

Hand-Held Controller Manual

1. System Characteristics of NK105

The NK105 is a standalone motion controller that is operated with a hand held controller that connects to a main control box. The software running the system is called Ncstudio. It is easy to use and provides many flexible functions.

Ncstudio™ system includes the following characteristics:

- Supports G code of ISO standard, file format of ENG/DXF/NC.
- Supports incremental feed in continuous and step mode. The user can adjust the feed rate and the length of step.
- Supports copying and processing of files on a USB flash disk.
- Supports spindle speed adjustment in 8 steps from High to Low. Spindle speed can be adjusted in real time during processing.
- Supports advanced processing modes, including processing by selecting file line number. (Skip function.)
- Supports pause during processing and skipping to pause point. 3 breakpoints can be saved after a pause, and user can select any of the 3 breakpoints to continue processing.
- Machine has a fixed reference point, mechanical origin, from which locations can be referenced.
- Supports multi-workpiece mode, rotary mirror image function and arc speed limit function.
- Supports auto tool calibration function, which allows the user to set Z-zero location quickly.
- Supports feed ratio on-line adjustment. In the machining process, user can adjust the feed ratio at any time. The least value is "0", meaning processing suspension, and the maximum ratio is 120%.
- Contains a high-speed smoothing algorithm. NCstudio contains an adaptive prediction algorithm for processing speed. In accordance with speed value, direction and maximum acceleration, and the function of forward prediction, the algorithm decides the connection speed between the current instruction and the next instruction. This increases the processing efficiency making the machine cut in faster times with a improved edge finish.
- Contains a flexible acceleration/deceleration control algorithm, which improves cut quality.
- Ncstudio has a special embedded processing file manager. The user can save files to internal memory on the controller and access files quickly.
- Parameters for the controller can be saved and easily recalled if the firmware becomes corrupt.

2. Basic Concepts of NK105

NK105 application system involves various concepts, such as workpiece coordinate system, mechanical coordinate system, operation mode, operation state etc. Therefore, it is necessary for user to grasp the meaning of these concepts before using NK105.

2.1. Operation Mode and State

2.1.1. Operation Mode

It is necessary for user to understand operation mode correctly. For Machine operation there are two modes.

Auto mode

Under automatic operation mode, the machine is taking commands from the G-code file. The user can pause, adjust the speed and feed, or stop the machine.

Manual mode

Under Manual mode the machine operation is being controlled by the user. There are four basic Manual modes: jog, stepping, origin and tool calibration motion modes.

- **[Jog] mode:** The machine is moved by the user with the directional arrows on the hand held controller. This is used to move the machine quickly and has low or high speed.
- **[Stepping] mode:** The machine will move by the exact amount as specified in the Step size. Used for precise positioning of the tool.
- **[Origin] mode:** This will send the machine to the Origin on the table.
- **[Tool calibration] mode:** This is used to set the Z zero position by touching the tool off the calibration tool.

2.1.2. Operation State

Operation state can be defined as the Status of the machine. Operation state will be shown on the top right of the display. The machine can be in one of the following states of operation.

IDLE state

Idle state is when the machine has no input and it is ready to accept commands for operation.

E-STOP state

E-stop state occurs when there is a hardware fault, such as a cable disconnected or the E-stop is pressed. Upon reaching such state the spindle and pump will turn off. The machine cannot be jogged until the e-stop state has been reset and a normal state is active.

Running state

When the machine is implementing any action, system enters into running state. Such as homing, running a g-code file or going to origin.

Pause state

When a file is running and the user presses “**pause**,” system will enter into this state and wait for further instruction. At this time, the user can let system enter the “running” state by pressing “pause/continue” key or stop system by pressing “stop/cancel” key.

LOCK state

Lock state is an internal state which occurs when soft limit operation is performed, i.e. an axis tries to move beyond the tables size of the machine.

2.2. Machine Coordinate System

The coordinate system describes the motion of the machine tool, in the XYZ plane. For the sake of unification, the standard coordinate system adopts the right-hand rule, as shown in **Fig.2-1**:

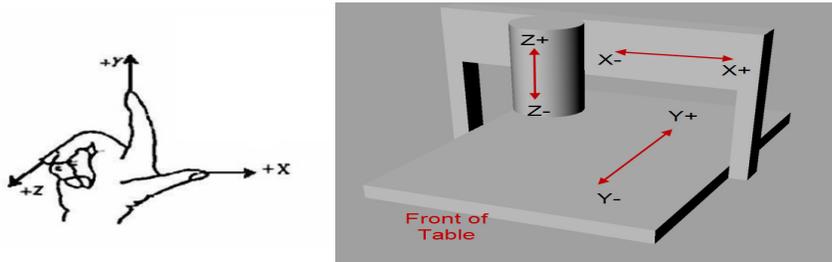


Fig. 2 1 Coordinate system abiding by right-hand rule

- **—Z-axis** is the up/down movement of the spindle. The direction of the cutter moving away from the workpiece is the positive direction of the Z axis (+Z).
- **—X-axis** is the left to right movement of the machine, if facing the spindle. X+ is right, X-is left.
- **—Y-axis** is the front to back movement of the machine, if facing the spindle. Y+ moves toward back, Y- to the front.

2.2.1. Mechanical Coordinate System (MCS)

Mechanical coordinates system (MCS) is the plane determined by the home switches. The **MCS** is a fixed system. The zero position will remain constant and never change in this coordinate system. In the **MCS** the XYZ zero position is at the extent of travel where the machine triggers the Home switches. Generally **MCS** is only used to reference an offset.

2.2.2. Workpiece Coordinate System (WCS)

Workpiece coordinates system (WCS) is the coordinate system that will be most often used when machining parts. **WCS** is the XYZ origin/zero point that is determined by the user on the table. In the CAD/CAM software the user determines where XY zero is in relation to the part they want to cut and if the Z zero is on the top, or bottom of the material. Generally this is in the bottom left corner of the material and the Z zero is on the top of the material.

On the machine the user will jog the machine where they want the **WCS** XYZ zero to be, and press the Zero XY and Zero Z button. The coordinate readout on the controller will now be zero and operating in the **WCS**

3. Functions & Operation Methods of Panel Keys

3.1. Brief Introduction

The arrangement of panel keys is shown in Fig. 4-1.



Fig. 4-2 Panel keys arrangement picture

3.2. Function Information of Each Single-key

NK105 operation panel is simple and easy to use. All the functions of the machine are carried by pressing one key, or by pressing a combination of two keys. Table1 displays the function of each key

Table 1 Single-key function table

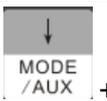
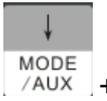
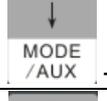
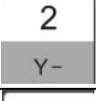
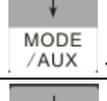
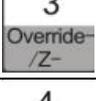
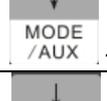
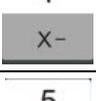
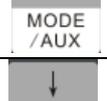
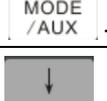
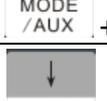
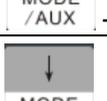
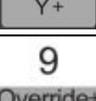
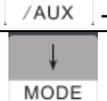
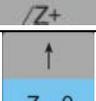
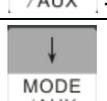
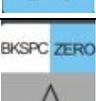
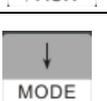
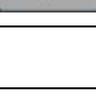
Key icon	Function
7 SPDL+	Increases spindle speed when the spindle is On / Input of number 7
8 Y+	Moves Y-axis in the positive direction. / Input of number 8
9 Override+ /Z+	Positive shift of Z-axis / Input of number 9 / Increases the feedrate when a file is running. (The range is from 0.1 to 1.2, each press increasing 0.1)
4 X-	Moves the X-axis in the negative direction / Input of number 4
5 SPDL ON/OFF	Start/stop of spindle under manual mode / Input of number 5
6 X+	Moves the X-axis in the positive direction. / Input of number 6
1 SPDL-	Decreases spindle speed when the spindle is on / Input of number 1
2 Y-	Moves the Y-axis in the negative direction / Input of number 2
3 Override- /Z-	Moves the Z-axis in the negative direction / Input of number 3 /Decreases the feedrate when a file is running. (The range is from 0.1 to 1.2, each press increasing 0.1)
0 HIGH SPEED	Selection of high or low speed movement when jogging / input of number 0
- XY→0	Moves X and Y axis to the zero position. / input of minus sign
· MENU	Entering menu setup/ input of decimal point/ forced entrance to BOOT program during system startup
BKSPC ZERO ^	Move XYZ to workpiece origin/ Backspace during input/ Page up in menu page/ Alternate between “plus-minus, yes-no and polarity” during input
↑ Z→0	Move Z-axis to Zero position/ up direction key
START V	Page down in menu page / Alternate “plus-minus, yes-no and polarity” during input / Start processing file in processing page
MACHINE ORIGIN OK	Move to mechanical origin under [origin] mode /Confirmation of various selections, inputs and operations
↓ MODE /AUX	Down direction key / Mode shift under processing page / Shortcut key button (mode button is valid when pop-up.)

