NK500 Integrated CNC System

Quick Start

1st Edition

Weihong Electronic Technology Co., Ltd.

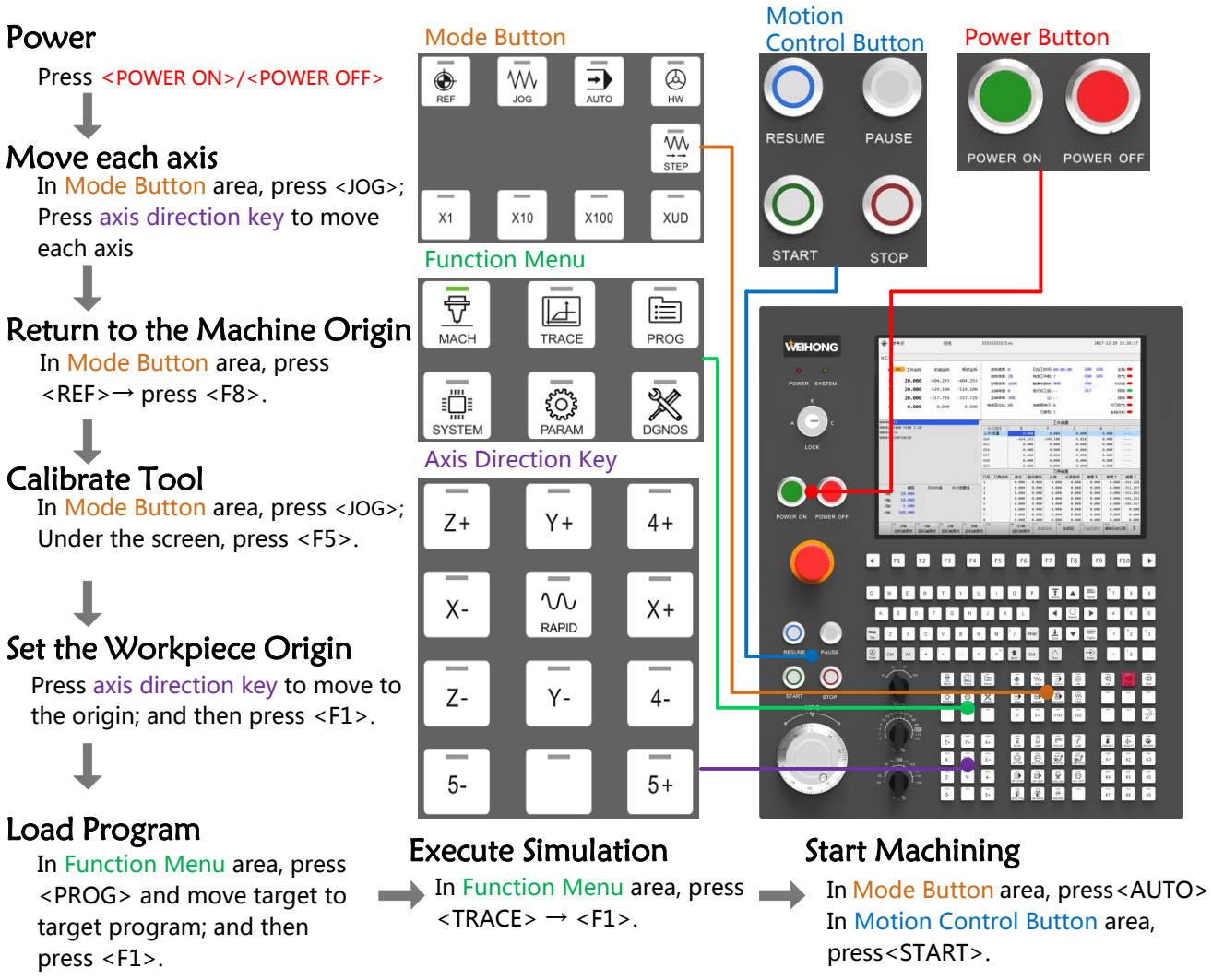
Preface

With this manual, you can quickly do the following:

1. According to common machining process, conducting machining. See section Common Machining Process for details.
2. Knowing about NK500 software. See Chapter 1 About NK500 for details.
3. Knowing about NK500 common operations. See Chapter 2 Common Operations for details.

Common Machining Process

This is the common machining process. Please refer to the following for details.



Contents

1 About NK500

This chapter mainly introduces the operating interface to help you know about NK500.

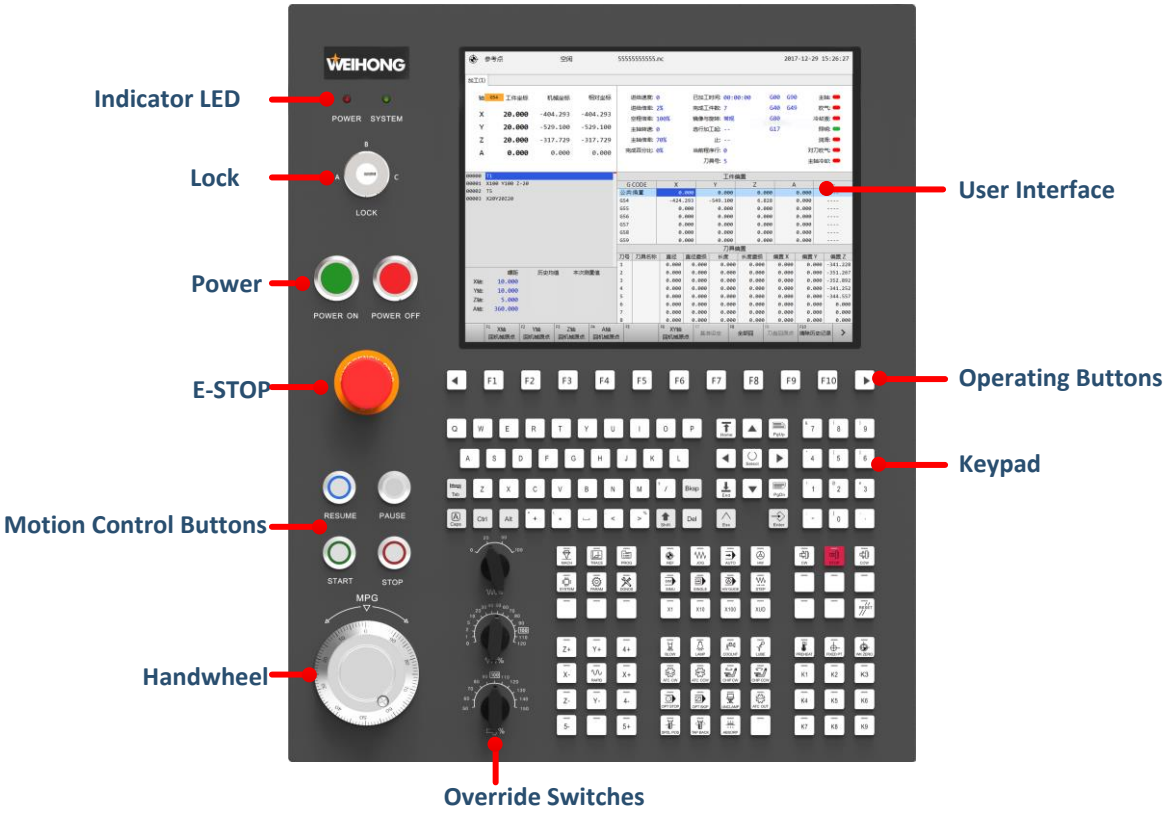


Fig. 1-1 NK500 Interface (1)

Indicator LED	• Red: indicator LED for power When power is supplied, LED is on; otherwise, it is off. • Green: indicator LED for running status ✓ If LED is off, reasons are as follows: ①The software is not started; ② The software crashes; ③ The software starts, but an connection exception occurs in the Lambda controller or the panel. ✓ If LED is on, then it indicates that the software starts and the controller and the panel are connected correctly. You can operate normally. ✓ If quick-flashing bright occurs, there is an exception in the firmware of NK500, which is unrecoverable. You must power off and restart.
Lock	This function is not supported for the time being.
Power	• Green switch: power on • Red switch: power off

E-stop	1) When the machine is in danger, press <E-stop> to stop the running of the machine tool. 2) After danger is removed, rotate the button clockwise to clear E-stop alarm.
Motion Control Buttons	• <PAUSE>: pause running the program. • <START>: start running the program. • <STOP>: stop running the program. • <RESUME>: resume running the program. In case of power interruption or emergency stop in machining, if the workpiece origin is secured, press the button to resume running the program from the exact interrupted position.
Override Switches	From top to bottom: G00 override; feed override; spindle override.
Keypad	Its usage is the same with the computer keyboard.
Operating Buttons	Buttons F1~F10 correspond to 10 software operations that are horizontally arranged under the user interface. : return to last page or menu, or close the dialog box. : enter into next page or menu.

