HD-II 2136
Tabletop CNC Router Manual

This document will provide a quick guide to the set up and operation of the Techno HD II Tabletop CNC Router equipped with the NCstudio controller.

TABLE OF CONTENTS

Safety Instructions

READ THESE INSTRUCTIONS THOROUGHLY BEFORE OPERATING MACHINE. DO NOT OPERATE MACHINE IF YOU ARE UNFAMILIAR WITH THESE SAFE OPERATING INSTRUCTIONS. DO NOT OPERATE MACHINE WITHOUT KNOWING WHERE THE EMERGENCY STOP SWITCH IS LOCATED.

WARNING: IMPROPER OR UNSAFE OPERATION OF THE MACHINE WILL RESULT IN PERSONAL INJURY AND/OR DAMAGE TO THE EQUIPMENT.

1. Keep fingers, hands, and all other objects away from machine while power is on.
2. Disconnect power to all system components when not in use, when changing accessories, and before servicing.
3. Do not loosen, remove, or adjust machine parts or cables while power is on.
4. Exercise care with machine controls and around keyboard to avoid unintentional starting.
5. Make sure voltage supplied is appropriate to specifications of components.
6. Machines must be plugged into three-pronged grounded outlets. Do not remove the grounding plug or connect into an ungrounded extension cord.
7. Keep cables and cords away from heat, oil, and sharp edges. Do not overstretch or run them under other objects or over work surfaces.
8. Use proper fixtures and clamps to secure work. Never use hands to secure work.
9. Do not attempt to exceed limits of machine.
10. Do not attempt to use machine for purposes other than what is intended.
11. Use machine only in clean, well-lit areas free from flammable liquids and excessive moisture.
12. Stay alert at all times when operating the machine.
13. Always wear safety goggles.
14. Do not wear loose-fitting clothing when operating machine. Long hair should be protected.
15. Always maintain proper balance and footing when working around the machine.
16. Maintain equipment with care. Keep cutting tools clean and sharp. Lubricate and change accessories when necessary. Cables and cords should be inspected regularly. Keep controls clean and dry.
17. Before using, check for damaged parts. An authorized service center should perform all repairs. Only identical or authorized replacement parts should be used.
18. Remove any adjusting keys and wrenches before turning machine on.
19. Do not operate the machine unattended.
20. Follow all safety instructions and processing instructions in the MSDS for the material being processed.
21. Use proper precautions with dust collection systems to prevent sparks and fire hazards.
22. Make sure to have proper fire extinguishing equipment on hand at all times.

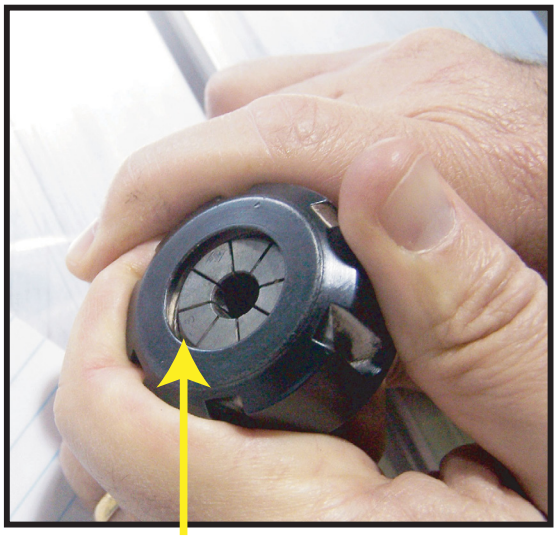
PREVENT FIRE HAZARDS by using the proper feeds, speeds, and tooling while operating your Techno machine. For example, setting feeds and speeds too low and/or using dull tool bits creates friction at the material. The friction generates heat which can result in a fire that can be drawn through the vacuum table or dust collector without warning. Fire hazard from friction heating caused by dull tools is possible when cutting certain materials, especially composite material such as wood composites, MDF and Particleboard.

WARNING!
THE SPINDLE WILL BE DAMAGED
IF UNBALANCED EQUIPMENT IS USED.

AIR SUPPLY MUST BE FILTERED AND DRY.

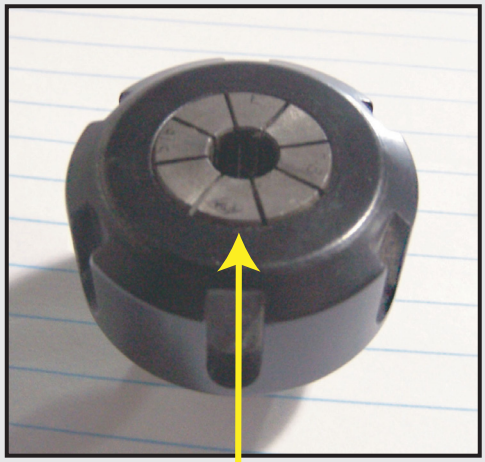
COLLETING GUIDELINES

WRONG!



This picture shows an improper assembly. Notice the gap and angle of the collet in relation to the nut. The collet is not flush to the end of the collet nut. Correct this assembly before using.

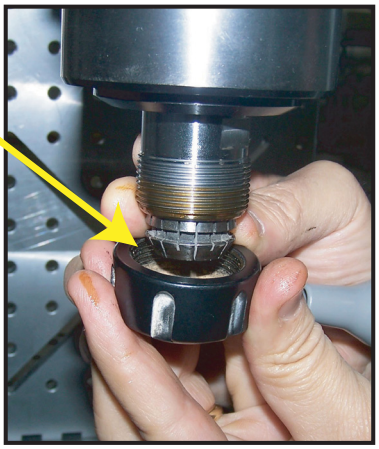
RIGHT!



The picture above is how your collet nut assembly should look: the end of the collet is flush with the bottom surface of the collet nut. You will hear and feel a "SNAP" as the collet properly goes into the collet nut. Once it is assembled, then "SCREW" the nut onto the threaded spindle end.

DO NOT PUSH THE COLLET INTO THE SPINDLE AT ANY TIME!

Only the proper assembly should be screwed onto the spindle.



FOR TOOLCHANGE AND FIXED COLLET SPINDLES:

ONLY USE TOOLHOLDERS, COLLET NUTS AND TOOLS THAT ARE BALANCED TO MEET OR EXCEED THE MAX RATED SPEED OF THE SPINDLE.

I. Techno HD II Tabletop Quick Set up

Fig. 1.1

1.1 - The electronics are housed in the controller cabinet located at the back of the machine displayed in Figure 1.1. The controller box may have to be moved or handled during the unpacking of the machine. For shipping purposes, the controller box may have been moved forward.

