



## HD II Tabletop CNC Router

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

The HD II Tabletop CNC Router is powered by high precision, stepper motors and controlled by a PC with a WinCNC Control System.

Techno CNC Systems, LLC. ©2017 (06/17)

**TABLE OF CONTENTS**

Safety Instructions ..... Page 3  
 Collecting Guidelines ..... Page 4

I Techno HD II Tabletop Quick Set up..... Page 5 - 7

II WinCNC Controller Overview ..... Page 8  
     2.1- WinCNC Program Operation ..... Page 9 - 29

III Operating Tutorials  
     3.1- Jogging the Machine and . ..... Page 30  
     3.2- Adjusting the XYZ Position/WCS/User Origin ..... Page 31

IV Advanced Tutorials  
     4.1- Alternating between Override and Programmed Feedrates .. Page 33  
     4.2- Setting Override speed for a G-code file..... Page 34  
     4.3- Setting the Table Size ..... Page 35  
     4.4- Command Reference ..... Page 36 - 63

V Messages and Error Codes ..... Page 64 - 69

Warranty ..... Page 70

## 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.**

© 2012

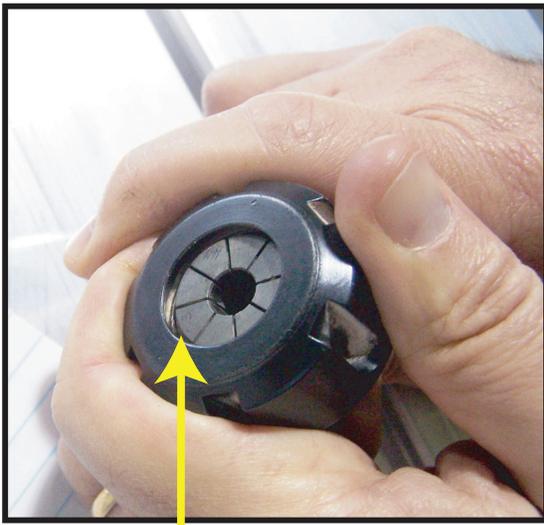


**WARNING!**  
**THE SPINDLE WILL BE DAMAGED**  
**IF UNBALANCED EQUIPMENT IS USED.**

**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.

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

Only the proper assembly should be screwed onto the spindle.



### **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.

**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 enclosure located at the back of the machine displayed in Figure 1.1. The enclosure does not need to be moved or handled during the unpacking of the machine.



**Fig. 1.2**

1.2 - Have a suitably qualified person connect power to the controller. The machine requires 220V single phase power in order to operate. Plug the power cable into the rear of the machine as shown in figure 1.2

### Connecting the Power

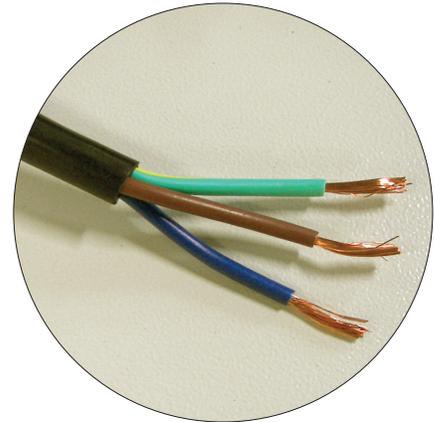
The customer is responsible to hardwire or install mating plug on supplied 10 ft. wire. You will need to supply your own plug. Have a suitable qualified person attach the correct plug in compliance with the wiring standards in your area. The machine must be connected to a 220V, single phase, 15A circuit.

The cable that we supply will be one of two types. It may either be a cable with a 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**

### Installing the Vacuum Generator



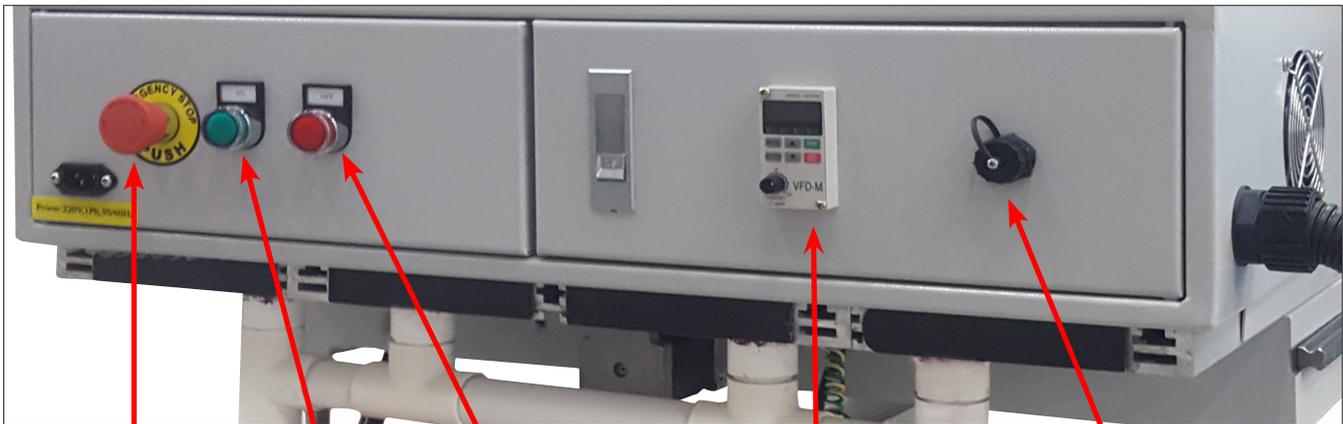
**Fig. 1.4**

To assist in the fixturing of material, the HD II Table Top is equipped with a vacuum generator that uses compressed air to produce a vacuum.

1.4 - To properly install the vacuum generator you must thread your pneumatic fitting into the female port on the unit labeled "P+" (3/8 FNPT connection). This is where you will be hooking up your air supply (80 psi @ 8CFM source). Thread the port labeled "P-" onto the vacuum fitting inside the vacuum hose on the machine. Once this process is completed, you can mount the vacuum generator to your own table top or to the one purchased from Techno using the holes provided.

**1.5** Control Panel Functions.

Figure 1.5 shows the machine control panel buttons and functions.



**Fig. 1.5**

**Emergency Stop**

**Power Enable**

**Power Disable**

**Spindle VFD Readout**

**USB Port**



**Fig. 1.6**

**1.6** Enabling The Machine.

First make sure the Emergency Stop is not depressed 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 now light up.

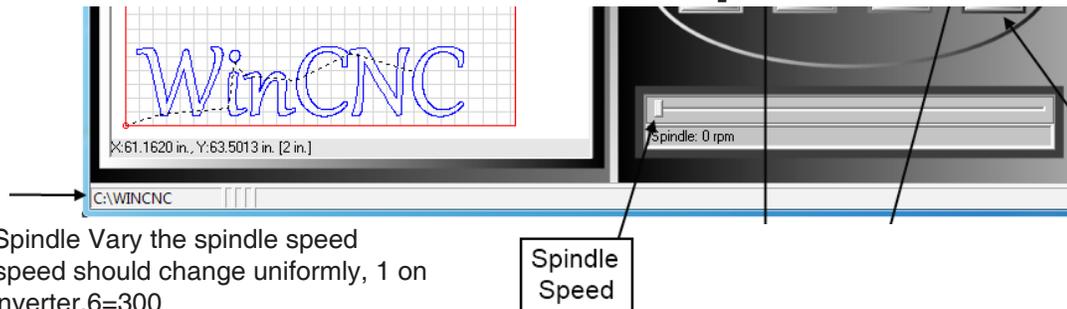
## 1.6 WinCNC Controller: Functionality

The front panel also contains the Spindle Inverter Readout.  
The number on this display multiplied by 60 gives the spindle speed in RPM.

The potentiometer and buttons on this display should NOT be touched.

The spindle is controlled by the WinCNC controller on your PC.

### Check rotation of the spindle.



Turn on the Spindle Vary the spindle speed  
The spindle speed should change uniformly, 1 on  
LCD=50 on inverter,6=300  
If looking down the spindle from above, tool should be  
turning Clockwise.

## 1.7 Start-Up/Home

### Homing the Machine

When WinCNC Controller is started the display is set to the last known position. If the machine has been moved manually or has drifted while powered down, this position will not be accurate. The G28 command must be used to home the machine. G28 moves the WZ motors up to the high limits, then moves the XY motors to their low limits. The heads are then moved away from the limits by the values specified in WINCNC.INI and each axis is set to zero. This position is Machine Zero.

It is important to use G28 to set Machine Zero since many WinCNC features are calculated from this position. Soft Limits and Boundaries cannot be used if Machine Zero is not properly set. G28 will search for the limits at 50" per minute. The "lim\_step=" setting in WINCNC.INI can help minimize the impact to the machine when stopping at the limit switches.

G28 can also be used to home only specified axis. (i.e. G28Z homes only the Z axis).

Normal machine operation using WinCNC Controller would be:

Start the program.

Enter G28. Push ENTER. (Machine Goes Home)

Enter part program to cut. Push ENTER. (Job file runs)

## II. WinCNC Controller: Overview

WinCNC Controller has advanced features to provide the smoothest possible cuts. G-Code input is constantly buffered to 'vector match' moves. This means your machine only slows down when it needs to and then only as much as needed to stay within the acceleration parameters programmed. WinCNC Controller also does S-Curve acceleration through arcs and matches arc tangential velocities with straight moves to provide smooth arc moves. The result of these features is simply smoother cuts and less need for finishing operations. WinCNC Controller has a highly customizable, user-friendly interface that lets you take control of your machine, providing features previously only found on custom industrial controllers.

The intuitive user interface is easy to learn and convenient to use. All the features you need and expect are readily accessible via industry standard G-Codes. Simple keystrokes provide the ability to start, pause, restart, jog, and rapid transit any axis without navigating multiple screens and windows. Programs may be run from any line as needed. The programmed feed rates may be adjusted from 1% to 200% without stopping machine motion.

### **Run the program by double clicking the WinCNC icon on your Windows desktop.**

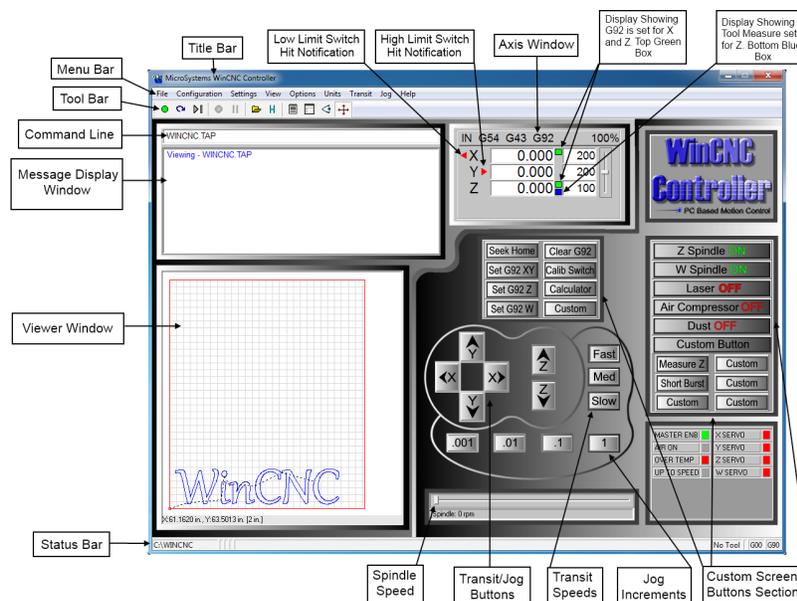
Program options may be selected from the pull down menu using the mouse, ALT+ menu key, or function keys while the machine is stopped. Files can either be typed into the command line or you can use the File menu's open option to bring the file name to the command line. Using the open command does not automatically run the file, it just brings the name to the command line.