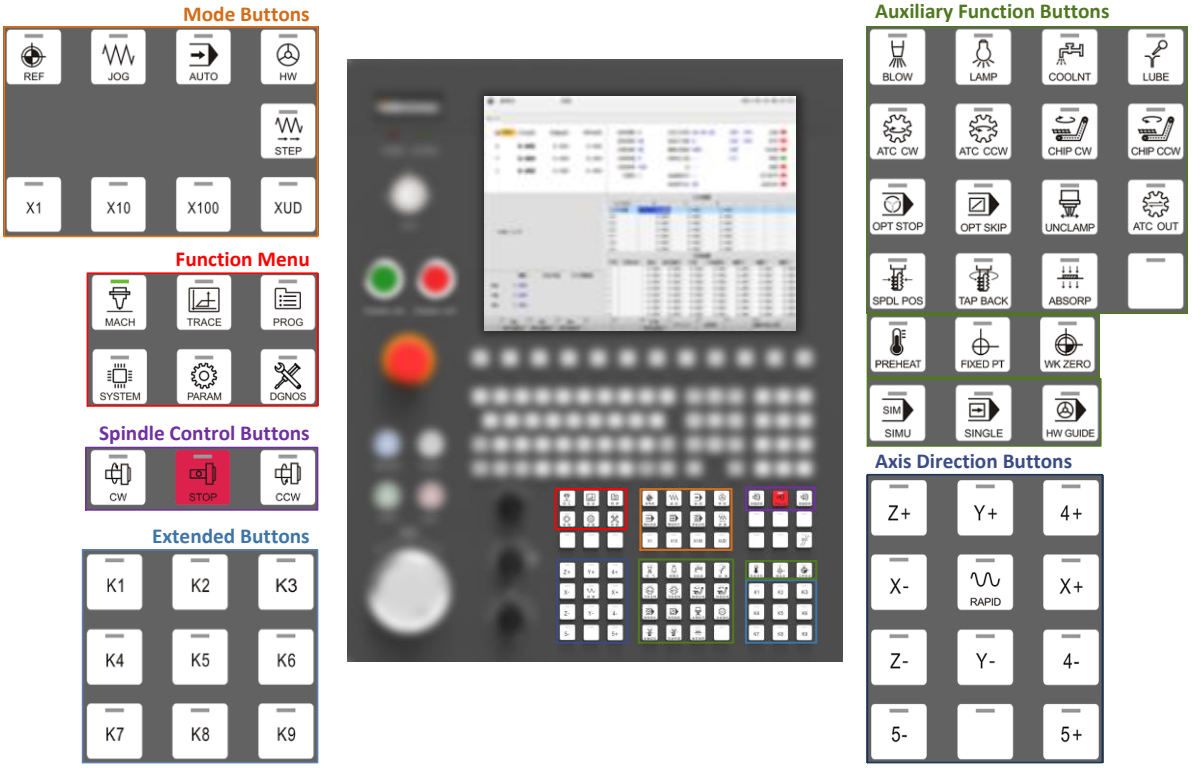


Fig. 1-2 NK500 Interface (2)

Mode Buttons	• It supports five modes: REF, JOG, AUTO, HW and STEP mode. • The default override for HW and STEP modes is X100. • Under STEP mode, X1, X10 and X100 correspond to 0.001, 0.01 and 0.1 (mm/inch) separately. XUD is for customizing the step length and you can set in “Machining” interface.
Function Menu	• MACH: it is for setting common parameters, executing common operations (like returning to the wokpiece origin, calibrating tool, returning to the fixed point, handwheel guide, executing single block, executing selective machining etc.), setting workpiece offset and public offset and managing tool. • TRACE: it is for simulating machining and checking machining information. • PROG: it is for loading, editing and deleting files from local/USB/internet. • SYSTEM: it is for registering, maintaining system, setting internet and checking software version, board card No. and internet information. • PARAM: it is for setting and checking all parameters. • DGNOS: it is for checking alarms and logs, managing ports and checking feedback pulse and coordinates.
Spindle Control Buttons	It is for controlling the rotation of spindle, including <CW>, <STOP> and <CCW>.
Extended Buttons	Buttons K1~K9 are used for customization.
Auxiliary Function Buttons	It can mainly divided into two categories: • Quickly executing some machining operations. For example, press <WK ZERO> to return to the workpiece origin; press <SIMU> to execute simulation. • Quickly opening/closing common ports. For example, after pressing <LUBE>, the indicator LED and lube will be on.
Axis Direction Buttons	• Under AUTO mode, press some axis direction button and <RAPID>, the machine tool moves at high jog feedrate. • Press some axis direction button alone, the machine tool moves at jog feedrate.

2 Common Operations

2.1 Returning to the Machine Origin

Machine origin is the origin of the machine coordinate system. Before machine tools leave factory, such an origin is determined after designing and commissioning. It is a fixed point.

After the controller system starts, returning to the machine origin either automatically or manually is required. After it is done, a related sign will appear before each axis name, then, the machine can continue to machining.

Here take “Combined Software---General 3-axis Configuration” as an example. Steps are as shown in Fig. 2-1

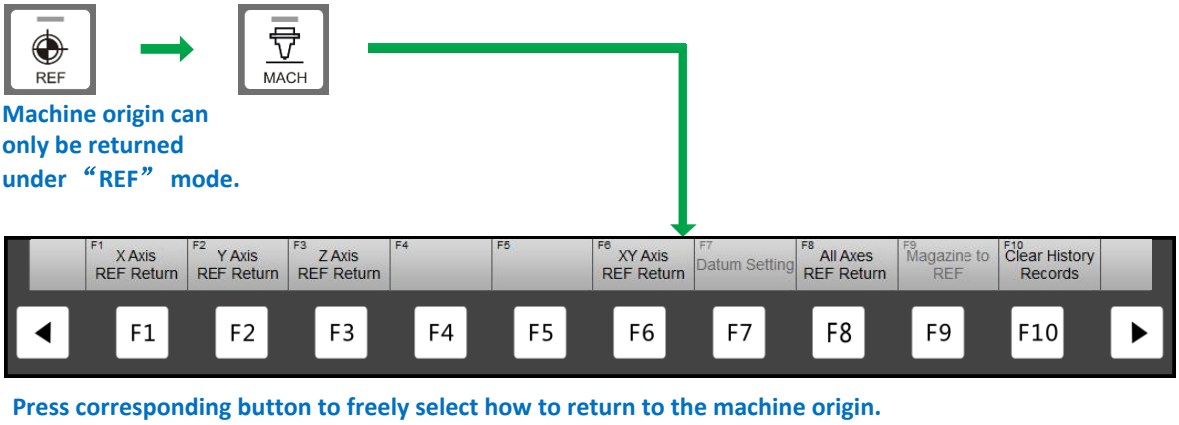


Fig. 2-1 Steps for returning to the machine origin

By default, Z-axis returning to the machine origin has the priority. If other axis returns to the machine origin before Z-axis, a dialog box for confirmation will pop up. You can select “No” to quit from the current operation or select “Yes” to continue.

2.2 Changing Tool

For the machine tools equipped with tool magazines, a tool magazine contains several tools. And each tool differs in usage; you can switch the tool according to your needs. Steps are as shown in Fig. 2-2:

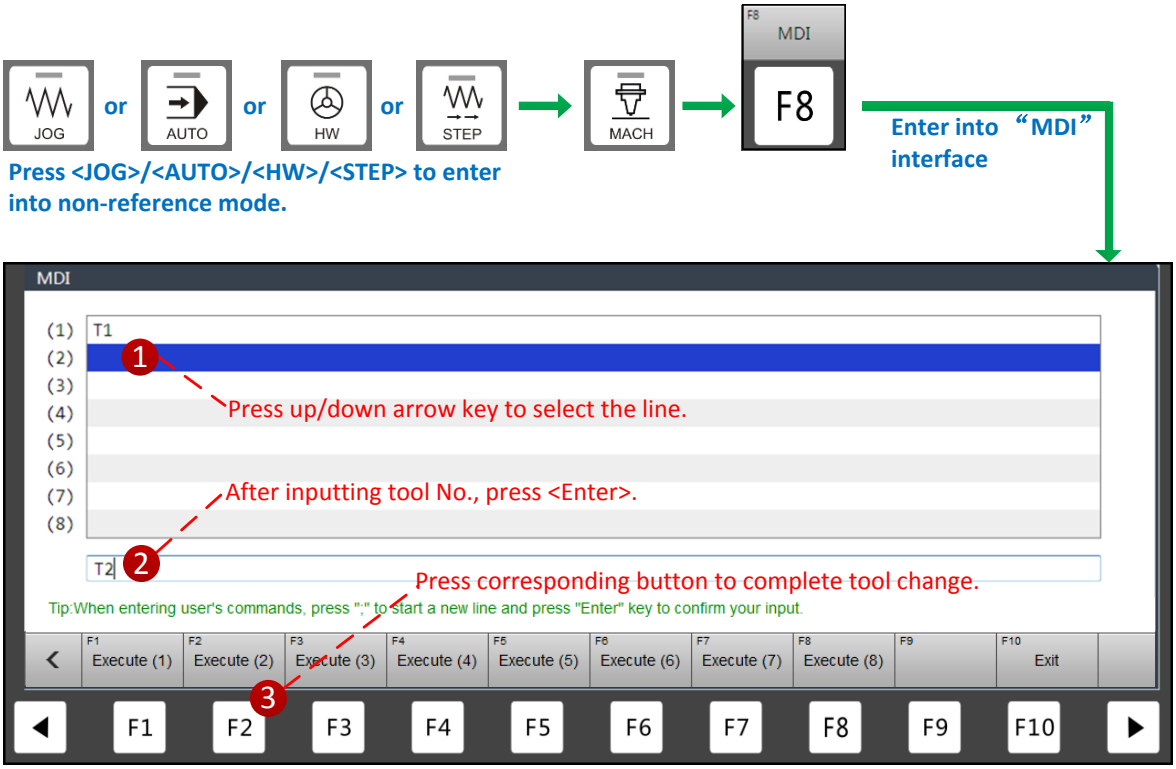


Fig. 2-2 Steps for tool change

2.3 Calibrating Tool

During actual machining, the tool length and tool holder position will change after tool change due to tool breakage or other reasons. On this occasion, you can calibrate tool to re-confirm the tool length offset.

