, spindle speed decided by the parameter “N20006 Spindle Speed when Centering”, whose value is 500 by default and should not be set too large.

When the parameter “N81004 Allow Spindle-On when centering” is set to “NO”, “F7 ENBL EdgeFinder” is not available. To turn on spindle, press “Spindle CW” or “Spindle CCW” at spindle speed set in the software.

3.10.1 Manual Center

Manual center, i.e. two-point centering, refers to the process of locating the midpoint of a line connected by two points, mainly used for locating the center of a blank.

In manual mode, press  ==> F5 “Center” to access the centering interface, as shown in Fig. 3-47. In this interface, press the letter key “F” to select “Manual Center(F)”.

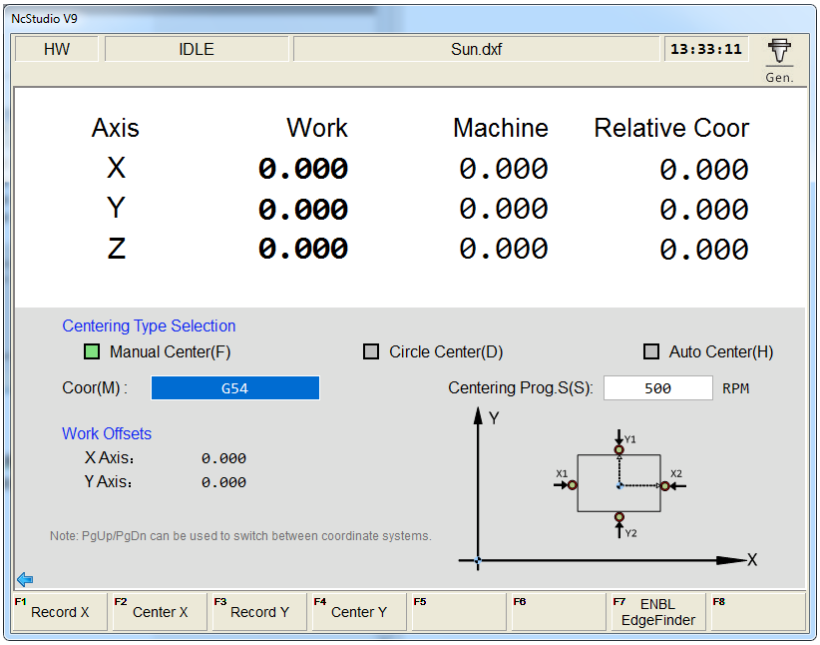


Fig. 3-47 Centering interface---Manual Center

The operation steps of manual centering are as below (An example of X-axis):

1) With the help of MPG usually, manually move the cutter to one side of workpiece, and then press F1

[Record X] to record the machine coordinate of current point.

2) Move the cutter to the other side of workpiece, and then press F2 [Center X] to calculate the midpoint coordinate based on the coordinate of current position and last recorded value and set it as workpiece origin.

CAUTION

In the process of centering of a certain axis, the other coordinate axis should keep motionless.

3.10.2 Circle Center

Circle center, i.e. three-point centering, means automatic calculation of center point coordinates (generally set as workpiece origin) of a circular blank in terms of the three recorded circle coordinates.

In manual mode, press  ==> F5 “Center” to access the centering interface. Press the letter key “D” to select “Circle Center (D)”, as shown in Fig. 3-49.

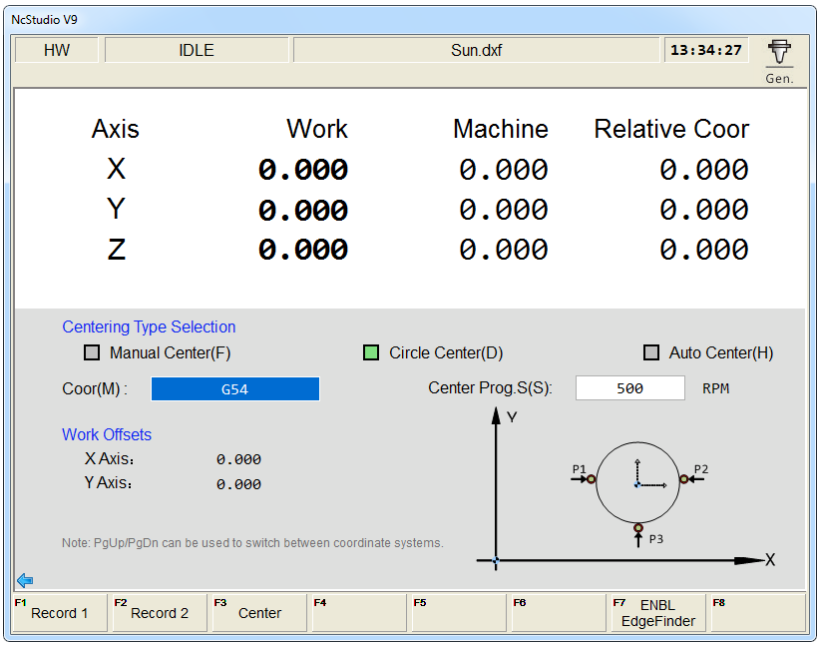


Fig. 3-48 Circle centering interface

The steps of circle centering are as below:

1) Manually move the cutter to one point on the circumference of a circular blank, and then press F1 [Record 1] to record the machine coordinates of current point as the first group of coordinate;

2) Move the cutter to another point on the circumference, and then press F2 [Record 2] to record the machine coordinates of current point as the second group of coordinate;

3) Move the cutter to the third point on the circumference, and then press F3 [Center] to calculate the circle center coordinates and set it as workpiece origin based on the current machine coordinates and the two groups of coordinate recorded previously.

3.10.3 Auto Center

In manual mode, press  ==> F5 “Center” to access the centering interface. Press the letter key “H” to select “Auto Center (H)”, as shown in Fig. 3-49.

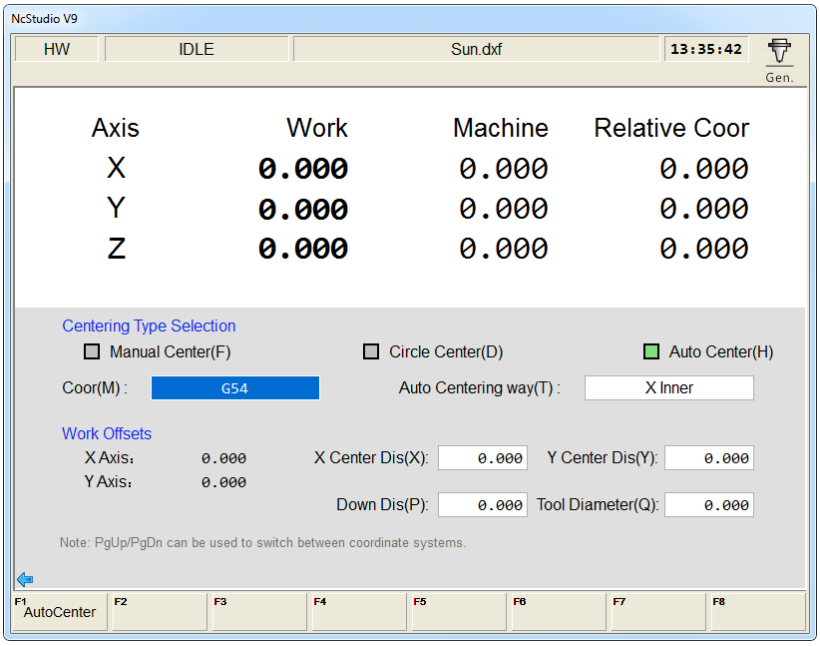


Fig. 3-49 Auto centering sub-screen

On auto centering screen, you can press letter key M to open a input dialog box, where you can set the WCSs, or you can directly press PageUp/PageDown keys to switch among the options.

Pressing letter key T can choose the auto centering way. There are altogether 10 types of auto centering, including “X Positive” “X Negative” “Y Positive” “Y Negative” “X Inner” “X Outer” “Y Inner” “Y Outer” “XY Inner” and “XY Outer”, which will be specifically introduced later.

- X Center Dis: the distance from midpoint of workpiece to X boundary. It must be a little larger than its actual value in outer center, while a litter smaller in inner center.
- Y Center Dis: the distance from midpoint of workpiece to Y boundary. It must be a little larger than its actual value in outer center, while a little smaller in inner center.
- Down Dis: tool down / up distance in tool measurement; in inner center, it must be smaller than the distance from the tool nose to workpiece surface, while larger in outer center.
- Tool Diameter: actual diameter of a tool.

The system offers three kinds of auto centering— inner center, outer center and boundary center (positive/ negative), switched by pressing the letter key “T” or clicking on the input box. Taking X-axis as an example:

◆ X Outer Center

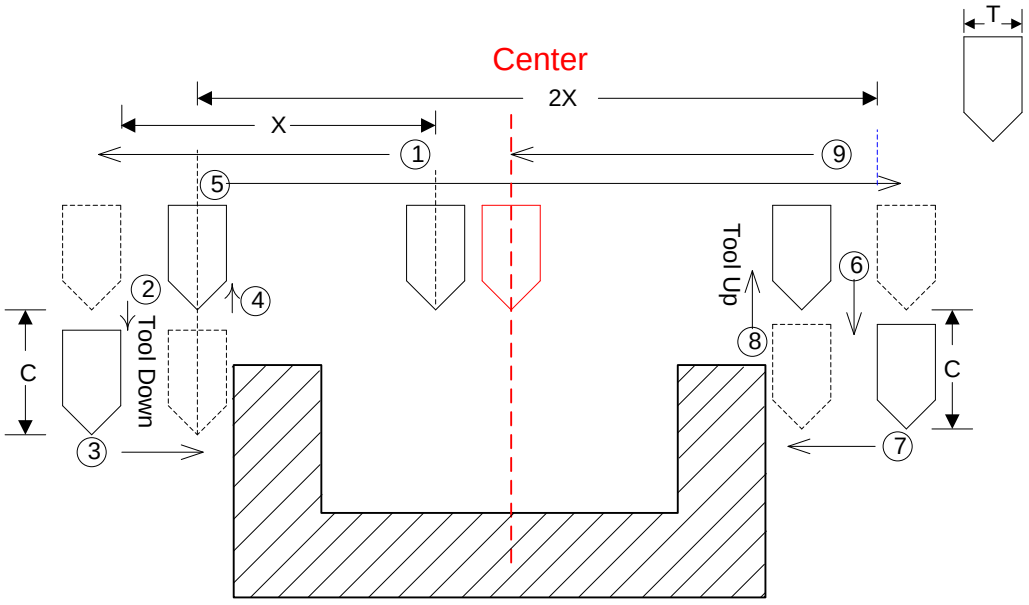


Fig. 3-50 The process of X outer center

Place the conducting workpiece (copper, iron, aluminum) on the insulated worktable, and connect it to the port CUT on the controller, while the cutter to COM. Put the cutter over the predicted center point position, press “H” to select “Auto Center(H)”, press “T” to switch to “X Outer”, and then press F1 “Auto Center” to start automatic centering. The cutter will move “X Center Distance”, shift down “Down Distance” and translate towards the workpiece a short distance until reaching the conducting workpiece so as to conduct the circuit and transfer the signal. At this time, the system will automatically record the current axial coordinate X1. Then the cutter will raise “Down Distance”, move two “Center Distance”, move down “Down Distance”, and translate towards the workpiece a short distance until reaching the conducting workpiece so as to conduct the circuit and transfer the signal. At this time, the system will automatically record the current axial coordinate X2 to calculate the coordinate of center point of workpiece and then move the cutter to this center point.

◆ **X Inner Center**

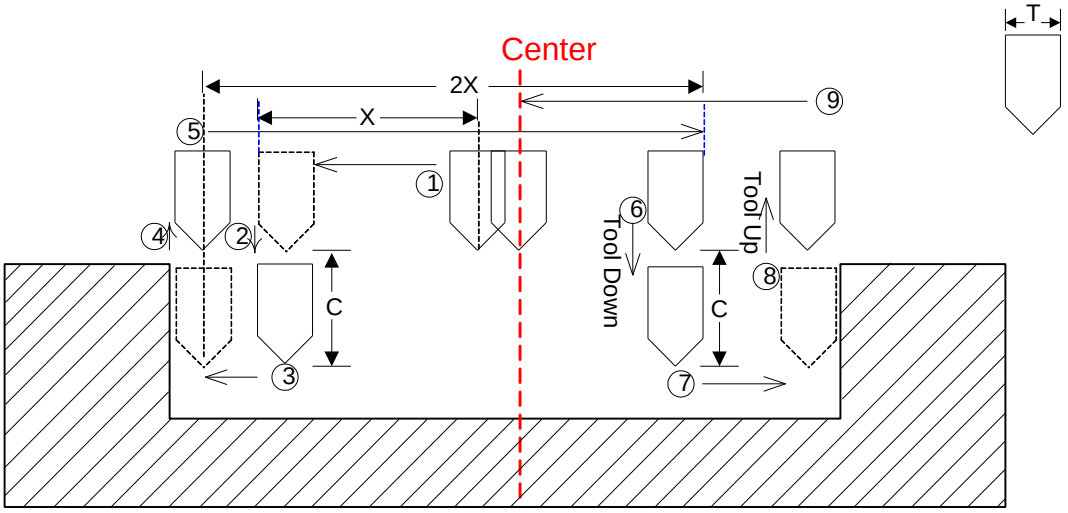


Fig. 3-51 Sketch map of X inner center

CAUTION

1) Before automatic centering, the tool nose must be over the predicted center position, and the value of center distance, down distance and tool diameter must be specified in advance.

2) Besides, the inner center distance must be smaller than workpiece radius, while the outer center distance must be larger than the workpiece radius.

3) When centering operation is used, it is recommended to use the active low.

◆ **Boundary Center**

Boundary center refers to setting the boundary point as workpiece origin, involving positive and negative boundary center, the process as shown in Fig. 3-52.

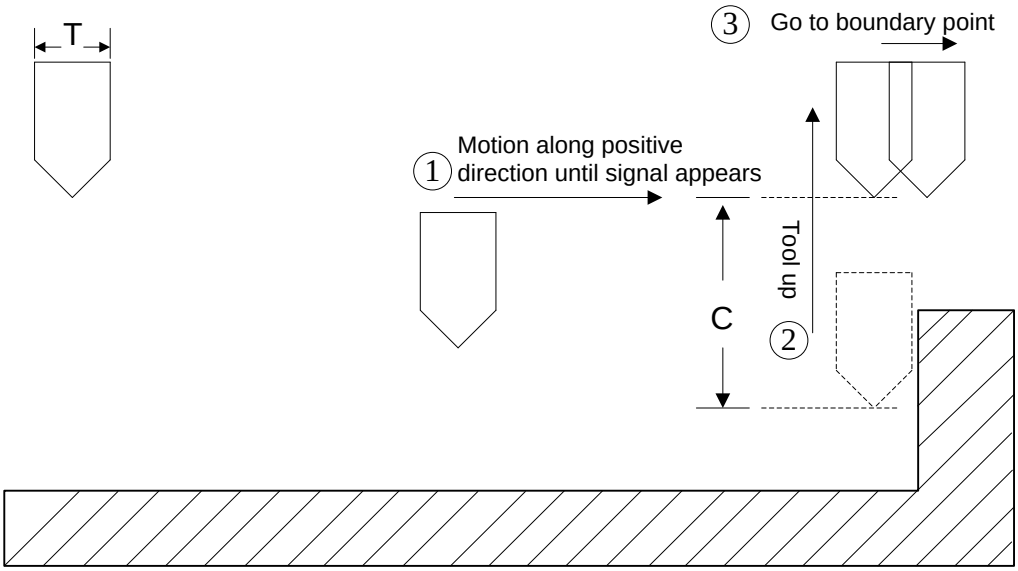


Fig. 3-52 Process of boundary center

3.11 Adjustment of Speed & Acceleration

3.11.1 Feedrate Setting

Feedrate (feed speed) can be set directly in the system interface.

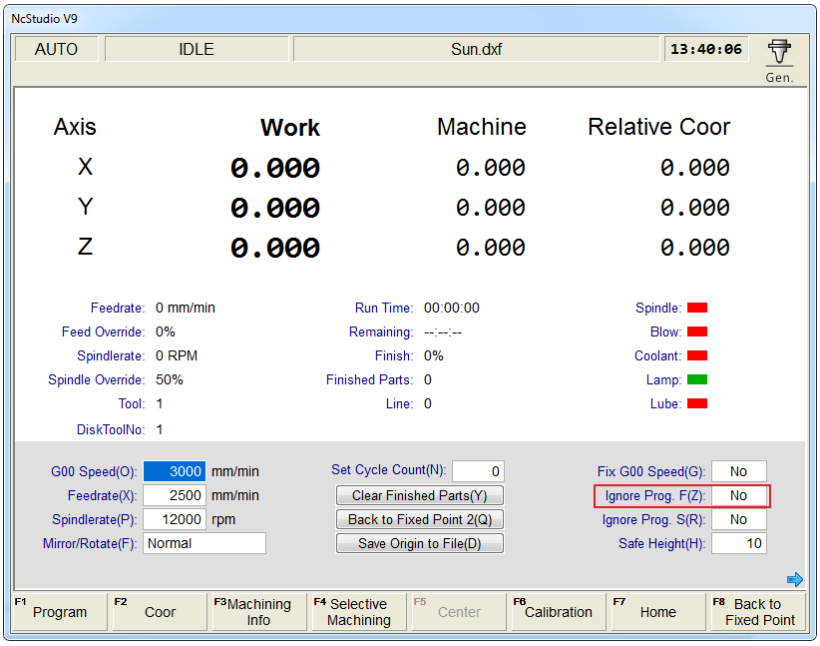


Fig. 3-53 Parameters setting zone-feedrate setting

In auto mode, press  to access the coordinate-auto interface. Feedrate can be directly set in the parameter setting zone above the manipulation button bar, as shown in Fig. 3-53. When the parameter N72001 “Ignore Programmed Feedrate” is set to “YES”, the system will adopt feedrate set in the system,

i.e. the value of “Feedrate”. When set to “NO”, the system will adopt the feedrate specified in the machining file.

There are two methods to select and set a parameter:

1) Press “↑” or “↓” to move cursor to the corresponding parameter, and then press Enter to eject an input box.
2) Press the corresponding shortcut key behind the desired parameter to eject an input box. Take “Prog.F(F)” as an example, pressing “F” will eject an input box for entering the desired value.

CAUTION

For feedrate settings on the new interface, operations and setting methods are the same while related shortcut keys next to each parameter may be different. See for new interface counterpart.

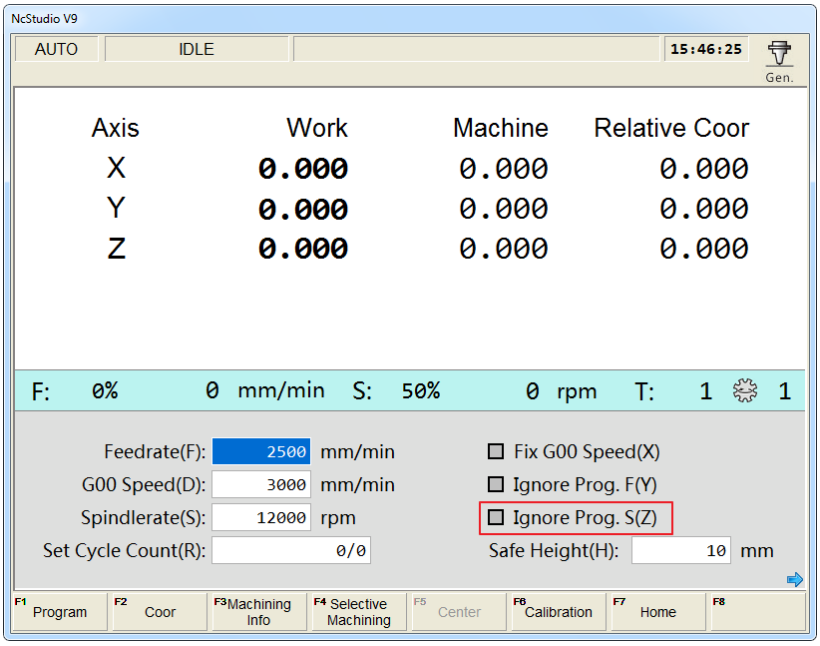


Fig. 3-54 Feedrate setting zone on new interface

Feedrate is also related with current feedrate override, so it can be controlled by adjusting the current feedrate override, and the formula is as below:

$$\text{Current Feedrate} = \text{Setting Feedrate} \times \text{Current Feedrate Override}$$

Feedrate override knob is on the operation panel, as shown in Fig. 3-55.

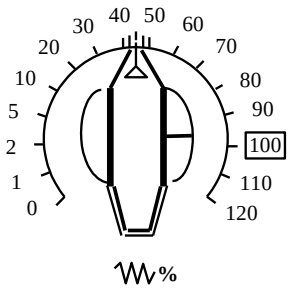


Fig. 3-55 Feedrate override knob

The adjusting range of feedrate override is “0% ~ 120%”.

3.11.2 G00 Speed Setting

G00 speed refers to the running speed of a machine tool under G00 command.

Similar to feedrate, G00 speed can also be set directly in the system interface. When the parameter N72003 “Fix Traverse Rate Override” is set to “YES”, the running speed of a machine tool under G00 command is fixed, i.e. the value of “G00 F”; when set to “NO”, the running speed of a machine tool under G00 command varies with the setting of feedrate override knob.

The concrete setting method is the same as that of feedrate.

3.11.3 Jog Speed/ Rapid Jog Speed

In manual-jog mode, press  to access the coordinate-jog interface. “Manual Low Speed” (jog speed) and “Manual High Speed” (rapid jog speed) can be set directly in the parameter setting zone above the manipulation button bar. See Fig. 3-56.

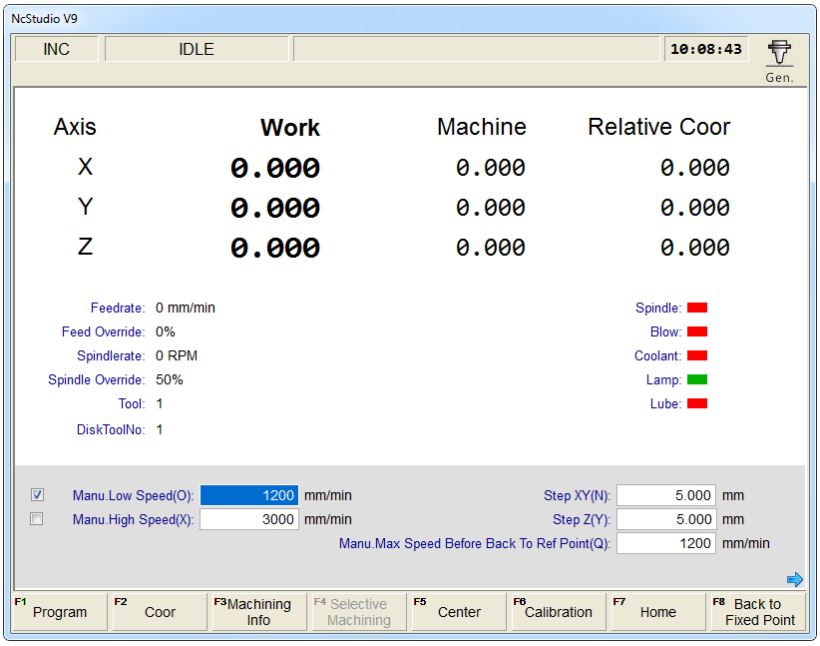


Fig. 3-56 Setting of jog speed and rapid jog speed

The concrete setting method is the same as that of feedrate.

CAUTION

For new interfaces, corresponding operations and setting methods are the same, except that the shortcut keys for each parameter are different. Manual high/low speed setting in new interface is as shown in Fig. 3-57 and user-defined step length setting in new interface is as shown in Fig. 3-58.

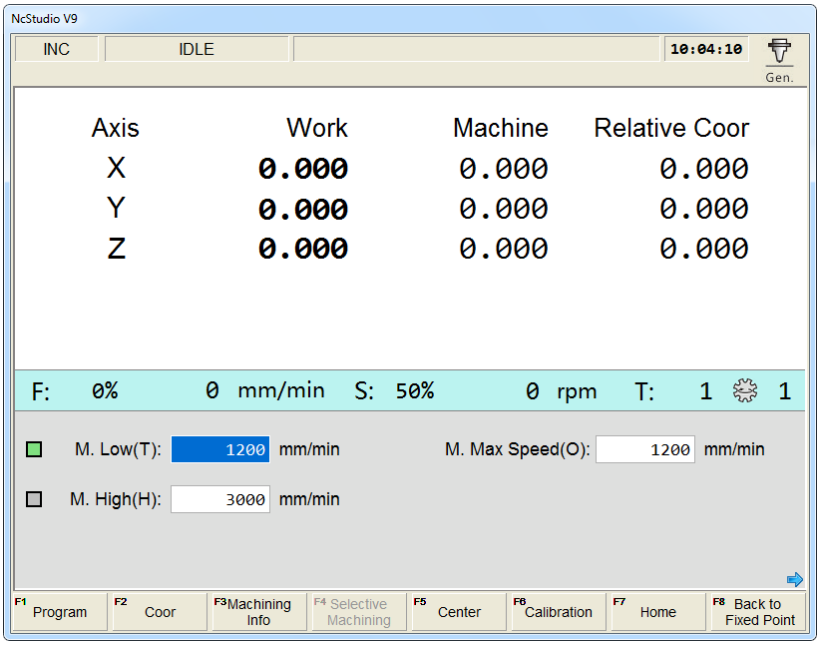


Fig. 3-57 Manual high/low speed setting in new interface

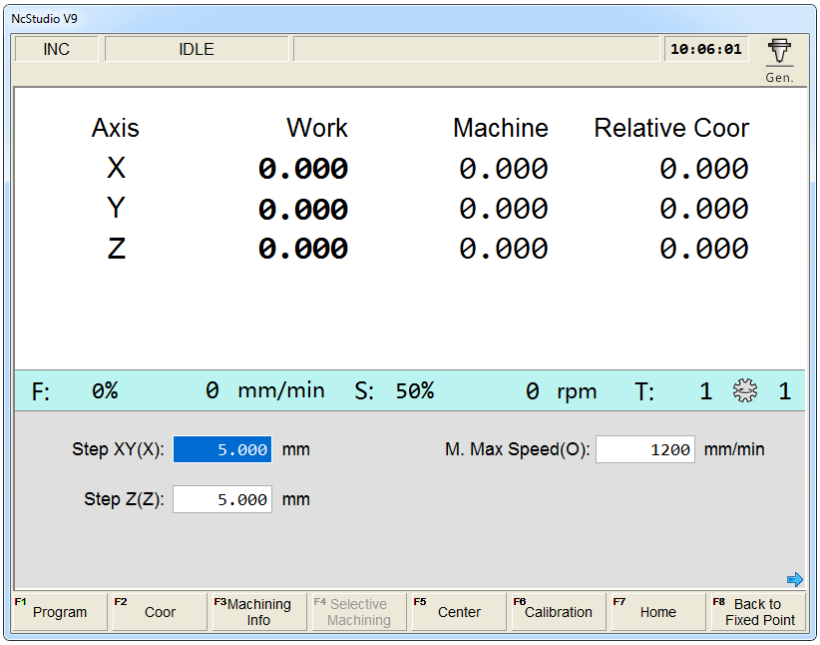


Fig. 3-58 User-defined step length setting in new interface

3.11.4 Parameter Specification

Except for feedrate and G00 speed, the other related parameters can be divided into following 5 types: velocity, acceleration, reference circle & circular speed limit, interpolation algorithm, and smooth setting.

◆ **Related Parameters for Velocity**

Parameter		Definition	Setting Range
N64000	Startup Speed	The max. achievable speed of a stepping motor in startup without acceleration	0 ~ 100000
N64020	Traverse Rate	The default speed of a machine tool in positioning (not in machining)	0 ~ 100000
N64060	Max Feedrate	The max. speed of a machine tool in machining	0 ~ 100000
N71000	Slow Jog Speed	There are two kinds of speed for option under manual mode: jog speed (Slow Jog Speed) and rapid jog speed, which can be switched by pressing the acceleration key on the operation panel. The system default running speed mode is jog speed.	0 ~ Rapid Jog Speed
N71001	Rapid Jog Speed		Slow Jog Speed~ Max Feedrate
N71002	Max Jog Speed Before Back to REF Point	The max velocity under manual mode before back to reference point.	0~ Rapid Jog Speed
Parameter N64000 “startup speed” applies to the startup frequency of a stepping & a servo driver, zero in default setting of driver. The startup frequency refers to the highest frequency of direct working startup without acceleration of motor. Reasonable setting of this parameter will improve machining efficiency, and avoid low speed segment with bad motion feature of motor. “Startup frequency” is generally included in the ex-factory parameters, but after installation, it will vary, especially in loading motion, thus, it should be set based on the actual measurement of motor power and inertia of a machine tool. Parameter confirmation method: set a lower value at first, and repeatedly make the machine execute typical motion & multi-axis synchronization motion, and then gradually increase this value until fixing the max. startup speed. The actual setting value of this parameter is half of the max. startup speed, with general setting range “300 ~ 400”.			

◆ **Related Parameters for Acceleration**

Parameter		Definition	Setting Range
N64101	Rapid Motion Axial Acceleration	The max. acceleration of each feed axis in machine positioning	0.001~100000
N64102	Z Axis Acceleration	The max. acceleration of Z-axis	0.001~100000
N64103	Speed Up Acceleration	Acceleration during speed up.	0.001~100000(mm/s^2)

Parameter		Definition	Setting Range
N64104	Speed Down Deceleration	Deceleration during speed down.	0.001~100000(mm/s^2)
N64120	Acceleration for Corners	The max. acceleration of feed motion on adjacent axes	0.001 ~100000
N64150	Axial Jerk	The change rate of acceleration of a single axis (acceleration's acceleration)	0.001 ~100000
N64204	Acc or Dec Time after Interpolation	The larger the value is, the smoother the speed will be. This parameter has no effect on track precision.	0 ~ 99999
"Acceleration for Corners" refers to the max. acceleration of feed motion on adjacent axes, and "1 ~4" times of "Axis Acceleration" is recommended, generally within "1200 ~ 5000". For higher speed requirement, "2 ~ 4" times of "Axis Acceleration" is recommended. "Axial Jerk" refers to growth rate of acceleration, i.e. the increment of acceleration in unit time, with unit "mm/s^3". It is available for S_type and LEP_type acceleration & deceleration, used to mitigate the bad effect caused by abrupt acceleration & deceleration of a machine.			

◆ **Related Parameters for Reference Circle and Circular Speed Limit**

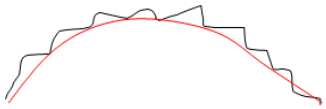
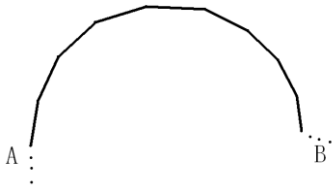
Parameter		Definition	Setting Range
N64207	Arc Velocity Limit	Only when this parameter is set to "YES" do N64208 and N64209 work.	YES: enabled NO: disabled
N64208	MAX Velocity of REF Circle	Reference circle is the reference of a machine in processing a circular workpiece. The max. speed of reference circle refers to the max. allowable speed of a machine in processing this circle without strong vibration.	0.001 ~ 100000
N64209	MIN velocity of REF Circle	Limit circular motion speed to avoid too low speed	0.001 ~ 100000
After installation of a machine completed, you can make the machine process a circle, in which vibration will occur due to centrifugal force. The higher the speed is, the stronger the vibration will be. Gradually increase the feed speed to see the state of vibration of the machine tool until the max. circular speed is achieved, i.e. the max. allowable speed of the machine tool without strong vibration. This circle is regarded as the reference circle, and its max. allowable speed is the max. speed of reference circle. Encountering other circles in machining, the system will calculate their max. centripetal acceleration in terms of the reference circle and its max speed to ensure the centrifugal force is within the debugging value, i.e. the vibration will not be stronger than that during ex-factory debugging. In processing a circle with small radius, even quite low feed speed of the circle will generate very high centripetal acceleration, thus the machining speed will be quite low caused by circle speed limit to limit the centripetal acceleration. To ensure machining efficiency, when the speed calculated by the system is lower than the setting value of N64209, the setting value of N64209 will be adopted in machining.			

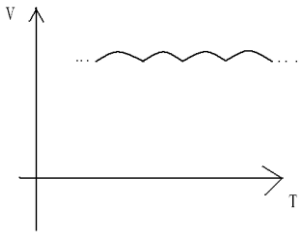
◆ **Related Parameters for Interpolation Algorithm**

Parameter		Definition	Setting Range
N64203	Path Interpolation Algorithm	Select the most suitable interpolation algorithm to reduce error after debugging.	0: Trapezoid algorithm 1: S_type algorithm 2: LEP algorithm 3: Acceleration trapezoid algorithm
N64205	MIN Velocity in LEP	The min. velocity in LEP algorithm interpolation	0 ~ 100000
N64203 is used for algorithm selection. The system currently supports trapezoid, S_type, LEP, acceleration trapezoid algorithms. Among them, trapezoid algorithm & S_type algorithm hold the highest efficiency, while LEP algorithm holds the highest machining quality in three-dimensional machining. When S_type algorithm is adopted, the max. acceleration of the system will reach the twice of single axis acceleration set in the system, so setting a smaller value for N64100 is recommended in S_type algorithm. "Acceleration Trapezoid Algorithm" means acceleration curve is a trapezoid. The relationship of acceleration and time: accelerate to the max. acceleration at "axial jerk", then keep this acceleration constant, and then decelerate to "0" at "axial jerk". Generally, if this algorithm is used, N64150 "axial jerk" can be set within "100000 ~ 200000" (mm/s^3). The flexibility of acceleration and deceleration in this algorithm is better.			

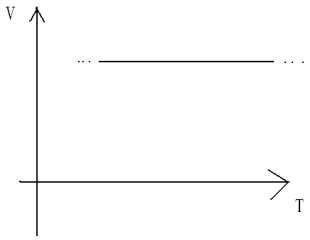
◆ **Related Parameters for Smooth Setting**

Parameter		Definition	Setting Range
N63002	Delay for Exact Stop	During machining, because of different inertia of each axis, the servo system may meet lag phenomenon at sharp turning corners. This parameter is used to overcome the lag phenomenon produced by the servo system by setting an extra stop time.	0.0 ~ 999
N63006	Path Smoothing Time	The larger the value of the parameter is, the smoother the workpiece surface will be. But if the value of the parameter is too large, it will affect the dimension of the workpiece. For a mold machine, it generally should be within 0.01, for a woodworking machine, within 0.03.	0.0 ~ 0.064
N64200	Smoothing the Path Velocity	If set to "NO", each motion instruction starts and ends at zero speed. If set to "YES", the system will set a proper start speed and end speed for each motion instruction according to the specific tool path to ensure smoothness of high speed machining.	YES: enabled NO: disabled

<table>
<tr><th colspan="2">Parameter</th><th>Definition</th><th>Setting Range</th></tr>
<tr><td>N64201</td><td>MAX Angle Smooth Velocity</td><td>The max. angle for the execution of “Smoothing the Path Velocity”</td><td>0 ~ 180</td></tr>
<tr><td rowspan="2">N64240</td><td>Smoothing Time Factor</td><td>It is used to specify the ratio of unit processing time to control periodic time when the speed is smooth.</td><td>0.01 ~ 10</td></tr>
<tr><td colspan="3">The larger the value is, the more ambiguous the details of workpiece are, i.e. the workpiece is smoother. But it will lead to reduction of arc radius in machining an arc. And it will also dwarf wave peak in machining workpiece resembling waves, as following. The range within 0.05s is recommended.
  </td></tr>
<tr><td>N64241</td><td>ConnectSpeed decreased at MaxConnetAngle</td><td>Whether to decelerate when the connection angle is approaching its max. value</td><td>YES: enabled
NO: disabled</td></tr>
<tr><td>N64245</td><td>Prepared number of path for optimizing performance</td><td>Segments for performance optimization, having no effect on the result of velocity planning.</td><td>1 ~ 2000</td></tr>
<tr><td>N64246</td><td>Slide speed for small lines</td><td>Eliminating velocity fluctuation when machining short segments.</td><td>YES: enabled
NO: disabled</td></tr>
<tr><td rowspan="2">N64247</td><td>Reference length of slide speed for small lines</td><td>Segments shorter than the value of this parameter will be executed speed smoothing.</td><td>0.001 ~ 10</td></tr>
<tr><td colspan="3">In machining an arc (or other curves) composed of short segments, velocity fluctuation, like frequent acceleration and deceleration obvious in our S-type algorithm, will occur at places where curvature is relatively large, as follows:
 

Assume that each segments is very short, and the curvature from A to B is large, the actual velocity planning will probably be as follows:
 

The above velocity curve (acceleration→ deceleration→ acceleration…) will lead to oscillation of a machine tool. At this time, the parameter “Slide speed for small lines”</td></tr>
</table>

Parameter		Definition	Setting Range
	should be set to “YES”, and the value of the parameter “Reference length of slide speed for small lines” should be set larger than the length of short segments in the tool path. When the short segments in the tool path are shorter than the reference length, the velocity will be executed smooth treatment. Otherwise, there is no treatment. The velocity after treatment is as follows:  In this way, frequent acceleration and deceleration is avoided, oscillation of a machine tool is eliminated, and machining quality is improved.		
N64248	Enable slide optimization	Optimizing the path smoothing and the Acc or Dec handling after interpolation, improving efficiency.	YES: enabled; NO: disabled
N64249	Velocity Smooth for Single Axis	With the function, moving speed of single axis will be restricted in order to get more smooth speed.	YES: enabled; NO: disabled

3.12 Simulation & Track

3.12.1 Simulation

The function of simulation provides a fast but lifelike simulation machining environment.

Running under the simulation mode, the system will not drive a machine tool to do the relative actions but only show the moving track of the cutter at high speed in the track window. By simulation, you can see the moving form of the machine tool in advance, avoiding machine tool damage due to programming mistakes in the machining file. And you can also learn other additional information.

Press  to access the program function area, where you can load a machining file into the system.

Press  ==> F3 “Machining Info” ==> F1 “Simu” to start simulation. You can see the whole machining track in the track window, and learn other info on the right side.

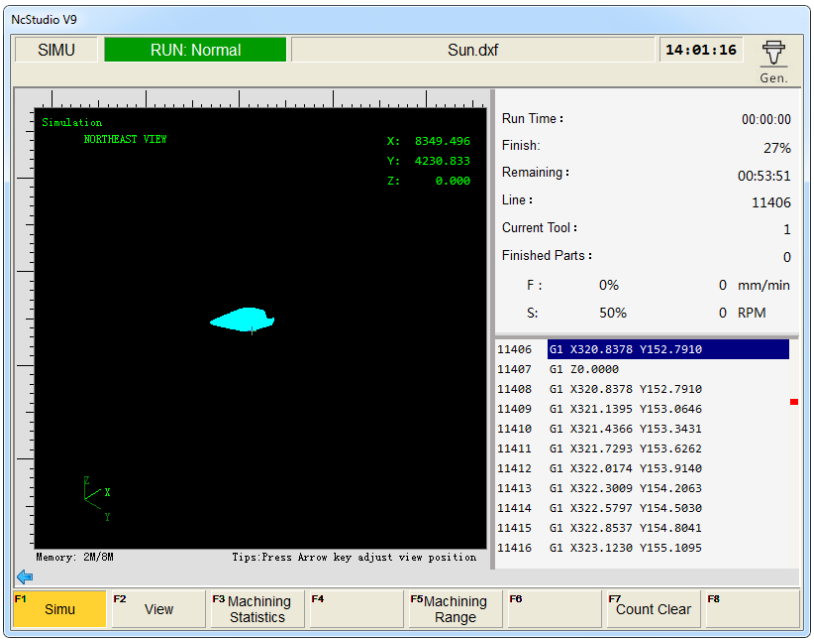


Fig. 3-59 Simulation and Track window

3.12.2 Motion Track

The track window displays the moving track of the cutter in real time. 3D display enables that you can view the tool path more intuitionally so as to ensure the accuracy of the loaded machining file. In the 3D tracking mode, abundant operation methods are offered by the system for the convenience of viewing the motion track from different perspectives and in an appropriate scaling.

