

ATLAS

Series CNC Router Manual

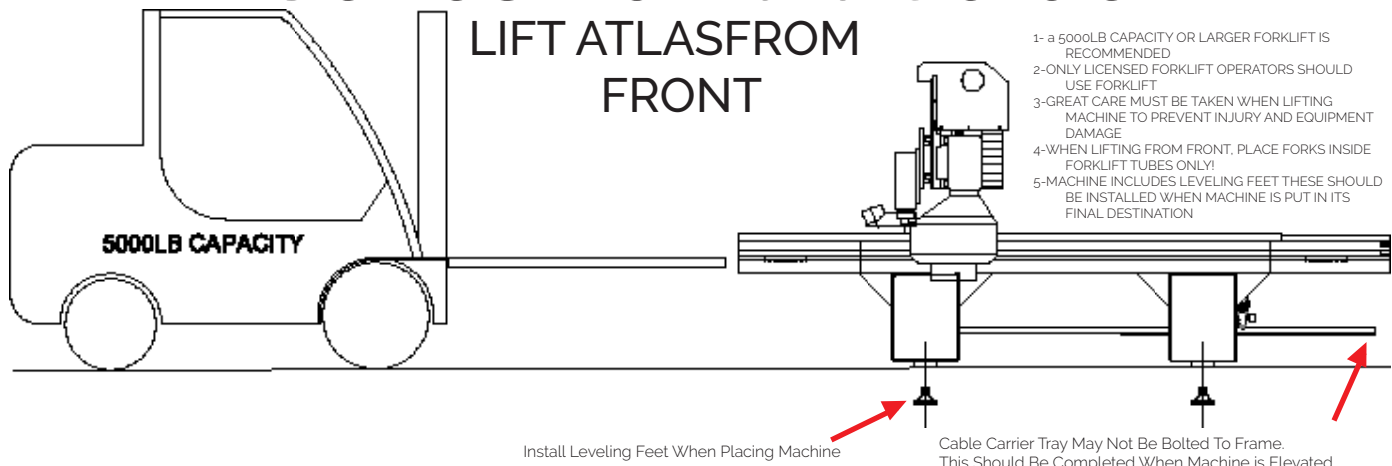


TABLE OF CONTENTS

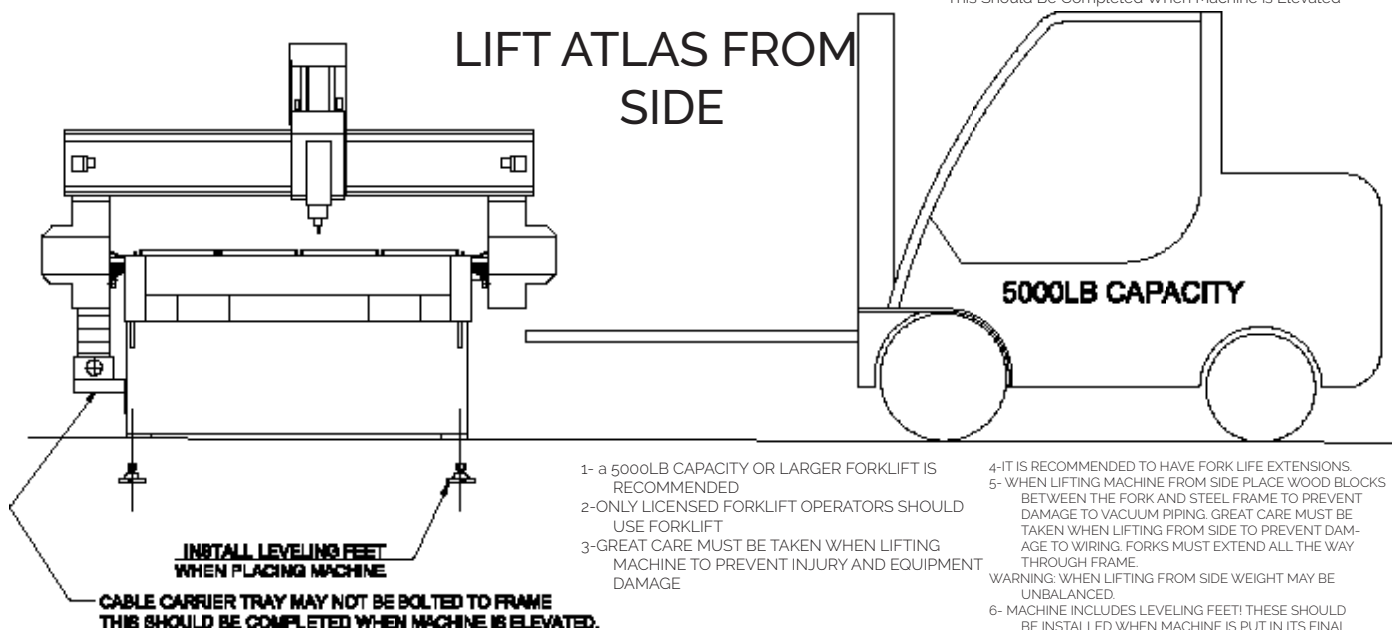
Forklift Guide	Page 3
Safety Instructions	Page 4
Collecting Guidelines	Page 5
I ATLAS Series Quick Setup	Page 6
I.I ATLAS Series Installation	Page 7-9
II Vacuum Pump Installation	Page 10
ATLAS Series Enabling the Machine	Page 11
III.I ATLAS Series Start Up	Page 12
Functions of the Keys	Page 13
Shift Commands / Combination Keystrokes	Page 14
III Operating Tutorials	
3.0- Switching Movement to Step or Jog	Page 15
3.1- Jogging the Machine/Changing from High/Low Jog Speed	Page 15
3.2- Stepping the machine	Page 15
3.3- Modifying the Jog Speed and Step Size	Page 16
3.4- Feedrate Override	Page 17
3.5- Adjusting the XYZ Position/WCS/User Origin	Page 18
3.6- Loading a G-code file	Page 19
3.7- Running a G-code File	Page 19
IV Advanced Tutorials	
4.1- Alternating between Override and Programmed Feedrates	Page 20
4.2- Setting Override speed for a G-code file	Page 21
4.3- Setting the Table Size	Page 22
Changing to Different Offset (a new XY Zero location)	Page 23
Notes on the G-code file Acceleration Set	Page 24
V Appendix	
ATLAS Series Settings	Page 26 - 30
Using the 4th Axis on an ATLAS Machine	Page 31
VI Atlas Machine Lubrication	Page 33
Becker Vacuum Pump Manual	Page 34-38
Becker Vacuum Pump Filter Inspection	Page 39 - 41
Becker Vacuum Pump Greasing Process	Page 42 - 46
VII Warranty	Page 47

Techno CNC Systems Atlas Series Forklift Guide

LIFT ATLAS FROM FRONT



LIFT ATLAS FROM SIDE



SAFETY WARNING: DO NOT LIFT MACHINE OR MOVE MACHINE BY THE GANTRY

For safety and to prevent damage to the machine, Lift Machine Using The Frame ONLY

** The side of the Atlas Series CNC Router does NOT have forklift tubes. Make sure forklift blades enter through the frame of the machine only. Use blocks when lifting machine to prevent damage to the vacuum table plumbing **

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

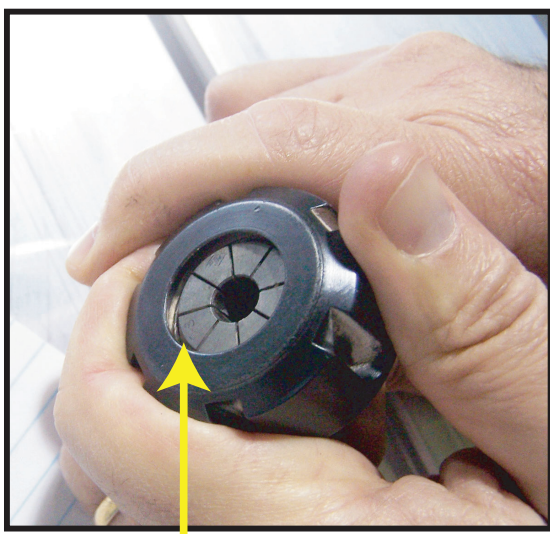


WARNING!
THE SPINDLE WILL BE DAMAGED
IF UNBALANCED EQUIPMENT IS USED.

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!**

The collet should be secured in the collet nut before threading 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.I Techno Atlas Series Unpacking and Placement



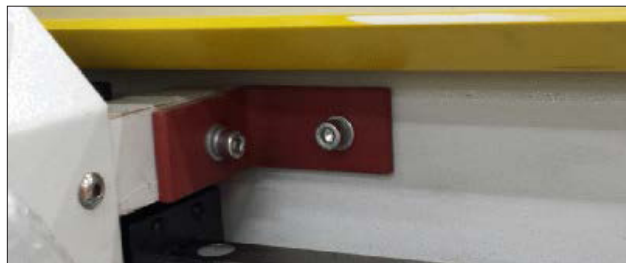
Remove all bubble wrap, foam, and strapping from the machine.

Carefully remove the Atlas from its wood pallet. Be sure to remove the bolts from its four leveling feet insertion holes, as well as anything stowed under the Atlas during shipping.

Attach the provided leveling feet to the legs and adjust accordingly until the table is level.

Fig.1.1a

Remove the four brackets used to stabilize the gantry during shipping using a M5 allen wrench. (Fig. 1.1a)



I. TECHNO ATLAS SERIES CONNECTIONS:

The Techno ATLAS Series CNC Router is powered by 208-230 V, 3 Phase. Unless specially requested, the electronics require 3-phase power.

1.1

The Electronics are housed in the large NEMA enclosure as shown in Figure 1.1. When unpacking the machine avoid twisting the cable conduit that guides the cables to the motors.



Fig. 1.1

1.2

Open the rear of the controller using the provided key located around the emergency stop button or inside of the tool box.

You will now have access to the electronics that drive the CNC. (shown in Fig. 1.2).

The terminals for the power connection are located at the bottom of the box (shown in Fig.1.3)



Fig. 1.2

1.3

Have a licensed electrician connect the 208-230 V, 3 ph power, to the shown terminals (Fig 1.3). Make sure that all local electrical codes are obeyed. For single phase power, connect power to L1 and L3 only.