The system supports three modes of calibration: fixed calibration, mobile calibration and first and exchange calibration. The default mode is the first one. This section only introduces the default mode---fixed calibration. Steps are as shown in Fig. 2-3:

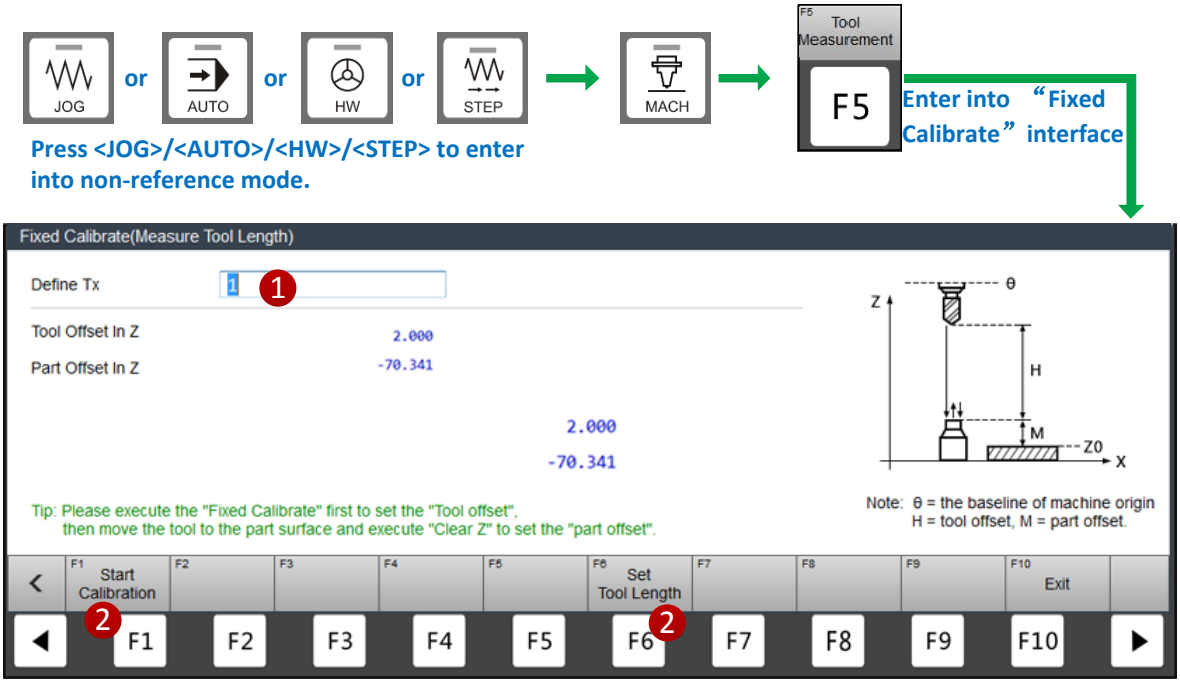


Fig. 2-3 Steps for fixed calibration

①	Press <Enter> to select a tool according to tool No.
②	• With tool sensor, press <F1> to automatically calibrate the selected tool. • Without tool sensor, press <F6> to manually set tool offset Z.

Repeat steps ① and ② for each tool.

After setting the tool offset through fixed calibration, select any tool and move its nose to the workpiece surface to clear Z-axis.

2.4 Setting the Workpiece Origin

You can set the workpiece origin through the following:

- Clearing
- Centering (including line centering and circle centering)

2.4.1 Clearing

When the demand for accuracy is not strict enough, and the shape of workpiece is irregular, you can manually set the workpiece origin by clearing. Steps are as shown in Fig. 2-4.

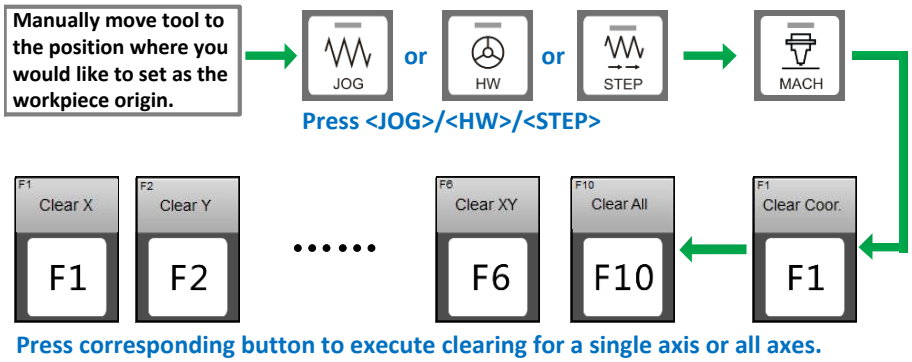


Fig. 2-4 Steps for setting the workpiece origin by clearing

2.4.2 Line Centering

Line centering is used to find the center point of a regular rectangle by two points and set it as the workpiece origin. Here take X-axis as an example to introduce steps for line centering. Steps as shown in Fig. 2-5.

(Note: during centering on some axis, the other axes should remain static.)

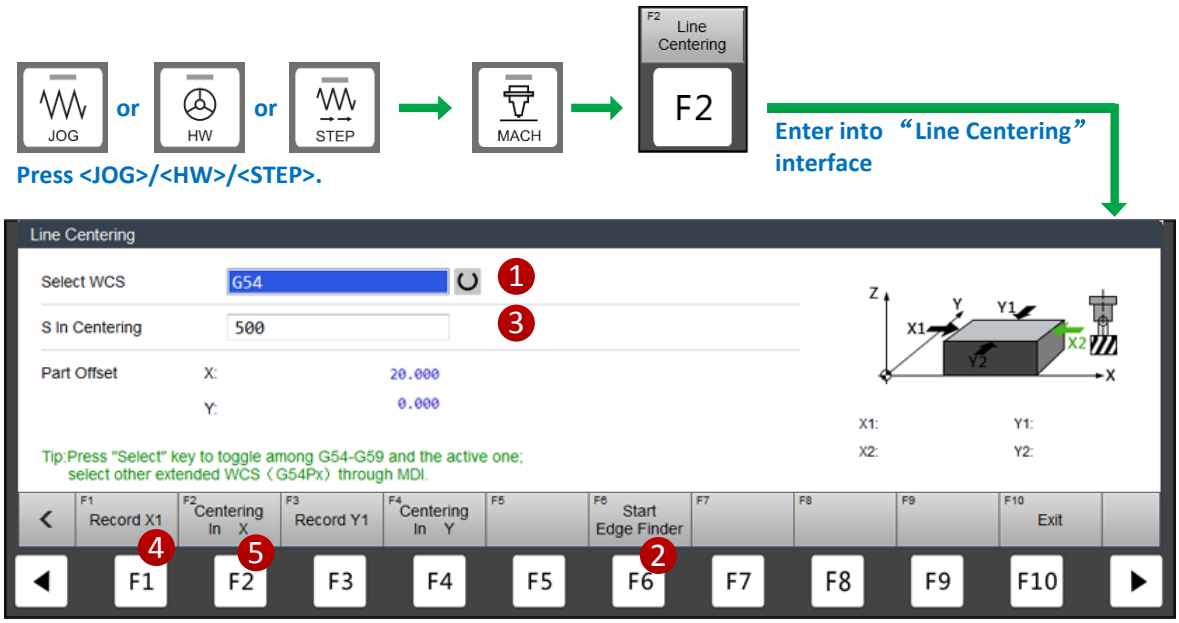


Fig. 2-5 Steps for line centering

1	Press <Select> to select the coordinate system from G54 to G59.
2	Optional: press <F6> to turn on edge finder. • During centering, the edge finder can help position precisely. At this time, the set speed in step ③ is valid. • With edge finder disabled, the set speed in step ③ is invalid. Press <CW>/<CCW> to turn on spindle. And its speed is the set value in the software or program file.
3	Press up/down arrow key to move to "S in Centering" and set its value (suggestion: the set value should not be too large.). The set value is the value of parameter [N20006 Spindle Speed in Centering].
4	Manually move the tool to one side of the workpiece, and press <F1>. The software records the machine coordinates of the current position.
5	Manually move the tool to the other side of the workpiece, and press <F2>. The software calculates the middle point and sets it as the workpiece origin.

2.4.3 Circle Centering

Circle centering is used to find the center point by three points on the workpiece. It is applicable to circular workpiece. Steps are as shown in Fig. 2-6.

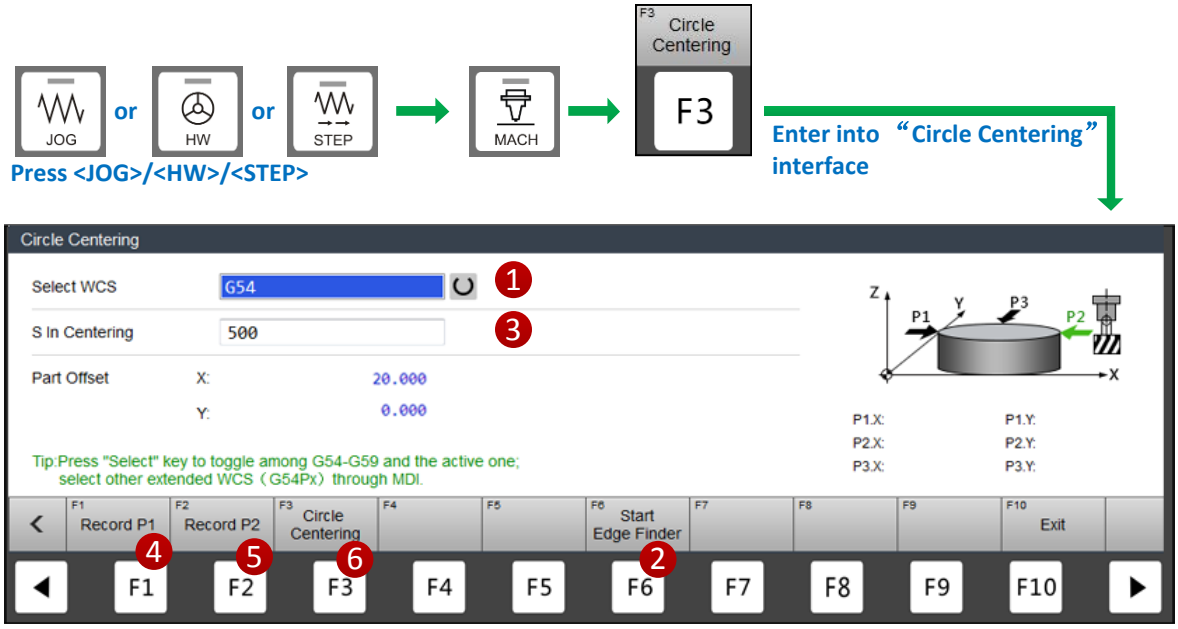


Fig. 2-6 Steps for circle centering

1	Press <Select> to select the coordinate system from G54 to G59.
2	Optional: press <F6> to turn on edge finder. • During centering, the edge finder can help position precisely. At this time, the set speed in step ③ is valid. • With edge finder disabled, the set speed in step ③ is invalid. Press <CW>/<CCW> to turn on spindle. And its speed is the set value in software or program file.
3	Press up/down arrow key to move to "S in Centering" and set its value (suggestion: the set value should not be too large.). The set value is the value of parameter [N20006 Spindle Speed in Centering].
4	Move the tool to one point on the circle (P1) → Press <F1>.
5	Move the tool to another point on the circle (P1) → Press <F2>.
6	Move the tool to the third point (P3) → Press <F3>. The system will calculate the center of a circle based on the two groups of recorded coordinates and the current machine coordinate and set it as the workpiece origin.

