



维宏股份 创业板: 300508

NK300HF Integrated CNC System

Operation Manual

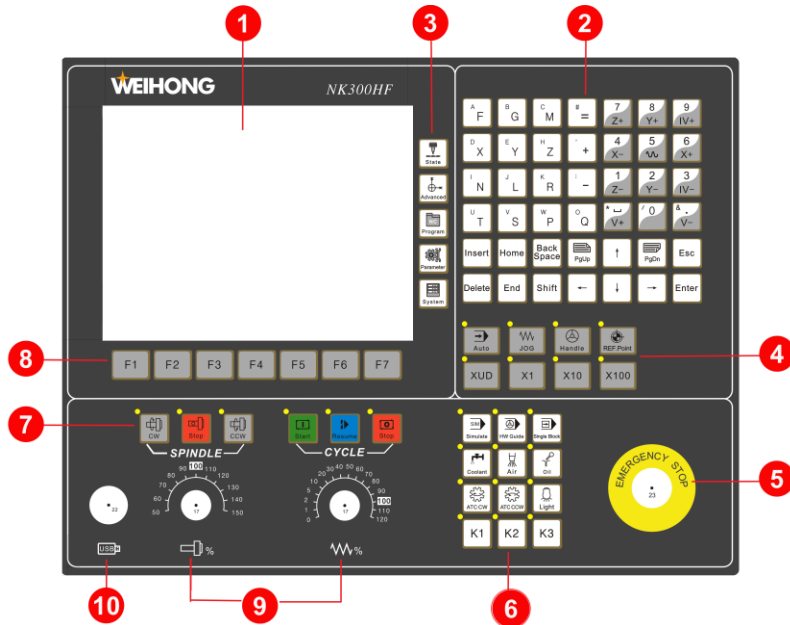
Table of Contents

Operational Panel	1
Led Display.....	2
Alpha-numeric Keypad	3
Main Function Access Buttons	4
Mode Buttons.....	5
E-stop Button	5
Buttons for Auxiliary Function.....	6
Buttons for Movement Control	7
Operational Buttons	7
Override Switches	7
Quick Start	8
Loading Program File	8
Loading Local File.....	9
Loading USB File	10
Loading Wizard File	11
Setting the Workpiece Origin	12
Clearing.....	12
Calibrating Tool.....	13
Centering.....	13
Selecting Machining Mode	16
Array Machining.....	16
Cycle Machining.....	17
Program Task.....	18
Running Program	19
Auto Run.....	19
Single Block.....	20

Selective Machining	20
Handwheel Guide.....	20
Executing Tool Lifting and Dropping.....	21
Checking Part Statistics.....	23
FAQs	24
What to do if the spindle does not rotate?.....	24
What to do if an axis does not move?	25
What to do if servo motor Z brake can't be opened?.....	25
What to do if machine tool returns to the machine origin abnormally?.....	26
What to do if the machine tool motions upward after arriving at the position of tool sensor during calibration?	27

Operational Panel

The operational panel of *NK300HF Integrated CNC System* is as follows::

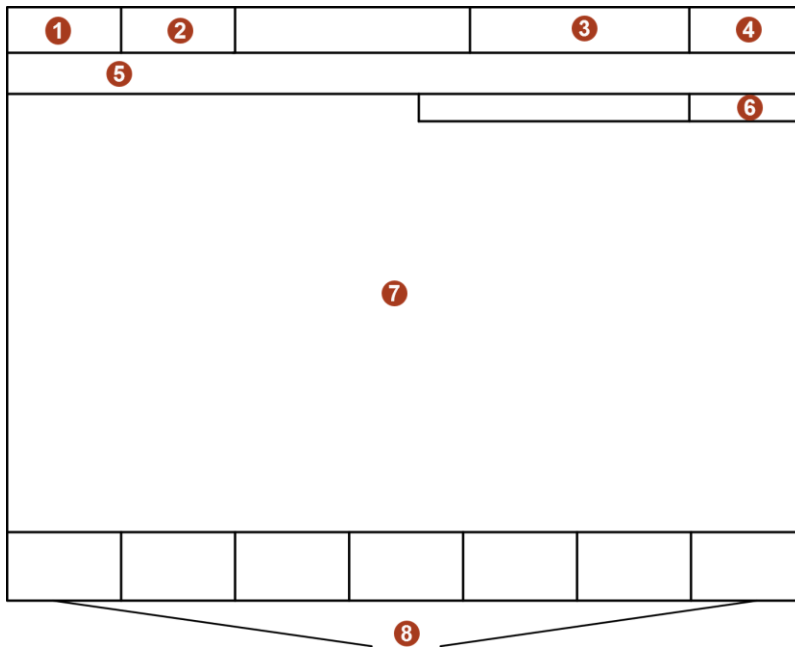


1. Led Display
2. Alpha-numeric Keypad
3. Main Function Access Buttons
4. Mode Buttons
5. E-stop Button
6. Buttons for Auxiliary Function
7. Buttons for Movement Control
8. Operational Buttons
9. Override Switches
10. USB Interface

Led Display

It shows different interactive operational interfaces of *NK300HF Integrated CNC System*.