## WinCNC Controller Program Operation

### Screen Display

The following section contains a sample of the WinCNC software display. Common components are labeled to provide clarity.

**NOTE** : This screen may differ from your screen layout, as it is highly customizable and may have been altered by the CNC machine manufacturer.



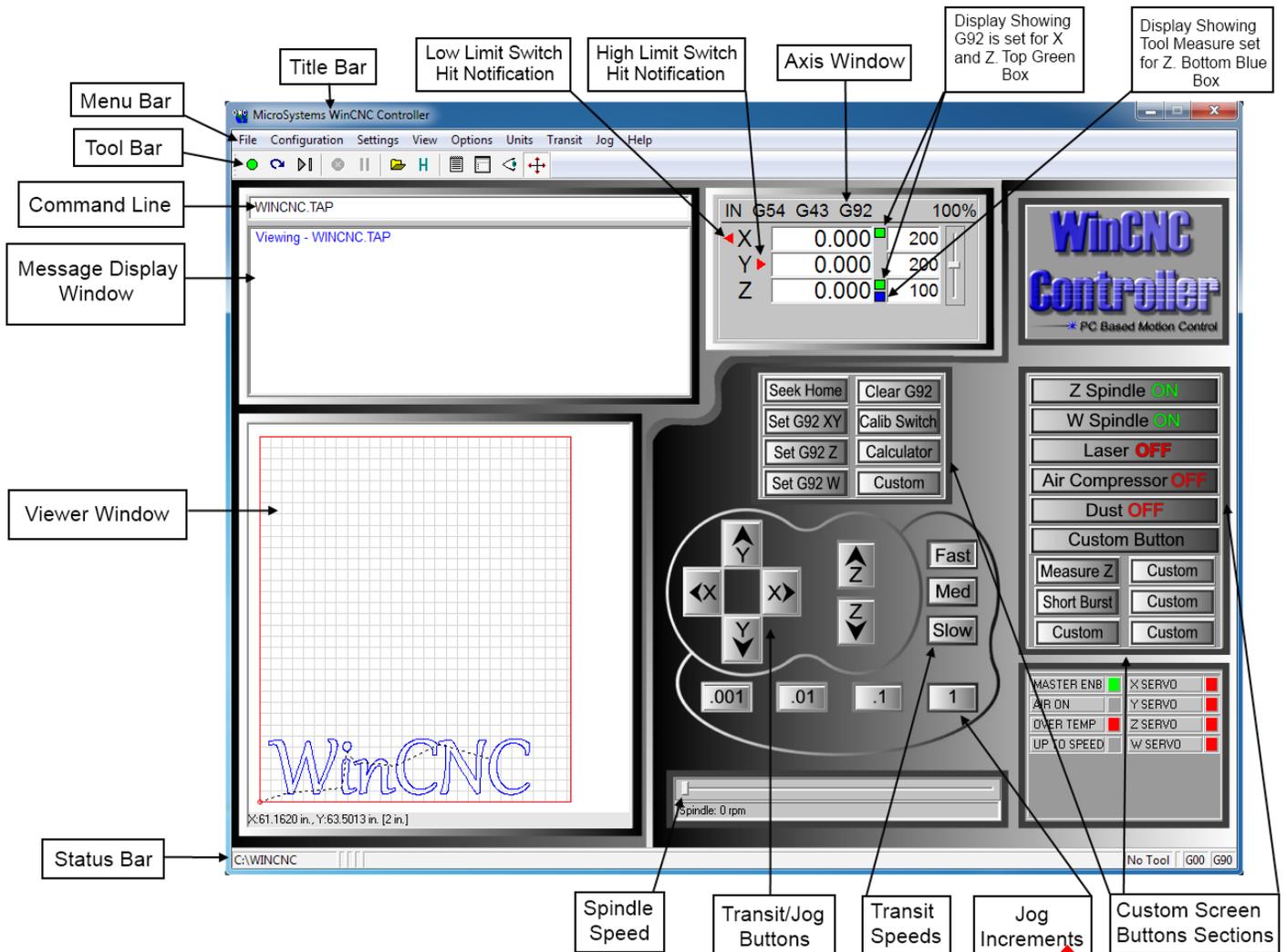
### Screen Display Component Breakdown

#### Menu Bar

The Menu Bar contains many of the main features of the WinCNC software. It is broken down into several generalized sections, as shown below. The menu bar selections are described in the following pages. If a menu option has a shortcut key, the shortcut key combination is listed in parenthesis ( ) after the name of the menu option.

## 2.1 Program Operation: Overview and Short Cuts

This section outlines the tasks required to operation your Techno HD CNC Router equipped with the WinCNC Control System



### Transit Menu

- Slow (F2) - Sets the transit speed to slow.
- Medium (F3) - Sets the transit speed to medium
- Fast (F4) - Sets the transit speed to fast.

### Keyboard (CTRL+K) -

Enable/Disable the keyboard transit/jog control used to move the machine with the keyboard.

**WinCNC Controller****Program Operation**

File Configuration Settings View Options Units Transit Jog Help

**File Menu**

**Open** (CTRL+O) - Opens a file.

**Edit** (CTRL+E) - Opens the editor. If a filename is in the command line, the editor opens that file.

**Simulate** (CTRL+S) - Simulates a file running to check for errors and estimate run time.

**View** (CTRL+V) - Views a file in the viewer window.

**Restart** (CTRL+R) - Opens the file restart dialog box to restart a file.

**Create Home File** (CTRL+H) - Creates a home file at the current machine position.

**Import => DXF** - Allows the user to import DXF (Drawing Exchange Format) file.

**Import => HPGL** - Allows the user to import HPGL (Hewlett-Packard Graphics Language) file.

\*Note: The DXF and HPGL import feature is intended for cutting simple designs. WinCNC does not compensate for tool diameter when converting these files. For more complicated design, where intelligent tool pathing is important, please use a compatible CAD/CAM program.

**Digitize => Manual** - Enable/Disable the Manual Digitizing Toolbar.

**Digitize => Automatic** - Automatic digitizing (L802, L803).

**Digitize => Frame** - Skeletal Digitizing (L810). This scanning method scans a center line or spine, and horizontal lines or ribs of the object.

**Digitize => Outline** - Creates an outline trace around an object.

## WinCNC Controller                      Program Operation

**Digitize => Laser => Run Scan** - Opens the laser digitization dialog box to set up laser scanning parameters.

**Digitize => Laser => Reprocess Data** - Opens a data file from a previous laser scan used to re-generate a 3D object with different filtering options to improve the quality of the 3D object.

**Exit** - Closes WinCNC.

### Configuration Menu

**D/A Calibration** - Calibration Settings for D/A (Digital to Analog) spindle speed control. This can only be used if D/A has been activated in the WinCNC Program.

### Settings Menu

**Positions** - Opens the positions dialog box with options to view and edit stored positions. (Ex: G53 P# to call position in G-Code)

**Home Positions** - Opens the positions dialog box with options to view and edit stored home positions. (Ex: G0 H# to call home pos.)

**Tool Positions** - Opens the tool positions dialog box with options to view and edit stored tool positions for use with automatic tool changers. (This requires an additional software feature to be enabled)

**Tool Library** - Edits and views the defined tool library.

**Local Coordinates** - View XYZ coordinates.

### View Menu

**Resolution** - View current resolution settings for each axis.

**Acceleration** - View current acceleration settings for each axis.

**Limits** - View current limit settings for each axis.

**Inputs** - Views real-time input states.

**Outputs** - Views real-time output states.

## WinCNC Controller                      Program Operation

**Toolbars** - Enable/Disable the main toolbar.

**Toolbars=> Size** - Change toolbar from small to large.

**Refresh View** (F5) - Refreshes the screen.

**Clear Screen** (CTRL+C) - Clears the message display window.

### Options Menu

**Soft Limits** (CTRL+L) - Enable/Disable the software limits.

**Keyboard** (CTRL+K) - Enable/Disable the keyboard transit/jog control used to move the machine with the keyboard.

**Buttons** - Used to customize user buttons.

**Auto Repeat** - Auto populate the command line with the last command executed.

**Auto Run** - Enable/Disable the auto run feature.

**Auto Preview** - Enable/Disable the auto preview feature.

**Single Step** (CTRL+P) - Enable/Disable single line execution of G-Code files.

### Units Menu

**Inches** - Sets the unit of measure to inches.

**Centimeters** - Sets the unit of measure to centimeters.

**Millimeters** - Sets the unit of measure to millimeters.

### Transit Menu

**Slow** (F2) - Sets the transit speed to slow.

**Medium** (F3) - Sets the transit speed to medium.

**Fast** (F4) - Sets the transit speed to fast.

## WinCNC Controller                      Program Operation

### Jog Menu

**0.001** (F6) - Sets jog increment to 0.001 units of measure.

**0.01** (F7) - Sets the jog increment to 0.01 units of measure.

**0.1** (F8) - Sets the jog increment to 0.1 units of measure.

**1** (F9) - Sets the jog increment to 1 unit of measure.

**Custom** (F10) - Allows the user to set a custom jog increment.

### Help Menu

**Help Topics** (F1) - Activates the in-program help menu.

**Update Program** - Opens the update utility.

**About** - Displays the WinCNC “About” box which contains important information about your specific software package. The About box displays your security key serial number, the software version number, your user level, the maximum number of axes, the table size limitation, and which additional features you have enabled. If you do not have a feature enabled that you would like or need, you can contact your software vendor for an upgrade.

### Tool Bar

The tool bar is a collection of shortcut buttons that perform specific actions. The toolbars in WinCNC are floating and can be positioned or docked to the user’s preference. The image below shows both the standard and manual digitizing toolbars both docked in the normal toolbar area. The additional images show each toolbar separately as a floating toolbar. The explanation of each button on the toolbar is as follows. The explanations move sequentially from left to right along the toolbar.



**WinCNC Controller****Program Operation**

**Start Motion** - This button will begin the command or job listed in the command line.

**Restart** - Allows you to pick a job file and the line to start that file on. This is useful if you want to skip over lines in a job file, or if you have aborted a job and want to start back at the point you aborted from. You can also choose to run the file in single step mode.

**Single Step** - When selected, this allows you to execute one line of a G-Code file each time you hit ENTER.

**Abort Motion** - Aborts a command or job that is running.

**Pause/Continue Motion** - Pauses/Continues a command or job that is running.

**Open File** - Opens a browse box that is used to open a job file.

**View History** - Opens the command history box, which allows a user to execute a command used previously. When an error is found in the WINCNC.INI file, the line containing the syntax error is displayed here in red when WinCNC starts.

**Edit** - Opens the default editor specified in the WINCNC.INI file. WinCNC uses Notepad by default. The editor can be used to open job files, listed in the command line in the editor.

**Simulate** - Simulates a file to check for errors and run-time.

**View** - Opens a file for viewing in the viewer window.

**Soft Limits** - Enables/Disables Softlimit features.

**Manual Digitize Buttons** - The seven buttons on the manual digitization toolbar are for use only with the Manual Digitize feature, and will only be visible after showing the manual digitize toolbar under the View->Toolbars section of the menu bar.