By pressing the letter key “G” in the “Track window”, you can see the pop-up “View Adjustment” window, as shown in Fig. 3-60 and press the displayed shortcut keys to view the machining track from a proper perspective.

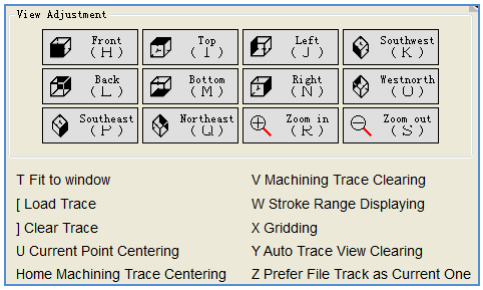


Fig. 3-60 View adjustment

3.12.3 Machining Statistics

As shown in Fig. 3-59, press F3 “Machining Statistics” to open the dialog box. This window mainly displays statistics info of all the previously processed machining files, including the machining file currently loaded. See Fig. 3-61. The upper part of the window displays the statistics info about the machining file currently loaded, such as name, start time, finished parts, total parts, run time, part run time, finished length and part total length. When a new file is loaded, all the statistics will be cleared.

The lower part displays the statistics info about all the previously processed machining files, including

name, start time, total time, total length and count.

Pressing the shortcut key F1 “Clear” will clear the selected history statistics in the list.

Pressing the shortcut key F3 “Save” will save the current history statistics to a txt file, with default name as “PartStat.txt”, under D:\Naiky\NK-300A\Config\std (varies with system configurations). The statistics of the machining file currently loaded will not be saved to the lower list, while those of processed machining files will be saved to the lower list automatically.

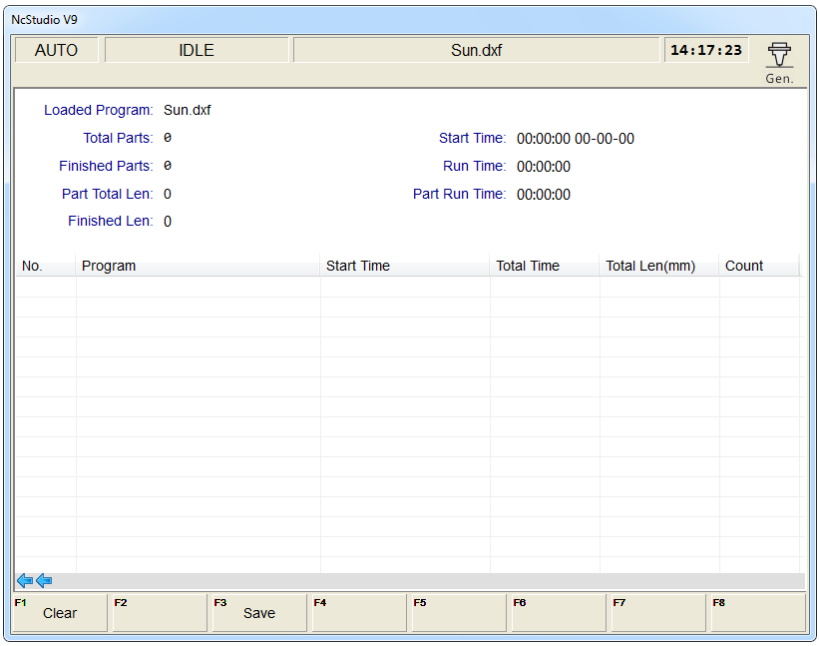


Fig. 3-61 Machining statistics window

3.12.4 Parameter Specification

◆ **Related Parameters (Under “Software Option”)**

Parameter		Meaning	Setting Range
N81000	Auto Load Graph	It sets whether the system will analyze the machining track automatically after a machining file is loaded.	NO: Not analyze YES: Analyze
N81001	Max File Size	It sets the file size limit in auto track loading. Only when the file size is smaller than or equal to this value can its track be loaded automatically, i.e. “Auto Load Graph” works.	0~10000
N81010	Gradient Fill	Setting whether to use gradient color fill in the track window	NO: Not use YES: Use
N81011	Draw Workbench	Setting whether to draw the boarder of the worktable in the track window	NO: Not draw YES: Draw
N81012	Draw Grid	Setting whether to draw grid in the track window	NO: Not draw YES: Draw
N81013	2D Mode	Setting whether to use 2D mode to view the	NO: Not use

Parameter		Meaning	Setting Range
		track in the track window	YES: Use
N81015	Clear on Loading	Setting whether to clear the contents of the current view when a new file is loaded	NO: Not clear YES: Clear
N81016	Draw WC Origin	Setting whether to display workpiece origin in the track window	NO: Not display YES: Display
N81017	Draw MC Origin	Setting whether to display machine origin in the track window	NO: Not display YES: Display
N81018	Bkground Color 1	Setting the background color for the track window	Select a color
N81019	Bkground Color 2	Setting the background color for the track window	Select a color
N81020 ~ N81023	G00/G01/G02/G03 Color (running)	Setting the color for motion track commanded by G00/G01/G02/G03 when running	Select a color
N81032 ~ N81035	G00/G01/G02/G03 Color (loading)	Setting the color for motion track commanded by G00/G01/G02/G03 when loading	Select a color
N81045	Grid Color	Setting grid color in the track window	Select a color
N81046	Coordinate Color	Setting coordinate color in the track window	Select a color
N81049	WC Origin Color	Setting a color for workpiece origin in the track window	Select a color
N81050	MC Origin Color	Setting a color for machine origin in the track window	Select a color

All these parameters are related with the attribute setting of the track window.

3.13 Compensation

Pressing the advanced function selection key  will access the following compensation interface, in which tool compensation, workpiece compensation and screw compensation are available.

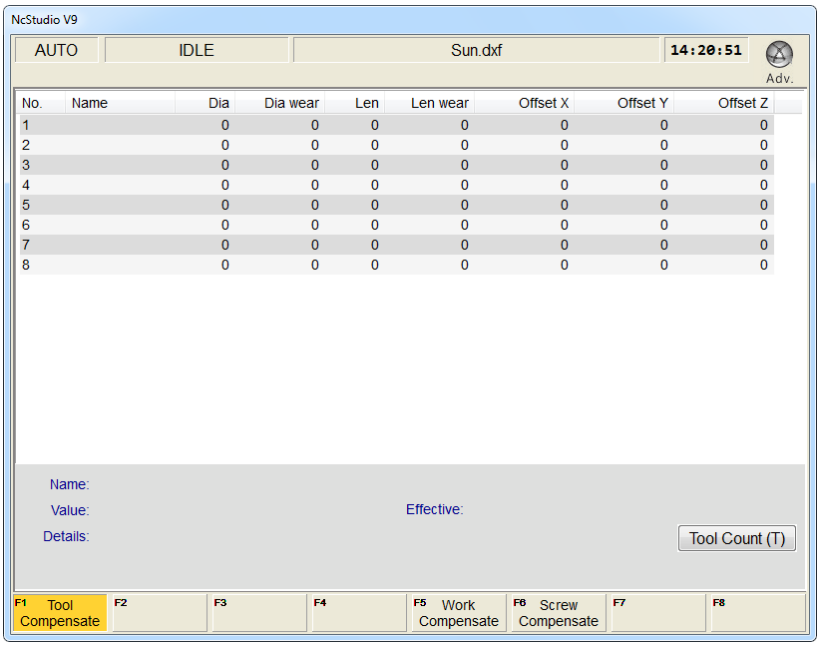


Fig. 3-62 Compensation interface

3.13.1 Tool Compensation

In CNC machining, the CNC system actually controls the tool center or the related point of the tool rest whose motion track is controlled directly to realize profile processing for the actual parts.

The cutting part actually used is the tool nose or the cutting edge which has dimensional variation with the tool center or the related point of the tool rest, so the CNC system has to compute the corresponding coordinates of the tool center or the related point of the tool rest according to the actual coordinate position of the tool rest or the cutting edge (namely the actual coordinate position of the part profile), which is called tool compensation.

Input the new tool parameter values in the tool compensation interface (see Fig. 3-62) if the tool nose radius is altered due to tool wear, tool sharpening or tool change, avoiding the trouble to modify the programmed machining file.

To make tool compensation (including tool radius compensation and tool length compensation) effective, parameter “N62410 Enable Cutter Compensation” should be set to “YES”. G43 (positive offset) and G44 (negative offset) are used for tool diameter compensation while G41 and G42 for tool radius compensation. And G40 (cancel tool radius compensation) and G49 (cancel tool diameter compensation).

The commands above must be used together with G00/G01 to make tool compensation.

◆ **Related parameters are:**

Parameter		Meaning	Setting Range
N62410	Enable Cutter Compensation	Setting whether to perform tool compensation	YES: enabled NO: disabled
N62411	Cutter Compensation Type	The type to establish and cancel cutter compensation	1: Normal type 2: Intersect type

Parameter		Meaning	Setting Range
			3: Insert type
N62412	Cutter Compensation Direction	Specifying the direction of tool compensation	0: No tool compensation 1: Left compensation 2: Right compensation
N62413	Num of Intervene Detected Graphics	See below for explanation.	1~5
	Interference here refers to over-cut caused by too large tool radius. Parameter N62413 decides interference detection among how many adjacent shapes. When interference phenomena detected, an interference alarm will be given. Generally, setting a smaller tool diameter will relieve the alarm. Note the default value of this parameter is 3. When set to 1, there is no interference detection and alarm.		
N62414	Intervene Evade Enable	To evade interference caused by some bad pos.	YES: enabled; NO: disabled
N65206	Force to Use Tool Compensation	If it is set to “YES”, when parameter “Enable Cutter Compensation” is set to “YES”, translation of an ENG file calls codes about tool length compensation or tool radius compensation; if it is set to “NO”, even though parameter “Enable Cutter Compensation” is set to “YES”, translation of an ENG file does not call codes about tool length compensation or tool radius compensation.	YES: Force to use NO: Not force to use

3.13.1.1 Tool Radius Compensation (Code G40~G42)

Tool radius compensation code, namely from G40 to G42, can make the tool moved by the offset value, see in Fig. 3-63.

To make the offset value is the same with the tool radius value, the system will firstly create an offset vector (known as “Starting”), whose length equals to radius of the tool.

Direction of the offset vector is perpendicular to the forward direction of the tool, looking into the tool center from the workpiece. If linear interpolation or circular interpolation is called after “Starting”, the system will contour with the tool moved by the offset, namely, with the tool compensated by radius value.

To end the compensation and make the tool return to the starting point, tool radius compensation code will be canceled and disabled.

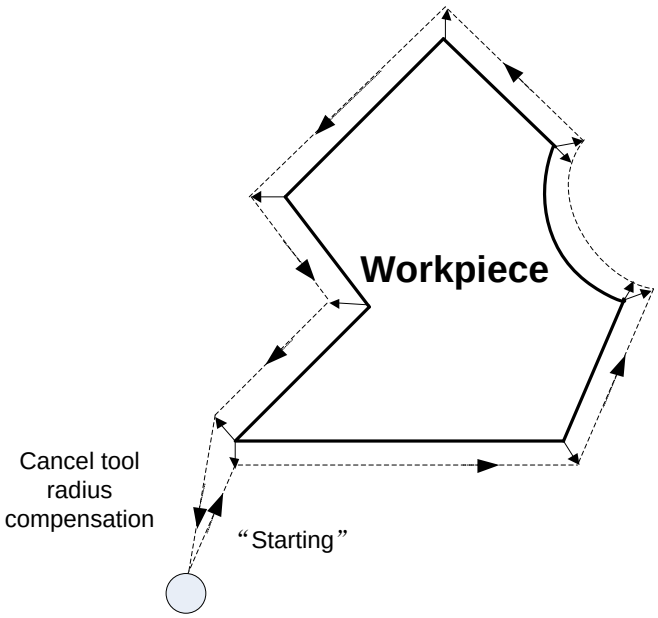


Fig. 3-63 Schematic Diagram for Tool Radius Compensation

3.13.1.2 Tool Compensation Type

Tool compensation should be established before executed, and cancelled after workpiece machining completed. To establish tool compensation is moving the tool to the edge of workpiece in a reasonable way, while to cancel tool compensation is moving the tool to the specified point from the edge of workpiece.

Generally speaking, tool compensation establishment consists of two segments, see segment 1 and segment 2 in Fig. 3-64. The software offers 3 ways to establish and cancel the tool compensation:

1) Normal type: the programming path is translated by 90 degrees to get the segment 2 for establishment, next, make the starting point of segment 2 the end point of the segment 1. Segment 1 and 2 constitute the tool nose path with tool radius compensated. Please note that this type is not available to arc command.
2) Intersect type: the programming path is translated in parallel to get the segment 2 for establishment, next, make the starting point of segment 2 the end point of the segment 1. Segment 1 and 2 constitute the tool nose path with tool radius compensated. Please note that this type is not available to arc command.
3) Insert type: after the programming path is translated, figure out the intersection point of segment 1 and 2. Insert a line from the starting point of segment 1 before translation and the starting point of segment 1 after translation, to get the tool nose path. It is available to arc command as well, but machining efficiency will be affected since an extra segment needs to be completed.

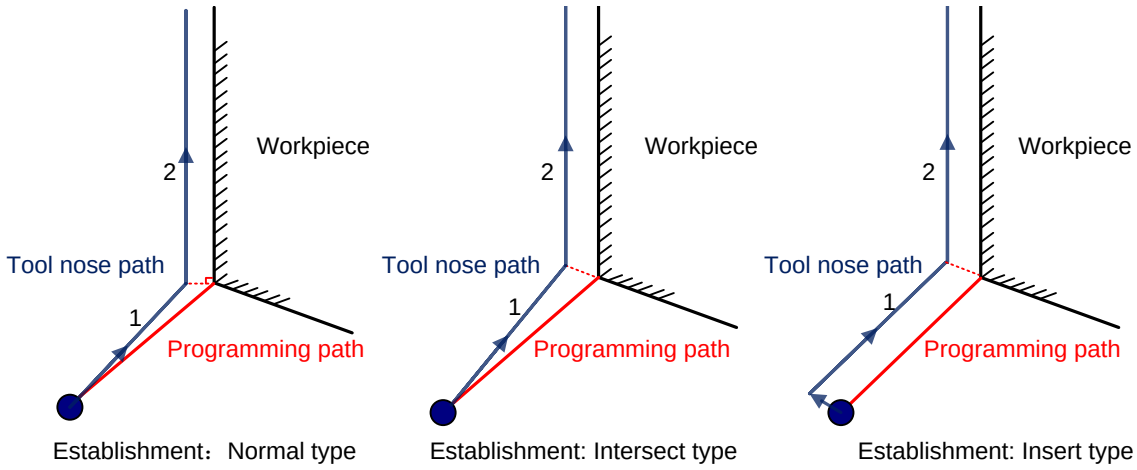


Fig. 3-64 Type to establish tool compensation

3.13.1.3 Tool Compensation Direction

The schematic diagram of tool compensation direction is as shown in Fig. 3-65.

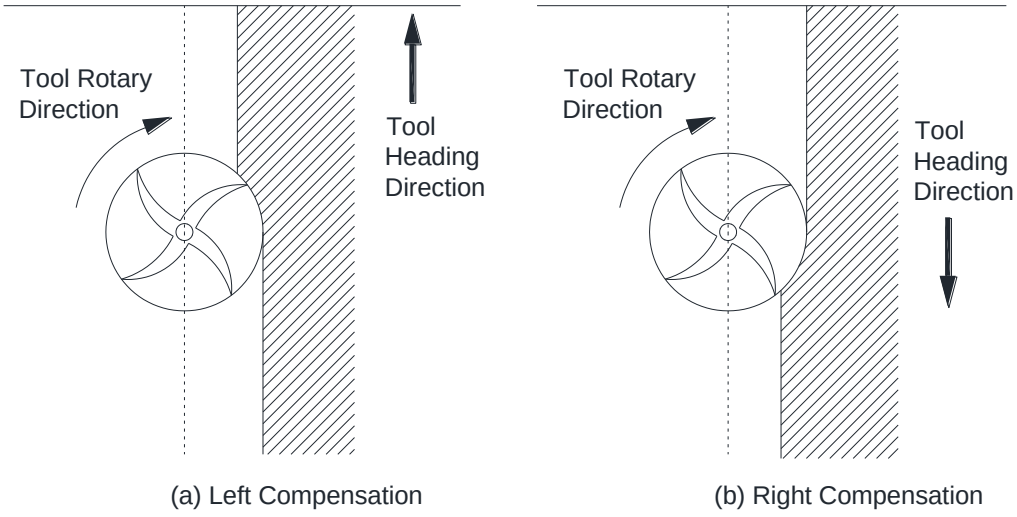


Fig. 3-65 Direction of tool compensation (a: left compensation b: right compensation)

Programming for tool radius compensation is as shown in Fig. 3-66:

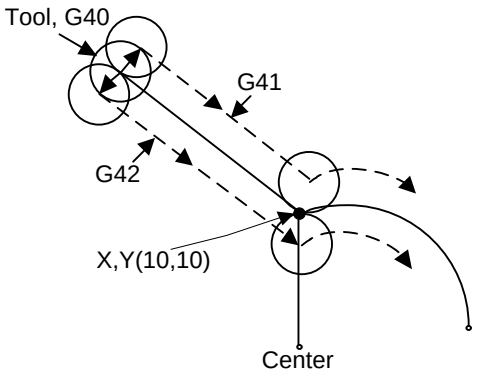


Fig. 3-66 Schematic diagram of tool compensation machining

```
G17 G01 G41(G42) X10 Y10 F1000 D01    'linear interpolation and tool radius compensation
G02 X_ Y_ I_ J_      'circular interpolation
```

Among the above programming, G41 means left compensation, namely the tool will deviate a distance towards the left side of tool heading direction and this distance is tool radius; G42 means right compensation, namely the tool will deviate a distance towards the right side of tool heading direction and this distance is tool radius. X10Y10 is the endpoint coordinates of linear motion. F1000 represents the tool moves at the speed of 1000 mm/min. D01 is the parameter of G41/G42, namely the tool compensation number. From D00 to D07, they have their own corresponding radius compensation value in the tool compensation table.

For the details of programming of tool compensation instruction, see *Programming Manual*.

3.13.2 Workpiece Compensation

The system includes single compensation and array compensation, as shown in Fig. 3-67. In single compensation, each workpiece is compensated separately, i.e. the compensation offset of each machining file can be different. In array compensation, the same rows or columns are compensated the same offset. Taking X01Y01 as an example, it compensates the first rows and columns.

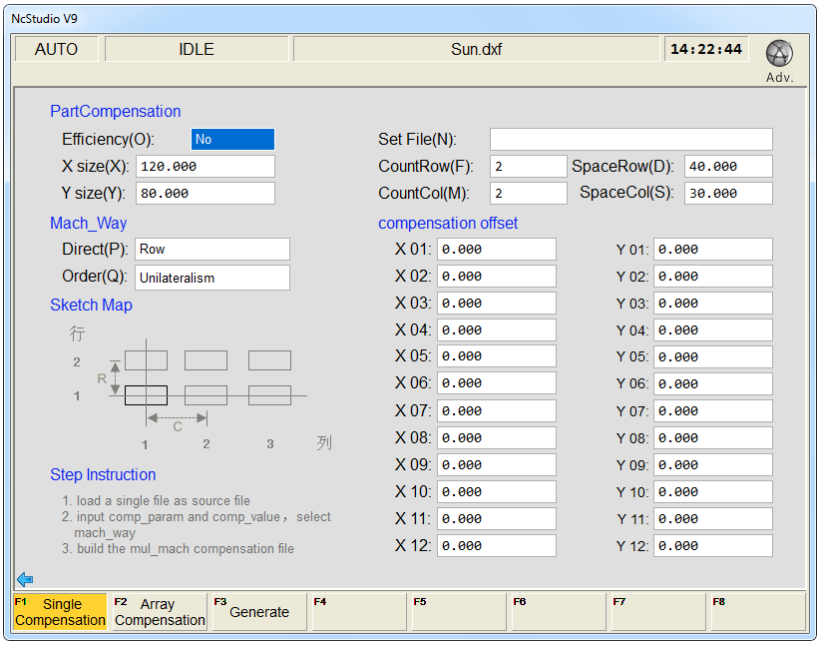


Fig. 3-67 Workpiece compensation interface

To enable workpiece compensation, set “Efficiency(O) to “Yes”.

Set File(N): load the desired single workpiece file into the system firstly, and then turn to this interface and press “Set File(N)” to load the file for work compensation. If the file is not loaded into the system firstly, pressing “N” will eject an error prompt, as shown in Fig. 3-68.

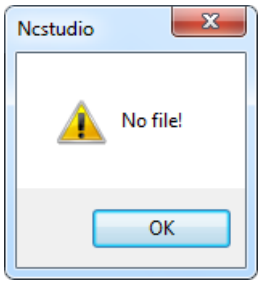


Fig. 3-68 Source file error prompt

After manually entering workpiece size (X size and Y size), rows (CountRow), columns (CountCol), row space (SpaceRow) and column space (SpaceCol), set machining direction and order to determine machining sequence of each workpiece, then enter the compensation offset for each workpiece, and then press F3 “Generate” to generate a multi-workpiece file. Enter the file name, and then press Enter to load the file into the system automatically. The new file is saved to “D:\NcFiles”.

CAUTION

1) After the file is loaded for workpiece compensation, the source file in the system will be deleted automatically, since the final machining file loaded into the system will be the compensation file newly generated.
2) G28, G29, G65, G92, M30 and M2, etc. are not supported in scale and array functions, neither are subprograms in the tool path. If there are codes mentioned before, the system will prompt manual or automatic deletion.

3.13.3 Screw Error Compensation

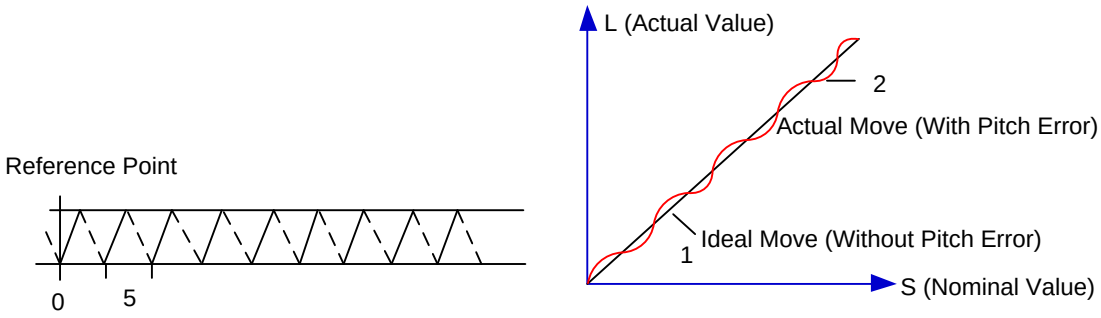
3.13.3.1 Causes of Screw Error and Compensation Method

Screw error consists of screw pitch error and errors caused by backlash. Generally, these two errors don’t need compensation, but backlash compensation is needed in high precision required situation, if higher precision is required, both the two compensations are needed.

◆ **Pitch Compensation:**

Pitch error is caused by screw defect and long-term wear, etc. In order to improve precision, pitch compensation is needed. The sketch of a screw is shown in Fig. 3-69(A). A coordinate system is established, based on “0” point on the screw as the reference point, nominal value as X-coordinate, and actual value as Y-coordinate. Then the ideal moving curve is as curve “1” in Fig. 3-69(B), however, the actual curve will be curve “2” due to pitch error. That is to say, the Actual value is not the same as its corresponding Nominal value, the actual moving curve deviating from the ideal one, and their difference is called error, i.e.:

Error = Nominal machine coordinate – Actual machine coordinate



(A) Sketch of Screw (B) Picture of Moving Curve

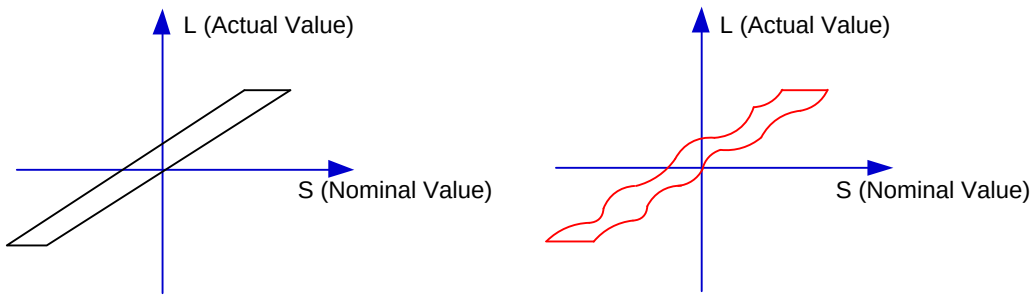
Fig. 3-69 Analysis of pitch error

◆ **Compensation Method:**

In pitch compensation, generally pitch error value isn't related to feed direction. That is, when the pitch is too small in positive feed, additional pulse is needed, and thus, when negative feed passes the same position, the same amount of feed pulse should be added. But if the pitch is large, deduction of pulse is needed, and neither is the reducing amount related to feed direction. In software compensation, correction of each point on the error curve should be tabulated and saved to the system memory. Then auto compensation for coordinates of each point is available in running, so as to improve machine precision.

◆ **Backlash Compensation:**

Hysteresis feature is caused by forward and reverse clearance. Assume that driving shaft drives driven shaft in negative (CW) rotation, servo motor will be idling without moving worktable because of mechanical driving chain backlash, when the driving shaft suddenly begins CCW rotation (positive motion). After staying at a certain position for some time, the worktable will move backward with the driving shaft; when the direction of the driving shaft changes again, the situation is the same, which is called Hysteresis. If pitch error doesn't exist, under ideal condition, the moving curve of worktable is shown in Fig. 3-70(A), in which the curve of horizontal section is during the idling of servo motor without worktable movement. The actual moving curve of worktable is shown in Fig. 3-70 (B).



(A) Hysteresis Feature (B) Actual Moving Curve

Fig. 3-70 Analysis of backlash

The popular explanation is: because spindle is generally fixed on the screw whose outer wire and the inner wire on the outer wire cannot be completely matched, backlash compensation compensates the

clearance between the screw of last direction that the spindle needs to finish after reversing its moving direction.

◆ **Measuring Method and Compensation Method**

Backlash can be measured by a specialized gauge. Firstly, fix the instrument nearby the spindle. Secondly, make the watch hand at the zero point position (machine origin). Thirdly, manually move “a” centimillimeter, then move back “a” centimillimeter, and then see the actual moving distance of watch hand- “b” centimillimeter. Therefore, the backlash is measured, namely (a-b) centimillimeter.

If one axis moves from positive to negative, “+Q” pulse will be output before reversal; conversely, from negative to positive, “-Q” pulse will be output before reversal (Q is backlash, preset by the program).

3.13.3.2 Screw Error Compensation Operation

Actually the system has already combined the above two errors (screw pitch error and backlash) to deal with and will execute error compensation automatically based on the error data in the file after the backward error and forward error of the corresponding nominal coordinate of each coordinate axis are listed into the screw error compensation file.

The detailed operation is: save the measured value of compensation in file “axeserr.dat” in directory of USB disk which is connected to the system. The system will execute compensation as the file described immediately.

◆ **Screw Error Compensation File “axeserr.dat”**

The name of the screw error compensation file is “axeserr.dat”, found under the installation directory. Modification to the data in the screw error compensation file will become valid after the software is restarted.

The file format is:

1) Firstly specify length unit, currently the supported length unit is mm and the style of writing is: unit = mm
2) Then specify error sequence of each axis. To work properly, the contents in this sequence must be in the ascending order of nominal machine coordinate value. Refer to Table 3 1 for details.
3) Annotation: it must be in a separate line and started with a semicolon. Its syntax is:

;<Annotation contents>

Note that a semicolon must be the first character of the separate line, that is, no other character should be in front of the semicolon, even blank space.

Table 3-1 Explanation about axis error sequence

Item	Specification	Remark
Axis Name	X, Y, Z, (Case-insensitive)	
Nominal Machine Coordinate	It is the machine coordinate with a sign with respect to reference point, which is calculated by the given pitch and pulse equivalent (i.e. the length calculated based on the nominal pitch, not on the	

Item	Specification	Remark
	actual physical one), arranged in ascending order. Nominal machine coordinate must be within the stroke range, or the compensation is invalid.	
Backward Error	The error generated by the motion towards decreasing direction of coordinate value.	
Forward Error	The error generated by the motion towards growing direction of coordinate value.	

1. The style of writing of each axis error sequence:

[Axis Name]

<Nominal Machine Coordinate>, < Forward Error>, < Backward Error>

<Nominal Machine Coordinate>, < Forward Error >, < Backward Error >

<Nominal Machine Coordinate>, < Forward Error >, < Backward Error >

2. The sign of nominal machine coordinate and actual machine coordinate

Pay special attention to the sign of nominal machine coordinate and actual machine coordinate, especially when equipment like laser interferometer is used to measure the length. Calculate after the measured length is converted to the corresponding machine coordinates, or a wrong result may occur.

Table 3-2 Example of screw error compensation file format

Condition	Example	Remark
Common cases	;unit=mm [X] -570.025, 0.027, 0.083 -450.020, 0.025, 0.077 -330.015, 0.015, 0.068 -210.010, 0.000, 0.057	
A certain axis only needs backlash compensation	;unit=mm [Y] 0.000, 0.000, 0.030 1000.00, 0.000, 0.030	Only the data of start point and end point of this axis needs writing down.

◆ **Related parameters are:**

Parameter		Meaning	Setting Range
N12000	Screw Error Comp	It sets whether to enable screw error compensation and decides compensation type.	0, 1, 2
N12001	Backlash Compensation Only	It sets whether to enable backlash compensation.	YES: enabled NO: disabled

There are three options for parameter N12000, which are 0 (no compensation), 1 (unidirectional compensation) and 2 (bidirectional compensation).

◆ **No compensation**

To disable compensation, set set N12000 to “0”, and N12001 to “NO”.

◆ **Unidirectional compensation**

To compensate by reading “Err Pos.” data (unidirectional error data) and backlash value in the screw compensation interface, set N12000 to “1” and N12001 to “YES”.
To compensate by only reading “Err Pos.” data (unidirectional error data) in the screw compensation interface, set N12000 to “1” and N12001 to “NO”.

◆ **Bidirectional compensation**

To enable bidirectional compensation, i.e. to compensate by reading “Err Pos.” (forward error) and “Err nEG.” (backward error) data in the screw compensation interface, set N12000 to “2”.

When parameter N12001 is set to “YES”, it means to enable the backlash compensation; when it is set to “NO”, it means that backlash compensation will not be enabled and comprehensive compensation will be made by reading backlash value and pitch error data from the error file.

3.13.3.3 Software Interface and Operation

Press the advanced function selection key  to access parameters interfaces, where you can find parameter N12000 “Screw Error Comp”. Set the parameter and decide which compensation type is to be enabled.

Press  ==> F6 “Screw Comp” to access the compensation interfaces, unidirectional or bidirectional compensation interfaces, which is decided by the setting of the parameter N12000 “Screw Error Comp”.

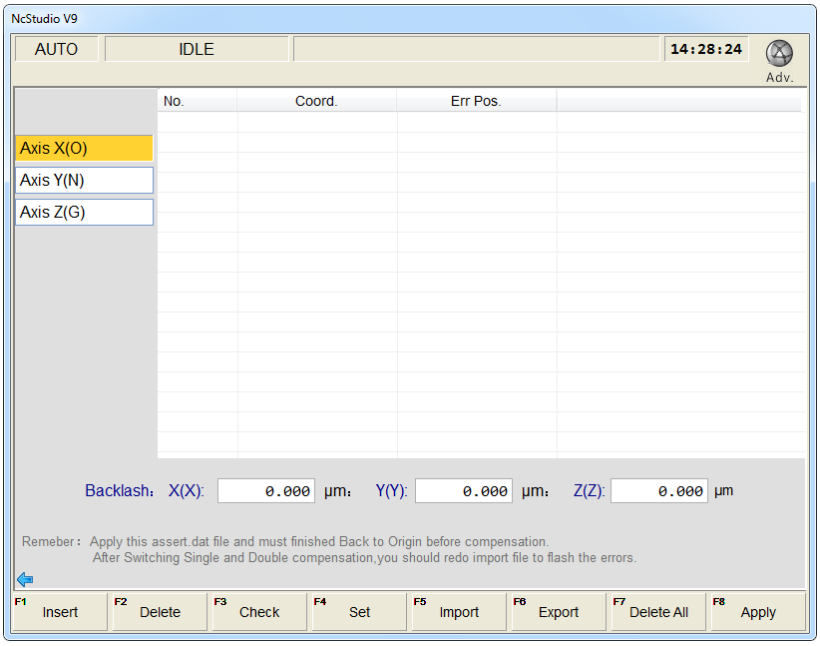


Fig. 3-71 Unidirectional compensation interface

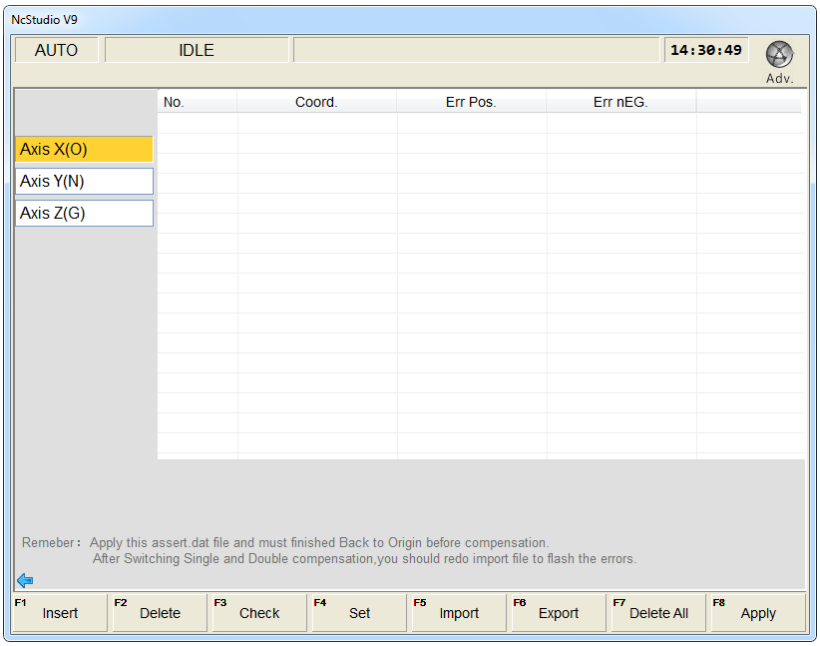


Fig. 3-72 Bidirectional compensation interface

Unidirectional compensation is fit for the situation that the forward error and backward error are relatively stable. If forward direction is the first in measure, enter the forward error (default) into “Err Pos.”; if backward direction the first in measure, enter the backward error into “Err Pos.” Backlash of each axis can be used together.

Bidirectional compensation reads forward error and backward error to execute comprehensive compensation, fit for the situation that forward error and backward error are not stable. Backlash is not available in the screw compensation interface when bidirectional compensation is enabled.

◆ **Unit**

Coord. (Position coordinate): mm

Err Pos. (unidirectional error), Err Pos. (forward error), Err nEG. (backward error), backlash: μm

◆ **Import and file format**

Three types of file can be imported from a USB disk. They are .lin, .rtl and axeserr.dat.

CAUTION

1) Check whether the .lin or .rtl file to be imported is generated correctly.
2) After manually modifying the axeserr.dat file, check whether the data of each axis is arranged in ascending order or descending order, and whether the data, including nominal coordinate, forward error, and backward error, are correct compared to the backlash of each axis.
3) After switching from unidirectional compensation to bidirectional compensation and restarting the software, load the file again to refresh the forward error and backward error of bidirectional compensation.

4) To improve the precision of the imported file, the data should be the average of multi-measurement (at least two times) on the same coordinate position.

Compensation Error Data= Measured Error Data- Error Data of Machine Origin

◆ **Delete All**

This key is used for deleting all data.

◆ **Apply**

After this key is pressed, the compensation data will be written into the drive, and the axeserr.dat file will be saved to the D disk.

CAUTION

After modification to Coord. (position coordinate), Err Pos. (unidirectional error), Err Pos. (forward error), Err nEG. (backward error), backlash, parameters N12000 and N12001 and execution of "Apply", there is a must to return to machine origin first to ensure compensation accuracy.

◆ **Remarks:**

- Error value= Actual machine coordinate- Nominal machine coordinate
- Ascending sequence and descending sequence can be set.
- Check whether there is any invalid data in the axeserr.dat file after opening the software and importing the file.

3.13.4 Quadrant Error Compensation

Quadrant error compensation, also called friction compensation, refers to the distortion, the most commonly seen is a spike, at the conversion part of two adjacent quadrants in circle machining of a machine tool.

To eliminate this kind of distortion, error compensation is necessary.

Quadrant error compensation parameters are used for spike compensation when machining arc passes quadrants. The setting method along positive and negative directions of X/Y/Z is similar.

◆ **Related Parameters:**

For there are 6 groups of parameters "time", "distance" (compensation amount), "delay" and "intensity", only one of them is listed in the following table.

Parameter		Meaning	Setting Range
N12020	Turn On AQE Compensation	Setting whether to enable quadrant error compensation	YES: enabled NO: disabled
N12030	Time(Group 0)	The bigger the value is, the larger the area will	0~10

Parameter		Meaning	Setting Range
		be influenced by the compensation. The recommended value is about 0.02 s.	
N12031	Distance(Group 0)		0~10
N12032	Delay(Group 0)		0~10
N12033	Intensity(Group 0)		0~1
To make quadrant error compensation effective, parameter N12020 should be set to “YES”. The larger the value of “Distance” is, the more obvious the compensation result will be. But note that too large value will make the arc concave, and too small value cannot decrease the arc height effectively. The recommended setting value is 0.3~3 times of the actual height of the spike measured by a measuring device like a laser interferometer (compensation result and compensation time are related to compensation intensity). Delay: the spike may not exactly appear at the conversion part on some machine tools due to the difference of mechanical property of each machine tool, but a distance away from the quadrant point. Estimate the time for finishing this distance and set it as the value of the “Delay”. Intensity has an influence on the compensation result: the bigger the value is, the more obvious the result will be.			

3.14 Log and Diagnosis

3.14.1 Log

To access the log interface, press the diagnosis function selection key  ==> F2 “Log”.

The log interface records important operations and system events. Not only can the log info since this time start-up be browsed, but also history records can be viewed. See Fig. 3-73.