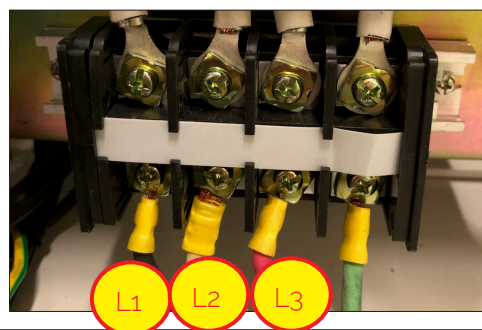


Fig. 1.3

1.4

Unpack the hand-held controller (shown in Fig 1.4) and carefully attach this to the controller board. (shown in Fig 1.5).



Fig. 1.4

1.5

Guide the cable through the hole on the side of the enclosure and attach the hand-held controller to the DB 15 terminal. (Fig 1.5)

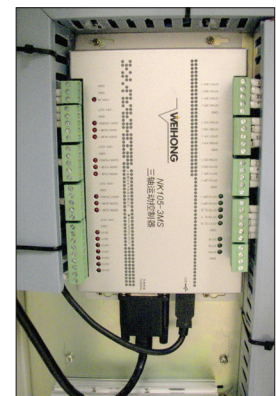


Fig. 1.5

1.6

If the machine has a vacuum hold down pump, there is a matching connector that will plug into the controller box (shown in Fig.1.6).

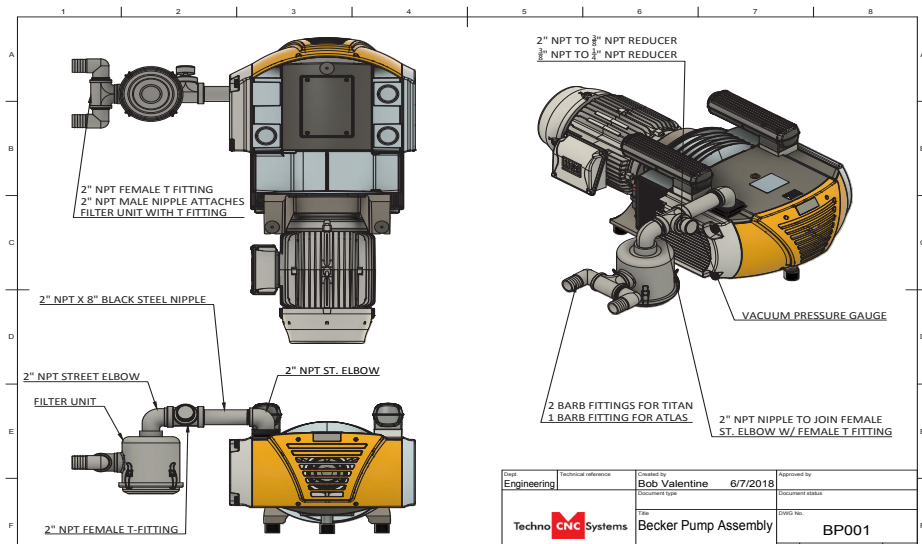
Vacuum Starter Connection



Fig. 1.6

III. Vacuum Pump Installation

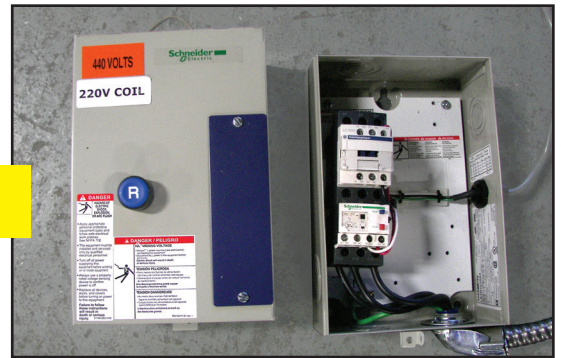
Single Phase Control Box connects to L1 and L2 and GND



Single Phase Control Box

Pump/Blower Motor Starter Box & Connector

NOTE:
The cover was removed from Motor Starter.



3 Phase Control Box

You will need to have the electrician connect AC power (208-230V 3PH) as specified on the unit to the Motor Starter or Single Phase VFD at L1, L2, L3 and Ground (PE).



WARNING: Direction of Pump Rotation is critical. Briefly start pump and check rotation (arrow on casing). Exchange phases if rotation is incorrect. IF YOU RUN THE PUMP/BLOWER CONTINUOUSLY IN THE WRONG DIRECTION, THE VANES WILL BE DAMAGED.

The starter box will have a round silver connector attached to a black cable coming out of it. The connector plugs into the socket on the side of the machine (shown in Fig. 1.7). The cable provides the signal to the starter coil to turn the vacuum pump on.



Fig 1.7 Matching connector that will plug into the controller box (shown left)



IV. Enabling the Atlas Series Machine

4.1 Control Panel Functions

Figure 4.1b shows the buttons and their functions.

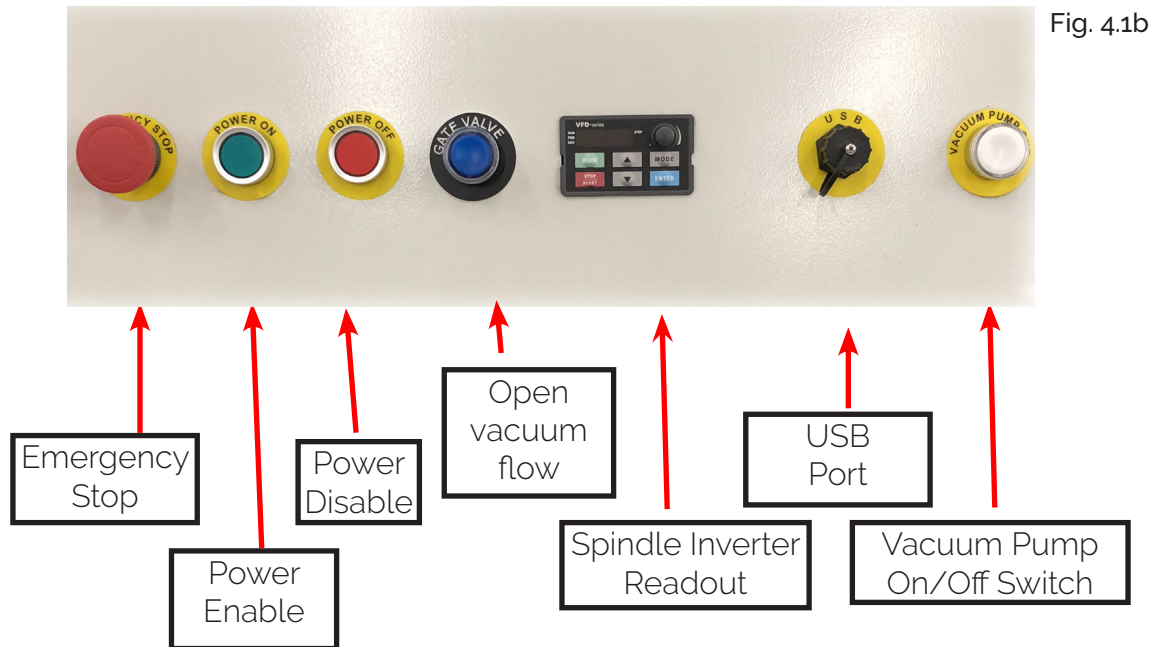


Fig. 4.1b

4.2 Powering On



Fig. 4.2a

Fig. 4.2b



Turn the machine on by turning the main power disconnect switch (Fig 4.2a) to the upright position. Note: Enclosure doors must be closed for system power to be enabled.

Power is now applied to the controller box. The red light will now light up indicating the machine is powered. Press the green button (fig. 4.2B) to enable system.

If system does not enable and red power button remains illuminated, check the Emergency Stop button to make sure it is not depressed. Turn Emergency stop counter clockwise to release.

Once the machine has been powered up and adequate air pressure has been applied, the PC and monitor may be turned on and booted up.

When the PC finishes loading, the Techno CNC Interface may now be launched.

Run the program by double clicking the Techno CNC Interface icon on your Windows desktop.



Techno CNC Interface

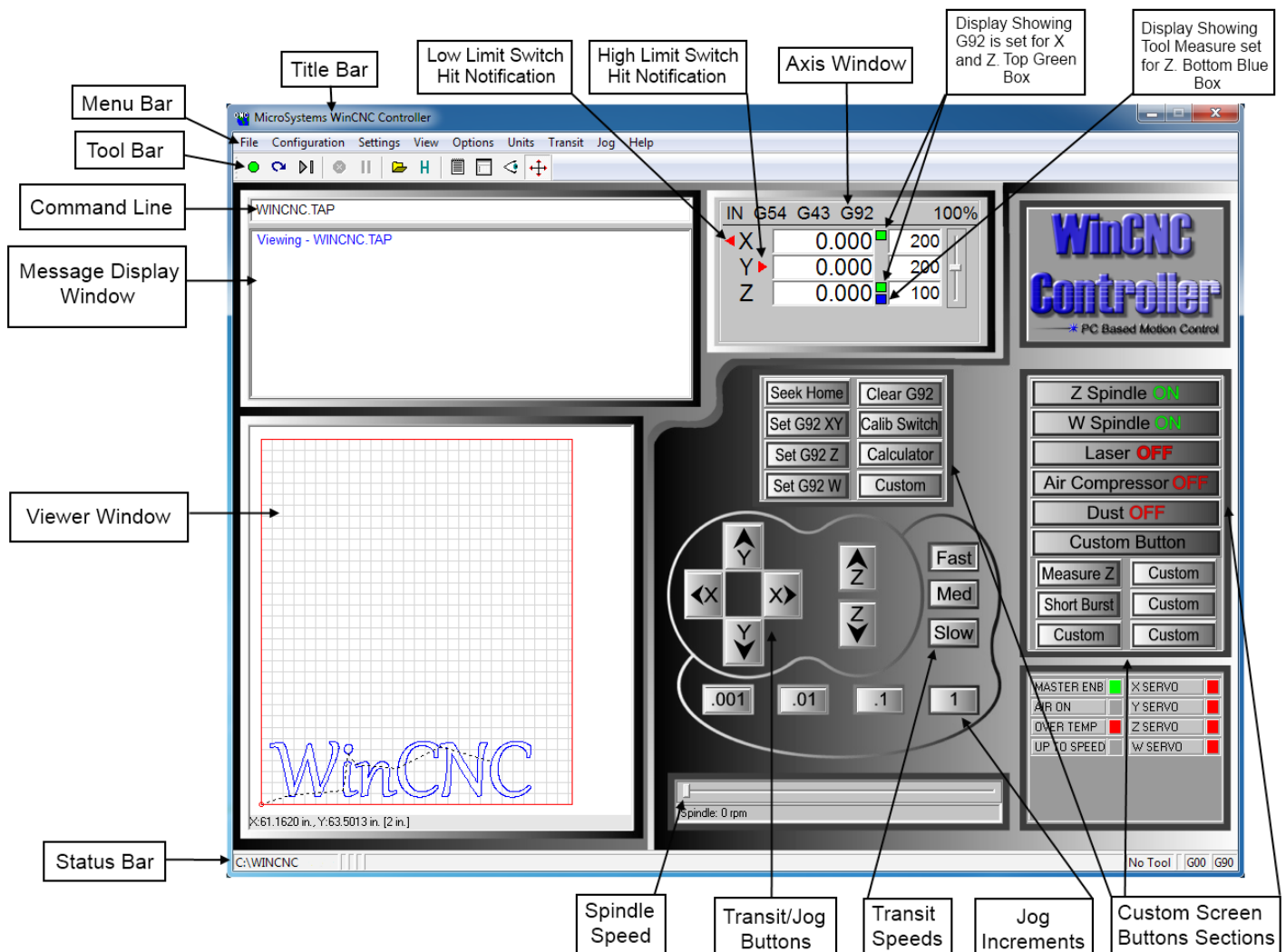
Techno CNC Software Controller: Overview

The Techno CNC 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. The Techno CNC 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. The Techno CNC 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.