See figure below for its layout:

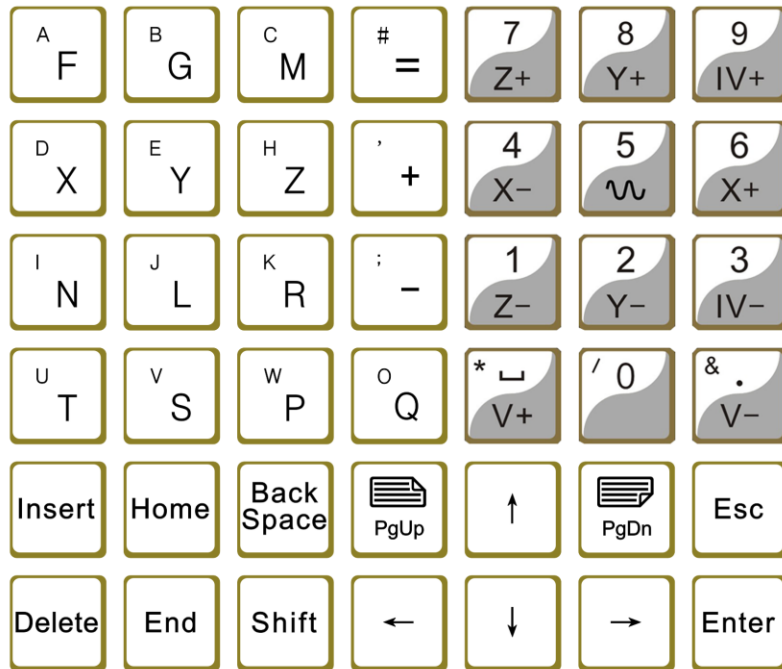


Description of each part is as follows:

1. It shows the current operational mode, such as Auto, Jog, Step, Ref, etc.
2. It shows the current status of the machine tool, such as Idle, Running, E-stop, etc.
3. It shows the name of the current machining file.
4. It shows running time.
5. It shows machining progress, alarms, etc.
6. It shows the name of the current main function access button.
7. It shows the child interfaces of each main function access button.
8. It shows the meaning of F1~F7 buttons under different child interfaces.

Alpha-numeric Keypad

See figure below for alpha-numeric keypad on the operational panel:



How to use:

- It is the same with the use of computer keyboard in general.
- To enter an alphabet:

Directly press the button to enter the alphabet on the right-lower part.





Press **Shift** + alphabet button to enter the alphabet on the left-upper part.

- In manual mode, use numeric buttons as axis direction buttons:

Only press the axis direction button to move the axis at jogging speed.

Press the axis direction buttons and rapid button **5** together to move the axis at rapid jogging speed.

Main Function Access Buttons

Button	Function
	<p>Used to enter into State interface. It includes the following child interfaces:</p> <ul style="list-style-type: none">Coor-AutoMotion TraceCur. ProgramPart Statistics <p>Keep pressing the button or press the shortcut key of the corresponding interface to switch the child interfaces.</p>
	<p>Used to enter into Advanced interface. It includes the following child interfaces:</p> <ul style="list-style-type: none">Coor-ManageCenteringUser Code InputOnekey Cali <p>Keep pressing the button or press the shortcut key of the corresponding interface to switch the child interfaces.</p>
	<p>Used to enter into Program interface. It includes the following child interfaces:</p> <ul style="list-style-type: none">Local ProgramUSB FileProg WizardProgram Task <p>Keep pressing the button or press the shortcut key of the corresponding interface to switch the child interfaces.</p>
	<p>Used to enter into Parameter interface. It includes the following child interfaces:</p> <ul style="list-style-type: none">Machine ParamParam BackupCompensationDriver ParamCoor Backup <p>Keep pressing the button or press the shortcut key of the corresponding interface to switch the child interfaces.</p>








Used to enter into **System** interface.
It includes the following child interfaces:


- Port
- Log
- System Info
- Data Collect

Keep pressing the button or press the shortcut key of the corresponding interface to switch the child interfaces.






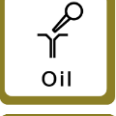





Mode Buttons

Button	Function
	Under this mode, the machine tool will automatically run machining program.
	Under this mode, the machine tool will keep running until you release the button.
	Under this mode, the machine tool will be controlled by handwheel.
	Under this mode, it is convenient for you to execute homing and set datum.
	Under this mode, the machine tool only move a fixed distance. "X100" is the default rate. "X1"/"X10"/"X100" corresponds to 0.001/0.01/0.1 mm(inch). You can customize the stepping length by "XUD" button.







E-stop Button

Button	Function
	Used to stop the machine completely when the machine is in danger. When the danger is cleared, rotate the button in clockwise direction to remove emergency alarm.