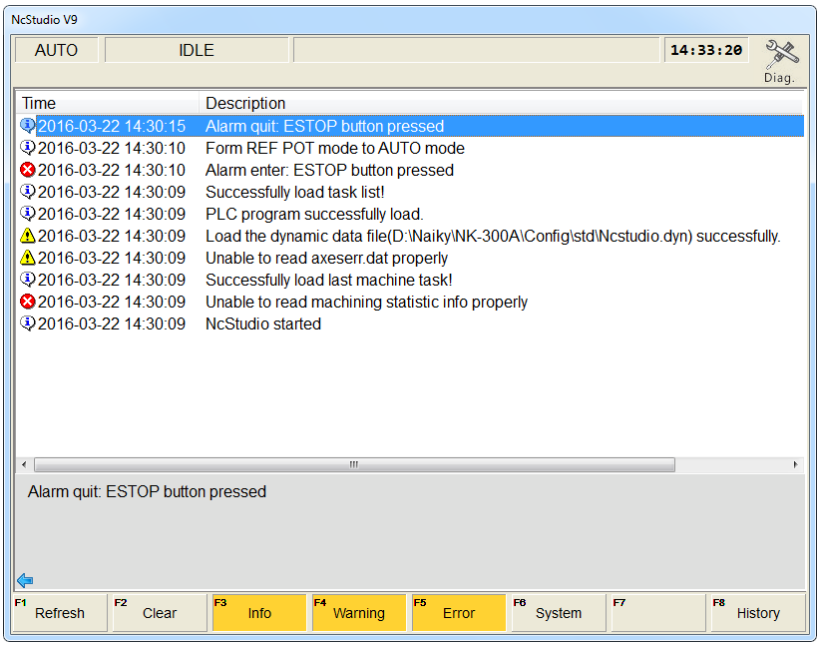


Fig. 3-73 Log interface

◆ **Refresh**

Pressing the shortcut key F1 will refresh the log list to make it synchronize with the system.

◆ **Clear**

Pressing the shortcut key F2 will clear all the current log information.

◆ **Info, Warning and Error**

Their shortcut keys are F3, F4 and F5 respectively.

Their default state is checked highlighted in yellow, namely the system displays normal info, warning info and error info by default. If you don't need certain info displayed, you just need to press the corresponding shortcut key to eliminate the yellow highlight. For example, pressing F5 (shortcut key of [Error]) will make the button bounced and the system hide the error info.

◆ **System**

Pressing F6 can view the system info, which needs password.

◆ **History**

Pressing the shortcut key F8 will display all the logs since recording.

3.14.2 Diagnosis

To access the diagnosis interface, press the diagnosis function selection key  ==> F5 "Diagnosis".

The diagnosis interface displays current feedback machine coordinates of each axis. After inputting a

valid sampling port into the channel and setting sampling interval, press F1 “Start” to diagnose the corresponding port. See Fig. 3-74.

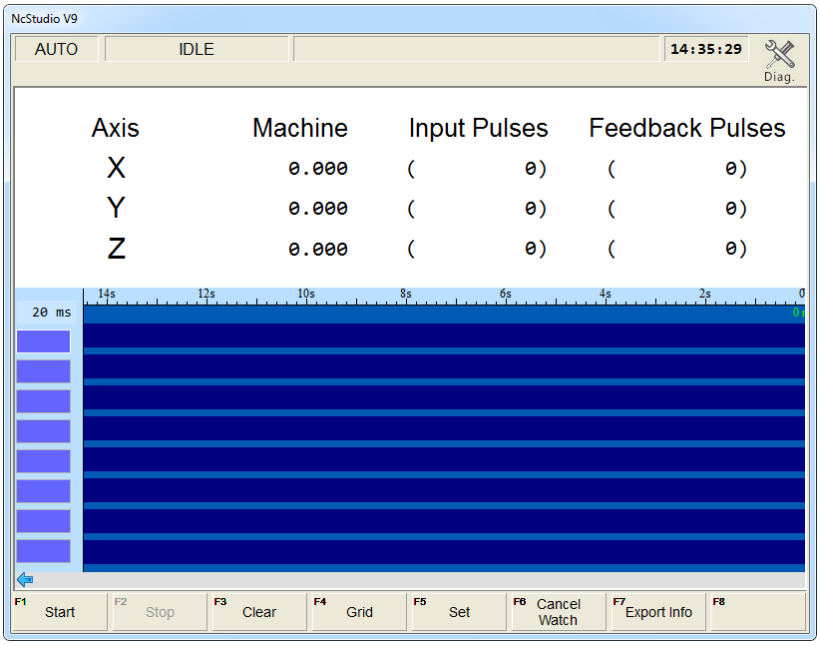


Fig. 3-74 Diagnosis interface

◆ **Start**

Pressing the shortcut key F1 will start diagnosing the corresponding port.

◆ **Stop**

Pressing the shortcut key F2 will stop diagnosing the corresponding port.

◆ **Clear**

Pressing the shortcut key F3 will clear the diagnosis result of the corresponding port.

◆ **Grid**

Pressing the shortcut key F4 will bring grid lines into the sampling window.

◆ **Set**

Sampling interval can be set after the shortcut key F5 is pressed, as shown in Fig. 3-75.

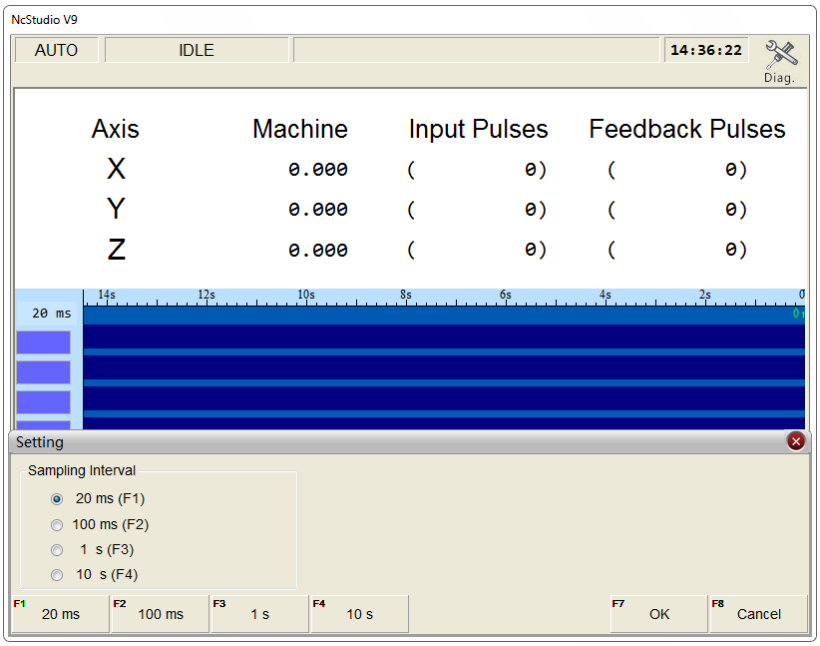


Fig. 3-75 Setting screen

Press F7 “OK” to confirm it after a sampling interval is selected (20ms, 100ms, 1s and 20s). And then the corresponding port or PLC address can be sampled periodically, realizing tracking detection to the port.

◆ **Cancel Watch**

Pressing the shortcut key F6 will cancel the monitoring to the corresponding port.

◆ **Export Info**

Pressing the shortcut key F7 will export the system information to D:\Naiky\NK-300A\Config\std (varies with system configurations), or to the removable disk if it is inserted.

3.15 Program File Management

Program file management manages the machining files in the system, related to operation of machining program.

3.15.1 Machining Wizard

NK300BX offers 5 basic machining program wizards: circular contour, circular pocket, rectangular contour, rectangular pocket and screw compensation. You just need to input some simple parameters to complete the operation of circular contour and rectangular contour, etc. Take circular contour milling as an example in the following:

Press  ==>  ==> F1 “Program Wizard” to enter the wizard screen, with the circular contour wizard screen as the default one, as shown in Fig. 3-76. To switch to other wizards, press the corresponding shortcut keys. To achieve the desired results, you can set parameters for the selected

machining shape, such as milling inner contour or outer contour (milling inner contour mills the region inside, and milling outer contour mills along the contour), workpiece diameter, initial (workpiece) coordinate X/Y, layer depth (of each cutting), engraving depth (of several accumulated cutting) and cutter diameter. After parameters are set, it is suggested to save them before loading the wizard into the system.

The operation method and parameter setting principle of circular pocket, rectangular contour and rectangular pocket are the same as those of circular contour, except for some parameters.

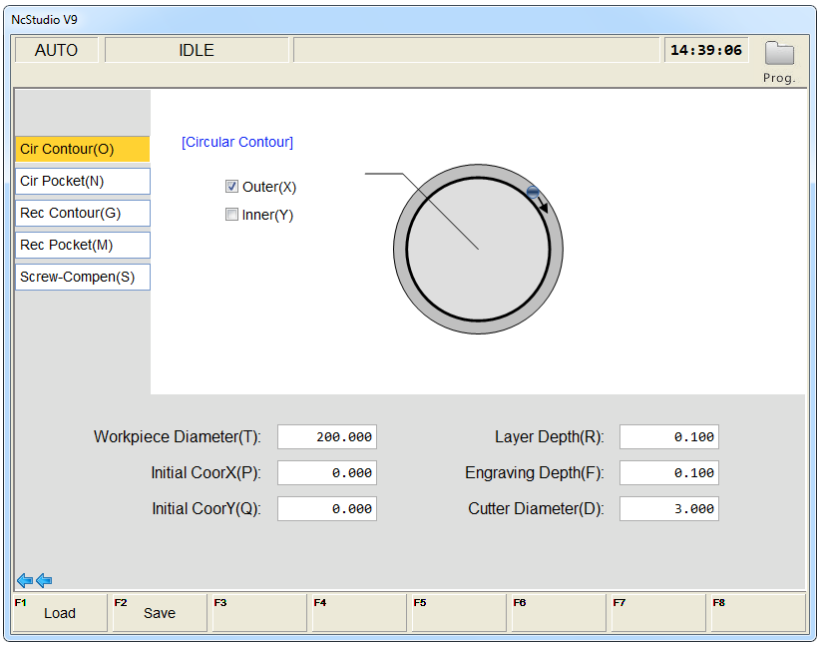


Fig. 3-76 Circular contour milling screen

In interface 「Wizard」, press key “S” to access interface 「Screw-Compen (S)」, as shown in Fig. 3-77. You can set relevant parameters for desired effect of machining drawing.

Wizard “Screw Measure” is used to measure screw error via laser interferometer.

Enter the values for start and end points, measuring points, repeats and dwell time, and then press F2 to save the setting. The system will generate a program file automatically to the directory D:\NcFiles\Wizards. Press F1 to load the file into the system.

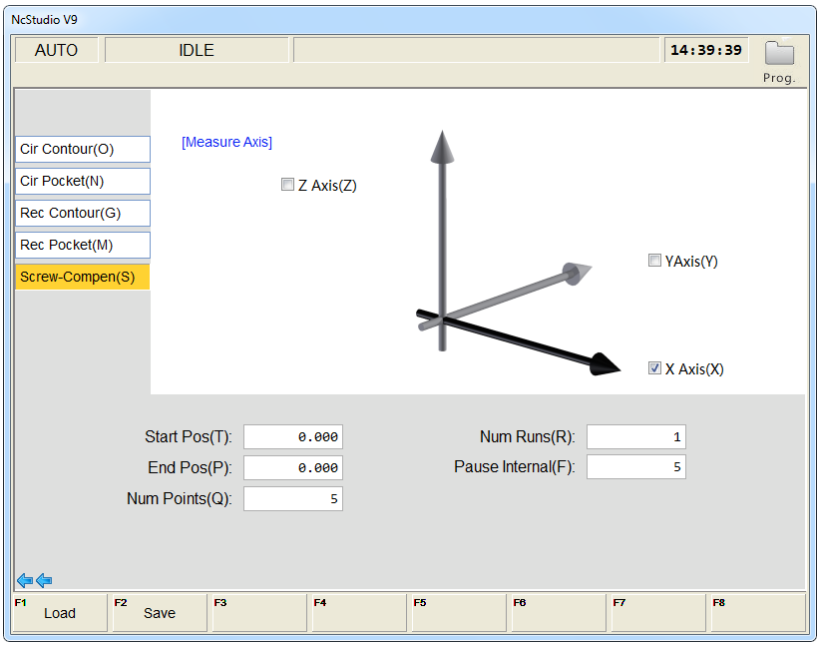


Fig. 3-77 Screw compensation screen

Or you can directly press F1 after setting parameters to save and load files. After the first time setting, if you modify the parameter values and press F1, a prompt will pop up, as shown in Fig. 3-78. Select “Yes” to save and load the newly generated file.

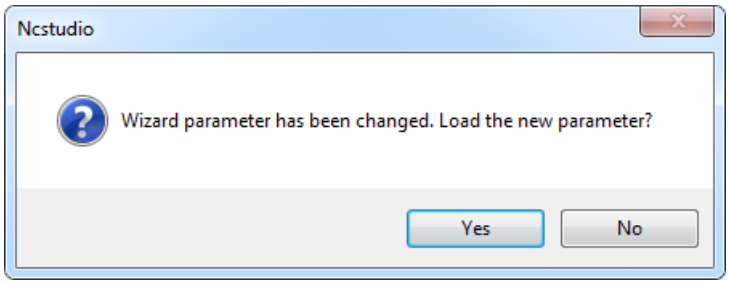


Fig. 3-78 Prompt for new parameter file

CAUTION

1) To begin with the operation, an axis should be selected, besides, X/Y/Z axis can only be selected alone at one time.
2) The “Start Pos” and “End Pos” should be located within the travel range and the latter must be larger than the former in absolute value.
3) One cycle refers to the process from the starting position to end position, during which, interferometer will record a group of data. However, a mean value will be used when written into the screw error file.
4) Measuring interval = (End position-Start position) / (Num of measuring points-1). To get an accurate measuring result, the starting position and end position should be calculated precisely and the number of measuring points should be an integer.

3.15.2 Program File

Pressing the program function selection key  will enter the machining file screen, and then pressing F1\F2 will switch between local disk program and removable disk program. See Fig. 3-79.

◆ **Local Program**

Press F1 to open a list of local program files, under the root directory D:\NcFiles. The upper part of this screen is a file list box, while the lower part prompts the path of the currently selected file and available space of the driver. To load a file into the system, press "↑" or "↓" to move the cursor onto the desired file, and then press "Enter", the loading progress displayed on the information bar. At the same time, the system will check the file being loaded. If an error found in the file, a specific prompt about the error will be displayed on the information bar. After successful loading, other operations can be executed.

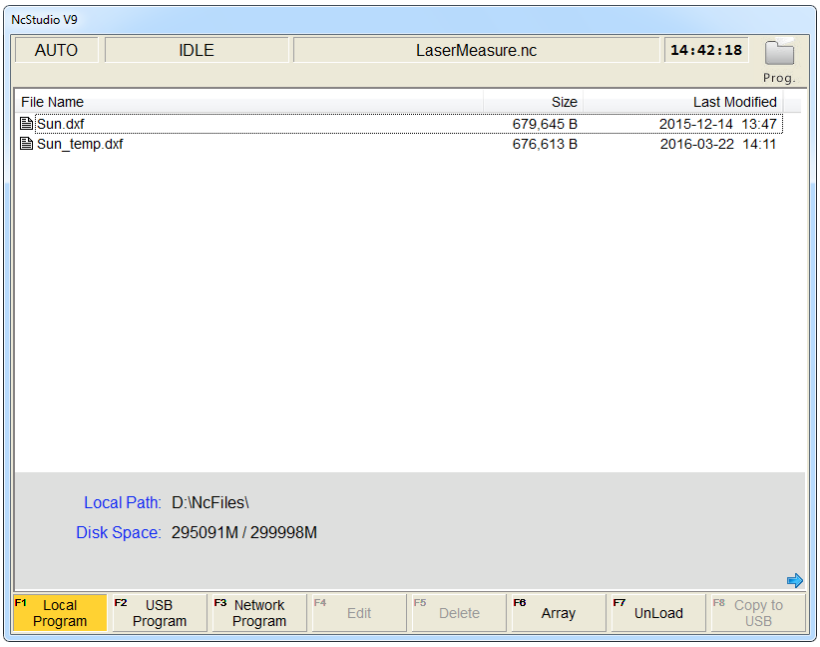


Fig. 3-79 Local program list screen

You can find the machining files under the default path of the hard disk (D:\NcFiles) and execute such operations as load, edit, delete and rename, etc. on them. In addition, you can create a new machining file under the default path and edit it.

◆ **USB Program**

Press F2 to open a program file list of removable disk where the program files under the root directory and subdirectory folders of USB disks, as shown in Fig. 3-80. The operation in the interface is similar to that in Local Program.

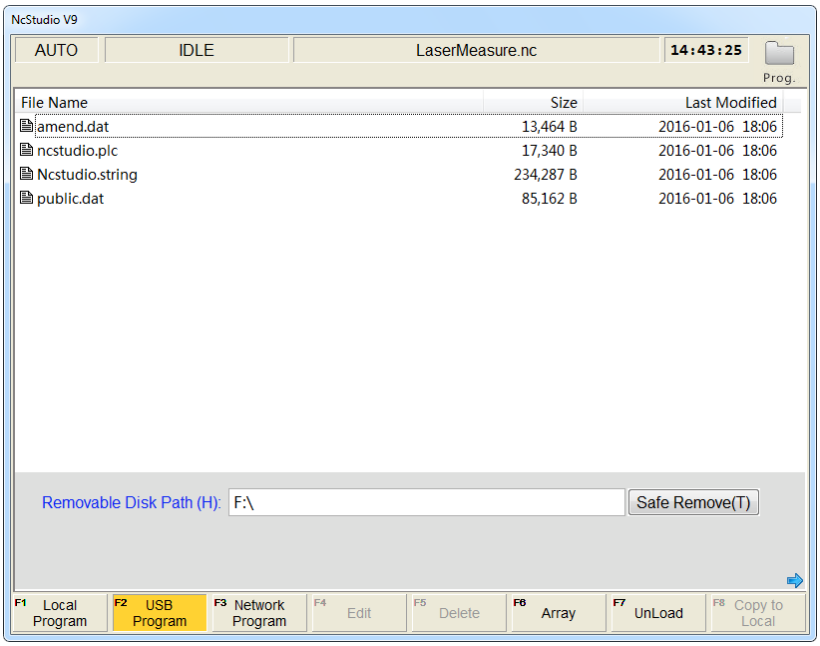


Fig. 3-80 USB program interface

◆ Network Program

Press F3 [Network Program]to visit among several machines sharing the same LAN are supported. Tool path files on the local directory or shared within the LAN are displayed in this interface, select button “Path(N)” to access files on the local directory or shared within the LAN, as shown in Fig. 3-81. The operation in the interface is similar to that in Local Program.

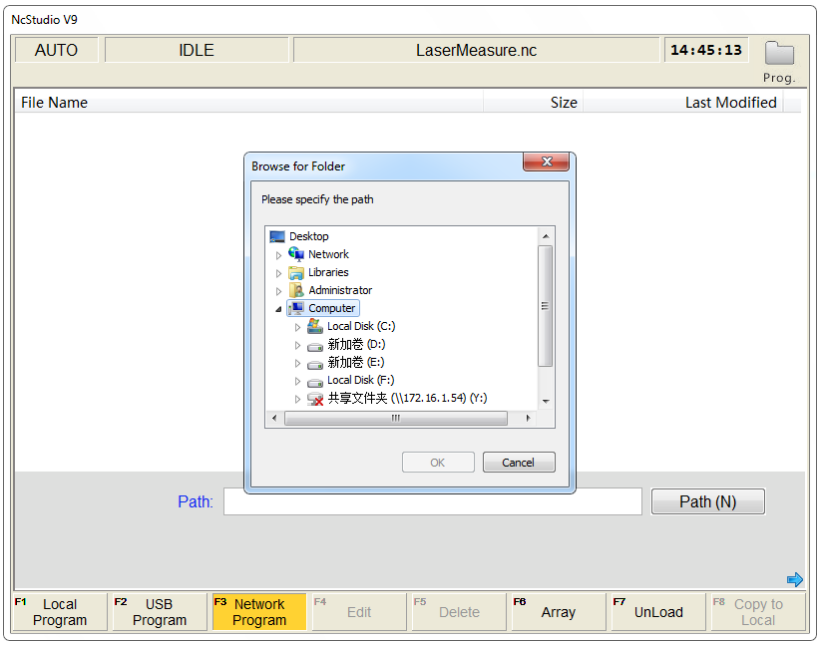


Fig. 3-81 Network program

CAUTION

1) Folder NcFiles is the default folder for sharing files or open files, for convenience of easy access to several machines.
2) To avoid mal-operations, delete function is beyond available for network programs. If you need to make any modification to files on the network, you can copy it to the local first before any further operations.
3) When the network is disconnected, program files loaded from the LAN network to the local (not copy) will be un-readable after power off or restart of the system or the software.

◆ Edit

After a machining file is selected, pressing the shortcut key F4 will make the system eject its embedded program editor automatically, in which you can do the following operations to the file, like “insert line”, “delete line”, “copy line”, “goto line”, “find”, “replace” and “save”.

After selecting a file, press F5 to eject a prompt box asking whether to delete the file.

CAUTION

1) Currently loaded file cannot be edited. Unload it before editing if necessary.
2) If the selected file is under the state of being loaded, edited or processed, deleting it is prohibited.

◆ Array

This function executes array machining for a machining file. Pressing the shortcut key F6 will display a new manipulation button bar, as shown in Fig. 3-82.

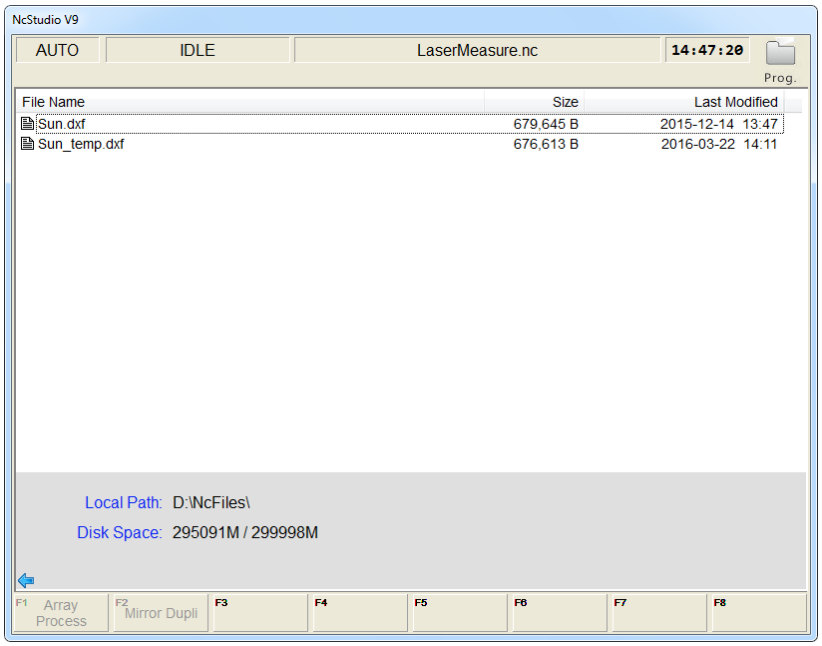


Fig. 3-82 Array machining

Press F1 “Array Process” to open a lower hanging dialog box, where you can set the rows, columns, row space and column space, etc. After parameter settings, press F7 “Generate” to generate a file, whose name can be user defined. After confirmation, the new generated file will appear in the programs list.

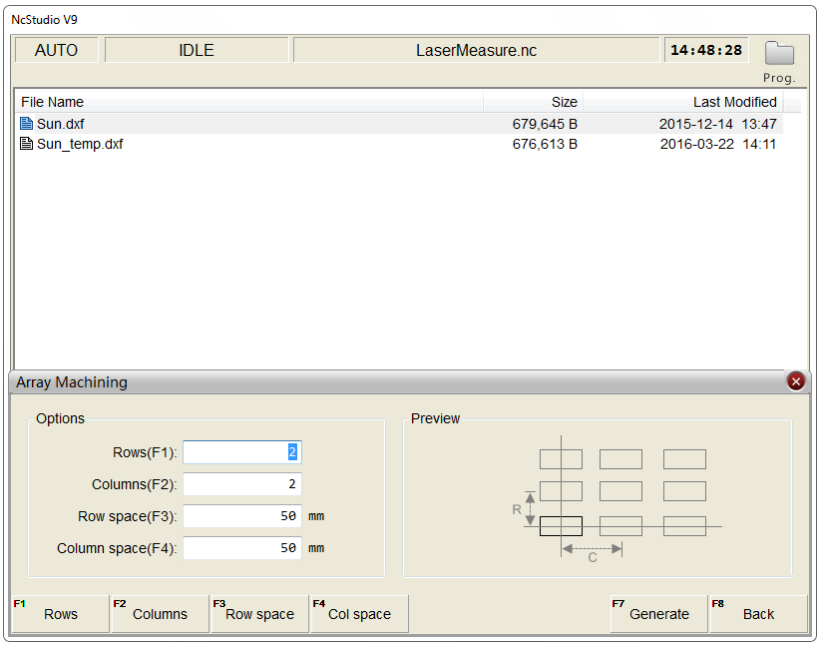


Fig. 3-83 Array machining process

In Fig. 3-82, press F2 “Mirror Dupli” to enable image mirror function. As shown in Fig. 3-84, you can set the “Axis X Mirror(F1)”, “Axis Y Mirror(F2)” and “Position Offset(F3)”. After setting, press “F7” to generate a file, whose name can be user defined. After confirmation, the new generated file will appear in the program list.

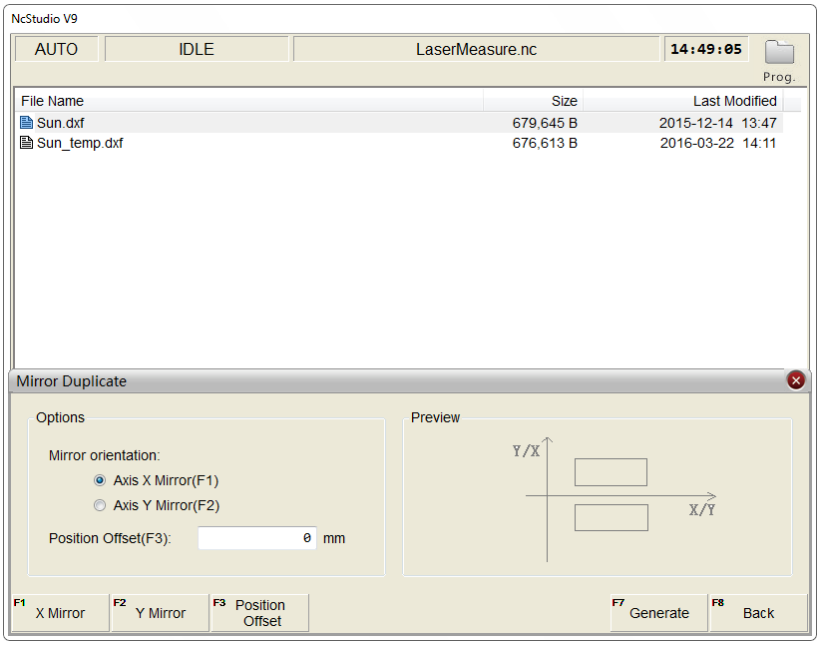


Fig. 3-84 Mirror and duplication

◆ **Unload and Copy to RemoveDisk**

Pressing the shortcut key F7 will unload the currently loaded machining file, opposite to the operation of “Load”.

Pressing the shortcut key F8 (the premise is that a removable disk has already been inserted) will copy the file selected to the removable disk.

Apart from NC files, the system also supports PLT files, DXF files and ENG files.

◆ **New**

After the shortcut key F4 (on the next buttons bar) is pressed, the system will create a .nc file under the path D:\NCFILES with the default file name “Untitle1.nc”. The system will then automatically enter the program editor for your programming.

◆ **Rename**

After selecting a machining file, press the shortcut key F5 (on the next buttons bar) to eject a file name input box. After entering the new name, press F1 “OK” to complete the operation.

3.15.3 Multi-task Machining

To meet the users’ demand in the field, multi-tasking can be enabled by simple parameter setting.

Press  ==>  ==> F1 “Software Option” ==> F8 “Manufacturer” to show parameters of MFR’s access. Find parameter N80003 “Support Program Task”, set its value to “Yes” and restart the software.

Press  ==>  ==> F3 “Task List” to open the setting dialog box, see Fig. 3-85.

The upper part of the screen shows the file list for the current machining task, while the lower part shows the files in the local disk.

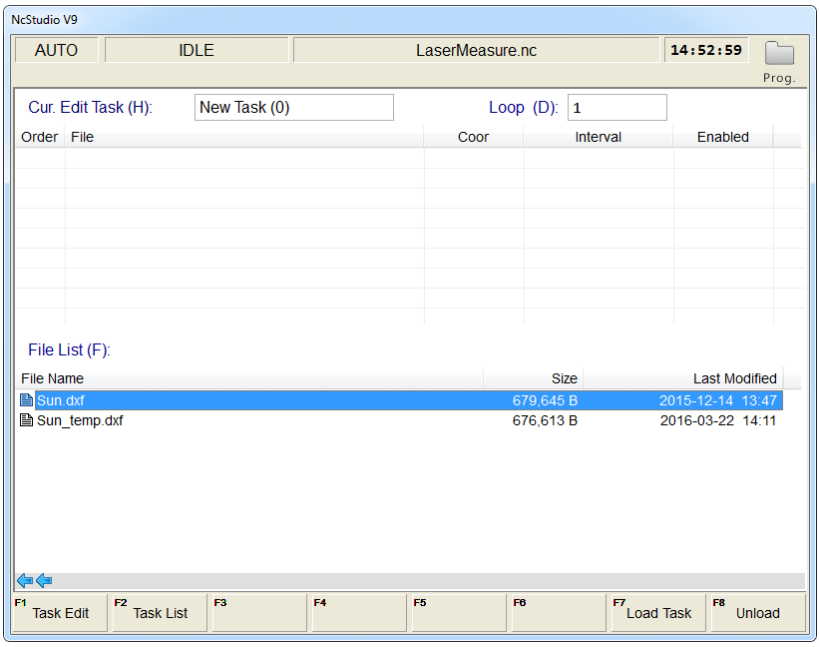


Fig. 3-85 Task List

◆ **Task List**

In Fig. 3-85, press F2 “Task List” to open a pop-up dialog box hanging over the lower part of the screen. As shown below, pressing F7 “Open Task” can open the selected task and jump to the Fig. 3-85. Pressing F1/F2/F3 can create/delete/rename a machining task.

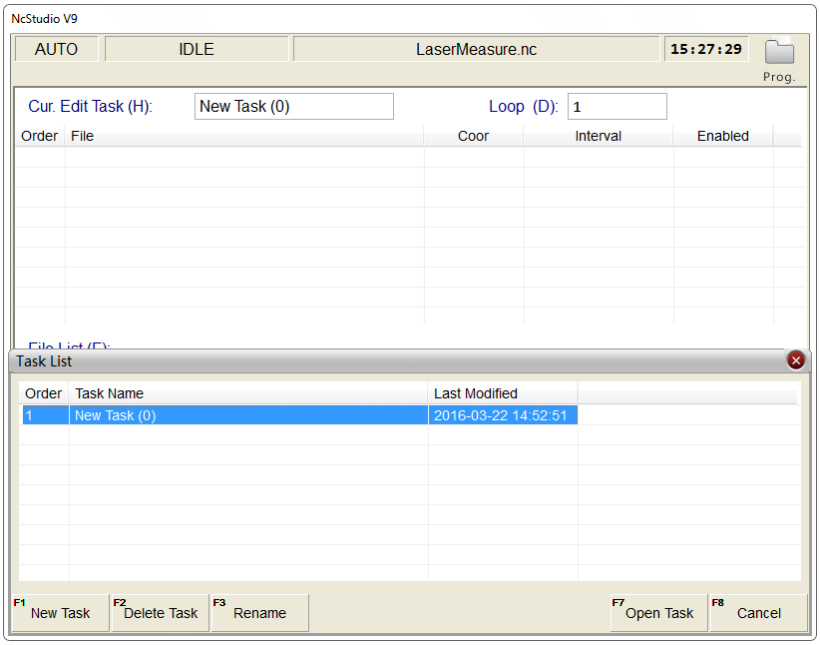


Fig. 3-86 Open task

◆ **Task Edit**

In Fig. 3-85, press F1 “Task Edit” to turn to the task editing screen, see Fig. 3-87.

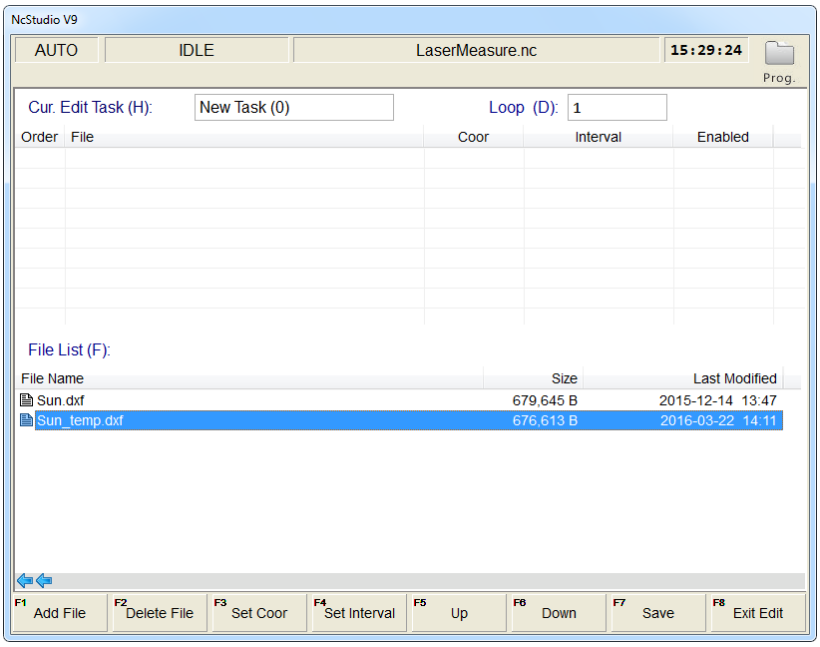


Fig. 3-87 Task editing

Select a program in the files list and press F1 “Add File”. A dialog box will pop up, as shown in Fig. 3-88, where you can select the WCS and extended WCS for the file of current task.

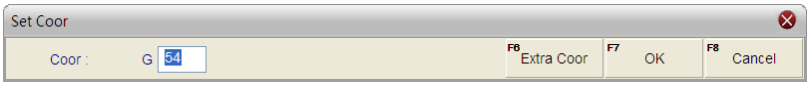


Fig. 3-88 Set WCS for file in the task

In Fig. 3-87, pressing F4 “Set Interval” can set the time period from the end of the selected program file to the beginning of next program file machining. See Fig. 3-89. Unit for the interval is second (s).

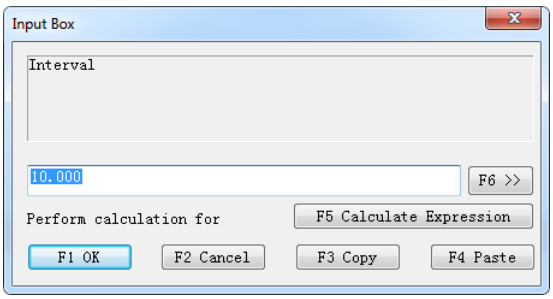


Fig. 3-89 Set interval

In Fig. 3-87, pressing F3 “Set Coor” can modify the WCS for the selected file. Pressing F5/F6 “Up” and “Down” can arrange the file in the current task list. Pressing F7 “Save” can save the settings. Pressing F8 can exit the current editing and back to main page of task list as shown in Fig. 3-85. Then press F7 “Load Task” to load the current task.

In real practices, the system will conduct machining according to the sequence of file in the task list. When one program is completed, the system will automatically check the remaining programs, wait the time period specified by “Task Interval” and continue next program machining, until all programs in the task list are completed.

3.15.4 Parameter Specification

◆ **Related Parameters**

Parameter		Meaning	Setting Range
N80003	Support Program Task	Show Program Task page in functional section Prog.	YES: show; NO: do not show

◆ **Related Parameters: PLT file translation**

Parameter		Meaning	Setting Range
N65000	Retract	It sets the tool lifting height during rapid traverse.	0~99999
N65001	PLT Units	Normally, 1plt=40.195mm, which can be magnified or reduced by setting this parameter.	0.001~99999
N65002	Tool Offset	To process the workpiece adequately, tool spacing set needs to make the parts between the adjacent tool paths overlapped based on the tool diameter. Tool offset here refers to the tool spacing in PLT file machining.	0.0001~99999
N65003	Cutting Depth	It specifies the machining depth for 2D files.	-99999~0
PLT file translation parameters are applied to translation of PLT files. PLT is a format of 2D machining files defined by an American company Hewlett Packard (HP), usually used in embossment and advertising carving, including such parameters as “retract”, “PLT units”, “tool offset” and “cutting depth”. At the same time, PLT is a kind of unit. Normally, 1plt=40.195mm, which can be magnified or reduced by setting the parameter N65001.			

◆ **Related Parameters: DXF file translation**

Parameter		Meaning	Setting Range
N65100	Retract	It sets the tool lifting height during rapid traverse.	0~99999
N65101	Cutting Depth	It specifies the machining depth for 2D files.	-99999~0
N65102	Layer Depth	It decides the cutting depth each time in 2D machining.	-99999~0
N65103	First Point as Origin	It sets whether to set the firstly met coordinate point as zero point when a DXF file is processed.	YES: Use the first point as zero point NO: Not use the first point as zero point
N65104	By Contour		YES: enabled NO: disabled
N65105	Enable Bottom Cutting	Valve operation is enabled only when [3D cutting] is on the workpiece surface.	YES: enabled NO: disabled

Parameter		Meaning	Setting Range
N65106	Use Metric	It forcibly sets a DXF file in metric size.	YES: Forcibly set in metric size NO: Not forcibly set in metric size
DXF file translation parameters are applied to translation of DXF files, including “retract”, “cutting depth”, “layer depth”, “first point as origin” and “by contour”, etc. When processing a Dxf file, the system treats the action of tool lifting as the separate mark for the adjacent shapes. If there is no tool lifting, the system will consider only one shape is being processed. If tool lifting occurs, it indicates the processing of a complete shape is finished. For example, process several circles adjacent to each other, but not overlapped. The depth of each circle is 10mm, and each feed depth of Z axis is 2mm. If parameter N65104 is set to YES, the machine tool will process the current circle 5 times, lift its tool, and then go to process the next circle. If it is set to NO, the machine tool will process the current circle once, lift its tool, and then go to process the next circle. After all the circles are processed once, this process will be re-executed 4 times to finish processing all the shapes.			

