S3CNC User's Guide

S3CNC 4 Axis Router and Plasma CNC Control Software

Features

- Fully configurable CNC control software for Router, Plasma, and other CNC equipment supporting 2, 3, and 4 axis machines
- 100 Mbps Ethernet motion controller (Ether-Mach) for robust communication and smooth, high-speed motion
- Graphical display of machine position and parts seamlessly links relation between physical environment and S3CNC controller
- Programmable Goto Positions for easy loading and unloading of parts, machine maintenance, and more
- Runs standard G-Code files compatible with most CAD/CAM software
- Plasma: Torch Height Control (THC) for optimal cut quality, Material Table loads and saves plasma settings for various materials
- Router: Automatic Tool Changer (ATC) for quick and easy tool changes, On-the-fly spindle speed control for optimal finish, Tool Table loads and saves parameters such as length and diameter
- Inputs and Outputs: step/direction motion control, drive resets, drive faults, digital or analog spindle speed, CW/CCW spindle, coolant, vacuum, home and limit switches, jog, start, stop, E-Stop, ATC inputs, THC inputs, plasma on and up/down controls, custom inputs, general CNC outputs

Product Brief

S3CNC is the latest CNC motion control software from Stepper3 LLC. It is a fully configurable, graphical, CNC control software package that allows easy configuration for a variety of CNC machines. It is feature rich, yet simple to use, allowing advanced features such as automatic tool changing, while still allowing the operator to run a G-Code file in four easy steps!

While specifically targeted for CNC routers and CNC plasma machines, the software can be used for laser, water jet, mills, and any other 2, 3, or 4 axis CNC machines.

S3CNC is high in performance, functionality, and versatility, making it a powerful tool for any CNC application.

S3CNC Contents

- 1 S3LIC, Stepper3 S3CNC hardware license dongle
- 1 S3CNC Software License
- 1 S3CNC Software Installer and this User's Guide

Table of Contents

1 Section 1: Installation and First Time Setup

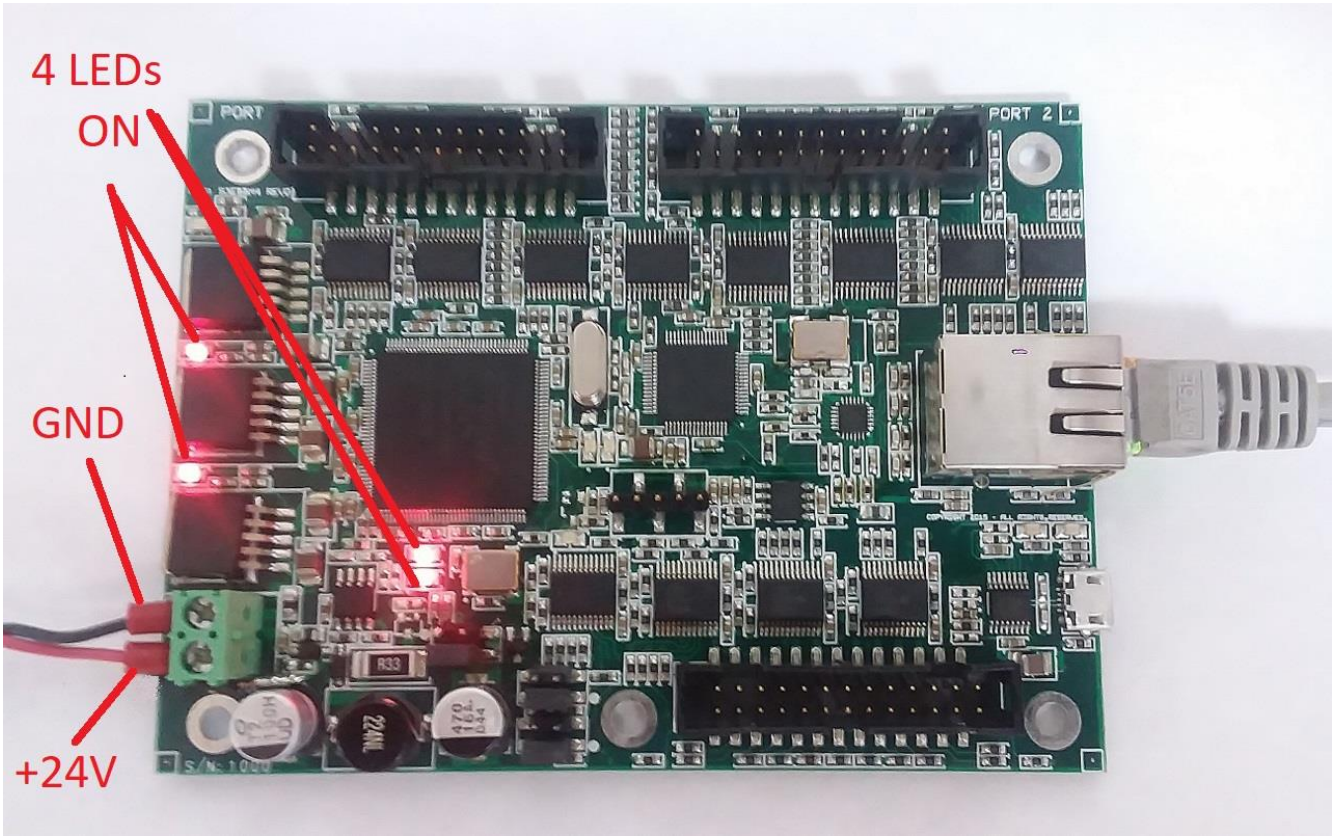
Before beginning this section, you should have the following:

- A computer
- The install package for S3CNC (S3CNC-Installer-X-Y.exe, where X and Y are the latest version numbers)
- An Ether-Mach board
- A 12-24v power supply for the Ether-Mach board
- S3LIC, The S3CNC licensing dongle

1.1 Setting up the Ether-Mach Board

Before running the S3CNC software, you must connect a powered Ether-Mach board to your computer; the Ether-Mach board is required in order to run S3CNC.

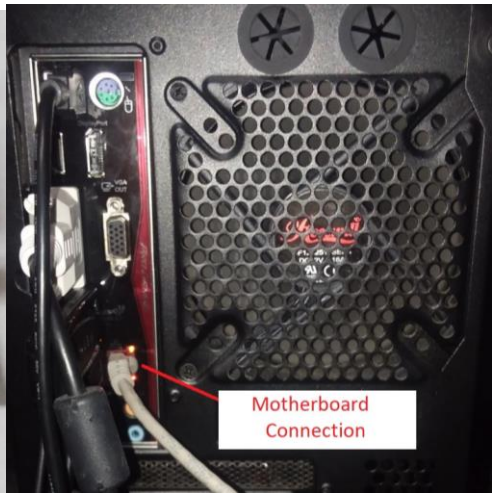
1.1.1 Step 1: Powering up the Ether-Mach breakout board



From a 9 to 24 volt DC power supply, connect the positive (e.g., +24v) line to the VM terminal and ground to the GND terminal of the Ethernet board. Turn on the supply and observe that the Power Indicator LEDs (they are along the same side as the power terminal block) on the Ethernet board light up, indicating that it was successfully powered. After about a second, the Board Ready and Power

Indicator LEDs will also light up, indicating that the board completed its initialization process and is ready to run.

1.1.2 Step 2: Connecting to your CNC System



Ether-Mach is designed to work with your existing CNC system; as such, it provides three parallel port headers. Plug your existing system into the parallel port headers. If you have uncommon pin assignments on the ports, do not worry – both inputs and outputs are fully configurable in the plugin and S3CNC configuration panels. After connecting your parallel ports, connect a CAT5 or CAT6 Ethernet cable to the Ethernet port of your board, and the other end to your computer.

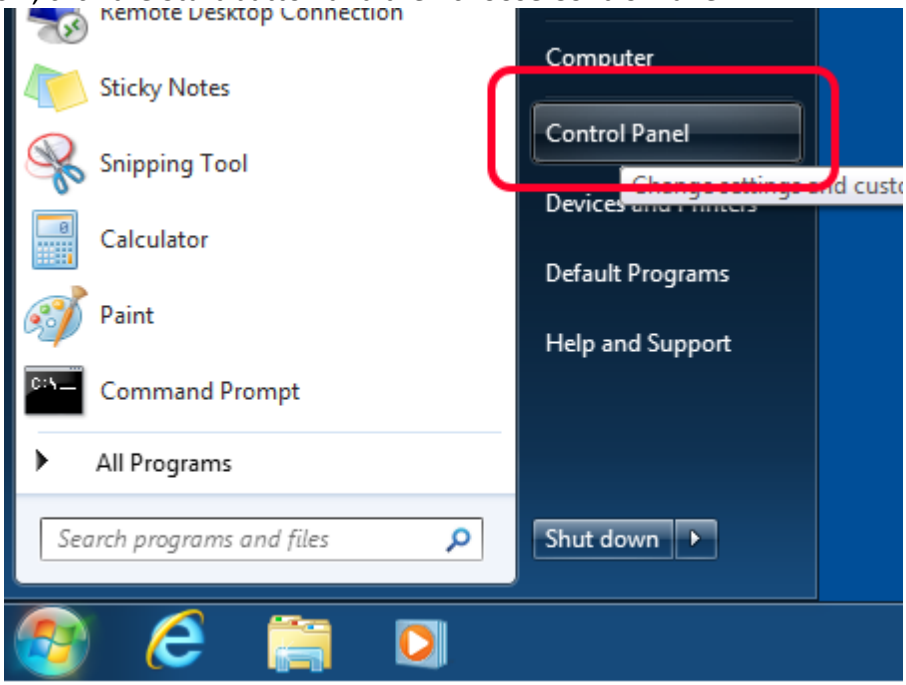
Note: it is required that the Ethernet connection to the Ether-Mach board is made directly to the PC motherboard, or to a PCI Ethernet adapter card. You may NOT use a USB to Ethernet adapter to connect to Ether-Mach, but you may for your INTERNET CONNECTION ONLY. If you do not follow these guidelines, it is extremely likely you will have connection issues.

1.1.3 Step 3: Configure your network settings

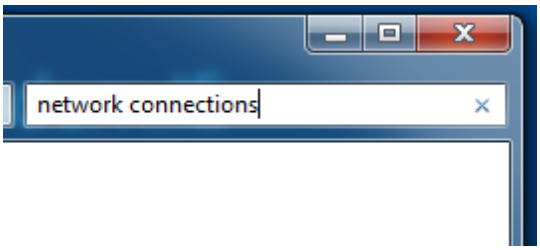
In order to function efficiently and avoid interference from other processes running on your system, you should follow the steps below to configure the Ethernet interface to disable all unnecessary services. If you do not do this, system services such as AutoIP or DHCP can interrupt or interfere with board connectivity, if you can connect at all. The instructions below include pictures for Windows 7, 8, and 10 (in steps where the process is the same, Windows 7 pictures are shown).