 	Adjustment of manual High/ Low speed in processing page/ Exit Processing page/ Cancel of various selections, inputs and operations
--	--

3.3. Function Information of Combination Key

The usage of each combination key is as follows: press the first key then the second; release the two keys simultaneously after the corresponding content appears.

Table 2 Combination key function table

Key icon	Function
 + 	Floating calibration under [tool calibration] mode
 + 	Run file from breakpoint 1
 + 	Run file from breakpoint 2
 + 	Run file from breakpoint 3
 + 	Unload file. This must be done before a new file is loaded into the system.
 + 	Pause during processing
 + 	Resume after pause
 + 	Release and recovery of hard limit
 + 	Switchover between WCS and MCS
 + 	Selection of WCS (from G54 to G59)
 + 	System reboot
 + 	Log-out
 + 	Selection of a processing file: after this combination key is pressed, system will load the processing file; if the file has been loaded, system will give a prompt that "please unload first", at this time, combination

3.4. Backing to Machine Origin

Machine origin is determined by the physical location of the limit/home switches on the machine. It is also called the Home Position. This is a fixed position that remains unchanged. Once the controller knows this position it can be used as a reference position to recall and save origins in the WCS, recall and save breakpoints etc. It is essential to move to the machine origin when the router starts.

3.4.1. Parameter Setting of Backing to Machine Origin

The [Back to machine origin] parameters include [back to mechanical origin speed], [back to mechanical direction] and [retract distance].

◆ **[Back to ref pt sp] (back to reference point speed)**

This parameter controls the speed at which the machine moves to the Home position. If the speed is too fast, the gantry can hit the hard stop possibly damaging the machine.

To adjust this speed:

After the system start-up, press  to enter menu page, move the cursor to

“machine param” item, then press   to enter configuration item.

Then press  to move the cursor downward to “back to reference point”.

Now press   to enter, move the cursor to “back to mechanical origin speed”.

Press   to set the speed. The setting picture is as shown in Fig. 4-3.

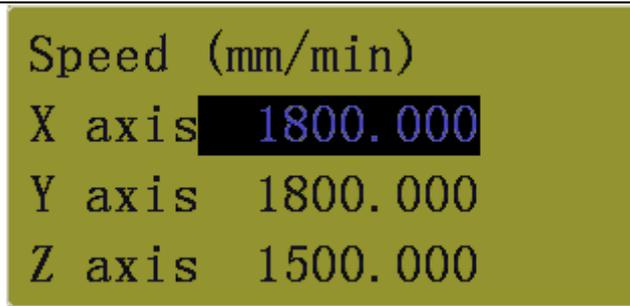


Fig. 4-3 Setting picture of [back to mechanical origin speed]

Press  to enter modification mode and clear all the figure of the selected item, then enter a new value. If the value is entered incorrectly, press  to delete the last character, or press  to delete the new entered value and return to show the original value.

Press  to save the parameter when the correct value is entered.

Press  or  to switch to Y and Z axes.

Note:

Data entry and modification can't be executed if  is not selected before input.

If the new value is not saved before switching to other axes during input, original value will be restored.

Pressing "cancel" during input will shift to modification mode. The previous input will be canceled, and the original value will be restored. Without selecting, user can input a new value again. Then press "cancel" again to exit from the current page and return to the upper page. If modification mode has not been shifted to, press "cancel" to exit from the current page directly and return to the previous page.

◆ **[Back to ref pt dir] (back to reference point direction)**

This parameter is determined by the direction settings of the motor, and the location of the Home switch on the machine. Unless additional home switches have been installed on the machine, the factory default for this setting is the only acceptable setting.

In the Machine Param menu.

Press  to move cursor to "back to mechanical origin direction" option.

Press  to enter the direction menu. The setting picture is as shown in Fig. 4-3.

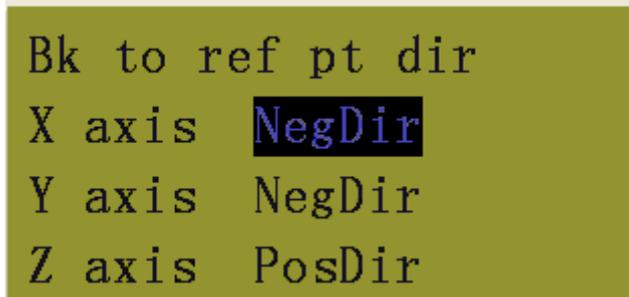


Fig. 4-4 Setting picture of [back to mechanical origin direction]

Press to modify the value, then press and to change the direction.

Press to save the parameter. Press or to move to Y and Z axes.

Note:

Attribute modification can't be executed if is not selected before input.

If you switch to other axes during input, original attribute will be restored and the new entered attribute will not be saved.

Pressing "cancel" during input will shift to modification mode. The previous input will be canceled, and the original value will be restored. Without selecting, user can directly change the attribute again. Then press "cancel" again to exit from the current page and return to the upper page. If modification mode has not been shifted to, press "cancel" to exit from the current page directly and return to the upper page.

◆ **[Retract distance]**

This parameter is the distance the machine backs off once it has triggered the Home/Limit switch.

The modification method of [retract distance] parameter is the same as that of [back to mechanical origin speed] parameter.

3.4.2. Operation Method of Backing to Mechanical Origin

Once the system has started, press to shift current mode to origin mode,

then press to execute the operation of backing to mechanical origin/ homing the machine. The direction, speed, and retract distance of backing to origin are decided by the

corresponding parameters previously set.

4. Processing Operation

There are two processing modes, Manual and Automatic.

4.1. Manual Processing

Manual processing is when the movements of the machine are controlled by the user. After backing to the home position, the system will enter manual state and the screen display is as shown in Fig. 6-1.

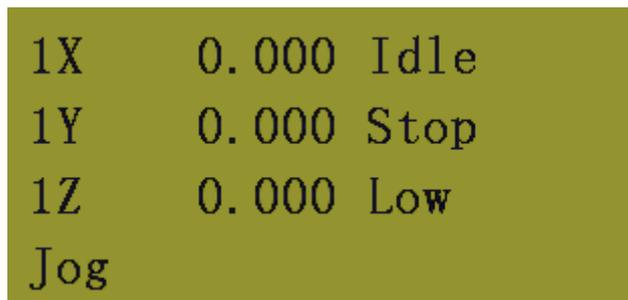


Fig. 6-19 Screen display of manual processing

4.1.1. Manual Processing Parameters Setup

There are two continuous speeds that the machine can be jogged at, high and low.