2.5 Returning to the Workpiece Origin

When the machine tool returns to the workpiece origin, Z-axis returns to the safety height first, and then X-axis and Y-axis return to the origin. The safety height for Z-axis is to avoid potential crash onto the workpiece surface during returning. Steps are as shown in Fig. 2-7.

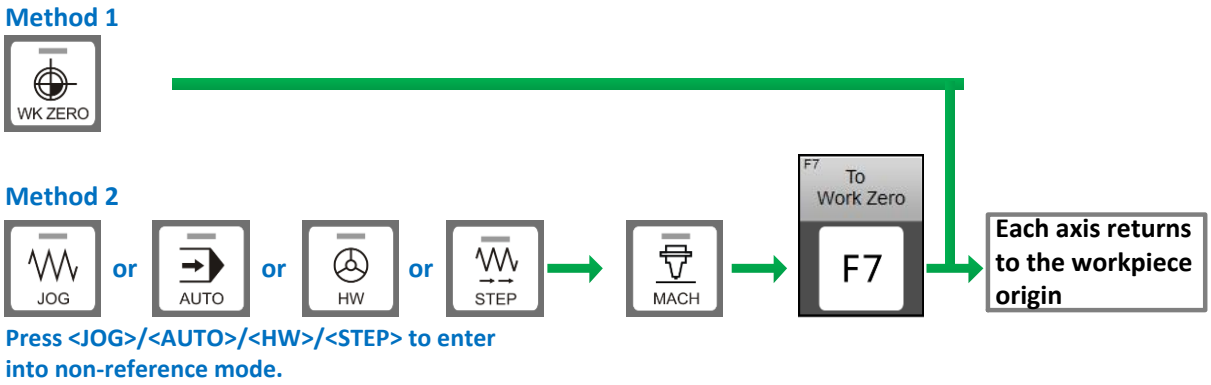


Fig. 2-7 Steps for returning to the workpiece origin

2.6 Loading Machining Program

You can load machining program from the following paths:

- Local: program files stored in the NK500 host.
- USB: program files stored in a USB flash disk.
- Network: program files stored in the local area network.
- Wizard: simple program file set in “Wizard” interface.
- History: in “History” interface, you can quickly load program files that have been loaded before.

Take loading files from the local as an example. Steps are as shown in Fig. 2-8.

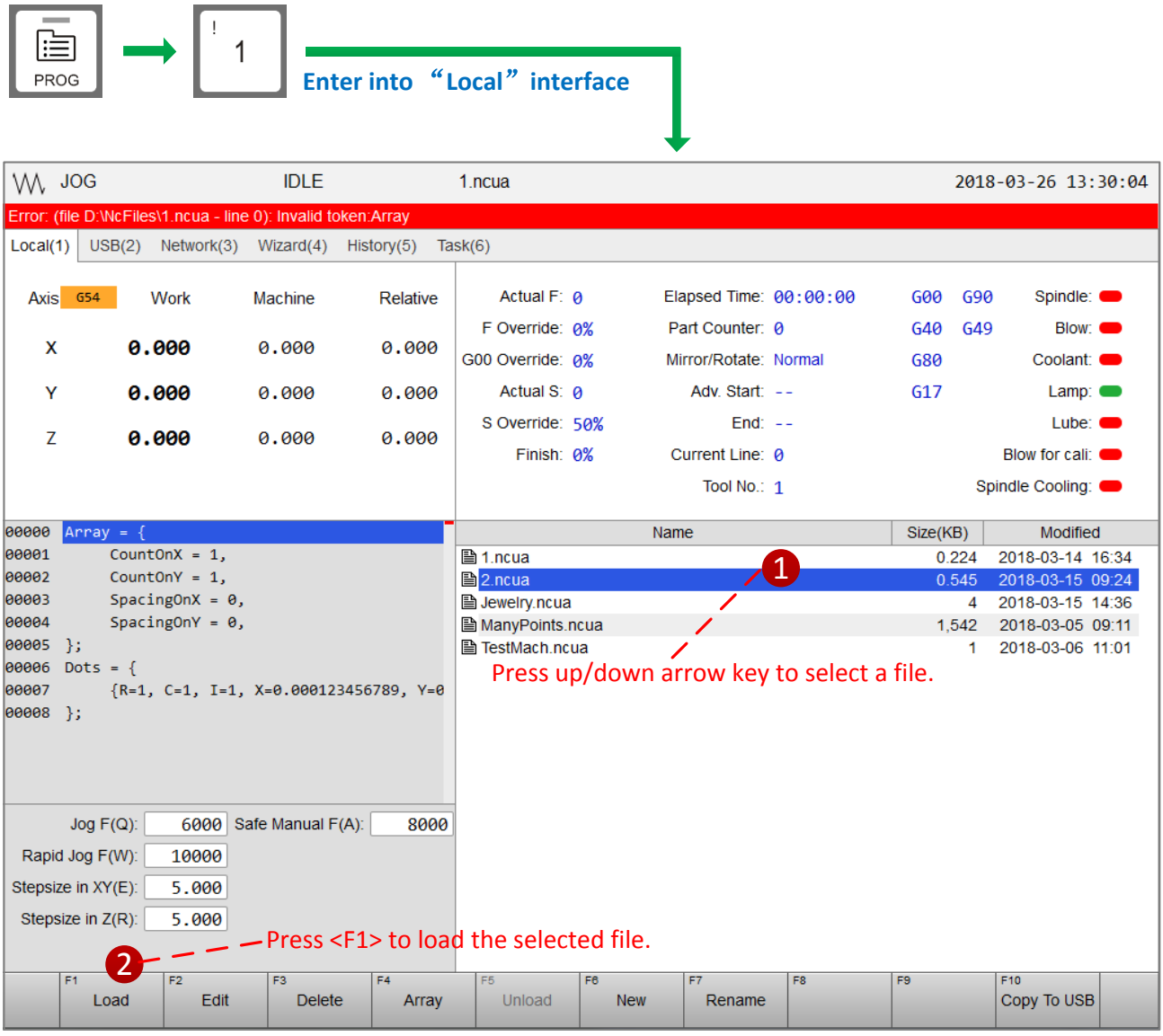


Fig. 2-8 Steps for loading file from the local

2.7 Quickly Setting Common Machining Parameters

In the main interface, you can set common machining parameters (e.g. spindle speed, feedrate, G00 speed and cycling repeats and etc.). Steps for setting related parameters are as shown in Fig. 2-9.

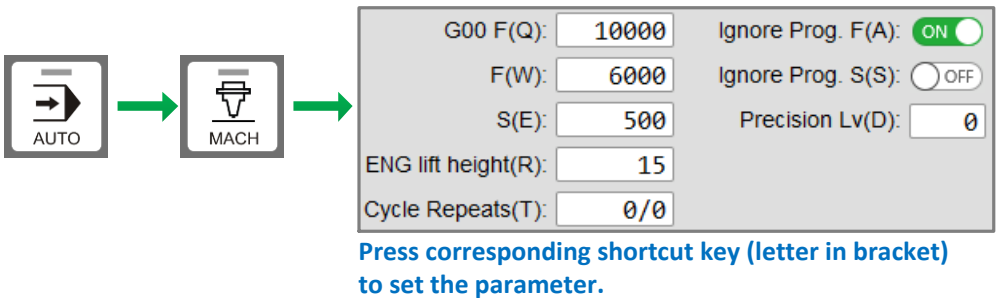
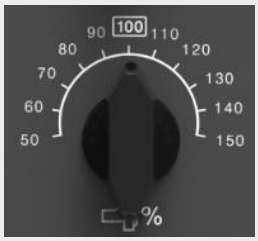
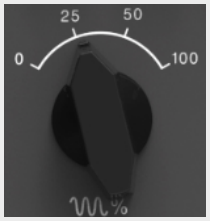


Fig. 2-9 Steps for setting related parameters

Spindle Speed	• Press <S> to enable (ON) or disable (OFF) the function of "Ignore Prog. S". ON: during auto machining, the spindle speed is the set value in the interface. OFF: during auto machining, the spindle speed is the value specified by program. Related parameter: [N72002 Ignore Prog. S] • Press <E> to set the spindle speed in the popup. • The spindle speed can be adjusted by the spindle override. Current spindle speed = Spindle speed * Spindle override	 Fig. 2-10 Spindle override switch
Feedrate	• Press <A> to enable (ON) or disable (OFF) the function of "Ignore Prog. F". ON: during machining, the feedrate is the set value in the interface. OFF: during machining, the feedrate is the value specified by program. Related parameter: [N72001 Ignore Prog. F] • Press <W> to set the feedrate in the popup. • The feedrate can be adjusted by the feed override.	 Fig. 2-11 Feed override switch

	Current feedrate = Feedrate * Feed override	
G00 Speed	• Press <Q> to set G00 speed in the popup. • When parameter [G00 F Fixed] is set to Yes, G00 speed is controlled by G00 override. When G00 speed is fixed and G00 override is 0, the actual G00 speed is controlled by parameter [N72005 Actual Override at Zero Traverse Override] and its range is [0,5]. Relation: Current G00 speed= G00 speed * G00 override • When parameter [N72003G00 F Fixed] is set to No, G00 speed is controlled by the feedrate.	 Fig. 2-12 G00 override switch
Cycling Repeats	Press <T> to set cycling repeats in the popup.	

2.8 Simulation

When simulation is executed, the system will not drive the machine tool to do actual actions but only show the machining trace of tool in the interface. Through this operation, you can see the motion of machine tool in advance as so to avoid the damage of machine tool as result of programming mistakes.

Steps for executing simulation are as shown in Fig. 2-13.