- Open "Network Connections"

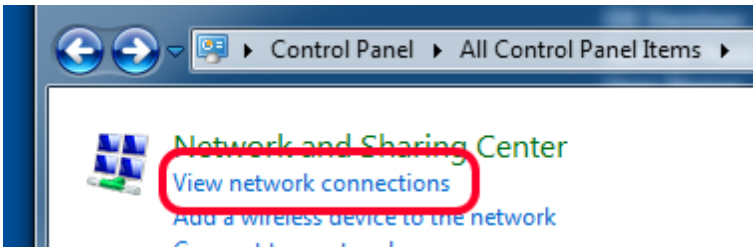
On Windows 7, click the Start button and then choose Control Panel:



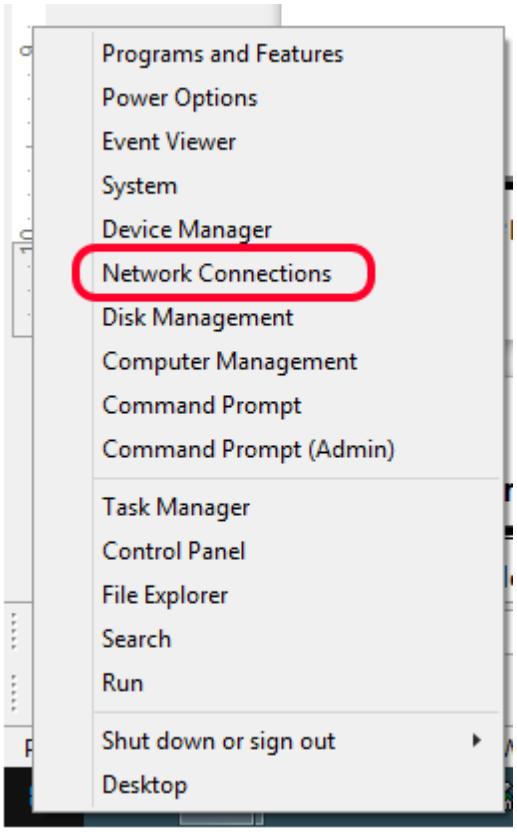
Then, in the top right of the control panel, search for “network connections” (no quotes):



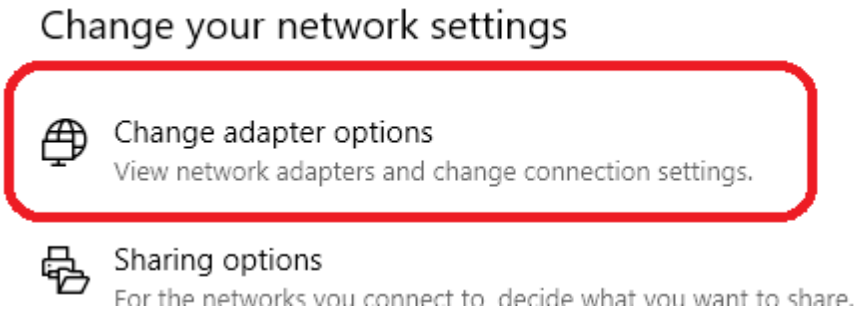
From the results, select “View network connections”:



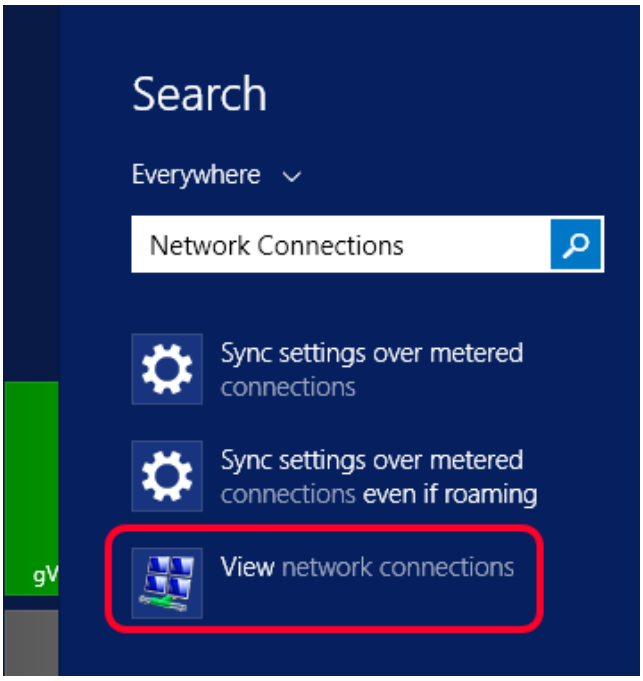
On Windows 8.1 or Windows 10, right click the start button and select “Network Connections”



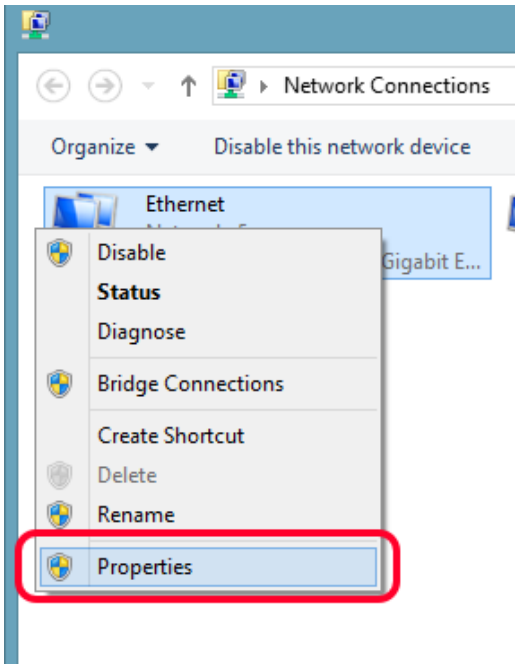
On newer versions of **Windows 10**, with the **Creator’s Update**, right click the start button and select “Network Connections”. In the window that opens, choose “Change adapter options”:



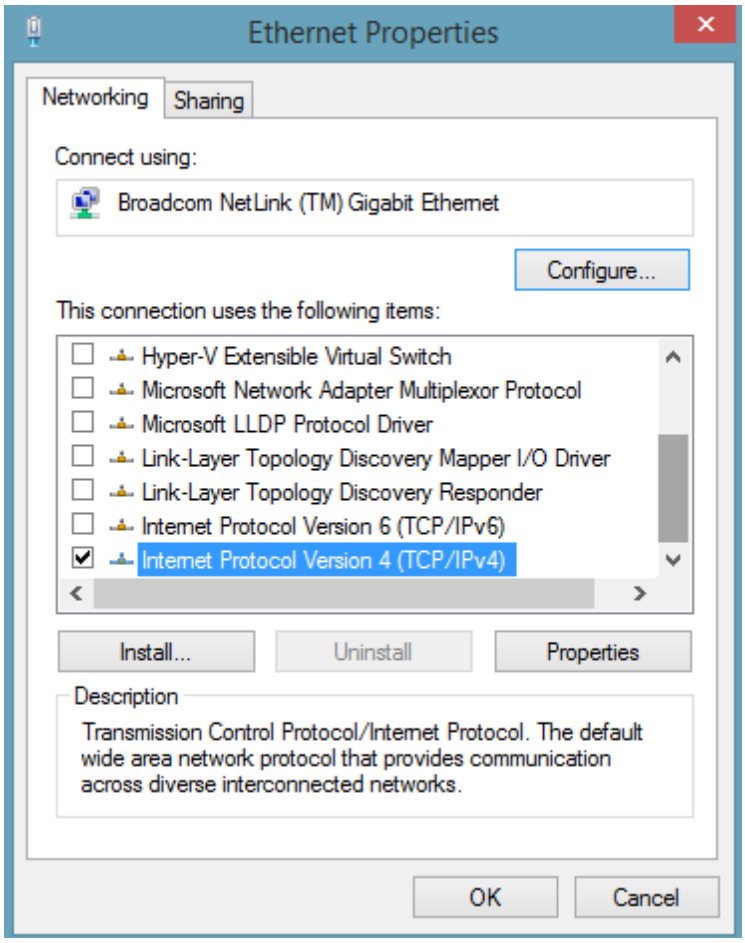
On regular Windows 8, press the windows key on your keyboard and then type “Network Connections” (no quotes). A search should begin on the right – choose the “View network connections” result:



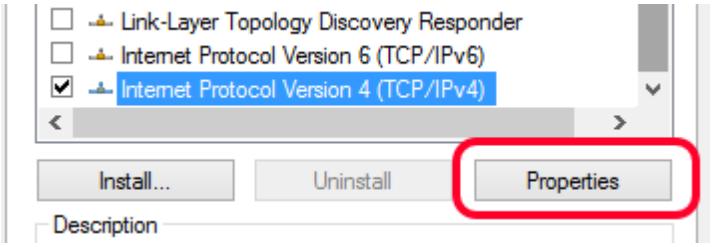
- Right click the icon for the Ethernet adapter that you connected the Ether-Mach board to and select properties



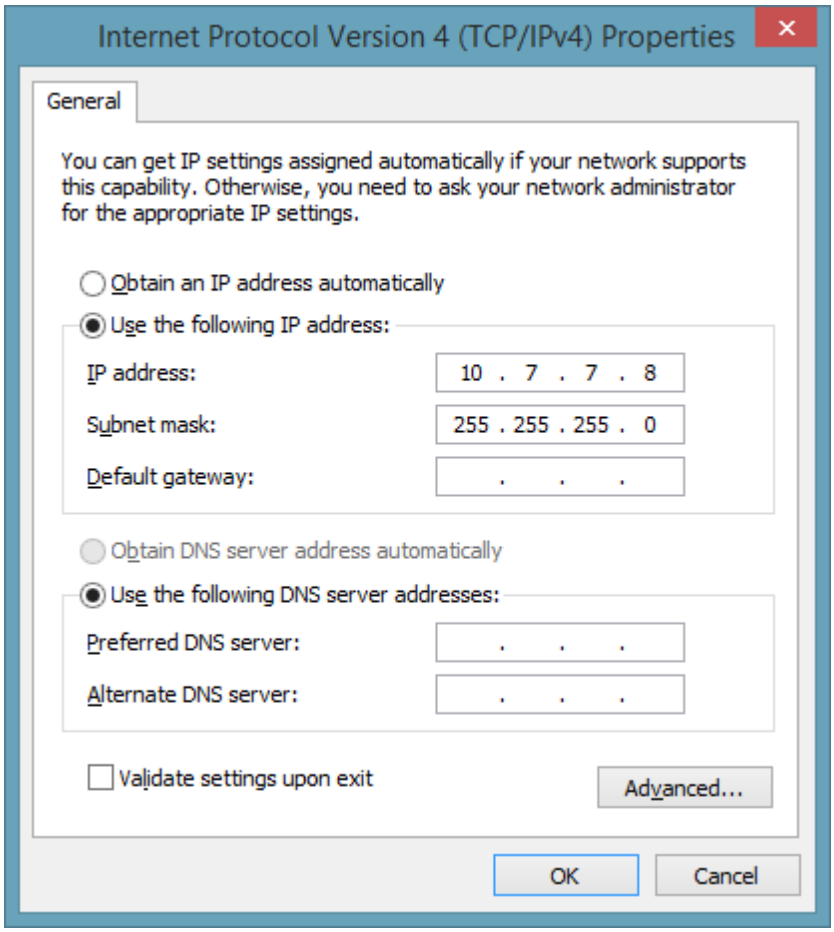
- Disable every check box except for "Internet Protocol Version 4"



- Select "Internet Protocol Version 4" and click properties

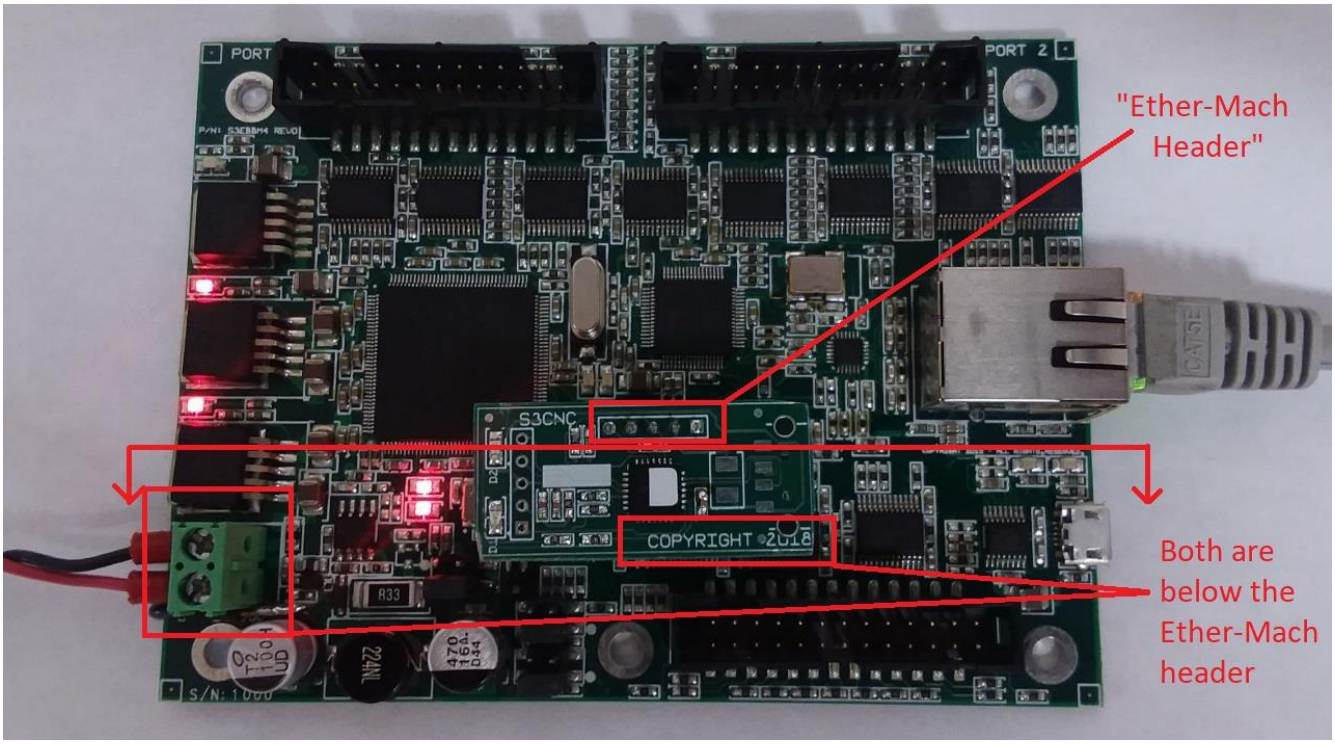


- Configure the dialog to look like the picture below



If you need to run the Ether-Mach board with a different configuration from the default, such as to change its IP to avoid subnet conflicts, contact your local system administrator and consult the full Ether-Mach users guide found at ether-mach.com.