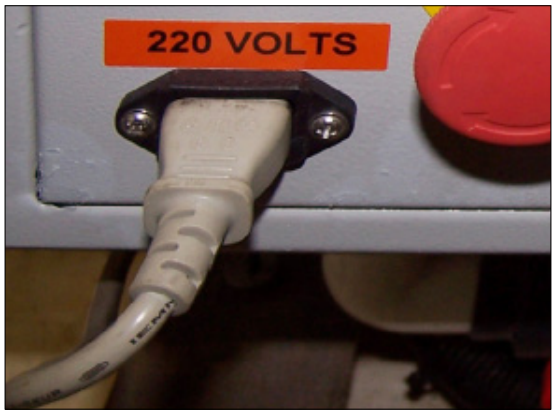


Fig. 1.2

1.2 - Have a licensed electrician connect power to the controller. The machine requires 220V single phase 15amp power in order to operate. Plug the power cable into the rear of the machine as shown in figure 1.2. The 10ft power cable supplied must be hardwired to the power source.

Connecting the Power

The cable provided will be one of two types: brown, blue, and green/yellow wire or a cable with a black, white, and green wire. (Fig 1.3)
Power is connected as follows:
Green or Green/Yellow conductor is always ground .
The two other conductors will be the hot leads.

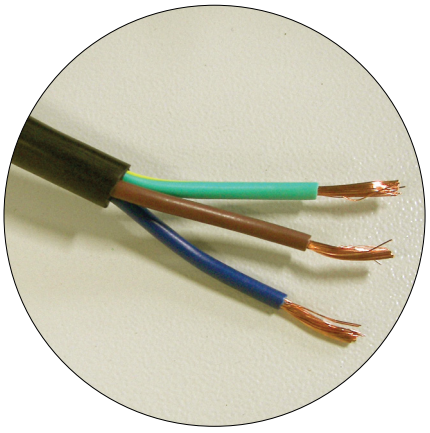


Fig. 1.3

Control Panel Functions

1.6 Enabling The Machine.
First make sure the Emergency Stop is not pressed by giving it a ¼ turn clockwise. Then, activate the machine by pressing the green POWER button. Power is now applied to the machine. The green light will illuminate.

IF EQUIPPED WITH SAFETY ENCLOSURE, CHECK E-STOP ON THE FRONT OF THE MACHINE NEAR THE DOOR!

II. HDII Tabletop Startup

When the machine first powers on, the display on the controller will light up and say "Starting System". (Fig. 2.1a)

Once the system has booted it will ask the user "Back to reference point?" Fig 2.2b

Fig. 2.1a

Fig. 2.2b

This is also known as 'homing' the machine. It refers to the process of the machine finding its mechanical home position.

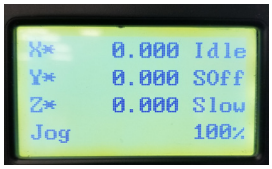
From this point, the user has two options;
Home the machine or cancel the homing process. We recommend that you home the machine every time you start up.

Pressing "OK" will initiate the homing process. The machine will first move the Z-axis to the top of travel and then the X and Y axis will move simultaneously until both are at the home / reference position at the front left corner of the machine bed.

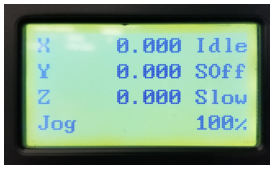
Press "ESC" will skip the homing process.
WARNING: There will be no reference position and break points, offsets and all functions that rely on a reference position will be invalid.

NOTE: The homing procedure can be aborted at anytime by pressing ESC.
Once the machine has moved to its homing position on each axis, it will stop and enter an IDLE state and will be ready to use.

Main Screens:



This picture shows that the machine ***has*** been homed due to the X* Y* Z* symbols



This picture shows that the machine ***has not*** been homed due to the X Y Z symbols (there are no *s next to the symbols)

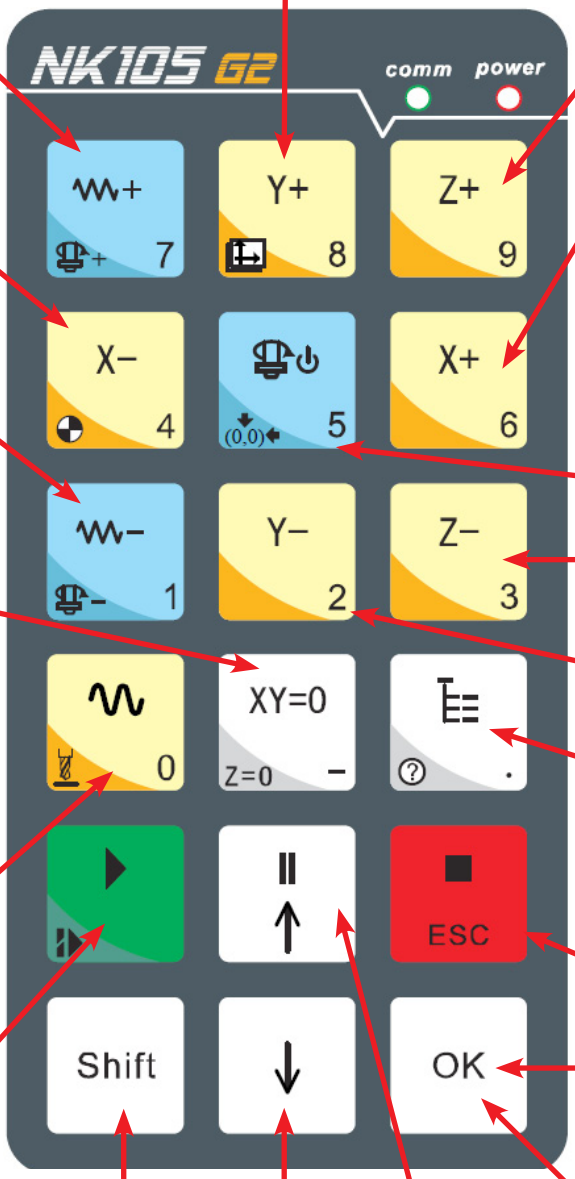
ADVANCED HOMING:
If you hit "ESC" by accident, or would like to reindicate the axes, there are two ways to "Home" again.
1) Press Menu > 3. Operations > 1. Back REF Point > 1. All Home

2) Shortcut

Single Keystroke Functions on the Handheld Pendant

Yellow Keys are movement related

Blue Keys are speed and feed related



Positive Feedrate Override (Input the number 7)

Y+ Movement (Input the number 8)

Z+ Movement (Input the number 9)

X- Movement (Input the number 4)

X+ Movement (Input the number 6)

Negative Feedrate Override (Input the number 1)

Spindle On/Off (Input the number 5)

Z- Movement (Input the number 3)

Y- Movement (Input the number 2)

Set XY Zero Position (Input a minus sign)

Menu Screen (Input a decimal point)

Jog Speed Select (Input the number 0)

Stop

Cancel

Escape

OK

Select

Start key

Shift Key

Switch between Jog and Step modes

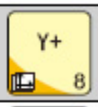
Down Directional Key

Up Directional Key

Pause

Enter Manual Parameters Screen

Change high/low speed and XYZ step increment

Key icon	Key name	Function
	Override+	Increase feedrate override; input the number 7
	Y+	Move Y axis in the positive direction (rear of table); input the number 8
	Z+	Move Z axis in the positive direction (top of travel); input the number 9
	X-	Move X axis in the negative direction (left of table); input the number 4
	Spindle ON/ OFF	Manually turn spindle on/off; input the number 5
	X+	Move X axis in the positive direction (right of table); input the number 4
	Override-	Decrease feedrate override; input the number 1
	Y-	Move Y axis in the negative direction (front of table); input the number 2
	Z-	Move Z axis in negative direction (bottom of travel); input the number 3
	Speed switchover	Toggle between manual jog high/slow speeds; input the number 0
	Clearing	Set XY=0 point (XY origin); input a minus sign
	Menu	Open the main menu; input a decimal point