Buttons for Auxiliary Function

Button	Function
	Used to make the program run in simulation mode.
	Used to enable handwheel guide.
	Used to execute program by lines, i.e. once the button is pressed, only one line of code block will be executed.
	Used to turn on/off the coolant.
	Used to turn on/off blowing.
	Used to turn on/off the lubricant.
	Used to make the tool magazine rotate in clockwise direction. Not customized yet.
	Used to make the tool magazine rotate in counter-clockwise direction. Not customized yet.
	Used to turn on/off the light. Not customized yet.
	Used to customize function. Not customized yet.
~	
	



Buttons for Movement Control

Button	Function
	Used to make the spindle rotate in clockwise direction.
	Used to make the spindle rotate in counter-clockwise direction.
	Used to stop the rotation of the spindle.
	Used to start programming running.
	Used to stop programming running.
	Used to resume programming running from the exact interrupted position when power interruption or e-stop occurs and the workpiece origin is secured.

Operational Buttons

Button	Function
	Used to execute the corresponding operation in the LED display.
~	
	

Override Switches

Button	Function
	Used to adjust spindle override from 50%~150%.
	Used to adjust feed override from 0%~120%.

Quick Start


Through this part, you can quickly know about how to use *NK300HF Integrated CNC System*.

Machining processes of the system include the following:

1. Loading Program
2. Setting Workpiece Origin
3. Selecting Machining Mode
4. Running Program
5. Executing Tool Lifting and Dropping
6. Checking Machining Statistics

Loading Program File



Press  to enter into **Program** interface.

NK300HF Integrated CNC System supports the following three ways to load the program file:

- [Loading Local File](#)
Used to load program files stored in NK300HF host.
- [Loading USB File](#)
Used to load program files stored in a USB flash disk.
- [Loading Wizard File](#)
Used to load program files generated by NK300HF program wizard.

The format of the program file can be one of the following:

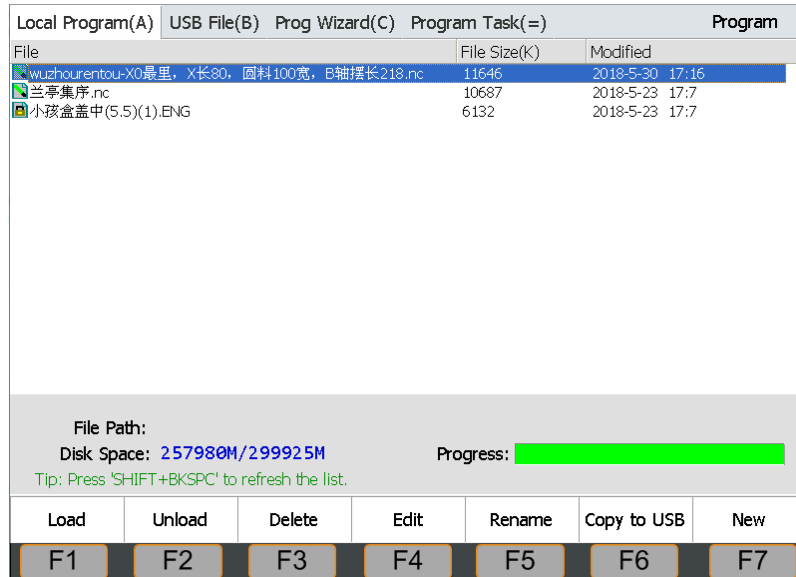
- .nc
- .plt
- .eng
- .dxf

Note: Only the program files with ".nc" format are editable. And the program files generated by program wizard are .nc format by default.

Loading Local File

To access program file stored in directory C:\NK280PC\Card, do the following:

1. Press **A** to enter into **Local Program** interface.

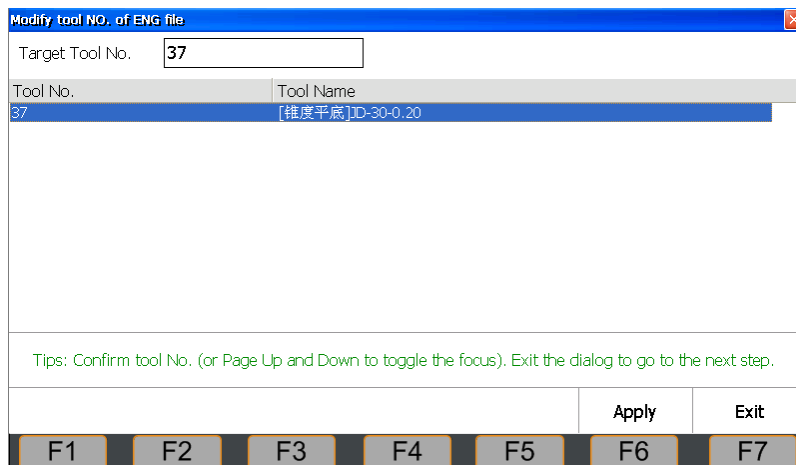


2. Press **F1** to load the selected program file into the system for machining.

If the following conditions are met:

- Parameter **65203 Enable ENG Tool Selection** is set to "Yes".
- Parameter **65207 Enable ENG Tool No. Change** is set to "Yes".
- The format of the loaded file is .eng format.