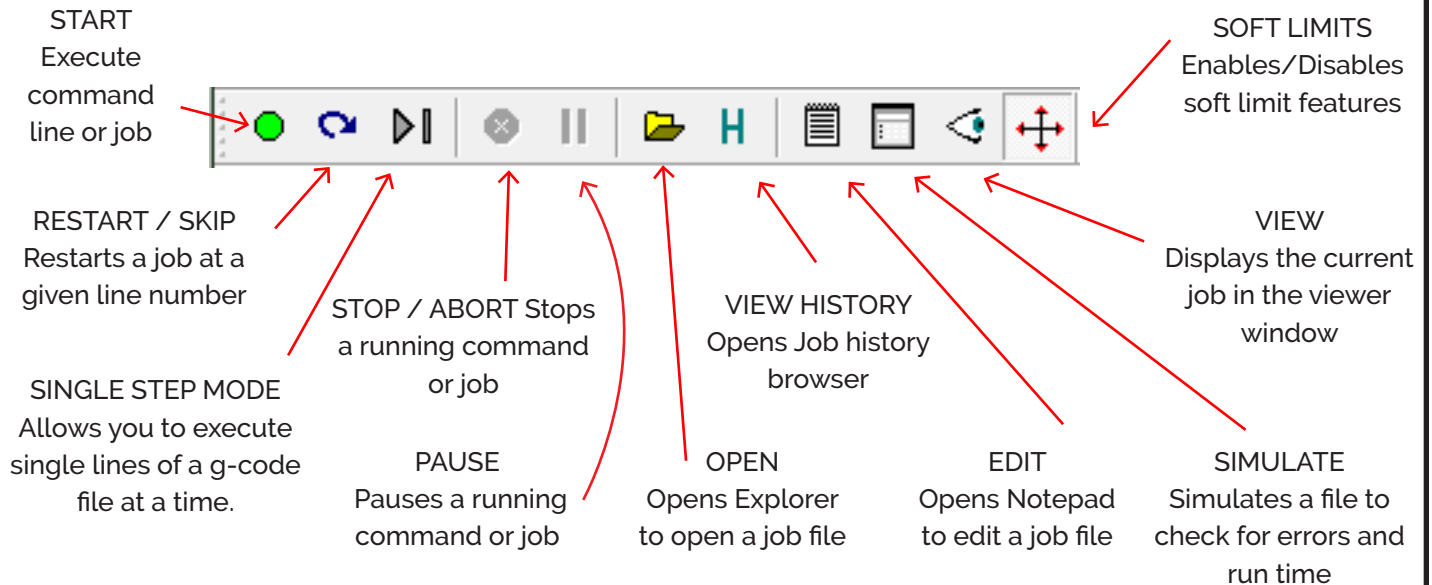
Program Operation: Overview

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



Tool Bar

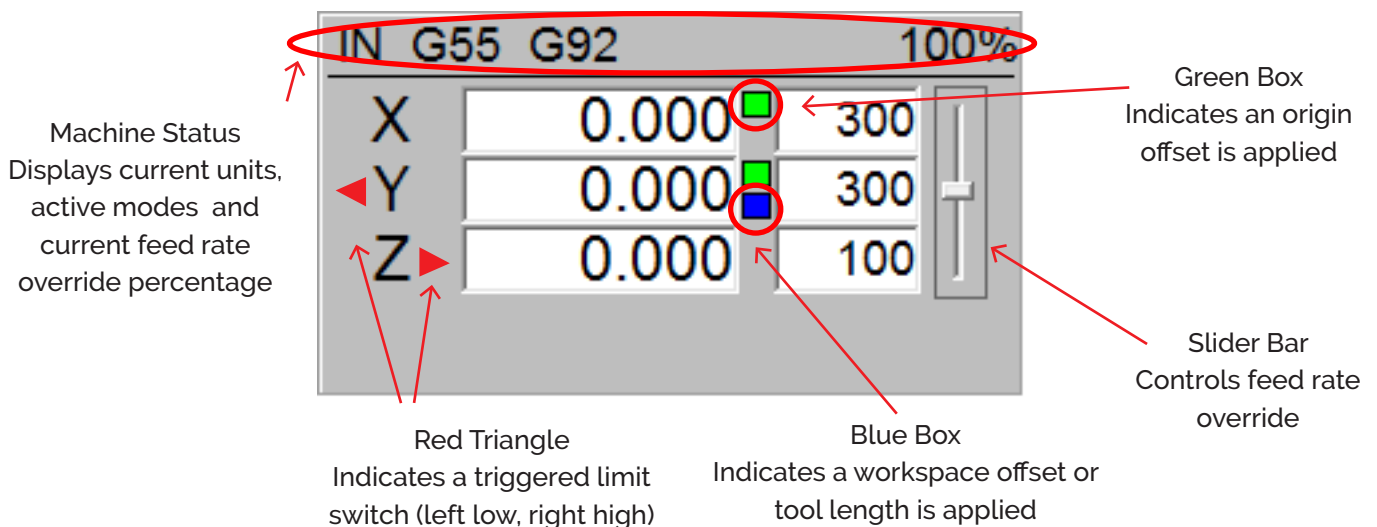
The tool bar is a collection of shortcut buttons that perform specific actions.



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.



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 run. By default, the viewer can also display a G-Code file line by line as it is run.

Go rapid moves are shown as a black dashed line. G1 feedrate moves are displayed as a solid blue line.

To view a file before you run it, open the file in the command line and then press the viewer button on the toolbar.

Once the object is loaded into the viewer the following controls can be used:

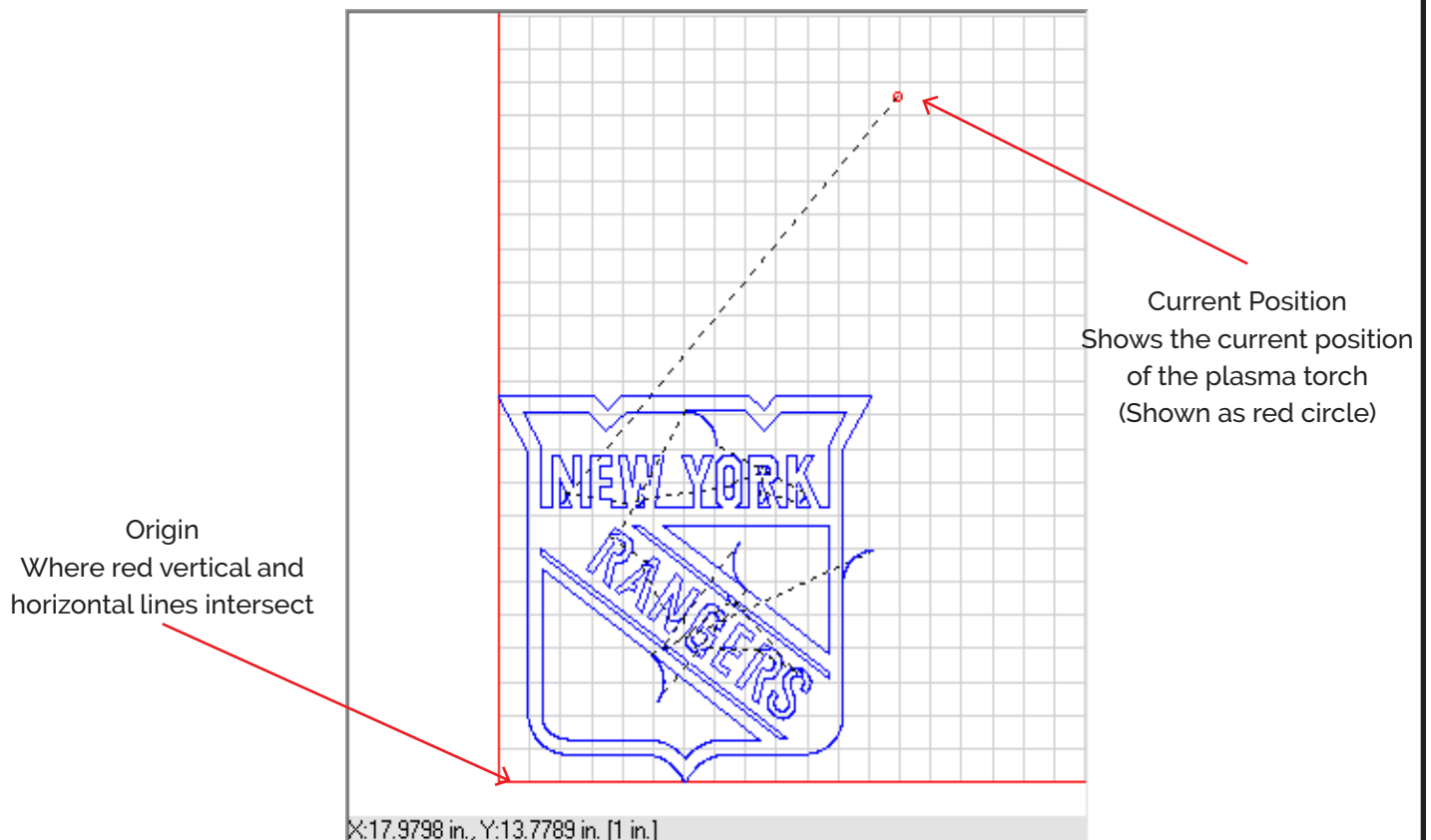
Zoom In - click the left mouse button.