1.1.4 Step 4: Affix the licensing dongle onto the Ether-Mach board



The 5 male pins on the Ether-Mach board should line up with the 5 female on the licensing dongle. The "COPYRIGHT 2018" text should be on the same side of the header as the power terminal. See the picture above for an example.

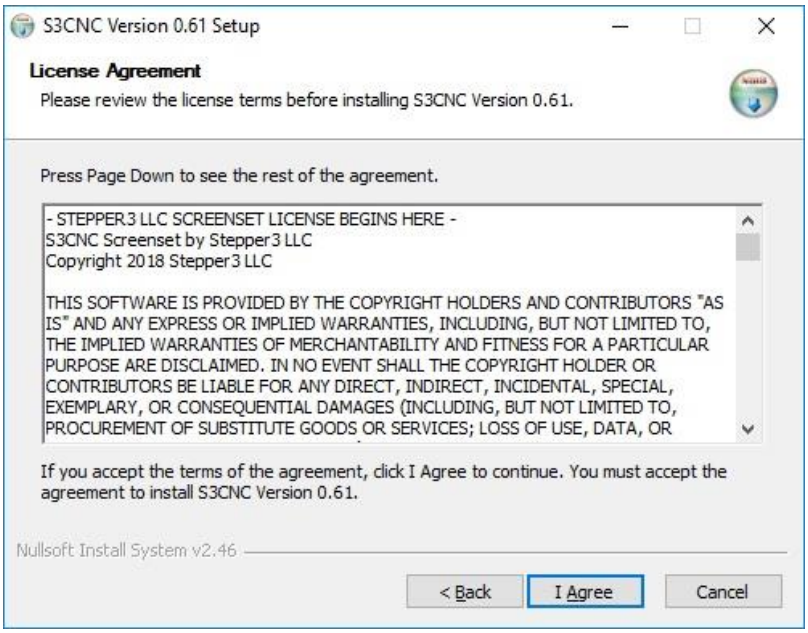
1.2 Part 2: Installing S3CNC

1.2.1 Step 1: Running the installer

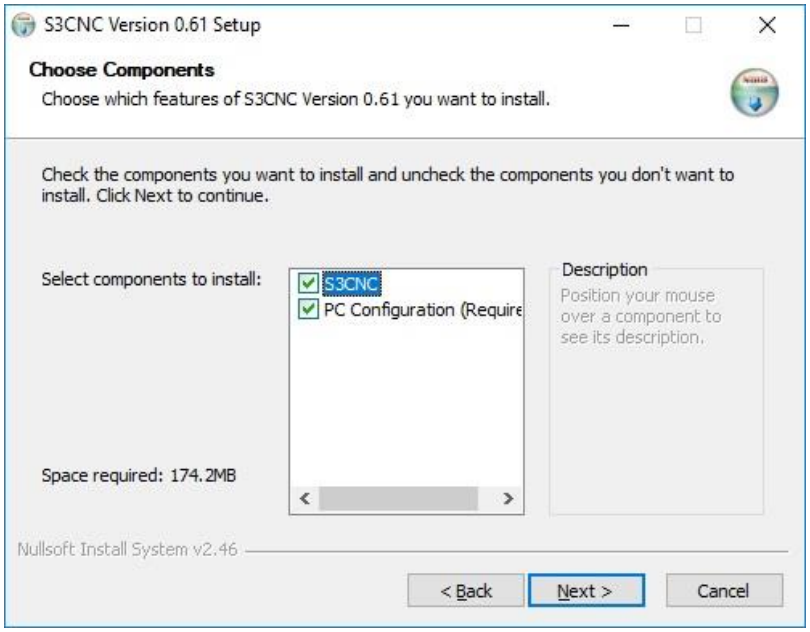
To install S3CNC, just run the installer. You can leave all the defaults, like installation directory, as they are. It is very important that the "PC Configuration" option remains on. This option disables Windows features that interfere with stable real time performance - specifically, automatic installation of updates and CPU frequency scaling.

- First page of the installer. Save your work in any open applications and close them, and then press “Next”

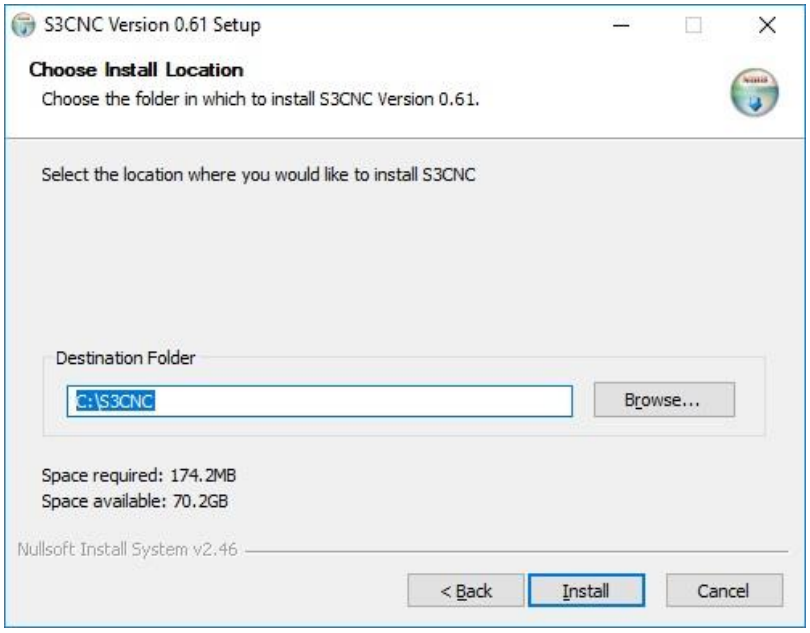
- Read the agreement and press “I Agree” if you agree to the terms presented



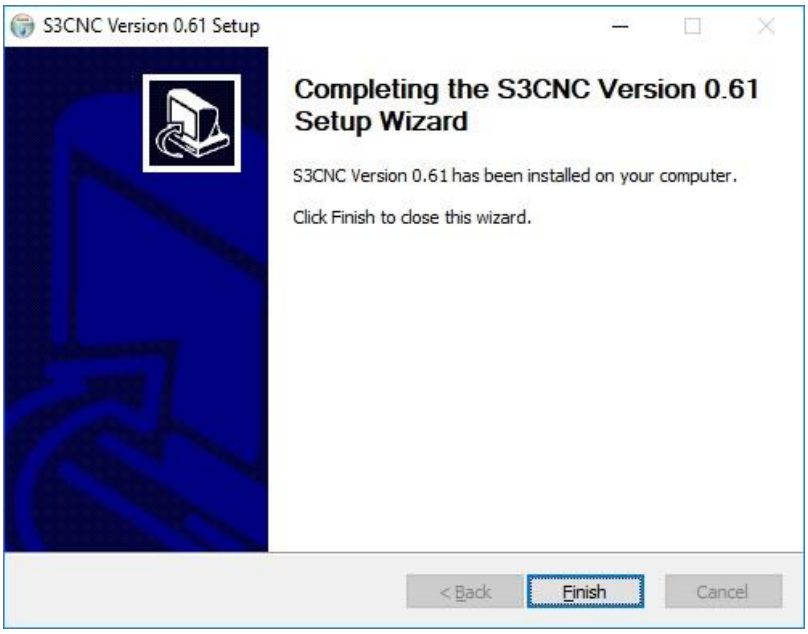
- Choose the S3CNC and PC Configuration (default selected) components to install and press “Next”



- Choose the installation directory (It is recommend to leave it as default C:\S3CNC) and press “Install”



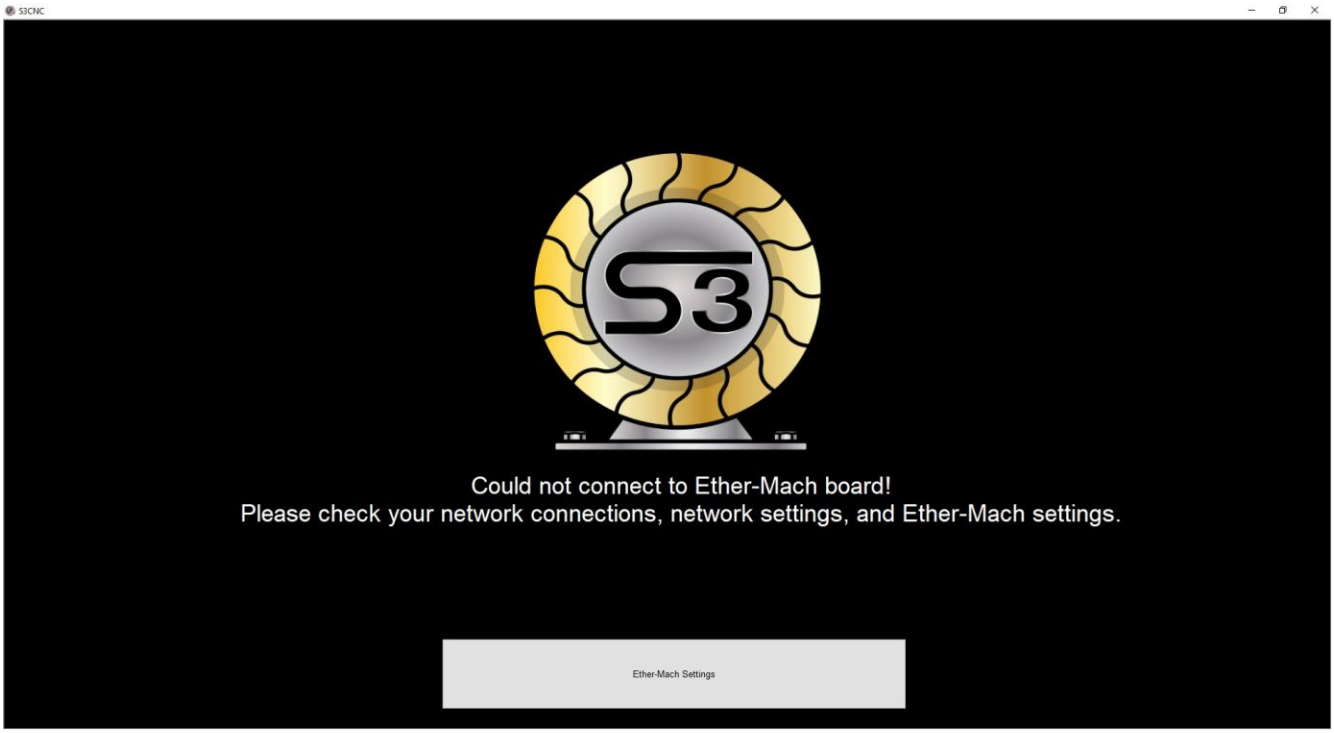
- The installation is now complete! Press “Finish” to exit the installer



After the installation completes, there should be an S3CNC icon on the desktop. Double click the icon to launch the software.

1.2.2 Step 2: Connecting to the board

When the software starts, it will first attempt to connect to the Ether-Mach board. If it fails to connect or no Ether-Mach board is present, it will fail to an error screen that provides access to the Ether-Mach settings dialog as shown below. This dialog can be used to change IP address or performance settings in order to get the board to connect successfully.