The dialog box for confirming tool number pops up automatically.



3. **Optional:** Do one of the following as needed:
 - press **F2** to remove the selected program file from the system.
 - press **F3** to delete the selected program file.

- press **F4** to edit the selected program file.
- press **F5** to rename the selected program file.
- press **F6** to copy the selected program file to a USB flash disk.
- press **F7** to create a new program file in the local.

Loading USB File

Before loading file stored in USB flash disk, you need to ensure the following:

- There are program files in the USB flash disk.
- The USB flash disk has been well connected with the NK300HF host.

To access file stored in USB flash disk, do the following:

1. Press **B** to enter into **USB File** interface.



2. **Optional:** Select the folder storing program file, and press **Enter** to open it.

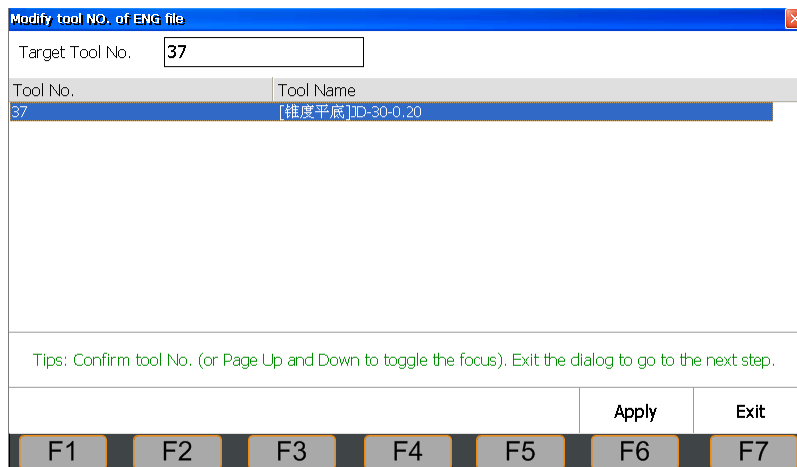
If the program file is under root directory, skip this step.

3. Press **F1** to load the selected program file into the system for machining.

If the following conditions are met:

- Parameter **65203 Enable ENG Tool Selection** is set to "Yes".
- Parameter **65207 Enable ENG Tool No. Change** is set to "Yes".
- The format of the loaded file is .eng format.

The dialog box for confirming tool number pops up automatically:



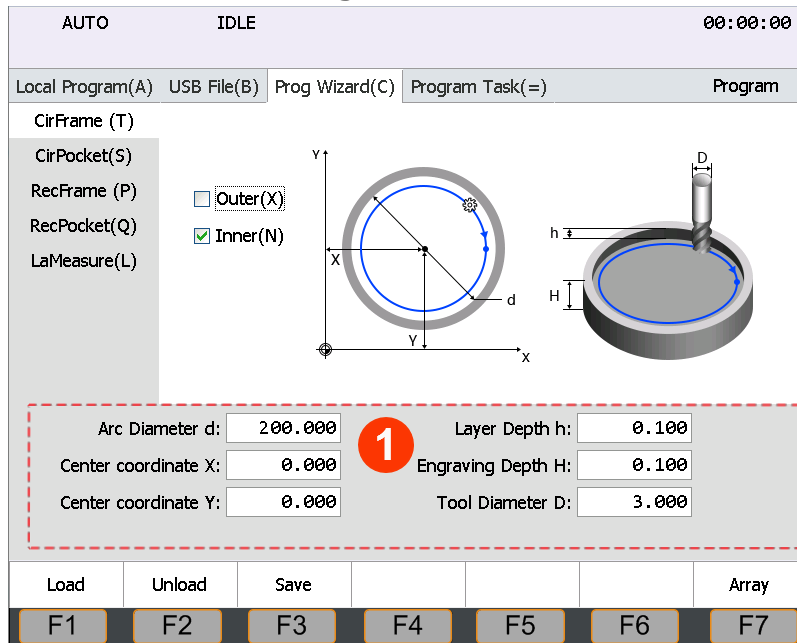
4. **Optional:** Do one of the following as needed:
 - Press **F2** to unload the selected program file from the system.
 - Press **F3** to delete the selected program file.
 - Press **F4** to edit the selected program file.

- Press **F5** to rename the selected program file.
- Press **F6** to copy the selected program file to the local.
- Press **F7** to create a new program file in the USB flash disk.

Loading Wizard File

To access program file generated by program wizard, do the following:

1. Press **C** to enter into **Program Wizard** interface:



2. Press **T/ S/ P/Q/ L** to select the type of program wizard.



3. Press direction keys + **Enter** to set parameters in area ①.
4. Press **F3** > **F1** to load the program file with .nc format generated by program wizard.
5. **Optional:** Press **F2** to unload the program file generated by program wizard.

Setting the Workpiece Origin

NK300HF Integrated CNC System supports the following three ways to set the workpiece origin:

- **Clearing**

Used to clear current workpiece coordinate of each axis and set workpiece offset value as corresponding machine coordinate of each axis when demand for accuracy is not strict enough.