Zoom Out - 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



Function Buttons

These buttons perform preset functions. They are used as shortcuts for many G-code operations.

Home All
Finds the machine's limit switches. Should be performed at start-up.

Park
Moves the machine to a clearance position



Clear
Clears all offsets.

Go XY0
Raises Z to its high limit and sends the machine to XY=0

Set XY
Sets the XY origin at the current position

Auto Z=0
Starts automatic touch off cycle to set Z=0

Man. Z=0
Sets the Z origin at the current position.

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.

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.

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.

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

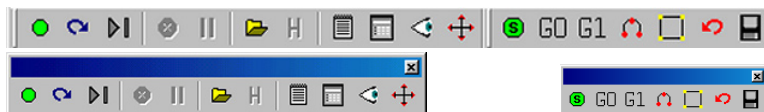
Help Topics (F1) - Activates the in-program help menu.

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.



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.

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.

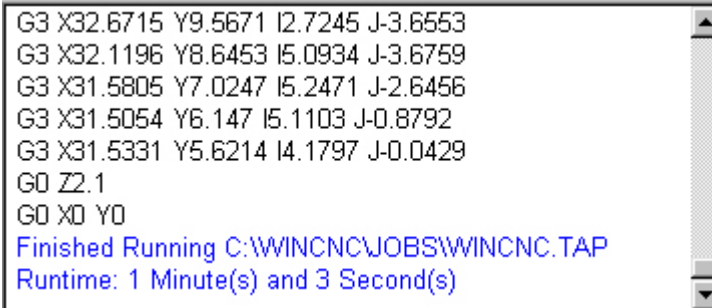
Save - Allows you to save the manual digitized file in either G-Code or DXF file format.

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.

Program Shortcuts

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

<u>Tool Bar Shortcuts</u>	
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
<u>Feed Rate Override Shortcuts</u>	
INSERT	Increases override rate
DELETE	Decreases override rate
CTRL+either	Resets feed rate to 100%. No over-ride settings
<u>Manual Digitize Shortcuts</u>	
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
<u>Spindle Speed and Laser Power Shortcuts</u>	
Spindle Speed Alone or Laser Power Alone	+ - (plus and minus)
Spindle Speed and Laser Power Together	Laser Power uses + - Spindle Speed uses Shift + -
<u>CTRL+(Function Key)</u>	
See "ctrlf#=" in configuration settings	

III. Operating Tutorials

3.1- Machine Startup - Homing

When starting the machine and WinCNC software, the first thing the operator should do is "Home" the machine. This allows the operator to check the functionality of the limit switches, and reference the machine so that previous origin positions can be used. Press "Home All" to home the machine.

Home All

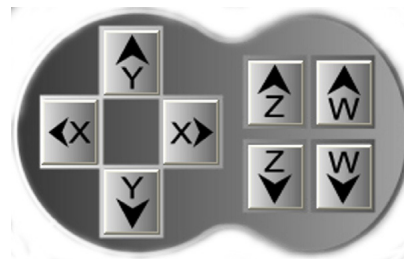
3.2- Manual Operation

The WinCNC controller supports two types of manual movement; Transit and Jog Increment modes.

Transit is a smooth, continuous motion and Jog Increment moves the machine in definite, precise steps.

Jog Buttons

These buttons are what allow you to initiate a manual jog movement. In jog mode, holding the button down will continuously move the machine. In step 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 step jog mode will move the machine only once.



Speed Buttons

When using these buttons, the machine will now be in jog mode and will move at a speed indicative of the button pressed.



Jog Increment / Step Buttons



When using these buttons, the machine will now be in Jog Increment/Step mode and will move at the distance specified on the selected button.


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.

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

3.3- 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 (IN), any currently active modes (G54), and the current feed rate override percentage (100%). 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 that a zero position has been set for that axis. A blue box displayed to the right of the position box indicates that an offset has been set for that axis. This is often the Z axis, indicating that the tool length has been learned.

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

To the far right side of the axis window is the feed rate override control slide bar.

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.

3.4- Park Position

Use this button to send the machine to a known Park position. The position is set to be toward the back of the table, convenient when loading and unloading material from the bed of the machine.

In its usage, the Z axis will lift up out of the way to its home position and then the machine will rapid to the park location.

Press the "Park" button to send the machine to the park location.



The location of the position can be adjusted in the "Park.mac" file in the WinCNC install directory.

3.5- Funtions



Main Function Windows

This frame contains the fundamental operations for the CNC machine. These buttons are discussed in detail throughout the manual.

Operator Panel

This frame contains the "Start", "Pause" and "Abort" controls.

Start - Executes the loaded the G-code file

Pause - Pauses a currently running G-code file. Press "Pause" again to continue

Abort - Stops/Cancels any machine movement. This includes running g-code files, tool changes, tool measuring cycles



Output Control

This frame contains the Output Override Controls. The buttons show the current status of the output. Click the button to change the status. For example, Off to On or Down to Up.

These are overrides, so they can only be performed when the machine is NOT moving.

When turning the spindle on and off, the spindle MUST have a tool loaded in the chuck and the RPM can be controlled by the slider bar at the bottom of the screen.



Status Indicators

This frame houses the machine status indicators.

If the described input is found, the box is green, if not, the box is red.

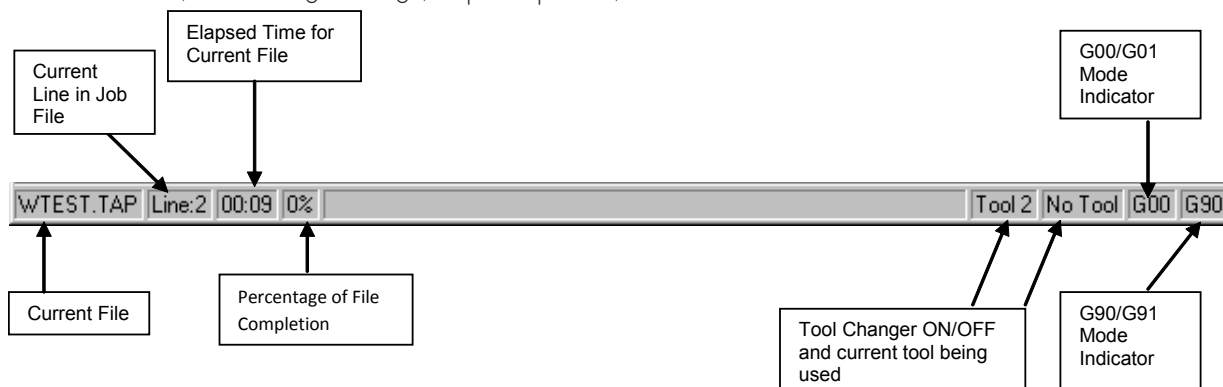
The Air pressure indicator and Touch ESTOP are necessary for machine operation.

Tool present and Shroud up, indicate if a tool is loaded and whether or not the shroud is in a safe position for tool changes.



Below that, in the main window status bar, will be the indication of what tool the machine thinks is in the spindle, "Tool 3" in this case.

It will also show, modes G90 vs G91, Rapids speeds, and units G20 vs G21.



3.6- 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 3.6a the word "BOXER" is located away from the XY zero.

In Fig 3.6b the object represents the material the letters will be cut from. The spindle, vertical with point, should be jogged to the corner of the material.

In most situations, the user will set X0 and Y0 together, and Z0 separately.

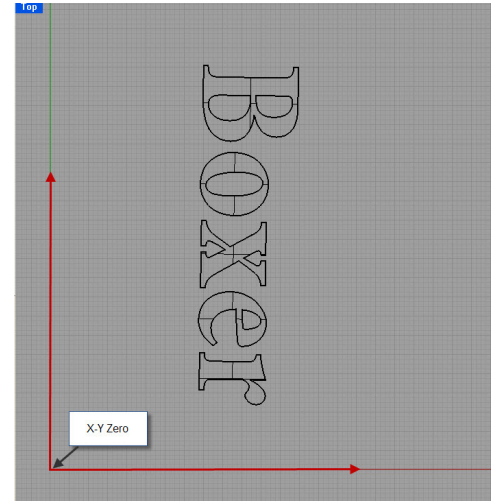


Fig. 3.6a

Setting the XY Zero Position

Jog the machine using the directional arrows so that the center of the bit is located directly above the position you would like to set as XY0. It is useful to utilize a small bit or a bit with a fine point when accurately setting this position.

Once the machine is in location, press the "Set XY" button, the coordinates will change to 0.000 and a green box will appear indicating the XY zero is now set.

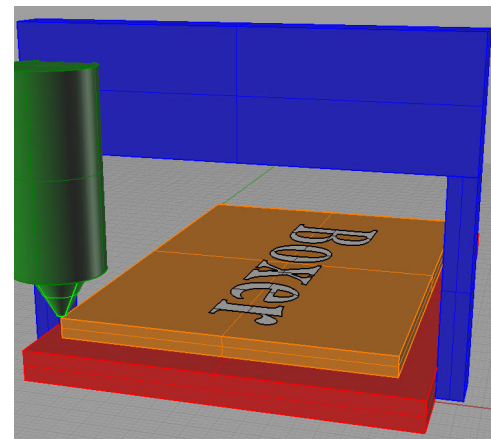


Fig.3.6b

IN	G54	G43	G92	100%
X	0.000		300	
Y	0.000		300	
Z	4.974		50	
B	0.000		25000	
C	0.000		25000	

Fig.3.6c

Figure 3.6c shows the axis window after setting the XY0 position.

Notice the green boxes indicating that the position has been set. This is also apparent by the G92 located at the top.

The coordinates now read 0.000

Setting the Z-axis Zero Position for Router usage.

The Z-axis zero position can be set in two ways:

Manually-

Use the Z-axis directional arrows to move the tip of the router bit to the location of your Z=0 position as indicated by your G-code program. This could be either the top of the material, or bottom of the material. Switch to Step Mode to slowly move the machine into position.