◆ **Related Parameters: ENG file translation**

Parameter		Meaning	Setting Range
N65200	Retract	It sets the tool lifting height of Z axis when a machine tool processes an ENG file in rapid traverse.	0~99999
N65201	Prompt for Tool Change	If it is set to YES, when tool change command is encountered, the machine tool will suspend machining and uplift its Z axis, and the prompt bar in the system will prompt tool change. At this time, you can perform the operation of tool change. If it is set to NO, when tool change command is encountered, the machine tool will not suspend machining, but the prompt bar in the system will still prompt tool change.	NO: disabled YES: enabled
N65203	Cutting by Tool Number	If this parameter is set to YES, opening an Eng file will eject a dialog box asking to select a tool (the tool specified in the Eng file instead of the system default tool) for machining based on the machining program.	NO: Not use YES: Use
N65204	Deep Hole Cutting Type	It sets the manner for processing deep holes.	0: Reciprocating chip removal 1: High-speed reciprocating chip removal 2: Up to safe height

Parameter		Meaning	Setting Range
N65205	Lifting Distance	It indicates the retract value after feed each time in the manner of high-speed reciprocating chip removal for deep hole drilling.	0~99999
These two parameters are related to processing of deep holes.			
N65206	Force to User Tool Compensation	Yes: Use the length or diameter compensation of tools to translate ENG when selected [Enable Cutter Compensation]; No: the length or diameter compensation commands are inactive in ENG translation.	YES: force; NO: do not force
N65207	Modify by Tool Number	With this function, ENG file machining can be executed by tool modify according to the number specified.	YES: modify; NO: do not modify
N65208	Tool Deepen Type	Tool deepen type of Z-axis when machining starts	0: deepen to moving height; 1: deepen to AreaMAx-1
N65209	Lifts when Change Tool	Whether to lift Z-axis tool before pausing tool change.	YES: lifts to moving height; NO: keep fixedly
N65210	Ignore Coordinate System Instruction	Ignore the coordinate system instruction in the ENG file.	YES: ignore; NO: do not ignore
N65211	Z Up Type after Drill	Z Up Type after Drill	0: up to R plane; 1: up to specified work coordinate position, just ENG file allowed.
N65212	Z Pos After Drill	When the value of parameter Z Up Type after Drill is 1 and the current file type is ENG, use the coordinate value.	-1000~1000
N65213	Depth Tool Speed Way	Depth Tool Speed Way	0: Machining speed way; 1: Fast across move speed way
N65215	Force G00 retract to [Lift Height]	In ENG5.5 file, when Z-axis retracts at G00, if the lifted position is lower than [Lift Height], force Z retract to position [Lift Height].	YES: force; NO: do not force

3.16 Handwheel Operation

3.16.1 Handwheel Mode

The system supports three operation modes—auto mode, reference point mode and manual mode, and manual mode is subdivided into jog mode, stepping mode and handle (handwheel) mode. You can turn the mode selection knob on the operation panel to “Handle”, i.e. to handwheel mode, as shown in Fig. 3-90.

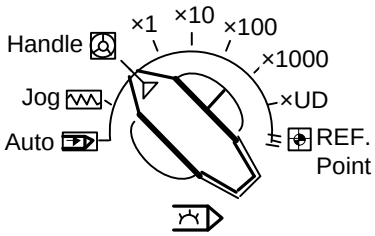


Fig. 3-90 Mode selection knob

In handwheel mode, you can configure a handwheel to control the machine tool. As shown in Fig. 3-91, select a motion axis by rotating “Axis Selection Button”, select handwheel override gear by rotating “Gear Selection Button”, and control the selected axis running at the selected handwheel override gear by rotating “Handwheel Control Rotation Disk”. Handwheel override gear regulates the displacement (linear displacement or rotation angle) of moving parts of a machine tool with each case turning of a handwheel. The displacement is set by parameters N52003, N52004 and N52005.

Fig. 3-91 A handwheel

◆ **Related Parameters:**

Parameter		Meaning	Setting Range
N52001	Precise Pulse Counting	When it is set as valid, the motion distance of a machine tool will correspond to handwheel counts strictly.	NO: disabled YES: enabled
If it is set as valid, when a handwheel turns too fast, even though handwheel stops, a machine tool will still move quite a long time since the driver receives all the pulse signals sent by the handwheel. If it is set as invalid, the system can respond to the turning of the handwheel rapidly. But if the handwheel turns too fast, the motion distance of the machine tool may not be in accordance with that indicated by the handwheel.			

◆ **And:**

Parameter		Meaning	Setting Range
N52002	Handwheel Direction	Positive/negative motion direction of a machine tool when turning a handwheel	1: Maintain the original machine motion direction in handwheel turning -1: Reverse the original machine motion direction in handwheel turning
N52003	Multiple at X1	The system will interpret 1 pulse is received when a handwheel sends 1 pulse.	0.001~10
N52004	Multiple at X10	The system will interpret 10 pulses are received when a handwheel sends 1 pulse.	0.001~10
N52005	Multiple at X100	The system will interpret 100 pulses are received when a handwheel sends 1 pulse.	0.001~10
N52010	Handwheel Acceleration	It sets the acceleration during handwheel mode (the smaller the value is, the more stable the motion will be).	1~1000
N52012	Deceleration when Switching Axis	If set to “YES”, oscillation of a machine tool may be reduced, but over-travel may occur. Otherwise, oscillation of the machine tool may occur.	YES: Decelerate; NO: Not decelerate

3.16.2 Handwheel Guide Control

NK300BX system supports handwheel guide control function.

Handwheel guide refers to a way of operation that the automatic execution speed of a machining

program is manually controlled during auto processing so as to guard against dangers caused by a wrongly loaded program or an inappropriate tool path.

In auto mode, press the handwheel guide control key 

on the operation panel. If the top-left indicator on, it means the function is activated. After machining starts, the system will execute the machining file with clockwise turning of the handwheel and stop machining with the stop of the handwheel. Machining speed varies with the handwheel turning speed.

The system also holds the function of handwheel guide reverse control. Turn the handwheel counterclockwise when an error is found in machining to make the machine tool reverse along the previous machining track. If you do not need the function, you can change the value of parameter N52013 “Forbid HW Reverse Guide” into “YES”, i.e., to forbid handwheel reverse guide function. When you turn the handwheel counterclockwise, the machine tool cannot reverse along the previous machining track and will not move.

◆ **Related Parameters:**

Parameter		Meaning	Setting Range
N52006	HW Lead Gear (Numerator)	This ratio is used to control the feed speed of a machine tool in handwheel mode.	1~1000
N52007	HW Lead Gear (Denominator)		
N52013	Forbid HW Reverse Guide	If yes, axis stops when HW is turning reversely in HW guide; otherwise, axis moves normally when HW is turning reversely.	YES: Forbid; NO: Do not forbid

3.17 System Management

To access the system info function section, press the system function selection key  . Under the function section, you can view system info, do system maintenance and switch to other configurations.

3.17.1 System Info

In the system info functions section, the default interface displays system info, including CNC software info and hardware info. Pressing F1 “Hardware” will eject a new manipulation button bar, in which “Register” can be found. See Fig. 3-92.

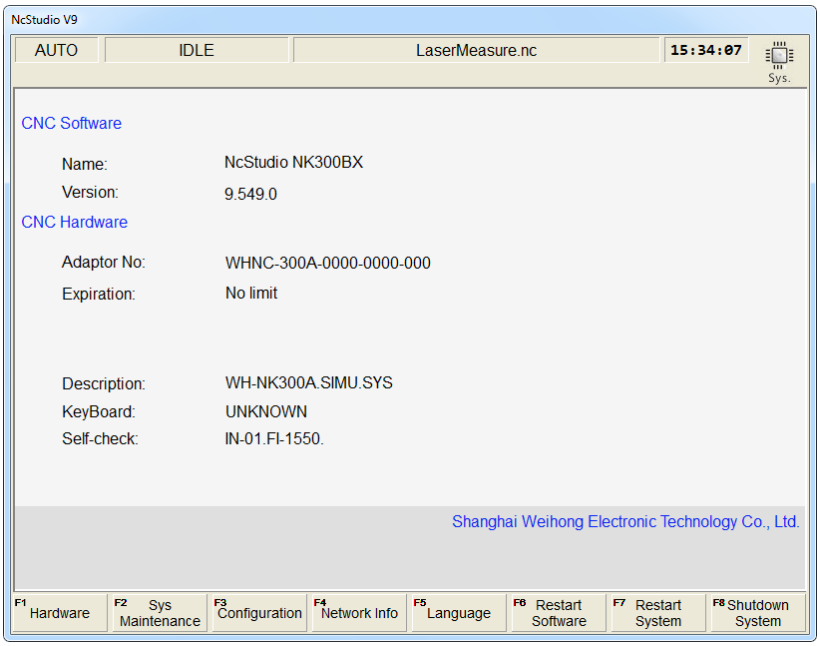


Fig. 3-92 System info

For more details about register, refer to section 3.17.4.

3.17.2 Configuration & Language Setting

◆ **Configuration Setting**

Currently NK300BX has two configurations for option—"Standard" and "Rotary Y". In addition, configurations can be added according to the user's requirements. To switch between "Standard" and "Rotary Y", press F3 "Configuration" in the system info function section. See Fig. 3-93.

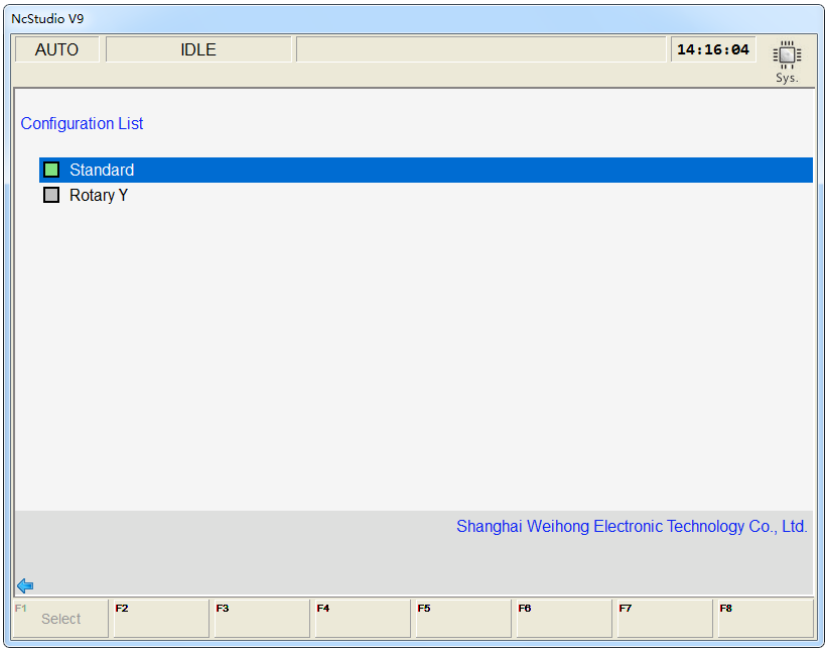


Fig. 3-93 Configuration management screen

To switch to other configurations, press " ↑ " or " ↓ " to move the cursor to the desired one, and then press

F1 “Select”, a dialog box asking “Configuration changes may cause damage to your machine, are you sure to change it” will pop out. Select “Yes”, a new dialog box showing “New active configuration has been selected. Please restart NcStudio to make it valid, do you want to go on?” will pop out. And select “Yes” again, the software will restart to enable configuration switchover. After the software restarted, you need to set relevant parameters again.

◆ **Language Setting**

To access the language setting interface, press F5 “Language” in the system info function section. At present, there are two languages for option—“Chinese” and “English”. You can select one by pressing “ ↑ ” or “ ↓ ” to move the cursor onto the desired language and then pressing F1 “Select”. The system will then give a prompt “Succeeded! Restart the software to take effect.” Press Enter for confirmation.

3.17.3 IP Setting

NK300BX supports network connection. You can obtain IP address automatically or set it manually.

◆ **Auto Obtain**

DHCP function is enabled here. In the system info function section, pressing F4 “Network Info” will access the network info interface. To access the network settings interface, press the letter key G. You can obtain IP address automatically. See Fig. 3-94. The system will obtain IP address automatically.

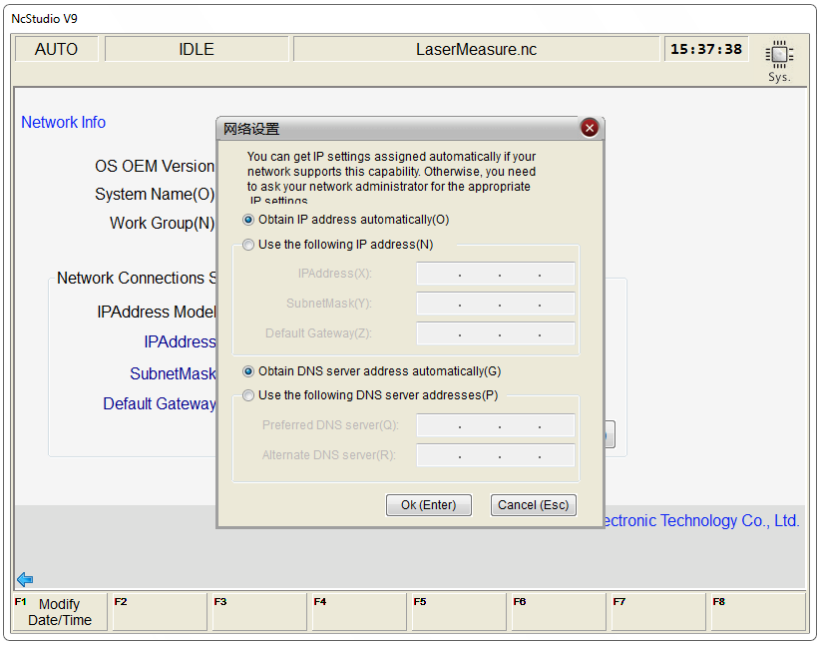


Fig. 3-94 IP address settings

◆ **Manual Setting**

In Fig. 3-94, press letter key N to manually set the IP address.

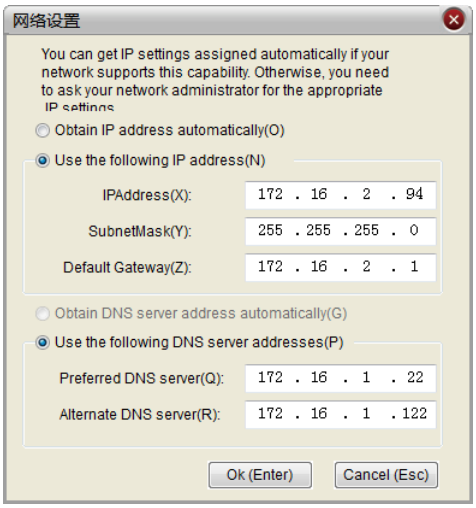


Fig. 3-95 Manual setting

As shown in Fig. 3-95, input the IP address manually:

IP address: 172.16.2.94 (within the same net range of that of the computer)

Subnet Mask: 255.255.255.0 (same with that of the computer)

Default Gateway: 172.16.2.1 (same with that of the computer)

After setting, press Enter for confirmation. And you can turn to the [Network Info] screen to view the setting.

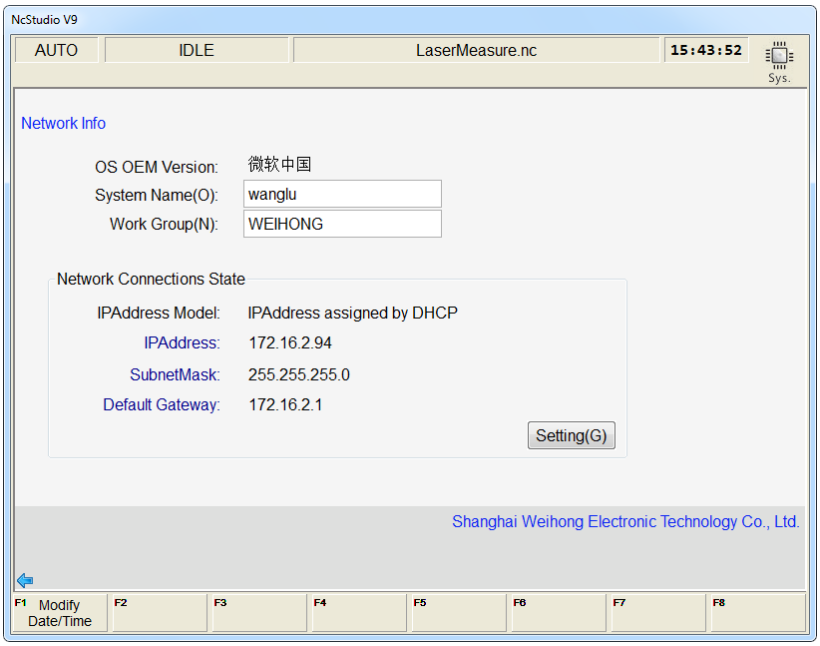


Fig. 3-96 Manually set network

CAUTION

Manual setting of NK300BX IP will reset the IP of the computer, please note that.

3.17.4 Register

“Register” under “Hardware” in the system info function section is used to decide system service time with the help of a registration code generated by the registration code maker.

The system supports register per hour or per day. Both two ways of register count service time according to system internal clocking, no matter whether the system is power off or not.

The steps to generate a registration code are as follows:

1) Turn to system info interface to get the adaptor serial number, as shown in Fig. 3-97. Or you can press F1 “Register” in Fig. 3-97 and get the serial number in the pop-up dialog box, as shown in Fig. 3-98.

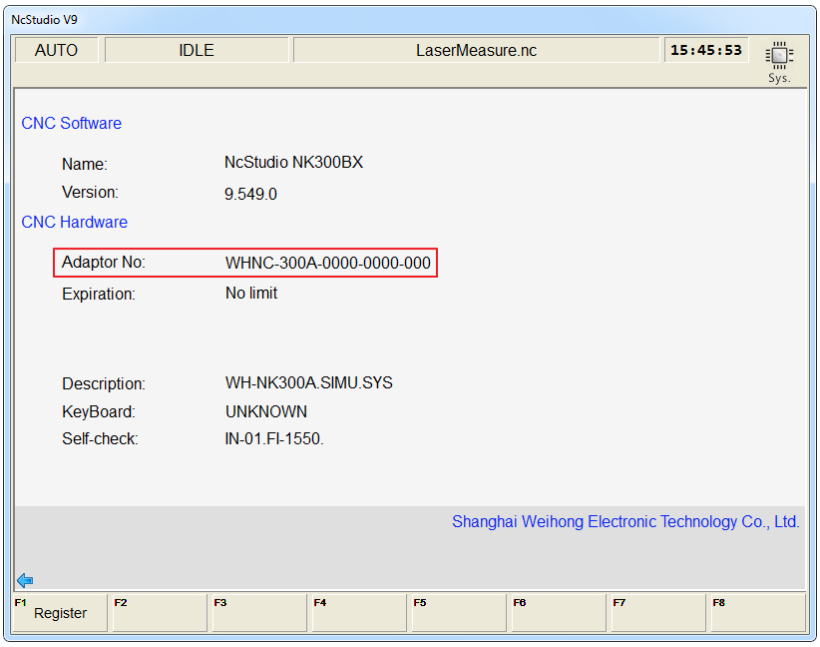


Fig. 3-97 Board serial number

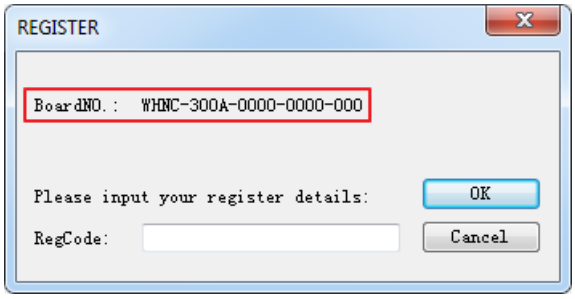


Fig. 3-98 Board serial number in the pop-up dialog box

2) Double click the registration code maker “GetRegCode.exe”, and then enter the password “ncstudio” (revisable) in the dialog box as shown in Fig. 3-99. Then press Enter, input control card serial number, registered times and limited service time, and then click “Generate” to generate a new code displayed on the lower part, as shown in Fig. 3-100. If service time does not need limiting, input “-1” in the limited service time box to generate an unlimited code.

Fig. 3-99 Registration code maker-1

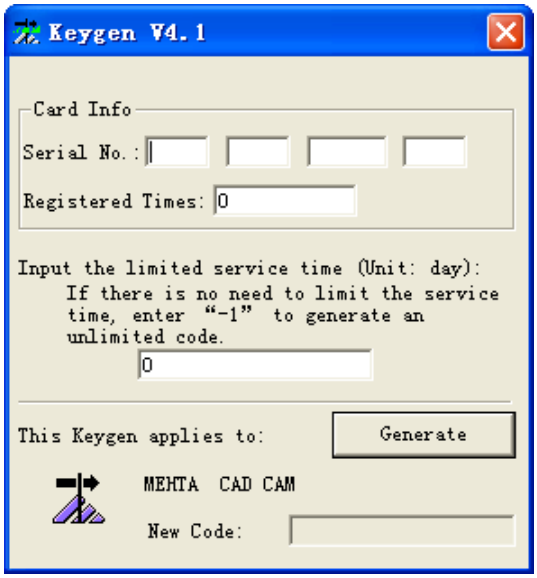


Fig. 3-100 Registration code maker-2

3) Press F1 "Register" under F1 "Hardware" in the system info function section, and then input the registration code generated in the first step into the input box, and then press Enter.

4) The system prompts "Register successfully".

CAUTION

1) Registration code maker are owned by machine tool builder.

2) ID of board varies as the registration times increases, which can be tell from the last three number of the serial number. For example, when registration times is 0, the last three number of the SN is 000, when registration times is 1, 001.

3.18 Auxiliary Function

3.18.1 Single Block Execution

You can set the machining task to be executed in single step mode, facilitating error diagnosis and failure

recovery. Once in single block mode, the system stops machining when resultant velocity of each axis is “0”.

When the single block key on the operation panel is pressed, the system will only execute the machining file for one line each time the START button is pressed, and then enter into the pause state. To go to the next line, you need to press the START button again.

3.18.2 Back to Workpiece Origin

The origin of WCS (workpiece coordinate system), i.e. workpiece origin, is fixed with respect to a certain point on the workpiece, while mobile with respect to machine origin. The selection of workpiece origin should meet the demands of simple programming, easy dimension conversion and small caused machining error, etc.

To back to workpiece origin, press the general function selection key  , and then press F7 “Home” will make the spindle return to workpiece origin automatically from the current position.

3.18.3 Jiggle

If machining is found not in position in auto machining, suspend machining and execute manual jiggle. Jiggle result is only available for the machining task this time, and becomes ineffective after machining stops.

Jiggle function can be found by pressing  ==>  ==> F3 “Jiggle”. After pressing the Pause key in auto machining, press F3 “Jiggle” to access the jiggle interface, as shown in Fig. 3-101, in which set a proper step, and then press an axis direction key to jiggle the corresponding axis. After satisfying jiggle result is obtained, press the START key to continue machining.

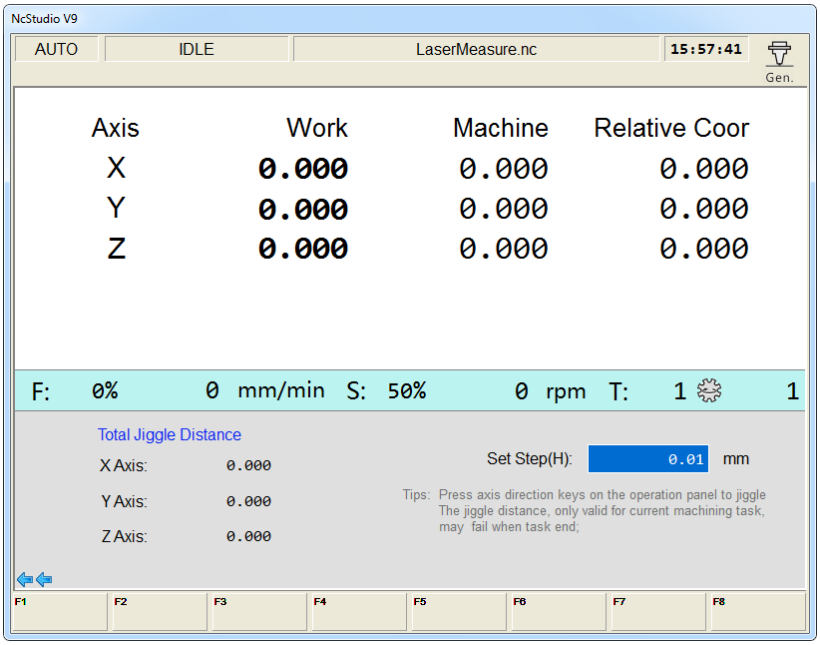


Fig. 3-101 Jiggle setup

3.18.4 Selective Machining

This function can select any blocks for machining.

Selective machining function can be found by pressing  and F4 “Selective Machining” to access the selective machining input box, as shown in Fig. 3-102.

Enter the desired start line number and end line number, and then press Enter for confirmation. At this time, pressing the START key will start machining the selected blocks.

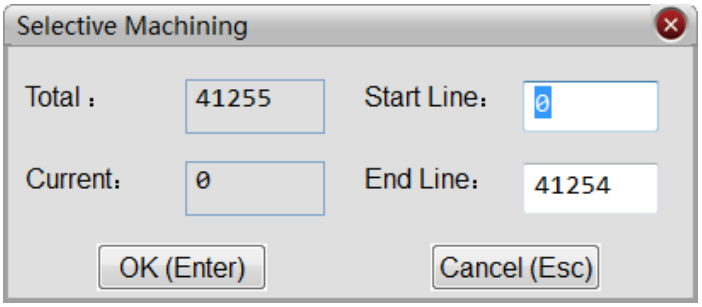


Fig. 3-102 Selective machining

3.18.5 Mirror and Rotation

This function can execute mirror and rotation on a machining file.

Mirror and rotation function can be set on the interface by pressing  . As shown in Fig. 3-103.

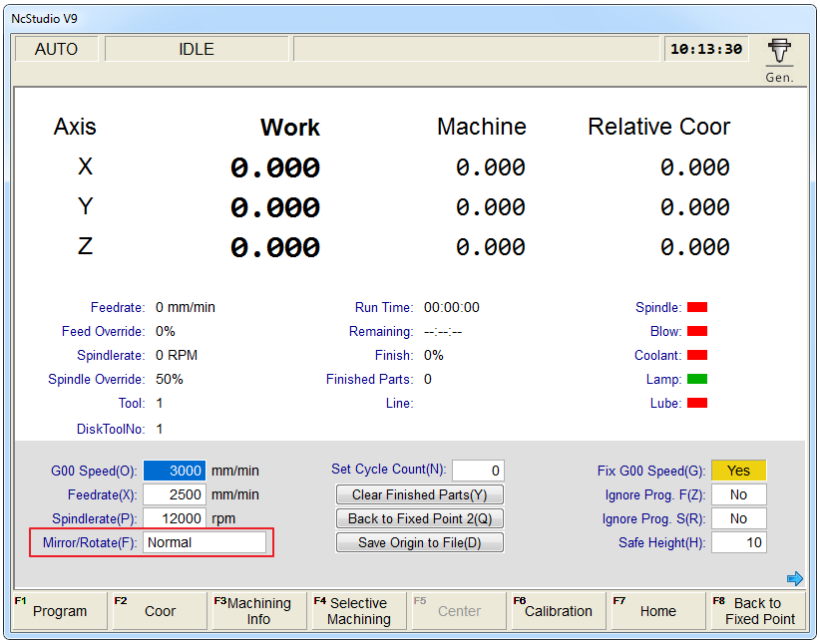


Fig. 3-103 Mirror and rotate setting on the main interface

Press letter key F to open the mirror and rotation machining dialog box, as shown in Fig. 3-104. Enter the sequence number of the desired machining mode into the machining mode input box, and then press

Enter for confirmation.

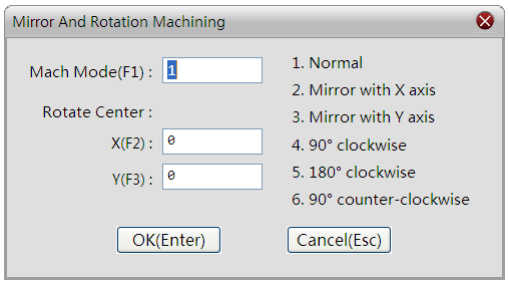


Fig. 3-104 Mirror and rotation interface

You can set rotating center in Fig. 3-104, if no center is specified, the workpiece origin will be the rotating center by default.

3.18.6 Breakpoint Resume

This function is executed by pressing the breakpoint resume key on the operation panel, which will make the system resume machining from the stop line number of last time machining automatically.

If power failure or emergency stop occurs during machining, and you are sure about the accuracy of the workpiece coordinates, you can execute this function to make a machine tool rapidly move to the breakpoint and resume machining, to save machining time.

3.18.7 Workpiece Length Sensing

Workpiece length can be sensed by the system.

The system senses workpiece length by workpiece coordinates. For instance, to sense the workpiece length in the X direction, the steps are as following:

1) Press  ==>  ==> F1 “Switch Coor” to switch to “Relative Coor Mode”;

2) Manually move the X axis to one side of workpiece, and then press F2 “Work” and F1 “Clear X Relative” in turn;

3) Manually move the X axis to the other side of workpiece. Workpiece length in the X direction is the X axis “Relative Coor” displayed on the interface.

CAUTION

Relative clear has no effect on absolute coordinates and machine coordinates, so you can still use the original coordinates for machining.

3.18.8 Parameter Auto Backup

The system boasts the function of parameter auto backup. If you forget to save the set parameters, you

can switch to this screen, in which you can restore parameters from the ex-factory date to system last shutdown.

Parameter auto back screen can be found by pressing the parameter selection key  ==>  ==> F2 “Param Auto Backup”, as shown in Fig. 3-105, in which press the Up or Down key to move the cursor to an active backup and press F1 “Restore” to restore the selected backup parameters.

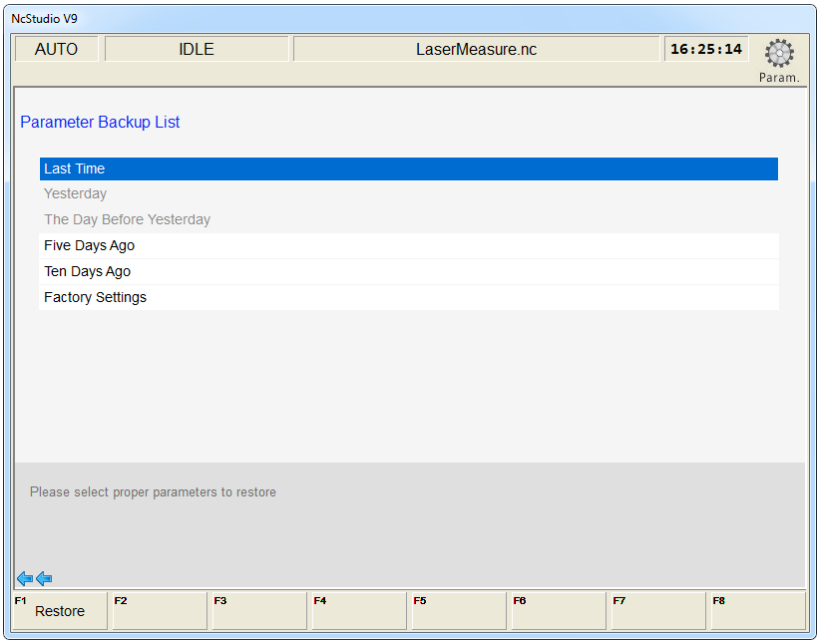


Fig. 3-105 Parameter auto backup

3.18.9 Manual Data Input (MDI)

Press  ==>  ==> F4 “MDI” in turn to access the MDI interface. See Fig. 3-106.

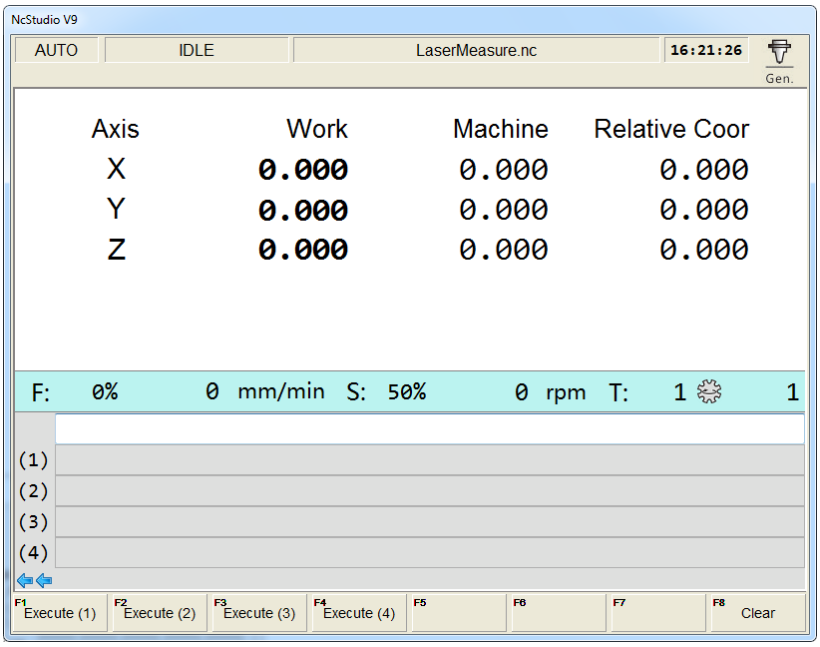


Fig. 3-106 Manual data input screen

At the upper part of the screen, displayed are machine coordinates and workpiece coordinates of each axis, while at the lower part of the screen, there are 4 input boxes for entering instructions. The newly entered instruction is at the top.

After entering the new instruction in the white input box, press Enter, and then press F1 “Execute (1) to execute the entered instruction. The system can save 4 newly entered instructions, and pressing F1, F2, F3 or F4 will execute an instruction in the corresponding input box.

3.19 Tool Magazine

3.19.1 Auto Tool Change of Linear Tool Magazine

Linear tool magazine stores tools in the form of array. For example, if a customer has 12 tools, he can select a 1-line 12-row tool magazine, or a 2-line 6-row tool magazine, etc. Auto tool change is realized by programming in the public.dat file according to the related information learned from the customer. The process of auto tool change for a linear tool magazine is as Fig. 3-107:

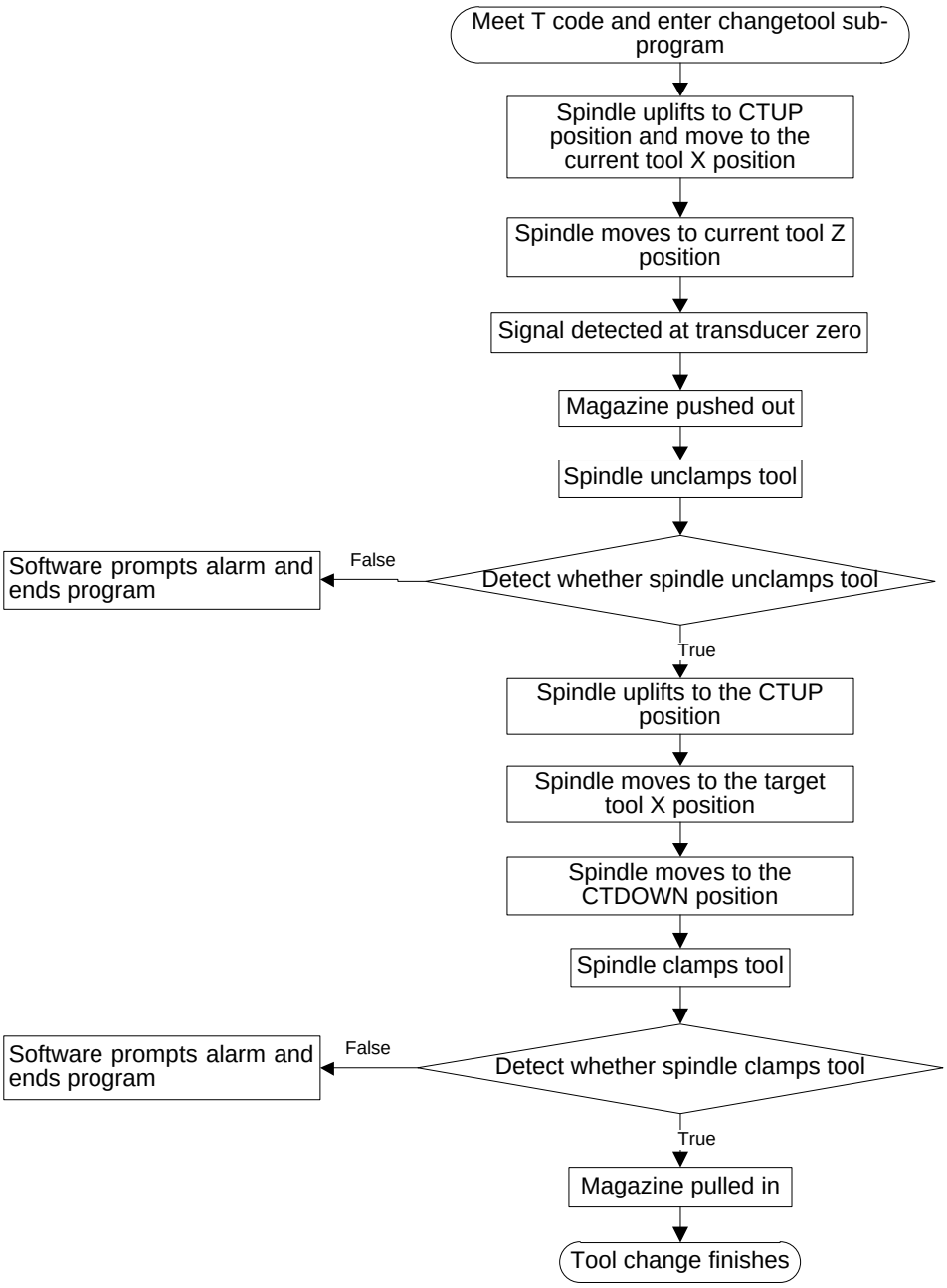


Fig. 3-107 Process of auto tool change for a linear tool magazine

3.19.2 Auto Tool Change of Circular Tool Magazine

When a machine tool is with the function of a circular tool magazine and auto tool change is needed during file machining, the process of auto tool change is as Fig. 3-108:

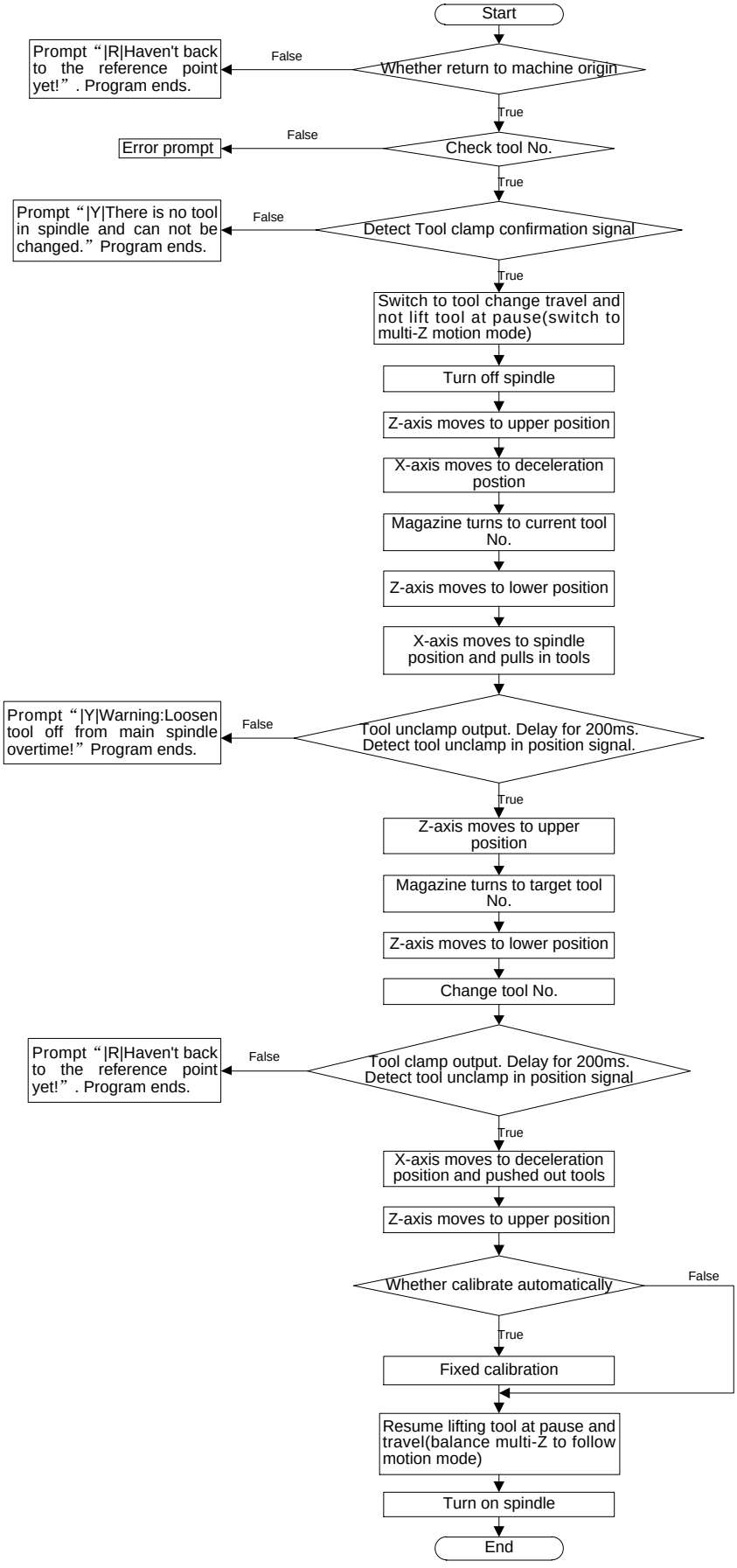


Fig. 3-108 Process of auto tool change for circular tool magazine

3.19.3 Parameters Specification

◆ **Related Parameters Are:**

Parameter		Meaning	Setting Range
N66074	Group 1 Tool Position (X)	Machine coordinate value of X axis of No. 1 tool.	-
N66075	Group 1 Tool Position (Y)	Machine coordinate value of Y axis of No. 1 tool.	-
N66076	Group 1 Tool Position (Z)	Machine coordinate value of Z axis of No. 1 tool.	-

Many tool coordinate positions are provided for selection, which will not be listed here.