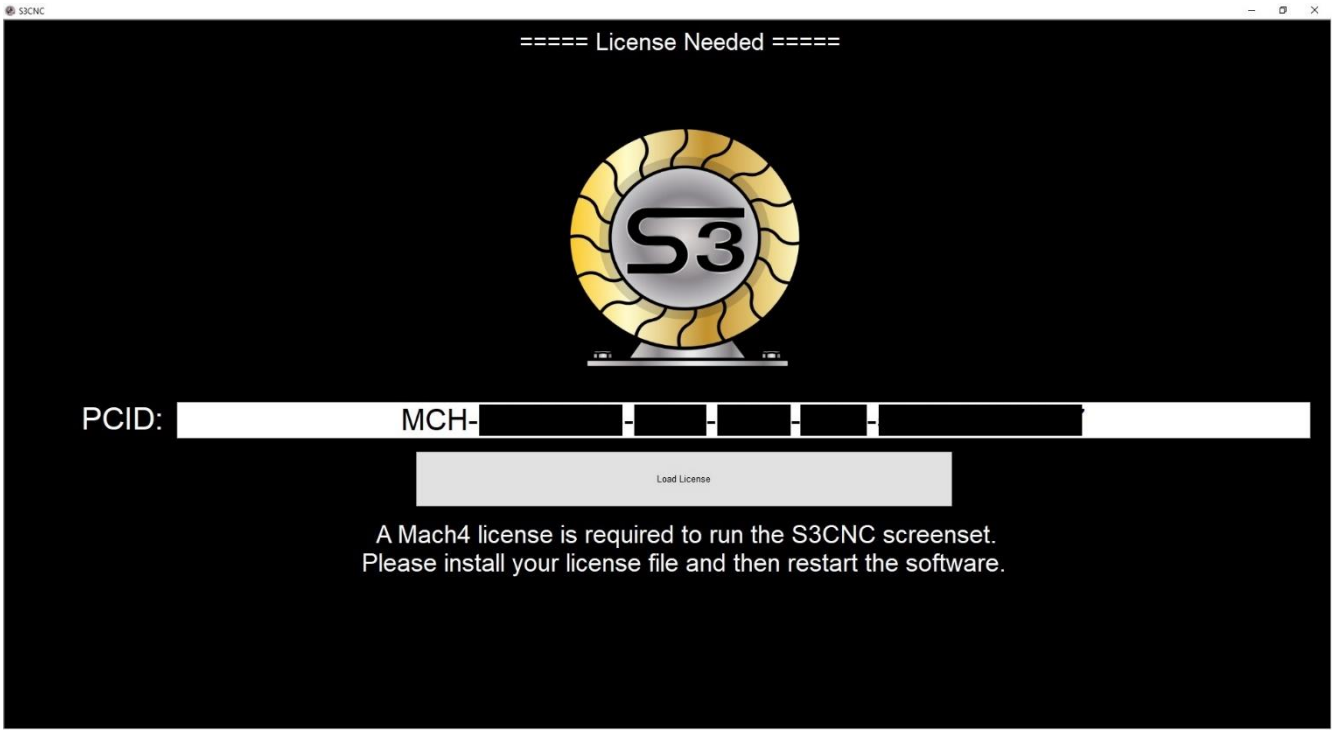


If you need to run the Ether-Mach board with a different configuration from the default, such as to change its IP to avoid subnet conflicts, contact your local system administrator and consult the full Ether-Mach users guide found at ether-mach.com.

To attempt to connect again after a failure, restart the software.

1.2.3 Step 3: Licensing

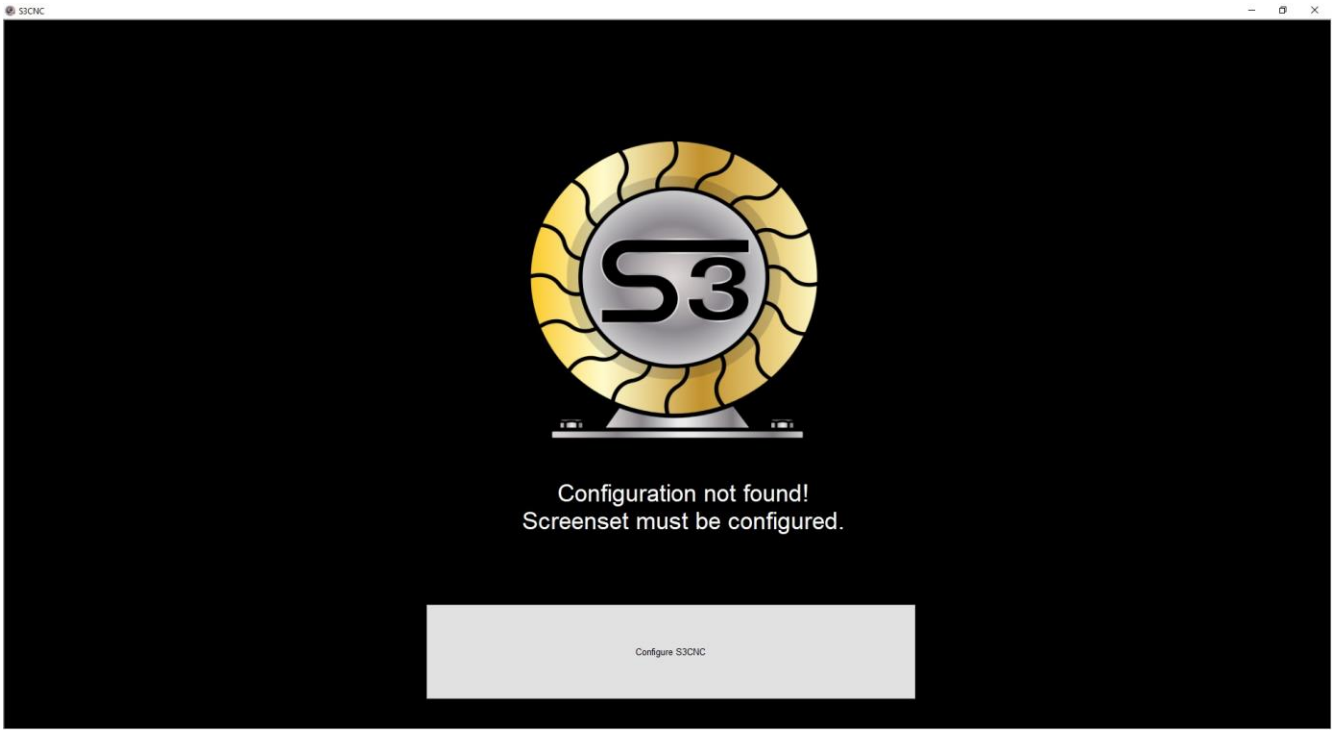
After connecting to the Ether-Mach board, the software will require you to provide a license key file. A PCID number is provided on the screen as partially shown below. This number must be given to Stepper3 to be turned into a corresponding license key file. Once you get your license key, click the "Load License" button and select your key file.



After loading your license key, you will have to restart the software.

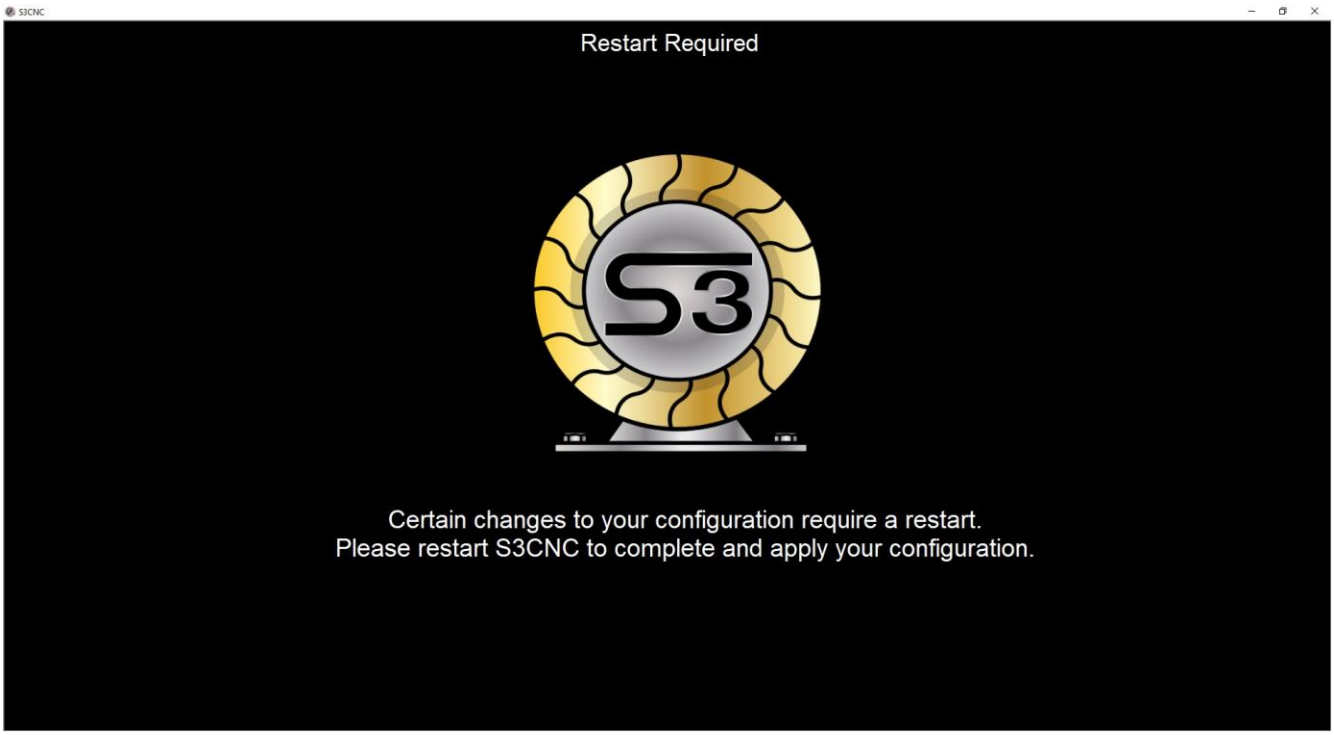
1.2.4 Step 4: First Time Configuration

Now that you have licensed the software, it will continue loading and attempt to load a CNC configuration. Since this is your initial installation, no configuration is present, and the screen below will be displayed. The software will require you to run through the configuration process. Click the "Configure S3CNC" button to bring up the Configurator (the S3CNC configuration wizard). You can either create a new configuration or load an existing one from an s3conf file.



For details on configuration, see Section 2: Software Configuration (Configurator).

Once you have completed your configuration, hit Finish at the end of the wizard. This first time configuration may require you to restart S3CNC one last time (screen below). If you don't have to restart, it should load the main page; otherwise, you will see the main page after you restart.



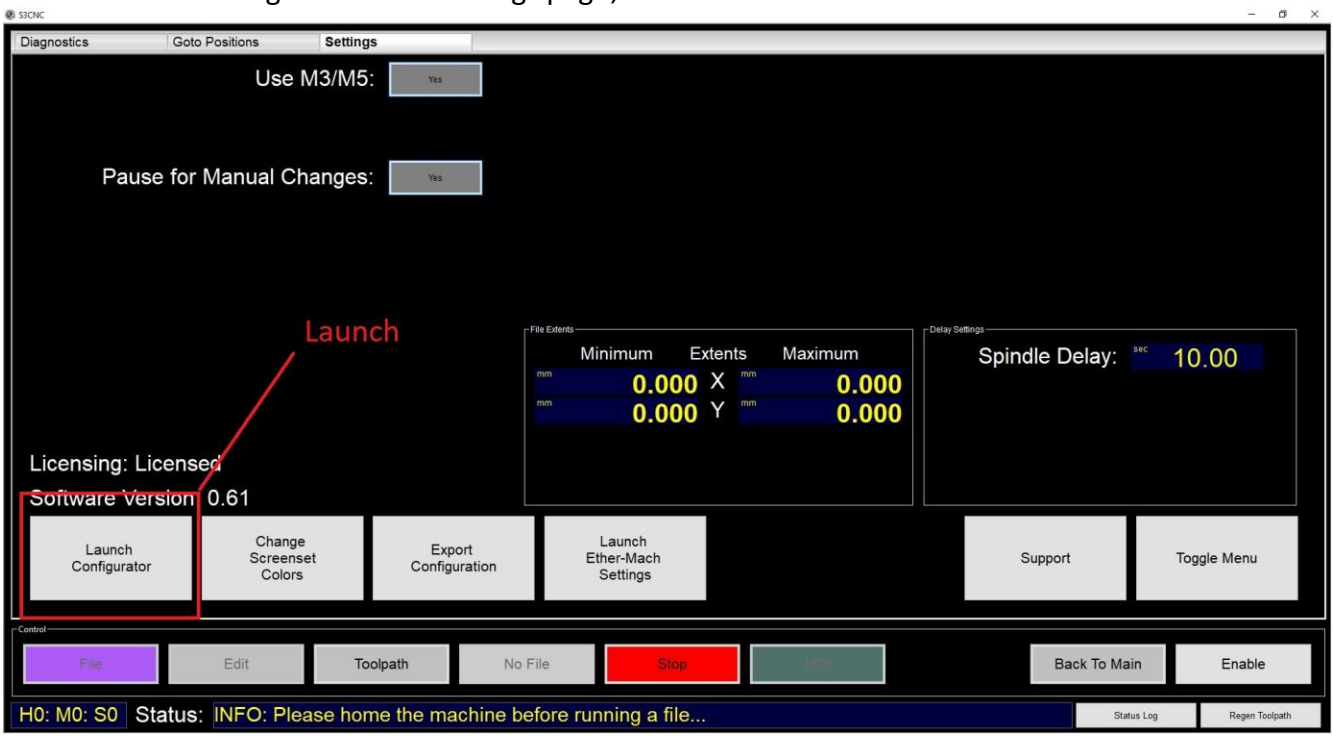
2 Section 2: Software Configuration (Configurator)