Once the bit is in position, use the "Man. Z=0" button to set this location as the Z origin for the Router. The Z-axis coordinate will change to 0.000 and a green box will appear indicating the Z zero is now set.

A rectangular button with a black background and white text that reads "Man. Z=0".

Using Touchpad-

Place the floating touchpad on top of the surface that will be your Z=0 position as indicated by your G-code program. This could be either the top of the material, or bottom of the material (top of the sacrificial spoilboard).

Using the directional arrows, center the router bit above the touchpad. Make sure there is some space between the bit and the pad.

Once the router is above the touchpad, use the "Auto Z=0" button to use the touchpad to set the Z origin of the router. The head will automatically descend, touch the pad and then retract. The Z-axis coordinate will change to 0.000 and a green box will appear indicating the Z zero is now set.

A rectangular button with a black background and white text that reads "Auto Z=0".

NOTE: Make sure your material is safely and properly secured. If you are using a vacuum pump, make sure the vacuum is on BEFORE setting your Z=0 position.

Clearing the Zero Position

Sometimes the position the user has set could be incorrect and needs to be redone. The operator could use the "Clear" button to start over or could simply choose the "Set XY" or "Set Z" buttons again to overwrite the current values.

A rectangular button with a black background and white text that reads "Clear".

Go to Zero Position

The user may want to physically see where the current XY zero position is set; whether to check its alignment to the material to be cut or to double check the position. The operator may do so by selecting "Go to XY0". The Z axis will home up for safety clearance and the spindle will move to XY0.

A rectangular button with a black background and white text that reads "Go XY0".

3.7- Running a G-code File

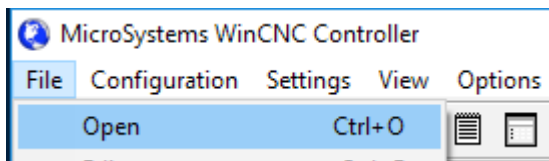
Once the XY and Z zero positions have been set, the machine is now ready to execute a G-code program.

Make sure your material is properly secured, the correct bit is inside the spindle and that the operator has the proper safety equipment on.

Warning: It is advised for first time users to run the file in the air to become more familiar with operations.

Loading a G-code File

To load a G-code file to be cut, select "File" from the Menu bar and then choose "Open".



Then using the explorer window, navigate to the G-code file you would like to run and choose "Open".

Once the G-code file has been selected and loaded, the network location of the file will be shown in the command window.



After loading a File

Once the file is loaded, the operator can View, Simulate, and then run the file.

View

(File>View) or



This option displays the toolpaths of the G-code file in the viewer window of the controller screen. It allows the user to see exactly what is being cut on a simulated view of the machine bed.

Simulate

(File>Simulate) or



This option simulates a file to check for errors and estimates run time.

Run

(Enter key) or



This option displays the toolpaths of the G-code file in the viewer window of the controller screen. It allows the user to see exactly what is being cut on a simulated view of the machine bed.

V. Controller Commands

Command Reference

Parameters in [Brackets] are optional.

XYZWIJ . . .	Axis Specification X# Y# Z#
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.	
Example: XYZ is equivalent to X0Y0Z0	

F	Feed Velocity - F#
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.	
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	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.	

{ }	Braces
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.	

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.

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.

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.</p> <p>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.</p> <p>Example: X1Y1 is equivalent to G1 X1Y1 if the mode is G1.</p> <p>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>	

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] WARNING: Never use Dwell to stop the machine while changing parts! 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>	

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.	

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.	

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>	

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.	

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.	

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.	

L8	Turns Backlash Compensation On, Restore INI values for all axes.
Examples: L8XY - Turn Backlash Compensation On, Restore INI values for specified axes. L8X.001 - Turns Backlash Compensation On, Set new value for specified axis.	

L9	Turns Backlash Compensation Off.
----	----------------------------------

L10	Cut Array - L10 R# C# X# Y# E#
Sets up array cutting. Repeats all code following until the end of the file or another L10 is reached. Program pointer is moved back to initial L10. The head is moved to the next column or row specified by XY at rapid velocity. Cycle repeats until all array points have been cut. L10 specified without RCXY values can be used to end an array cut. Lines that follow will not be repeated. If an 'E' is found on the L10 command line then WinCNC will skip the final move back to the starting position at the end of the array. If no 'E' is found then WinCNC will move back to the starting position at the end of the array.	
**A G92XY is required after the L10 Line for the cut array to work. X# is the units to move the x axis before starting the next cut. Y# is the units to move the x axis before starting the next cut. R# is the number of rows. C# is the number of columns.	
Example: L10 r5 c5 x4 y4 e1 or L10 r1 c4 x6 y0 G92xy G92xy G90 G90 G01 x3 m98 Texas.tap G01 y3 G01 x0 **End with L10 if using more G01 y0 than one L10 array.	

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.</p> <p>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.</p> <p>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.</p> <p>Examples:</p> <p>L13 ZW (Z is controlling axis, W follows Z.)</p> <p>L13 WZ (W is controlling axis, Z follows W.)</p> <p>L13 XY (X is controlling axis, Y follows X.)</p>	

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.	

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 Knifemode= . 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.	

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. NOTE: 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. ONLY 1 PARAMETER CAN BE USED.	

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

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.	

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#.	

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.	

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#
<p>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.</p>	
L250	Vector Laser Off
Disables vector laser cutting.	

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.	

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"
[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. < 16 characters long.	
L500.2	- L500.2 T# N# A# C# O# H# M#
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.	
L501	Store Local Positions - L501 [X, Y, Z...] [P H T] L# C# N"Name"
[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. < 16 characters long.	

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.

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#
<p>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. Only one axis can be scanned at a time: X or Y. Uses up/down, and side-to-side probe type.</p> <p>FILENAME.EXT - replace with your filename. Do not surround with quotation marks.</p> <p>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.</p> <p>Y# - works like X#, except on the Y axis.</p> <p>I# - is the maximum increment for the horizontal axis.</p> <p>K# - is the vertical increment (how high the probe raises).</p>	

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.	

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. Note: For use with a Drill Bank 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.	

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.	

M28	Disable Limit Abort
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.	
M29	Disable Limit Abort
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.	
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#
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.	

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# ...
<p>M37.3 is used to shift workspace offsets for tool measures, tool measure switch heights, or individual axis.</p> <p>S# is the shift amount to be used for tool measures and tool measure switch positions. (usually a negative #).</p> <p>H# is the H register number to be shifted by the shift amount. H0 will shift all H registers.</p> <p>T# is the tool number height to be shifted by the shift amount. T0 will shift all tool numbers.</p> <p>M# is the measure switch height to be shifted by the shift amount. M0 will shift both switches.</p> <p>X# Y# Z# ... shifts each axis workspace offsets by the specified value.</p>	
M98	Subprogram Call - M98 <filename>
<p>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.</p>	
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.	

VI. 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.

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.

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.

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.

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.

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.

VIII. ATLAS Machine Lubrication

8.1 Lubricating the X-Y Rack and Pinion

Lubrication is important with rack and pinion gearing systems. A thin film of grease should always be present on the contacting tooth flanks to minimize metal to metal contact.

Lithium grease lubrication is recommend over oil, as the oil lubrication will flow away from tooth flanks.

The grease should be applied to the rails at regular intervals, depending on the usage of the machine. Use a small brush to coat both rails on the side of the Y-axis and the single rail across the X-axis. Fig 8.1

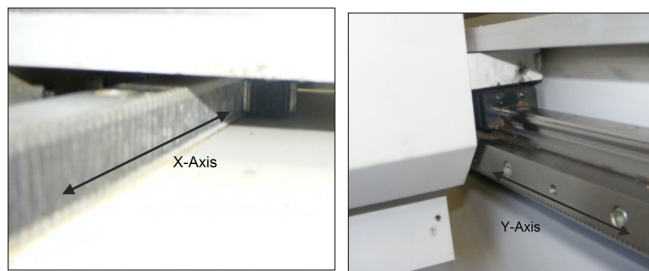
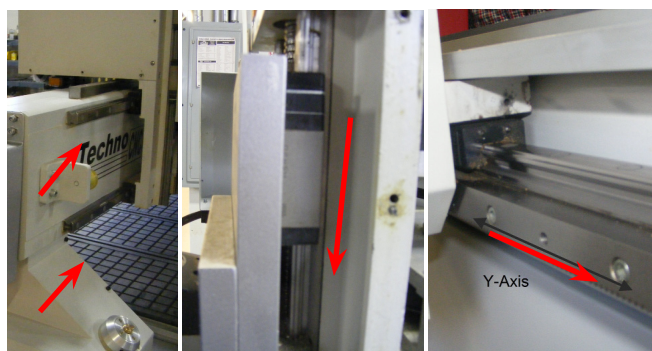


Fig 8.1

8.2 Lubricating the X-Y-Z Rails

The rail carriage bearings are sealed and protected with wipers. The rails should be lightly oiled to allow smooth operation. Avoid a build up of debris on the rails by blowing them off with air, or wiping them down with a rag. The rails do not need to be lubricated as often as the rack, once a month should be sufficient.



X-Axis

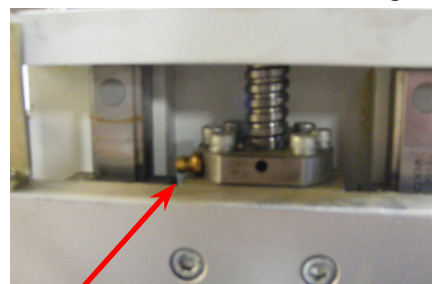
Z-Axis

Y-Axis

Fig 8.2

8.3 Lubricating Z Ballscrew

The Z axis uses a ballscrew and ballnut instead of a Rack and Pinion. The ballnut has a nipple for applying lubrication to the mechanism. Fig 8.3a



Lubrication Point

Fig 8.3a

Lithium grease is pumped into the lubrication point with the application gun provided with the machine. Fig 8.3b

Fig 8.3b

8.4 Recommended Lubricants

Lithium Based Grease:

Alvania Grease No. 2(Shell)

or Equivalent.

Techno Part No.

H90Z00-8670T8

Oil:

Vactra No. 2s(mobile)

Tonner Oil or Equivalent.

Techno Part No.

H90200-LUBE002

Oil and Grease Kit:

Techno Part No.