Key icon	Key name	Function
	Start	Start machining
	Up	Pause machining; Up direction arrow when navigating menus
	ESC	Stop machining; Cancel; Escape
	Shift	Switch between jog and stepping modes; Auxilliary key
	Down	Down direction arrow when navigating menus
	OK	OK; Select; Open manual jog/step adjustment screen

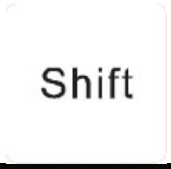
Shift Commands / Combination Keystrokes

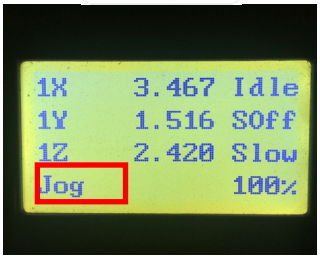
To use the shift commands, you must press and hold the shift key and then select a second key.

Key icon	Function
Shift + W+ 7	Increase spindle RPM
Shift + Y+ 8	Switch between work (relative) and machine (absolute) cooridinates **When it reads X1 Y1 Z1, that is relative. When it reads X* Y* Z*, that means absolute.**
Shift + X− 4	Go to XYZ home (mechanical origin)
Shift + 5	Go to current work origin (relative origin)
Shift + W− 1	Decrease spindle RPM
Shift + 0	Set ZO position using touch-off pad
Shift + XY=0 Z=0	Set ZO position manually
Shift +	Resume from breakpoint MO command

III. Operating Tutorials

3.0- Switching Movement to Step or Jog.





There are two modes that allow the user to control the movement of the machine: Jog and Step. To switch between these modes press the "Shift" button. The mode will be displayed on the bottom left of the screen.

Jog- Also known as continuous mode. When a directional arrow is pressed, the machine will move in that direction until the button is released.

Stepping- Also known as step mode. When a directional arrow is pressed, the machine will move an exact amount, as dictated by the manual parameters page. To move again, you must release the button and press it again.

NOTE: See section 3.3 to learn how to change jog speeds and step sizes.

3.1- Jogging the machine and changing from High/Low Jog Speed.

Select between high and slow Jog speeds

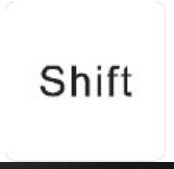
To Jog the machine, hold down one of the Yellow directional keys on the keypad while in Jog mode. The keypad has X+,X-,Y+,Y-,Z+,Z- printed on the keys to indicate direction.

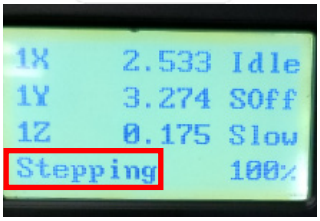
The machine has two speeds, High and Slow.
When the machine starts it will be in the Slow speed.

To toggle between low and high speed press the Jog Speed Select Button. You can only toggle speed when in Jog Mode.
The LCD will display High or Slow on the right of the screen.

Press 'OK' to change the default high and slow speeds, see section 3.3.

3.2- Stepping the machine.





To move the machine in increments, press the shift key once so that the controller indicates "stepping" . When in stepping mode, press down one of the Yellow directional keys on the keypad The keypad has X+,X-,Y+,Y-,Z+,Z- printed on the keys to indicate direction.

This will move the machine in predetermined increments for the axis selected.
By default, the X and Y axes will move in .005 inches and the Z axis will move in .001 inches.

3.3- Modifying the Jog Speed and Step Size.

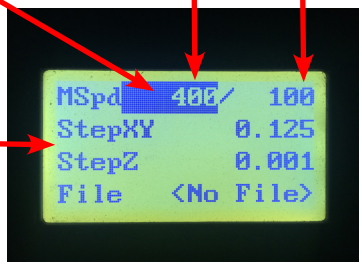
The machine can be jogged at two speeds, slow and high. You can also change the increments in which the machine will move in Step mode. These speeds are set in the Manual Parameters page.

To access the Manual Parameters page press OK from the Main Screen (Not menu)

The value being changed will be highlighted as shown

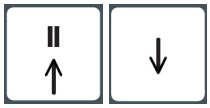
Low Speed

High Speed



Step Sizes

To move the cursor, use the Up and Down directional arrows.
Enter a new value.
Press OK to accept that value.



Set the High and Slow speed to a suitable value.
Adjust the Step value as needed.

To Exit out of this screen and return to the main menu press ESC.

WARNING:

Adjust the step size carefully. If you set the step size to an excessive value, the machine will move by that value and could damage the machine.

When inputting a decimal increment, you must enter the value as 0.### <Zero+decimal+(your increment)>

3.4- Feedrate Override.

While running a G-Code file, the user can manually override the feedrate or cutting speed of the program. The range of the override goes from 0% to 120% of the original feedrate.

Example: 50% of 100 IPM = 50 IPM

The user can override the feedrate using the following keys:

Increase Feedrate

Decrease Feedrate

DO NOT MAKE
0% OR
THE MACHINE
WILL NOT MOVE

3.5- Adjusting the XYZ Zero position/WCS/User Origin.

XYZ zero position, Working Coordinate System (WCS), and User Origin are all the same thing.

Different CAM systems and users just name the concept differently. For convenience XYZ zero position will be used in the rest of this manual.

XYZ zero position is the location point on a drawing in a CAD/CAM package where X,Y and Z all equal zero.

Generally, XY zero is on the bottom left corner and Z zero is the top of the part. In fig 3.5a the letters are located away from the XY zero, all points representing positive integers.

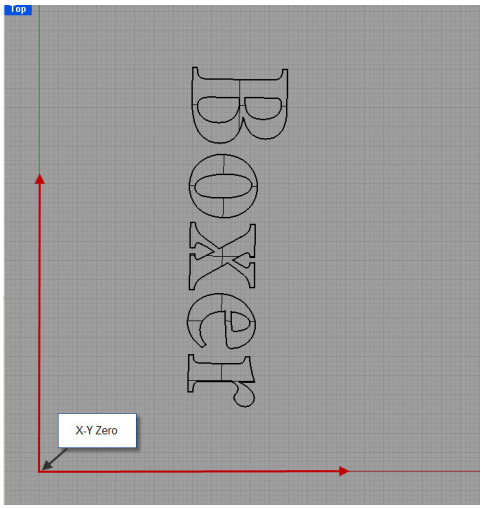
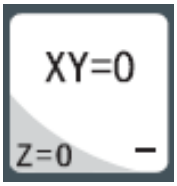


Fig. 3.5a

In Fig 3.5b the object represents the material the letters will be cut from. The machine should be jogged to the corner of the material by using the directional arrows on the keypad. Once the machine is in location press to set XY zero. The coordinates on the controller will change to 0,0.XY zero is now set.