The S3CNC software is set up and configured using a step by step wizard called the S3CNC Configurator. The Configurator provides a straightforward, streamlined configuration process that focuses on keeping important settings together on easy-to-use wizard pages. With the configurator, you can set up a machine in a variety of configurations, such as:

- 2 axis router (e.g., laser engraving, water jet)
- 3 axis router (milling)
- 3 axis router with automatic tool changing
- 2 axis plasma
- 3 axis plasma (THC)
- and more

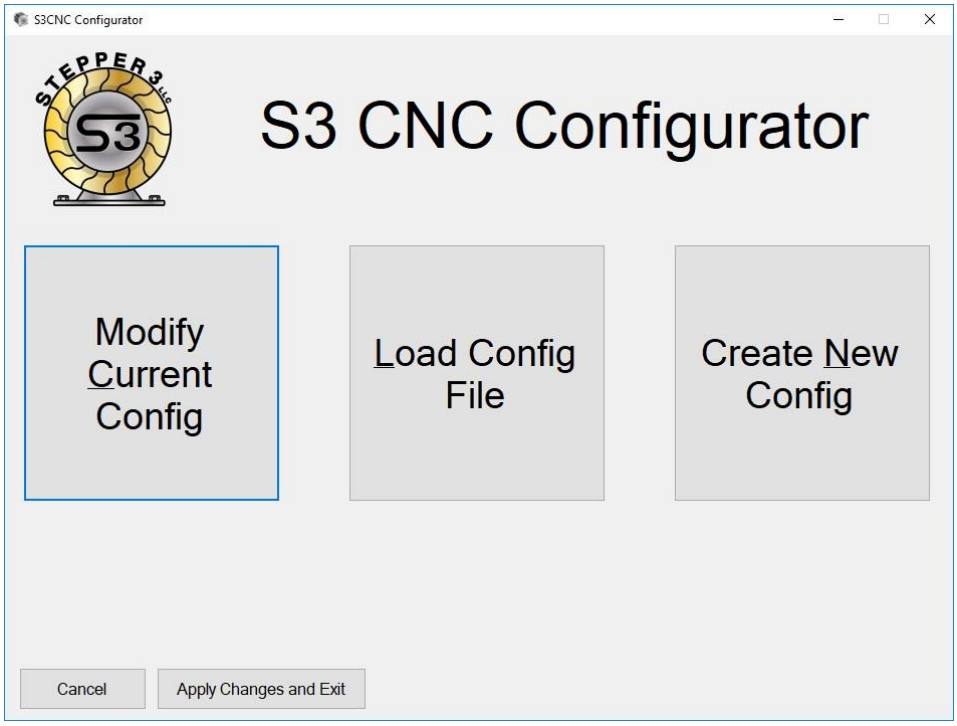
2.1 Part 1: Launching the Configurator

During the first time installation of the software, you will be required to launch the Configurator and create an initial configuration before entering the software. On the other hand, if you already have a configuration and are already running the software, you can launch it with the "Launch Configurator" button on the Settings tab of the Settings page, as shown below



.

2.2 Part 2: Choosing a configuration



Once you launch the Configurator, you are presented with three large buttons:

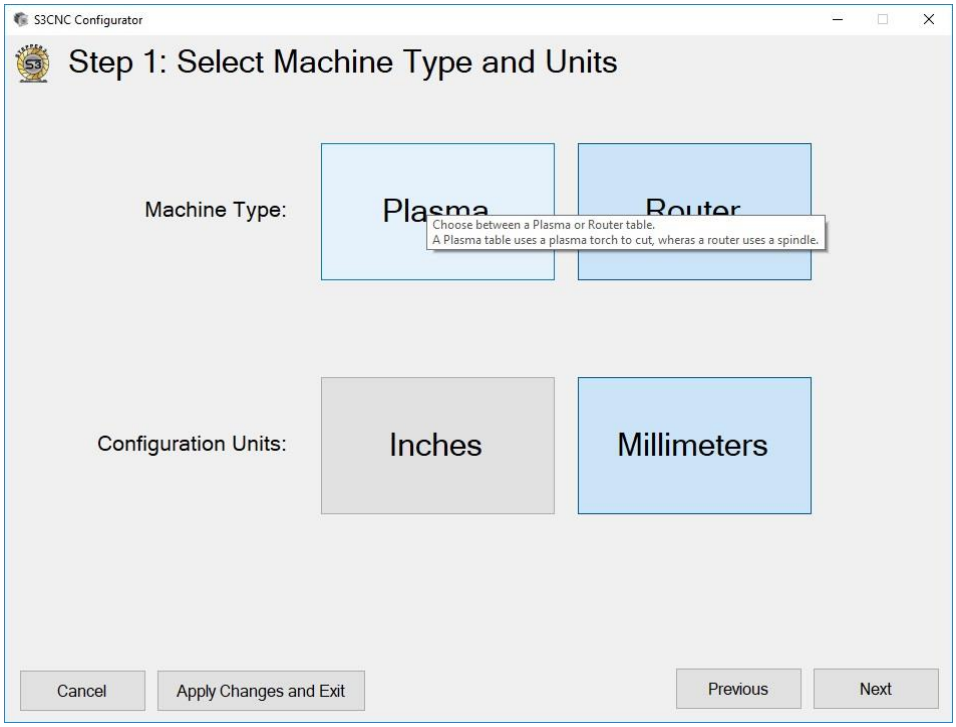
- *Modify Current Config* - Re-run the wizard with the existing configuration loaded. Use this if you just want to make some changes to your existing settings.
- *Load Current Config* - Select and load an s3conf configuration file. This will overwrite your existing configuration, if any, with the configuration in the selected file.
- *Create New Config* - This will overwrite your configuration with a new empty configuration.

Once you have selected one of these options, you will begin the configuration wizard. The wizard consists of numerous pages each pertaining to one aspect of your CNC system. At the bottom of each page are the following buttons:

- *Cancel* - Exit the Configurator without saving/applying any changes
- *Apply Changes and Exit* - This button is not available during first time configuration. This saves/applies all the existing settings and exits, without requiring you to reach the end of the wizard. If there is an error on any wizard page, it will stop at that page, report the error, and leave you at that page in order to fix it.
- *Previous* - Goes back one page. If there is an error on your current page, then you cannot save its settings. It will warn you about the error and give you the option to ignore it and go back any way or to fix it.

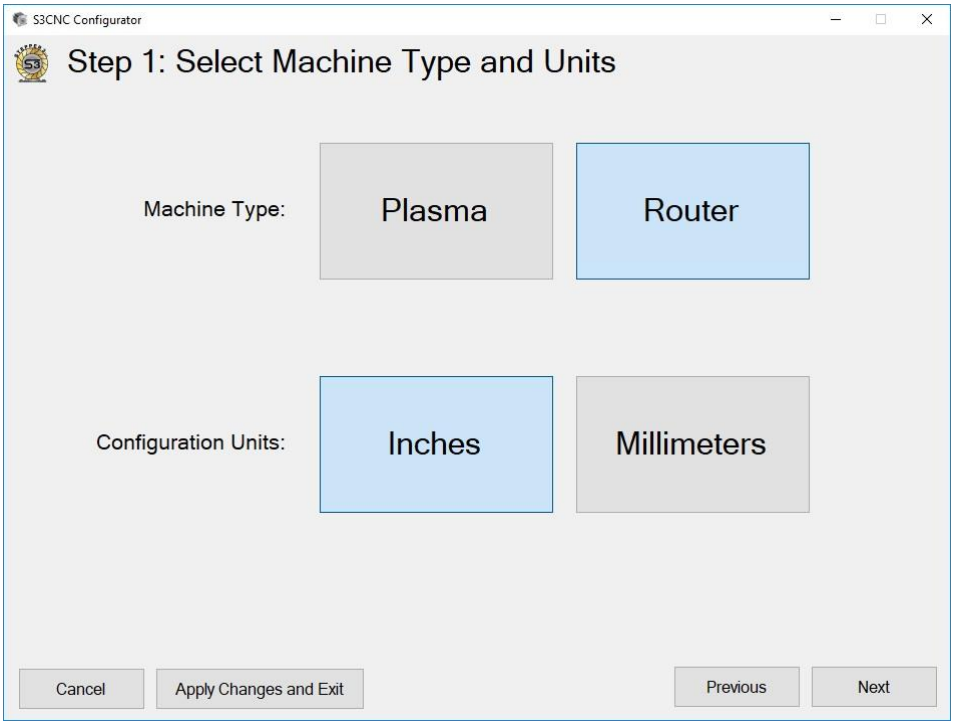
- *Next* - Checks for any errors on the current page and then goes to the next page. If there is an error, a message will pop up describing the error and you will be required to fix it before moving on to the next page.

If you need help remembering what a setting does, in addition to this guide, each setting in the configurator has hover text that provides a brief explanation if you rest your mouse on it, like below.



2.3 Part 3: Complete the wizard

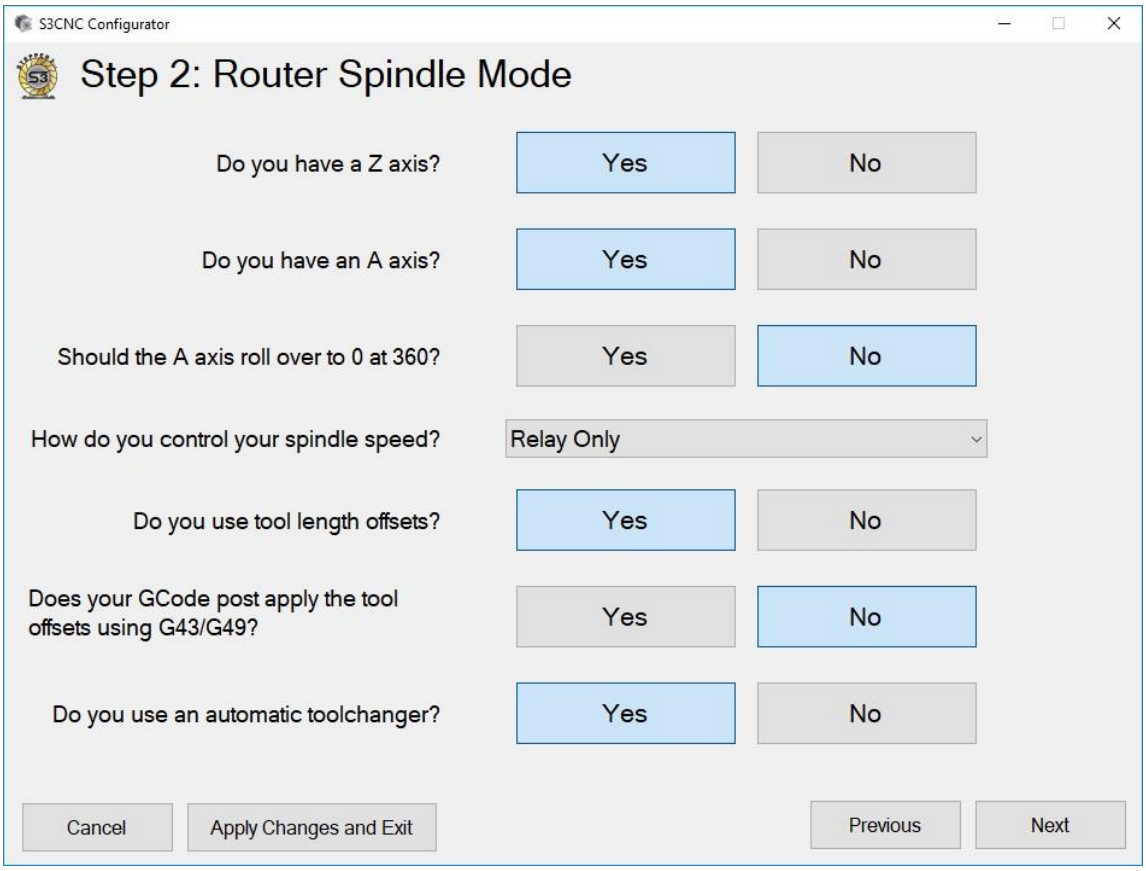
2.3.1 Step 1: Select Machine Type and Units



S3CNC supports two **machine types**: Router and Plasma. A Router machine type is used for machines that operate a spindle, with or without speed control, for tasks such as milling and routing. A Plasma machine type is for plasma machines, with or without Torch Height Control (THC).