**Start a Manual Digitized File** - Starts a manual digitized file and enables the manual digitize mode.

**WinCNC Controller****Program Operation**

**Add a Rapid Move** - Adds a rapid move to the manual digitized file.

**Add a Feed Move** - Adds a feed move to the manual digitized file.

**Add an Arc Point** - Used to add arcs into a manual digitized file.

**Close Shape** - Used to close the last move in a shape without moving the machine.

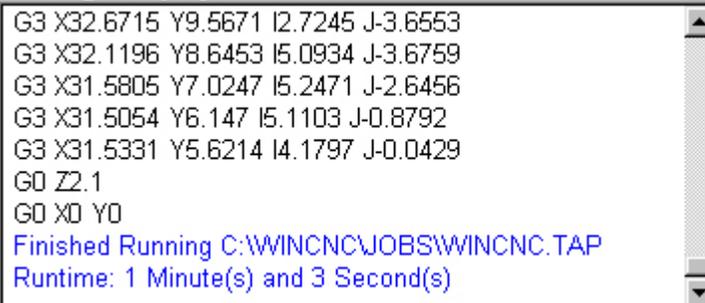
**Undo** - Used to undo previous moves.

**Save** - Allows you to save the manual digitized file in either G-Code or DXF file format.

**Command Line**

WTEST.TAP

The command line is the input line for users to enter commands or job files to be executed.

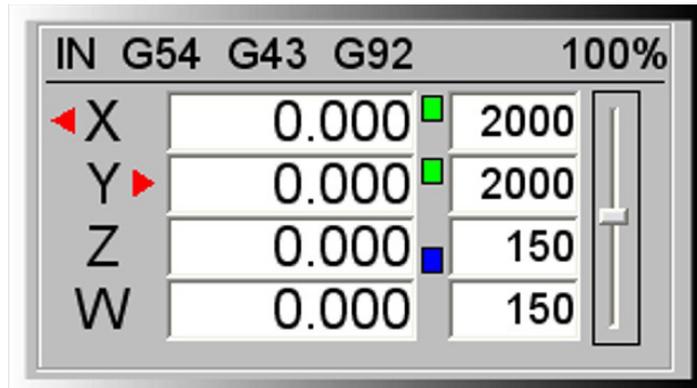
**Message Display Window**

```
G3 X32.6715 Y9.5671 I2.7245 J-3.6553
G3 X32.1196 Y8.6453 I5.0934 J-3.6759
G3 X31.5805 Y7.0247 I5.2471 J-2.6456
G3 X31.5054 Y6.147 I5.1103 J-0.8792
G3 X31.5331 Y5.6214 I4.1797 J-0.0429
G0 Z2.1
G0 X0 Y0
Finished Running C:\WINCNC\JOBS\WINCNC.TAP
Runtime: 1 Minute(s) and 3 Second(s)
```

The message display window is the main output window displaying the commands that have been executed, messages to the user, or errors that have occurred during an operation.

**WinCNC Controller**  
Axis Window

**Program Operation**



The axis window is the primary display for current information about each axis on your machine. The current positions and velocities for each axis are displayed in the text boxes. Displayed above the position and velocity boxes are the current units of measure, any currently active modes, and the current feed rate override percentage. Red triangles to the left and right of each axis label indicate low or high limit switch hits. A red triangle to the left of an axis label indicates a low limit switch hit, and to the right indicates a high limit switch hit. A green box displayed to the right of the position box indicates a temporary workspace (G92) setting, and a blue box indicates a tool length measure or workspace setting. To the far right side of the axis window is the feed rate override control slide bar.

**Transit Speed Buttons**



These buttons allow you to specify the transit speed of your machine.

Fast transit is the default.

In the `cncscrm.ini` file, change the "default speed" field to 1, for the speed you would like to be default, all others must stay 0.

Ex: for default medium speed, use the following:

`"JogMode", 0,5,35,25,7,1,"Med", "", "", -1,-1,"med.bmp"`

**Jog Increment Buttons**

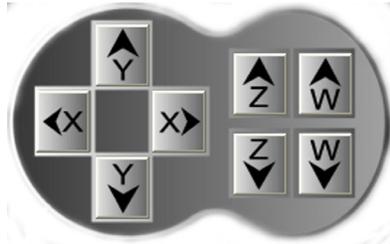


These buttons allow you to specify the increment of a jog move.

## WinCNC Controller

## Program Operation

### Transit/Jog Buttons



These buttons are what allow you to initiate a manual transit or jog movement from the console of the computer that runs the machine. In transit mode, holding the button down will continuously move the machine. In jog mode, each time a button is pressed the machine will move according to the jog increment that has been selected.

Holding the button down in jog mode will move the machine only once.



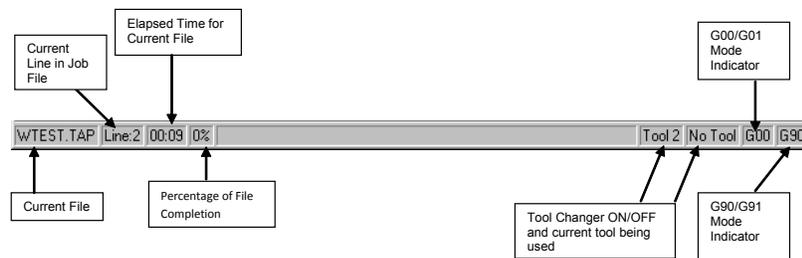
### Custom Screen Buttons Section

This section of controls is a customizable button section that allows the user to place shortcut buttons to activate heavily used commands.

**Note:** See Section 4 Screen Configuration

### Status Bar

The status bar is the label along the bottom of the WinCNC window that looks similar to the one shown below. This bar provides the user with the status of several features in WinCNC. Each section in the example below is labeled for clarity.



\* Note: There are TWO tool changer indicators

**WinCNC Controller** **Program Operation**

Shortcut Keys

<b><u>File Menu Shortcuts</u></b>	
Open	Enter (with blank command line), CTRL+O
Edit	CTRL+E
Simulate	CTRL+S
View	CTRL+V
Create Home File	CTRL+H
<b><u>View Menu Shortcuts</u></b>	
Refresh View	F5
Clear Messages	CTRL+C
<b><u>Options Menu Shortcuts</u></b>	
Keyboard	CTRL+K
Soft Limit	CTRL+L
<b><u>Transit Menu Shortcuts</u></b>	
Slow	F2
Medium	F3
Fast	F4
<b><u>Jog Menu Shortcuts</u></b>	
.001	F6
.01	F7
.1	F8
1	F9
Custom	F10
<b><u>Help Menu Shortcuts</u></b>	
Help	F1

**WinCNC Controller**

**Program Operation**

Shortcut Keys continued

<b><u>Tool Bar Shortcuts</u></b>	
ESC	Aborts the current file or command
SPACE	Pauses a file or command
ENTER	Starts/restarts a file or command
TAB	Opens the command history box
CTRL+R	Opens the restart file box
<b><u>Feed Rate Override Shortcuts</u></b>	
INSERT	Increases override rate
DELETE	Decreases override rate
CTRL+either	Resets feed rate to 100%. No override settings
<b><u>Manual Digitize Shortcuts</u></b>	
After opening the Manual Digitize toolbar and clicking the green start button, the following shortcut key combinations become active for adding moves to the digitized file.	
Add Rapid Move	CTRL+J
Add Feed Move	CTRL+F
Add Arc Midpoint/Endpoint	CTRL+A
Undo Last Move	CTRL+U
Close Current Shape	CTRL+D
Save	CTRL+W
<b><u>Spindle Speed and Laser Power Shortcuts</u></b>	
Spindle Speed Alone or Laser Power Alone	+ - (plus and minus)
Spindle Speed and Laser Power Together	Laser Power uses + - Spindle Speed uses Shift + -
CTRL+(Function Key)	See "ctrlf#=" in configuration settings

**WinCNC Controller**

**Program Operation**

**Limit Switches**

The default channel settings will normally provide proper interfacing with the machine's limit switches. Before attempting to move the machine, verify that all limit switches are functioning properly. Limit switch status is displayed in the position window using a red triangular indicator to the left or right of the axis label.

IN G54		100%		IN G54		100%	
◀ X	0.000	200		X	0.000	200	
Y	0.000	200		Y ▶	0.000	200	
Z	0.000	50		Z	0.000	50	
W	0.000	50		W	0.000	50	

The indicator to the left of the axis label shows low limit status for that axis. The indicator to the right of the label shows high limit status. If the indicator is present then the limit switch is closed, otherwise it is open. The lim\_mode setting is used to control how WinCNC responds to limit switch triggers.

First toggle your limit switches by hand. Verify that the appropriate limit display toggles. If it does not toggle then you need to adjust WINCNC.INI. See the WINCNC.INI reference sections of the manual if the default settings are incorrect for your machine.

It is recommended that you verify limit switch operation at the beginning of each session. Do not attempt movement until the limit switch displays toggle correctly. Once the limits are working try a small move. If nothing moves, the wrong axis moves, or movement is in the wrong direction then adjust WINCNC.INI.

Perhaps the easiest way to do preliminary testing of movement is with the Jog mode. Select "Jog .1", and select the KEYBOARD option under the OPTIONS menu. If you are using the default jog key configuration, use the LEFT and RIGHT arrow keys to move X, the UP and DOWN arrows to move Y, the PAGEUP and PAGEDOWN keys to move Z, and the HOME and END keys to move U. Otherwise use the buttons that are defined for your system.

## WinCNC Controller

## Program Operation

### Homing the Machine

When WinCNC Controller is started the display is set to the last known position. If the machine has been moved manually or has drifted while powered down, this position will not be accurate. The G28 command must be used to home the machine. G28 moves the WZ motors up to the high limits, then moves the XY motors to their low limits. The heads are then moved away from the limits by the values specified in WINCNC.INI and each axis is set to zero. This position is Machine Zero.

It is important to use G28 to set Machine Zero since many WinCNC features are calculated from this position. Soft Limits and Boundaries cannot be used if Machine Zero is not properly set. G28 will search for the limits at 50" per minute. The "lim\_step=" setting in WINCNC.INI can help minimize the impact to the machine when stopping at the limit switches.

G28 can also be used to home only specified axis. (i.e. G28Z homes only the Z axis).

Normal machine operation using WinCNC Controller would be:

Start the program.

Enter G28. Push ENTER. (Machine Goes Home)

Enter part program to cut. Push ENTER. (Job file runs)

### Feed Rate Override

WinCNC accepts feed overrides to increase/decrease the feed rate. The feed rate can be adjusted from 1% to 200% of the programmed rate. The override rate cannot be set higher than the max velocity or G0 rate for a given axis. Use the Insert key to increase and the Delete key to decrease the programmed feed rate, holding CTRL and pressing either Insert or Delete will result in the feed rate being reset to the programmed value. There is also a slider bar to the right of the feed rate screen display that can be used to initiate a feed rate override. The override feed rates will be displayed in place of the programmed feed rates in the display. The feed rate will change colors to depict that an override is in use. If the override rate is higher than the programmed rate the color of the feed rate display will be red, if it is lower than the programmed feed rate it will be blue.

**WinCNC Controller****Program Operation****Coordinate System**

The Machine Zero (MZ) set by homing the machine becomes the anchor point for all positions specified in subsequent G-Code commands. This is also considered absolute zero, not taking into account any temporary homes (G92's) or tool measures.

Local Zeros (LZ) are set using the G92, G92.1 and G92.2 commands.

The position display box displays the current axis coordinates and the axis coordinate modes. The axis coordinate mode is displayed to the right of the axis position.

A green box indicates that the axis has a Local Zero applied. (G92, or temporary home).

A blue box on a horizontal axis indicates that a workspace other than G54 is in effect. (G55, G56, G57: ex: using W axis instead of Z).

A blue box on a vertical axis indicates that tool length measure is in effect.

The coordinate modes show how the axis values in a given G-Code command will be interpreted.

MZ Coordinates - values are relative to MZ. (Or absolute zero)

G92 Coordinates - values are relative to LZ specified by G92  
(Ex: enter the command G1 X10 Y10, then enter. Next type G92, then enter. X & Y will now show as x0, Y0)

G55, G56, G57 Coordinates for horizontal axes - horizontal axes values are offset for vertical axes heads.