- **Calibrating Tool**




Used to set tool offset so as to determine the specific location for workpiece coordinate system.


- **Centering**

Used to determine the workpiece origin of regular-shaped workpiece and set the workpiece offset.

Clearing

To do clearing, do the following:

1. Press  /  /  to activate manual mode.
2. Move tool to the position where is to be set as the workpiece origin.

3. Press  > **A** > **F6** to enter into **Clear** interface:

X Clear	Y Clear	Z Clear	A Clear	B Clear	All Clear	Back
F1	F2	F3	F4	F5	F6	F7







4. Do clearing on each axis:
 - Press **F1** to clear current workpiece coordinate of X-axis to zero.
 - Press **F2** to clear current workpiece coordinate of Y-axis to zero.
 - Press **F3** to clear current workpiece coordinate of both X-axis and Y-axis to zero.
 - Press **F4** to clear current workpiece coordinate of Z-axis to zero.
 - Press **F5** to clear current workpiece coordinate of all axes to zero.

After doing clearing, the workpiece coordinate turns to zero.

Calibrating Tool

To calibrating tool, do the following:

1. To enter into **Tool Cali** interface, do one of the following:

- Press  >  > **A**.
- Press  /  /  >  > **A** > **F5**.

		Fixed Cali	Set ToolLen	Mobile Cali	Set XY Offset	Back
F1	F2	F3	F4	F5	F6	F7

2. To select calibration method, do one of the following:

- Press **F3** to select fixed calibration.

With this method, the machine tool calibrates tool on a fixed point so as to reconfirm tool offset. This is applicable to multi-tool mode.

- Press **F5** to select mobile calibration.

With this method, the machine tool calibrates tool on current position.

Dialog box for confirmation pops up.


3. Press **F6** to execute calibration.

Centering

To do centering, do the following(take X-axis as an example):

1. Press  /  /  to activate manual mode.

Handle mode is suggested.

2. Press  > **B** to enter into **Centering** interface.

3. To select centering method, do one of the following:

- If the workpiece is regular rectangular workpiece, press **T** to select line center.

With this method, the system finds the center point by two points on the workpiece.

AUTO		IDLE		小孩盒盖中(5.5)(1).ENG		00:00:18	
Coor-Manage(A)	Centering(B)	User Code Input(C)	OneKey Cali(=)	Advanced			
Center(T)		Axis	WorkCoor	MachCoor			
CircleCenter(S)		X	0.000	140.878			
		Y	0.000	7.972			
		Z	10.000	10.000			
<p>1)Move the tool to one side of the workpiece, and press [Record] button; 2)Move the tool to one side of the workpiece, and press [Centering] button.</p>							
Record 1:		Record 2:		Centering			
X:	140.878	140.878	0.000	0.000			
Y:	7.972	7.972	0.000	0.000			
Record X	X Centering	Record Y	Y Centering				
F1	F2	F3	F4	F5	F6	F7	

- If the workpiece is circular workpiece, press **S** to select circle center.

With this method, the system finds the center point by three points on the workpiece.

AUTO		IDLE		小孩盒盖中(5.5)(1).ENG		00:00:08	
Coor-Manage(A)	Centering(B)	User Code Input(C)	OneKey Cali(=)	Advanced			
Center(T)		Axis	WorkCoor	MachCoor			
CircleCenter(S)		X	140.446	281.324			
		Y	0.185	8.157			
		Z	10.000	10.000			
<p>1)Move tool to a point on the edge of the workpiece and press [Record 1] button 2)Move tool to another point on the edge of the workpiece and press [Record 2] button 3)Move tool to a third point on the edge of the workpiece and press [Centering] button</p>							
Record 1:		Record 2:		Record 3:		Centering	
X:	267.712	281.324	0.000	0.000			
Y:	7.072	8.157	0.000	0.000			
Record 1	Record 2	Centering					
F1	F2	F3	F4	F5	F6	F7	

- Optional: Press **F5** to enable the edge finder.
 - Enable: the system can position precisely. At this time, the spindle speed is controlled by parameter **20005 Spindle Speed in Centering**.

- Disable: parameter **2005 Spindle Speed in Centering** is invalid. The spindle speed will be the setting value for the spindle in the software or the rotational speed specified by S command in the program file.
5. Manually move the tool to one side of the workpiece and press **F1**. The system records the machine coordinate of the current point.
 6. Manually move the tool to the other side of the workpiece and press **F2**. The system calculates the middle point according to the second position and the last recorded values, and sets it as the workpiece origin.

Note:

- The 4th and 5th steps belong to the operation of line centering. These two ways of centering are similar. Please refer to the steps on the corresponding interface.
- When X-axis is centered, the other axis should keep still.