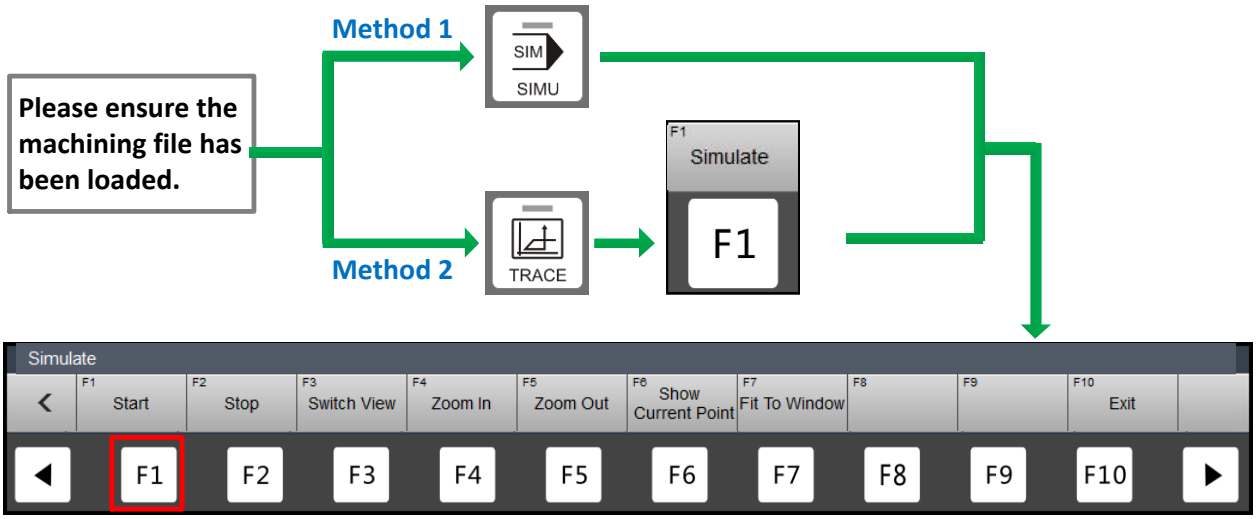


Fig. 2-13 Steps for executing simulation

2.9 Editing Program

This operation is to edit the currently loaded program. Steps are as shown in Fig. 2-14.

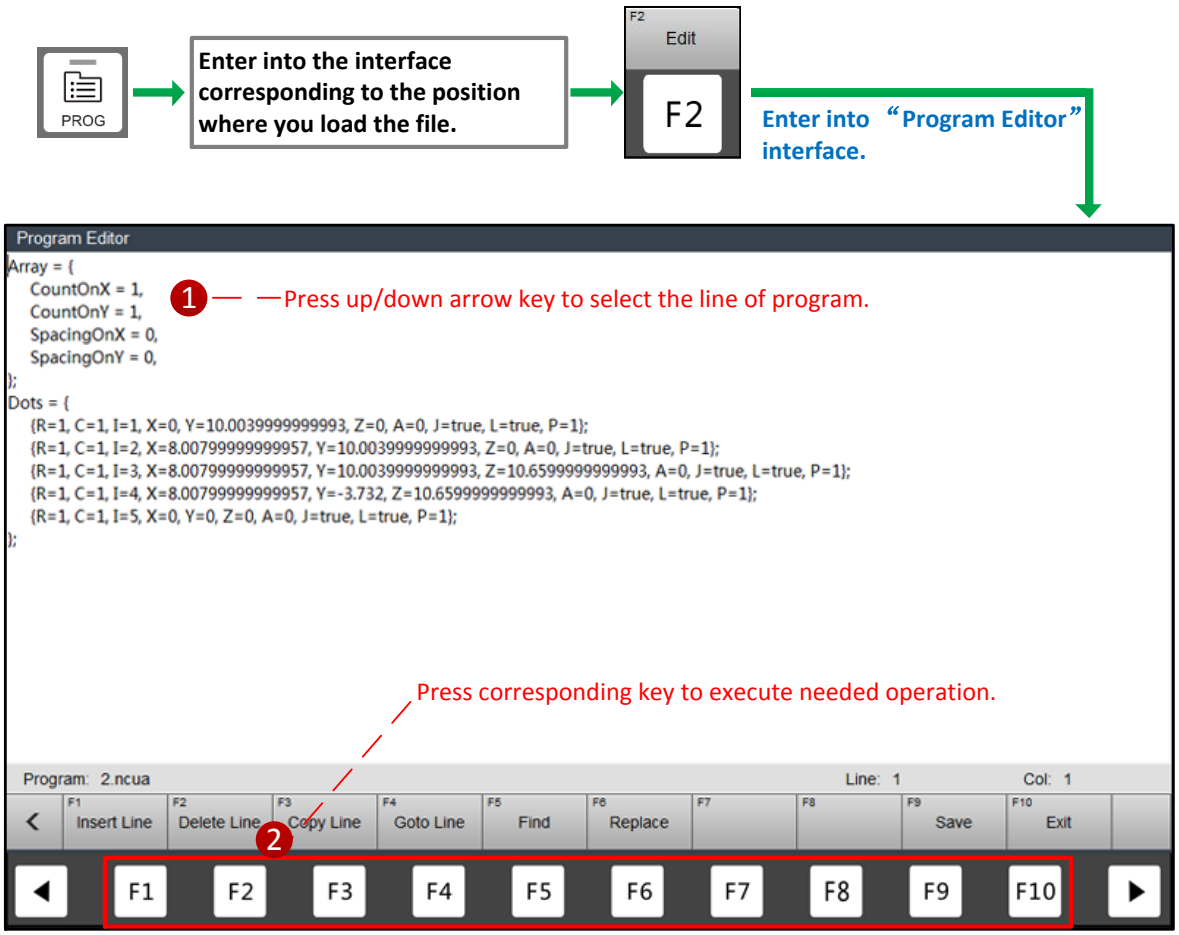


Fig. 2-14 Steps for editing program

2.10 Selecting Tool

When a machining file contains several tools, you can select the tool you need to machine. Such tool selection function is available when parameters [N62022 Enable Tool Selection for G Code File] and [N65203 Enable Tool Selection for ENG File] are set to “Yes”.

When the program file you loaded contains several tools, a dialog box of tool selection will pop up automatically. Steps for selecting a tool are as shown in Fig. 2-15 . Steps for selecting several tools are as shown in CAUTION part.

Fig. 2-15 Steps for selecting a tool

CAUTION

Steps for selecting several tools are as follows:

Select the tool No. → Press <F2> → Select the tool No. → Press <F2>……(till you have selected all the needed tools) →Press <F9> → System machines the machining area corresponding to the selected tools.

2.11 Setting the Workpiece Offset

Workpiece offset is the offset corresponding to the machine origin. Its setting steps are as shown in Fig. 2-16.

Please note that the positive and negative values of workpiece offset will influence the position of the workpiece origin.

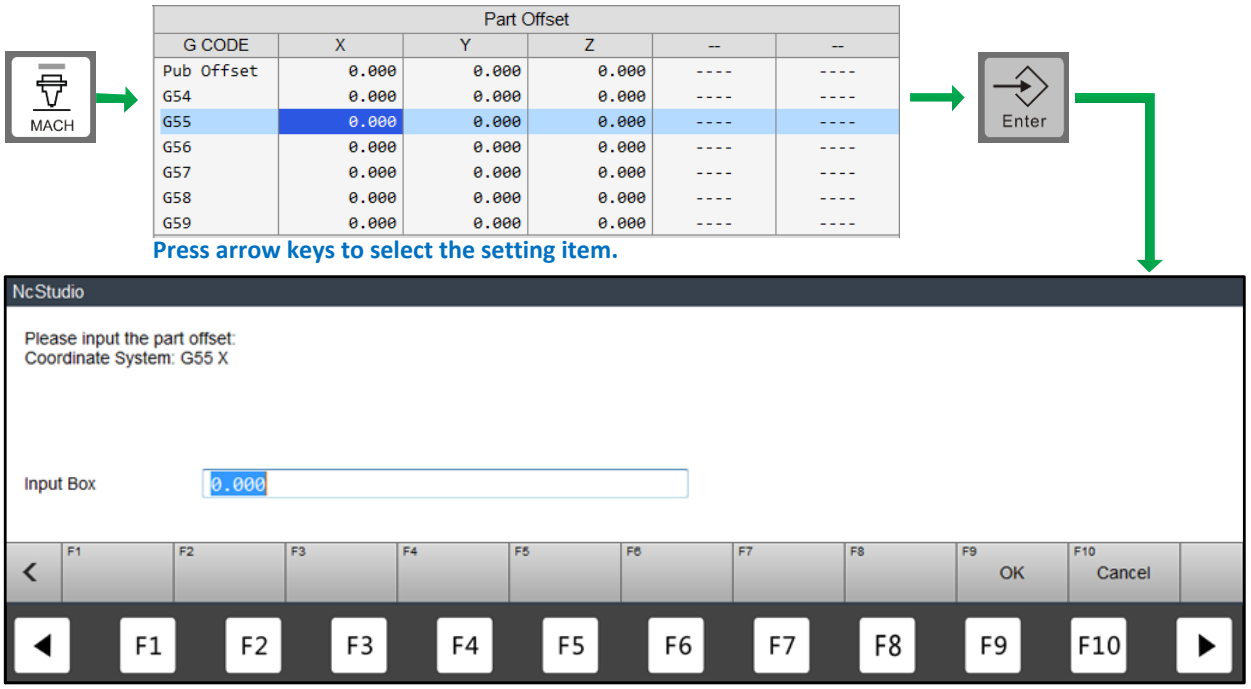


Fig. 2-16 Steps for setting the workpiece offset---modify directly

2.12 Setting Tool Compensation

Due to tool wear, tool change or other causes, the radius of tool nose will change. Therefore, you need to set tool compensation in “Tool Offset” area. Steps are as shown in Fig. 2-17.

Please note that parameter [N62410 Enable Tool Compensation] should be set to “Yes” before setting tool compensation.