G43 Coordinates for vertical axes - vertical axes values are offset for M37 tool measure values.

G92 and G55, G56, G57, G43 - values are offset for both.

**WinCNC Controller****Program Operation****G91 Mode**

G91 mode is called relative or incremental mode. In G91 mode values specify distances. For instance if the X position is currently 20 and G0 X8 is specified the machine would move +8 units of measure in X to X28.

A G-Code program written in G91 mode may be run from any position. Since the moves are relative, the starting position does not matter. The program will run properly from any starting position. However, to get the proper results, the machine must be set to the proper position in relation to the workpiece before the piece is run.

The axis coordinate status does not have any effect on how the program runs. It does however affect the coordinate display while the program runs.

**G90 Mode**

G90 mode is called absolute mode. In G90 mode values specify positions. If the X position is 20 and G0 X8 is specified the machine would move -12 units of measure in X to X8.

In G90 mode the situation is different. The initial of the machine position will not affect where the part runs since the first move will go to the absolute position specified on the table. Instead of setting the machine position properly before running a G90 program.

The G92 commands are used to change the coordinate system so that any position may be temporarily made to be Local Zero (LZ).

Most G90 programs are written relative to a starting position of X0Y0 with Z0W0 being either the bottom or top of the workpiece. All that is then necessary to run the program is to insure that the current LZ is set to match the program before running. There are several ways to do this.

The machine may be positioned to the proper position and the G92 command used to set the proper coordinates for running.

**WinCNC Controller**

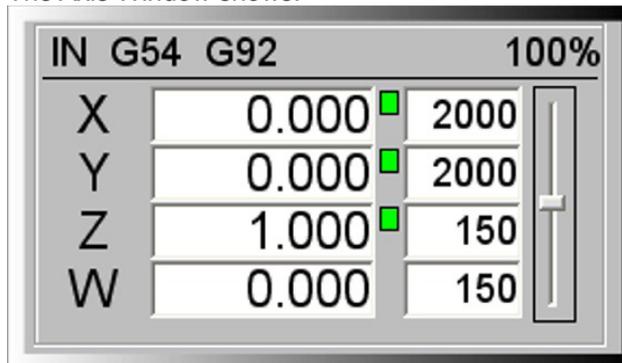
**Program Operation**

**G90 Mode continued**

Example: The workpiece is fixtured at X20Y20, is 1" thick and Z0 in the part file refers to the top of the workpiece.

G90	Commands in G90 Mode
G92	Turn off any current G92s
G0Z2	Set Z 2" above the table and 1" above the workpiece
G0X20Y20	Go to corner of workpiece
G92X0Y0Z1	Set this position to be 1" above LZ

The Axis Window shows:



Alternately LZ may be set by shifting MZ the desired amount using G92.1. Using G92.1 it is not necessary to move to the workpiece first.

**Example**

G90	Commands in G90 mode
G92.1 X20Y20Z1	Move LZ 20" in XY and 1" above the table

The coordinate display shows the current position relative to the workpiece.

When writing a G-Code program there must always be a starting point. Typically it would be the lower left corner (in XY) of the piece to be machined and either the upper or lower surface of the workpiece or a position a known distance above the workpiece (ZW). In a G91 mode program this point does not need to be given a coordinate value.

## WinCNC Controller                      Program Operation

In G90 mode this point is assigned a coordinate value (usually 0,0,0) and all positions are specified relative to this starting point.

When running the program the machine is positioned to this starting point and G92 is used to set position to the start position for the program. Alternately G92.1 may be used to shift MZ.

### Workspace

The G54/G55/G56/G57 functions are used to allow for easy switching between workspace coordinates. Each G54/G55/G56/G57 line you wish to use must be specified in the WINCNC.INI file.

**WINCNC.INI Setup:** For descriptions see the Systems Configuration Section of the manual.

G54=X# Y# T# A# R# O#

After configuring the G54/G55/G56/G57, the G54/G55/G56/G57 commands can be used to switch workspace coordinates, or for rotating a rotational axis.

If using the type 0 G54/G55/G56/G57 for switching vertical heads just use the G54/G55/G56/G57 commands alone to switch workspace coordinates.

If using the type 3 G54/G55/G56/G57 for rotational axis command G54/G55/G56/G57 X# Y#, where X and Y are the positions to rotate towards.

### Head Swap

L12 provides a means of cutting with a head not specified within a program and of using multiple heads simultaneously.

Example:

L12WZ moves Z whenever W is specified and moves W when Z is specified.

L12ZZ moves Z and W together whenever Z is specified.

L12 alone or L12ZW sets normal operation.

**WinCNC Controller**                      **Program Operation**

**Tool Length Offset**

M37 is used to measure tool length and set ZWUV workspace.

By using M37 it is not necessary to measure tool length for every workpiece thickness change. Using M37 in combination with Soft Limits and Boundaries also provides a means of protecting the table from being routed accidentally. After M37 is set G28 will set the ZW to the actual position of the tool tip above the table.

Example: (without automatic tool changer)  
Move the Z head tool tip to the table.  
Command M37 Z0.

M37 turns on G43 mode, indicating that tool length offsets are active. Tool length offsets can be disabled using G49 and re-enabled with G43 without re-measuring.

**Soft Limit / Boundaries**

Soft Limits and Boundaries are used to keep programs and command from moving into the limit switches. Values must be set in WINCNC.INI before using.

**Soft Limits** check absolute position command against the limit switch positions. If a position is commanded that would result in a limit switch being hit, a limit error will be displayed and the move will not be run.

lolim=X# Y#	Distance from MZ (machine zero) to XY low limit switches less .1"
hilim=X# Y# Z# W#	Distance from MZ (machine zero) to XYZW high limit switches less .1"

ZW low limits normally are not specified since desired positions vary with tool length.

**WinCNC Controller**

**Program Operation**

**Boundaries** check workspace positions. In G54 mode the Z head will not be allowed to leave the table. In G55 mode the Z head might move off the table but the W head will be kept on the table.

lobound=X# Y# Z# W#	Distance from MZ to XY table edge and top
hibound=X# Y#	Distance from MZ to XY table edge

ZW high boundaries normally are not specified since desired positions vary with tool length.

ZW low boundary checking is available if M37 is used. Use M37 to set Z0W0 to the tabletop. The ZW hi limits and low boundaries are then both set to 0. This allows movement between the limits and the table.

Limits and Boundaries may be enabled or disabled in the Settings Menu and in WINCNC.INI using softlim=0 or 1 and boundary=0 or 1.

**Repeat Command**

The command or file name history may be viewed by pushing the TAB key. To repeat a command from the history list select the desired command or file name using the mouse, and then click OK.

**Simulate**

WinCNC supports the simulation of files to check for errors, estimate runtime, and check min/max positions. To use the simulate function of WinCNC make sure the file name is typed into the command line and instead of running the file, you will use the simulate call to simulate the file. This is done by using either the Simulate command found under the File menu, by pressing CTRL+S, or by pressing the simulate button on the toolbar. Once you initiate the simulate feature, the software will display a screen showing the specifics of the file simulated which includes the starting (x,y) positions, the ending (x,y) positions, the min and max (x,y) positions, the number of lines ran in the file, and the total time it will take to run the file.

**NOTICE:** The time function in the simulation feature does not take into account smoothing. If you have smoothing set up on your machine, the time it takes for the file to run should be less than what is displayed in the simulation data.

## WinCNC Controller

## Program Operation

### Viewer

WinCNC contains a built in viewer window that will allow you to display the output of G-Code files before the job is actually ran. The viewer can also display a G-Code file line by line as it is run by WinCNC. When the file is displayed in the viewer you can distinguish G0 moves from G1 moves by the color of the line. By default, G0 moves are shown as a black dashed line. G1 moves are displayed as a solid blue line. See the Screen Configuration section if a different color configuration is desired.

You can also re-center and zoom in/out of the display within the viewer. To view a file before you run it, enter the file name into the command line and then press CTRL+V, select View from the File menu, or press the viewer button on the toolbar. The file will be displayed line by line automatically when you run the file. Once the object is loaded into the viewer the following controls can be used:

Zoom In Incrementally - click the left mouse button.

Zoom Out Incrementally - click the right mouse button.

Selected Zoom - hold down the left mouse button and drag the box around the area to zoom in on.

Pan or Re-Center Object - hold CTRL and click either mouse button.

Reset Image - hold shift and click either mouse button.

To set up the viewer window you must include lines in both the WINCNC.INI and the CNCSCRN.INI files.

WINCNC.INI Setup (for descriptions see the System Configuration section of the manual)

table=X# Y# W# H# B#

CNCSCRN.INI Setup (for descriptions see the Screen Configuration section of the manual)

“Display”, “Viewer”, 370, 150, 270, 270

### III. Operating Tutorials

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

**Start Motion** - This button will begin the command or job listed in the command line.

**Restart** - Allows you to pick a job file and the line to start that file on. This is useful if you want to skip over lines in a job file, or if you have aborted a job and want to start back at the point you aborted

#### Jog Increment Buttons

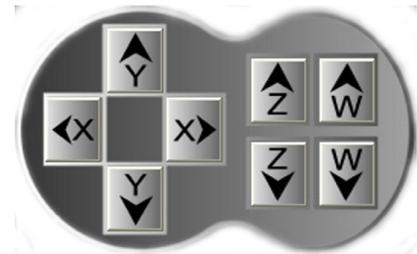


#### Jog Menu

- 0.001 (F6) - Sets jog increment to 0.001 units of measure.
- 0.01 (F7) - Sets the jog increment to 0.01 units of measure.
- 0.1 (F8) - Sets the jog increment to 0.1 units of measure.
- 1 (F9) - Sets the jog increment to 1 unit of measure.
- Custom (F10) - Allows the user to set a custom jog increment.

#### Transit/Jog Buttons

These buttons are what allow you to initiate a manual transit or jog movement from the console of the computer that runs the machine. In transit mode, holding the button down will continuously move the machine. In jog mode, each time a button is pressed the machine will move according to the jog increment that has been selected. Holding the button down in jog mode will move the machine only once.



**Single Step** - When selected, this allows you to execute one line of a G-Code file each time you hit ENTER.

#### Transit Speed Buttons

These buttons allow you to specify the transit speed of your machine. Fast transit is the default.

In the cncscrn.ini file, change the "default speed" field to 1, for the speed you would like to be default, all others must stay 0.

Ex: for default medium speed, use the following:  
"JogMode", 0,5,35,25,7,1,"Med","","",-1,-1,"med.bmp"

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



**NOTE:** 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.

### 3.2- 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 3.2a the letters are located away from the XY zero.

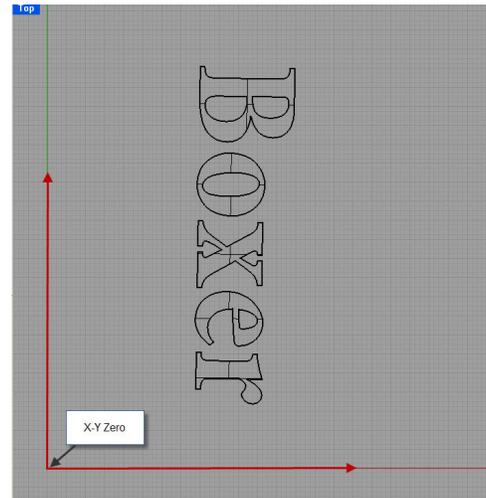
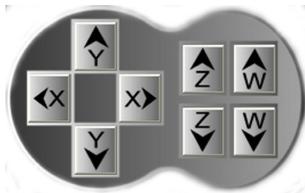


Fig. 3.2a

In Fig 3.3b the object represents the material the letters will be cut from. The machine should be jogged to the corner of the material. Once the machine is in location set XY zero. The coordinates will change to 0,0. XY zero is now set.