Manual high speed and manual low speed can be switched by pressing high speed key



The machine can also be jogged in Step mode. In step mode the machine will move an exact amount with every push of the jog button. This is used to move the machine with high precision.

The High/Low speed and the step size can be adjusted in the parameter menu, access this

menu by pressing  , as shown in Fig. 6-2.

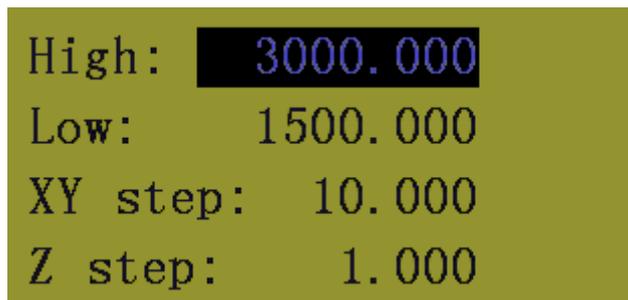


Fig. 6-20 Manual processing parameters setting page

Move cursor to select the parameter to be by pressing up key  or down key . After selection, press  to enter modification mode and clear the figure of the selected option. Then input the new value. If the input is not correct, press backspace key  to delete the last character, or press cancel key  to delete the input value, The original figure will be restored. Confirm the modification and save the parameter by pressing  key. Then press down key  or up key  to switch to Y axis or Z axis.

4.1.2. Manual Processing Mode

There are four modes for manual processing. To switch between these modes press . The mode will be displayed at the bottom of the screen.

◆ Jog motion mode

In jog mode the machine can be moved in continuous motion with the numeric pad. Files can also be loaded and XYZ zero position can be set.

◆ Stepping motion mode

In step mode the machine will move an exact amount with every push of the jog button. This is used to move the machine with high precision.

◆ Origin motion mode

In this mode the machine will move to the Home position when  is pressed.

◆ Tool calibration motion mode

This mode is used to touch off the Z-zero position with the Tool Calibration block. The

combination key  + , is used to activate the Touch off sequence. The Z axis will slowly move down and the tool will touch the calibration block. After the tool calibration is completed, the Z axis will move 5mm away from the surface of calibration block.

4.1.3. Switchover between WCS and MCS

WCS and MCS can be switched by pressing the combination key  + .

The screen display is shown in Fig. 6-3.

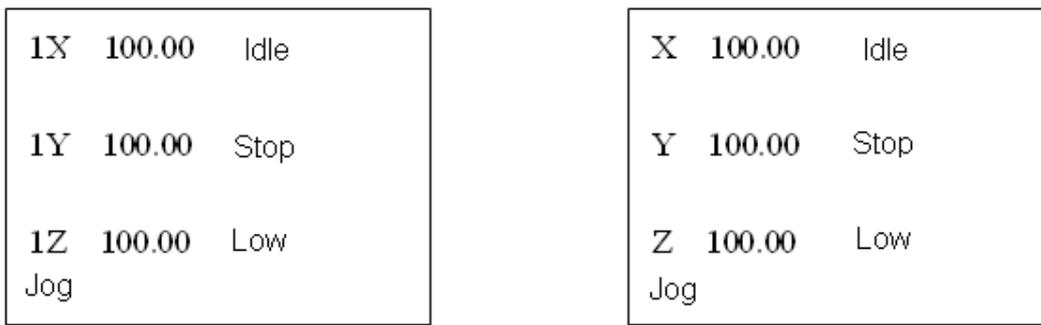


Fig. 6-21 Screen display diagram of WCS and MCS

4.1.4. Selection of WCS

The 6 workpiece coordinate systems can be saved. They can be selected by pressing the

combination key  + , and their screens are shown below.

1X 100.00 Idle	2X 100.00 Idle	3X 100.00 Idle
1Y 100.00 Stop	2Y 100.00 Stop	3Y 100.00 Stop
1Z 100.00 Low Jog	2Z 100.00 Low Jog	3Z 100.00 Low Jog
4X 100.00 Idle	5X 100.00 Idle	6X 100.00 Idle
4Y 100.00 Stop	5Y 100.00 Stop	6Y 100.00 Stop
4Z 100.00 Low Jog	5Z 100.00 Low Jog	6Z 100.00 Low Jog

Fig. 6-22 Screen display diagram of different workpiece coordinate systems

4.2. Auto Processing

Auto processing is when the machine is running a g-code file.

4.2.1. Operation Steps of Auto Processing

◆ Confirm the workpiece origin

Jog the machine so the tool is in the location that you want to set as the XY zero position, also

called WCS origin. Press  for zero the XY coordinates. The display will now read X 0, Y 0.

There are two ways to set the Z zero position.

1. Slowly jog the tool down until the tool is on top of the material. Press  to zero the Z coordinate.
2. Place the Calibration Tool Block on top of the material and activate the Touch off Z-zero as described previously.

◆ Load the File

Once the origin is set, press this combination key  +  to enter file selection page.

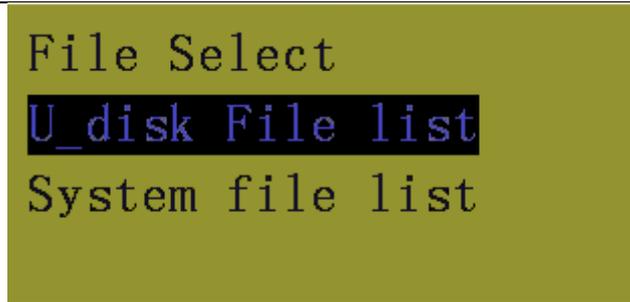


Fig. 6-23 File selection page

Move cursor to select a file list by pressing  or . Then press  to enter the selected file list. Move cursor to select a file by pressing  or . Press the ok key  to load the selected g-code file. System will automatically switch to the processing page when the file is loaded.

◆ **Start processing**

Press the start key  to start auto machining in the processing page.

Information such as feed override and feed rate are scrolled on the screen during file processing.

4.2.2. Machining Process Adjustment

◆ **Feed override adjustment**

Feed override can be adjusted by pressing  or  during file processing.

Current feed rate= machining speed * current feed override

The range of feed override is 0.0 ~ 1.2.

◆ **Spindle speed adjustment**

Spindle speed is adjusted by pressing  or .

◆ **Suspend processing and adjust position**

Pause the file by pressing the combination key  + .

◆ **Continue after pause**

Resume the file by pressing the combination key  + .

4.2.3. Processing Exit, Breakpoint Saving & Resuming

To abort a file before it completes press  , the following screen will be displayed.

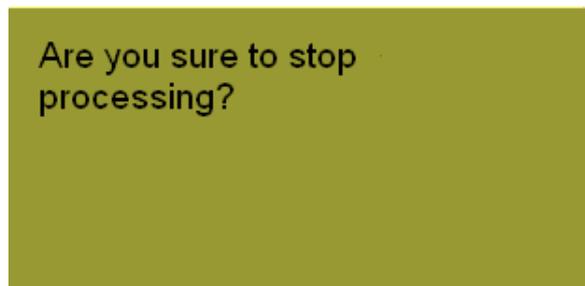


Fig. 6-24 Processing stop and system exit prompt page

Press , and the screen display is as following:

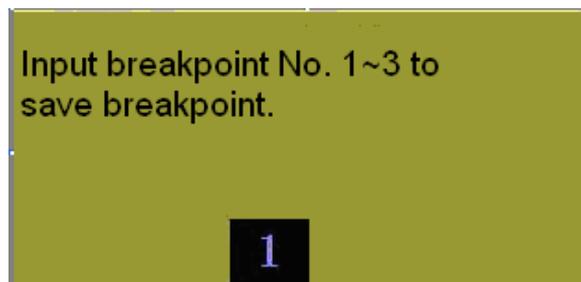


Fig. 6-25 Breakpoint saving prompt page

System can save three breakpoints. User can enter the breakpoint No. by pressing number key 1 ~ 3 when the breakpoint prompt appears. Then press the ok key to save the current breakpoint and stop processing. Processing page will display:

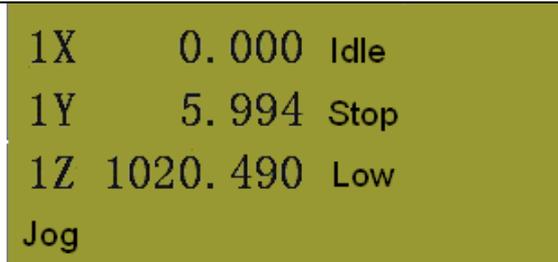


Fig. 6-26 Page after exit from processing

To resume from breakpoint 1 press  + . The system will display a prompt “are you sure to start breakpoint resume?”. Press OK to continue processing from breakpoint 1.