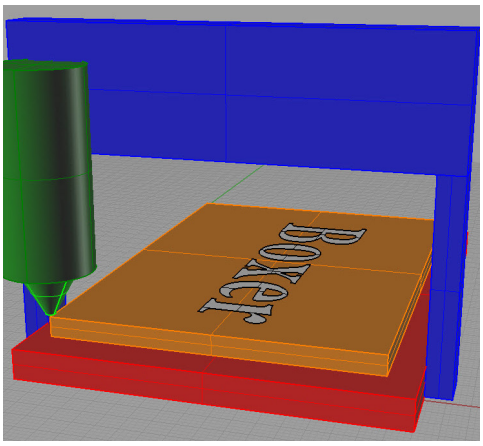


Fig.3.5b

There are two methods for setting the Z-axis zero position:

1. Manual Method: Use the Z-axis directional arrows on the keypad to move the router to the top of the material. Switch to Step Mode to slowly move the machine into position. When the router bit is in position press shift/aux and the Z=0 button as shown.

Zero Z-axis Manually

2. Tool Calibration Pad: Place the touch off pad on top of the material and under the cutter. Press shift and 0 simultaneously. The spindle will slowly move down until it touches the touchpad. The Z axis will now be set to the top of the material.

Activate Z-Touch off procedure.

The Z coordinate will now read 'Z 0.000', according to the Relative Work Origin.

3.6- Loading a G-code File.

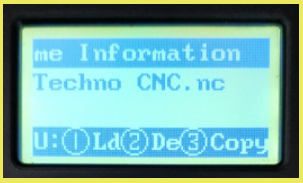
Press the Menu button. 

Select "2.USB files" to access the flash drive.
Only a G-code file with an "nc" extension with show.

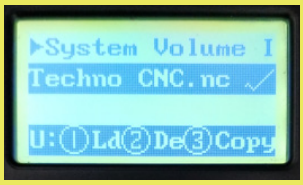
Scroll through the files with  and 

Select file by pressing OK.

Then load the file by pressing 1. 



Loading G-Code file: Before pressing OK (notice that there is no checkmark)



Loading G-Code file: Notice the check mark next to the file I want to work with. The check mark appears after pressing OK. Finally, press 1 to load file, 2 to delete the file, or 3 to copy it.

Note:
Files can be copied from this USB to the controller using the "2" button
Local disk space is limited!

Once a file is copied locally, it can also be selected from the jog speed /step size screen

3.7- Running a G-code file.

Once the XYZ origin has been set as per section 3.5 and the file has been loaded as per section 3.6, the user is ready to run the G-code file.

To run the G-code file simply press the start button 

Once the spindle has reached its programmed speed, the machine will move into position to start the first cut.

The file can be paused while running by pressing the pause button. 

To resume the file press 

To abort the file at any time press 

NOTE: When the machine pauses, the spindle will stop and the Z axis will move to the Z clearance/Safe height to allow inspection of the part.

If the machine is jogged off the part during a pause, it will lose its position and when the file is resumed it will start from the new position.

IV. Advanced Tutorials.

4.1- Alternating between Override and Programmed Feedrates.

The controller can run G-code files with speed set by the user on the keypad, override speed, or with speed set in the CAM package/G-code file, programmed speeds.

To determine what speed protocol will be used, do the following:

In the main screen, press menu  to enter the menu screen .

Use the  and  key to scroll the cursor and highlight 4. Oper Param

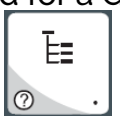
Press OK to select.

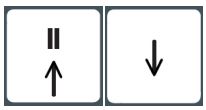
Use the  and  key scroll the cursor and highlight 8. Ignore F code / 9. Ignore S code

Press OK to select.

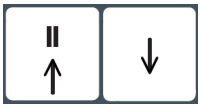
NOTE:
The F or S Option. F stands for Feed rates, and S stands for Spindle RPMs.
Highly Recommended "No" means speed in the G-code file will be obeyed.
"Yes" means speed will be overrode by the controller.

4.2 Setting the Override Speed for a G-code file.

From the main screen, press Menu  to access the Menu screen.

Use the up and down  to move the cursor and highlight 4. Oper Param

Press OK to select this option and enter the Operations Parameters screen

Use the up and down keys to move between each option + 
press enter to select the option.
Press OK to edit the data and use the number keys to enter data.
Press OK to save data and Cancel to exit out of the screen.
Keep pressing cancel until you return to the main screen. 

1. G00 Speed
2. GXX Speed

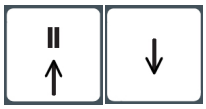
G00 Speed is the rapid speed, or the speed the machine moves when the cutter is above the material.
GXX Speed is the speed the machine moves when the cutter is in the material.

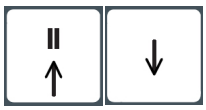
This speed will vary with cutter size, material, cutter type, etc. Great care must be taken when setting feedrates and spindle RPM, otherwise risk broken tools and loss of material.

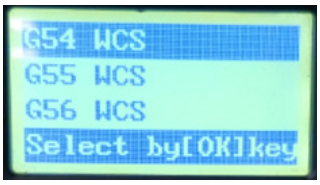
4.3 How to Use all 6 Work Coordinates

This controller allows a user to have up to 6 work coordinates saved at a time. They are labeled as G54, G55, G56, G57, G58, and G59. The controller should be automatically set for G54 (work coordinate system 1) because next to X, Y and Z on the main screen should be a number 1. In order for the controller to use another coordinate system like G55 (work coordinate system 2), there needs to be a number 2 next to X, Y, and Z. In order to change between each coordinate system go to the Main Menu, then go to the 3.Operations, then go to 6.Select WCS.

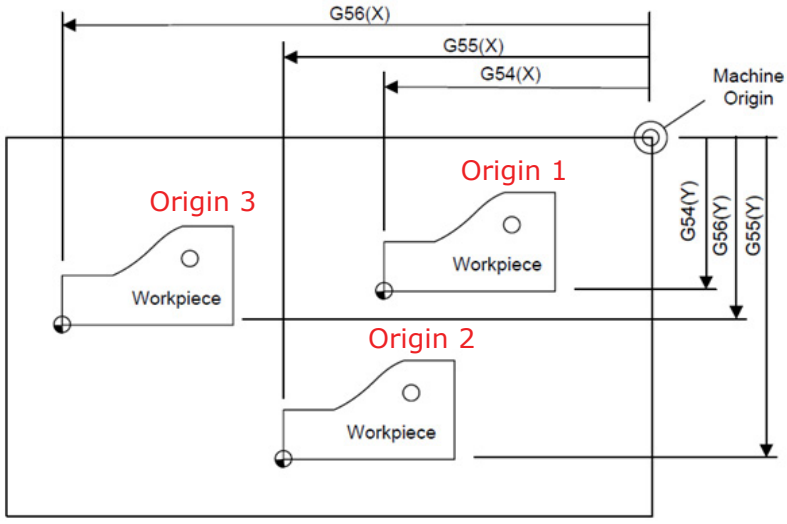
From the main screen, press Menu to access the Menu screen. 