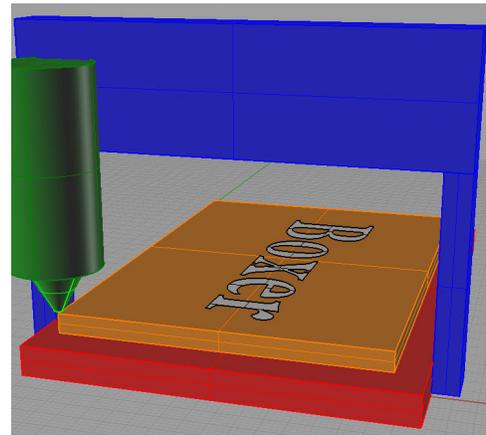


Fig.3.3b

#### Setting the Z-axis zero position.

Use the Z-axis directional arrows 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, G28 can also be used to home only specified axis. (i.e. G28Z homes only the Z axis).

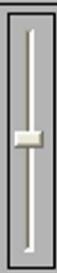
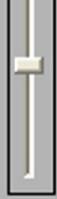
#### Axis Window

The axis window is the primary display for current information about each axis on your machine. The current positions and velocities for each axis are displayed in the text boxes. Displayed above the position and velocity boxes are the current units of measure, any currently active modes, and the current feed rate override percentage. Red triangles to the left and right of each axis label indicate low or high limit switch hits. A red triangle to the left of an axis label indicates a low limit switch hit, and to the right indicates a high limit switch hit. A green box displayed to the right of the position box indicates a temporary workspace (G92) setting, and a blue box indicates a tool length measure or workspace setting. To the far right side of the axis window is the feed rate override control slide bar.

IN G54		100%	
X	0.000	200	
Y	0.000	200	
Z	0.000	50	
W	0.000	50	

## Limit Switches

The default channel settings will normally provide proper interfacing with the machine's limit switches. Before attempting to move the machine, verify that all limit switches are functioning properly. Limit switch status is displayed in the position window using a red triangular indicator to the left or right of the axis label.

IN G54		100%		IN G54		100%	
 X	0.000	200		X	0.000	200	
Y	0.000	200		 Y	0.000	200	
Z	0.000	50		Z	0.000	50	
W	0.000	50		W	0.000	50	

The indicator to the left of the axis label shows low limit status for that axis. The indicator to the right of the label shows high limit status. If the indicator is present then the limit switch is closed, otherwise it is open. The `lim_mode` setting is used to control how WinCNC responds to limit switch triggers.

First toggle your limit switches by hand. Verify that the appropriate limit display toggles. If it does not toggle then you need to adjust `WINCNC.INI`. See the `WINCNC.INI` reference sections of the manual if the default settings are incorrect for your machine.

**It is recommended that you verify limit switch operation at the beginning of each session.** Do not attempt movement until the limit switch displays toggle correctly. Once the limits are working try a small move. If nothing moves, the wrong axis moves, or movement is in the wrong direction then adjust `WINCNC.INI`.

Perhaps the easiest way to do preliminary testing of movement is with the Jog mode. Select "Jog .1", and select the `KEYBOARD` option under the `OPTIONS` menu. If you are using the default jog key configuration, use the `LEFT` and `RIGHT` arrow keys to move X, the `UP` and `DOWN` arrows to move Y, the `PAGEUP` and `PAGEDOWN` keys to move Z, and the `HOME` and `END` keys to move U. Otherwise use the buttons that are defined for your system.

## IV. Advanced Tutorials.

### 4.1- Alternating between Override and Programmed Feedrates.

#### Feed Rate Override

WinCNC accepts feed overrides to increase/decrease the feed rate. The feed rate can be adjusted from 1% to 200% of the programmed rate. The override rate cannot be set higher than the max velocity or G0 rate for a given axis. Use the Insert key to increase and the Delete key to decrease the programmed feed rate, holding CTRL and pressing either Insert or Delete will result in the feed rate being reset to the programmed value. There is also a slider bar to the right of the feed rate screen display that can be used to initiate a feed rate override. The override feed rates will be displayed in place of the programmed feed rates in the display. The feed rate will change colors to depict that an override is in use. If the override rate is higher than the programmed rate the color of the feed rate display will be red, if it is lower than the programmed feed rate it will be blue.

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

### **override=E# L# H#**

Gives the user the options of enabling or disabling the feed rate override, and setting the low and high limitations for the feed rate.

**E#** enables or disables the feed rate override. Can be set to 0 or 1. 0 is disabled, 1 is enabled. The default is enabled.

**L#** sets the low limitation of the override percentage. Can be 1 to 200, but must be lower than the high value.

**H#** sets the high limitation for the override percentage. Can be 1 to 200, but must be higher than the low value.

**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.

**For G Code Command Reference see pages 49 - 55.**

## 4.3 Setting the Table Size.

**table=X# Y# W# H# B# Z#**

Configures table dimensions for the G-Code/BMP Viewer.

X# is the low X coordinate to start viewer area.

Y# is the low Y coordinate to start viewer area.

W# is the width of the table area to view in viewer.

H# is the height of the table area to view in viewer.

B# is the border size in units (keeps table outline viewable).

Z# is the auto-zoom for the G-Code Viewer. 1 (default) - auto zoom out only. 2 - auto zoom in and out. 3 - no auto zoom

## 4.4

### WinCNC Controller

### Command Reference

#### Command Reference

Parameters in [Brackets] are optional.

XYZWIJ . . .	Axis Specification X# Y# Z#
<p>Axis values are specified with a decimal point. A value with no decimal is read as an integer value. No value is equivalent to specifying 0.</p> <p>Example: XYZ is equivalent to X0Y0Z0</p>	

F	Feed Velocity - F#
<p>Velocity is stored separately for Linear XY, Linear ZW. Arc's Rapid and Feed Velocity is stored separately for Linear Moves. Independent velocities are stored based on the XYZW specified in the line containing the F# command. Velocity is specified in units of measure per minute.</p> <p>Example: F60 sets all axes in the feed rate override command group to a Feed Velocity of 60 (see "vgroup=").  G1 X F60 sets the X axis and all other axes grouped with X to a Feed Velocity of 60.  F100 X Y sets the X and Y axes to a Feed Velocity of 100. It also sets any other axes that are grouped with either axis to a Feed Velocity of 100. For information about axis grouping, see "vgroup=".  Feed Velocity changes affect all axes in that vgroup.</p>	

P	Stored Positions - P#
Move to the selected stored position.	

H	Stored Home Positions - H#
Move to the selected stored Home position.	

S	Spindle Speed - S#
Sets the spindle speed to the given value.	

[ ]	Comment
Used to add comments to programs. A closing bracket is optional.	

## WinCNC Controller

## Command Reference

{ }	Braces
<p>Used to substitute axis or parameter values in a line of G-Code. Internally defined variables as well as constant numbers can be used within the braces. Following is a list of internally defined variables and what they represent.</p>	

### Internally Defined Variables

TP1	Tool number, tool changer 1.
TP2	Tool number, tool changer 2.
TN1	Tool number 1 + 10.
TN2	Tool number 2 + 20.
TS1	Tool number 1 + tool changer 1 offset.
TS2	Tool number 2 + tool changer 2 offset.
TH1	Tool height for tool changer 1.
TH2	Tool height for tool changer 2.
TM1	Height of a tool length measure switch. Set using M37.1.
TM2	Height of a second tool length measure switch.
TMX	X position of a tool length measure switch.
TMI	X position of a second tool length measure switch. (In rare cases, all heads cannot reach the same tool measure switch.)
TMY	Y position of a tool length measure switch.
TMJ	Y position of a second tool length measure switch.
TMD	Rapid move distance for auto tool measure.
TMF	Feed rate for auto tool measure.
LZ0	Current G92 value for axis 0.
LZ1	Current G92 value for axis 1.
LZ2	Current G92 value for axis 2.
LZ3	Current G92 value for axis 3.
LZ4	Current G92 value for axis 4.
LZ5	Current G92 value for axis 5.

**WinCNC Controller** **Command Reference**

**Internally Defined Variables continued**

THM	1 if THC Manual Voltage Mode is enabled, 0 if disabled.
THA	1 if THC Auto Voltage Mode is enabled, 0 if disabled.
THO	The specified output channel used to turn the torch on/off for THC.
THI	The specified input channel used to monitor the “arc good” signal for THC.
THP	1 if pierce height for THC is enabled, 0 if disabled.
THC	1 if cut height for THC is enabled, 0 if disabled.
THZ	The specified pierce height for THC.
THR	The specified cut height for THC.
THS	The specified pierce delay for THC.
THE	The specified end delay for THC.
THD	The specified switch compensation amount.
THL	The specified torch lift amount.
THF	The feed rate specified for THC.

Additional variables can be defined in the CNC.MAC file. Defined variables and constants can be used together with simple math operators. ( \* ) multiplication, ( / ) division, ( + ) addition, ( - ) subtraction.

Example: G0Z{TH1+2} moves Z to tool height 1, plus 2 units.

**NOTE:** ANY use of braces requires a value or axis spec and a number. The second parameter inside the braces must be a number.

**WinCNC Controller** **Command Reference**

**G Codes**

G0	Rapid Move - G0 X# Y# Z# W#
<p>Moves to the position specified at Rapid velocity. G0 is modal. After a G0 is executed lines with no G-Code command are executed as a G0. Example: X1Y1 is equivalent to G0 X1Y1 if mode is G0.</p>	

G0.1	Rapid Move with Vertical Lift - G0.1 [X#] [Y#]
<p>First lifts all vertical heads then moves the position specified at Rapid velocity then drops the vertical heads back to their previous positions.</p>	

G1	Feed Move - G1 L# X# Y# Z# W#
<p>Moves to the position specified at Feed velocity. G1 is modal. After a G1 is executed lines with no G-Code command are executed as a G1. Example: X1Y1 is equivalent to G1 X1Y1 if the mode is G1. L# is used to allow setting laser power in vector cutting (1-100.)</p>	

G2	Clockwise Arc - G2 L# X# Y# I# J# Z# W# K# R#
<p>Moves to the position specified at Feed velocity. I is the X distance to the center point. J is the Y distance to the center point. If no XY move is specified, a full circle is cut. If no I or J is specified, previous I J values are kept. Any additional axis spec that is not part of the arc itself will move that axis simultaneously throughout the arc. L# is used to allow setting laser power in vector cutting (1-100). R# is the radius, if R# is used IJK can not be used.</p>	

G3	Counter Clockwise Arc - G3 L# X# Y# I# J# Z# W# K# R#
<p>Moves to the position specified at Feed velocity. I is the X distance to the center point. J is the Y distance to the center point. If no XY move is specified, a full circle is cut. If no I or J is specified, previous I J values are kept. Any additional axis spec that is not part of the arc itself will move that axis simultaneously throughout the arc. L# is used to allow setting laser power in vector cutting (1-100). R# is the radius, if R# is used IJK can not be used.</p>	

**WinCNC Controller**

**Command Reference**

G4	Dwell - G4 X#
<p>Stops movement for the time specified by the X value in seconds. There is no limit to delay time. If no time is specified, then the machine will be stopped until the operator pushes ENTER. Place a comment after the dwell to prompt the operator.                  Example: G4 [Ready To Start Section 2]  <b>WARNING: Never use Dwell to stop the machine while changing parts!</b> Instead program a single part and use the TAB key at the Program prompt. This will repeat the last part cut.</p>	

G9	Smoothing - G9 S# A#
<p>Used to set the smoothing factor. When the XY direction of motion changes, this setting can reduce the “slowdown” to improve the smoothness of motion. Increasing the S# decreases the slowdown of the machine when turning corners. Too high a factor can result in missed steps or motor stalls. Smoothing can also be set in the WINCNC.INI file using G09=S#.                  S# - smoothing factor to be used. (not a specific unit of measure)                  A# - used to change the angle used in the velocity matching algorithm. The A# value represents an angle. If a small angle, such as say 20, is specified, the G9=# will not apply to any angle smaller than 20.</p>	

G20	Converts unit of measure to inches.
G21	Converts unit of measure to centimeters.
G22	Converts unit of measure to millimeters.
G23	Returns to the last used unit of measure.