4.2.4. Limit Treatment

◆ Software limit

Software limit occurs when a file exceeds the set “machine stroke range”, table size. Jog the machine in the opposite direction if the limit is reached.

◆ Hardware limit

If the machine hits the hard limit, press the combination key  +  to release the limit. Jog the machine in the opposite direction of the limit once it has been released.

5. Menu Page

5.1. Summarization

The default mode after system start-up is manual mode. The page for this mode displays the following information: coordinate axis, coordinate value, operation state, spindle state, type of manual speed and processing mode, as illustrated in Fig. 5-1.

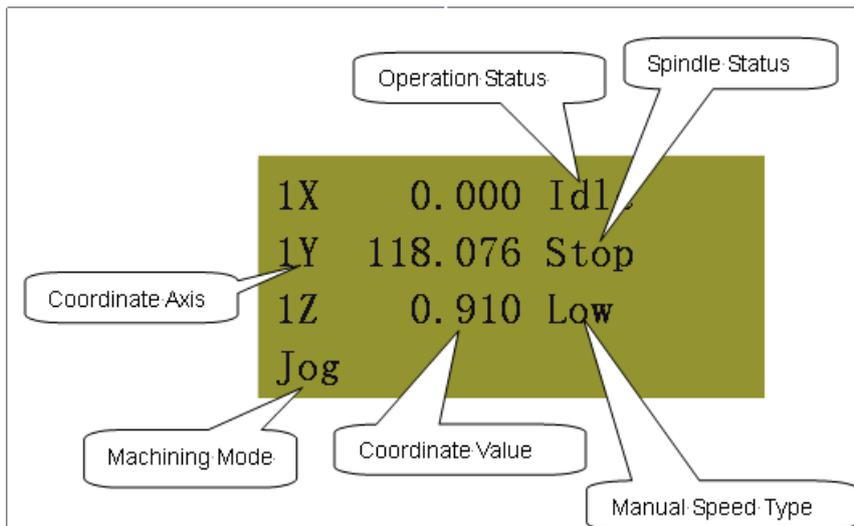


Fig. 5-1 Processing page

The coordinate system displayed will be the WCS by default and will be indicated by a number and the axis ie 1X. The MCS will be indicated by an asterisk (*) beside the axis. Alternating

between the coordinate systems is done by pressing the combination key  and .

Fig. 5-1 is WCS, and the number 1~6 before X/Y/Z axis indicates the coordinate system that can be called up by G54 to G59 in the G-code.

Machining mode can be selected by pressing . The mode can either be jog, stepping, origin and tool calibration.

Operation state includes idle, E-stop, running, pause and lock states.

The spindle can be manually controlled by pressing .

Manual speed can be either manual high or low speed, which can be switched by .

Please refer to chapter 6.1.1 manual processing for further information.

Press  to enter menu page. There five parameter menu items, but the LCD can only show 4 of them, as shown in Fig. 5-2.

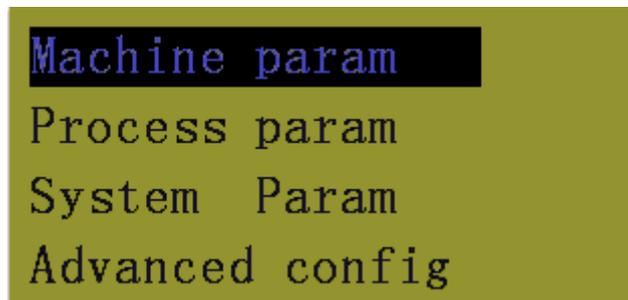


Fig. 5-5 Menu parameters page

Move cursor to select a parameter item by pressing , ,  and . Then

press  to enter its submenu.

5.2. Machine Parameter Configuration

These parameters are related to mechanical operation of the machine. Please note that improper setting of these parameters can cause abnormal execution of file operation, mechanical failure and even injury to the operator. Therefore, great care should be taken when modifying these parameters. If it is necessary to change them, please carry out it under the guidance of Techno Tech Support.

The menu level included in machine parameter configuration is as follows.

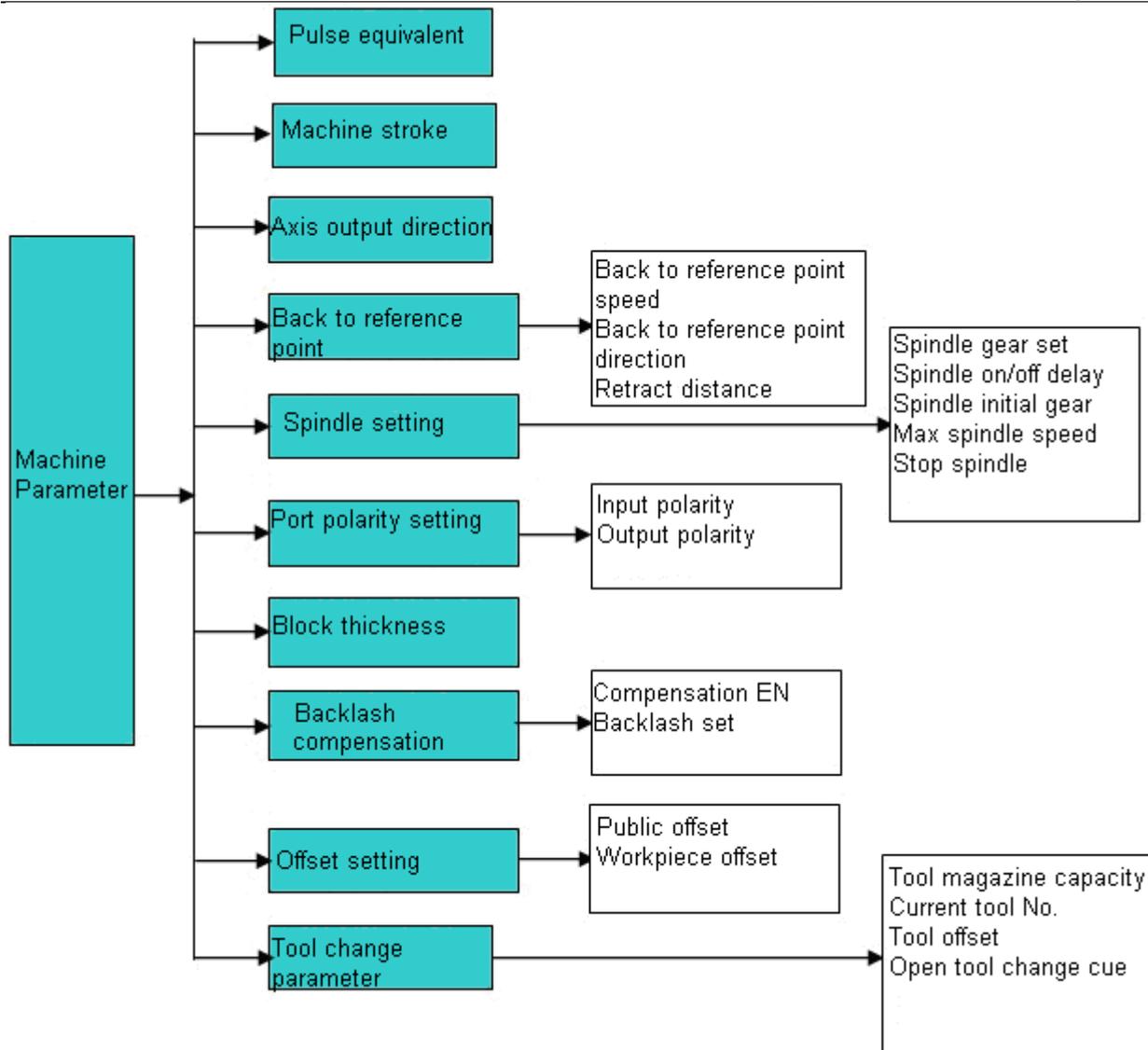


Fig. 5-6 Submenus of all different levels under machine parameter configuration page