Use the up and down  to move the cursor and highlight 3. Operations

Use the up and down  to move the cursor and highlight 6. Select WCS



Scroll through each of the WCS and press OK to the work coordinate system that needs to be used.

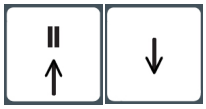


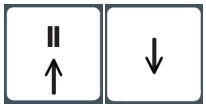
Labeled here is the multiple work coordinate origins that can be saved for certain parts or fixtures. G54 represents Origin 1, G55 represents Origin 2, and G56 represents origin 3, etc

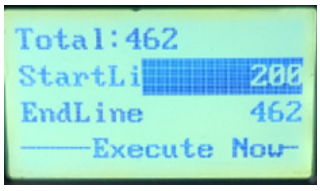
4.4 How to Use the Select Line Number Function

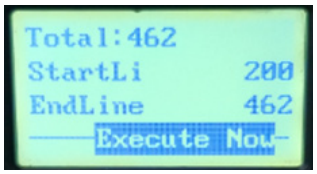
A file that is currently loaded to the machine may be ran using only certain line numbers of the G-Code if the operator chooses to do so. If the operator accidentally presses STOP, they can use this function to run from the last ran G-code line number (For ex: N100). This can be done by using the select line number function on the controller.
First, go to the Main Menu, then go to the 3.Operations, then go to 3.Select Line No., and then type in the corresponding line numbers you would like to start and stop at.

From the main screen, press Menu to access the Menu screen. 

Use the up and down  to move the cursor and highlight 3. Operations

Use the up and down  to move the cursor and highlight 3. Select Line No.





Go to the Start Line and End Line and enter in where you would like to start and end the file. The End Line will automatically be set as the last line in the G-Code unless you decide to change it. Scroll down to Execute Now and press OK for the file to run from the Start Line to the End Line.

```
200 G1X0.176Y8.369
201 G1X0.162Y8.246
202 G1X0.151Y8.123
203 G1X0.142Y8.000
204 G1X0.134Y7.876
205 G1X0.129Y7.751
206 G1X0.126Y7.626
207 G1X0.125Y7.499
208 G3X7.500Y0.125I7.3750J0.0005
209 G3X14.875Y7.499I0.0000J7.3750
210 G1X14.874Y7.624
211 G1X14.871Y7.748
212 G1X14.866Y7.871
213 G1X14.859Y7.994
214 G1X14.849Y8.116
215 G1X14.838Y8.238
216 G1X14.825Y8.359
217 G1X14.810Y8.480
218 G1X14.793Y8.601
219 G1X14.774Y8.721
220 G1X14.753Y8.840
221 G1X14.730Y8.959
222 G1X14.705Y9.077
223 G1X14.678Y9.195
224 G1X14.649Y9.312
225 G1X14.619Y9.428
226 G1X14.587Y9.544
227 G1X14.552Y9.659
228 G1X14.516Y9.773
229 G1X14.478Y9.887
230 G1X14.439Y10.000
```

```
433 G1X0.162Y14.260
434 G1X0.175Y14.138
435 G1X0.190Y14.017
436 G1X0.208Y13.897
437 G1X0.227Y13.777
438 G1X0.248Y13.657
439 G1X0.271Y13.538
440 G1X0.296Y13.420
441 G1X0.323Y13.302
442 G1X0.351Y13.185
443 G1X0.382Y13.069
444 G1X0.414Y12.953
445 G1X0.448Y12.838
446 G1X0.485Y12.723
447 G1X0.522Y12.610
448 G1X0.562Y12.497
449 G1X0.604Y12.385
450 G1X0.647Y12.274
451 G1X0.692Y12.163
452 G1X0.739Y12.054
453 G1X0.787Y11.945
454 G1X0.837Y11.837
455 G1X0.889Y11.730
456 G1X0.943Y11.624
457 G1X0.998Y11.519
458 G1X1.055Y11.415
459 G1X1.114Y11.311
460 G0Z0.750
461 G0Z0.755
462 M5
463 M30
```

If we take a look at this G-code, we can see that we will be starting from line 200 and ending the G-code file at line 463

USING THE 4TH AXIS ON THE TECHNO HD II 2136 TABLETOP MACHINES:

Note: The 4th axis on the Techno HDII 2136 Tabletop machine is not a true 4th axis. You can only use this to do "wrapping" tool paths. This means that the file is designed as a regular, flat, 3-axis file, which is scaled so that the width matches the circumference of round stock. Then, instead of cutting flat, the rotary is substituted for the X-axis and the cut follows the circumference of the stock, as if it is being "wrapped" around it.

To change from normal 3-axis operation to rotary operation, you must change some settings in the controller:
1. Press the menu button on the keypad. Go to and press OK to select "5. MFR Param". The password is 33587550.

2. Go to and press OK to select "3. Pulse Equiv". Make note of the X-axis value, it should be 0.0031250.

5. MFR Param

3. Calculate the new pulse equivalent value based on the diameter of the cylindrical stock being used through the following equation:
Rotary Pulse Equivalent = (25.4 * π * D) / 32,000
Where D is the diameter of the rotary stock in inches.

3. Pulse equiv

4. Enter the calculated value for Rotary Pulse Equivalent in the location for X under Pulse Equiv. To input a decimal number, please press 0 (zero) first, then the button for the decimal point and then the numbers.

5. Exit the menu and restart the machine. The new settings will now be applied.

6. Now jog to your starting point and set your X and Y origin. This position should be above the rotary part. Note: The X-axis will most likely move at a different speed than normal and the coordinates will not look right.

7. Flip the switch in the front of the machine into Rotary mode.

8. Run your part
To revert back to normal 3-axis operation, follow the first two steps and then put the original value, 0.0031250, into the X-axis pulse equivalent variable, then reboot the machine to apply the changes.

Notes On the G-code File

If a part requires multiple tools, it is best to output a different file for each part.

If the G-code file references a tool number higher than T10, then the controller will give an error at the start of the file. M6 T1 to M6 T10 are allowed.

In general it is best to remove T commands by telling the CAM package that the machine is not a tool changer machine, or insuring that the Tool number does not exceed 10.

G92 is the Axis presetting command, when this command is encountered in the G-code file the XYZ zero position is set at the position the machine is in at that time.

In general it is best to remove this from the G-code file, or if it is in the G-code file, make sure the machine is at the origin before you press start.

The controller will recognise G54 to G59 offset commands.

Acceleration Set

Under the menu MFR Params, there is a sub menu called Velocity.
This menu controls the acceleration and cutting motion of the machine.
The Defaults for these parameters are:

Jerk	310
Single Axis Acc	25
Max Turn Acc	100

A low Max Turn Acc will result in arcs that move in a jerky motion or at a slow speed.