H90Z00-LUBEKIT2

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



Betriebsanleitung
Operating Instructions
Instructions de service
Istruzioni d'uso
Handleiding
Instrucciones para el manejo
Manual de instruções
Naudojimosi instrukcija
Kasutusjuhend
Lietošanas instrukcija
Οδηγίες χρήσης
取扱説明書
사용설명서

Driftsinstruks
Driftsinstruktioner
Käyttöohje
Driftsvejledning
Instrukcja obsługi
Kezelési útmutató
Návod k obsluze
Navodilo za uporabo
Návod na obsluhu
El Kitabi
Инструкция по эксплуатации
使用说明书

VTLF 2.200
VTLF 2.250

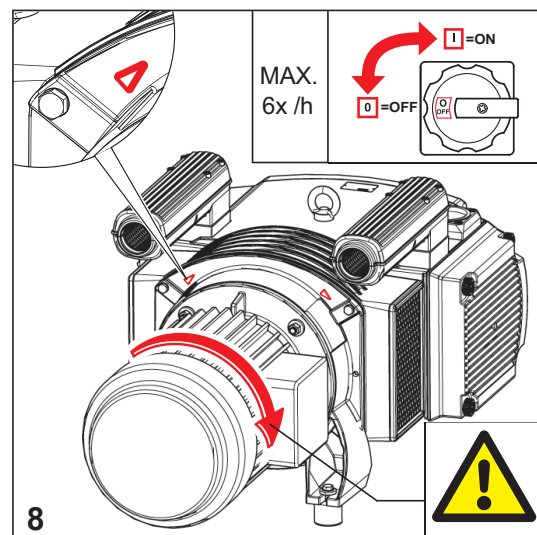
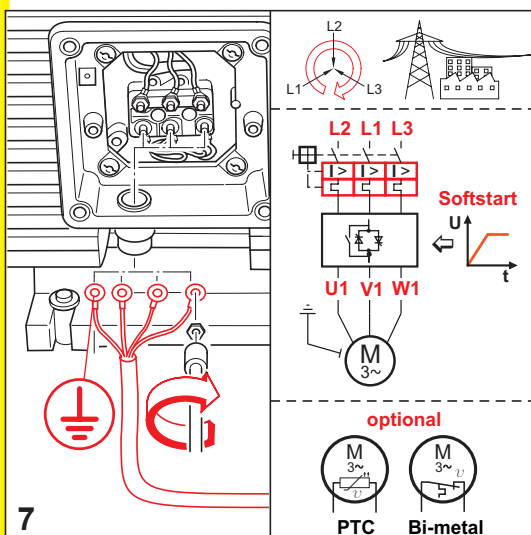
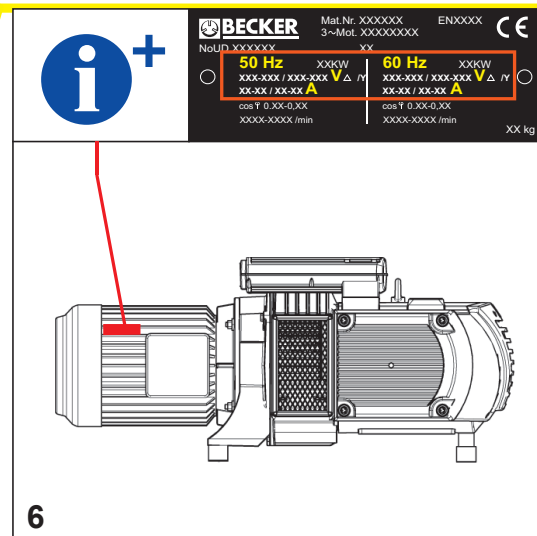
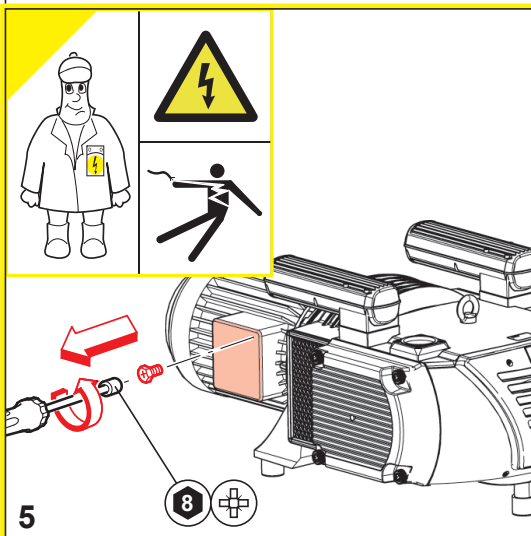
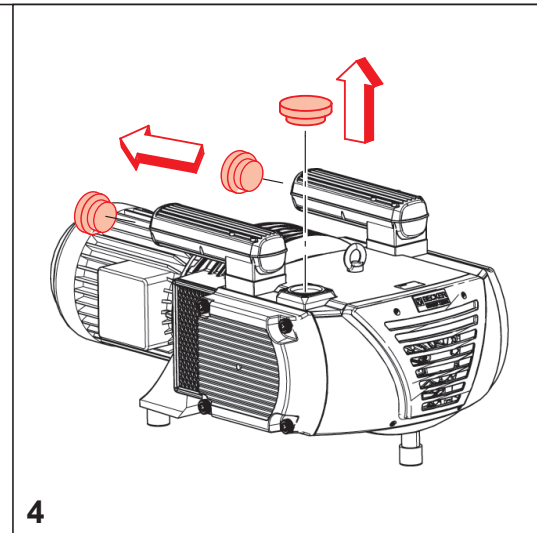
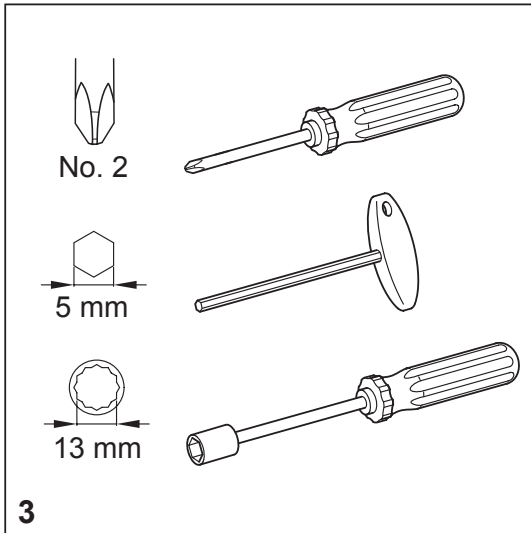
98/37 EG
2006/95 EG

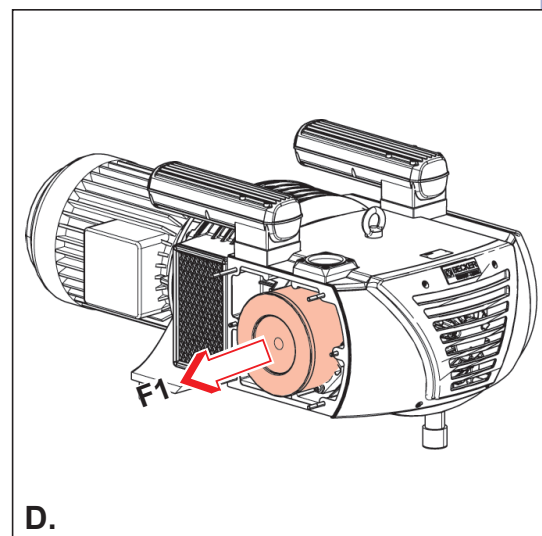
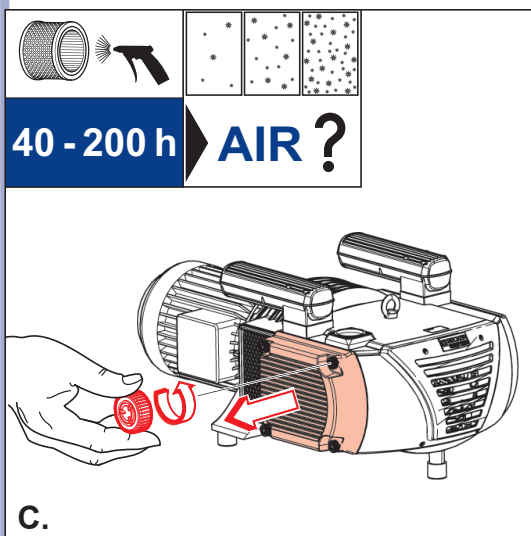
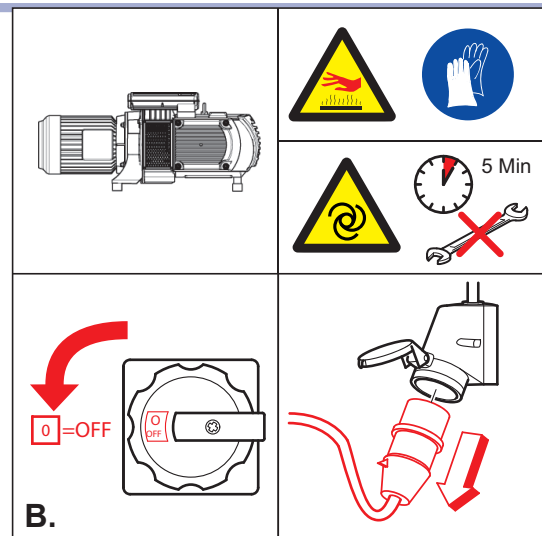
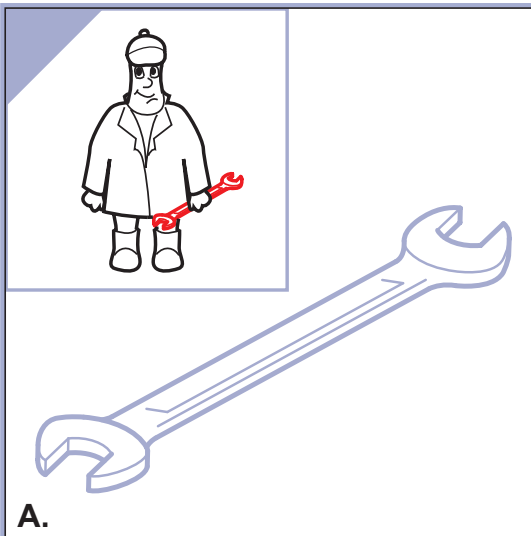
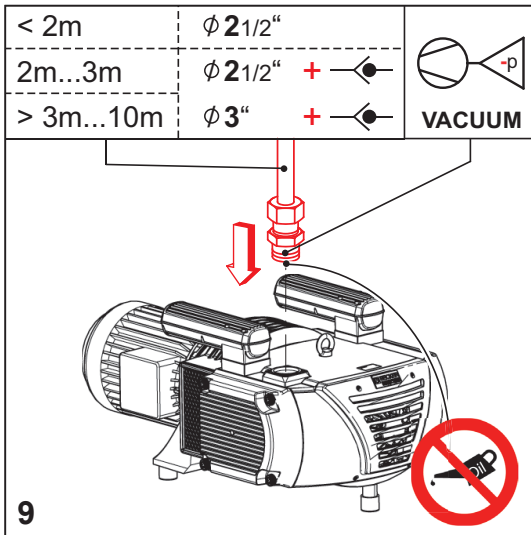