5.2.1. Pulse Equivalent

Pulse equivalent is how much the machine moves when one pulse is applied to the motor. Its unit is millimeter/ pulse and its calculation formula is as follows:

$$\text{Pulse equivalent} = \text{lead screw pitch} / ((360 \text{ degree} / \text{stepping angle}) * \text{subdivision number})$$

If the value of pulse equivalent is not correct the dimensions of parts will be inaccurate.

WARNING: DO NOT CHANGE THESE VALUES UNLESS INSTRUCTED TO DO SO BY TECHNO. INCORRECT VALUES COULD DAMAGE THE MACHINE.

When cursor is on the configuration item of pulse equivalent, press   to enter password page which will prompt user to enter the password. Default password is 123456.

The procedure for modification of this value is the same as modifying [back to mechanical origin speed].

Pulse equivalent menu is shown in Fig. 5-4. The sign “*” before the title bar indicates modification to this parameter is valid after system reboot.

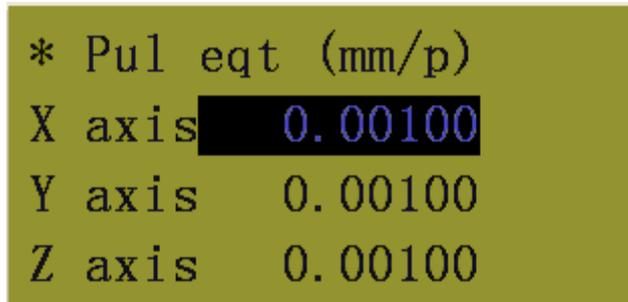


Fig. 5-7 Setting page of pulse equivalent

5.2.2. Machine Tool Stroke

Machine tool stroke refers to the size of the table. This will set the table size from the MCS zero position. This value must be accurate and should be the dimensions from the Home position to the opposite end of the gantry.

If the file had dimensions that exceed the table size, a message will appear saying the processing is out of range, i.e. “X-axis positive limit”. This will eliminate the possibility of the machine hitting the end of travel.

Modification of this parameter is valid after reboot.

5.2.3. Axis Output Direction

This setting will change the direction the machine jogs. Changing this setting will require modifications to homing direction, machine tool stroke, and other settings. Modifying this setting will essentially flip the jog direction of the machine. -X will now be +X etc.

Modification to this parameter is valid after reboot.

5.2.4. Backing to Mechanical Origin Setup

See section 1.4. for an explanation of this function.

5.2.5. Spindle Setup

This menu contains the parameters for control of the spindle, including parameters of spindle gear/speed settings, spindle on/off delay, spindle initial gear and maximum spindle speed. Spindle

speed can be changed while the spindle is running by using the handheld controller or by using the controller on the inverter, depending on the inverter and controller programming.

◆ **[Spindle gear setting]**

To access this setting: move cursor to “spindle gear setting”, press  to open the menu.

This number determines speed table of the spindle. There are 7 steps available. Press  to confirm the input.

Then set the line state of each gear speed according to state definition of multi-section speed control interface of inverter. “○” corresponds to the OFF state of transducer multi-section speed,

while “●” corresponds to the ON state of transducer multi-section speed. Press  or  to

move cursor, and  or  to alter the attribute. After setup finishes, press  to confirm

the modification and shift to the modification page of the next line state, or press  to cancel the modification and return to the upper menu.

Note:
Modification to this parameter becomes valid after reboot.

◆ **[Spindle on/off delay]**

This is the amount of time the machine will wait for the spindle to reach the required speed, before the machine starts running a g-code.

This page only has one parameter. Input the value directly without selecting it, and then press

 to save the value and return to the upper menu, or press  to cancel the input value and return to the upper menu.

◆ **[Spindle initial gear]**

This is the default speed for the spindle.
Modification to this parameter becomes valid after system reboot..

◆ [Maximum spindle speed]

This sets the maximum speed of the spindle. This should be the speed that is programmed into the inverter.

Modification to this parameter becomes valid after system reboot.

5.2.6. Port Polarity Setup

Port polarity defined the Input/Output settings on the controller board. It determines is an i/o is active high, or active low. The port number of system corresponds to the ports on the terminal board as Table 3 and Table 4.

Table 3 Corresponding signals of system input ports

System terminal No.	Terminal name	Corresponding signal	Remark
0	GX01(XORG)	Mechanical origin signal of X axis	Externally connected with mechanical, photoelectric, proximity switches, etc
1	GX02(XLM+)	Positive limit signal of X axis	Externally connected with mechanical, photoelectric, proximity switches, etc
2	GX03(XLM-)	Negative limit signal of X axis	Externally connected with mechanical, photoelectric, proximity switches, etc
3	GX04(YORG)	Mechanical origin signal of Y axis	Externally connected with mechanical, photoelectric, proximity switches, etc
4	GX05(YLM+)	Positive limit signal of Y axis	Externally connected with mechanical, photoelectric, proximity switches, etc
5	GX06(YLM-)	Negative limit signal of Y axis	Externally connected with mechanical, photoelectric, proximity switches, etc
6	GX07(ZORG)	Mechanical origin signal of Z axis	Externally connected with mechanical, photoelectric, proximity switches, etc
7	GX08(ZLM+)	Positive limit signal of Z axis	Externally connected with mechanical, photoelectric, proximity switches, etc
8	GX09(ZLM-)	Negative limit signal of Z axis	Externally connected with mechanical, photoelectric, proximity switches, etc
9	GX10	Extended input 0	
A	GX11	Extended input 1	

B	GX12	Extended input 2	
C	GX13	Extended input 3	
D	GX14	Extended input 4	
E	GX15	Extended input 5	
F	GX16	Tool calibration signal input	

Table 4 Corresponding signals of system output ports

System terminal No.	Terminal name	Corresponding signal	Remark
0	GY013(SP-)	Control port of spindle reverse rotation	Control ports of multi-section spindle gears with at most 8 gears of speed control; COM of spindle connected with GND of terminal during wiring
1	GY014(SP+)	Control port of spindle forward rotation	
2	GY15(S_2)	Output port of spindle speed gear 2	
3	GY16(S_1)	Output port of spindle speed gear 1	
4	GY17(S_0)	Output port of spindle speed gear 0	
5	GY18	Output port of workpiece cooling	
6	GY19	Output port of spindle coolant	
7	GY20	Output port of auto lubricating fluid	

Move the cursor by pressing  or  after shifting to polarity setting, and modify the value

by pressing  and . Press  to save the modification and return to the upper menu,

or press  to cancel the modification and return to the upper menu.

5.2.7. Calibration Block Thickness

The calibration block is used to quickly set the Z-zero point in the WCS. The block is placed between the tool and the material (if the Z-zero is on the top of the material,) the machine is then instructed to move down until the tool touches the block, completing a circuit. The block thickness is then accounted for and Z-zero position is set as been on top of the material.

The user can calculate the Calibration Block Thickness by doing the following:

- 1) Set the Calibration Block Thickness to 0.