Configuration units are the units used in the Configurator. If you selects Inches, any setting in the following wizard pages labeled "units" or "units/sec" expect values in "inches" and "inches/sec". Likewise, selecting Millimeters makes them "mm" and "mm/sec".

2.3.2 Router Step 2: Router Spindle Mode



This page is used to set up the spindle and tool settings for your router. It asks the following questions:

Do you have a Z axis? Most routers will have a Z axis, especially if they have a spindle. On the other hand, some machine types, like a water jet, will operate with only 2 axes, with the water jet just firing straight down.

Do you have an A axis? The A axis is an additional rotary axis used to either rotate the material or the spindle.

How do you control your spindle speed? There are 4 different options for spindle speed control:

- *Relay Only*: There is no speed control, just the spindle on/off relay
- *Analog Speed Control*: The spindle speed is output using PWM. This can then be converted into a voltage or 4-20 mA signal for spindle speed control
- *Digital Speed Control*: Up to three outputs are used to select one of up to 8 different speeds
- *Stepper Motor Spindle*: A stepper motor is used at the spindle

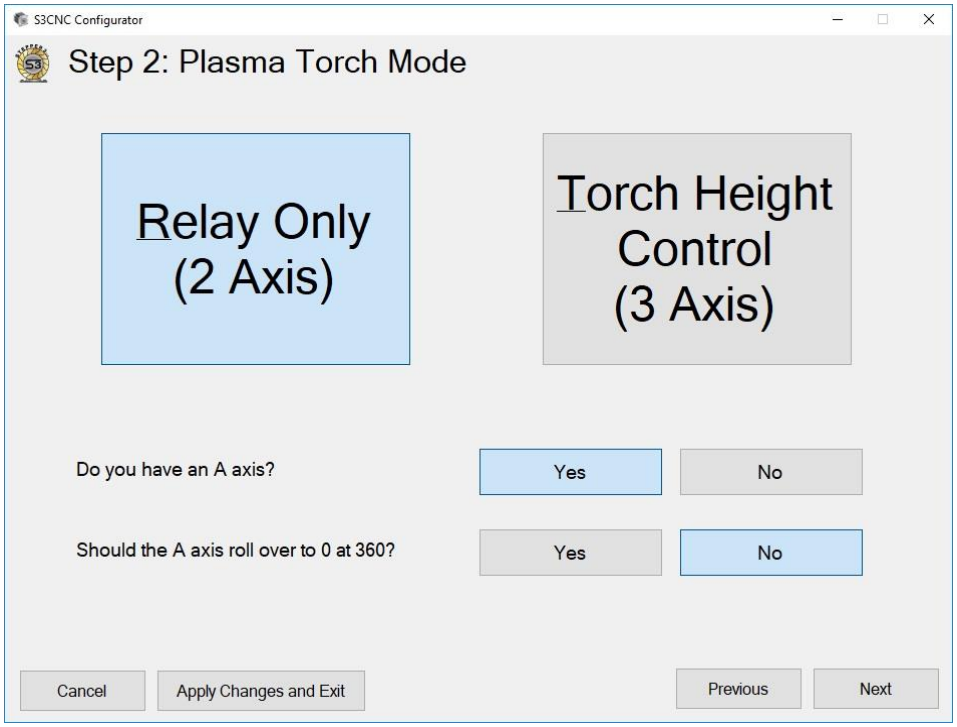
Do you use tool length offsets? Tool length offsets are measurements of each tools' length that are

stored in the software's tool table. When an M6 is used to select a particular tool, an offset will be applied to shift the work coordinates such that the tip of the selected tool gets placed at the coordinates specified by the G-Code. Users that do not use tool length offsets typically only use one tool per file and re-zero their work coordinates every time they change tools. Those that do use tool length offsets have multiple tools in a single file. They zero once at the start of the file and rely on the offsets to ensure that even when the tool length changes, the Z height of the tip stays the same.

If you answer **Yes** to *Do you use tool length offsets*, you will be asked, **Does your GCode post apply the tool offsets using G43/G49?** In normal G-Code, tool length offsets are explicitly enabled and disabled using the G-Code G43 to select a tool offset and G49 to disable the offset. Many posts will output G43 commands in addition to the M6 (tool change) to specify a tool offset. Others, however, do not output these G-Codes and instead expect the M6 M-Code to automatically apply the length offset for the selected tool.

Lastly, **Do you use an automatic tool changer?** An automatic tool changer has a set of stands for tool holders. The spindle chuck is software controlled, so the software can automatically take and return tool holders from the stand.

2.3.3 Plasma Step 2: Plasma Torch Mode

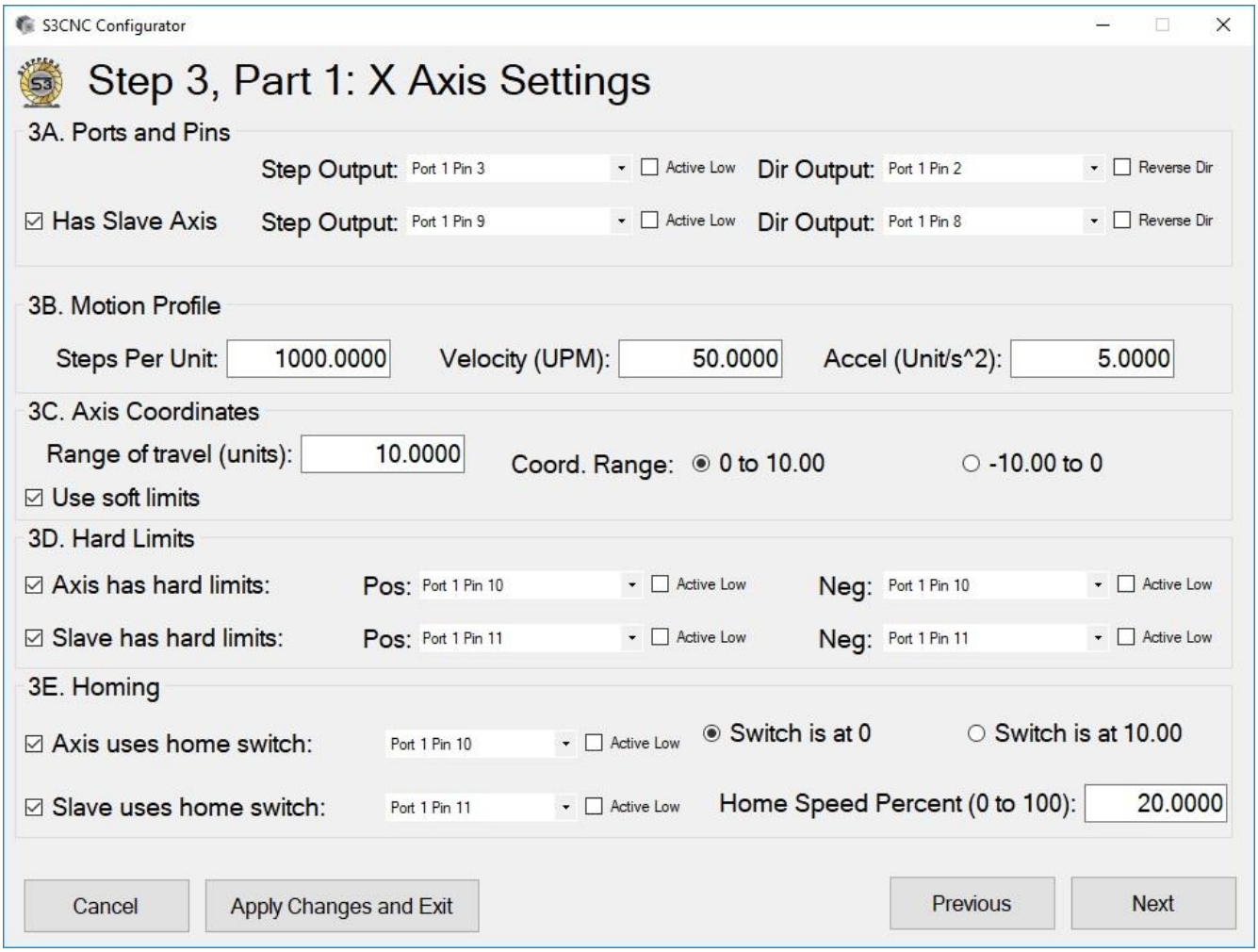


You can select between two plasma torch modes: *Relay Only* and *Torch Height Control*. *Relay Only* is for two axis machines that only use a relay to enable/disable the torch and do not use height control. It can also be used for systems with existing external height control solutions. *Torch Height Control* has 3 axes, the third of which is a Z axis that is used for height control.

The other options here are:

- *Do you have an A axis?* - The A axis is a fourth axis that is typically used in a rotary manner to rotate materials
- *Should the A axis roll over to 0 at 360*? - That is, is it a rotary (Yes) or linear axis (No)?

2.3.4 Step 3: Axis Settings



The next two to four wizard pages are all identical - each page is used to set up a single machine axis. There will be one axis configuration page for each axis, with a max of four (X, Y, Z, A). Each page contains the following settings:

3A. Ports and Pins

- *Step Output* - Select the step output pin for the axis's main motor. As with all pin selection in the wizard, checking Active Low will invert the signal
- *Dir Output* - Select the direction output pin for the axis's main motor. The Reverse Dir checkbox is the equivalent of making the signal active low
- *Has Slave Axis* - Check this if the axis has a slave axis. If checked, an additional Step Output and

an additional Dir Output pin selection will appear where you must provide the step and direction pins for the slave axis

3B. Motion Profile

The motion profile defines the limitation on motor motion with regards to speed and acceleration. It also defines the relationship between step pulses and real world units.

- *Steps Per Unit* - The number of steps per unit (inch or mm) of travel
- *Velocity (UPM)* - The velocity in units per minute
- *Accel (Units/s^2)* - The acceleration in units per second squared

The S3CNC trajectory planner is more modern and more efficient than those of older CNC systems like Mach3. When Mach3 ran in constant velocity mode, each axis was effectively limited to half of the given acceleration for the axis. However, the smarter, more efficient planner in S3CNC allows it to get closer to the ideal acceleration as specified by your motion profile. As a result, the same acceleration values from Mach3 may lead to higher accelerations in S3CNC. If you find that your motion profile is causing slipping or step loss, try lowering the acceleration first, and then the velocity afterwards. It is likely that your previous software did not run the axis as close to the numbers you entered as this system does.

3C. Axis Coordinates

- *Range of travel (units)* - This is the number of units that the axis can travel from one table end to the other. For example, if, from one limit switch, the axis can travel 60 inches to the other side before hitting the opposing limit switch, the axis has 60 inches of travel. However, it is recommended that you make this value a bit smaller than the full switch-to-switch range, so that soft limits can stop before hitting the hard limit. So, in this example, a good value to enter would be 59.5 inches
- *Coord. Range* - The coordinate range specifies the coordinates of each end of the axis's range of travel. One end is always 0, so the options are a positive range (0 to X) or a negative range (-X to 0)
- *Use soft limits* - This enables or disable soft limits for this axis. Soft limits are a software feature that prevents the axis from moving outside of its range of travel after homing. It is highly recommended that you leave this feature enabled for all axes

3D. Hard Limits

Hard limits are input signals from switches at the far ends of the axis's travel. These switches are used to detect when an axis has travelled too far and stop it before any damage occurs.

If you check *Axis has hard limits*, two pin selection boxes will appear (*Pos* and *Neg*) where you should select the input pin for the positive and negative limits. The positive and negative limits can use the

same pin and this can be the same pin as the home switch. Many tables have a single input signal for home, positive limit, and negative limit. In such a case, you would set all three to the same input pin.

If you have a slave axis, there will be an additional checkbox, *Slave has hard limits*, to enable hard limits for the slave axis as well.

3E. Homing

Homing is a startup process used to move the axis to a known, repeatable location before establishing a coordinate system. By homing your axes, the software can use machine coordinates to refer to exact locations on your table, relative to the home location that is established.

Like with hard limits, there is an *Axis uses home switch* checkbox to enable homing for the axis. Additionally, there may be a *Slave uses home switch* checkbox to enable homing for the slave axis. When checked, a pin selection box will appear where you must select the home switch input pin.