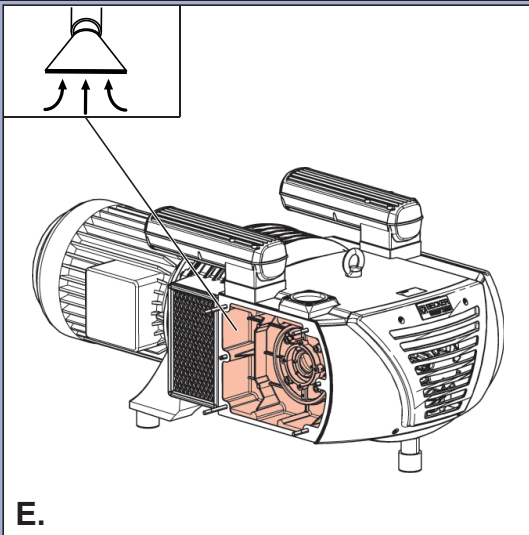
 	 	 MAX. VACUUM	 MAX. VACUUM mbar
		 AIR	 MAX.
		$L_{pA} = 75-77 \text{ dB(A)} - 50\text{Hz}$ $L_{pA} = 77-79 \text{ dB(A)} - 60\text{Hz}$ $K_{pA} = 3 \text{ dB(A)}$	

 			$A > 400\text{mm}$ $A > 16"$	 $> 5^\circ\text{C}/41^\circ\text{F}$ $< 45^\circ\text{C}/113^\circ\text{F}$	 max. 90%	 max. 800m
1 	2 					

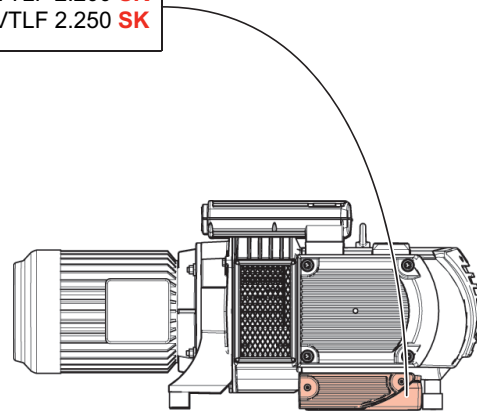
BPC 28100052202 04/09



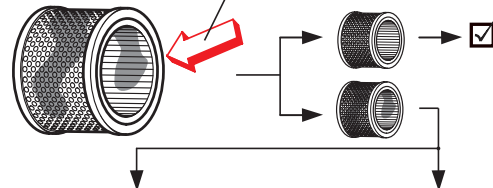
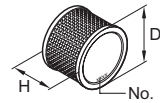
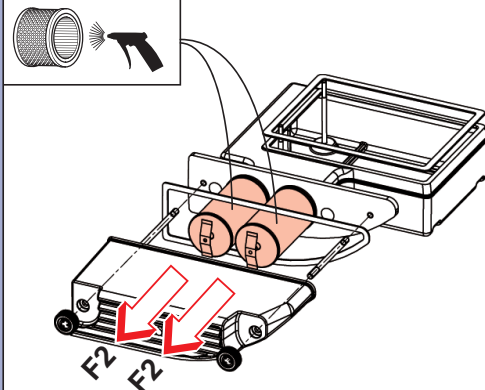




VTLF 2.200 **SK**
VTLF 2.250 **SK**



VTLF 2.200 **SK**
VTLF 2.250 **SK**



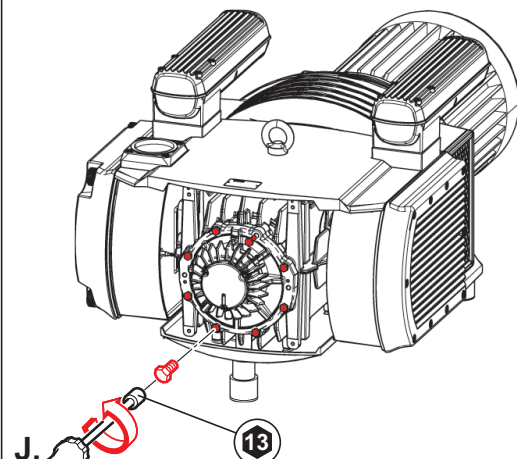
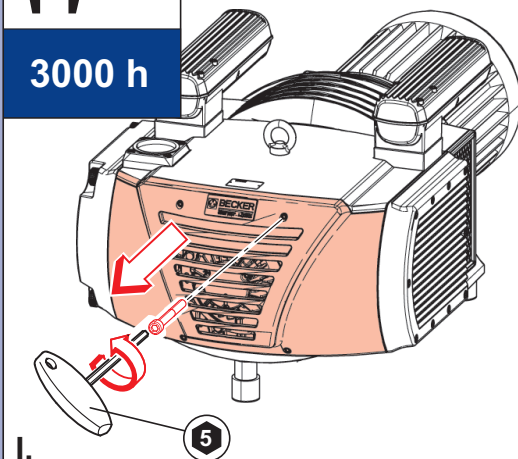
F1
D: 218 mm
H: 122 mm
No.: **909512**

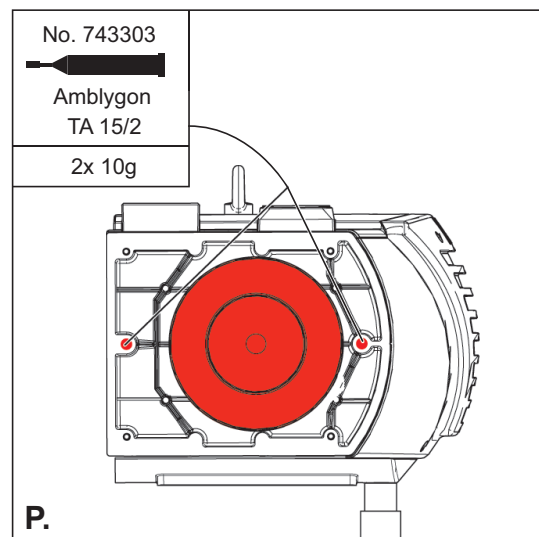
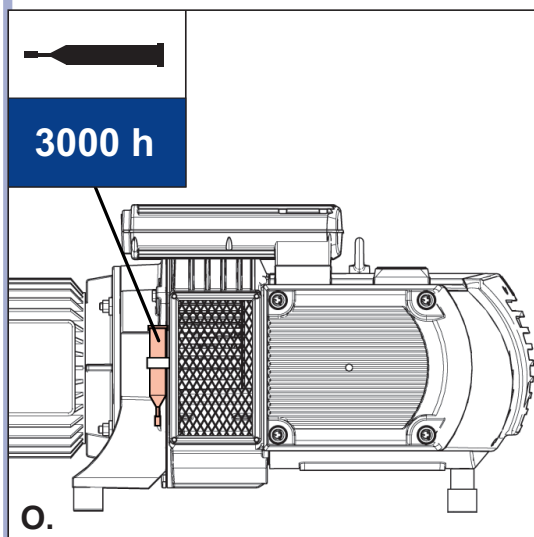
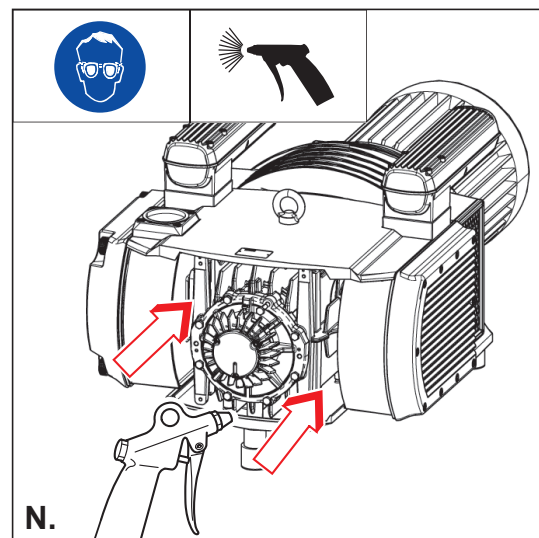
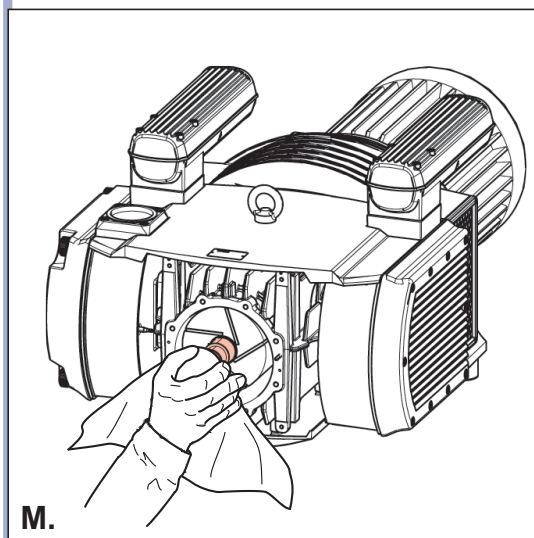
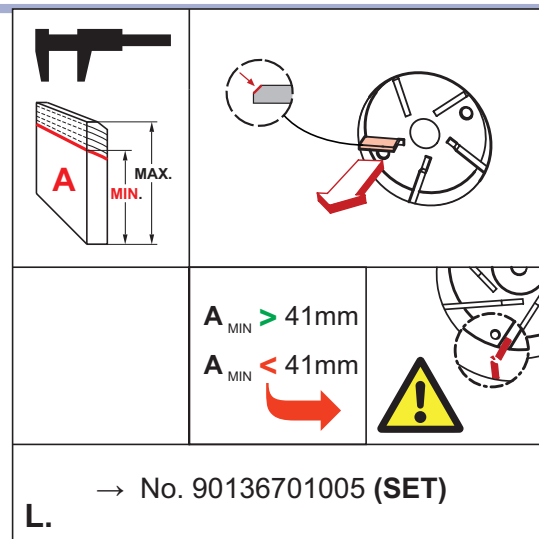
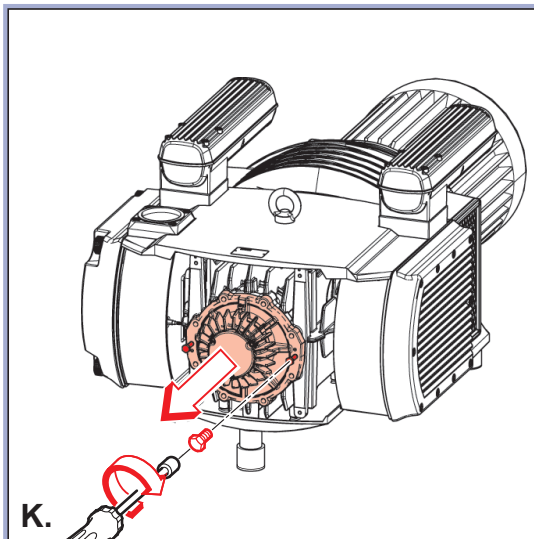
F2 (VTLF - SK)
D: 64 mm
H: 120 mm
No.: (2x) **909510**

H.