- 2) Exit to the main screen.
- 3) Place the block on the table, under the tool.

- 4) Press  until Tool Calibration Mode is active. Press  +  and the tool will slowly move down until it touches the calibration block.
- 5) Remove the block. Slowly and carefully jog the machine until the cutter is touching the table. Record the value the Z axis displays, approximately 1.58 with most blocks.
- 6) This value is the Calibration Block Thickness, enter this value using the same method for modifying [spindle on/off delay]

Please refer to chapter 6.1.2 for the detailed operation method of tool calibration.

5.2.8. Backlash Compensation

Generally, spindle is fastened to the lead screw whose outer lead can't be exactly coincidental with its inner lead (it is attached to the outer lead). When spindle reverses its direction abruptly after moving towards a direction, it has to finish the lead screw backlash of the last direction, and this is called backlash error. To reach the specified processing accuracy, it is necessary to compensate this error, which is called backlash compensation. The compensation amount is set according to the actual test value.

This page includes two parameters: backlash compensation EN and backlash setting. The setup method of "compensation EN" is the same as that of "spindle on/off delay", while the setting method is the same as that of "back to mechanical origin speed". Modification to "backlash setting" is valid after system reboot.

5.2.9. Offset Setup

There are two Offset parameter, Public and Workpiece.

Workpiece offset: Sets X and Y offset relative to mechanical origin or Home position. These offsets are called by G54 to G59 commands in the g-code.

Public offset: Is used to adjust the WCS origin. Entering a value in this parameter will adjust the zero position by that value. eg X 5 will change the X zero position by positive 5.

Also, if coordinates are zeroed in the jog menu, whatever value is in this parameter will be applied to that zero. Usually these values will be zero.

The setup method of these two parameters is the same as that of "back to mechanical origin speed".

5.2.10. Tool Change Parameter Setup

The parameters related to tool change in NK105 system are all set here.

Currently the Techno NK105 configuration is manual tool change only, so these parameters are not relevant.

◆ **[Tool magazine capacity]**

Determines the number of tool holders in the system. Modification to this parameter becomes valid after system reboot.

◆ **[Current tool No.]**

It is used to select the currently tool in the spindle.

◆ **[Tool offset]**

Sets the tool offset for XYZ of the tool.

◆ **[Start tool change prompt]**

Determines if the machine will stop and create a breakpoint when a T command is encountered in

the g-code file. After entering the page, select “yes” or “no” by pressing  or .

5.3. Processing Parameter Configuration

Processing parameters mainly deal with acceleration, speed parameters and how the controller interprets Feedrates in a g-code file .

The menu level included in processing parameters configuration is as follows.

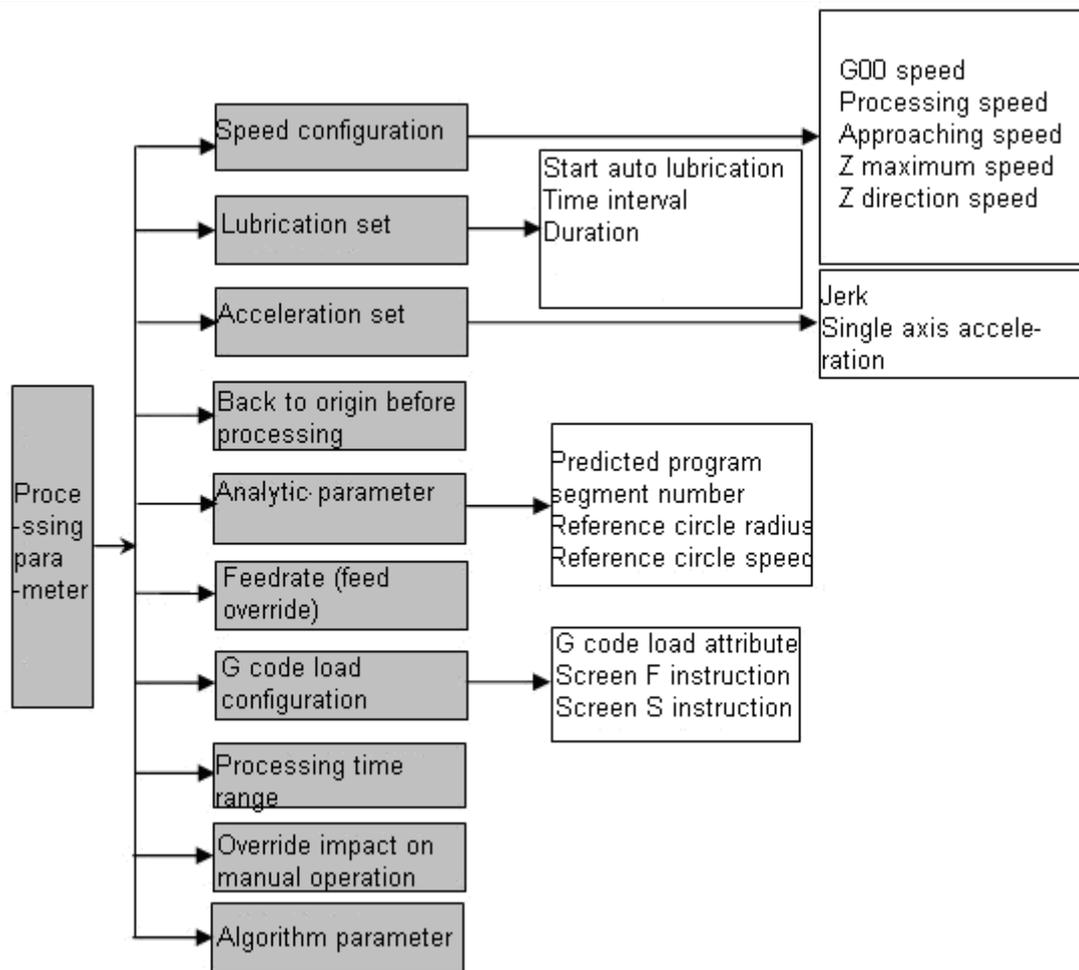


Fig. 5-8 Submenus of different levels under processing parameters configuration page

5.3.1. Speed Configuration

Speed configuration menu consists of:

Rapid Traverse Speed-This is the G00 speed when a g-code file is running.

Processing Speed- Is the speed the machine moves XY, when cutter is in the material.

Startup Speed-Is the speed the machine will be in on start-up.

Approaching speed -Is the speed the Z axis moves as it enters the material.

The adjustment of the parameters in this page is the same as that of "spindle on/ off delay".

5.3.2. Lubrication Setup

This function is not valid in the current setup.

5.3.3. Acceleration Configuration

Sets the values of jerk and single-axis acceleration. This determines the acceleration and smoothness of curves when a file is running.

5.3.4. Backing to Origin before Processing

This determines if the machine will move to the Home/Machine origin before a file will run. Generally this should be 'false'.

5.3.5. Analytic Parameter

This contains the parameters:

Forward-looking segments, reference circle radius and reference circle speed.

5.3.6. Feed Override

This sets the amount the feedrate will override when a g-code file is running.

5.3.7. G Code Load Setup

This menu has two parameters:

F instructions-If the value is 'true', default feedrates from the Speed Configuration menu are used and the feedrate in the g-code are ignored.

S instruction-If the value is 'true,' default spindle speeds are used and the values in the g-code file are ignored.

5.3.8. Processing Time Range

This menu will display the file envelope and the amount of time the file will take to run.

After entering this menu, a prompt to choose a file (from a USB flash disk or the system) will pop

up. Move cursor to select “USB file list” or “system file list” by pressing  or .

Press  to access the selected file list. And then move cursor to select a file by pressing .

or . Press  to load the file and analyze it, after which the file processing time and the maximum processing range of X, Y and Z axes will be displayed. The screen display is as shown in Fig. 5-6.