G28	Return to machine zero - G28 XYZW
<p>Moves specified axes to Lo Limit for XY - Hi Limit for ZW (these are default, they can be reversed). Moves specified axes to WINCNC.INI specs from limits. Sets all axis positions to 0. All axes are moved if none are specified.</p>	

G31	Stop file parsing
<p>Must be used with the M28 command to measure machine position after touching a limit switch.                  Example: L91 G1 Z-10 F20 M28 G31 M37 Z1.</p>	

**WinCNC Controller**                      **Command Reference**

G40	Cutter Compensation Off
Turns off cutter compensation.	

G41	Cutter Compensation Left - G41 O#
Compensates for differences in nominal and actual tool diameters. Starts to the left on the first move. O# is the compensation amount on machines without a tool changer.	

G42	Cutter Compensation Right - G42 O#
Compensates for differences in nominal and actual tool diameters. Starts to the right on the first move. O# is the compensation amount on machines without a tool changer.	

G43	Tool Length Offset On - G43 ZWUV
Sets tool length offsets to the values set by the last M37 command. G43 alone turns all offsets on.	

G49	Tool Length Offset Off - G49 ZWUV
Cancels tool length offsets for axis specified. G49 alone turns all offsets off.	

G50	Scaling/Rotation Mode Off
Cancels the scaling or rotation mode.	

G51	Scaling Mode On - G51 X# Y# Z# I# J# K# C# W#
Enables the scaling factor. Scale each axis type based on the values specified by X# Y# Z#. The object will be centered based on the I# (X center) J# (Y) K# (Z). You can center all axes using the C# line.	

G52	Rotation Mode On - G52 I# J# A#
Enables G-Code rotation. G-Code commands or files will be rotated by the specified number of degrees around the specified center point. I# is the X center point of rotation. J# is the Y center point of rotation. A# is the degrees to rotate.	

**WinCNC Controller** **Command Reference**

G53	Rapid Move - G53 X# Y# Z# W#
Moves to the position specified at rapid velocity, ignoring tool measures and local coordinates. G53 alone will lift all vertical or type 3 axes.	
G54	G54 Workspace - Rotational: G54 X# Y#
Select the Z head. If G54 is commanded from G55, G56, or G57 mode, the Z head will be moved to the current XY position.	
G55	G55 Workspace - Rotational: G55 X# Y#
Select the W head. If G55 is commanded from G54, G56, or G57 mode, the W head will be moved to the current XY position.	
G56	G56 Workspace - Rotational: G56 X# Y#
Select the U head. If G56 is commanded from G54, G55, or G57 mode, the U head will be moved to the current XY position.	
G57	G57 Workspace
Select the V head. If G57 is commanded from G54, G55, or G56 mode, the V head will be moved to the current XY position.	
G73	Chip Break Cycle - G73 X# Y# Z# R# Q# F#
Moves to XY specified at Rapid velocity. On the first peck, moves to R retract height at Rapid velocity, moves to Q peck depth at F feed velocity, lifts 0.05" at Rapid velocity to break the chip, moves to next peck depth at Feed velocity. The cycle repeats until Z depth is reached. Then moves back to retract height R at Rapid velocity. Without this cycle, some materials can produce a long "stringer" which can become a safety issue to the operator.	
G80	End Drill Cycle
Used to disable the G81 Drill Cycle mode. Note: Switching to G0 or G1 will also break out of the drill cycle mode.	

**WinCNC Controller**                      **Command Reference**

G81	Drill Cycle - G81 X# Y# Z# R# F#
<p>Moves to XY specified at Rapid velocity. Moves to R (clearing height) at Rapid velocity. Moves to Z specified at Feed velocity. Moves back to R at Rapid velocity. F allows you to set the feed rate.</p>	
G82	Dwell Cycle - G82 X# Y# Z# R# P# F#
<p>Moves to XY specified at Rapid velocity. Moves to R (clearing height) specified at Rapid velocity. Moves to Z specified at Feed velocity. If P is specified, bit pauses that amount of time at the bottom of the hole, then moves back to R at Rapid velocity. P is measured in milliseconds (thousandths of one second). F allows you to set the feed rate.</p>	
G83	Peck Drill Cycle - G83 X# Y# Z# R# Q# F#
<p>Moves to the XY specified at Rapid velocity. On the first peck, moves to R retract height at Rapid velocity, moves to peck depth Q at Feed velocity, then moves back to retract height at Rapid velocity. On each subsequent peck, moves at Rapid velocity to within 0.05" of previous peck depth, then moves at Feed velocity to next peck depth. Cycle repeats until Z depth is reached. Then moves back at Rapid velocity to retract height R. The F allows user to set the feed rate.</p>	

**WinCNC Controller**

**Command Reference**

G90	Absolute Mode
Can be specified with other G-Codes on any line. XYZW values from the current line forward are read as absolute coordinates. IJ values are always relative to the current XY position, not absolute positions regardless of G90/G91 mode.	
G91	Relative Mode
Can be specified with other G-Codes on any line. XYZW values from the current line forward are read as relative movements from the current position.	
G92	Set Local Coordinates - G92 X# Y# Z# W#
Used to specify a new coordinate system for running absolute mode programs. Use G92 alone to restore the Machine Coordinates. G92 X0 Y0 Z0 W0 sets the current position to zero. G92 then restores the Machine Coordinates values.	
**If running a .tap, .mac, .nc etc file, and there is a G92 in the file, and the user wishes to maintain that G92 after the file is done and closed, then a L92 must be added at the end of the file.	
G92.1	Shift MZ Coordinates - G92.1 X# Y# Z# W#
Similar to G92 except that the coordinate system produced shifts Machine Zero by the amount specified. This is useful since a given local coordinate system can be set without positioning the head to a certain position first. G92 X10 Y10 sets the absolute position X10 Y10 to X0 Y0 in local coordinates. G92.1 may also be used to restore a single axis to absolute coordinates. G92.1 X0 leaves YZW local coordinates but sets X back to absolute coordinates.	

**WinCNC Controller**                      **Command Reference**

G92.2	Shift LZ Coordinates - G92.2 X# Y# Z# W#
Similar to G92.1 except that the coordinate system shift is added to the current local coordinates instead of replacing them. This is useful for 'jogging' a local position. If the Z head has been set for running a G90 mode file but then needs to set to cut .010" deeper, G92.2 Z-.01 will accomplish this with a single command. Can be useful as a macro.	
G92.3	Stores and Restores G92 values
Allows the user to switch back and forth between local and absolute coordinates. Use the G92.3 command without any parameters or axis characters. If G92 positions are set on any axis then they will be stored and cleared. The next G92.3 call will restore the previous G92 values even if they have been cleared by using G92.	
G93	Inverse Time Feed Mode
In inverse time feed rate mode, an F word means the move should be completed in [one divided by F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute. When the inverse time feed rate mode is active, an F word must appear on every line which has a G1 motion, and an F word on a line that does not have G1 is ignored. Being in inverse time feed rate mode does not affect G0 motions. It is an error if: inverse time feed rate mode is active and a line with G1 does not have an F word.	
G94	Cancels the Inverse Time Feed Mode
Turns off the Inverse Time Feed Mode.	

**WinCNC Controller** **Command Reference**

**L Codes**

L1	Save Positions - L1 N# V#
Store values is the saved positions file. Can store a value in one of the 30 existing 'H' registers. N# is the desired 'H' register number, can be 0 to 29. V# is the desired value. Can be the result of a math operation or a constant.	
L1.1	Save 'H' Register
Save the 'H' register value to a file.	
L1.2	Load 'H' Register
Load the 'H' register value from a file.	
L3	Set Home Position - L3 XYZW
Stores current position of each axis specified. Values specified are ignored.	
L4	Return to Home
Moves each axis specified to the last L3 position stored. Values specified are ignored.	
L5	Set Contouring Velocity - L5 XYZ
Sets the contouring velocity in units per minute.	
L6	Set Acceleration - L6 XYZ
Sets the acceleration in units per minute per second. L6 alone restores the original values.	
L7	Set Acceleration Percentage L7 P#
Sets the acceleration to a percentage of the current acceleration. P# is the percentage (1-100) to set. L7 alone restores the original.	



**WinCNC Controller**                      **Command Reference**

L11	Set Axis Mapping - L11 XYZW
<p>Allows re-mapping of axis inputs. Specify the axes that should receive the input from XYZW in that order. Specify L11 alone to return to normal parsing.                  Example: L11 XYWZ swaps the W and Z axes, L11 XYZZ moves Z and W together.</p>	

L12	Set Axis Mapping - L12 ZW
<p>Allows re-mapping of axis inputs. Specify the axes that should receive the input from ZW in that order. This command works like L11 except that the XY inputs are not used. Specify L12 alone to return to normal parsing.                  Example: L12 WZ swaps the W and Z axis, L12 ZZ moves Z and W together.</p>	

L13	Set Axis Mapping - L13 XYZW
<p>Allows re-mapping of axis on a pulse basis. This means when the controlling axis takes a step, the slave will follow in that direction. Specify L13 alone to return to normal mapping.                  Examples:                  L13 ZW (Z is controlling axis, W follows Z.)                  L13 WZ (W is controlling axis, Z follows W.)                  L13 XY (X is controlling axis, Y follows X.)</p>	

**WinCNC Controller**                      **Command Reference**

L20	Enable Soft Limits - L20 [X1, X2]
Enable Soft Limit and Boundary Checking: L20 X1 enables Boundaries only. L20 X2 enables Soft Limits only. L20 alone enables both.	

L21	Disable Soft Limits
Disable Soft Limit and Boundary Checking Soft Limits and Boundaries are used to define a cutting area which is checked during parsing of a file or command. This effectively keeps the machine from moving out of a defined area. L21 disables Soft Limit and Boundary checking.	

L22	Set Low Boundaries - L22 X# Y# Z# W#
Set Low Boundaries to values specified. If no values are given then all Low Boundaries are set to WINCNC.INI defaults.	

L22.1	Restore Low Boundaries - L22.1 XYZW
Restores default Low Boundaries to the specified axes.	

L23	Set High Boundaries - L23 X# Y# Z# W#
Set High Boundaries to values specified. If no values are given then all High Boundaries are set to WINCNC.INI defaults.	

L23.1	Restore High Boundaries - L23.1 XYZW
Restores default High boundaries to the specified axes.	

L28	Set Position - L28 X# Y# Z# W#
Sets Machine Coordinates. Machine Coordinates specify the fixed machine zero point for your machine. L28 is normally used only to write a custom homing program. L28 does not move the machine, but sets the current position to the values specified. Use G92 to set a local coordinate system for running absolute mode programs from any table position.	