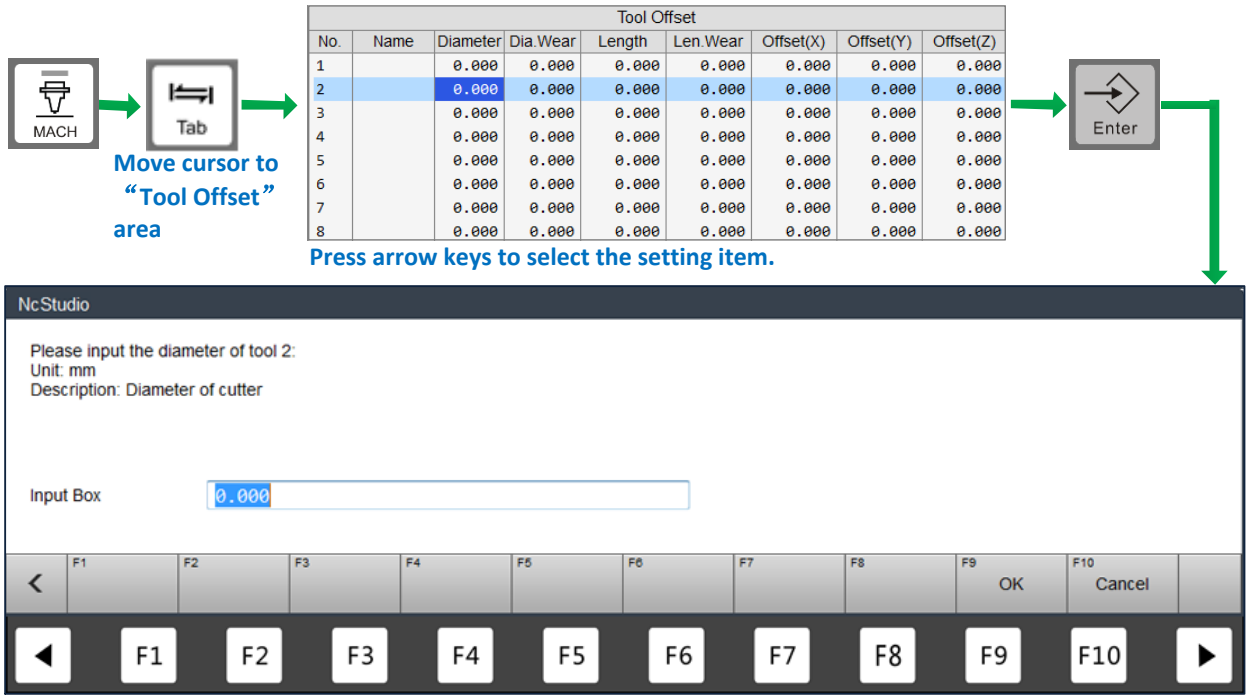


Fig. 2-17 Steps for setting tool compensation

2.13 Updating/Packing up the Software

You can update the software and common file and pack up the software through this function. Steps are as shown in Fig. 2-18.

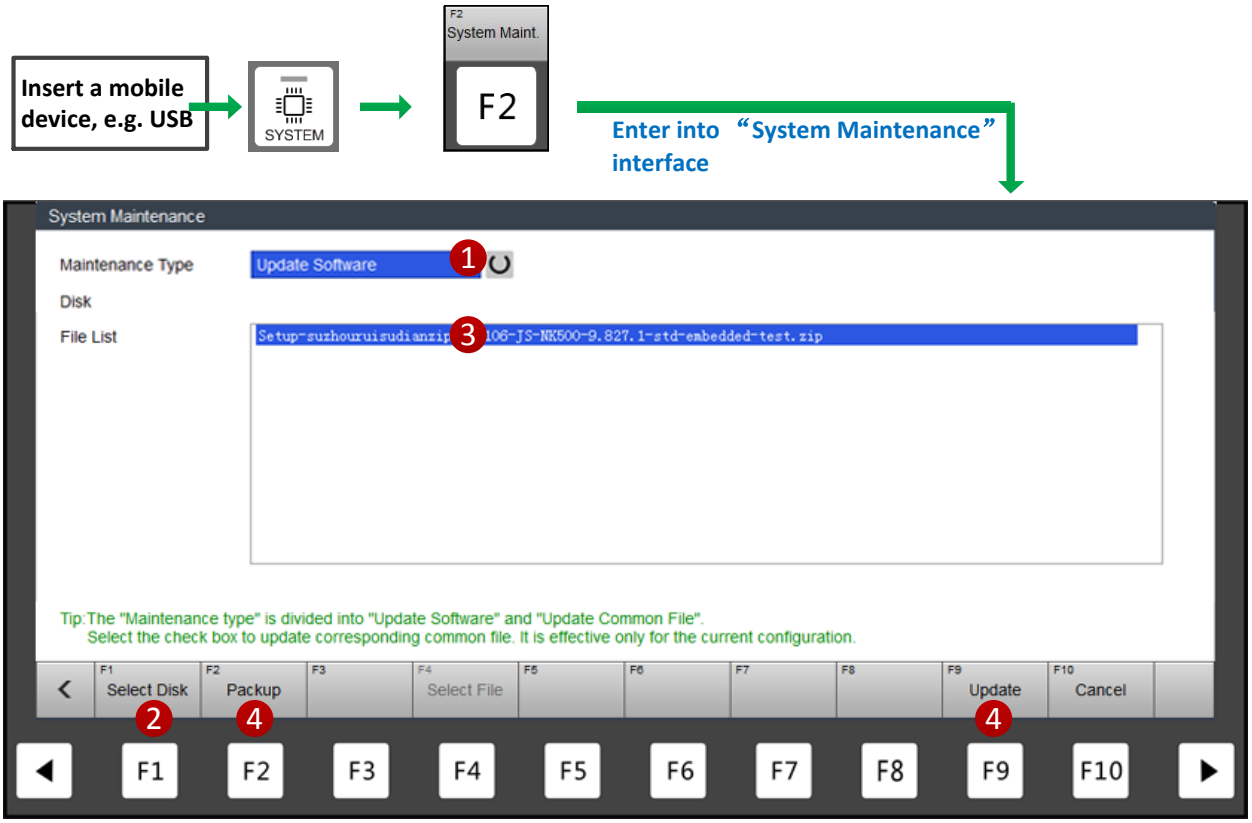


Fig. 2-18 Steps for update/pack up software

1	Press <Select> to select "Update Software".
2	• With several USB devices, please press <F1> to select the file path. • With only one USB device, ignore this step.
3	Press up/down arrow key to select a file.
4	Press <F2> to pack up the software. Press <F9> to update the software.

2.14 Registering

This function is used to limit the service time. Steps are as shown in Fig. 2-19.

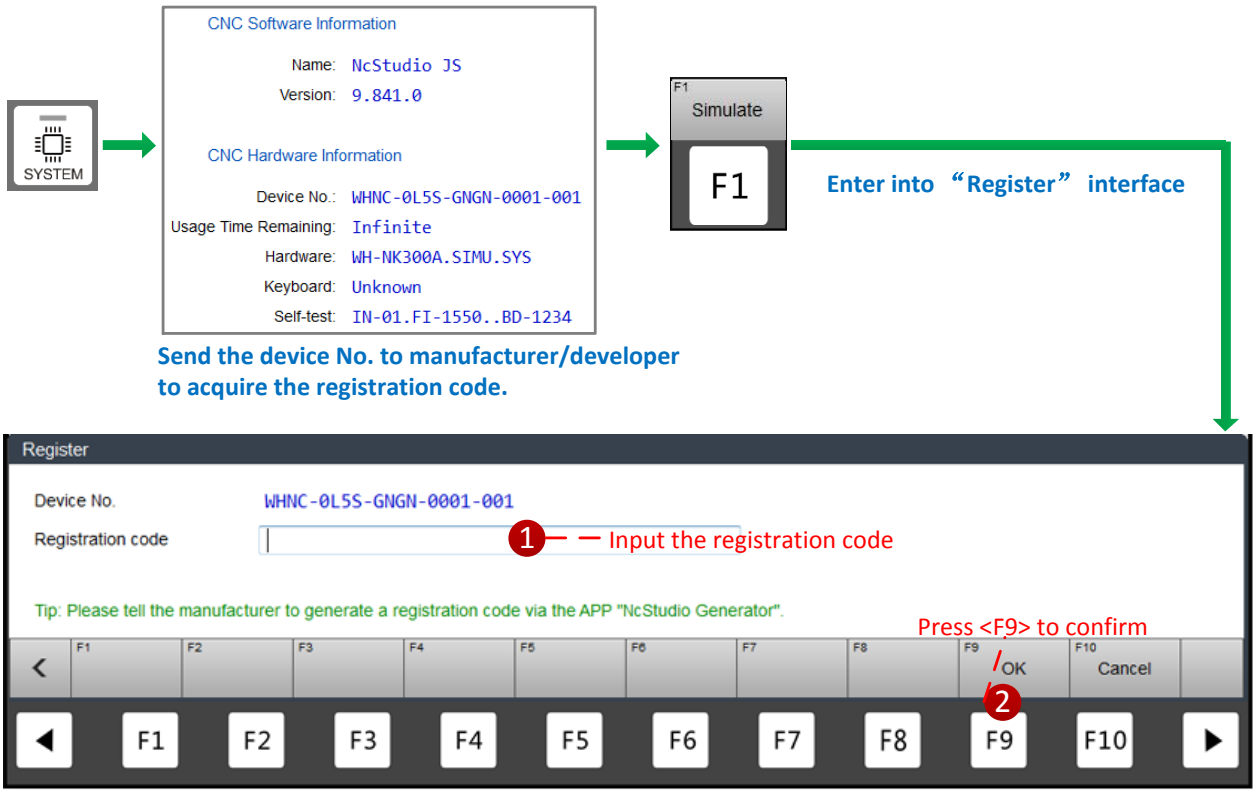


Fig. 2-19 Steps for registration

2.15 Exporting Data

Through this function, you can import/export data of datum, screw error compensation and servo parameters.

2.15.1 Exporting Data of Three Types

Steps are as shown in Fig. 2-20.