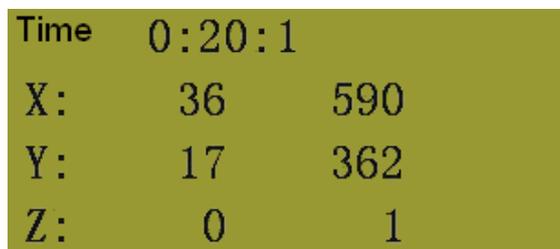


Fig. 5-9 Processing time and range page

5.3.9. OR Imp on Man Op (Manual Mode Affected by Override)

This setting determines if Jog and step speeds are influenced by Feed Override. Modification to this parameter is valid after reboot.

5.3.10. Algorithm Parameter

This activated the S type algorithm for acceleration deceleration. S type algorithm provides uniform acceleration/deceleration and allows for smooth movement on curves.

Press  to enter the modification page.

Select “yes” or “no” to modify the parameter by pressing  or .

5.4. System File Parameter

This menu provides access to system and file parameters.

File levels included in the system file parameters page is as shown in Fig. 5-7.

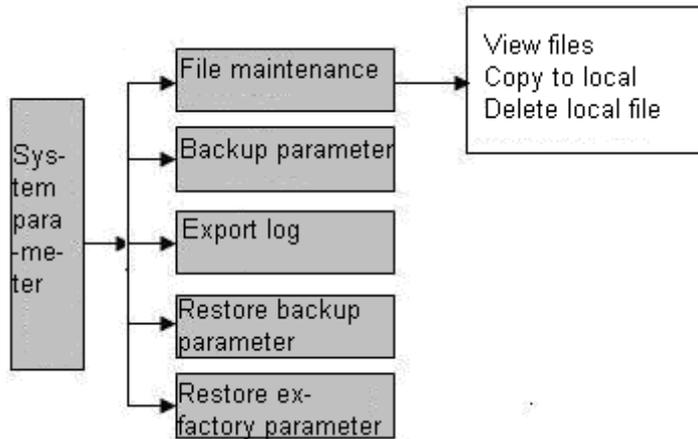


Fig. 5-10 Submenus of each different level under system file parameters page

5.4.1. File Maintenance

There are three items in this menu screen.

◆ **[View file]**

It shows G code line by line. A newline can be displayed by pressing



and

◆ **[Copy file to local]**

Will copy a file from the USB onto the controller.

◆ **[Delete local file]**

Will delete a file from the system.

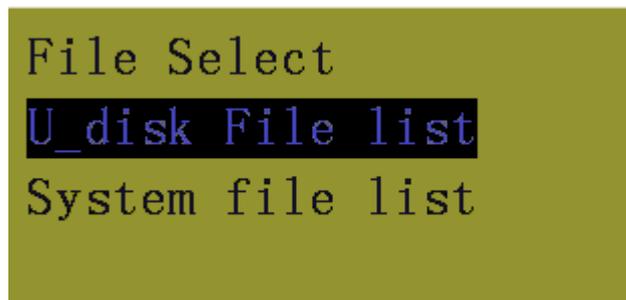


Fig. 5-11 File selection interface

After finding the desired file, press  to execute the corresponding operation. If there is no file under this path, “no processing file is found” will be displayed.

Return to the upper menu by pressing  or .



5.4.2. Backup Parameter

This will backup the system parameters.

Press  to confirm backing up the parameter.

5.4.3. Log Export

A Log.txt will be generated and exported to the USB disk.

Return to the main page by pressing  or .

5.4.4. Backup Parameter Recovery

This will restore the parameters that have been previously backed up.

Once the parameters are restored the system will require a reboot.

Restart the system by pressing  +  or restart later. Or return to the upper page by pressing  or .

5.4.5. Ex-factory Parameter Recovery

This will restore the factory parameters, this is necessary if the parameter file becomes corrupt, or after a system upgrade.

Move cursor to this option, then press  to enter.

Follow the on screen instructions and press  . Once the factory parameters are restored a reboot is required and the message in Fig. 5-9 is displayed.

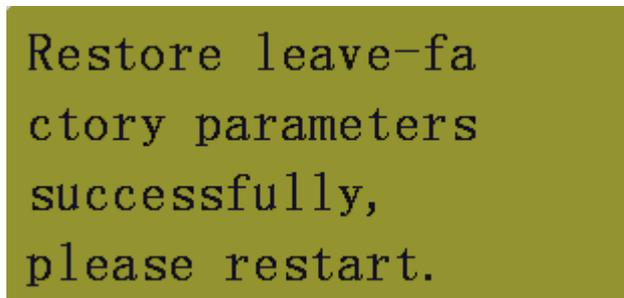


Fig. 5-12 Page of recovering ex-factory parameter successfully

Reboot the system by pressing  + , or return to the upper page by pressing   or  . Backup parameters will not be deleted by this action.

5.5. Advanced Processing Configuration

Advanced processing contains program functions

5.5.1. Stalling Setting

This setting dictates what the machine does after a g-code file has come to an end.

◆ **[Stalling method]**

Move the cursor to “stalling method” item, then press  , as shown in Fig. 5-10.

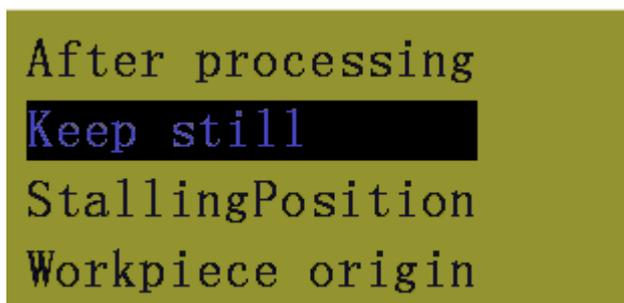


Fig. 5-13 Action after auto processing page

Move cursor to select and then press .

Keep Still-The machine will remain in the position of the last coordinate in the g-code file.

Stalling Position-The machine will move to a specified position when the G-code file ends.

Workpiece Origin-The machine will move to Xyz zero position.

◆ **[Stalling position]**

Move cursor to “stalling position” then press  to enter the “stalling position” page, as Fig.

5-11.

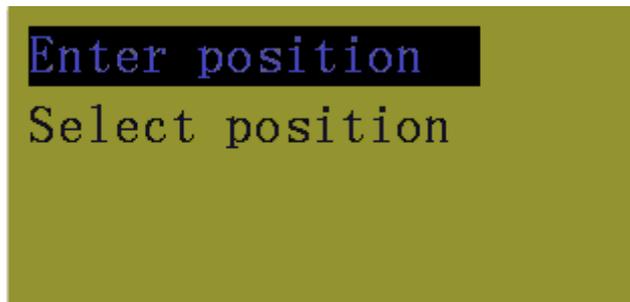


Fig. 5-14 Stalling position page

Enter position allows the user to enter coordinates in the MCS. The machine will move to this position when a g-code file is complete.

Select Position allows the user to save the location the machine is in at that point as the position the machine will move to at the end of a g-code file.

5.5.2. Rectangle Planar Milling Processing

The user can create a basic rectangle milling file from the controller. This can be used to plan a piece of material.

◆ **[Processing parameters setup]**

The following parameters must be entered.

Engraving depth: This is the step over of the tool path.

Cutting depth: This is how deep the cutter will go into the material.

Tool diameter: This is the diameter of the tool to be used.

Nose distance: This is the distance the tool lifts above the material.

Height, Width: This is the size of the rectangle to be planned.

XY start coordinate: Is where the file will start from, usually 0.

Processing manner: This determines if the tool path will be horizontal milling or vertical milling.

◆ **[File generation and loading]**

Once the parameters are selected the file will be created and loaded onto the memory from

this screen. If there is a file already loaded remove it by Pressing   or   to enter the

processing page and then unload the file by pressing  +  .

5.5.3. Region Selection Processing

This section allows the user to skip to a block of text in a g-code file.