If either the main axis or slave axis has a home switch (i.e. either checkbox is checked), two options will appear:

- *The home switch location* - This selects at which end of the table the home switch is. For example, if your coordinate range is 0 to 10, you can choose "Switch is at 0" or "Switch is at 10"
- *The home speed percent* - The speed the axis moves at during homing while it searches for the home switch, as a percentage of the max velocity (as specified in the motion profile)

2.3.5 Step 3 continued: Additional Axis Inputs and Outputs

After each axis settings page, there is a final page for configuring some less common inputs and outputs for each axis and its slave. For each, there is a check box to enable or disable the pin. If enabled, a pin selector box will appear for you to choose the proper input or output.

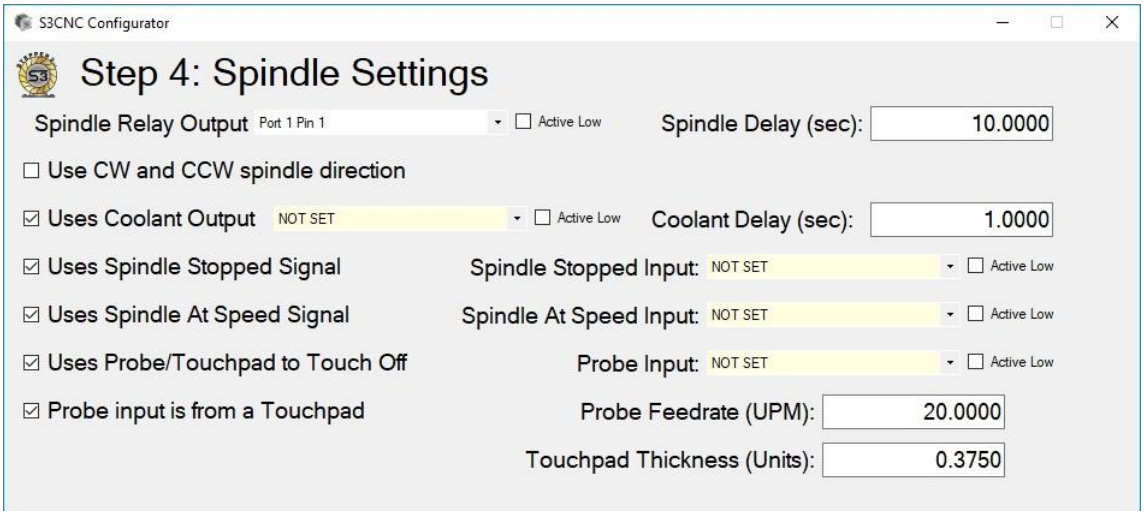
The pin types are as follows:

- *Enable pins* - These are used to enable/disable drives whenever you enable/disable the software
- *Reset pins* - These are used to reset drives (particularly servo drives) in order to clear any drive faults that are preventing the drives from enabling
- *Brake pins* - These pins are used to control the brakes. Normally, the brakes are disengaged outside E-Stop and engage when you enter it

There is an option regarding reset outputs: *Always reset drives during hard limit recovery*. If this is selected, the software will reset your drives when recovering from hard limit errors even if there is no drive fault input active.

Lastly, if any of your axes have a home switch, there is the option: *How much should an axis move to clear the switch after homing?*. This is the distance that the axis backs off of the home switch before setting the machine origin.

2.3.6 Router Step 4: Spindle Settings



You must always select a *Spindle Relay Output*, which is the output pin used to enable the spindle. You can also configure the *Spindle Delay*, which sets the number of seconds the software waits when turning the spindle on. This should be set to ensure that the spindle can always reach its target speed before the delay expires. You can change this setting directly in the software for fine tuning.

Next, there are 4 or 5 checkbox options:

- *Uses Coolant* - If checked, you must set an output pin used for turning on and off the coolant, as well as a coolant delay. Coolant is controlled with the M7 and M9 M-Codes
- *Uses Spindle Stopped Signal* - If checked, you must set an input pin as the spindle stopped signal. The spindle stopped signal indicates that the spindle is stopped. When turning the spindle off, instead of just assuming that the spindle is stopped after the spindle delay, the software will check this signal to verify. If the signal does not change, an error is reported to the user so they can stop running the machine and investigate the problem
- *Uses Spindle At Speed Signal* - Another input pin, like the spindle stopped signal. This one indicates that the spindle is at the requested speed and is no longer accelerating. If enabled, when turning the spindle on, instead of just assuming that the spindle reached its speed after the spindle delay, the spindle at speed signal is checked. If it never changes, an error is reported to the user so they can stop running the machine and investigate the problem
- *Uses Probe/Touchpad to Touch Off* - Before running a file on a router, a Z zero position must be established at the top of the material. This can be done manually by jogging the tip of your tool to the top of the material and zeroing; however, many tables support using a probe or a touchpad to perform this operation automatically. If enabled, you must provide an input pin for your probe/touchpad signal, as well as a probing feed rate

- If you selected to use a probe/touchpad to touch off, you can also choose *Probe input is from a Touchpad*, which if selected, requires you to enter the thickness of the touchpad

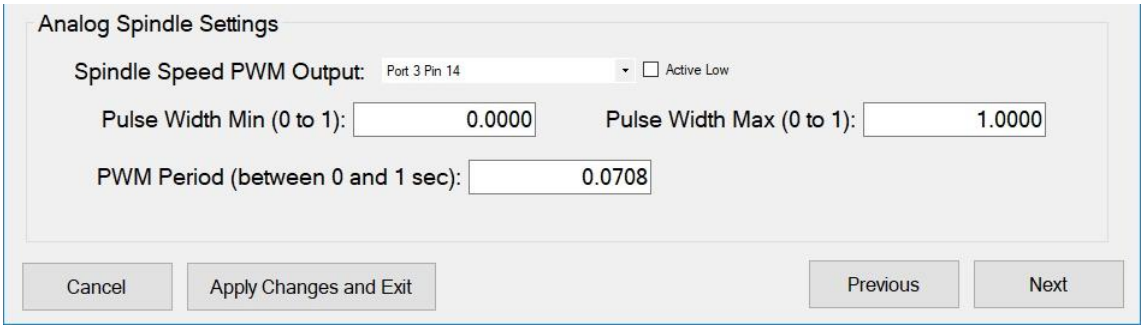
Next, you can enter the *Minimum RPM* and *Maximum RPM* of the spindle. Note that these are values that the software uses simply to display spindle speed and interpret file or MDI speed commands. Your spindle drive must be properly setup such that the physical signals commanded by S3CNC correspond to the true RPM of the spindle. For example, if S3CNC is configured to use analog speed control with a 4-20 mA output, a minimum RPM of 6000, and a maximum RPM of 18000, the spindle drive must also be programmed to spin at 6000 RPM when receiving a 4 mA signal, and 18000 RPM when receiving a 20 mA signal.

The remaining options vary based on the spindle type you selected:

2.3.6.1 Relay Only

If you chose relay only spindle, there are no further options (see the screenshot at the beginning of this section).

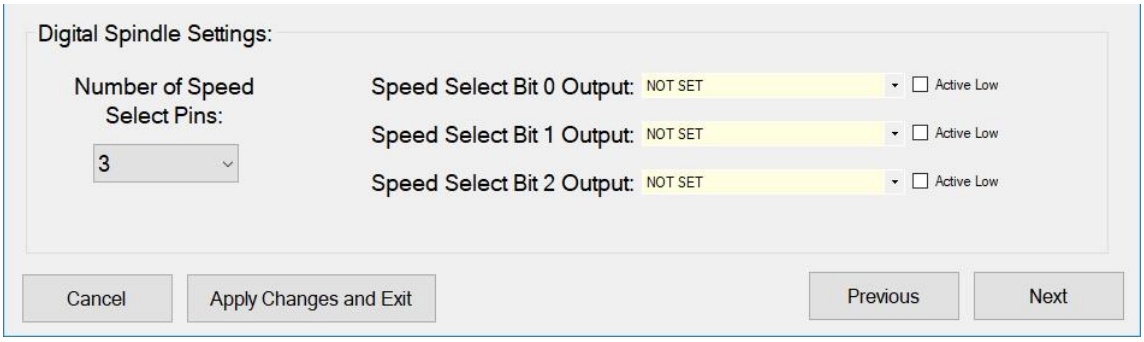
2.3.6.2 Analog Spindle Settings



If you chose an analog spindle, you will get the following options:

- *Spindle Speed PWM Output* - The pin on which to output the spindle speed PWM
- *Pulse Width Min* - The minimum duty cycle of the PWM, range 0 to 1
- *Pulse Width Max* - The maximum duty cycle of the PWM, range 0 to 1, and it must be greater than the minimum
- *PWM Period* - The period of the PWM (the time in seconds for one cycle to complete), can be in the range of 0 to 1 seconds. The default value, 0.0708, is set for compatibility with Stepper3's 4 to 20 mA board as well as the 0 to 10V analog output on CNC4PC's C62 breakout board.

2.3.6.3 Digital Spindle Settings



A digital spindle requires you to select the number of outputs used to indicate the speed (*Number of Speed Select Pins*) and the output pin for each one. The S3CNC software counts upward in binary and maps the digital output accordingly, as shown in the tables below, where SS in "SS Bit 0" stands for Speed Select. An example is shown if the *Maximum RPM* is set to 18,000 and *Minimum RPM* is set to 0.

- Number of Speed Select Pins: 1 – Two Speed Settings

S3CNC Commanded RPM (0-18000 RPM)	SS Bit 0
0-8999	Off
9000-18000	On

- Number of Speed Select Pins: 2 – Four Speed Settings

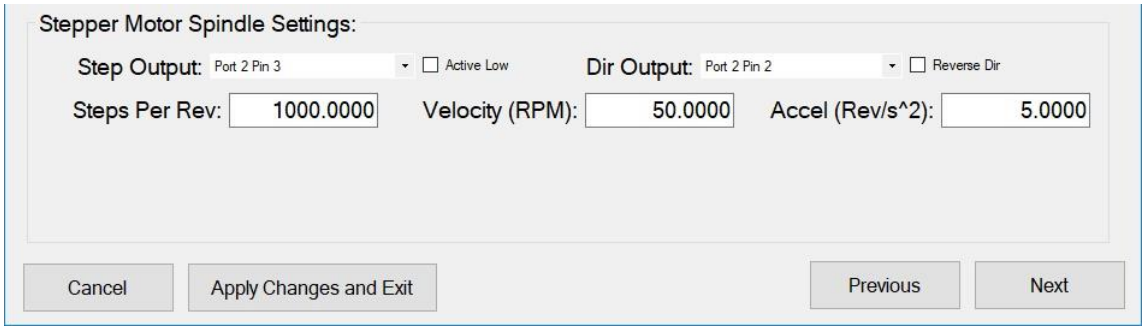
S3CNC Commanded RPM (0-18000 RPM)	SS Bit 1	SS Bit 0
0-4499	Off	Off
4500-8999	Off	On
9000-13499	On	Off
13500-18000	On	On

- Number of Speed Select Pins: 3 – Eight Speed Settings

S3CNC Commanded RPM (0-18000 RPM)	SS Bit 2	SS Bit 1	SS Bit 0
0-2249	Off	Off	Off
2250-4499	Off	Off	On
4500-6749	Off	On	Off
6750-8999	Off	On	On
9000-11249	On	Off	Off
11250-13499	On	Off	On
13500-15749	On	On	Off
15750-18000	On	On	On

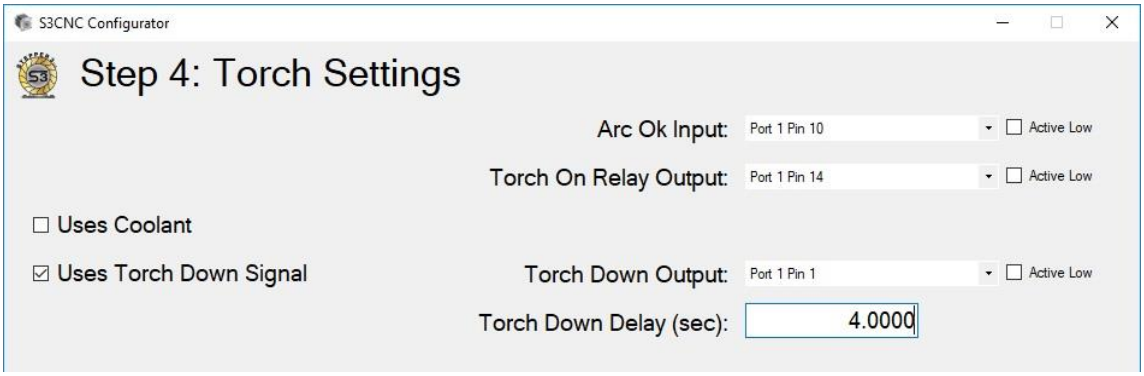
You can see the commanded RPMs follow a stepwise function that control the outputs on. These ranges are evenly divided within the minimum and maximum RPM ranges.