High/Low Speeds and Step Distances (from main screen, press 'OK')

MSpd:	240 / 100	
Step XY:	0.005	
Step Z:	0.005	
File:	(active file name)	Note: These numbers can vary.

All following settings can be found by pressing the 'Menu' key and are worded/abbreviated as you would see them on screen.
Note: All settings with '' on screen requires reboot to take effect.*

1. LOCAL FILES
2. USB FILES
3. OPERATIONS
    1. Back to REF Point
        1. All Home
        2. Z Home
        3. X Home
        4. Y Home
    2. Rect Machining
        1. Params Setting
            Engr Depth
            0.004
            Each Depth
            0.004
            Tool Dia
            0.118
            Nose Gap
            0.079
            Height
            3.937
            Width
            3.937
            X Init
            0
            Y Init
            0
            Mode Horiz Mill
            LOAD NOW
        2. Load the Last
    3. Select Line No
        Total: 106
        Start Line: -1
        End Line: 106
        EXECUTE NOW
    4. Machining Info
        Time: 0:0:0
        X: 0.06 3.88
        Y: 0.06 3.88
        Z: -2.02 -1.82
    5. Park MCS Site
        1. Park Mode
            Not Move
            To Park Site
            To WCS Origin
        2. Park Site

1. Input Site
   Input Park Site
   X: 0
   Y: 0
   Z: -1.00
2. Select Site
   Select Current Position As
   Park Pos by [OK] Key
   Return ny [ESC] Key

6. Select WCS
   G54 WCS
   G55 WCS
   G56WCS
   G57WCS
   G58WCS
   G59WCS
   Select by [OK]
7. Array Process
   File
   Rows 2
   Columns 2
   Row Space 1.969
   Col Space 1.969
   Delay 50
   LOAD NOW
8. Origin Last
   1 X:0 Y:0 Z:0
   2 ____ , _____
   3 ____ , _____ (repeats through 8)
9. Nearby Process

4. Oper Param
   1. G00 Speed May vary
      200.00 in/min **(HD II Tabletop)**
   2. GXX Speed
      100.00 in/min **(HD II Tabletop)**
   3. Back REF First
      NO
   4. Lifts on Pause*
      1.81
   5. Offset →
      1. Public Offset
         1. X = 0
         2. Y = 0
         3. Z = 0
      2. Work Offset
         1. G54 Offset →
            1. X = 0
            2. Y = 0
            3. Z = -2.021
         G55 to G59 Offset →
            1. X = 0
            2. Y = 0
            3. Z = 4.273
   6. Cycle Process →
      1. Cycle Process
         NO
      2. Cycle Times
         2
      3. Cycle Interval

0 ms
4. S-Off in Interv
NO
7. G73-G83 Retract
0.0 inch
8. Ignore F Code
NO
9. Ignore S Code
NO
10. Spindle Stop →
1. S off at Pause*
YES
2. S off at Stop*
YES
3. S off at End
YES
11. Ratio on Manu*
YES
12. DXF Params →
1. Lifting Height
0.039
2. Process Depth
-0.039
3. 1st Point as 0*
YES
4. Shape Process*
NO
5. Bottom Process*
NO
6. Metric Size*
NO
13. ENG Params →
1. Lifting Height*
0.039
2. Tool Change Tip*
YES
3. Cycle Times*
1
4. Deep Hole Mode*
0
5. Retract Amount*
0.039
6. Select Tool No*
YES
14. PLT Params →
1. Lifting Height
0.197
2. Plt Unit
40
3. Tool Step
0.001
4. Process depth
-0.039 in
15. Tool Change →
1. ATC Capacity*

10
2. Current Tool No
1
3. Tool Offset
1. Tool 1
X: 0
Y: 0
Z: 0 *(settings repeat through tool 10)*

4. Tool Change Tip
NO
5. Cali Coordinates
1. X Cali Coor = 0
2. Y Cali Coor = 0
3. Z Cali Coor = -0.039

16. Process End Tip
NO
17. Cali Height
.512
18. ENG Unit
YES

5. MFR Param PASSWORD: 33587550
1. Velocity →
1. Decel Dist
0.394 inch
2. Approach Speed
20.00 in/min
3. Run Acc
20.00 in/sec^2
4. Dry Run Acc
20.00 in/sec^2
5. Max Turn Acc
30.00 in/sec^2
6. Jerk
300.00 in/sec^3
7. Max Speed
X = 276.00 in/min Y = 276.00 in/min Z = 118.00 in/min
8. Short Seg Spd Lmt
YES
9. SPDLMT Length
0.1 inch
10. Z Down Option
0
11. Z Plunge Cut Spd
11.8 in/min
12. REF Circle Radius
0.1 inch
13. REF Circle Speed
100 in/min
14. Jump Speed
0
15. Look Ahead IS
0
2. Axis Output Dir* →
X: Negative

Y: Negative
Z: Positive

3. Pulse Equiv* → ***DO NOT CHANGE***
   X: 0.0031250 **(HD II Tabletop)**
   Y: 0.0031250 **(HD II Tabletop)**
   Z: 0.0031250 **(HD II Tabletop)**
4. Machine Stroke → ***May Vary***
   1. Strk Upper Lmt →
      X: 23.071
      Y: 35.039
      Z: 0.100
   2. Strk Lower Lmt →
      X: 0.787
      Y: 0.000
      Z: 8.465
5. Change Stroke →
   1. Strk Upper Lmt →
      X: 15.748
      Y: 15.748
      Z: 0
   2. Strk Lower Lmt →
      X: 0
      Y: 0
      Z: -3.937
6. Ref Point Set →
   1. RefP Speed →
      X: 70 in/min
      Y: 70 in/min
      Z: 60 in/min
   2. RefPDir
      X: Negative
      Y: Negative
      Z: Positive
   3. Retract Dist
      1. X Retract Dist
         0.079 inch
      2. Y Retract Dist
         0.079 inch
      3. Z Retract Dist
         -0.079 inch
7. Spindle Set →
   1. Spindle Gears*
      7
   2. On/Off Delay
      5000 ms
   3. Initial Gear*
      6
   4. Max Spdl Speed*
      18000
8. Y Rotary Axis →
   1. Y as Rotary Axis*
      NO
   2. Rotary Y Pulse
      0.006 deg/pulse
   3. MM as Unit

NO
4. Rev Work Radius
0.394
5. Rotary Takeoff
0.291 rad/s
6. Rotary Y Acc
6.98 rad/s^2
7. Max Rotary Vel
30 r/min

9. Compensation
1. Screw Error Comp
NO
2. Enable Backlash
NO
3. Axis Backlash* →
X: 0.0
Y: 0.0
Z: 0.0
10. Calib Thickness
0.669 inch *(will vary slightly)*
11. Algorithm
YES
12. Arc Incriment
YES
13. Arc Tolerance
0.079
14. Forward Look Seg
50
15. Sign of BK REF
YES
16. Safety Height*
1.81
17. Lube →
1. Enable Auto Lube
NO
2. Time Interval
5000s
3. Duration
5s
18. G00 Feed 100%*
YES
19. Smoothing Time
0.0s
20. Corner Option
0
21. Corner Tolerance
0.004
22. Control Cycle
NO
23. Soft Limit Time
0.500
24. User Param
1. user param 2
YES