◆ **[Processing file selection]**

Load a file from the usb or internal memory. If there is a prompt to unload the file, it is

necessary to return to processing page manually and press  +   to unload the file.

◆ **[Processing parameter setting]**

In this menu the starting line and the end line are selected.

◆ **[Start machining]**

This will start the file from the parameters entered in “processing parameter setting”

5.6. System Diagnosis

This menu is used by the manufacturer to diagnose the controller and by the user to upgrade the system.

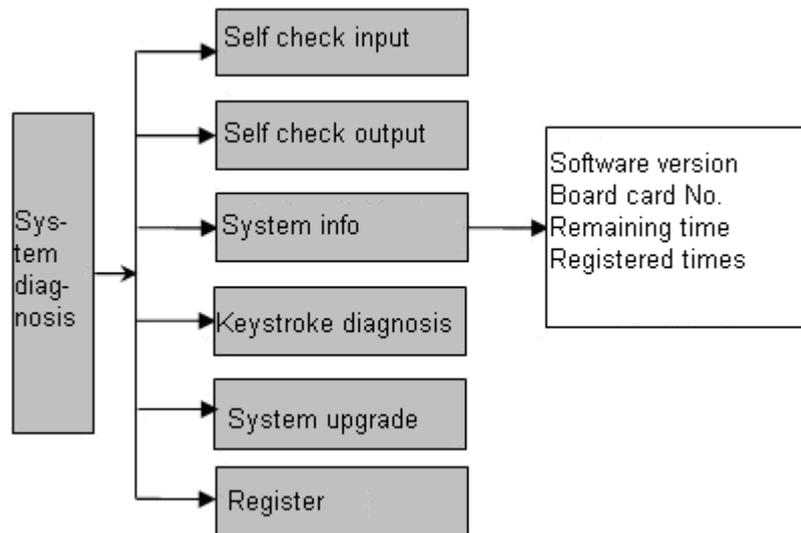


Fig. 5-15 Submenus of all different levels in system diagnosis page

5.6.1. Self-detection Input

This displays the polarity of input ports. Nothing in this menu can be modified by the user.

5.6.2. Self-detection Output

This displays the polarity of output ports. Nothing in this menu can be modified by the user.

5.6.3. System Info.

In this page, user can view system software version, control card No. and registration details.

5.6.4. Keystroke Diagnosis

This is used to diagnose faulty keys. After entering, the system will display a prompt “press the

next button”. Press any key (except  ) to show the name of the pressed key on the screen, as shown in Fig. 5-13.

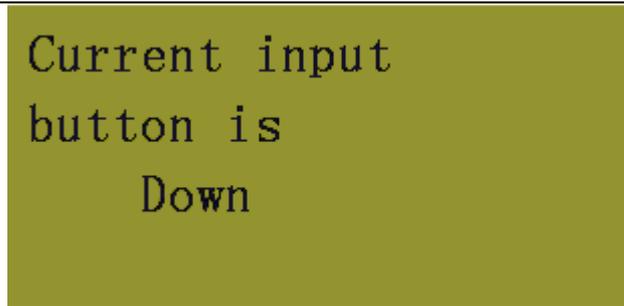


Fig. 5-16 Currently pressed button display page

Press   to exit from this page.

5.6.5. System Upgrade

Press   twice, and then the prompt “please reboot the system!” will be displayed on the

screen. Upgrade can be aborted by pressing   or  .

To upgrade the system, he can press  +  to enter the upgrade page, as shown in Fig. 5-14.

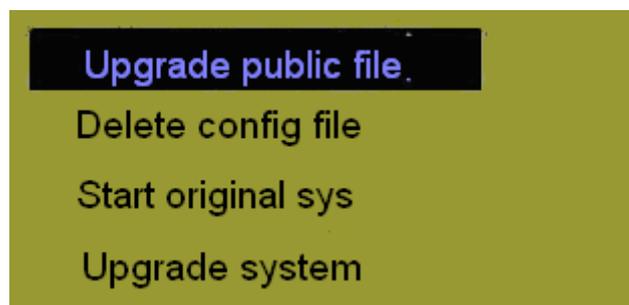


Fig. 5-17 System upgrade page

Select a corresponding operation in this page by pressing down key  or up key .

“Update public file” will upgrade the Public.dat file;

“delete config file” will delete the configuration file in BOOT, This must be done before performing “upgrade system”;

“start the original system” will start the system without upgrading it;

“upgrade system” means deleting the original system and upgrading the system with the new application file in the USB flash disk.

Please refer to chapter 2.2.3 for the operation of system upgrade.

Exit from the system upgrade page and enter start page by pressing



on “start the original system”. Or exit from the system upgrade page by pressing



after selecting “upgrade system”. Then restart the system

5.6.6. System Registration

Move cursor on this item, and then press  to enter a registration code, as shown in Fig.

5-15.

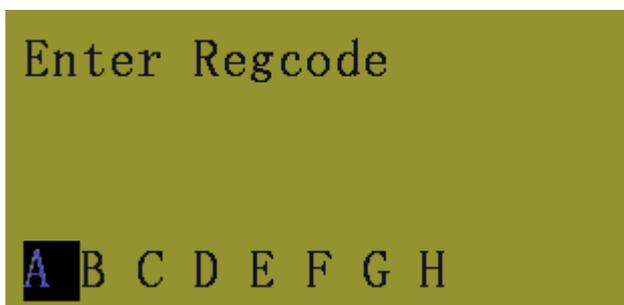


Fig. 5-18 Registration page

6. System Installation

6.1. Basic Configuration of Ncstudio™

◆ Basic configuration of NK105 system

Memory:	128M
Flash:	256M
Display:	128*64 graphic LCM

6.2. Recovery of Ncstudio™

When the NK105 unit leaves the Techno factory the correct settings are installed and the machine can be used as soon as power is attached. However if a failure occurs in the units memory, the system can be refreshed or upgraded easily

6.2.1. Configuration Update

An USB flash disk is needed to store the system image and updated data.

6.2.2. System Image Update:Upgrading Firmware.

- 1) Prepare an USB flash disk with required system image EBOOT105.nb0, NK105.nb0 and system application under its root directory. Please contact Tech Support at Techno if you do not have these files. Insert the flask disk into USB interface on NK105 control box.



- 2) Power on NK105, then press “OK + MENU” to show the update selection screen. Then press “3” to select “Image” and start updating system image. This process can take up to 3 minutes, so please be patient. Once the update is complete, the screen will display “USB device can be used now!” Press “OK” to access system update screen, and then select “delete config file”. Once the configuration file is deleted, select “system update” to start updating system application. When the update is finished, the system will start automatically.

NOTE: It is necessary to restore factory parameters after each system update. If “delete config file” has not been selected first during update, factory parameters needs to be restored after system update completes. To

restore Factory settings: after system automatic start-up, press  key to enter menu page, and then select “system param” then ”restore factory param” , follow the instructions on the screen to complete.

6.2.3. System Application Update

Updating the system image/firmware will also update the System Application, ie the interface software that is on the Handheld controller. If it is unnecessary to update system image, the system application software can be updated by the following steps:

- 1) Prepare a flash disk with the system application to be updated under its root directory. Insert it into USB interface on NK105 control box.

- 2) Power on NK105. When you enter the system interface, press  to enter menu items, and then select "system diagnosis" and then "system upgrade." Follow the prompt on the screen until the screen shows "USB device can be used now!" press "OK" to shift to system update interface, then select "delete config file." Once the file is deleted, please choose "system update" to start updating system application. When update is finished, the program will switch over to system automatically.

NOTE: It is necessary to restore factory parameters after each system update. If "delete config file" has not been selected first during update, factory parameters needs to be restored after system update completes. To

restore Factory settings: after system automatic start-up, press  key to enter menu page, and then select "system param" then "restore factory param" , follow the instructions on the screen to complete.