3000 h





TLF 2.250-2.500 Internal Filter Inspection

-Tools required-
Flashlight

ATTENTION

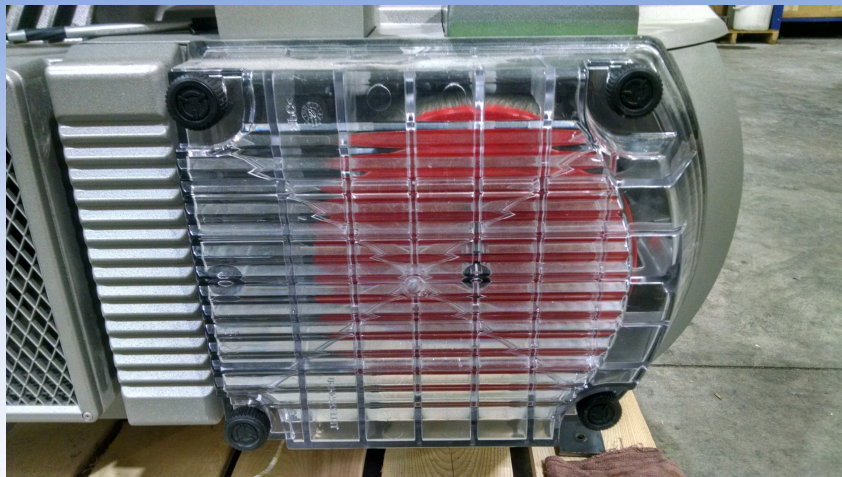
VISUAL CLUES REGARDING VTLF 2.250 FILTER MAINTENANCE SHOULD NOT ALWAYS BE THE SOLE INDICATOR OF WHETHER A FILTER IS "CLEAN".

THOUGH THE FILTER HAS TREMENDOUS SURFACE AREA, THE DEEP PLEATING OF THE FILTER MAY DISGUISE WHETHER THE FILTER IS CLOGGED.

A PERIODIC PHYSICAL INSPECTION SHOULD BE PERFORMED TO MAKE SURE THERE IS A GOOD FLOW OF AIR THROUGH THE FILTER.

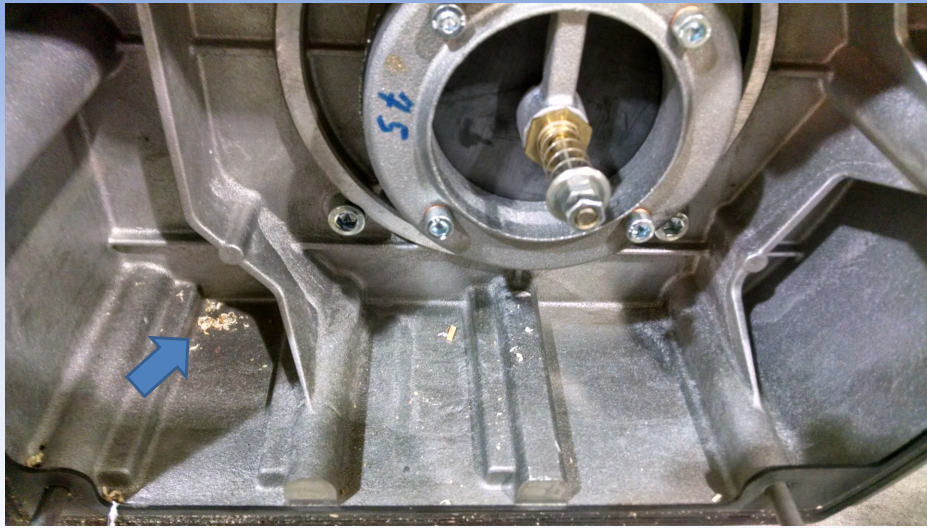
A CLOGGED FILTER IS ALMOST ALWAYS THE CAUSE OF PRE-MATURE VANE WEAR OR IN SOME CASES, PUMP FAILURE

-Remove the (4) black knobs by hand-



-Remove the internal filter and look for debris-

-Check for large debris deposits. This is an indicator that the filter caught the smaller particles-



-Use a flashlight on the outside of the filter-



If light **cannot** be seen on the inside, the filter is clogged and needs replaced.



-If you **can** see light, then blow out the filter using compressed air and replace-

- This needs to be a modest amount of light.
- Light should be present through each pleat.



Greasing TLF 2.200-2.360

-Tools required-
X1 – 7433050000
(50 gram grease gun)



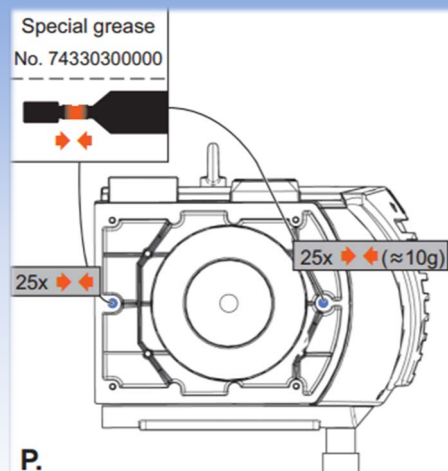
Author: Mike Ruff
Becker Pumps Corp.

Greasing instructions

The greasing instructions can be found on step "P." in the operation manual sent with each pump.

Or they can be found at
www.Beckerpumps.com

Bearings are to be grease every 3000 – 4000 hours



P.



Author: Mike Ruff
Becker Pumps Corp.

**All new units come with new grease guns.
(Found in either of the two places below)**



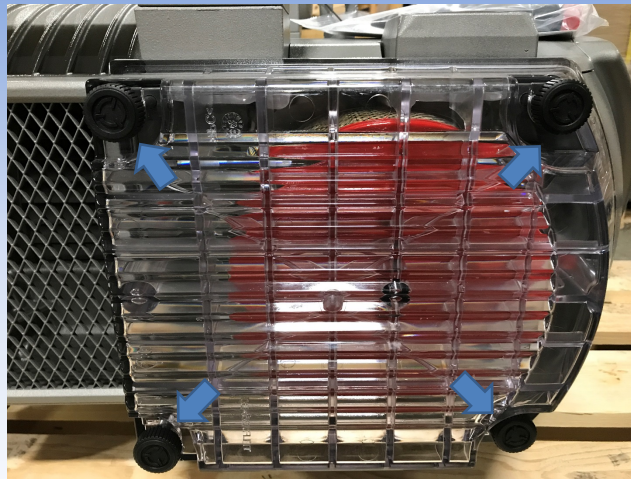
Author: Mike Ruff
Becker Pumps Corp.

GREASING PROCEDURE



Author: Mike Ruff
Becker Pumps Corp.

Remove the filter cover by loosening the black hand knobs.



Author: Mike Ruff
Becker Pumps Corp.

Remove the internal filter and replace if needed.

**Grease fittings are found next to the filter.
(Remove the red caps.)**



Author: Mike Ruff
Becker Pumps Corp.

Remove the black cap from the grease gun



Author: Mike Ruff
Becker Pumps Corp.

Prime all new grease guns by placing them at an angle against a hard surface.

Pump a few times until the grease is visible at the tip.



Author: Mike Ruff
Becker Pumps Corp.

Place the grease gun against the push fitting

Pump 10x into each bearing

(New or dry bearings = 25 times per bearing)



Author: Mike Ruff
Becker Pumps Corp.

**Once the pump is ran, the grease will evenly
distribute between the rollers and ball bearings.**



Author: Mike Ruff
Becker Pumps Corp.

DAILY MAINTENANCE

Check the machine before start up and clean it after every use.

Check to ensure chips and dust are not caught in the X, Y or Z ball screws. Dislodge all chips and thoroughly clean dust.

Make sure the machine bed is clear of obstacles.

REGULAR MAINTENANCE

Generally, this maintenance should be completed every month

Moving parts on the machine may loosen or displace over time from regular operation and dust that forms during regular machine operation will often stick to the lubrication oil used on the moving parts of the machine (ballscrew, linear rails). This dust can cause premature wearing and damage to the precision bearings.

Inspect the hardware of each component on the machine, checking for loose bolts in case they have loosened over time. Pay special attention to linear rails, bearing blocks, spindle plates, limit switches and gantry uprights.

Loose hardware can result in the machine going out of square, poor accuracy and repeatability, damage to components and broken wires.

Check to make sure that any wires are not being pinched or crushed by moving mechanical parts.

Listen for abnormal noises during operation, Grinding noises, squeals and/or banging sounds are NOT normal,

Clean grime and any accumulated lubricating oil off the guide rail and bearing blocks.

Turn on power to the machine and move it to clean and lubricate everywhere along the guide rail and rack and pinion.

Lubricate ball screw assemblies monthly to ensure peak mechanical performance.

Thoroughly clean components of any excess lubrication before fresh coating.

Ball screw assemblies should be lubricated via the grease nipple using a grease gun.

Check your machine for squareness and for backlash. Out of square machines and machines with backlash are indicative of a crash or loose hardware.

IX. Atlas Troubleshooting

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

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.