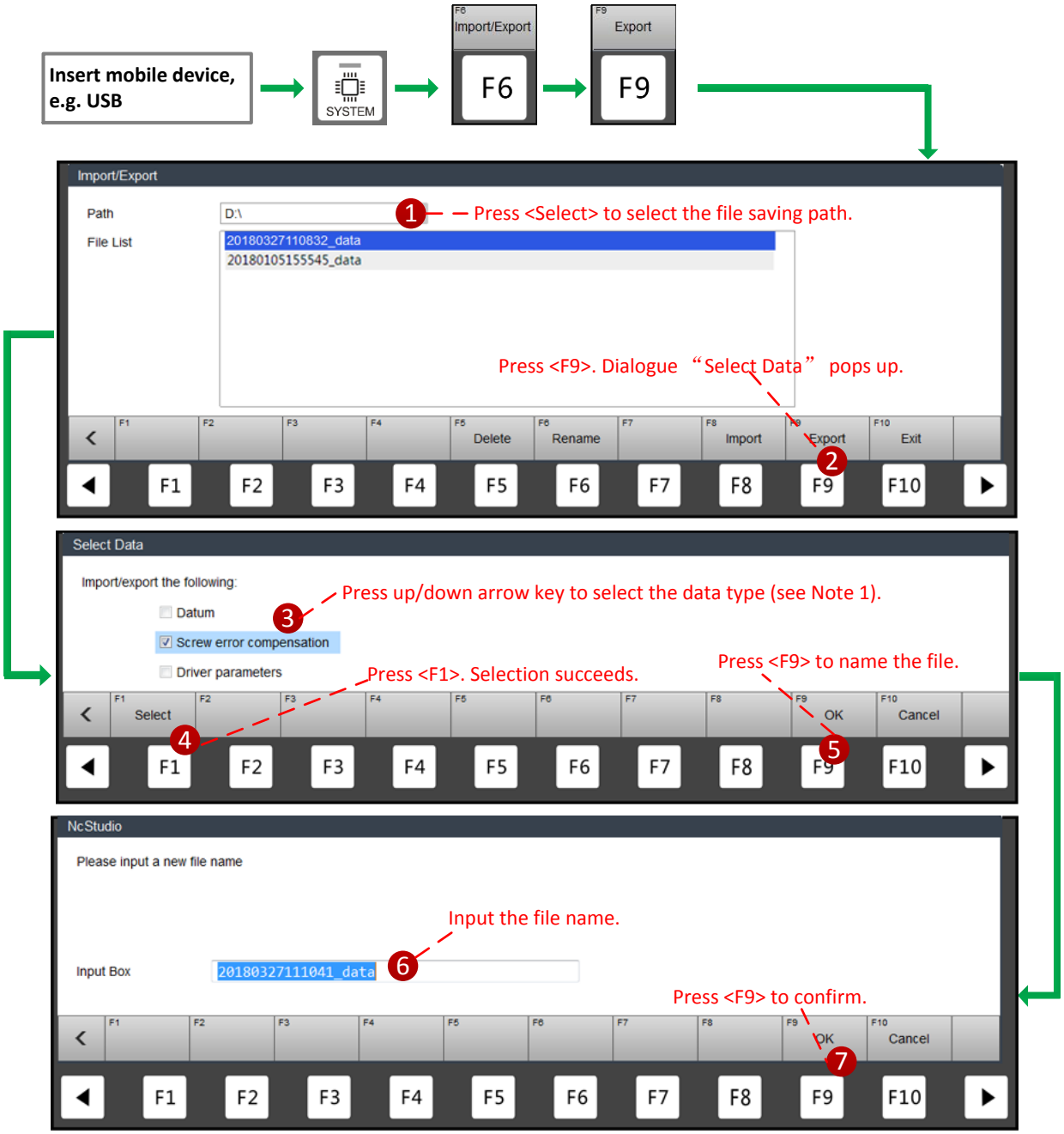


Fig. 2-20 Steps for exporting data

Note 1:

- “Datum” data is from the datum set by absolute encoder. You can select it only when the absolute encoder is used.
- “Screw error compensation” data is from backlash and pitch error compensation. You can also export data in “Screw Err Comp” interface. See section 2.15.3 for detail.
- “Servo parameters” data can be selected only when the value of parameter [N50000 Control System Type] is set to 1. You can also export data in “Servo Parameters” interface. See section 2.15.2 for detail.

2.15.2 Exporting Servo Parameters

When the value of parameter [N50000 Control System Type] is set to 1 (Bus Control System), you can modify/ import/ export servo parameters in NK500 panel directly.

Exporting servo parameters is to save the set parameters in “Servo Parameter” interface to mobile devices, e.g. USB, for later use. Steps are as shown in Fig. 2-21.

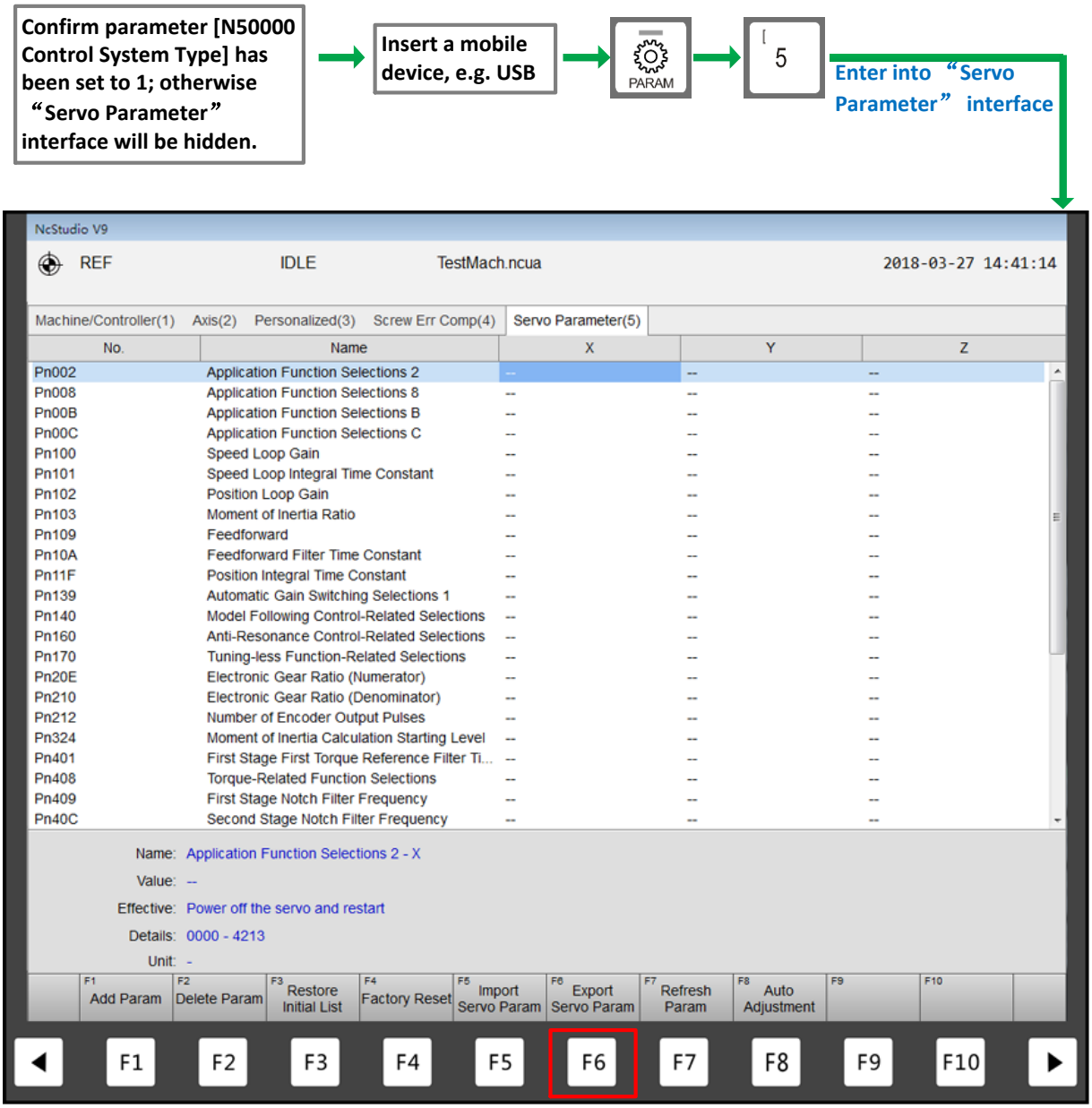


Fig. 2-21 Steps for exporting servo parameters

2.15.3 Exporting Screw Error Compensation

When high machining accuracy is required, backlash compensation is necessary; when higher machining accuracy is required, both backlash compensation and screw error compensation are necessary. In fact, the system has combined these two compensations. Through putting error data into error compensation file, the system will compensate automatically during running.

You can export current compensation data into the USB device for direct use in following machining. Steps are as shown in Fig. 2-22.