6. Param Upkeep

1. Backup Params
2. Restore Params
3. Factory Params
4. Export Params
5. Import Params
6. Import Err Data

7. System Upkeep
    1. Language
        1. Chinese
        2. English
    2. Export Log
    3. System Update
    4. Register
    5. Help
        Spec: Help Message Show Delay
        Value: -1
        Unit: s
    6. Reboot
    7. Exit
    8. Delete Log
    9. Disk Space
    10. Delete Info
    11. Modify Code
8. Diagnosis
    1. System Info
        1. Software Version NK105G3I_20_72f2
        2. Card No WHNC-0105-TD56-10C4
        3. Remaining Time Limitless
        4. Register Times 1
    2. Port List
    3. Keypress Diag
    4. Import Diag
    5. Outport Diag

V. HDII Machine Lubrication.

NOTE: AVOID A BUILD UP OF DEBRIS ON MOVING PARTS. CLEAN OFF ANY DEBRIS TO AVOID DAMAGING THE MACHINE.

The X and Y axis should be lubricated every 100 hours of use, the Z axis lubricated every 200 hours.

Before applying lubrication, clean off any debris from the machine and parts to be lubricated.

Apply oil with a clean cloth or brush.
Do not put a heavy amount of oil on the machine, just a light layer will be sufficient.

Lubricating the X Axis.

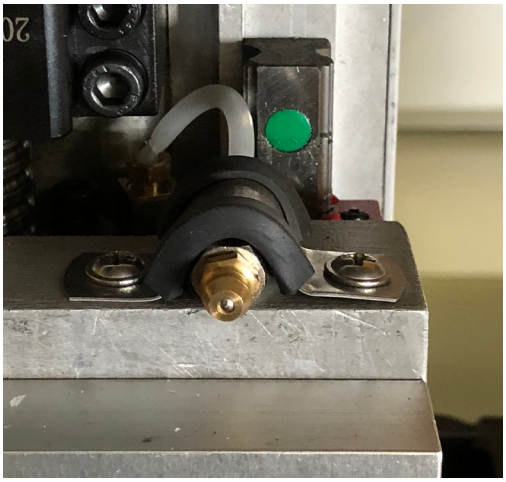
The grease fitting for the X axis ballnut is located on the rear face of the gantry head.

Lubricating the Y Axis.

The grease fitting for the Y axis ballnut is located on under the table on the gantry beam. Jog the Y axis to the middle of travel to find it.

Lubricating the ZAxis.

The grease fitting for the Z axis ballnut is located on the top of the spindle plate



Recommended Lubricants.

Oil:
Vactra No. 2(mobile)
Tonner Oil or Equivalent.
Techno Part No.
H90200-LUBE002

DAILY MAINTENANCE

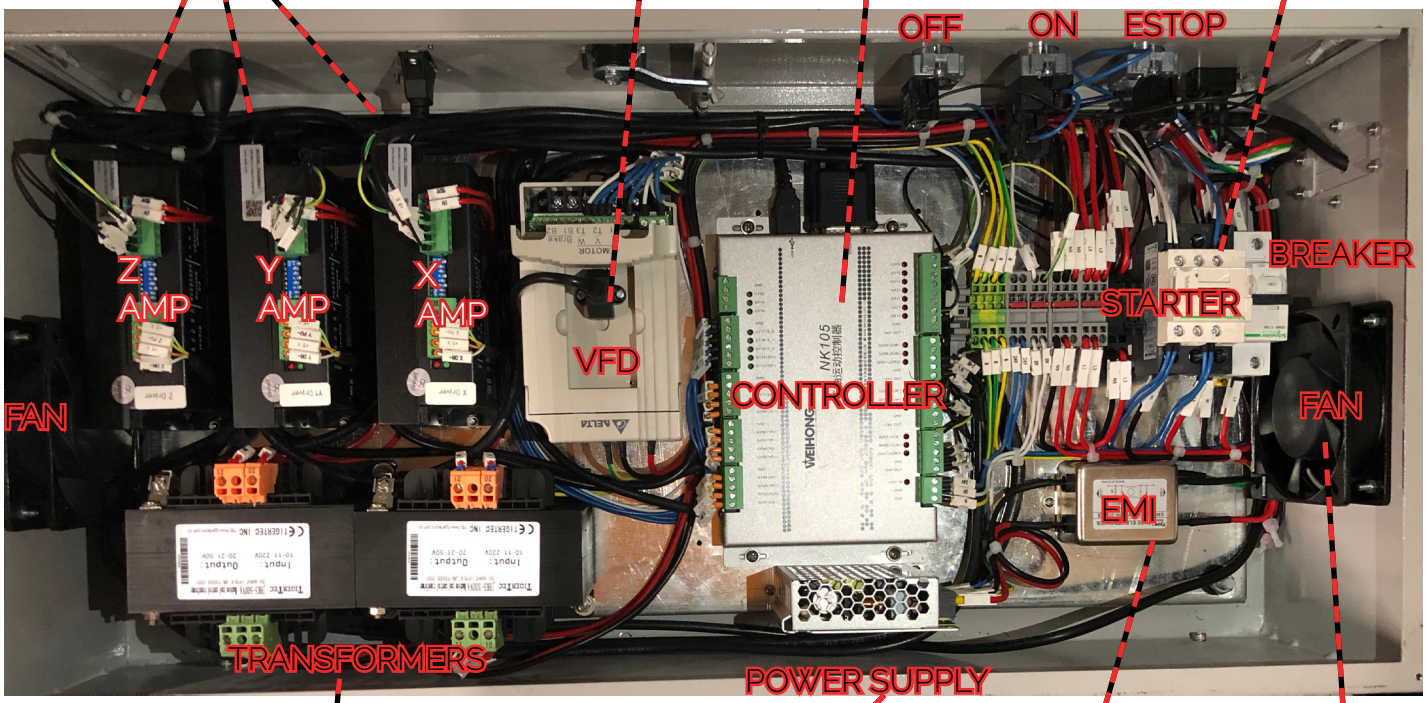
Check the machine before start up and clean it after every use.
Check to ensure chips and dust are not caught in the X, Y or Z ball screws. Dislodge all chips and thoroughly clean dust. Make sure the machine bed is clear of obstacles.