◆ **Related Parameters Are:**

Parameter		Meaning	Setting Range
N66020	Tool Magazine Type	0: No Tool Magazine; 1: Disk Tool Magazine; 2: Linear Tool Magazine	0; 1; 2
N66021	Tool Magazine Capacity	The capacity of tool magazine.	1~255
N66022	Check Change ToolNo	Whether to check tool number in tool change is proper or not.	YES: The tool number must be within (0, 256) in tool change command. NO: The range of the tool number is not limited, and the tool number remains the same.
N67000/N67001/ N67002	Change Tool Workbench Range Lower Limit X/Y/Z	Machine coordinate value of worktable stroke lower limit of X/Y/Z during tool change.	-
N67010/N67011/ N67012	Change Tool Workbench Range Upper Limit X/Y/Z	Machine coordinate value of worktable stroke upper limit of X/Y/Z during tool change.	-
N67020	Enable Change Tool Travel Limits(MCS)	-	YES: enabled; NO: disabled.
This group of parameters sets the worktable stroke range for tool change to avoid tool damage caused by over travel during tool change.			

◆ **Related Parameters**

Parameter		Meaning	Setting Range
N66000	Prompt for Tool Change	Whether to pause and prompt tool change when meeting tool change command.	NO: disabled; YES: enabled
N66002	Pause in ToolChange for Same Active and Target Tool No.	On Condition that N66000[Prompt for ToolChange] is enabled and this parameter set to YES, machine will pause when reach the same T No. command in tool change. When set to NO, machine will not pause if the active target T No. is the same.	YES: Pause; NO: Do not pause
N66005	Upper Position	Z-axis machine coordinate when a tool moves to tool magazine for tool change, or CTUP position	-99999~99999
N66006	Lower Position	Z-axis machine coordinate of tool change position when a tool moves downwards from Upper Position, or CTDOWN position	-99999~99999
N66007/N66008/N66009	Spindle Position X/Y/Z	Machine coordinate value when spindle changing a tool, usually used for circular tool magazine parameter setting.	-99999~99999
N66010/N66011/N66012	Deceleration Position X/Y/Z	Machine coordinate value of spindle position before tool change	-99999~100000
N66015	Tool Change Speed	Movement speed of spindle during tool change	0~100000
N66016	Z Axis Speed	The speed of Z axis moving to Upper and Lower Position during tool change	0~13740
N66017	Automatic Tool Measure	Whether to execute auto tool measurement after tool change	NO: Not execute YES: Execute

CAUTION

Since the machine structures of tool magzines vary, the above tool change flow charts apply only for general situations. Please contact with the manufacturer for part adjustment if the actual operation is diiferent from the operation described above.

4 Maintenance

4.1 Operation System Maintenance

When you receive product NK300BX, it is ready for use with all systems installed and setup. If any failure occurs, you can restore the software to leaving-factory state.

4.1.1 Preparation

1) An USB flash disk (above 1G)
2) The backup and restore kit

4.1.2 Creating a Setup Disk

◆ **Creating a Startup Disk with an USB**

You can create an USB startup disk which will help the system access DOS interface, backup and restore the system SSD with DOS tools in the USB disk.

Steps to create USB startup disk are as follows:

1) Insert an USB into the PC, and double click file “HPUSBFW.EXE" in file folder “hpUpgsh” on the desktop of PC. An interface as shown in Fig. 4-1 will pop up.

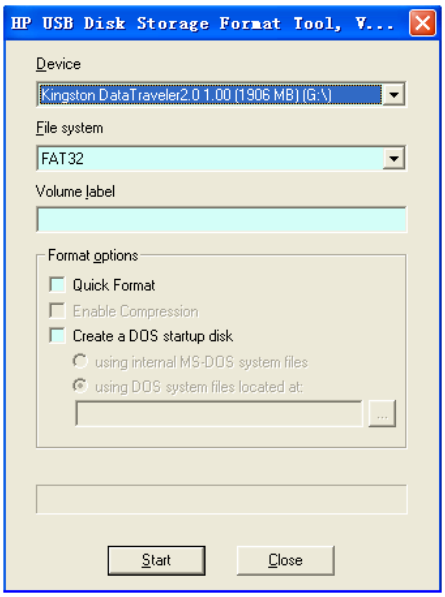


Fig. 4-1 USB format tool

- Under “Device”, select the USB flash disk which needs to be formatted; .
- The default for “File system” is FAT32;
- Select items “Quick Format” and “Create a DOS startup disk;
- Below the item “using DOS system files located…”, specify and locate the path “Desktop\hpUpgsh\boot”.

- Click “Start” to initiate formatting. After two successive confirmations, creation of USB startup disk is successfully completed.

2) On the desktop of PC, double click the application “USB backup and restore tool kit”, and interface as shown in Fig. 4-2 will appear.

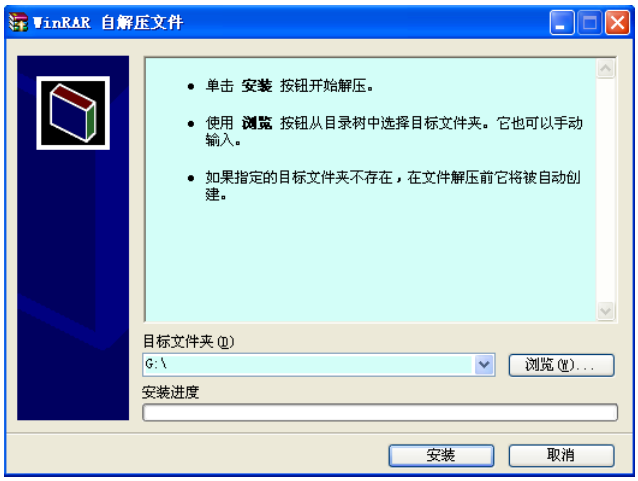


Fig. 4-2 USB restore tool kit

3) Locate the USB startup disk which has been created successfully in the target file box, and then click “Install”. After installation, all files contained in “USB backup and restore tool kit” will be unzipped into this USB. .

4) Conduct anti-virus check on the USB to secure it is safe from viruses.

4.1.3 Restore System

Restore system refers to the process of mirror image installation of the system SSD. USB well created before and a system SSD is needed.

1) Insert the USB flash disk to the USB slot.

2) Restart the system, and press Delete key to enter BIOS interface. Accessing “Boot→Hard Drive BBS Priorities→Boot Option #1”, set USB setup disk as “Boot Option #1”.

3) After setting start orders, press F4 to save the setting and restart. After normal startup, interface as shown in Fig. 4-3 will appear.

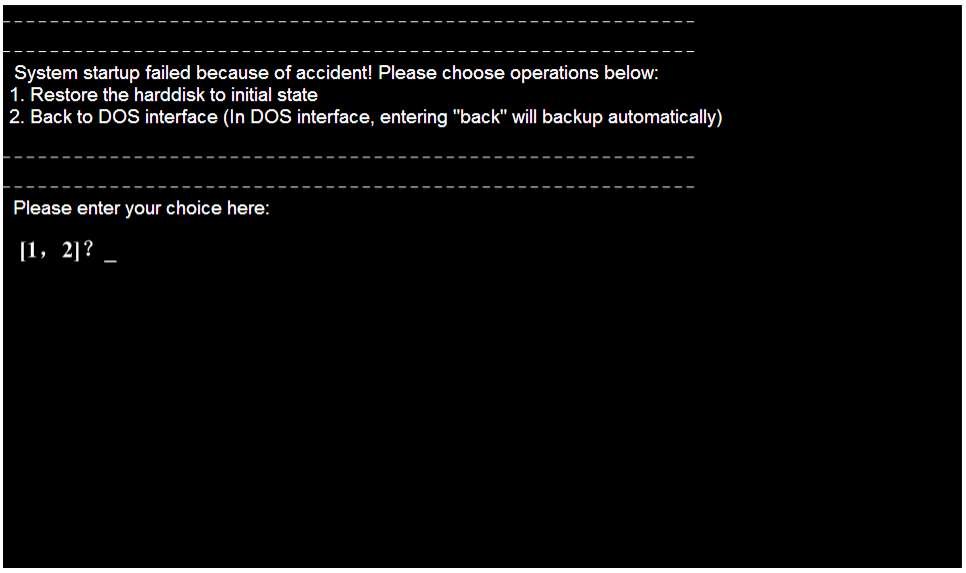


Fig. 4-3 DOS startup interface-1

4) Enter “1”, and interface shown as Fig. 4-4 will pop up.

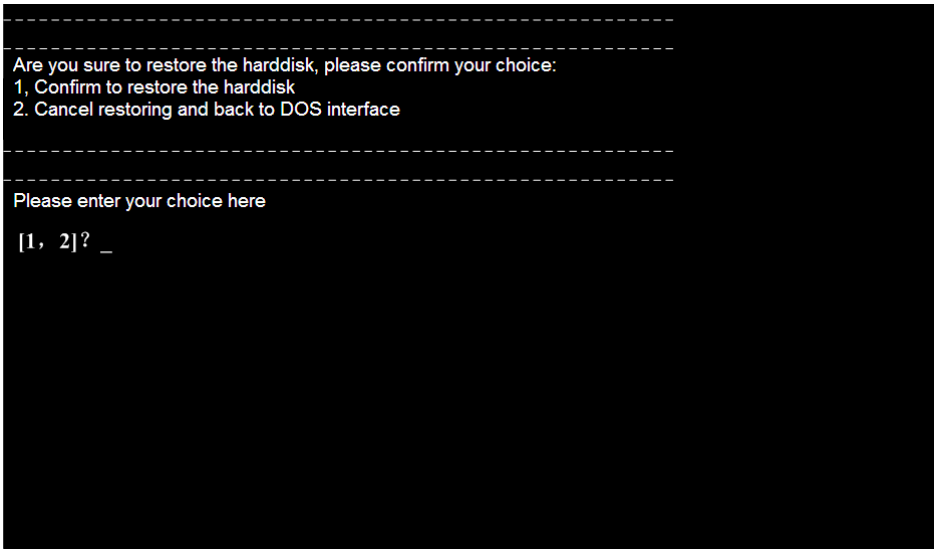


Fig. 4-4 DOS startup interface-2

5) Enter 1 again, and the system will execute Ghost restoration. Pull out the USB disk the moment the system restarts. System installation is completed.

4.1.4 Backup OS System to USB

Back up the system to USB disk as follows:

1) Insert the USB flash disk to the USB slot.
2) Restart the system, and press Delete key to enter BIOS interface. Accessing “Boot→Disk BBS Priorities→Boot Option#1”, and set USB setup disk as “Boot Option #1”.
3) After setting start orders, press F4 save the setting and restart. After normal startup, interface as shown in Fig. 4-5 will appear.

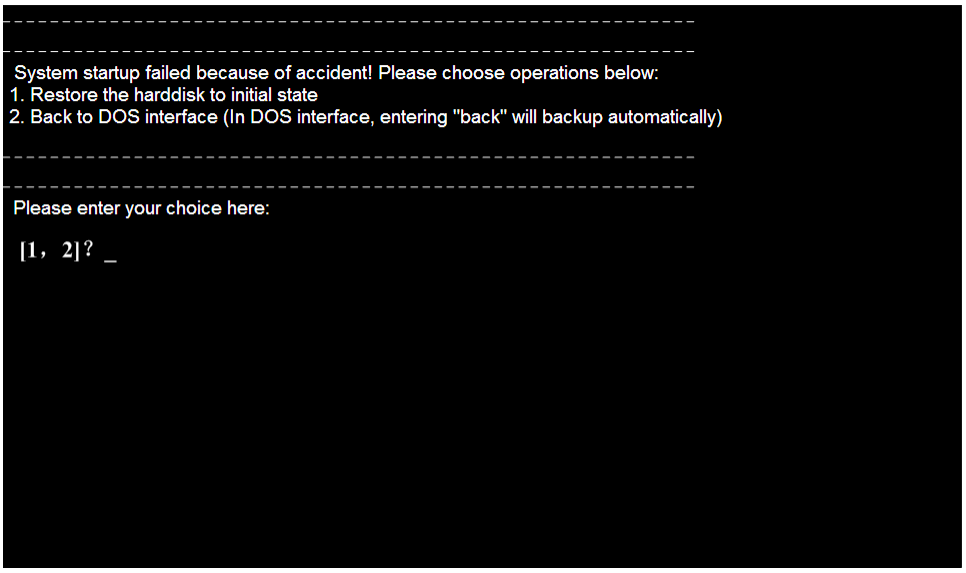


Fig. 4-5 DOS startup interface-3

4) Enter “2 →back → 2”, and the system will execute Ghost backup process. Pull out the USB disk and turn off the power when the process finishes. System backup is completed.

4.1.5 System Re-backup and Restoration

Backup of operating system of NK300BX has been completed before leaving factory. You can use “Windows Ghost” Restoration function to restore the system when problems occur. In addition, the software has also been installed in CNC system, but its backup is not done in CNC system. We suggest that you immediately back up the BIOS system and the software again the first time the machine is power on or after debugging is completed.

◆ **System Re-backup**

Steps to back up the system again are as follows.

1) Power on the machine, access the operating system choice interface, and select “Windows Ghost”, as shown in Fig. 4-6.

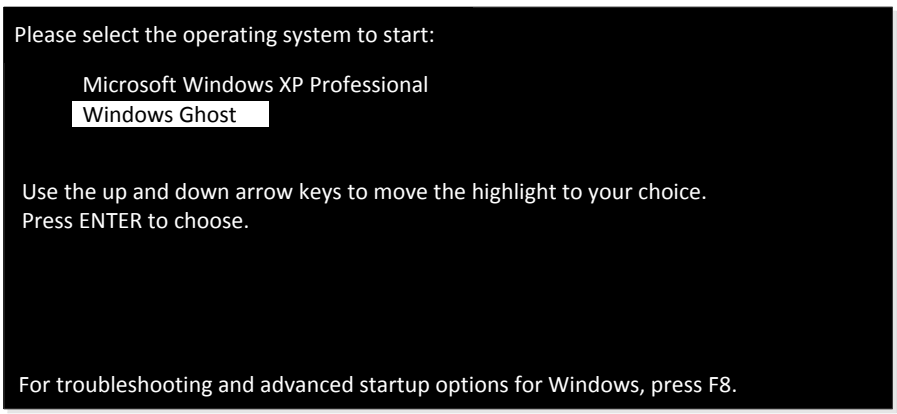


Fig. 4-6 Select “Windows Ghost” operating system

2) It jumps to restoration confirmation interface, as shown in Fig. 4-7. Select “Cancel” and press “Enter” to access the operation interface as shown in Fig. 4-8.

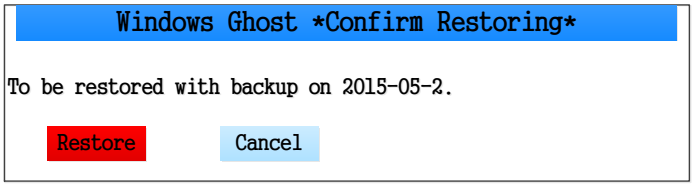


Fig. 4-7 Restoration confirmation

3) Select “2 Re-Backup” as shown in Fig. 4-8.

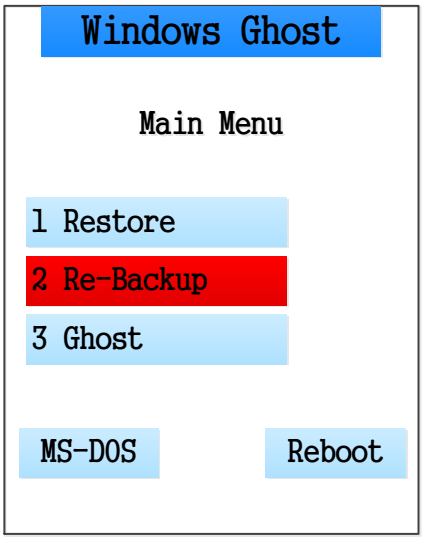


Fig. 4-8 Re-backup

4) After confirmation, backup progressing indictor will show up.

5) When it finishes, backup is completed.

◆ **System Restoration**

After re-backing up the system, if you need to use windows ghost restoration function, select “Restore” in Fig. 4-7, and do as the tips to restore to the leaving factory state.

Note that if you use windows ghost restoration function while you have not re-back up the system, the system will be re-started automatically. A dialog box titled with “FirstRun” will pop up noticing you that you should install the software, as shown in Fig. 4-9 and Fig. 4-10. You can select a disk to install the software as directed.

Fig. 4-9 FirstRun Notice 1

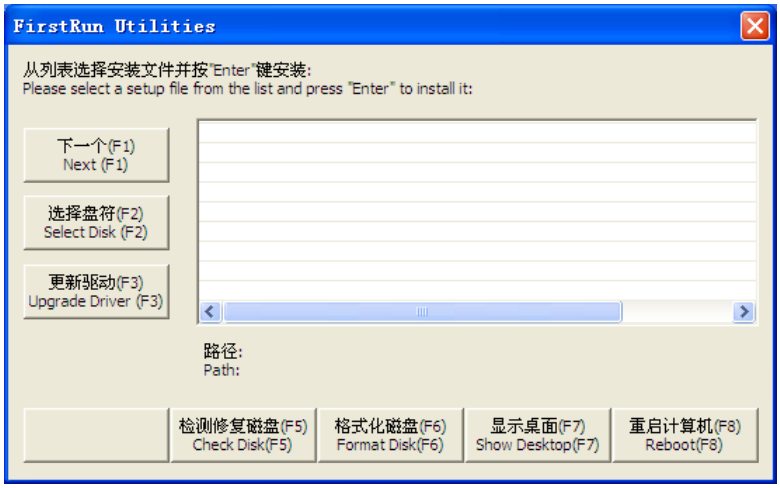


Fig. 4-10 FirstRun Notice 2

CAUTION

1) When exceptions occur during NcStudio system backup and restoration, you should consider the following causes at priority.
   - ✓ Is the guidance order of hardware in BIOS correct?
   - ✓ Is there any problem occurred during system backup?
   - ✓ Is the storage of USB disk enough during system backup?
   - ✓ The backup progress will exit automatically if image file exists in the USB disk.
   - ✓ During system backup, if there is mirror image file in USB disk, the process will exit.

To avoid the problems listed above, it is recommended that you conduct system disk security check and rerpair before proceeding with system backup and restoration. Otherwise, system performance may be influenced. So as data disk.

2) Do not power off the PC during backup of NcStudio system. Otherwise, the system can be damaged.

3) When the prompt about installing software in FirstRun dialog box appears, only .exe file is supported.Compressed files of format such as .zip, .rar are not supported. They must be unzipped for installation

4.2 NcStudio System Maintenance

4.2.1 Package and Upgrade

The system holds the function of software backup. After installing the software and setting various parameters corresponding to a specific machine tool, you can backup and save the software with proper parameter settings as the original data. And the backup software can be directly installed on a machine tool of the same type. This function is realized in system maintenance. The following will introduce the system maintenance in detail.

Press  to enter the system info function section, and then press F2 “Sys Maintenance” to enter the system maintenance interface, as shown in Fig. 4-11.

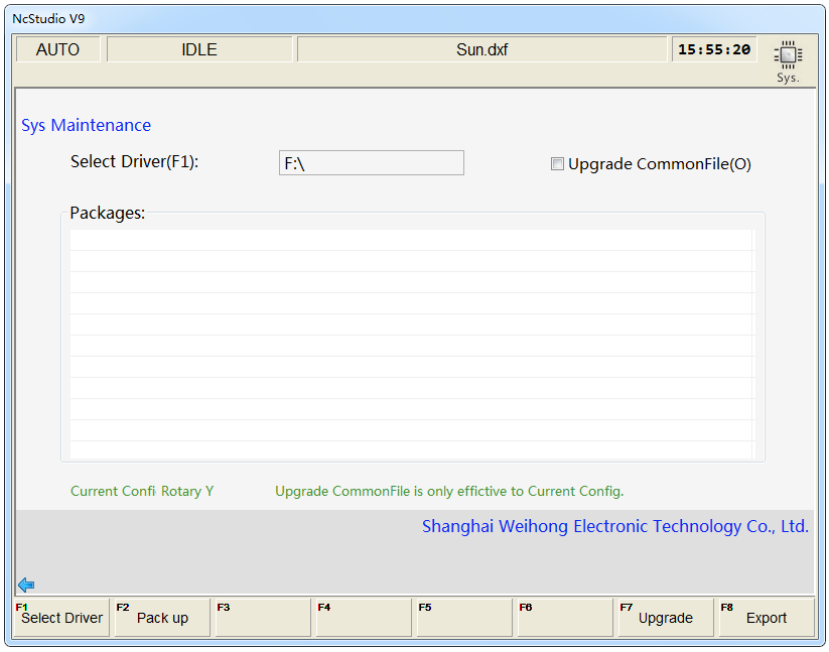


Fig. 4-11 System maintenance interface

- **Select Driver**

Press F1 “Select Driver” will eject an input box for entering the drive letter of the removable disk with the update package.

- **Pack up**

Pressing F2 “Pack up” will pack the software automatically and save the packaged software to the selected disk.

- **Upgrade**

All the update packages are listed in “Packages”. Press “ ↑ ” or “ ↓ ” to move the cursor to the desired one, and then press F7 “Upgrade” to start software installation.

- **Export**

Press F8 “Export” to export the public file such as amend.dat, ncstudio.plc, ncstudio.string, and public.dat files to removable disk. Prompt for successful exportation will pop up as shown in Fig. 4-12.

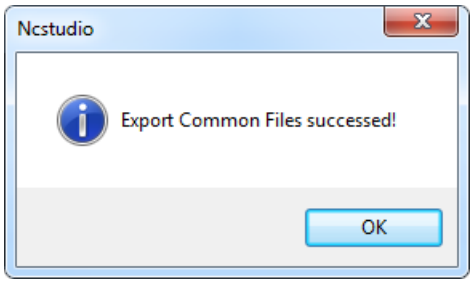


Fig. 4-12 Prompt for successful exportation

Pressing shortcut key O can display all common files for the system in the list of “Packages”, where you can select desired files to save them to the directory of removable disk. As shown in Fig. 4-13, select the common file you need to upgrade.

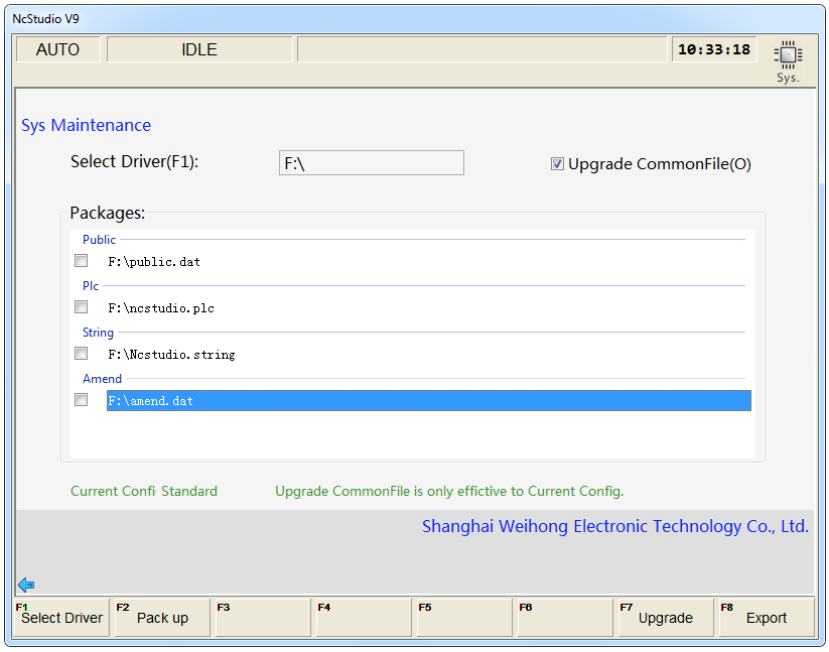


Fig. 4-13 Common file upgrade

Check in the check box in front of the public files you need to upgrade, and press F7 “Upgrade” to initiate. After confirmation, upgrading succeeds Restart the system.

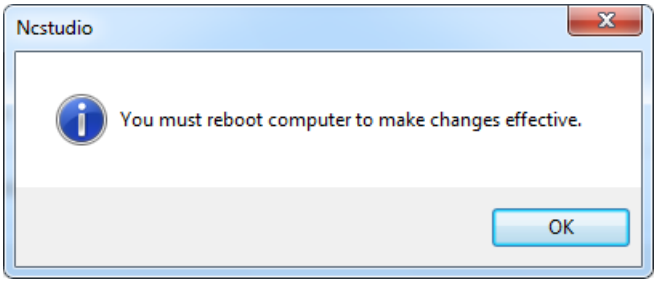


Fig. 4-14 Prompt for restarting software after updating common file

If the public files to upgrade contains amend.dat file, before final upgrading, a prompt dialog box will

show up, as shown in Fig. 4-15. Choose “Yes” to confirm and continue upgrading and choose “No” to cancel it.

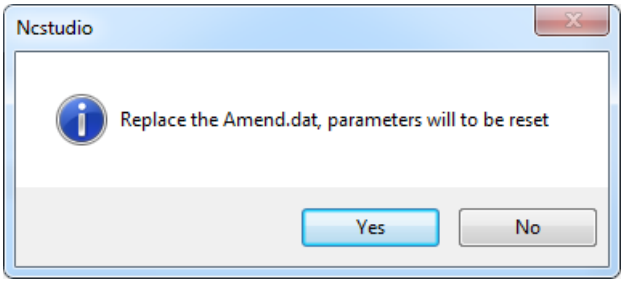


Fig. 4-15 Prompt before amend.dat file upgrading

CAUTION

Function “Upgrade Common File” is only effective for the current configuration, please notice that.

4.2.2 Software Installation

Software installation can be divided into following steps:

1) Accessing the desktop. Press combination key “Ctrl + Alt + Delete” to enter the task manager interface ==> press “Alt + F” to select “New Task” ==> in the new task dialog, input “explorer” and press “Enter” ==> press “Alt + Tab” to switch to NcStudio ==> press “Alt + F4” to close it and enter the desktop.

2) Insert the USB flash disk with the saved software into the USB interface on the operation panel of NK300BX host. Enter the desktop as operations in step one. Find the software to install in My Computer and double click it to initiate installation. First step in software installation is the language selection, as shown in Fig. 4-16, which decides the running language of the system. You can make your own choice here. Of course, language can be switched in the software later in use.

Fig. 4-16 Language selection dialog box

3) To avoid the interference of last version of the software to the current software installation, before formal installation begins, a dialog box prompting previous parameter setting saving will pop up, shown in Fig. 4-17. Choose “Yes” to save the parameters and delete the old version software before current software installation begins.

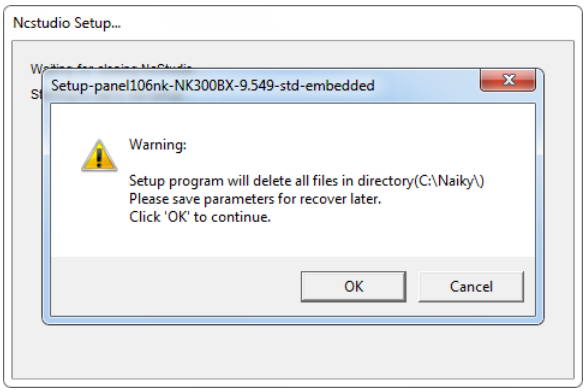


Fig. 4-17 Save parameters before installation

4) Click [Yes] to continue. If software of other version has been installed before and its parameters have been modified, there will be a prompt dialog box for confirmation of parameter settings saving, as shown in Fig. 4-18. Note that if it is the first time to install the software, this prompt will be omitted. Jump to step 5) directly to go ahead.

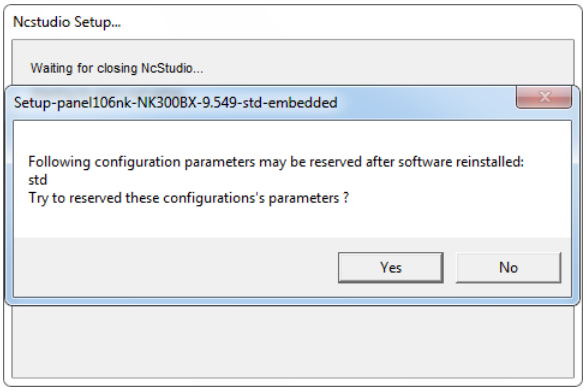


Fig. 4-18 Prompt for parameter saving

5) Click [Yes]. The system will be installed under directory C:\Naiky. Installation progress is shown in progress bar, as shown in below.

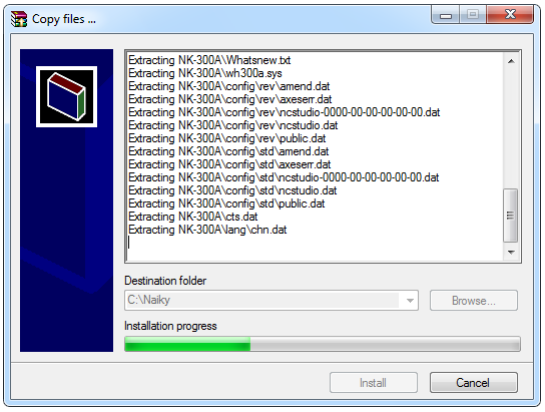


Fig. 4-19 Installation Progress

6) Software installation is completed.

CAUTION

The above installation introduction is for situation where the software has been damaged and cannot work normally. If the software can be launched normally, please refer to 4.2.1 to upgrade software instead of newly installing one.

4.3 Warning Information

Type	Warning Content	Cause	Solution
Warning	Simulation result shows that running scope of the program exceeds the mechanical stroke.	The scope of machining file exceeds upper and lower limits of work table, decided by setting “N10020”& “N10030” in system.	Modify the value of “N10020” and “N10030” to expand stroke range of work table (see section 3.4.3).
	The function can’t be used since backing to machine origin has not completed.	System has not returned to machine origin yet. If “N74001” is set “yes”, returning to machine origin before machining is a must.	Execute [back to machine origin] before using this function.
	The error of X (Y\Z) axis backing to machine origin is out of allowable range.	An error occurs in the precision of origin switch.	Detect the precision of origin switch.
		An error occurs in the precision of encoder origin.	Detect if origin signal of system encoder is right.
	This operation can’t be executed under current processing state.	Such illegal operation is executed in processing state as adjusting parameter.	Stop processing or execute those operations under idle state.
	This operation can’t be executed under simulation state.	Such illegal operation is executed in simulation mode as modifying parameters or pressing some shortcut keys.	Quit simulation mode or execute those 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 [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (refer to section 3.3).
		X-axis runs into limit switch directly during motion.	Manually move X-axis away from limit switch.
		There is an error in limit switch itself.	Check if limit switch works normally.

Type	Warning Content	Cause	Solution
Return to machine origin alarm	The distance is too close between coarse and fine positioning switches if X (Y\Z) axis backing to machine origin.	The actual installation distance between coarse and fine positioning switches is smaller than the setting value of parameter “N74110”.	Re-adjust the actual position of origin switch and encoder origin to make the space within the range: [0 + “N74110” ~ screw pitch – “N74110”] (see section 3.5).
Servo alarm	Servo alarm of X (Y\Z) axis	The polarity of X-axis servo alarm port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (refer to section 3.3).
		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 [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (refer to section 3.3).
		The E-stop button is pressed.	Turn the E-stop button clockwise make it pop-up.
Oil level alarm	Oil level alarm	The polarity of oil level alarm port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (see section 3.3).
		When the oil level line in the oil pump is below a certain value, a signal will be sent to the system to give an alarm.	Check if the oil mass is too small in the oil pump.
Spindle alarm	Spindle alarm	The polarity of spindle alarm port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (see section 3.3).
		There is an error in inverter.	Check if the inverter works normally.
File error	There is no file.	Start file machining with no file loaded in the system in advance.	Load a machining file before start machining.

Type	Warning Content	Cause	Solution
Pulse feedback alarm	The pulse feedbacks exceed the setting range of parameter.	It is used to detect if the D-value between the sent pulses and received pulses exceeds the setting value of parameter.	Check if the servo system is stable or the motor encoder is damaged.
Change tool over-travel alarm	Alarm for over-travel in tool change	Alarm signal occurs in tool change over-travel protection port	Check if the tool presetter works normally. During tool changing, Z-axis keeps moving downward for receiving no calibration signal, and triggers the over-travel protection port. Hardware faulty, which may result in continuous signal of the port.
Terminal board not connected	The terminal board is not well connected with the NK300 system	Wiring is not well or hardware fault of Lambda controller.	Re-plug the connection wire and restart the software. Something wrong with the port polarity. Invert the polarity and restart the software. Analyze possible causes according to the state of SYSTEM LED indicator. Change a new Lambda controller.
Panel not connected	Operation panel is not well connected	Something wrong with the port polarity. Wiring is not well. Operation panel fault.	Something wrong with the port polarity. Invert the polarity and restart the software. Re-plug the connection wire and restart the software. Change the operation panel.

CAUTION

Some alarms shown in table above are alarms added for the the machine structure of of a machine tool, and it is not discribed in general warning information. Please consult with the machine tool manufacturer if you have any questions.

4.4 Common Troubleshooting

4.4.1 What should you do if the spindle does not rotate?

1) Check if the software can work regularly. Press the Spindle CW/CCW button and see if the dot before the output port “SPIN” in [I/O Port] screen under [Diagnosis] becomes green. If it does, the software works regularly.
2) Start spindle, and check if the spindle start indicator lamp on the controller is on. If not, check whether the connection cable of the controller becomes loose. Close the host machine and power off the machine tool, and then re-plug the connection cable of the controller. If it still does not light, check if the cable of the controller, the controller or the system goes wrong. If it lights, measure if the SPIN port is conducted with a multimeter. If conducted, the spindle start output port (Y00) works normally; if it is not conducted, there is an error in the relay of spindle start.
3) Test whether the analog voltage output is normal between SVC and GND with a multimeter. If not, check if the connection cable of the controller becomes loose. Still not, check if the cable of the controller, the controller or the system goes wrong.
4) Check whether the parameter setting of the inverter is right, whether the spindle and the inverter have been damaged, or whether the wiring of the spindle and the inverter is correct.

4.4.2 What should you do if an axis does not move?

1) Check if there is output for the “Axis × Servo Enable” output port (in green) in [I/O Port] screen under [Diagnosis]. If there is output, the software works regularly. Check if the port polarity (it should be NO) is set correctly.
2) Check if the parameter setting of servo driver is correct (like setting control mode as position control, selecting a pulse input port for Panasonic driver, etc.).
3) Check if the servo cable of this axis is well contacted at the joint with the system host machine and the servo driver.
4) Check if something is wrong with servo driver, motor, servo cable or control system (e.g. exchange servo cable and servo driver with those of other axes working normally).

4.4.3 What should you do if servo motor Z-axis brake does not work?

Start the system and power on the machine tool (removing system alarm signal), and see if the brake output indicator lamp on the controller turns light.

1) If light, test whether there is 24V voltage between brake output ports (Y01-C01) with a multimeter. If there is 24V voltage, check whether the wiring of motor brake cable is correct. The motor brake cable should be connected to brake output port on the controller directly.
2) If not, directly conduct the brake input ports on the controller with a conducting wire. If light at this time, check whether the servo driver is enabled, whether the parameter setting related to brake output of servo driver is right, and whether brake output line of driver is correctly wired to the controller (black line is wired to COM, and only X20 is connected to with only one line); if still not light, please change the controller.

4.4.4 What should you do if homing is abnormal?

- Limit alarm or servo driver alarm occurs during homing (backing to machine origin).

1) See whether “Home Search Dir” “Home Latch Dir” and “Back Off Distance” in “Return Machine Home” are set properly. Sign of “Home Latch Dir” and “Back Off Distance” should be the same, but opposite to that of “Home Search Dir”.
2) Check if the software can receive the reference point signal of the axis. Trigger the home switch, while observing if the color of the dot before the “Reference Point of × axis” changes from red to green. If there is no color change, it indicates the software can’t receive the reference 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, directly conduct the reference point signal and COM port on the controller with a conducting wire, while observing whether the color of the dot before “Reference Point of × axis” changes in [I/O Port] screen.
3) 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 a machine tool.

- When backing to 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 input port “Reference Point of × axis” is correct in [I/O Port] screen under [Diagnosis]. When the 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 zero signal of this axis. The solutions are as below:

1) See if the servo cable of this axis is well contacted at the joints with the system host machine and servo driver.
2) Set the value of driver parameter “pre-scaler (frequency divider) of encoder” as 1/2 or 1/4 of the original one if YASKAWA or TECO, etc. driver is used.
3) Check if there is an error in the driver, motor, servo cable, or the control system (e.g. exchange servo cable and servo driver with those of other axes able to return to machine origin normally).

4.4.5 What should you do if a machine tool moves upward after arriving at the position of tool presetter in measurement?

1) View and tell whether the polarity of “Tool Presetter Signal” is right in [I/O Port] screen under [Diagnosis]. The color of the dot before the “Tool Presetter Signal” is red when the system does not receive tool measurement signal.
2) Manually press the tool presetter and check the color of the dot before the “Tool Presetter Signal” changes or not. If it remains the same, it tells that the tool presetter has been damaged.

5 Driver

5.1 Driver Parameters

Parameters listed in this chapter can make the machine work normally without ensuring machining results. Relevant parameters need adjusting according to the specific machine type.