**WinCNC Controller**                      **Command Reference**

L29	Clear Home Flags - L29 XYZ
Clears all require home flags. Axes can be used with the command to clear require home flags only on the specified axes.	
L29.1	Ignore Home Flags
Ignores require home flags for all axes during execution of the currently running file.	
L30	Knife Mode On
Enables support for the Knife Mode.	
L30.1	Knife Parameters L30.1 A# Z# P#
Set Lift Angle, Lift Position, and Plunge depth for knife. When amount of change in the angle of the knife exceeds A# the knife will lift to make the turn. A# is also used for angle correction in Target Mode: see <b>Knifemode=</b> . Z# is the position to lift to. P# is the position that the knife will plunge to.	
L31	Knife Mode Off
Disables support for the Knife Mode.	
L32	Saw Mode On
Enables support for the Saw Mode.	
L33	Saw Mode Off
Disables support for the Saw Mode.	
L34	AutoAuxOut Enable - L34 D#
Enables an auxiliary output that will turn on during G1/G2/G3 moves, and will turn off for all other moves. L34 D# sets the dwell time in seconds used when the channel is set to on and L35 D# sets the dwell time in seconds when the channel is set to off. When the 'D' parameter is used with the L34 command the "AutoAuxOut" mode does not change. L34 parameters are set in the WINCNC.INI file by the "autoaux=" line.	

**WinCNC Controller**                      **Command Reference**

L35	AutoAuxOut Disable - L35 D#
Disables the auxiliary output that is turned on by the L34 command. L35 D# sets the dwell time in seconds when the channel is set to off. When the 'D' parameter is used with the L35 command the "AutoAuxOut" mode does not change.	

L36	Torch Height Control On.
L36.1	Auto Voltage Set Mode On.
L36.2	Reset Auto Voltage Set Status.
L37	Torch Height Control Off.
L37.1	Auto Voltage Set Mode Off.
L38	Torch On.
L39	Torch Off.

L40	Feed Rate Override - L40 P#
Allows user to set the feed rate override by command. P must be within the bounds specified in the INI file. If not, the command is ignored. The L40 command without a P# resets the feed rate to the default.	

L41	Shell Command - L41 C:\Path\Program.exe
Allows the user to run an executable program from within WinCNC. <b>NOTE:</b> There is no error reporting for this command. If the path or filename are incorrect the command will fail without aborting.	

L48	Laser Height Control
Sets the target voltage to the current input voltage.	

L51	Set D/A - L51 V# P# S#
Sets the output voltage for DA. The V parameter is used to set the DA by voltage, the P to set the DA by a Percentage, and the S is used to set the DA by spindle speed. <b>ONLY 1 PARAMETER CAN BE USED.</b>	

L53	Set Vertical Head Height
Raises all vertical heads as high as possible while keeping the bits at equal height.	

**WinCNC Controller**                      **Command Reference**

L55	Pump Velocity Mode On
Enables velocity matched pump control.	
L56	Pump Purge Mode On
Sets pump to purge mode.	
L57	Pump Off
Stops the pump.	
L58	Pump Off with Auto Reverse
Stops the pump, using auto reverse.	
L59	Set Up Pump Max - L59 M#
Sets the pump max to the specified M parameter.	
L60	Turn Virtual Input Off - L60 C#
Turn specified virtual input off, # can be 1-64.	
L61	Turn Virtual Input On - L61 C#
Turn specified virtual input on, # can be 1-64.	
L70	Re-Enter Last Command
When L70 is used in a file, the last used command will automatically be re-entered on the command line after file completion.	
L82	Measures the Surface - L82 Z#
L82 Z# measures the surface with the probe and sets the measured point to the Z value given. L82 alone will set the measured point to Z0.	
L82.1	Sets the Surface for Probe Digitization
Sets a G92 for vertical axis based on the L82.3 calibration.	
L82.2	Moves the Surface for Probe Digitization
Adjusts G92 positions set by the last L82.1 based on a new measurement.	

**WinCNC Controller**                      **Command Reference**

L82.3	Calibrate Digitizing Probe
Calibrates, or stores, the distance from the probe contact point to the table surface.	

L83	Measures Z Depth - L83 X# Y#
L83 command will do a one-shot Z probe at the specified location, and record the XYZ position values in a comma delimited text file named SCAN.CSV. Where X# and Y# specify the (X,Y) coordinates to move to before doing the depth probe.	

L90	One Time Absolute
L90 is used in a command to specify G90 coordinates on one line without changing program mode to G90.	

L91	One Time Relative
L91 is used in a command to specify G91 coordinates on one line without changing program to G91.	

L92	Cancel G92 Restore
The current G92 Local Coordinate system is saved every time a program is started or a subprogram is called using M98. When the program is finished, the original G92 system is restored. This is not desirable if the program is being used to preset a system configuration for other programs. L92 causes the normal G92 coordinate restore to be skipped after the program containing the L92 is finished.	

L110	Tool Change - L110 T#
Sets current tool number for machines with an automatic tool changer. The tool number is available as {TP1} in subsequent commands, or {TN1} and {TN2} for dual tool changer machines. The stored tool measure is reloaded for the new tool. (See M37 command for storing tool measures)	

L111	Test Tool Number - L111 T#
Run next line if tool number is equal to T#.	

L111.1	Test Tool Number with Dual Tool Changers - L111.1 T#
Run next line if both tool numbers are equal to T#.	

**WinCNC Controller** **Command Reference**

L112	Test Tool Number - L112 T#.
Skip next line if tool number is equal to T#.	

L113	Spindle Speed Conditions - L113 L# H# V#
Skip the next line if the V# is not greater than the L# and less than the H#.	

L120	Tool Change - L120 T#
Sets current tool number for machines with an automatic tool changer. The tool number is available as {TP2} in subsequent commands.	

L201	Set 201 Signal Command - L201 FileName T# C# S#
Sets the command used with the 201 signal type.	

L202	Set 202 Signal Command - L202 [Command]
Sets the command used with the 202 signal type.	

L204	Activate Signals
L204 C# activates signals and enables the specified channel. L204 activates all signals and enables.	

L205	Deactivate signals
L205 C# deactivates signals and enables the specified channel. L205 deactivates all signals and enables.	

L210	Select Alternate Low Limits - L210 XYZW
Selects axis to use alternate low limits.	

L210.1	Select Alternate Low Limit Pin - L210.1 XYC#
Selects alternate low limit pin from the specified C# channel. If no channel is specified, the original settings are restored.	

**WinCNC Controller**                      **Command Reference**

L211	Select Alternate High Limits - L211 XYZW
Selects axis to use alternate high limits.	

L211.1	Select Alternate High Limit Pin - L211.1 XYC#
Selects alternate high limit pin from the specified C# channel. If no channel is specified, the original settings are restored.	

L212	Select Primary Limits
Selects the primary limits for all axes.	

L213	Dual Axis, Select Main - L213 XYZW
Selects the main motor on a dual motor axis.	

L214	Dual Axis, Select Secondary - L214 XYZW
Selects the secondary motor on a dual motor axis.	

L215	Dual Axis, Select Both Axes - L215 XYZW
Selects both motors on a dual motor axis.	

L249	Laser Power Setting - L249 S# P# D# O# M# I#
Using the L249 command allows you to specify the laser power. Example: L249 P50 (sets laser power to 50). S# - 0 or 1 sets smart engrave off/on. P# - sets the laser pulse width in percentage (1-100). D# - sets the laser pulse duty cycle (1-100). O# - sets the bitmap engraving offset. M# - sets the bitmap engraving margin. I# - reverses color polarity.	

L250	Vector Laser Off
Disables vector laser cutting.	

**WinCNC Controller** **Command Reference**

L251	Vector Laser On - L251 P# D# L#
Enables vector laser cutting. P# - laser power (1-100). D# - sets the distance for the distance based vector laser firing. L# - sets the distance for the off signal when perforating.	
L251.1	Laser Dwell - L251.1 D#
D# - time in seconds for dwell.	
L252	Bitmap Engraving
L252.1 Stores rotation for future use.	
L253	Bitmap Engraving rotated 90 degrees clockwise.
L253.1 Stores rotation for future use.	
L254	Bitmap Engraving rotated 180 degrees clockwise.
L254.1 Stores rotation for future use.	
L255	Bitmap Engraving rotated 270 degrees clockwise.
L255.1 Stores rotation for future use.	
L256	Offset for bitmap engraving row alignment L256 R#
Offset on positive motions to compensate for laser firing delays to line up bitmap rows during dual motion engraving. R# sets the radius used for rotary engraving in the current unit of measure.	
L300	Lock Axis - L300 XYZW
Locks specified axis motion. Any motion commanded to a locked axis will be ignored.	
L301	Unlock Axis - L301 XYZW
Unlocks specified axis motion.	

**WinCNC Controller**                      **Command Reference**

L400	Create Home File
Creates a home file at the machine's current position.	

L401	Recall Positions
Moves all axes to the positions where they were at previous to the last file ran.	

L500	Store Absolute Positions - L500 [X, Y, Z...] [P H T] L# C# N"Name"
<p>[X, Y, Z...] - axes to set positions for, if none specified then all axes positions are stored.                  [P H T] - type of position (P)osition, (H)ome Position, (T)ool Position. Can only use one value.                  L# - 0 no vertical head lift, 1 include head lift.                  C# - Clearance for vertical heads if L1.                  N"Name" - Name for specified position. &lt; 16 characters long.</p>	

L500.2	- L500.2 T# N# A# C# O# H# M#
<p>T# specifies the tool number.                  N# specifies the nominal diameter for the tool.                  A# specifies the actual diameter for the tool.                  C# specifies the diameter compensation for the tool.                  O# specifies the height offset for the tool.                  H# specifies the tool height.                  M# specifies the maximum current allowed for the tool.                  This command sets up a tool library instead of using the dialog boxes.</p>	

L501	Store Local Positions - L501 [X, Y, Z...] [P H T] L# C# N"Name"
<p>[X, Y, Z...] - axes to set positions for, if none specified then all axes positions are stored.                  [P H T] - type of position (P)osition, (H)ome Position, (T)ool Position. Can only use one value.                  L# - 0 no vertical head lift, 1 include head lift. Default L1.                  C# - Clearance for vertical heads if L1.                  N"Name" - Name for specified position. &lt; 16 characters long.</p>	

**WinCNC Controller**                      **Command Reference**

L600	Disable Mapping Adjustments
Disable mapping adjustment for the specified axis. If no axis is specified, mapping adjustments for all axes are disabled.	

L601	Enable Mapping Adjustments
Enable mapping adjustment for the specified axis. If no axis is specified, mapping adjustments for all axes are enabled.	

L602	Start Mapping Setup
Starts mapping setup for specified axis. First axis specifies the axis to map and the second axis specifies the axis to adjust. The axis map will automatically move to the start map position. No axis spec stops the mapping setup.	

L603	Store Current Adjustment and Move - L603 {NPSE}
Stores the current adjustment and moves the map axis to the specified map increment. N is next, P is previous, S is start, and E is end. The default move is to the next map increment position.	

L604	Move Map Axis By Increment - L604 {NPSE}
Moves the map axis to the specified map increment. N is next, P is previous, S is start, and E is end. The default move is to the next map increment position.	

L605	Store Current Adjustment
Stores the current adjustment.	

L800	Disable Probe
Disables the probe.	

L801	Enable Probe
Enables the probe.	

When probe is enabled (L801) it acts similarly to a limit switch. When it comes in contact with an object, it will stop motion. If it is disabled (L800) it no longer functions.

**WinCNC Controller**                      **Command Reference**

L802	Scan Given Rectangle - L802 X# Y# I#
Scans a given rectangle XY with an increment of I in and up/down, and side-to-side style scan. (Use automatic scan)	

L803	Scan Given Rectangle - L803 X# Y# I#
Scans a given rectangle XY with an increment of I in an up/down style scan.	

L810	Scan Slice (Drop-off Detection) - L810 FILENAME. EXT X# Y# I# K#
Digitizes a slice (line), either in X direction, or Y direction and saves the points to a text file. If a drop-off is encountered, or distance stated in X or Y is reached, the slice is saved and the scan is ended. <b>Only one axis can be scanned at a time: X or Y.</b> Uses up/down, and side-to-side probe type. FILENAME.EXT - replace with your filename. <b>Do not surround with quotation marks.</b> X# - is the actual X axis distance that you want the probe to travel looking for an edge. If it finds one, the file is saved and the scan ends. Y# - works like X#, except on the Y axis. I# - is the maximum increment for the horizontal axis. K# - is the vertical increment (how high the probe raises).	