REGULAR MAINTENANCE

Generally, this maintenance should be completed every month
Moving parts on the machine may loosen or displace over time from regular operation and dust that forms during regular machine operation will often stick to the lubrication oil used on thhe moving parts of the machine (ballscrew, linear rails). This dust can cause premature wearing and damage to the precision bearings.
Inspect the hardware of each component on the machine, checking for loose bolts in case they have loosened over time. Pay special attention to linear rails, bearing blocks, spindle plates, limit switches and gantry uprights. Loose harware can result in the machine going out of square, poor accuracy and repeatbility, damage to components and broken wires.
Check to make sure that any wires are not being pinched or crushed by moving mechanical parts.
Listen for abnormal noises during operation, Grinding noises, squeals and/or banging sounds are NOT normal,
Clean grime and any accumulated lubricating oil off the guide rail and bearing blocks.
Turn on power to the machine and move it to clean and lubricate everywhere along the guide rail and rack and pinion. Lubricate ball screw assemblies monthly to ensure peak mechanical performance.
Thoroughly clean components of any excess lubrication before fresh coating.
Ball screw assemblies should be lubricated via the grease nipple using a grease gun.
Check your machine for squareness and for backlash. Out of square machines and machines with backlash are indicative of a crash or loose hardware.

V. HDII Troubleshooting

Problem	Solution
Hand held controller display is blank with no power	Ensure the machine has power and is turned on. Ensure that the 24V power supply has its green LED indicating it is on. Make sure the NK105 controller has power. Make sure the handheld conroller is plugged in. Make sure the Emergency stop switches are not pushed in. Make sure the wire harness from enclosure is plugged in.
Machine will not jog or move	If the handheld controller say “ESTOP” in the top right hand corner, the machine has moved beyond its limit. Shut down and manually push the machine off its limit switches Ensure the feedrate override is not set to 0%
Machine crashes during homing process	To confirm the limit switches are working properly, place a metal screw driver over any of the (3) axis limit switches, and the LED light on the switch itself should turn on and off. When the machine is moving, ensure its metallic striker plate is passing close enough to the switch for it to trip.
Axis motor has loud bang or grinding sound	Motor has stalled, lower speed, re-home
Spindle will not turn on	Ensure enclosure is closed. Ensure VFD display does not display error, if so, full power restart
Touchpad is no working, spindle crashes down	Ensure touchpad works by touching pad to ground on machine and showing GX16 input tripping Wire may be broken. Spindle may have worn bearings. Tool may not be suitable for touchpad


SCHNIEDER LCID25
WEIHONG NK105
LEADSHINE DMA860H
on-on-off-off-on-off-on-on
3 times
DELTA YFD015M21A
SCHENIDER ZB2-BE102C
OFF
ON
ESTOP
Z AMP
Y AMP
X AMP
VFD
CONTROLLER
BREAKER
STARTER
FAN
FAN
EMI
TRANSFORMERS
POWER SUPPLY
TIGER TECH
JBK5-500VA
220 V IN 50 V OUT
MEAN WELL WES-75-24
220V IN, +24VDC OUT 3.2A
TY212S-20
SCHNEIDER IC65N C10A

Techno CNC Systems, LLC., Terms and Conditions For Limited Warranty and Repairs Warranty

WARRANTY
All Techno CNC Systems, LLC., mechanical components are warranted against manufacturer's defects in material and workmanship for a period of one (1) year from the time of shipment from Techno CNC Systems, LLC., facilities. All Techno CNC Systems, LLC., electrical components are similarly warranted for a period of one (1) year from the time of shipment from Techno CNC Systems, LLC., facilities. Techno CNC Systems, LLC.,'s sole obligation under this warranty is limited to repairing the product or, at its option, replacing the product without additional charge, provided the item is properly returned to Techno CNC Systems, LLC., for repair as described below. The provisions of this warranty shall not apply to any product that has been subjected to tampering, abuse, improper setup or operating conditions, misuse, lack of proper maintenance, or unauthorized user adjustment. Techno CNC Systems, LLC., makes no warranty that its products are fit for any use or purpose to which they may be put by the customer, whether or not such use or purpose has been disclosed to Techno CNC Systems, LLC., in specifications or drawings previously or subsequently provided, and whether or not Techno CNC Systems, LLC.,'s products are specifically designed and/or manufactured for such a purpose. NOTE: Drive motors (servo or stepper) are considered "mechanical components".

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES EXPRESSED OR IMPLIED. ALL OTHER WARRANTIES, INCLUDING, BUT NOT LIMITED TO, ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, WHETHER EXPRESSED, IMPLIED, OR ARISING BY OPERATION OF LAW, TRADE USAGE, OR COURSE OF DEALING, ARE HEREBY DISCLAIMED. THERE ARE NO WARRANTIES THAT EXTEND BEYOND THE DESCRIPTION ON THE FACE HEREOF.

LIMITATION OF REMEDY
In no event shall Techno CNC Systems, LLC., be liable for any incidental, consequential, or special damages of any kind or nature whatsoever. Techno CNC Systems, LLC., is in no way liable for any lost profits arising from or connected to this agreement or items sold under this agreement, whether alleged to arise from breach of contract, expressed or implied warranty, or in tort, including, without limitation, negligence, failure to warn, or strict liability.

RETURN PROCEDURE
Before returning any equipment in or out of warranty, the customer must first obtain a return authorization number and packing instructions from Techno CNC Systems, LLC.,. No claim will be allowed nor credit given for products returned without such authorization. Proper packaging and insurance for transportation is solely the customer's responsibility. After approval from Techno CNC Systems, LLC., the product should be returned with a statement of the problem and transportation prepaid. If, upon examination, warranted defects exist, the product will be repaired or replaced at no charge, and shipped prepaid back to the customer. Return shipment will be by common carrier (i.e., UPS). If rapid delivery is requested by customer, then such transport is at the customer's expense. If an out-of-warranty situation exists, the customer will be notified of the repair costs immediately. At such time, the customer must issue a purchase order to cover the cost of the repair or authorize the product to be shipped back as is, at the customer's expense. In any case, a restocking charge of 20% will be charged on all items returned to stock.

FIELD SERVICE
Repairs are ordinarily done at Techno CNC Systems, LLC.,'s Ronkonkoma, New York facility, where all necessary instrumentation is available. This instrumentation is difficult to transport, so field service is severely limited, and will only be supplied at Techno CNC Systems, LLC.,'s discretion. If field service is required and is performed at Techno CNC Systems, LLC.,'s sole discretion, all relevant expenses, including transportation, travel time, subsistence costs, and the prevailing cost per hour (eight hour minimum) are the responsibility of the customer.

UNFORESEEN CIRCUMSTANCES
Techno CNC Systems, LLC., is not liable for delay or failure to perform any obligations hereunder by reason of circumstances beyond its reasonable control. These circumstances include, but are not limited to, accidents, acts of God, strikes or labor disputes, laws, rules, or regulations of any government or government agency, fires, floods, delays or failures in delivery of carriers or suppliers, shortages of materials, and any other event beyond Techno CNC Systems, LLC.,'s control.

ENTIRE AGREEMENT/GOVERNING LAW
The terms and conditions contained herein shall constitute the entire agreement concerning the terms and conditions for the limited warranty described hereunder. No oral or other representations are in effect. This Agreement shall be governed in all respects by the laws of New York State. No legal action may be taken by any party more than one (1) year after the date of purchase.

TECHNO CNC SYSTEMS, LLC., RESERVES THE RIGHT TO CHANGE DESIGNS, SPECIFICATIONS, PRICES, AND ANY APPLICABLE DOCUMENTATION WITHOUT PRIOR NOTICE.