5.1.1 Parameter Setting of WISE Servo Driver

Para. No.	Function	Value	Description
Pr528	LED initial status	6	Monitor if the number of sent and received pulses is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
Pr008	Command pulse No. per motor circle	0	When it is set to “0”, parameters Pr009 and Pr010 are valid.
Pr009	1st numerator of command pulse frequency division/ multiplication	Need calculation 0~2^{30}	Typical value: pitch 5 mm, encoder resolution 10000, deceleration ratio 1:1, pulse equivalent 0.001 mm: Pr009＝10000 Pr010＝pitch 5mm/ pulse equivalent 0.001mm＝5000 Pr009/Pr010=10000/5000=2/1
Pr010	Denominator of command pulse frequency division/multiplica-tion	Need calculation 0~2^{30}	
Pr100	1st position loop gain	480 (default)	Unit: 0.1/s. Set it according to the actual situation.
Pr101	1st velocity loop gain	270 (default)	Unit: 0.1Hz. Set it according to the actual situation.
Pr102	1st velocity loop integrated time constant	210 (default)	Unit: 0.1ms. Set it according to the actual situation.
When the value of Pr008 is not “0”, it should be calculated according to the following formula: $\text{Command Pulse No. per Motor Circle} = \frac{\text{Screw Pitch}}{\text{Pulse Equivalent} \times \text{Mechanical Deceleration Ratio}} = \frac{5\text{mm}}{0.001\text{mm/p}} = 5000$ That is to say, when screw pitch is 5mm and pulse equivalent is 0.001, the value of Pr008 is 5000.			

◆ **Attached list: the relationship among parameters Pr0008, Pr0009 and Pr010**

Pr008	Pr009	Pr010	Description
0~2^{20}	– (no influence)	– (no influence)	Command Pulse Input → Encoder Resolution / Setting Value of Pr008 → Position Command As shown above, the process is undergone in terms of the setting value of Pr008, not affected by the settings of Pr009 and Pr010.
0	0	0~2^{30}	Command Pulse Input → Encoder Resolution / Setting Value of Pr010 → Position Command When the values of Pr008 and Pr009 are both set to “0”, as shown above, the process is undergone in terms of the setting value of Pr010.
	0~2^{30}	0~2^{30}	Command Pulse Input → Setting Value of Pr009 / Setting Value of Pr010 → Position Command When the value of Pr008 is “0”, but the value of Pr009 is not “0”, as shown above, the process is undergone in terms of the setting values of Pr009 and Pr010.

5.1.2 Parameter Setting of YASKAWA Σ–Ⅱ Servo Driver

Para. No.	Function	Value	Description
Fn010	Set password (to prevent arbitrary modification to parameters)	0000	Set [0000]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] permitted; Set [0001]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] prohibited.
Un00C	Surveillance mode	LXXXX (Hexadeci-mal system)	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
Pn000	Direction selection Control mode selection	0010	Bit 0: Set 0, “CCW” rotation is forward rotation (viewed from the load end of screw ball); Set 1, the rotation direction of the motor is reversed. Bit 1: Set 1, position control mode (calculate pulse instruction all the time).
Pn200	Select pulse	0005	Bit 0: Set 5, select the instruction input mode as

<table>
<tr><th>Para. No.</th><th>Function</th><th>Value</th><th colspan="3">Description</th></tr>
<tr><td></td><td>instruction mode</td><td></td><td colspan="3">“pulse + direction”, negative logic.
Bit3: Set 0, input differential signal into filter.</td></tr>
<tr><td>Pn50A</td><td>Selection function</td><td>8100</td><td colspan="3">Bit 1: Set 0, Servo ON /S-ON, input from the 40th pin; Set 7, Servo ON all the time.
Bit 3: Set 8, forward rotation not used and signal input (P-OT) prohibited.</td></tr>
<tr><td>Pn50B</td><td>Selection function</td><td>6548</td><td colspan="3">Bit 0: Set 8, reverse rotation not used and signal input (N-OT) prohibited.</td></tr>
<tr><td>Pn50F</td><td>Selection function</td><td>0300</td><td colspan="3">Set it when servo motor with brakes.
Bit 2: Set 3, brake interlock signal “/BK” is output from CN1-29, CN1-30 to control 24V relay for brake.</td></tr>
<tr><td>Pn50E</td><td>Selection function</td><td>0211</td><td colspan="3">Set it when servo motor with brakes.
To avoid of CN1-29 and CN1-30 being used for other function and leading to brake ineffective, “3” is not allowed to appear in the 4 digits.</td></tr>
<tr><td>Pn506</td><td>Servo off, time delay of brake when motor stops</td><td>Depended</td><td colspan="3">Set it when motor with brakes.
Default setting is “0”, setting unit is 10ms.</td></tr>
<tr><td rowspan="4">Pn201</td><td rowspan="4">Encoder cycle-divided ratio (Pulse output No. per motor cycle after cycle-divided)</td><td rowspan="4">See right-side</td><td rowspan="4">Gain
Encoder</td><td>Type</td><td>Pulse No. per Motor Circle (PPR)</td></tr>
<tr><td>A</td><td>13bit 2048</td></tr>
<tr><td>B</td><td>16bit 16384</td></tr>
<tr><td>C</td><td>17bit 32768</td></tr>
<tr><td>Pn202</td><td>Electronic gear ratio (numerator)</td><td>Need Calculation</td><td colspan="3" rowspan="2">Pn202 = pulse No. of each encoder circle × 4 × mechanical deceleration ratio.
Pn203 = (screw pitch/ pulse equivalent).
Typical value: pitch 5mm, encoder 17-bit, coaxial connection between motor and screw, pulse equivalent 0.001mm, Pn202＝16384; Pn203＝625.
Pitch 5mm, encoder 17-bit, coaxial connection between motor and screw, pulse equivalent 0.0005mm, Pn202＝8192; Pn203＝625.</td></tr>
<tr><td>Pn203</td><td>Electronic gear ratio (denominator)</td><td>Need Calculation</td></tr>
</table>

5.1.3 Parameter Setting of YASKAWA Σ-Ⅴ/Σ-7 Servo Driver

Para. No.	Function	Value	Description
Fn010	Parameter input prohibition setting	0000	Set [0000]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] permitted. Set [0001]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] prohibited.
Pn000	Function selection basic switch 0	0010	Bit 0: Set 0, positive rotation at positive rotation command; Bit 1: Set 1, position control mode (pulse sequence command)
Pn200	Format selection switch of position control command	0005	Bit 0: Set 5, select the instruction mode as “pulse + direction”, negative logic.
Pn50A	Input signal selection 1	8100	Bit 1: Set 0, Servo ON /S-ON, input from the 40th pin; Set 7, Servo ON all the time. Bit 3: Set 8, positive rotation not used and signal input (P-OT) prohibited.
Pn50B	Input signal selection 2	6548	Bit 0: Set 8, negative rotation not used and signal input (N-OT) prohibited.
Pn50F	Output signal selection 2	0300	Set it when servo motor with brakes. Bit 2: Set 3, brake interlock signal “/BK” is output from CN1-29, CN1-30 to control 24V relay used for brake.
Pn50E	Output signal selection 1	0211	Set it when servo motor with brakes. To avoid of CN1-29 and CN1-30 being used for other function and leading to brake ineffective, 3 is not allowed to appear in the 4 digits.
Pn506	Brake instruction-servo OFF time delay	Depended	Set it when motor with brakes Default setting is “0”, setting unit is ms.
Pn20E	Electronic gear ratio (numerator)	Need Calculation	$\frac{Pn20E}{Pn210}=\frac{Encoder\ Resolution\times Pulse\ Equivalent\times Deceleration\ Ratio}{Screw\ Pitch}$ For example, screw pitch 5mm, 20-bit encoder, coupling direct drag, pulse equivalent 0.001mm, $\frac{Pn20E}{Pn210}=\frac{2^{20}\times 0.001}{5}=\frac{1048576}{5000}=\frac{131072}{625}$ When screw pitch is 10mm, $\frac{PN20E}{PN210}=\frac{1048576}{10000}=\frac{65536}{625}$ For a rotary axis with 13-bit encoder and deceleration ratio as 60,
Pn210	Electronic gear ratio (denominator)	Need Calculation	

Para. No.	Function	Value	Description
			$\frac{Pn20E}{Pn210}=\frac{2^{13}\times 0.001\times 60}{360}=\frac{8192}{6000}=\frac{512}{375}$

5.1.4 Parameter Setting of PANASONIC MINAS A4 Servo Driver

Para. No.	Function	Value	Description
Pr01	LED initial status	12	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
Pr02	Select control mode	0	0: position mode 1: velocity mode 2: torque mode
Pr40	Selection of command pulse input	1	1: input by differential exclusive circuit
Pr42	Select command pulse input mode	3	Set command pulse input mode: pulse + direction, negative logic
Pr48	1st numerator of command pulse frequency multiplication	Need calculation Range: 1~10000	Typical value: pitch 5 mm, encoder resolution 10000, shaft coupling direct drag, pulse equivalent 0.001 mm: Pr48＝10000 Pr4B＝pitch 5mm / pulse equivalent 0.001mm＝5000 Pr48/Pr4B=10000/5000=2/1
Pr4B	Denominator of command pulse frequency multiplication	Need calculation Range: 1~10000	

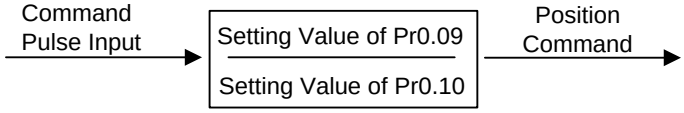
5.1.5 Parameter Setting of PANASONIC MINAS A5 Servo Driver

Para. No.	Function	Value	Description
Pr5.28	LED initial status	6	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control

Para. No.	Function	Value	Description
			card is detected by pulse inspection in order to determine whether there is electrical interference.
Pr0.01	Select control mode	0	0: position mode 1: velocity mode 2: torque mode
Pr0.05	Selection of command pulse input	XX	0: Photo-coupler input (PULS1, PULS2, SIGN1, SIGN2) 1: Exclusive input for line driver (PULSH1, PULSH2, SIGNH1,SIGNH2) Note: generally, “1” is selected for this parameter.
Pr0.07	Command pulse input mode setup	3	Set command pulse input mode: pulse + direction, negative logic.
Pr0.08	Command pulse counts per one motor revolution	0	When it is set as “0”, parameters Pr0.09 and Pr0.10 are valid.
Pr0.09	1st numerator of command pulse frequency multiplication	Need calculation Range: $0\sim2^{30}$	Typical value: pitch 5 mm, encoder resolution 10000, shaft coupling direct drag, pulse equivalent 0.001 mm: Pr0.09＝10000 Pr0.10＝pitch 5mm/ pulse equivalent 0.001mm＝5000 Pr0.09/Pr0.10=10000/5000=2/1
Pr0.10	Denominator of command pulse frequency multiplication	Need calculation Range: $0\sim2^{30}$	
When the value of Pr0.08 is not “0”, it can be calculated in terms of the following formula: $\text{Command Pulse No. per Motor Circle} = \frac{\text{Screw Pitch}}{\text{Pulse Equivalent} \times \text{Mechanical Deceleration Ratio}} = \frac{5\text{mm}}{0.001\text{mm/p}} = 5000$ When screw pitch is 5mm and pulse equivalent 0.001mm/p, the value of Pr0.08 is “5000”.			

◆ **Attached List: the relationship among parameters Pr0.08, Pr0.09 and Pr0.10.**

Pr0.08	Pr0.09	Pr0.10	Description
$0\sim2^{20}$	— (no influence)	— (no influence)	Command Pulse Input → Encoder Resolution / Setting Value of Pr0.08 → Position Command The process shown above is undergone in terms of the setting value of Pr0.08, not affected by the settings of Pr0.09 and Pr0.10.
0	0	$0\sim2^{30}$	Command Pulse Input → Encoder Resolution / Setting Value of Pr0.10 → Position Command When the values of Pr0.08 and Pr0.09 are both set as “0”, as

Pr0.08	Pr0.09	Pr0.10	Description
			shown above, the process is undergone in terms of the setting value of Pr0.10.
	$0\sim2^{30}$	$0\sim2^{30}$	 When the value of Pr0.08 is “0”, but the value of Pr0.09 is not “0”, as shown above, the process is underdone in terms of the setting values of Pr0.09 and Pr0.10.

5.1.6 Parameter Setting of MITSUBISHI MR-JE Servo Driver

Para. No.	Code	Function	Value	description
PA01	*STY	Operation mode	XXX0	_ _ _x: select position control mode.
PD24	MBR	Output assignation to CN1-23 pin	XX05	_ _ xx: select MBR (electromagnetic brake interlock).
PA06	CMX	Electronic gear numerator	Need calculation	CMX/CDV=command unit × servo motor resolution × mechanical deceleration ratio / pitch of screw. E.G., pitch 5 mm, encoder resolution 10000, deceleration ratio 1:1, pulse equivalent 0.001 mm, CMX/CDV=10000×0.001/5 = 2/1; When pulse equivalent = 0.0005mm, CMX/CDV = 1/1. Electronic gear ratio range: 1/50 ~ 500
PA07	CDV	Electronic gear denominator	Need calculation	
PC36	*DMD	Status display selection	00XX	_ _xx: status display selection at power-on. This is used to select a status display shown at power-on. 00: cumulative feedback pulses 01: servo motor speed 02: droop pulses 03: cumulative command pulses 04: command pulse frequency
PA13	*PLSS	Command pulse input form	0011	Set command pulse input form: pulse train+ sign, negative logic.
PD03	*DI1L	Input assignation to CN1-15 pin	XX02	_ _xx: select SON under position control mode.

5.1.7 Parameter Setting of MITSUBISHI MR-E Servo Driver

Para. No.	Code	Function	Value	Description
0	*STY	Control mode selection and regenerative fittings	X0X0	Bit 0: set 0: select position control mode. Bit 1, select motor series: 0: HC-KFE; 1:HC-SFE; Bit 3, select regenerative apparatus, set 0: not use. Bit 4, select motor power.
1	MBR	Function selection 1	001X	Bit 0: input signal filter. If external input signal causes chattering due to noises, etc., input filter is used to suppress it. Bit 1: CN1-12 function selection, set “1”: electromagnetic brake interlock (MBR); set “0”: zero speed detection signal.
3	CMX	Electronic gear numerator	Need calculation	CMX/CDV=command unit × servo motor resolution × mechanical deceleration ratio / screw pitch. E.G., pitch 5 mm, encoder resolution 10000, shaft coupling direct drag, pulse equivalent 0.001 mm, CMX/CDV=10000×0.001/5 = 2/1; When pulse equivalent = 0.0005mm, CMX/CDV = 1/1. Electronic gear ratio range: 1/50 ~ 500
4	CDV	Electronic gear denominator	Need calculation	
18	*DMD	Status display selection	00XX	3: cumulative command pulses E: load inertia When the parameter is set [3], monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection to determine if there is electrical interference.
21	*OP3	Function selection 3 (command pulse format selection)	0001	Set pulse command input form: pulse train+ sign, negative logic
41	*DIA	Signal input SON-ON, LSP-ON and LSN-ON automatically selection	0110	Bit 0: Servo-ON selection. [0]: servo on by external input; [1]: servo on all the time inside. Bit 1: last signal of positive rotation range (LSP): [1]: auto servo on inside, without external wiring. Bit 3: last signal of negative rotation range (LSN): [1]: auto servo on inside and no need of external wiring.

5.1.8 Parameter Setting of DELTA ASDA-A Servo Driver

Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection to determine if there is electrical interference.
P1-00	External pulse input type	ZYX	002	X=2: pulse + direction; Z=0: positive logic
P1-01	Control mode setup	ZYX1X0	0000	Z=0: during control mode switching, DIO is maintaining the set value. Since switching control mode is not used, Z=0 Y=0: forward rotation (CCW) (in terms of load); Y=1: the rotation direction is reversed. X1X0=00: position control mode
P1-32	Motor stop mode selection	YX	00	Y=0: when there is no servo enabled, motor dynamic brake occurs; Y=1: motor is free. X=0: motor stops instantly, X=1: motor stops with deceleration.
P1-44	Electronic Gear Ratio (Numerator)(N1)	1~32767	Need calculation	N1/M= encoder pulses × 4× pulse equivalent× mechanical deceleration ratio/ pitch Representative value: encoder pulses=2500, pitch=5mm, pulse equivalent=0.001, deceleration ratio=1, calculation as below: N1/M= 2500×4×0.001/5 = 2 / 1, N1=2, M=1; When the multi-electronic gear ratio is not used, P2-60~ P2-62 are not required.
P1-45	Electronic Gear Ratio (Denominator) (M)	1~32767	Need calculation	
P2-10	Digital Input Pin DI1	X2X1X0	101	X1X0=01: digital input (DI1=SON) corresponds to 9th pin of CN1. X2 = 1: set DI1 input as NO (normally open) a-contact point.
P2-15	Digital Input pin DI6	X2X1X0	100	Default factory setting of DI6 and DI7 are NC (normally closed) limit signal input pins;

Para. No.	Function	Format & Range	Value	Description
P2-16	Digital Input Pin DI7	X2X1X0	100	driver can't run without being connected to pin 32 and pin 31 of CN1. X2=1: set DI6 and DI7 inputs as NO (normally open) a-contact points; X1X0=00, limit signal input of the driver is not used.
P2-17	Function setting for digital input pin DI8	X2X1X0	100	External EMG stop input is not used.
P2-21	Function setting for digital output pin DO4	X2X1X0	108	DO4 corresponds to pin 1 & pin 26, used as clamping-position brake signal of Z-axis; X2=1: set DO4 output as NO (normally open) a-contact point; X2=0: set DO4 output as NC (normally closed) b-contact point; X1X0=08: set pin 1 and pin 26 as BK+ and BK- respectively.
P2-22	Function setting for digital output pin DO5	X2X1X0	007	DO5 corresponds to pin 28 & pin 27, used as servo alarm signal. X2=0: set DO5 output as NC (normally closed) b-contact point. X1X0=07: set pin 28 and pin 27 as ALRM+ and ALRM- respectively.
P2-51	Servo ON (SON) setup		0	0: Servo ON must be triggered by numerical input signal. 1: when servo is powered, if there is no alarm signal, servo will be automatically on. Set 1 when there is no SON signal wire.

5.1.9 Parameter Setting of DELTA ASDA-B Servo Driver

Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.

Para. No.	Function	Format & Range	Value	Description
P1-00	External pulse train input type	ZYX	002	X=2: pulse + direction; Z=0: positive logic
P1-01	Set control mode	YX1X0	000	Y=0: forward rotation (CCW) (from the view of load) Y=1: the rotation direction is reversed. X1X0=00: position control mode
P1-32	Motor stop mode	YX	00	Y=0: when there is no servo enabled, motor dynamic brake occurs; Y=1: motor is free. X=0: motor stops instantly; X=1: motor stops with deceleration.
P1-44	Electronic Gear Ratio (Numerator) (N1)	1~32767	Need calculation	N1/M= mechanical deceleration ratio × 4 × encoder pulses × pulse equivalent / pitch. Representative value: encoder pulses=2500, pitch =5mm, pulse equivalent=0.001 mm/p, deceleration ratio = 1, calculation as below: N1 / M = 2500×4×0.001/5 = 2/1, N1=2, M=1; When the multi-electronic gear ratio is not used, P2-60 ~P2-62 are not required.
P1-45	Electronic Gear Ratio (Denominator) (M)	1~32767	Need calculation	
P2-10	Digital Input Pin 1 (DI1)	X2X1X0	101	X1X0=01: digital input (DI1 = SON) corresponds to 17th pin of CN1. X2=1: set DI1 input as NO (normally open) a-contact point.
P2-15	Function setting for digital input pin DI6	X2X1X0	100	Default factory setting of DI6 is NC (normally closed) limit signal input; driver can't run without being connected to pin 32 and pin 31 of CN1. X2=1: set DI6 input as NO a-contact point. X1X0=00, limit input of driver is not used.
P2-18	Function setting for digital output pin DO1	X2X1X0	108	DO1 corresponds to the 16th pin, as clamping-position brake signal of Z-axis; X2=1: set DO1 output as NO a-contact point; X2=0: set DO1 output as NC b-contact point; X1X0=08: set the 16th pin as BK+.
P2-20	Function setting for digital output pin DO3	X2X1X0	007	DO3 corresponds to pin 1, used as servo alarm signal. X2=0: set DO3 output as NC b-contact

Para. No.	Function	Format & Range	Value	Description
				point; X1X0=07: set pin 1 as ALRM+.

5.1.10 Parameter Setting of DELTA ASDA-A2 Servo Driver

Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
P1-00	External pulse train input type	ZYX	002	X=2: pulse + direction; Z=0: positive logic
P1-01	Set control mode	ZYX1X0	0000	Z=0: during control mode switching, DIO is maintaining the set value. Since switching control mode is not used, Z=0; Y=0: positive rotation (CCW) (from the view of load); Y=1: negative rotation (CCW) X1X0=00: position control mode
P1-44	Electronic Gear Ratio (Numerator) (N1)	1~32767	Need calculation	$\frac{\text{P1-44}}{\text{P1-45}} = \frac{\text{Encoder Reso.} \times \text{Pulse Equiv.} \times \text{Mech. Dece. Ratio}}{\text{Screw Pitch}}$
P1-45	Electronic Gear Ratio (Denominator) (M)	1~32767	Need calculation	Assuming encoder resolution is 1280000, pitch 5mm, pulse equivalent 0.001, and non-cascade connection, then: $\frac{\text{P1}-44}{\text{P1}-45} = \frac{1280000 \times 0.001}{5} = \frac{256}{1}$ When the multi-electronic gear ratio is not used, P2-60 ~P2-62 are not required.
P2-10	Digital Input Pin 1 (DI1)	X2X1X0	101	X1X0=01: digital input (DI1 = SON) corresponds to 9th pin of CN1. X2=1: set DI1 input as NO (normally open) a-contact point.
P2-15	Function setting for digital input pin DI6	X2X1X0	100	Default factory setting of DI6 and DI7 is NC (normally closed) limit signal input; driver can't run without being connected to pin 32

Para. No.	Function	Format & Range	Value	Description
				and pin 31 of CN1. X2=1: set DI6 and DI7 input as NO a-contact points. X1X0=00, limit input of driver is not used.
P2-16	Function setting for digital input pin DI7	X2X1X0	100	External EMG stop input is not used.
P2-17	Function setting for digital input pin DI8	X2X1X0	100	
P2-21	Function setting for digital output pin DO4	X2X1X0	108	DO4 corresponds to pin 1 & pin 26, used as clamping-position brake signal of Z-axis; X2=1: set DO4 output as NO (normally open) a-contact point; X2=0: set DO4 output as NC (normally closed) b-contact point; X1X0=08: set pin 1 and pin 26 as BK+ and BK- respectively.
P2-22	Function setting for digital output pin DO5	X2X1X0	007	DO5 corresponds to pin 28 & pin 27, used as servo alarm signal. X2=0: set DO5 output as NC b-contact point. X1X0=07: set pin 28 and pin 27 as ALRM+ and ALRM- respectively.

5.1.11 Parameter Setting of DELTA ASDA-B2 Servo Driver

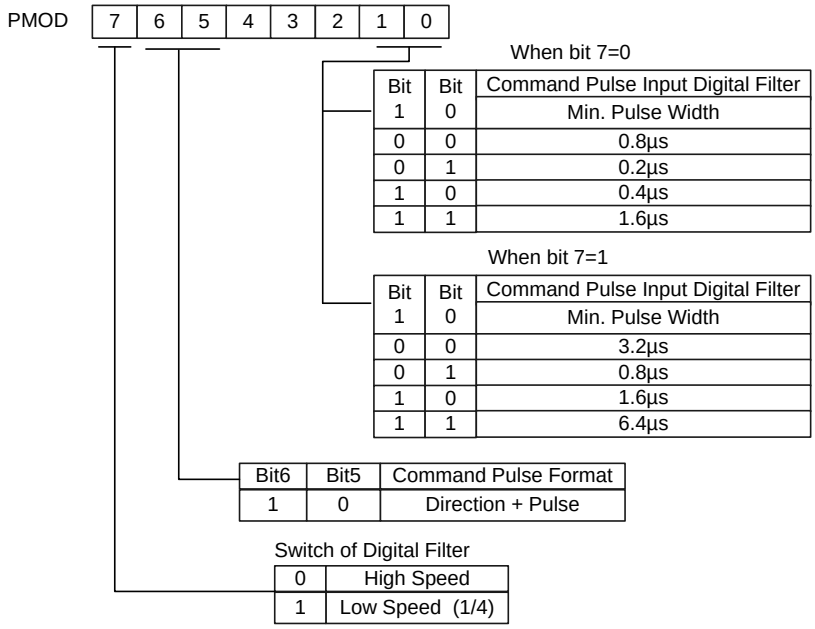
Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
P1-00	External pulse train input type	ZYX	002	X=2: pulse + direction; Z=1: positive logic

Para. No.	Function	Format & Range	Value	Description
P1-01	Set control mode	ZYX1X0	0000	Z=0: during control mode switching, DIO is maintaining the set value. Since switching control mode is not used, Z=0; Y=0: forward rotation (CCW) (from the view of load); Y=1: the rotation direction is reversed. X1X0=00: position control mode
P1-44	Electronic Gear Ratio (Numerator) (N1)	1~32767	Need calculation	N1/M= mechanical deceleration ratio × 4 × encoder pulses × pulse equivalent / pitch. Representative value: encoder pulses=40000, pitch =5mm, pulse equivalent=0.001, deceleration ratio = 1, calculation as below: N1 / M = 40000×4×0.001/5 = 32/1, N1=32, M=1; When the multi-electronic gear ratio is not used, P2-60 ~P2-62 are not required.
P1-45	Electronic Gear Ratio (Denominator) (M)	1~32767	Need calculation	
P2-10	Digital Input Pin DI1	X2X1X0	101	X1X0=01: digital input (DI1 = SON) corresponds to 9th pin of CN1. X2=1: set DI1 input as NO (normally open) a-contact point.
P2-15	Function setting for digital input pin DI6	X2X1X0	000	Default factory setting of DI6 and DI7 is NC (normally closed) limit signal input; driver can't run without being connected to pin 32 and pin 31 of CN1. X2=0: set DI6 and DI7 inputs as NC b-contact point. X1X0=00, limit input of driver is not used.
P2-16	Function setting for digital input pin DI7	X2X1X0	000	
P2-17	Function setting for digital input pin DI8	X2X1X0	000	External EMG stop input is not used.
P2-18	Function setting for digital output pin DO1	X2X1X0	108	DO1 corresponds to pin 6 & pin 7, used as clamping-position brake signal of Z-axis; X2=1: set DO1 output as NO (normally open) a-contact point; X2=0: set DO1 output as NC (normally closed) b-contact point; X1X0=08: set pin 6 and pin 7 as BK- and BK+ respectively.
P2-22	Function setting	X2X1X0	007	DO5 corresponds to pin 28 & pin 27, used

Para. No.	Function	Format & Range	Value	Description
	for digital output pin DO5			as servo alarm signal. X2=0: set DO5 output as NC b-contact point. X1X0=07: set pin 28 and pin 27 as ALRM+ and ALRM- respectively.

5.1.12 Parameter Setting of SANYO PY Servo Driver

Para. No.	Abbr.	Name	Standard Value	Setting Range	Unit	Remark
1-2	EGER	Electronic gear ratio	4/1	1/32767 to 32767/1		Depends on the specific encoder resolution. The formula of electronic gear ratio of servo driver is as below: Electronic gear ratio numerator =mechanical deceleration ratio × 4× pulse No. per encoder circle; Electronic gear ratio denominator = (screw pitch / pulse equivalent) E.G. In Weihong system, the default pulse equivalent is 0.001mm/p, screw pitch is 5mm, pulse number per encoder circle is 2000 shaft coupling direct drag, currently the numerator of the electronic gear ratio is 8, and the denominator is 5. (Select incremental type encoder)
1-16	MENP	Pulse amount of the motor encoder 1. Set the pulse amount of the motor encoder; 2. Standard configuration of the encoder pulse No. is as below. Incremental encoder omitting wiring: --2000P/R Absolute		500 to 65535	P/R	

<table>
<tr><th>Para. No.</th><th>Abbr.</th><th>Name</th><th>Standard Value</th><th>Setting Range</th><th>Unit</th><th>Remark</th></tr>
<tr><td></td><td></td><td colspan="2">encoder:--2048P/R</td><td></td><td></td><td></td></tr>
<tr><td>2-0</td><td>PMOD</td><td colspan="5">Pulse format of position command:
Our system uses: direction + pulse format, the parameters are shown as following:
 
</td></tr>
<tr><td>4-3</td><td>TYPE</td><td colspan="3">Control mode:
*Select one control mode from position, velocity, and torque modes.
<table><tr><th>Selection Item</th><th>Content</th></tr><tr><td>Position</td><td>Position control mode</td></tr><tr><td>Velocity</td><td>Velocity control mode</td></tr><tr><td>Torque</td><td>Torque control mode</td></tr><tr><td>Velo ↔Torq</td><td>Velocity↔ Torque switch mode</td></tr><tr><td>Posi ↔Torq</td><td>Position↔ Torque switch mode</td></tr><tr><td>Posi ↔Velo</td><td>Position ↔Velocity switch mode</td></tr></table>Referring to the switch type, the requisite control mode can be selected from pin 36 or 35 of the CN1.
Func3, set Bit7 as 0: pin 36 is enabled.
set Bit7 as 1:pin 35 is enabled.
$ $ $: standard value varies with the reset setup (leave factory setting).</td><td>6 types</td><td>Our system selects position control mode.</td></tr>
</table>

5.1.13 Parameter Setting of SANYO R Servo Driver

Para. No.	Parameter Name	Set Value	Remarks
Group 0, parameter setting of tuning mode			
00	Setting of tuning mode	00	Set as auto tuning mode
Group 8, setting of the control parameters			
00	Polarity of position input	00	Position command mode: positive rotation effective
11	Input command mode	02	Pulse train + negative logic, negative logic
15	Setting of electronic gear	8/5	It depends on the resolution of the specific encoder. E.G.: incremental encoder 2000, motor needs 2000 ×4=8000 pulses per circle. And pulse equivalent of Weihong control card is 0.001mm/p, it needs 1000 pulses to move 1mm along line, in other words, if the screw pitch is 5, so, to move 5mm along line needs 5000 pulses, so F=8000/5000=8/5.
Group 9, setting of function effective			
05	Servo ON selection	02	Select servo ON state.
02	Servo alarm elimination	10	Make the function of servo alarm effective
Setting of the system parameters			
02	Encoder selection	00	Standard incremental encoder. The parameter depends on the specific situation, what we list is only the representative one.
03	Encoder resolution	2000	500－65535, set the encoder resolution manually.
08	Control mode selection	02	Select position control mode

5.1.14 Parameter Setting of SANYO Q Servo Driver

Para. No.	Parameter Name	Set Value	Remarks
Group 1			
GER1	Electronic gear ratio 1	1/1	Set electronic gear ratio for position command pulse. E.G., incremental encoder 2000, motor needs 2000 ×4=8000 pulses per circle. And pulse equivalent of Weihong control card is 0.001mm/p, it needs 1000 pulses to move 1mm along line, in other words, if the screw pitch is 5, so, to move 5mm along line needs 5000 pulses, so F=8000/5000=8/5.

Para. No.	Parameter Name	Set Value	Remarks
GER2	Electronic gear ratio 2	1/1	This setting is the same as that of electronic gear ratio 1 and activated during electronic gear switching.
Group 4			
PA400	Command pulse selection	00H	Set position command pulse as “pulse + direction”.
Group 8			
S-ON	Servo ON	02H	Select servo ON state.
AL-RST	Alarm reset	10H	Make the function of servo alarm effective
Setting of the system parameters			
01	Encoder selection	00	Standard incremental encoder. The parameter depends on the specific situation, what we list is only the representative one.
03	Incremental encoder resolution	2000	500－65535, set the encoder resolution manually.
08	Control mode selection	02	Select position control mode

5.1.15 Parameter Setting of KT270 Servo Driver

Para. No.	Parameter Name	Value	Description
PA4	Control mode selection	0	The control mode of the driver can be set through this parameter: 0: position control mode; 1: speed control mode; 2: trial run control mode; 3: JOG control mode.
PA12	Numerator of position command pulse ratio	2	Set the ratio of the position command pulse (electronic gear). Under position control mode, with the setting of the PA12 and PA13, it is convenient to match with pulse source of each type, which can reach your perfect control resolution (that is angle/pulse) Expression: $P \times G = N \times C \times 4$ P: pulse amount of the input command; G: electronic gear ratio, G=ratio numerator / ratio denominator. N: circle number that the motor rotates; C: each circle line number of photo electricity encoder, C of our system =2500.

Para. No.	Parameter Name	Value	Description
			E.G.: input 6000 command pulses to make the servo motor rotate one circle, $G=\frac{N\times C\times 4}{P}=\frac{1\times 2500\times 4}{6000}=\frac{5}{3}$ So set PA12 as 5 and PA13 as 3. We recommend the range of electronic gear ratio as: $\frac{1}{50}\le G\le 50$
PA13	Denominator of the position command pulse ratio	1	Refer to parameter PA12.
PA14	Input mode of the position command pulse	0	Set the input mode of the position command pulse; there are following three modes can be selected by setting the parameter: 0: pulse + symbol; 1: positive rotation pulse/ negative rotation pulse; 2: two orthogonal pulses inputs Default setting is 0: pulse + symbol, negative logic.
PA20	Invalid input on the end of the stroke	1	0: Valid stroke end of LSP, LSN positive rotation, negative rotation. When switch LSP is connected, driving of the positive rotation is allowed; When switch LSP is disconnected, driving of the positive rotation is prohibited (torque of the positive direction is 0). LSN is the same as LSP. If LSP and LSN are all disconnected, the abnormal alarming of driving prohibited will occur (NO.7). 1: Invalid stroke end of LSP, LSN positive rotation, negative rotation. No matter which state of the switch LSP and LSN is in, driving of positive rotation and negative rotation are all allowed. Simultaneously, even if LSP and LSN are all disconnected, abnormal alarming of driving prohibited will not occur (NO.7). 2: Invalid stroke end of LSP, LSN positive rotation, negative rotation, and SON is forced to be effective. (Note: SON forcedly effective is only used for motor debugging. In normal use, we suggest controlling the state of SON by input port.)

Para. No.	Parameter Name	Value	Description
			3: Valid stroke end of LSP, LSN positive rotation, negative rotation. When switch LSP is connected, driving of the positive rotation is allowed; When switch LSP is disconnected, driving of the positive rotation is prohibited (the speed of positive direction is 0, but the torque is not 0). LSN is the same as LSP. When LSP and LSN are all disconnected, abnormal alarming of driving prohibited will not occur (NO.7).

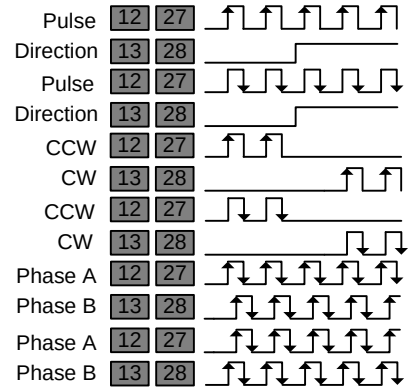
5.1.16 Parameter Setting of FUJI FALDIC-β Servo Driver

Para. No.	Name	Value	Description
01	Command pulse numerator α	Need calculation 1~32767	Command pulse numerator and denominator are equal to those of the electronic gear ratio. α/ β=encoder resolution× pulse equivalent× mechanical deceleration ratio / screw pitch. Typical value: encoder resolution 65536, pitch 5mm, pulse equivalent 0.001, mechanical deceleration ratio 1, α / β=65536×0.001 / 5=8192 / 625, So α=8192, β=625.
02	Command pulse denominator β	Need calculation 1~32767	
03	Pulse string input form	0	Set the input mode of pulse string as: instruction + symbol, that is 'pulse + direction'.
04	Direction of rotation switch	0 or 1	Set 0: Positive direction: Forward rotation (CCW) Set1: Positive direction: Reverse rotation (CW)
10	CONT1 signal distribution	1	CONT1 is distributed as RUN (i.e. SON); if not distributed, CONT1 will be auto ON if there is no alarming when powered.
11	CONT2 signal distribution	2	CONT2 is distributed as RST (i.e. servo alarming clearance CLR). When 12, 13, 14 are 0, that is CONT3, CONT4 and CONT5 can't be distributed as OT over-travel or EMG (external emergency stop).
15	OUT1 signal distribution	1	Set 1, OUT1 is distributed as a-contact point of alarming output; Set 2, OUT1 is distributed as b-contact point of alarming detection.
27	Parameter	0 or 1	Set 0, write-enable.

Para. No.	Name	Value	Description
	write-protection		Set 1, write-protected.
74	CONT always ON 1	1	Initial value: 0. when set “1”, servo is activated (RUN).

5.1.17 Parameter Setting of STONE GS Servo Driver

<table>
<tr><th>Para. No.</th><th>Para. Name</th><th>Value</th><th>Description</th></tr>
<tr><td>F0f</td><td>Electronic gear ratio numerator</td><td>2</td><td rowspan="2">Electronic gear ratio of position mode: 4× pulse frequency fed back by servo encoder = command pulse frequency× F0f / F10; value of F0f / F10 must be within 1/100~100. (calculation with pitch 10mm)</td></tr>
<tr><td>F10</td><td>Electronic gear ratio denominator</td><td>1</td></tr>
<tr><td>F00</td><td>Control mode selection</td><td>2</td><td>0: External speed running mode; make sure the value and direction of motor speed according to the external analog -10V ~ +10V signal of CN2-16, 17;
1: Internal speed running mode; make sure the value and direction of motor speed according to the setting of parameter F33, F35, F37, F39 and the port status of CN2-9, CN2-25;
2: Position pulse running mode; accept the input of external position pulse and direction level signal;
3: Jog mode; make sure the motor speed in terms of parameter setting of F3b, and control the rotation direction by the direction keystroke ▼ and ▲;
4: Torque mode; make sure the value and direction of motor torque according to the external analog -10V ~ +10V signal of CN2-43, 1;
5~10: Mixed mode; select mode according to the port input status of CN2-24:
<table>
<tr><th rowspan="2">F00 Value</th><th colspan="2">CN2-24 Interface Status</th></tr>
<tr><th>OFF (Mode One)</th><th>ON (Mode Two)</th></tr>
<tr><td>5</td><td>Position Pulse Mode</td><td>External Speed Running Mode</td></tr>
<tr><td>6</td><td>Position Pulse Mode</td><td>Internal Speed Running Mode</td></tr>
<tr><td>7</td><td>Position Pulse Mode</td><td>Torque Mode</td></tr>
<tr><td>8</td><td>Internal Speed Running Mode</td><td>External Speed Running Mode</td></tr>
<tr><td>9</td><td>Internal Speed Running Mode</td><td>Torque Mode</td></tr>
<tr><td>10</td><td>External Speed Running Mode</td><td>Torque Mode</td></tr>
</table>
</td></tr>
</table>

Para. No.	Para. Name	Value	Description
F2e	Pulse input mode selection	2	Command pulse string mode selection of position mode: 1 – Single pulse train positive logic 2 – Single pulse train negative logic 3 – Double pulse train positive logic 4 – Double pulse train negative logic 5 – Orthogonal pulse positive logic 6 – Orthogonal pulse negative logic 

5.1.18 Parameter Setting of TECO TSDA Servo Driver

Para. No.	Function	Value	Description		
Pn010-1	Set control mode	1	Value	Control mode	
				CN1 Pin12 open circuit	CN1 Pin12 closed circuit
			0	Speed control	Speed control
			1	Position control	Position control
			2	Torque control	Torque control
			3	Speed control	Speed control
			4	Position control	Position control
			5	Torque control	Torque control
Pn010-2	Set the pulse input format under position control mode	0	Value	The format of pulse input	
			0	Pulse + direction	
			1	Dipulse	
			2	A/B phase difference	
Pn010-3	Set rotation direction of motor	1	Value	Function	
			0	Input positive order, motor rotates CCW.	
			1	Input positive order, motor rotates CW.	
Pn021	Electronic gear ratio numerator	Need calculation	The input pulse amount will be multiplied with this number before output. Ratio range of parameter 21 to 22: 1/127＜parameter 21/ parameter 22＜127		
Pn022	Electronic gear ratio denominator		The input pulse amount will be multiplied with this number before output. Ratio range of parameter 21 to 22:		

Para. No.	Function	Value	Description	
			1/127＜parameter 21/ parameter 22＜127	
Pn011-4	Set the value of Pin20 of CN1	1	Value	Function
			0	Output of “0” speed signal
			1	Output of brake signal
Pn013-1	Set the maximum pulse frequency received by the driver under position control mode	7	It can correct the phenomenon of unauthorized over-travel. Received frequency is divided into 8 segments from 500Kpps to 200Kpps. “0” indicates 500Kpps while “7” 200Kpps.	