Selecting Machining Mode

NK300HF Integrated CNC System supports the following three machining modes:

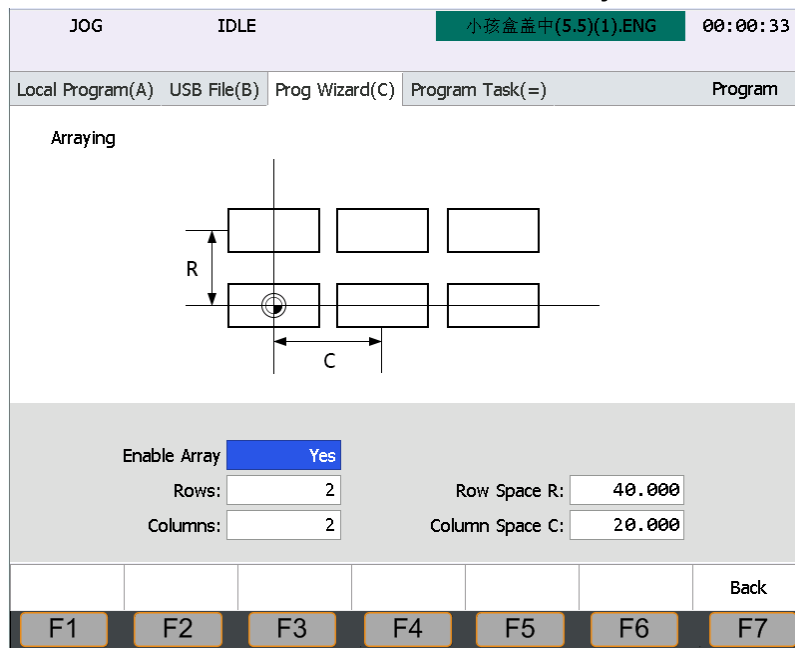
- **Array Machining**
Used to generate complex machining program so as to improve efficiency. And it is suitable to workpiece arranged regularly.
- **Cycle Machining**
Used to complete all actions of machining with a program block under auto mode.
- **Program Task**
Used to put several tool paths into a task.

Array Machining

To execute array machining, do the following:




1. Press **C** > **F7** to enter into **Array** interface:



2. Modify the value of parameter **Enable Array** to "Yes" to enable array machining.
3. Set parameters **Rows**, **Columns**, **Row Space R** and **Column Space C** for array machining.






Press . The system executes array machining on current machining program.

Note: This mode is applicable to all types of program files, not just program files generated by program wizard.


Cycle Machining

To execute cycle machining, do the following:

1. Press  >  > **A** to enter into **Coor-Auto** interface.
2. Press direction buttons to switch to **Prog Cycle Times**.

3. Press  to input program cycle times.



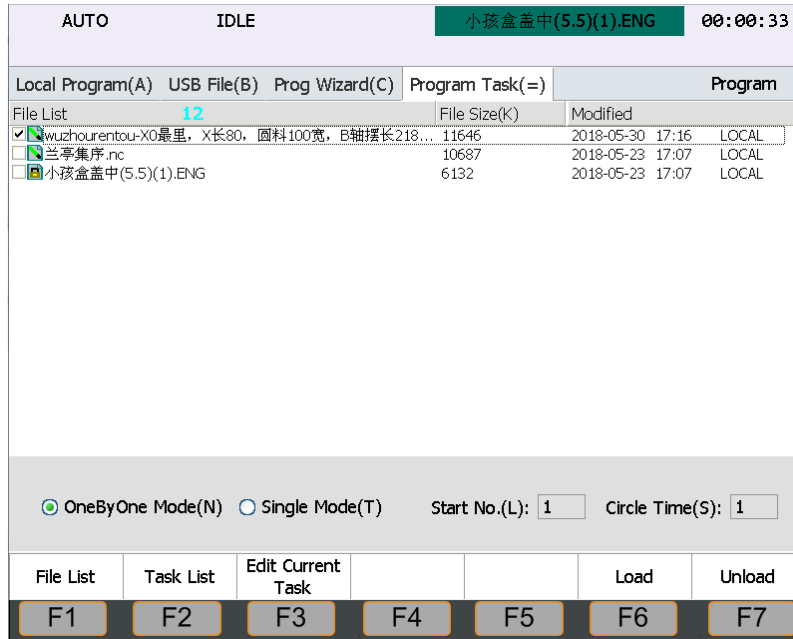
Press . The system automatically machines the set times on the current program file.

Program Task

To choose program task, do the following:



1. Press **Program** > = to enter into **Program Task** interface:

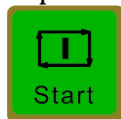


2. To operate on program task, do one of the following:
 - Press **F1** to enter into file list to add or select tool path file.
 - Press **F2** to enter into task list to create new task, rename/delete task or open task.
 - Press **F3** to edit current task to delete/save path tool file, adjust machining order, etc.
3. To select machining mode, do one of the following:
 - Press **N** to select **One by One Mode**.

After machining the currently loaded tool path, the system automatically unloads current tool path and loads another one. After waiting for the set interval, the system continues to machine the new tool path until all tool paths have been machined.

- Press **T** to select **Single Mode**.

After machining the currently loaded tool path, the system automatically unloads current tool path and loads another one, and then enters into



pause status. Press **Start** to continue to machine the new tool path.

4. Press **F6** to load a program task.

Running Program

During running the program, you can check file content, file size, machining range, etc. in the **Cur. Program** interface.