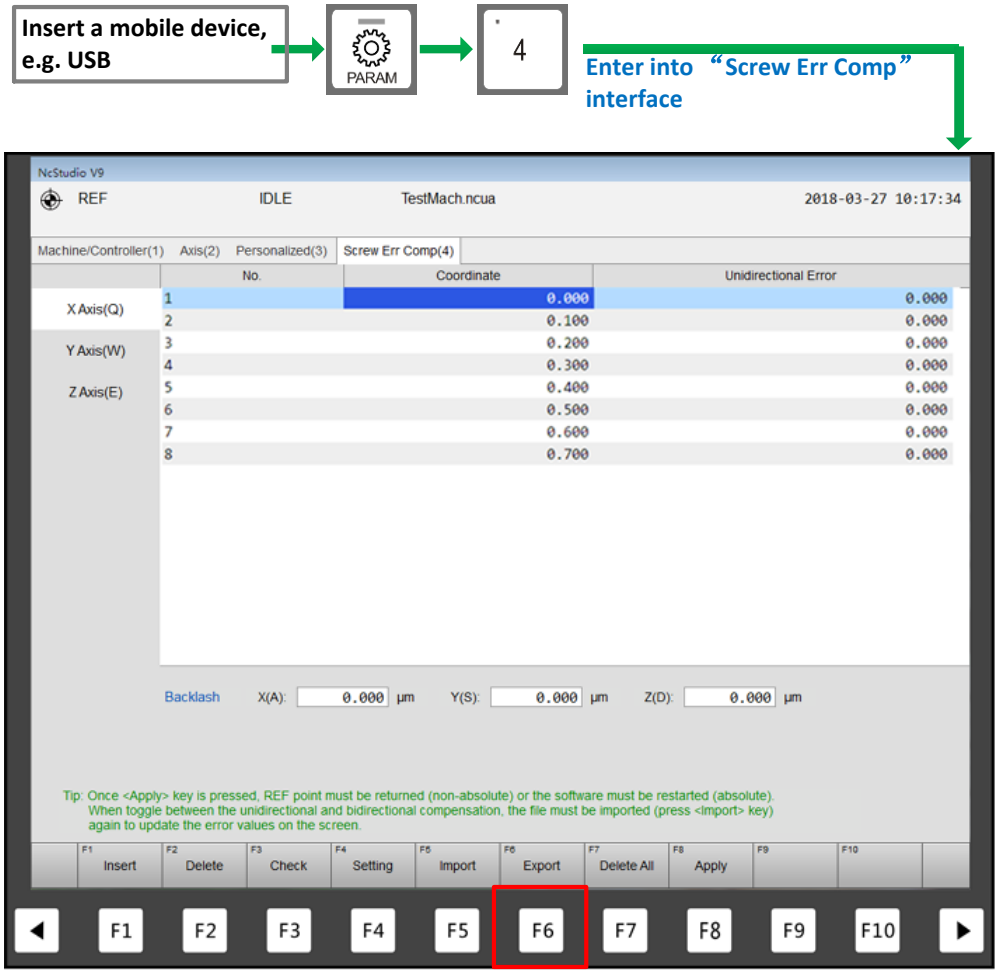


Fig. 2-22 Steps for exporting data of screw error compensation

2.16 Checking Alarms

This function is for checking system alarms so as to find solutions. Steps are as shown in Fig. 2-23.

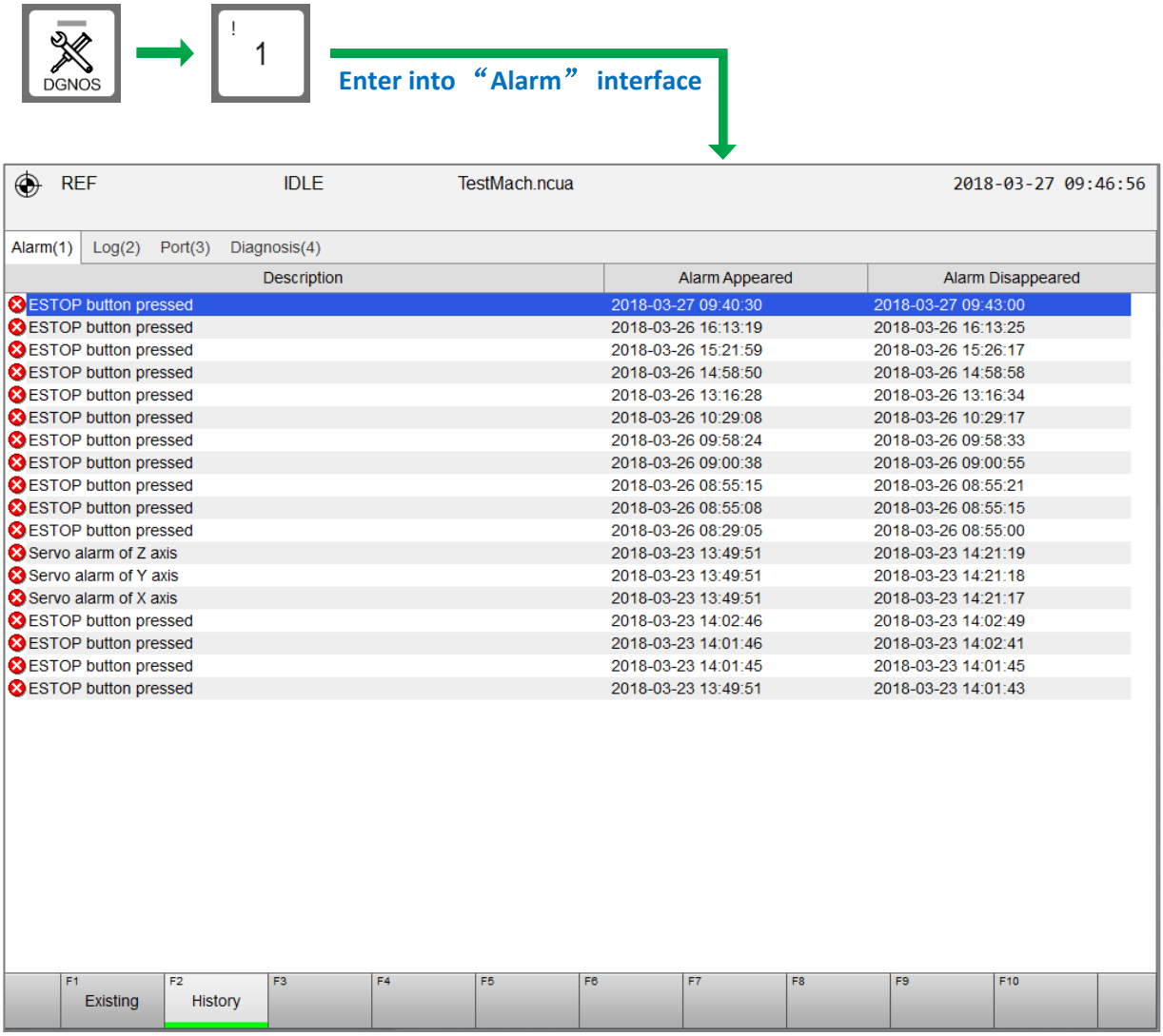


Fig. 2-23 Steps for checking alarms

2.17 Checking Logs

"Log" interface records your operations and the system events. Steps are as shown in Fig. 2-24.

In this interface, you can conduct operations by pressing corresponding buttons.

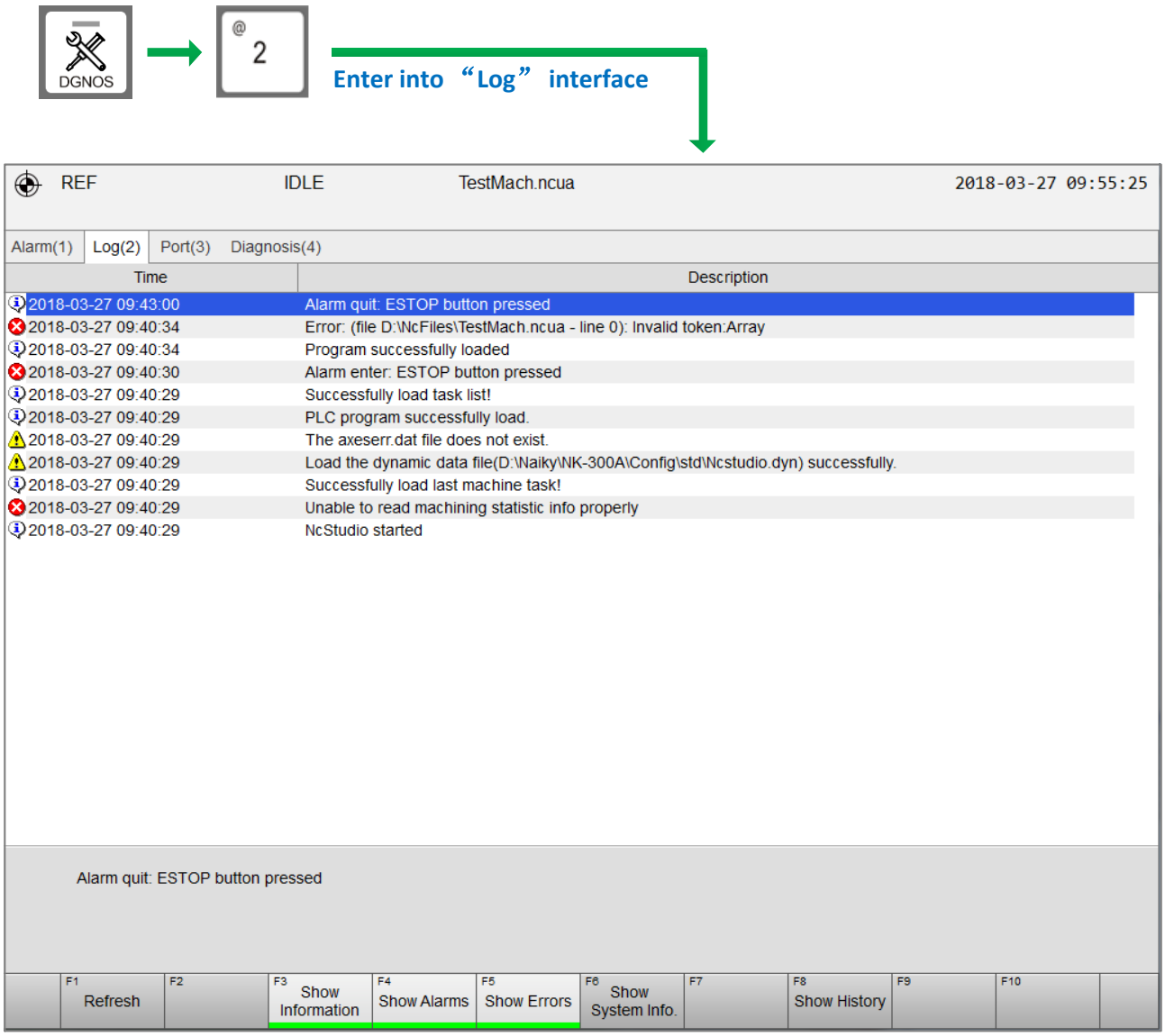


Fig. 2-24 Steps for checking logs

Revision History

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

Date	Edition	Revision
2018.03	R1	This edition is released for the first time.

Contact Us

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

Company Name: Weihong Electronic Technology Co., Ltd.

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

Tel: +86-21-33587550

Fax: +86-21-33587519

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