CAUTION

For the parameter setting of driver of various brands, refer to the driver manual of specific brand.

5.2 Wiring Diagram of Driver and Terminal Board

Wiring diagrams in this part are the wiring diagrams of control system-axes control-driver motion. When you want to use one axis of the control system to control the motion of two drivers, the wiring diagram is as shown in Figure 2 in section 5.2.2 and Figure 4 in section 5.2.6 (take YASKAWA driver and DELTA driver as an example; for YASKAWA server, its alarm signal wiring is NC type, while for DELTA server, its alarm signal wiring is NO type).

5.2.1 Wiring Diagram of WISE Servo Driver

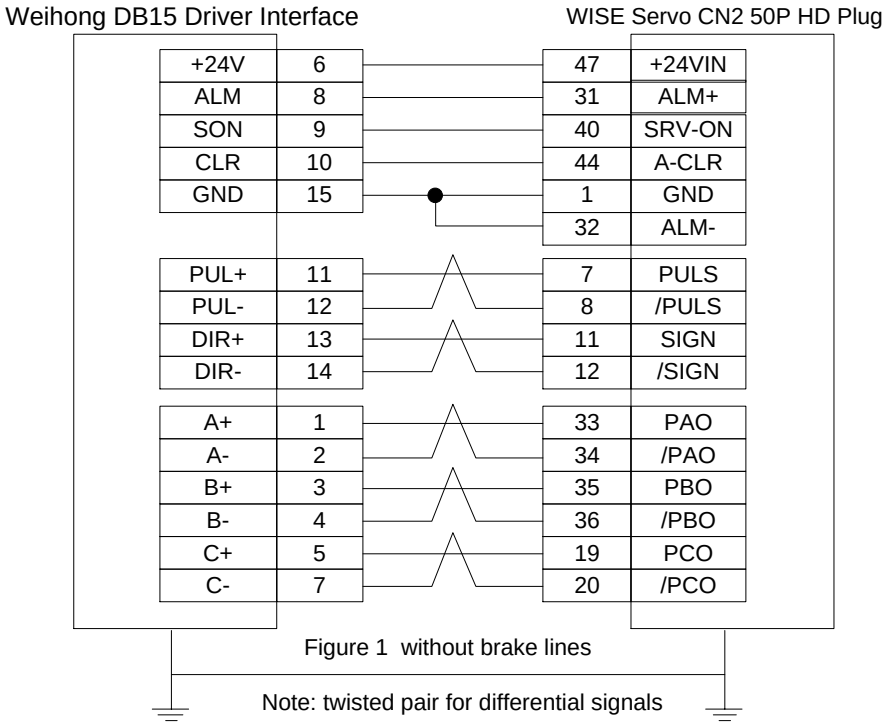


Figure 1 without brake lines

Note: twisted pair for differential signals

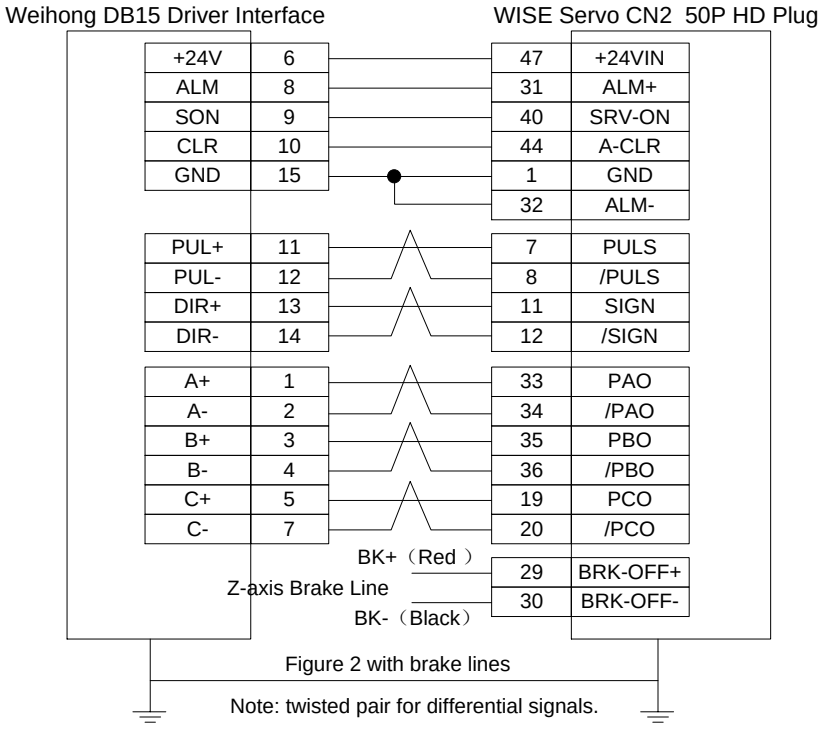


Figure 2 with brake lines

Note: twisted pair for differential signals.

5.2.2 Wiring Diagram of YASKAWA AC Servo Driver

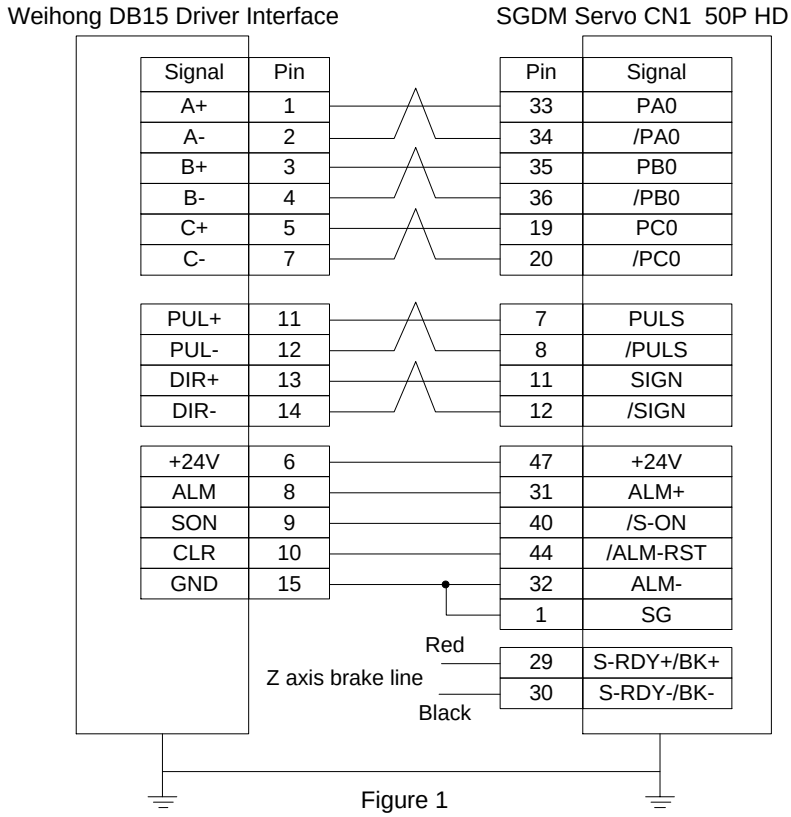


Figure 1

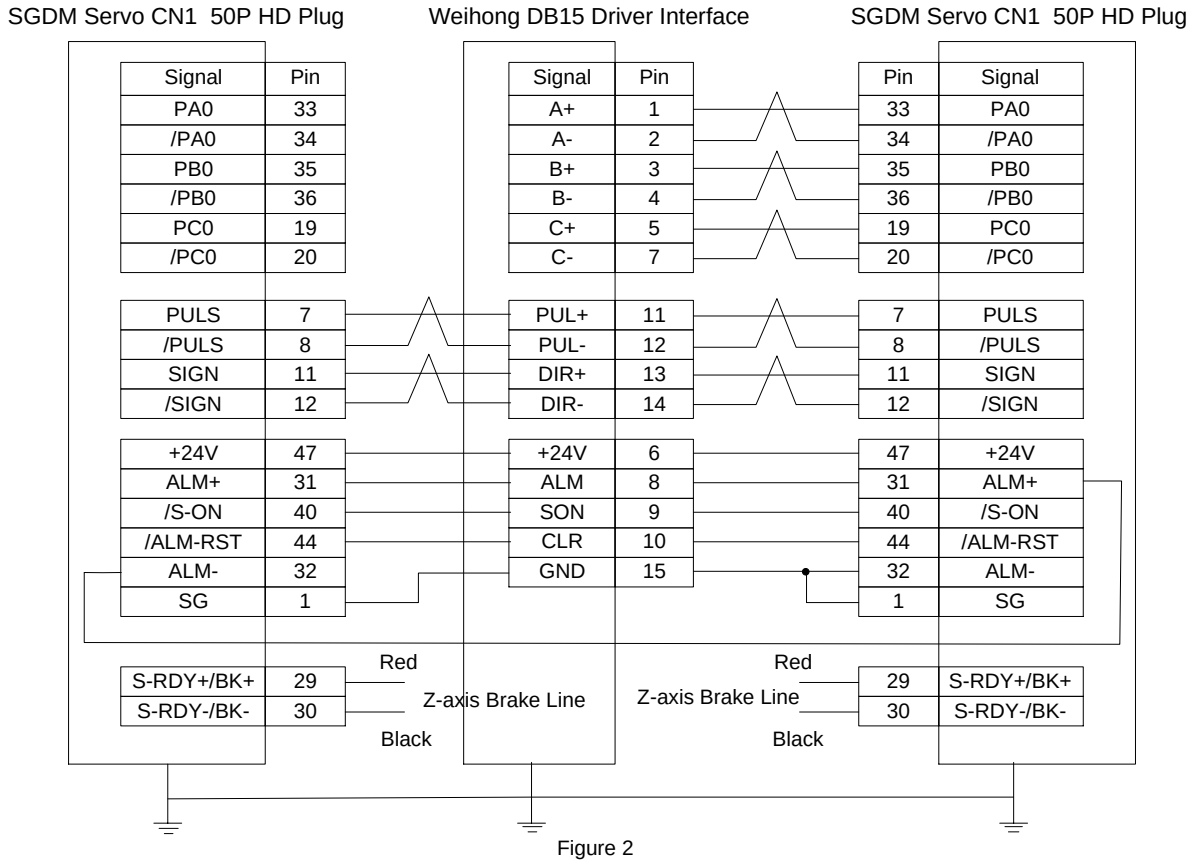
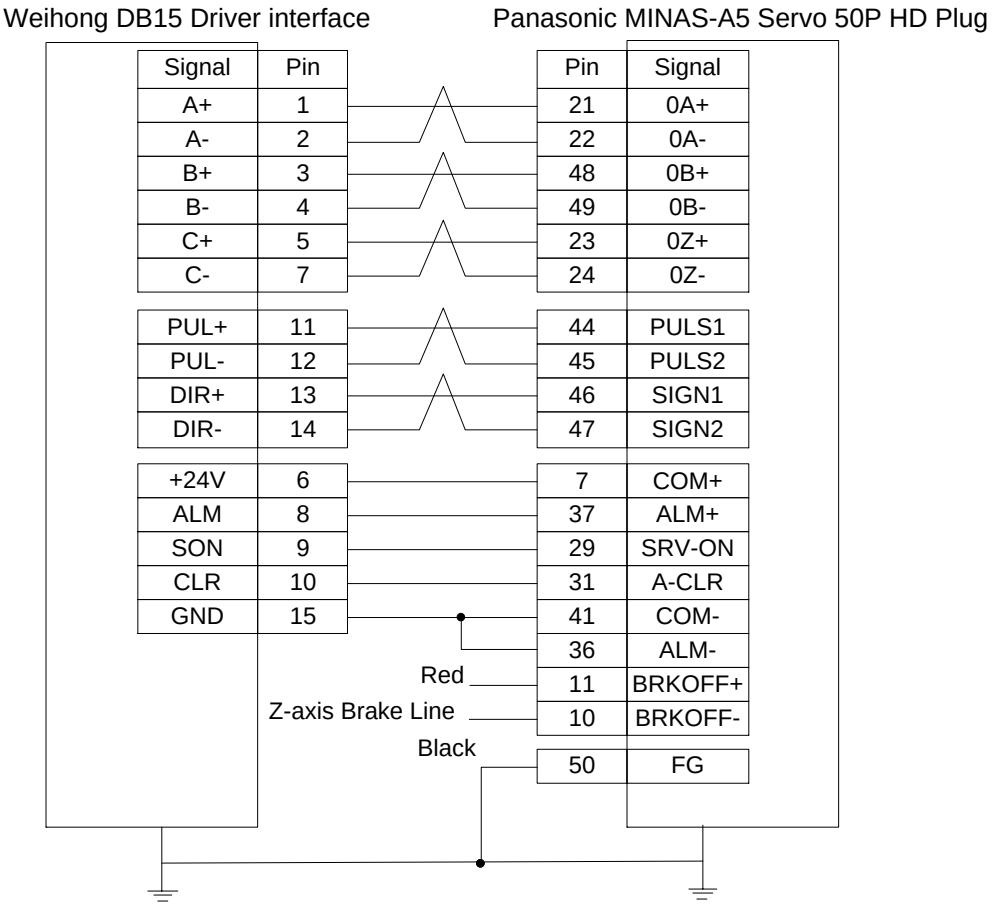


Figure 2

5.2.3 Wiring Diagram of PANASONIC AC Servo Driver



5.2.4 Wiring Diagram of MITSUBISHI MR-JE Servo Driver

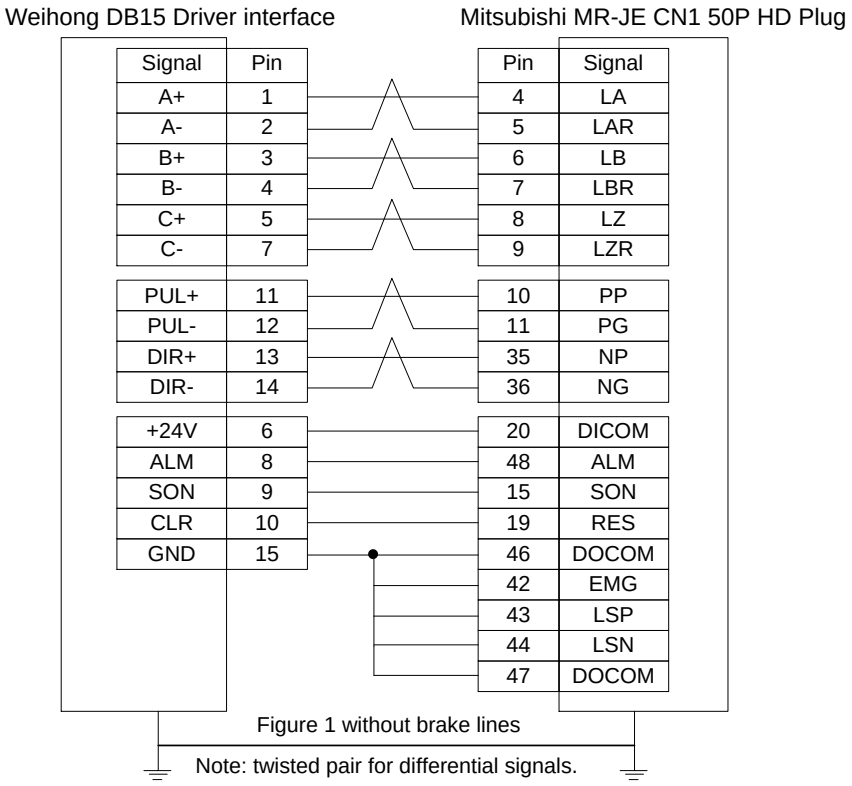


Figure 1 without brake lines

Note: twisted pair for differential signals.

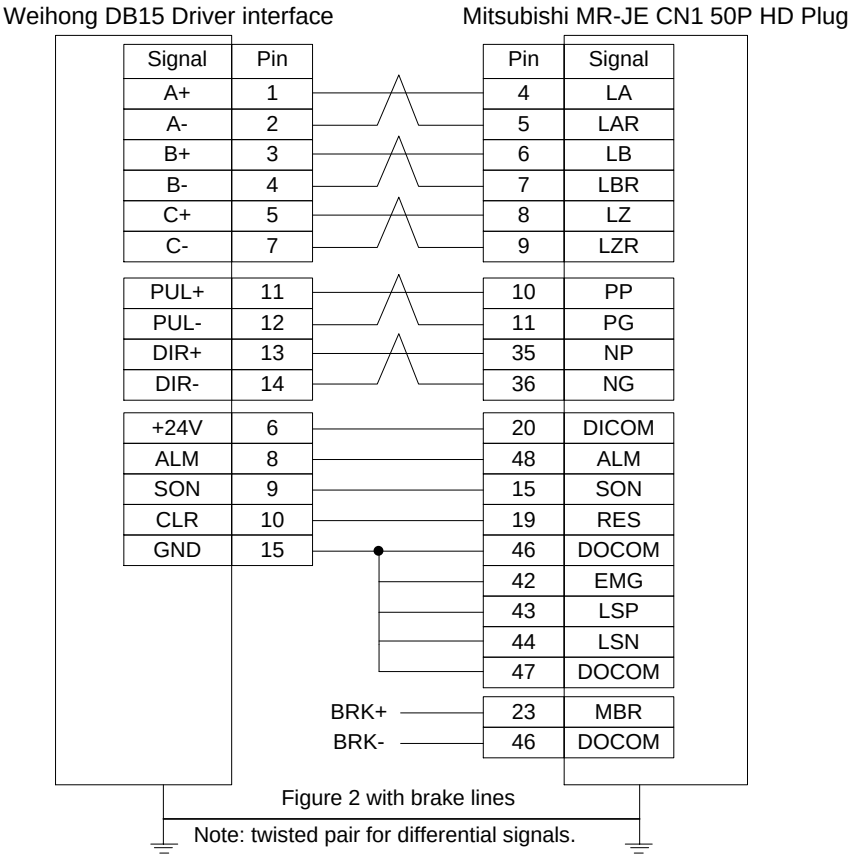
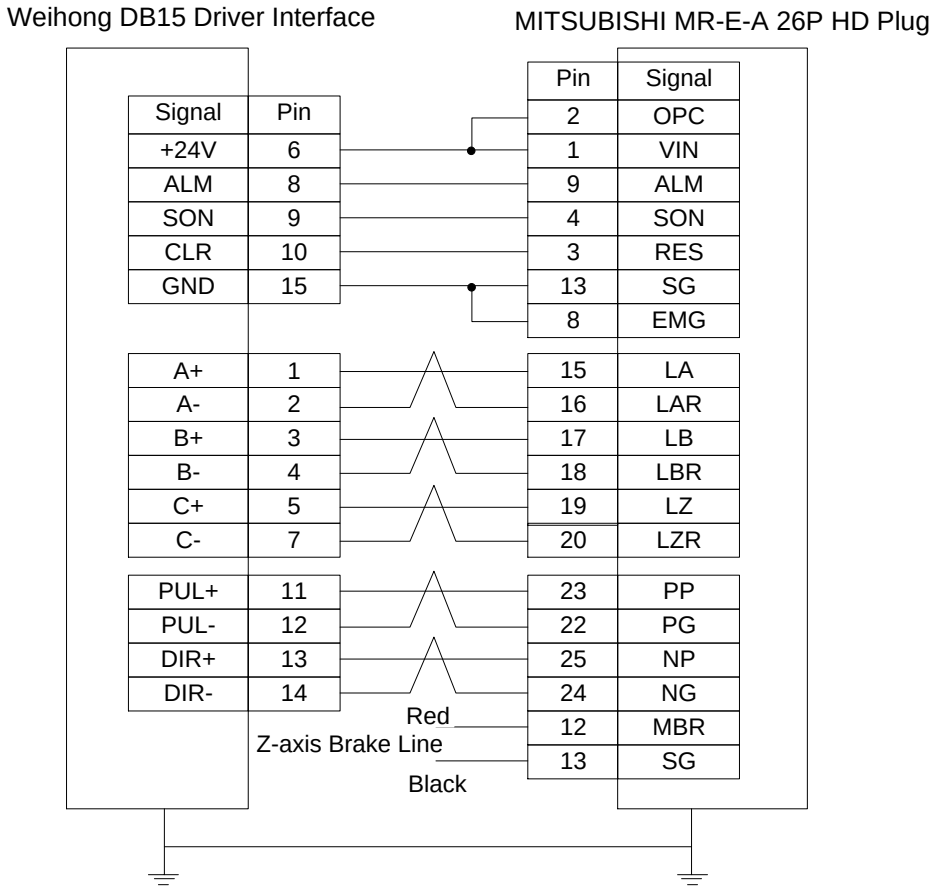


Figure 2 with brake lines

Note: twisted pair for differential signals.

5.2.5 Wiring Diagram of MITSUBISHI MR-E Servo Driver



5.2.6 Wiring Diagram of DELTA Servo Driver

DELTA ASDA-A, ASDA-A2, ASDA-AB share the same wire. Among them, ASDA-A2 and ASDA-AB have the same wiring pin while ASDA-A has the contrary pulse pin, with PULSE 41, /PULSE 43. For detailed parameter setting, refer to section 5.1.8 and section 5.1.9.

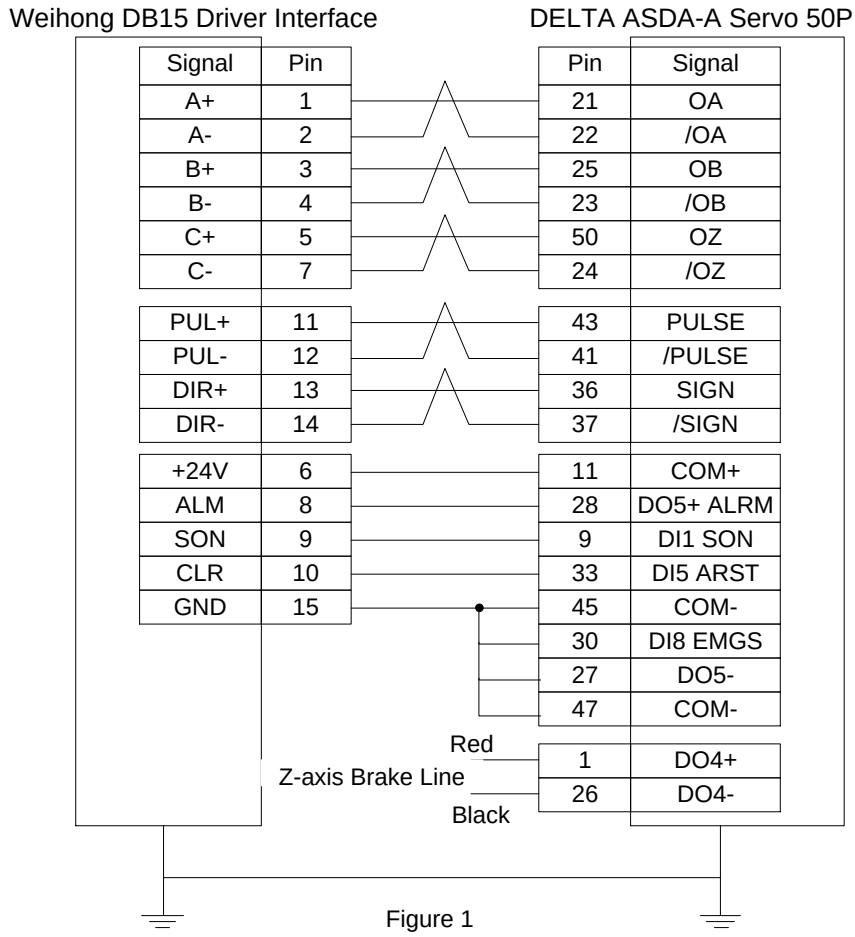


Figure 1

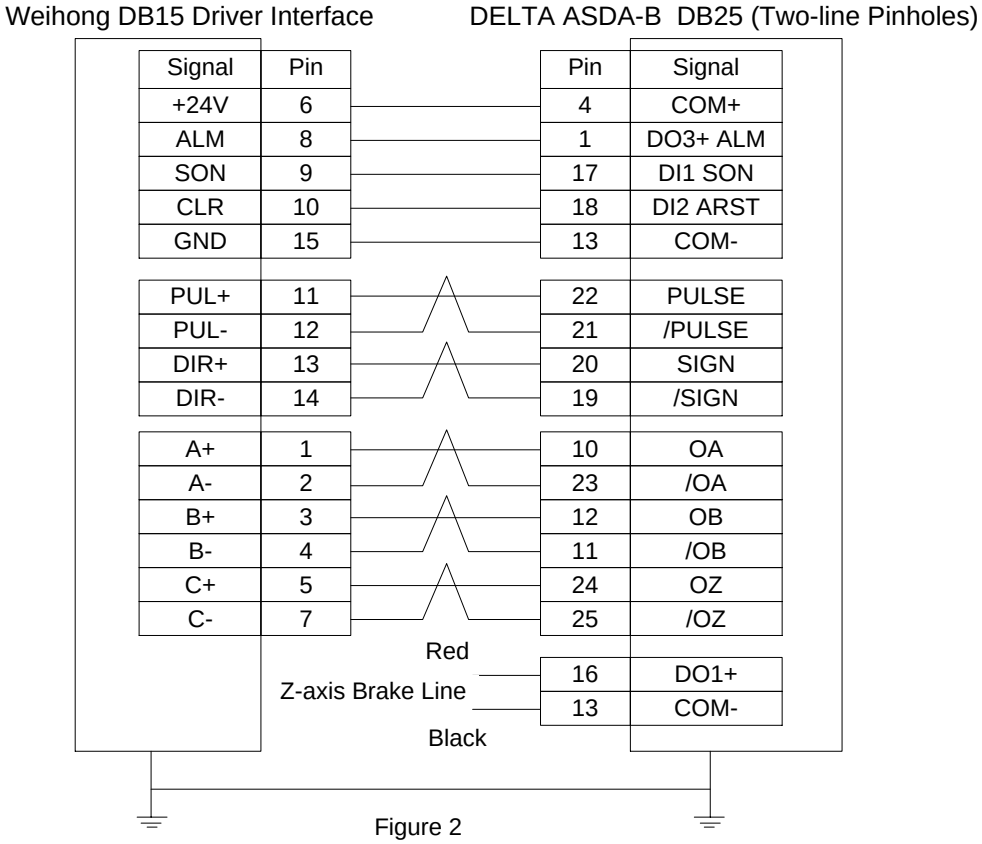


Figure 2

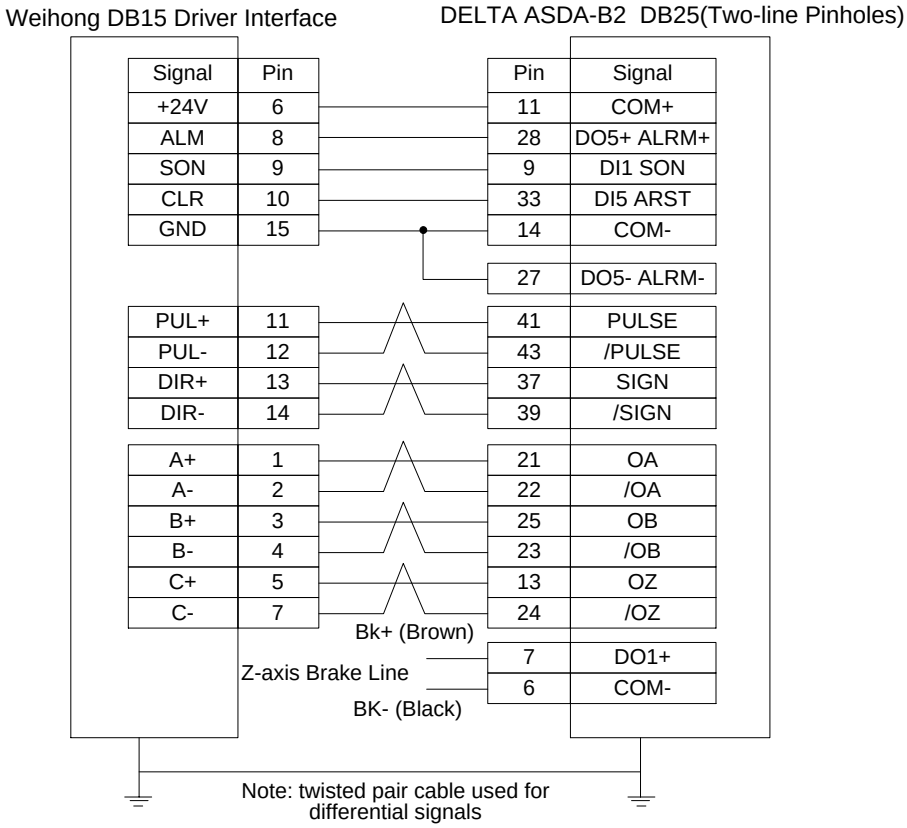


Figure 3

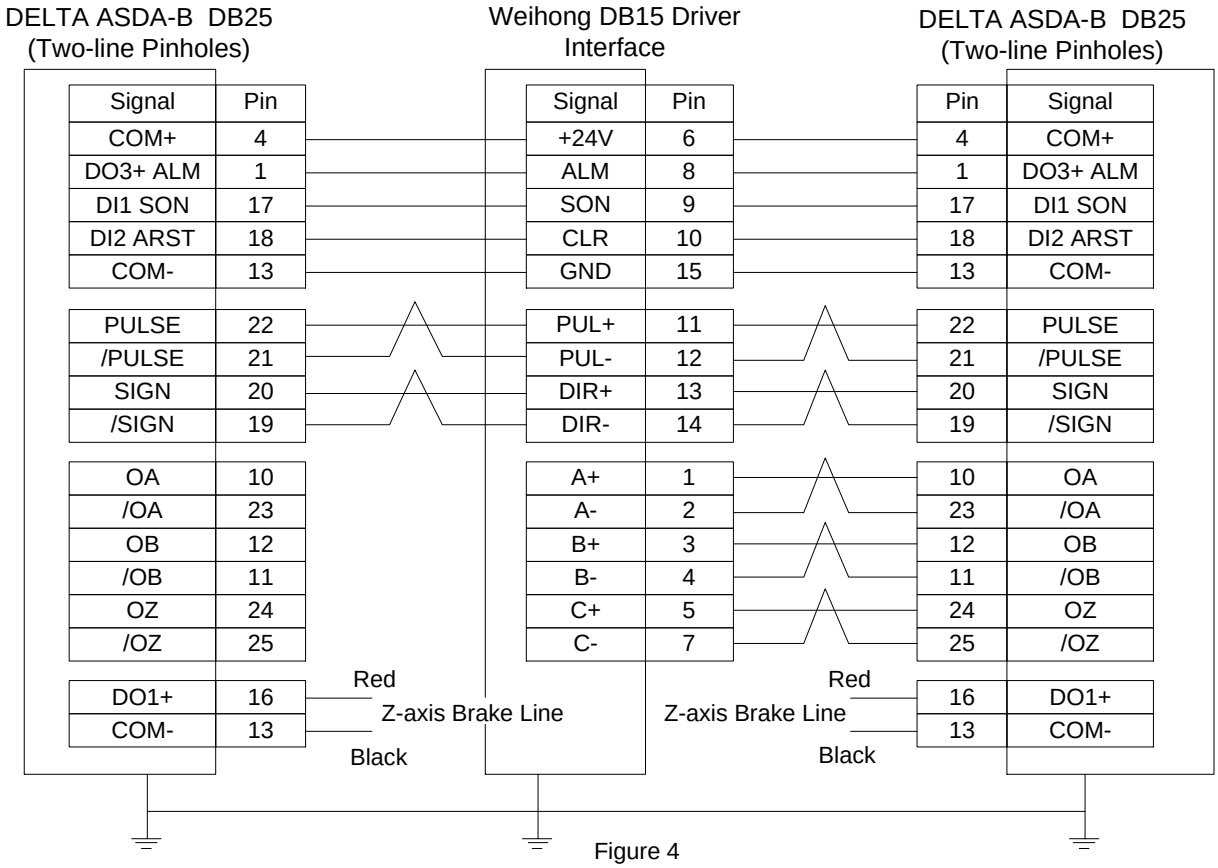
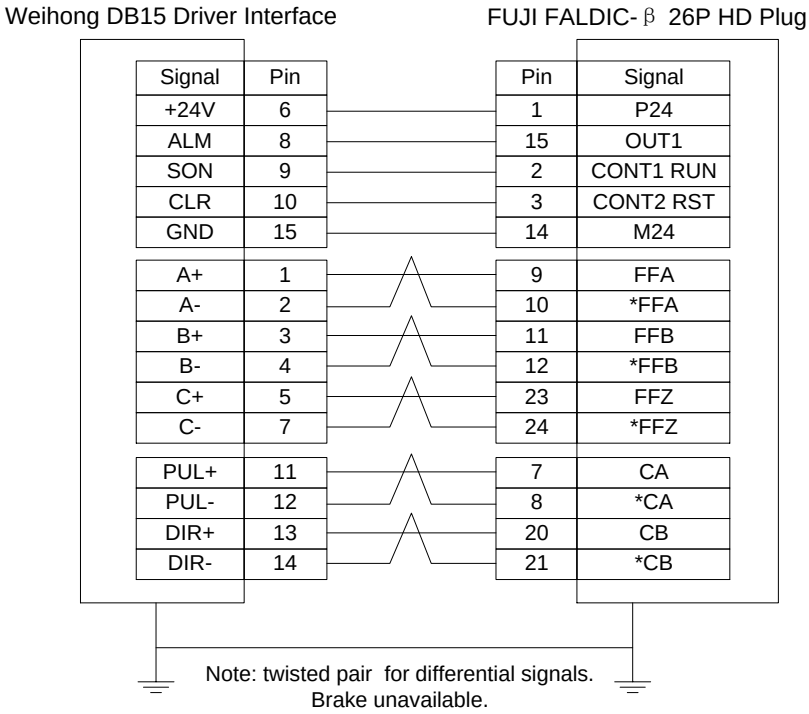
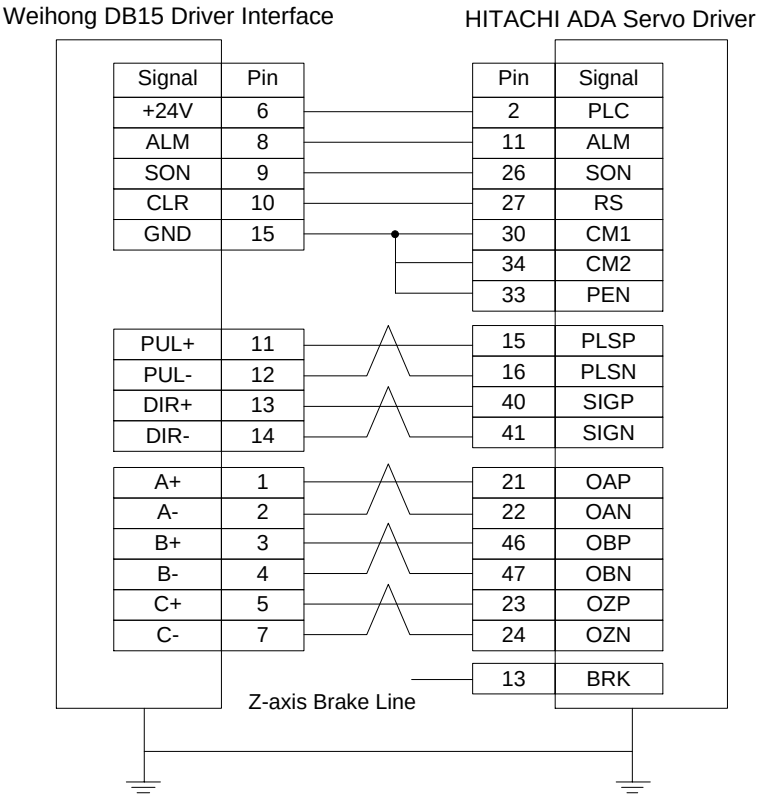


Figure 4

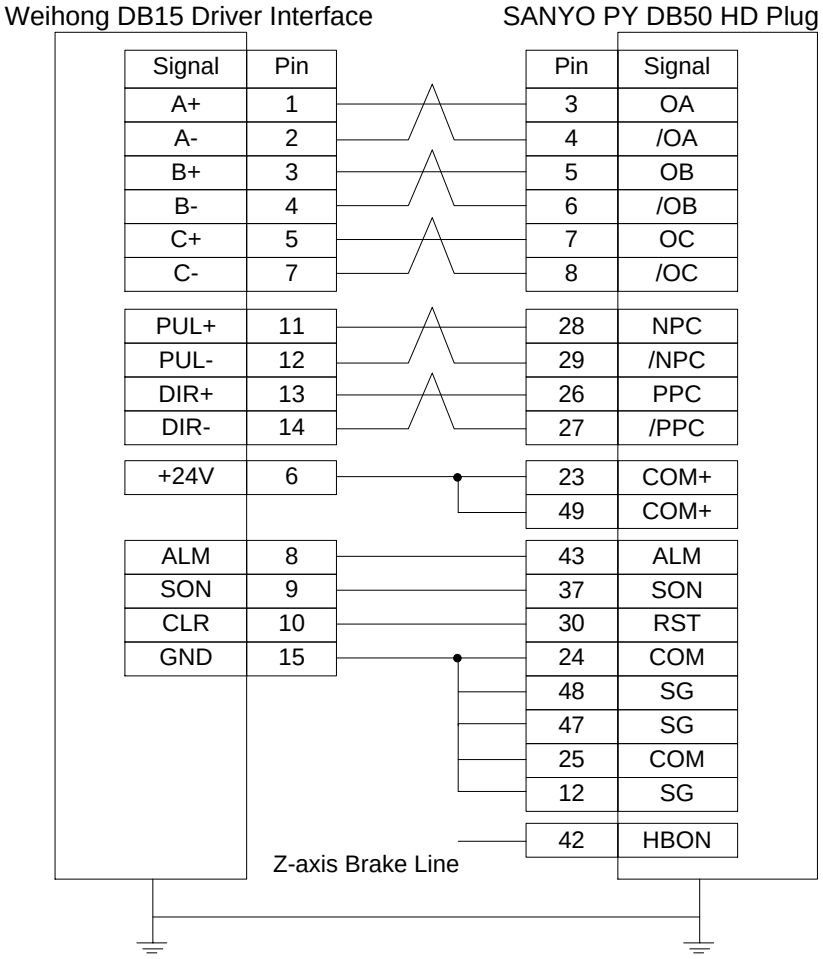
5.2.7 Wiring Diagram of FUJI Servo Driver