NK300HF Integrated CNC System supports the following four ways to run the program:

- **Auto Run**

Used to automatically run the program from beginning to the end.

- **Single Block**

Used to execute the program by segments. After the execution of a segment, the system will enter into pause status. It can be used for error diagnosis and troubleshooting.

- **Selective Machining**



Used to realize optional skip of the program.

- **Handwheel Guide**

Used to control the execution speed manually during auto run so as to avoid tool damage from wrongly loaded program or inappropriate tool path.





Auto Run

To execute auto run, do the following:

1. Press  to activate auto mode.
2. Press . The system automatically executes the program until end of block or any intentional interruption (e.g. **Stop** or **Pause** button is pressed).




Single Block

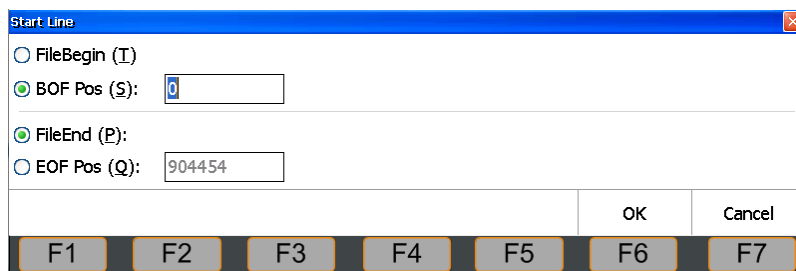
To execute single block, do the following:

1. Press  to activate auto mode.
2. Press . The indicator light turns on.
3. Press . The system stops after executing one block of the program and enters into Pause status.
4. Press  again to continue to run another block of the program until the whole program has been executed.

Selective Machining

To execute selective machining, do the following:

1. Press  to activate auto mode.
2. To open dialog box **Selective Machining**:
 - Press  > **C** > **F1**.
 - Press  > **A** > **F2**.



3. Press shortcut buttons **T**, **S**, **P** and **Q** to set program start/end position.

Handwheel Guide

To use handwheel guide, do the following:



1. Press **Auto** to activate auto mode.
2. To enable handwheel guide, do one of the following:



- Press **HW Guide**. The indicator light on the upper left of the button turns on.



- Press **State** > **A** > **F1**.

Executing Tool Lifting and Dropping

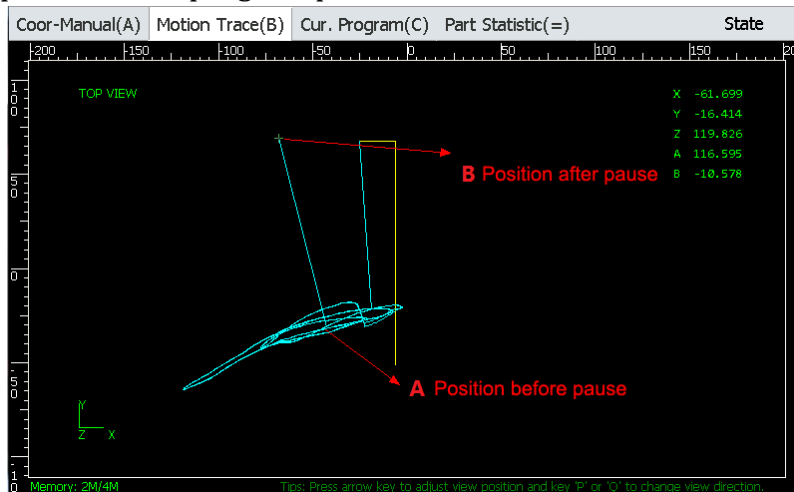
It is used to avoid workpiece damage from triggering commands like Pause and Program Stop during machining under **Five_Axes(AB Model)** and **Five_Axes(Z+1 Model)** configurations.

Function of each axis is as follows:


- A-axis: control the rotation of the workpiece.
- B-axis: control the rotation of the tool.
- XZ-axis: control tool lifting and dropping.

To execute tool lifting and dropping, do the following:


1. Set the following parameters:
 - Set parameter **73003 Enable Five Axis Lift-drop When Pause and Resume** to "Yes".
 - Set parameter **73004 Z-axis Lift Distance on Pause** as needed, such as 10.
2. During machining, press **F3** when Z-axis moves to A point. Then Z-axis lifts to B point and the program pauses.






3. Press . Z-axis returns to A point first and the system continues machining.



Note: The movement of Z-axis is also applicable to the operations: press 



during machining > 【Operation, e.g. move axis】 > .

Checking Part Statistics

It is used to show machining statistics of both history and current programs. It is convenient for you to do statistics after exporting all of data.