L820	Disable axisadj
Disables automatic axis position adjustment.	

L821	Enable axisadj
Enables automatic axis position adjustment.	

L901	Limit Switch Debounce - L901 D#
Changes the limit switch debounce. D# can be 1-30.	

L910	Disable G-Code Line Display
Disabling G-Code display when running large G-Code files can reduce CPU and memory usage resulting in better overall computer system performance.	

**WinCNC Controller**                      **Command Reference**

L911	Enable G-Code Line Display
Enabling G-Code display when running large G-Code files can increase CPU and memory usage resulting in worse overall computer system performance.	

**M Codes**

M11	Auxiliary Output On - M11 C#
Controls auxiliary outputs. Channel specified by # is turned on. Port and bit addresses are set up in WINCNC.INI file. <b>Note: For use with a Drill Bank</b> A d# can be added to the M11 command for use with a drill bank. The d# is the binary number of channels up to the n# specified in the drill bank line to turn on. Example: M11D7 would turn on the first 3 channels of a ten channel drill bank.	

M12	Auxiliary Output Off - M12 C#
Controls auxiliary outputs. Channel specified by # is turned off. Port and bit addresses are set up in WINCNC.INI.	

M13	Auxiliary Output On - M13 C#
Same as M11, except the output is turned on without pausing motion.	

M14	Auxiliary Output Off - M14 C#
Same as M12, except the output is turned off without pausing motion.	

M17	Pause Until Input On - M17 C#
Wait for channel specified by # to come on. Port and bit addresses are set up in WINCNC.INI.	

M17.1	Run Next if Input On - M17.1 D# C#
Run next command only if channel specified by # is on. Port and bit addresses are set up in WINCNC.INI. D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.	

**WinCNC Controller** **Command Reference**

M17.2	Run Next if AuxOut is On - M17.2 D# C#
Run the next command only if channel specified by the # is on. Port and bit addresses are set up in WINCNC.INI file. D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.	

M17.3	Run Next if Virtual Input On - M17.3 C#
C# is a "virtual input" channel, will run the next line if the specified "virtual input" channel is on and skip the next line if the channel is off. # can be 1-64.	

M17.4	Run Next if C# is 1, Otherwise Skip. M17.4 C#
C# can be 0 or 1. 1 will run the next line, 0 will skip the next line.	

M18	Pause Until Input Off - M18 C#
Wait for channel specified by # to go off. Port and bit addresses are set up in WINCNC.INI.	

M18.1	Run Next if Input Off - M18.1 D# C#
Run next command only if channel specified by # is off. Port and bit addresses are set up in WINCNC.INI. D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.	

M18.2	Run Next if AuxOut is off - M18.2 C#
C# is an "auxiliary output" channel, will run the next line if the specified output channel is off and skip the next line if the channel is on. # can be 1-64.	

M18.3	Run Next if Virtual Input is Off - M18.3 C#
C# is a "virtual input" channel, will run the next line if the specified "virtual input" channel is off and skip the next line if the channel is on. # can be 1-64.	

M18.4	Run Next Line If C# is 0, Otherwise Skip - M18.4 C#
C# can be 0 or 1. 0 will run the next line, 1 will skip the next line.	

**WinCNC Controller**

**Command Reference**

M28	Disable Limit Abort
<p>M28 is used to write your own home program instead of using G28 to home the machine. WinCNC normally aborts a program when a limit switch is encountered. Adding M28 to a G0 or G1 command disables the program abort and instead stops the limited axis while allowing other axes to continue. Limit Cushions are disabled while using M28 so any move done with M28 should be done slowly to avoid machine damage. M28 must be specified on each line separately.</p>	

M29	Disable Limit Abort
<p>M29 is used to write your own home program instead of using G28 to home the machine. Adding M29 to a G0 or G1 command disables the program abort and instead stops the current G-Code line but continues on the following line. M29 must be specified on each line separately.</p>	

M37	Measure Tool - M37 O# Z# W# H# T#
<p>M37 is used to set Tool Length Offsets. The end of the tool is first set to a known position. M37 is then specified to set the tool length offset.</p> <p>O# manually specifies the tool height offset.</p> <p>H# is the tool number of the currently loaded tool. Use this with mode 0 only, M0 on the ATC line in the WinCNC.ini file.</p> <p>T# is the tool number of the currently loaded tool. Use this with mode 1 only, M1 on the ATC line in the WinCNC.ini file.</p> <p>Example: Jog the Z head to the spoil board. Enter M37Z0 to the Z work coordinate to 0. Once tool length offsets are set, Boundaries may be used to insure that the table is not routed by accident. Normally zero values are set to the top of the spoil board. Use G92 to set material thickness if programs reference zero from the top of material. This eliminates re-measuring tool for different thickness materials. G43 and G49 are used to enable and disable offsets.</p>	

M37.1	Measure Tool for Calibrating G37 - M37.1 Z# W#
<p>M37.1 is used to store a tool measure in the TM1 variable. This command is used primarily with calibrating G37 for automatic tool measuring.</p>	

**WinCNC Controller** **Command Reference**

M37.2	Measure Tool for Calibrating G37 - M37.2 Z# W#
M37.2 is used to store a tool measure in the TM2 variable. This command is used primarily with calibrating G37 for automatic tool measuring.	

M37.3	Shift Workspace Offsets M37.3 S# H# T# M# X# Y# Z# ...
M37.3 is used to shift workspace offsets for tool measures, tool measure switch heights, or individual axis. S# is the shift amount to be used for tool measures and tool measure switch positions. (usually a negative #). H# is the H register number to be shifted by the shift amount. H0 will shift all H registers. T# is the tool number height to be shifted by the shift amount. T0 will shift all tool numbers. M# is the measure switch height to be shifted by the shift amount. M0 will shift both switches. X# Y# Z# ... shifts each axis workspace offsets by the specified value.	

M98	Subprogram Call - M98 <filename>
Run a subprogram. A subprogram may be any G-Code file. Control is passed back to the calling program when finished. To run a G90 mode subprogram from any position on the table use G92X#Y#Z#W# to set the local coordinates at the beginning of the file.	

M98.1	Subprogram Call - M98.1 <filename>
Same as M98, except line numbers in the subprogram are not counted. This is most useful for macros that call subprograms.	

M99	End Subprogram
Ends a subprogram.	

M99.1	Abort
Initiates a commanded program abort.	

**WinCNC Controller Messages and Error Codes****Messages and Error Codes****Error - Line Too Long**

Input line more than 256 characters.

**Error - Unsupported G Code**

G Code in input line which is not supported.

**Error - Unsupported L Code**

L Code in input line that is not supported or ignored.

**Error - Unsupported M Code**

M Code in input line which is not supported.

**Error - Multiple Commands**

Input line contains more than one command.

**Error - Syntax - Command or File Not Found**

Non-supported code or invalid file name entered.

**Error - Arc Radius**

Distance from start point to center is not equal to distance from endpoint to center. Arc Radius errors can occur from running a G2 or G3 intended for G91 mode in G90 mode and vice versa.

**Error - Soft Limit Exceeded**

Move specified would result in the head being moved outside the soft limits.

**Error - Boundary Exceeded**

Move specified would result in the head being moved outside the workspace boundaries.

**Error - Acceleration Out of Bounds**

Acceleration rate too high for resolution. Maximum (acc\*res) in steps is 65535.

**WinCNC Controller Messages and Error Codes****Error - Illegal Value**

The value specified is invalid.

**Error - Unknown Position Specified**

The position specified is invalid.

**Error - Subprogram Nesting Too Deep**

Too many nested subprogram calls.

**Error - L10 in Subprogram**

L10 command found in subprogram.

**Error - File Not Found**

The specified file name cannot be found.

**Error - Parameter Not Specified**

A required parameter is not specified.

**Error - Unknown**

An unknown error has occurred.

**Error - Memory Error**

Insufficient or bad memory.

**Error - User Level 5 Required**

A Level 5 user license is required.

**Error - Invalid L10 Command**

L10 command has invalid parameters.

**Error - Invalid P1 Command**

P1 command has invalid parameters.

**WinCNC Controller Messages and Error Codes****Error - Command Not Implemented**

This command has not yet been fully implemented.

**Error - Tool Changer Not Enabled**

Tool changer is not set up in the WINCNC.INI file.

**Error - Parameter Out of Range**

Specified parameter is out of range.

**Error - D/A Not Enabled**

D/A channel is not set up in the WINCNC.INI file.

**Error - Arc Too Small**

Specified arc move is too small.

**Error - Device Not enabled**

The requested device is not enabled.

**Error - Could Not Open File**

Requested file could not be opened. May be in use by another program.

**Error - Pump Control Not Enabled**

The pump control feature has not been enabled.

**Error - Soft Limits Not Enabled**

Softlimits are set as disabled.

**Error - Tangential Knife Not Enabled**

The tangential knife feature is not enabled.

**Error - Forced Limit Exceeded**

Move specified would result in the head being moved outside the forced limits.

**WinCNC Controller Messages and Error Codes****Error - Digitizer Probe Not Enabled**

The digitizer probe feature has not been enabled.

**Error - Arcs Disabled When X or Y scaled**

When scaling is enabled, arcs are not supported.

**Error - Rotating Saw Not Enabled**

The rotating saw feature is not enabled.

**Error - Macro Expansion Error**

Occurs when macro commands exceed 256 characters.

**Aborted - Limit Switch**

Limit switch encountered while running.

**Aborted - Aborted By User**

Escape button pushed while running.

**Key Not Found**

The software key cannot be found.

**Wrong Key Found**

The software key found is incorrect.

**Key Error #**

A software key read error has occurred.

**Errors Reading WINCNC.INI**

The WINCNC.INI file contains errors or is not found.

**Errors Reading CNCSCRN.INI**

The CNCSCRN.INI file contains errors or is not found.

**WinCNC Controller Messages and Error Codes****CNC.MAC Not Found**

The CNC.MAC file is not found.

**Upgrade needed for Key Version 5, or 6 Axis card**

And upgrade of the daughterboard is needed.

**Pausing.....**

Motion is pausing.

**Continuing.....**

Motion is continuing.

**Finished Running "Filename"**

The file "Filename" is complete or has been aborted.

**Runtime: #d Minute(s) and #f Second(s)**

Run time for the last file in minutes and seconds.

**File Not Found**

A specified file cannot be found.

**Error - Demo Count Exceeded**

Demo mode only. 25 commands have been completed.

**Error - Helical Interpolation Not Supported**

No Z or W moves during an arc.

**Escape Button Pushed While Running**

User has aborted motion with the ESC key.

**Error - Not a BMP File**

Tried to open a non-bmp file for bitmap engraving.

**WinCNC Controller Messages and Error Codes****Error - BMP File Invalid Header**

Tried to open an incompatible bmp file.

**Error - 2-Bit Bitmap Engrave Not Enabled**

Tried to open a 2-Bit bitmap without the feature being enabled in the software.

**Error - 8-Bit Bitmap Engrave Not Enabled**

Tried to open an 8-bit bitmap without the feature being enabled in the software.

**Error - Vector Laser Not Enabled**

Vector laser feature not enabled in the software.

**Error - Auto-AuxOut Not Enabled**

User has tried to use an Auto-AuxOut without the "autoaux=" line being set in the WINCNC.INI file.

**Error - Depth Must Be Lower Than Retract**

Tried to retract a tool to a height that was below current depth.

**Error - Unknown**

An error has occurred that is undefined in WinCNC.

## 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.**