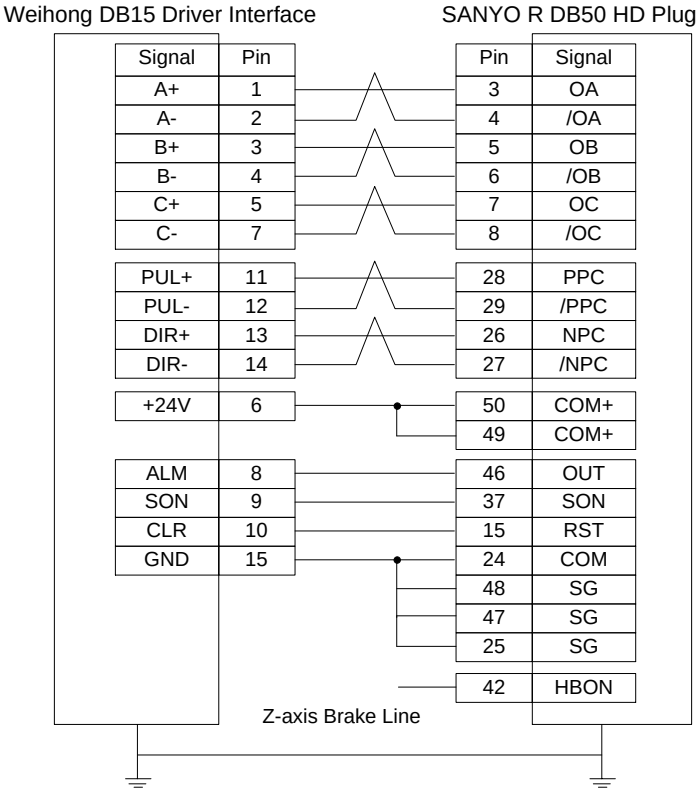
5.2.8 Wiring Diagram of HITACHI Servo Driver



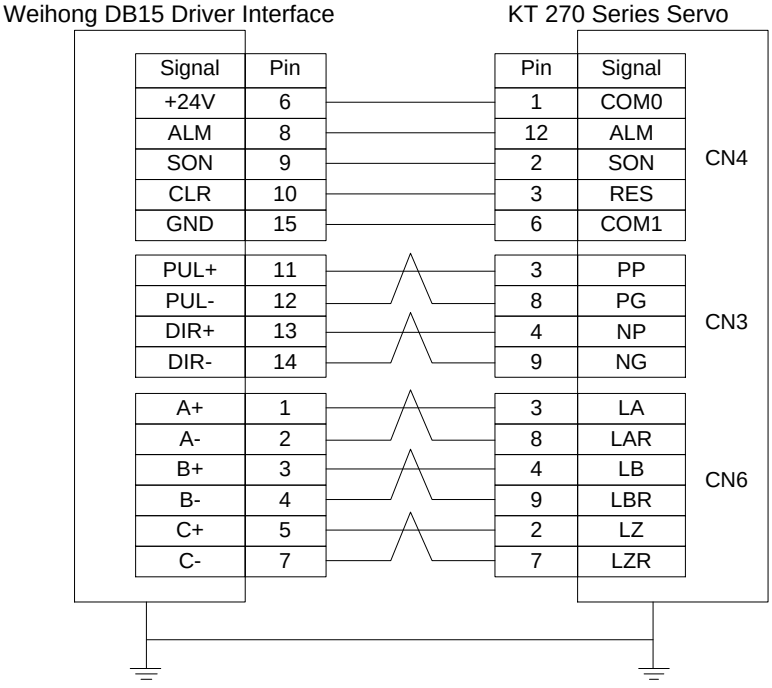
5.2.9 Wiring Diagram of SANYO PY Servo Driver



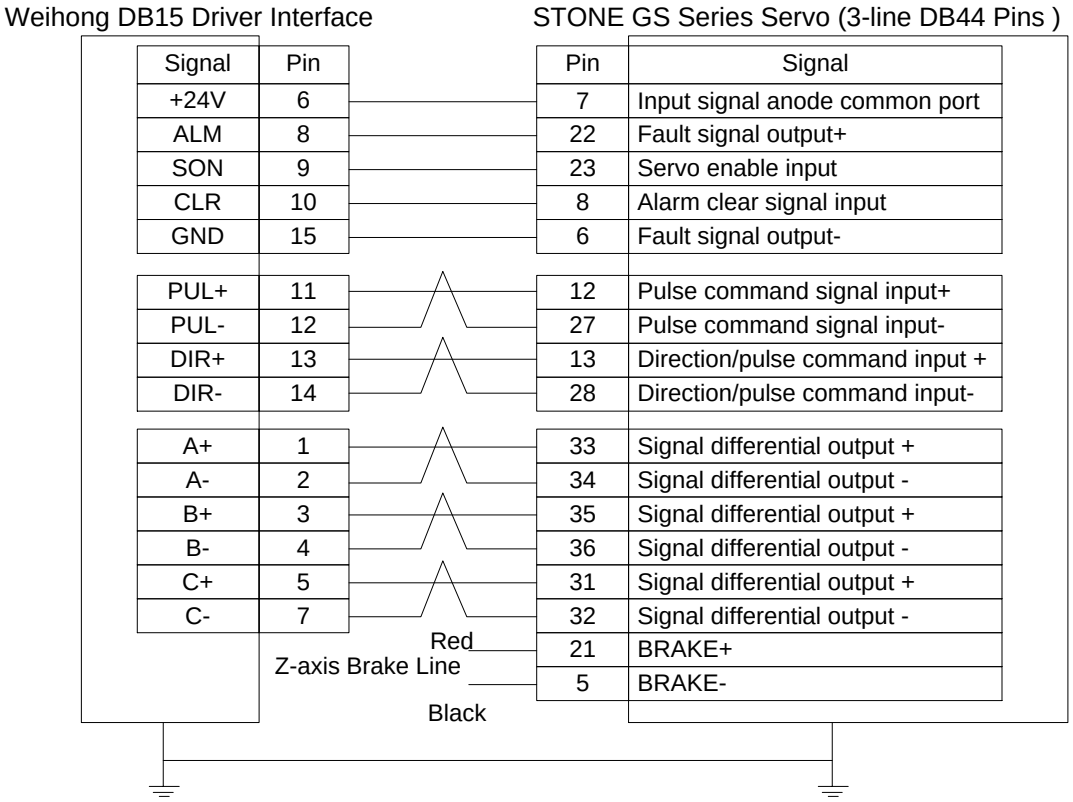
5.2.10 Wiring Diagram of SANYO R Servo Driver



5.2.11 Wiring Diagram of KT270 Servo Driver

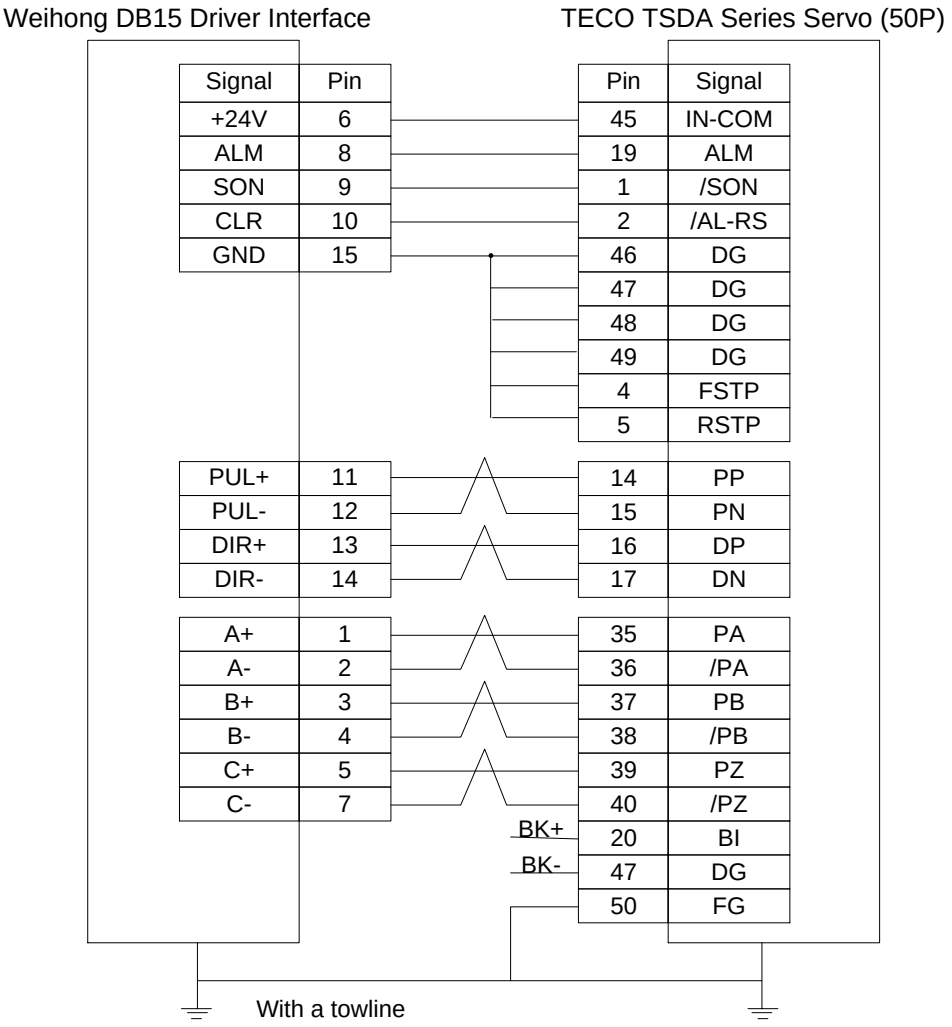


5.2.12 Wiring Diagram of STONE GS Servo Driver

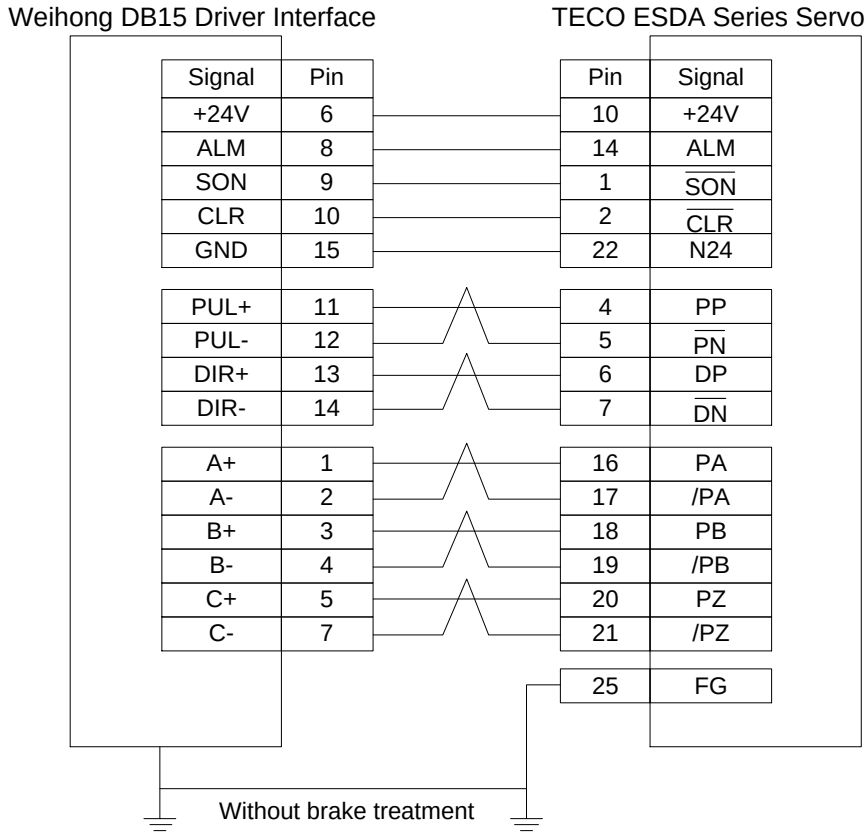


Signal	Pin	Pin	Signal
+24V	6	7	Input signal anode common port
ALM	8	22	Fault signal output+
SON	9	23	Servo enable input
CLR	10	8	Alarm clear signal input
GND	15	6	Fault signal output-
PUL+	11	12	Pulse command signal input+
PUL-	12	27	Pulse command signal input-
DIR+	13	13	Direction/pulse command input +
DIR-	14	28	Direction/pulse command input-
A+	1	33	Signal differential output +
A-	2	34	Signal differential output -
B+	3	35	Signal differential output +
B-	4	36	Signal differential output -
C+	5	31	Signal differential output +
C-	7	32	Signal differential output -
		21	BRAKE+
		5	BRAKE-

5.2.13 Wiring Diagram of TECO TSDA Servo Driver



5.2.14 Wiring Diagram of TECO ESDA Servo Driver



6 Parameters Overview

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
1.0 Axis					
N10000	Axis Direction (X/Y/Z)	1: Positive direction -1: Negative direction	-1	Restart	3.4.1
N10010	Pulse Equivalent Resolution (X/Y/Z)	-0.0000009~999 (mm/p)	0.001	Restart	3.4.2
N10020	Travel Limits-Negative(X/Y/Z)	-99999~99999 (mm)	X: 0 Y: 0 Z: -100	Restart	3.4.3
N10030	Travel Limits-Negative Positive(X/Y/Z)	-99999~99999 (mm)	X: 800 Y: 600 Z: 0	Restart	3.4.3
N10040	Enable Travel Limits(X/Y/Z)	YES: enabled; NO: disabled	YES	Restart	3.4.3
N10050	Positive ToolMeas. Travel Limits (X/Y/Z)	-99999~99999 (mm)	9999	Restart	3.8.4
N10055	Negative ToolMeas. Travel Limits (X/Y/Z)	-99999~99999 (mm)	-9999	Restart	3.8.4
N10060	Enable ToolMeas. Travel Limits	YES: enabled; NO: disabled	NO	Restart	3.8.4
1.1 Encoders					
N11110	Axis Encoder Dir(X/Y/Z)	1: Increasing encoder value; -1: Decreasing encoder value	1	Restart	3.5.2
N11130	Check Encoder Error(X/Y/Z)	YES: enabled; NO: disabled	YES	Restart	3.5.2
N11140	Static Tolerance(X/Y/Z)	1~999999(p)	500	Restart	3.5.2
N11150	Dynamic Tolerance(X/Y/Z)	1~999999(p)	10000	Restart	3.5.2
N11160	Frequency Division Pulses of PG (X4) (X/Y/Z)	1~999999	10000	Restart	3.5.2
N11303	Delay for Stopping On Estop	0.001~10 (s)	1	Immediate	-
	The time for a machine stops completely after E-stop.				
N11304	Encoder Feedback	YES: enabled;	YES	Restart	3.5.2

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
		NO: disabled			
N11309	Delay in Setting REF Sign	0.5~5(s)	0.5	Immediate	-
	The wait time for a machine tool stopping completely after backing to machine origin completed. The REF. Point mark will not be set until after the wait time.				
1.2 Compensation					
N12000	Screw Error Comp	0: No compensation; 1: Single compensation; 2: Double compensation	0	Restart	3.13.3
N12001	Enable Backlash Compensation	YES: enabled; NO: disabled	YES	Restart	3.13.3
N12020	Turn On AQE Compensation	YES: enabled; NO: disabled	NO	Immediate	3.13.4
N12030	Time(Group 0 of 6 groups)	0~10(s)	0	Restart	3.13.4
N12031	Distance(Group 0 of 6 groups)	0~10(mm)	0	Restart	3.13.4
N12032	Delay(Group 0 of 6 groups)	0~10(s)	0	Restart	3.13.4
N12033	Intensity(Group 0 of 6 groups)	0~1	0.75	Restart	3.13.4
1.3 Velo/Acc limits					
N13000	Max Axis Velocity(X/Y/Z)	0.001~100000 (mm/min)	48000	Immediate	3.11.1
1.4 Rotary axis					
N14001	Programming Units	0: Angle(unit: degree); 1: Surface distance of the rotary workpiece(unit: mm)	1	Immediate	-
N14002	Workpiece Diameter	1~3000 (mm)	10	Immediate	-
N14003	Rotary Y Max Velocity	0.001~9999(rpm)	600	Restart	-
N14004	Angular Acceleration	(deg/s^2)	500	Restart	-
N14005	Coordinate Units	0:angle; 1: mm	1	Immediate	-
2.0 Spindle					
N20001	Max Spindle Speed	0~999999 (rpm)	24000	Restart	3.7.1
N20005	Spindle Cool Off Delay Time	0~600 (s)	5	Immediate	3.7.1
N20006	Spindle Speed when Centering	0~100000(rpm)	500	Immediate	3.10.3

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N20010	Spindle On Delay Time	0~60(s)	5	Immediate	3.7.1
N20011	Spindle Off Delay Time	0~60(s)	5	Immediate	3.7.1
4.1 Lubricate					
N41000	Auto Lubricate	YES: Auto on; NO: Not auto on	NO	Immediate	-
	It sets whether the system automatically opens lubrication pump periodically and fills lube.				
N41001	Lubricating Interval	1~1000000 (s)	18000	Immediate	-
	It is the time interval between two start-ups of lubrication pump.				
N41002	Lubricating Duration	1~100 (s)	5	Immediate	-
	It is the filling time of lubrication pump each time.				
4.2					
N42000	Cycle End Inform Type	0; 1; 2	2	Immediate	-
	0: No light; 1: Red lamp lights 3 seconds; 2: Red lamp lights until there is any input from mouse or keyboard.				
N42001	G28 Enable	YES: enabled; NO: disabled	YES	Immediate	-
N42002	Modify Popedom Check	YES: check; NO: do not check	NO	Immediate	-
N42004	Show machining range type	0: show machining range include G00; 1: show machining except G00	0	Immediate	-
N42007	Select Tool Mode	0: Easy mode; 1: Full mode.	0	Immediate	-
N42021	Set delay of prepare workpiece position	0~100000(ms)	0	Restart	-
5.2 Handwheel					
N52001	Precise Pulse Counting	YES: Adopt; NO: Not adopt	NO	Restart	3.16.1
N52002	Handwheel Direction	1: same direction; -1: opposite direction	1	Restart	3.16.1
	The connection between the direction of handwheel and the moving direction of axis: 1: same direction; -1: opposite direction				
N52003	Multiple At X1	0.001~10 (mm)	0.001	Restart	3.16.1
N52004	Multiple At X10	0.001~10 (mm)	0.01	Restart	3.16.1
N52005	Multiple At X100	0.001~10 (mm)	0.1	Restart	3.16.1
N52006	HW Lead Gear (Numerator)	1~1000	1	Restart	3.16.1
N52007	HW Lead Gear	1~1000	1	Restart	3.16.1

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
	(Denominator)				
N52010	Handwheel Acceleration	1~1000 (mm/s^2)	200	Restart	3.16.2
N52012	Deceleration when Switching Axis	YES: decelerate; NO: do not decelerate	YES	Restart	3.16.1
N52013	Forbid HW Reverse Guide	YES: forbid; NO: do not forbid	NO	Restart	3.16.2
5.3 Operation panel					
N53004	Enable Jog Override	YES: enabled; NO: disabled	YES	Restart	-
6.2 G code options					
N62000	Deceleration Distance	0~999 (mm)	2	Immediate	-
N62001	Approach Speed	0.001~99999 (mm/min)	300	Immediate	-
N62020	Enable Arc IJK Programming	YES: enabled; NO: disabled	YES	Immediate	-
N62021	Arc Radius Tolerance	0~9999 (mm)	1	Immediate	-
N62022	Enable Tool Selection by G-code File	YES; NO	NO	Reload program	-
N62090	Exact Stop Tolerance(X/Y/Z)	0~99(mm)	0.001	Immediate	-
N62410	Enable Cutter Compensation	YES: enabled; NO: disabled	NO	Immediate	3.13.1
N62411	Cutter Compensation Type	1: Normal type; 2: Intersect type; 3: Insert type	1	Immediate	3.13.1
N62412	Cutter Compensation Direction	0: None 1: Left 2: Right	1	Immediate	3.13.1
N62413	Num Of Intervene Detected Graphics	1~5	3	Immediate	3.13.1
N62414	Intervene Evade Enable	YES: enabled; NO: diabled	NO	Immediate	3.13.1
N62730	G73_G83 Lifting Distance	-99999~99999 (mm)	0	Immediate	-
N62760	G76_G87 Stop Orientation	0: G17 +X; 1: G17 –X; 2: G17 +Y; 3: G17 –Y	0	Immediate	-
N62765	T0 Process Way	0: Invalid; 1: Empty tool, into a none tool state; 2: Starting tool number 0	0	Restart	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
6.3 Trajectory					
N63000	Look Ahead Distance	0~999(mm)	0.5	Immediate	-
N63001	Connect Speed LEP Look Ahead Distance	0~0.05(mm)	0	Immediate	-
N63002	Delay for Exact Stop	0~999 (s)	0	Immediate	0
N62003	Max COON Angle Num	0~1000	100	Immediate	-
N63006	Path Smoothing Time	0~0.064 (s)	0	Immediate	0
N63007	Trace Pretreatment Options	0, 1, 2	0	Immediate	-
N63008	Trace Pretreatment Precision	0~0.1(mm)	0	Immediate	-
N63009	Max Angle of Trace Pretreatment	0~180(deg)	180	Immediate	-
6.4 Speed/Acc					
N64000	Startup Speed	0~100000 (mm/min)	0	Immediate	0
N64020	Traverse Rate	0~100000 (mm/min)	3000	Immediate	0
N64060	Max Feedrate	0~100000 (mm/min)	48000	Immediate	0
N64100	Axial Acceleration	0.001~100000 (mm/s^2)	800	Immediate	0
N64101	Rapid Motion Axial Acceleration	0.001~100000 (mm/s^2)	800	Immediate	0
N64102	Z Axis Acceleration	0.001~100000 (mm/s^2)	800	Immediate	0
N64103	Speed Up Acceleration		800	Immediate	0
N64104	Speed Down Deceleration		800	Immediate	0
N64120	Acceleration for Corners	0.001~100000 (mm/s^2)	3800	Immediate	0
N64150	Axial Jerk	0.001~1*10^{11} (mm/s^3)	150000	Immediate	0
N64200	Smoothing The Path Velocity	YES: enabled; NO: disabled	YES	Immediate	0
N64201	MAX Angle Smooth Velocity	0~180	90	Immediate	0
	When the connection angle of two segments is larger than the value of the parameter, the system will start at startup speed, instead of smoothing the path velocity.				
N64203	Path Interpolation Algorithm	1; 2; 3	3	Immediate	0
	Its setting range: 0: trapezoid algorithm 1: S-type algorithm 2: LEP algorithm 3: acceleration trapezoid algorithm.				
N64204	Acc or Dec Time after Interpolation	0~99999	0.005	Immediate	0

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
	The longer the time is, the smoother the velocity will be. This parameter has no effect on the track precision.				
N64205	Min Velocity in LEP	0~100000	60	Immediate	0
N64207	Arc Velocity Limit	YES: enabled NO: disabled	YES	Immediate	0
N64208	Max Velocity of Reference Circle	0.001~100000 (mm/min)	3600	Immediate	0
N64209	Min Velocity of Arc	0.001~100000 (mm/min)	180	Immediate	0
N64240	Smoothing Time Factor	0.01~10	1	Immediate	0
N64241	Connect Speed decreased at Max Connect Angle	YES: enabled; NO: disabled	YES	Immediate	0
N64245	Prepared number of path when optimizing performance	1~2000	300	Immediate	0
N64246	Slide speed for small lines	YES: enabled; NO: disabled	NO	Immediate	0
N64247	Reference length of slide speed for small lines	0.001~10	1	Immediate	0
N64248	Enable slide optimization	YES: enabled; NO: disabled	YES	Immediate	0
N64249	Velocity Smooth for Single Axis	YES: enabled; NO: disabled	YES	Restart	0
6.5 File translation					
PLT File Param					
N65000	Retract	0~99999 (mm)	5	Reload program	3.15.4
N65001	PLT Units	0.001~99999	40	Reload program	3.15.4
N65002	Tool Offset	0.0001~99999 (mm)	0.025	Reload program	3.15.4
N65003	Cutting Depth	-99999~0 (mm)	0	Reload program	3.15.4
DXF File Param					
N65100	Retract	0~99999 (mm)	5	Reload program	3.15.4
N65101	Cutting Depth	-99999~0 (mm)	0	Reload program	3.15.4
N65102	Layer Depth	-99999~0 (mm)	0	Reload	3.15.4

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
				program	
N65103	First Point As Origin	YES: use; NO: do not use	YES	Reload program	3.15.4
N65104	By Contour	YES: enabled; NO: disabled	NO	Reload program	3.15.4
N65105	Enable Bottom Cutting	YES: enabled; NO: disabled	NO	Reload program	3.15.4
N65106	Use Metric	YES: Forcibly use; NO: Not forcibly use	NO	Reload program	3.15.4
ENg File Param					
N65200	Retract (ENG)	0~99999 (mm)	5	Reload program	3.15.4
N65201	Prompt For Tool Change	YES: Prompt; NO: Do not prompt	YES	Reload program	3.15.4
N65203	Cutting By Tool Number	YES: use; NO: do not use	NO	Reload program	3.15.4
N65204	Deep Hole Cutting Type	0: Reciprocating chip removal 1: High-speed reciprocating chip removal; 2: Up to a safe altitude	0	Reload program	3.15.4
N65205	Lifting Distance	0~99999(mm)	1	Reload program	3.15.4
N65206	Force To Use Tool Compensation	YES: force; NO: do not force	YES	Reload program	3.15.4
N65207	Modify by Tool Number	YES: enabled; NO: disabled	NO	Reload program	3.15.4
N65208	Tool Deepen Type	0; 1	1	Reload program	3.15.4
	The type of Z-axis downward feed at the beginning of machining an ENG file: 0: deepen to moving height; 1: deepen to AreaMax -1.				
N65209	Lifts when Change Tool	YES: enabled; NO: disabled	YES	Reload program	3.15.4
N65210	Ignore Coordinate System Instruction	YES: Ignore; NO: Do not ignore	NO	Immediate	3.15.4
N65211	Z Up Type After Drill	0;1	0	Immediate	3.15.4
	0:Up to R Plane; 1: Up to specified workpiece coordinate position, just ENG file allowed.				

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N65212	Z Pos After Drill	mm	10	Immediate	3.15.4
N65213	Depth Tool Speed Way	0: Machining speed way; 1: Fast across move speed way	0	Reload Program	3.15.4
N65215	Force G00 Retract to [Lift Height]	YES: Force; NO: Do not force	NO	Reload Program	3.15.4
6.6 Change tool					
N66000	Prompt for Tool Change	YES: enabled; NO: disabled	NO	Immediate	3.19.3
N66002	Pause in Tool Change for Same Active and Target Tool No.	YES: Pause; NO: Do not pause	NO	Immediate	3.19.3
N66005	Upper Position	-99999~99999 (mm)	0	Immediate	3.19.3
N66006	Lower Position	-99999~99999 (mm)	0	Immediate	3.19.3
N66007	Spindle Position X	-99999~99999 (mm)	9999	Immediate	3.19.3
N66008	Spindle Position Y	-99999~99999 (mm)	9999	Immediate	3.19.3
N66009	Spindle Position Z	-99999~99999 (mm)	9999	Immediate	3.19.3
N66010	Deceleration Position X	-99999~100000 (mm)	0	Immediate	3.19.3
N66011	Deceleration Position Y	-99999~100000 (mm)	0	Immediate	3.19.3
N66012	Deceleration Position Z	-99999~100000 (mm)	0	Immediate	3.19.3
N66015	Tool Change Speed	0~100000 (mm/min)	3000	Immediate	3.19.3
N66016	Z-axis Speed	0~13740 (mm/min)	1800	Immediate	3.19.3
N66017	Automatic Tool Measure	YES: enable Automatic tool measure NO: disable Automatic tool measure	YES	Immediate	3.19.3
N66020	Tool Magazine Tye	0: No Tool Magazine; 1: Disk Tool Magazine;	0	Restart	3.19.3

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
		2: Linear Tool Magazine			
N66021	Tool Magazine Capacity	1~255	8	Immediate	3.19.3
N66022	Check Change ToolNo	YES; NO	YES	Immediate	3.19.3
	YES: Limit DeskToolNo in range of (0,256); NO: Limit range, keep ToolNo unchanged.				
N66030	Tool Count Port	PLC address	NA	Immediate	-
N66031	Tool Mag. Back to Origin Port	PLC address	NA	Immediate	-
N66032	Tool Mag. CW Port	PLC address	NA	Immediate	-
N66033	Tool Mag. CCW Port	PLC address	NA	Immediate	-
N66034	Mag. CW to Origin Delay	0~5000(ms)	0	Immediate	-
N66035	Mag. CCW to Origin Delay	0~5000(ms)	0	Immediate	-
N66036	Tool Count CW Delay	0~5000(ms)	0	Immediate	-
N66037	Tool Count CCW Delay	0~5000(ms)	0	Immediate	-
Matrix Tool Slot Param (1st goup of 21 goups)					
N66074	Tool Position X	(mm)	0	Immediate	3.19.3
N66075	Tool Position Y	(mm)	0	Immediate	3.19.3
N66076	Tool Position Z	(mm)	0	Immediate	3.19.3
6.7					
N67000	Negative Change Tool Travel Limits X	(mm)	-10000	Immediate	3.4.3
N67001	Negative Change Tool Travel Limits Y	(mm)	-10000	Immediate	3.4.3
N67002	Negative Change Tool Travel Limits Z	(mm)	-10000	Immediate	3.4.3
N67010	Positive Change Tool Travel Limits X	(mm)	10000	Immediate	3.4.3
N67011	Positive Change Tool Travel Limits Y	(mm)	10000	Immediate	3.4.3
N67012	Positive Change Tool Travel Limits Z	(mm)	10000	Immediate	3.4.3
N67020	Enable Change Tool Travel Limits(MCS)	YES: enabled; NO: disabled.	NO	Restart	3.19.3
7.1 Manu					
N71000	Slow Jog Speed	0~N71001 (mm/min)	1200	Immediate	0
N71001	Rapid Jog Speed	0~N13000 (mm/min)	3000	Immediate	0
N71002	Max Jog Speed Before	0~ “Rapid Jog Speed”	1200	Immediate	0

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
	Back to REF Point				
7.2 Auto					
N72001	Ignore Programed Feedrate	YES: ignore NO: do not ignore	NO	Immediate	3.11.1
N72002	Ignore Programed Spindlerate	YES: ignore NO: do not ignore	NO	Immediate	3.7.1
N72003	Fix Traverse Rate Override	YES: fix NO: do not fix	NO	Immediate	3.11.2
N72004	Spindle Off when Cycle Stop	YES: off; NO: on	YES	Immediate	3.7.1
N72008	Spindle On when Cycle Start	YES: on; NO: off	YES	Immediate	3.7.1
N72009	Cycle Machining Interval	0~1000	10	Immediate	-
N72010	Enable Work Coordinate Limits(WCS) (X/Y/Z)	YES: enable; NO: disable	YES	Immediate	-
N72020	Negative Work Coordinate Limits(X/Y/Z)	(mm)	-99999	Immediate	-
N72030	Positive Work Coordinate Limits(X/Y/Z)	(mm)	99999	Immediate	-
7.3 Pause					
N73000	Z Axis Return Feedrate after Pause	0~100000 (mm/min)	600	Immediate	-
N73001	Z Axis Lifting Feedrate on Pause	0~100000 (mm/min)	600	Immediate	-
N73002	Z Axis Lifting Mode on Pause	0; 1; 2; 3	0	Immediate	-
	Setting range: 0: lift to distance set in parameter; 1: lift to work coordinate set in parameter; 2: lift to Mach coordinate set in parameter; 3: lift to fixed position set in parameter.				
N73003	Z-axis Lifting Pos in WCS	0~9999 (mm)	10	Immediate	-
N73004	Lifting Distance on Pause	0~500 (mm)	10	Immediate	-
N73005	Stop Spindle On Pause	YES: stop; NO: do not stop	YES	Immediate	3.7.1
N73006	Z-axis Lifting Pos in MCS	-100~0 (mm)	0	Immediate	-
N73007	Return to Fixture X on Pause	-99999~99999	0	Immediate	-
N73008	Return to Fixture Y on Pause	-99999~99999	0	Immediate	-
N73009	Return to Fixture Z on Pause	-99999~99999	0	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
7.4 Return Machine Home					
N74000	Cancel REF Sign when reset	YES: cancel; NO: do not cancel	YES	Immediate	3.6.5
N74001	Back to REF Required	YES: required; NO: not required	YES	Immediate	3.6.5
N74002	Cancel REF Sign when E-stop	YES: cancel; NO: do not cacel	YES	Immediate	3.6.5
N74010	Home Offset (X/Y/Z)	0~N10030 (mm)	0	Restart	3.6.5
N74020	Home Search Dir. (X/Y/Z)	1: Positive direction -1: Negative direction	X: -1; Y: -1; Z: 1	Immediate	3.6.5
N74030	Home Search Velocity(X/Y/Z)	0.001~10000 (mm/min)	1800	Immediate	3.6.5
N74040	Home Switch Inport Adddress(X/Y/Z)	X: 00117; Y: 00120; Z: 00123	X: 00117; Y: 00120; Z: 00123	Immediate	-
	The PLC addresses of input port of coarse positioning switch in each axis.				
N74050	Home Latch Dir. (X/Y/Z)	1: Positive direction -1: Negative direction	X: 1 Y: 1 Z: -1	Immediate	3.6.5
N74060	Home Latch Velocity (X/Y/Z)	0.001~10000 (mm/min)	60	Immediate	3.6.5
N74070	Index Pulse Inport Address(X/Y/Z)	X: 00000; Y: 00001; Z:00002	X: 00000; Y: 00001; Z: 00002	Immediate	-
	The PLC addresses of input port of fine positioning switch in each axis.				
N74080	Back Off Distance (X/Y/Z)	-1000~1000 (mm)	2	Immediate	3.6.5
N74090	Home Latch Count	1~100	1	Immediate	3.6.5
N74100	Leadscrew Pitch(X/Y/Z)	0~360 (mm)	5	Immediate	3.6.5
N74110	Coarse/Fine Switches Min Dist(X/Y/Z)	0~thread pitch/2 (mm)	1	Immediate	3.6.5
N74120	Coarse/Fine Pos Distance Tolerance	0~100 (%)	10	Immediate	3.6.5
N74130	Max Distance during REF Positioning(X/Y/Z)	Thread pitch~99999	50	Immediate	3.6.5-
7.5 Measure					
N75000	Probe Input Port Addr	00016	00016	Immediate	-
	The PLC address of the input port Tool Presetter Signal.				
N75001	ToolMea Fine Speed	(mm/min)	60	Immediate	3.8.4
N75002	ToolMea Fine time	-	1	Immediate	3.8.4

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N75020	ToolMea Result Tolerance	0~10	0.1	Immediate	3.8.4
N75023	Disconnect Probe Input Port	-1; 1	-1	Immediate	-
	Specifying whether the software with the function of disconnecting with the wire of tool presetter. If there is this function, this port will be given a signal to disconnect with the wire of tool presetter in machining, while connecting with the wire in tool measurement. 1: Without the function of disconnecting with the wire of tool presetter; -1: With the function of disconnecting with the wire of tool presetter				
N75024	ToolMea Overtravel Port Addr	00124	00124	Restart	-
	The PLC address of the tool presetter over-travel protection port.				
N75025	ToolMea Overtravel Alarm	YES: enabled; NO: disabled	YES	Restart	3.8.33. 8.4
N75100	Mobile Probe Surface to WCS Z0	-1000~1000 (mm)	0	Immediate	3.8.4
N75201	Fixed Probe Surface to WCS Z0	(mm)	10	Immediate	3.8.3
N75203	Fixed Preset Speed	(mm/min)	300	Immediate	3.8.3
7.9 Operation others					
N79000	Z Down Feedrate Limitation Mode	0; 1; 2	0	Immediate	-
	Setting range: 0: No limitation; 1: Limit when only Z-axis down; 2: Limit when including Z-axis down motion.				
N79001	Z Down Federate Limitation	0~100000 (mm/min)	480	Immediate	-
N79003	Safe Height	0~1000 (mm)	10	Immediate	-
N79004	Forced to Use the Z Feed	YES; NO	YES	Immediate	-
N79100	Stop Mode when Cycle Completed	0: Stay where it is; 1: Back to fixed point 2: Back to workpiece origin.	0	Immediate	-
N79101	Run T and M3, M4, M5 Code Before Resume	YES: run; NO: do not run	NO	Immediate	-
	Whether to run T code and M3, M4, M5(Spindle On/Off code) when breakpoint resume or advanced start.				
N79110	Fixed Point Position(X/Y/Z)	-99999~99999 (mm)	0	Immediate	-
N79120	Second Fixed Point Position(X/Y/Z)	-99999~99999 (mm)	0	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
8.0 User interface					
N80002	Support Extension Workpiece Offsett	YES: Support; NO: Not support	NO	Restart	3.9.2
N80003	Support Program Task	YES: Support; NO: Not support	NO	Restart	3.15.4
N80004	Print Info	YES; NO	NO	Immediate	3.5.2
N80007	Page Jump After Load File	0; 1	0	Immediate	-
	0: Jump to “Coor-Auto” page; 1: Jump to “Coor-Program” page				
N80018	Use New Frp Algorithm	YES: Use; NO: Do not use	NO	Immediate	-
N80030	Use Old Mainpage	YES: Use; NO: Do not use	YES	Restart	3.2
N80032	Calibration Type	0: Mob Calibration Type; 1: Fixed Calibration Type; 2: First Exchanged Calibration Type	1	Immediate	-
8.1 Position view					
N81000	Auto Load Graph	YES; NO	NO	Immediate	3.12.4
N81001	Max File Size	(KB)	1000	Immediate	3.12.4
N81004	Allow Spindle-On when Centering	YES; NO	YES	Restart	3.10
N81005	Show Remain Time	YES; NO	YES	Immediate	3.17.4
N81006	Keep Register Type Remain	YES; NO	NO	Restart	3.17.4
N81007	Auto Restart After Register	YES; NO	YES	Immediate	3.17.4
N81010	Gradient Fill	YES; NO	YES	Immediate	3.12.4
N81011	Draw Workbench	YES; NO	NO	Immediate	3.12.4
N81012	Draw Grid	YES; NO	NO	Immediate	3.12.4
N81013	2D Mode	YES; NO	NO	Immediate	3.12.4
N81015	Clear On Loading	YES; NO	YES	Immediate	3.12.4
N81016	Draw WC Origin	YES; NO	NO	Immediate	3.12.4
N81017	Draw MC Origin	YES, NO	NO	Immediate	3.12.4
N81018	Bkground Color 1	Select a color	0×00000000	Immediate	3.12.4
N81019	Bkground Color 2	Select a color	0×00000000	Immediate	3.12.4
N81020	G00 Color (running)	Select a color	0×0000FFFF	Immediate	3.12.4

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N81021	G01 Color (running)	Select a color	0×00FFFF00	Immediate	-
N81022	G02 Color (running)	Select a color	0×00FFFF00	Immediate	-
N81023	G03 Color (running)	Select a color	0×00FFFF00	Immediate	-
N81032	G00 Color (loading)	Select a color	0×04000000	Immediate	3.12.4
N81033	G01 Color (loading)	Select a color	0×00600000	Immediate	-
N81034	G02 Color (loading)	Select a color	0×00600000	Immediate	-
N81035	G03 Color (loading)	Select a color	0×00600000	Immediate	-
N81045	Grid Color	Select a color	0×00800080	Immediate	3.12.4
N81046	Coordinate Color	Select a color	0×0000FF00	Immediate	3.12.4
N81049	WC Origin Color	Select a color	0×0000FFFF	Immediate	3.12.4
N81050	MC Origin Color	Select a color	0×0000FFFF	Immediate	3.12.4

7 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 maloperation. 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.