To check machining statistics, do the following:



1. Press **State** > **=** to enter into **Part Statistics** interface:

No.	Program Name	Signal Total Time	Total Time	Signal Total Len.	Total Len.	Count
2	兰亭集序.nc	00:35:02	05:49:57	43557.231	174228.924	4
1	兰亭集序.nc	00:35:08	00:35:08	43557.231	43557.231	1

Cur. Program: 小孩盒盖中(5.5)(1).ENG	Single Time: 00:00:00
Cycles: 0	Total Time: 00:00:00
Part Count: 0	Single Len.: 0
	Total Len.: 0

Clear	Export					
F1	F2	F3	F4	F5	F6	F7

- ① shows machining statistics of history programs.
 - ② shows machining statistics of the current program.
 - ③ shows operations you can do, including clearing and exporting workpiece statistics.
2. **Optional:** Press **F1** to clear all statistics in area ①.
 3. **Optional:** Press **F2** to export all statistics in area ① to external storage device (e.g. a USB flash disk). And the statistics will be saved as a .txt document.

FAQs

During usage of *NK300HF Integrated CNC System*, you may encounter following questions:

What to do if the spindle does not rotate?

What to do if an axis does not move?





What to do if servo motor Z brake can't be opened?

What to do if machine tool returns to the machine origin abnormally?

What to do if the machine tool motions upward after arriving at the position of tool sensor during calibration?

What to do if the spindle does not rotate?

Troubleshoot as follows:

- Press  /  >  > **A** to see if the dot before the **SPIN/Spindle CCW** becomes . If it does, the software works normally.
- Start spindle start output **Y00 SPIN/Y02 Spindle CCW**, and check if the spindle start indicator lamp on the terminal board is on.


If not, check whether the connection cable of terminal board becomes loose : close the host machine and power off machine tool, and then re-plug the connection cable of terminal board.

- If it still does not light, check if there is an error in cable of terminal board, terminal board or system.
- If it lights, measure if the SPIN port is connected with a multimeter: If it is connected, the spindle start output port works normally. if it is not conducted, there is an error in the relay of spindle start.
- Test whether the analog voltage output is normal between AVC and GND with a multimeter. If not, check if the connection cable of terminal board becomes loose. If it is not loose, check if there is an error in the cable of terminal board, terminal board and system.
- Check whether the parameter setting of inverter is right, the spindle and inverter has been damaged, or the wiring of the spindle and inverter is correct.

What to do if an axis does not move?

Troubleshoot as follows:



- Press **System** > > **A** to check if the signal of × **servo enable** output port becomes :
 - Yes: the software works normally.
 - No: check if the port polarity (it should be NO “N”) is set correctly.
- Check parameter setting, like setting control mode as position control, selecting pulse input port for Panasonic driver, etc.).
- Check if the servo cable of this axis is well contacted at the joint with system host machine and servo driver.
- Check if something is wrong with servo driver, motor cable, servo cable or control system. (e.g. exchange servo cable and servo driver with those of other axes working normally).

What to do if servo motor Z brake can't be opened?

Troubleshoot as follows:

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

- If it turns light, test with a multimeter to see whether there is 24V voltage between brake output ports (BRAKE-COM). 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 terminal board directly.
- If not, directly conduct the brake input ports (K+, K-) on terminal board with a conducting wire.
 - If it turns light at this time, check whether the servo driver is enabled, the parameter setting related to brake output of servo driver is right, and brake output line of driver is correctly wired to terminal board (black line is wired to K-, and only K+ is connected to with only one line).
 - If it is still not light, please change the terminal board.

What to do if machine tool returns to the machine origin abnormally?

Common abnormal situations and corresponding solutions are as follows:



Abnormal Situation 1:

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

Solution:

1. Check if the software can receive the REF. point signal of this axis.





- Trigger the home switch and press  > **A** to see if the color of the dot before the **× Machine Origin** port turns to . If the color does not change, it indicates the software can't receive the REF. point signal. Then, check the following:
 - Home switch
 - Wiring of home switch. Make the REF. point signal on the terminal board and COM port into conduction directly with a conducting wire, and then check whether the color of the dot before **× Machine Origin** changes.
2. 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.
 - The position of home switch is out of the mechanical stroke of machine tool.

Abnormal Situation 2:

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

Solution:



Trigger this home switch, press  > **A** to see if the dot before the **× Machine Origin** input port is .

Abnormal Situation 3:

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

Solution:

This is due to that the system can't detect the encoder REF. point signal of this axis. Do the following:

1. See if the servo cable of this axis is well contacted at the joints with NK300HF host and servo driver.
2. If the driver brand is YASKAWA or TECO, set driver the value of parameter **PG Pulses** to half or quarter of the original.
3. Troubleshoot the driver, motor, encoder cable, servo cable or the CNC system.


e.g. exchange servo cable and servo driver with those of other axes able to return to the machine origin normally in turn.

What to do if the machine tool motions upward after arriving at the position of tool sensor during calibration?

Troubleshoot as follows:



Press **System** > **A** to observe the dot before **X13 Cut** input port.

The dot is  when the system does not receive tool sensor signal.

SPECIALIZED / CONCENTRATED / FOCUSED



Shanghai Weihong Electronic Technology Co., Ltd.

Address: No. 1590, Huhang Rd., Fengxian, Shanghai, China, 201401

Hot-line: 400 882 9188

Website: www.weihong.com.cn/en