2.3.6.4 Stepper Motor Spindle Settings



A spindle motor spindle requires a *Step Output* and *Dir Output* just like an axis, as well as a motion profile: *Steps Per Rev* - Steps per revolution, *Velocity (RPM)* - Max velocity in revolutions per minute, *Accel (Rev/s^2)* - Acceleration in revolutions per second squared.

2.3.7 Plasma Step 4: Torch Settings



There are two required pins for all torch configurations:

- *Arc Ok Input* - This input is used as an Arc Ok feedback to ensure that the plasma arc has formed. If this signal goes inactive during a file run, a torch failure occurs and the user has options to recover or stop the file run
- *Torch On Relay Output* - This is the output pin used to turn the torch on/off

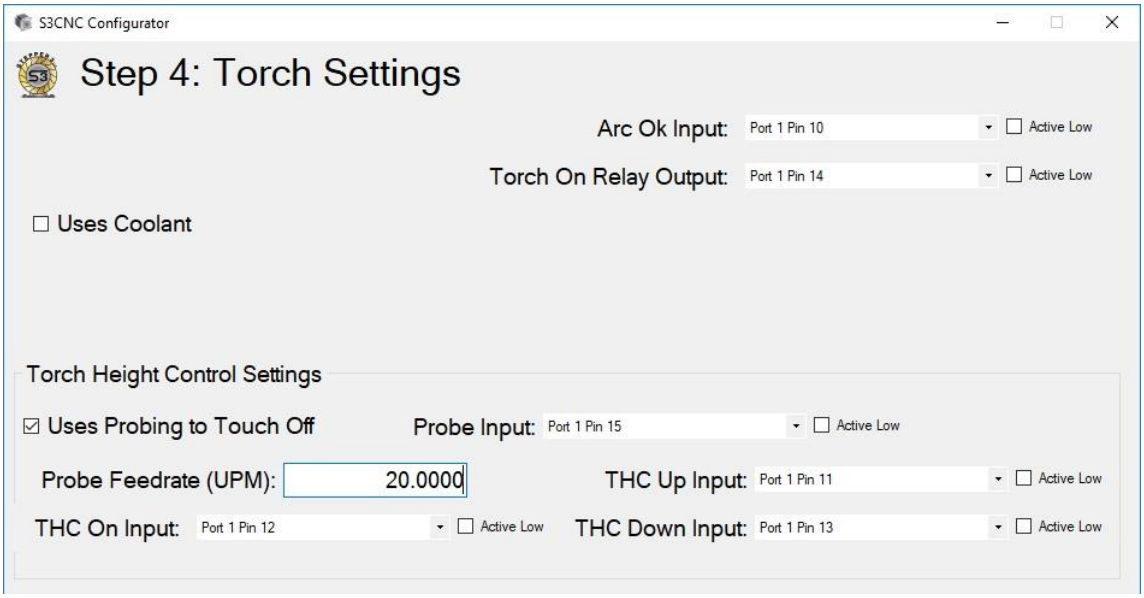
There is also a coolant option: *Uses Coolant* - If checked, you must set an output pin used for turning on and off the coolant, as well as a coolant delay. Coolant is controlled with the M7 and M9 M-Codes.

If you are not using Torch Height Control (THC), you also have the option:

- *Use Torch Down Signal* - If enabled, you can select an output pin that is used to command the torch to move down. This can be used to direct an external height control system to move the plasma torch down to the material. There is an additional *Torch Down Delay* that provides a delay between when the torch down signal is activated and when the torch on signal is activated

The torch sequence when not using THC is found in the Torch Behavior section of Guide 2: 2-Axis Plasma Table.

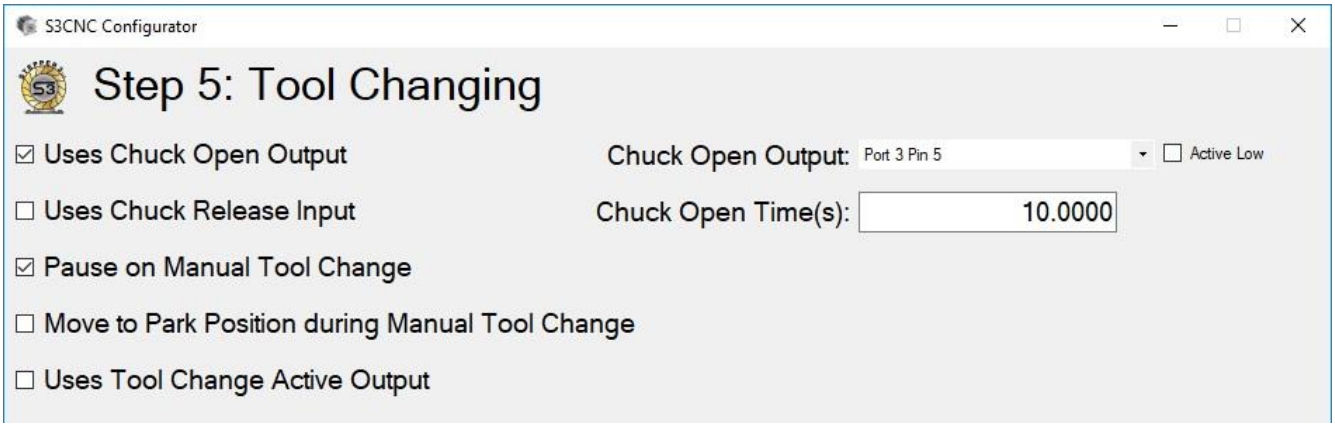
2.3.7.1 Torch Height Control Settings



Torch Height Control (THC) requires three inputs: THC Up, THC Down, and THC On. These three signals should come from your THC controller (ex, the Proma 150).

There is an option, *Uses Probing to Touch Off*, which, if enabled, requires a probe input pin and a probe feed rate. This option allows you to use probing when referencing the Z axis for THC. For more information on plasma and torch height control, see Guide 3: 3-Axis/THC Plasma Table.

2.3.8 Router Step 5: Tool Changing



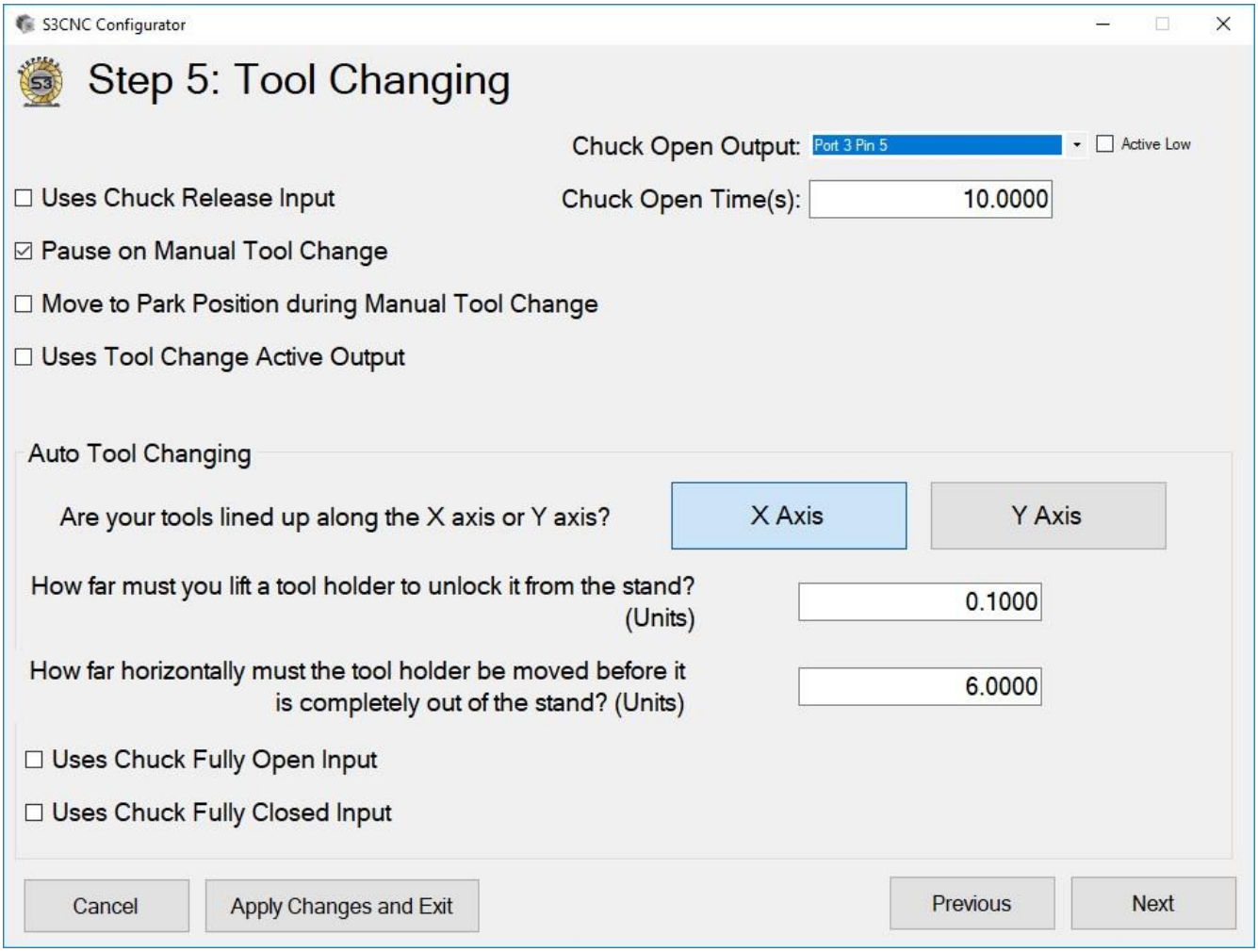
There are 4 options available regardless of whether or not automatic tool changing is used. If you are using auto tool changing, the option *Uses Chuck Open Output* is not present because it must be enabled.

- *Uses Chuck Open Output* - If enabled, you will have to select an output pin that is used to control the chuck. When performing tool changes, the chuck can be opened by the software

with a timer to automatically close it. The time until close is set with the *Chuck Open Time* option that appears when this checkbox is enabled

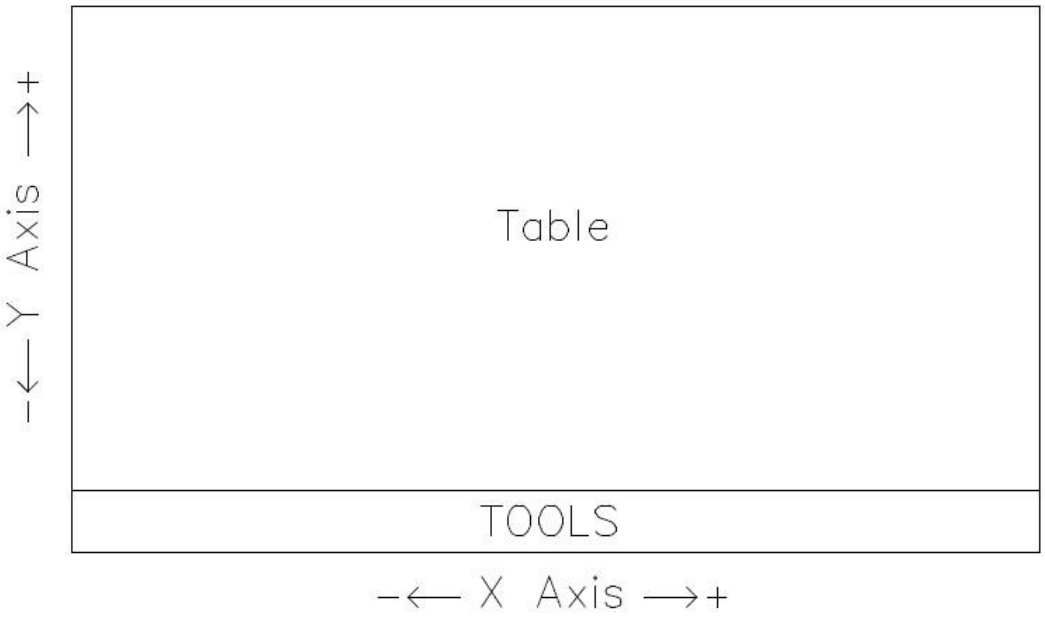
- *Uses Chuck Release Input* - If enabled, you will have to select an input that is used to control a chuck release. When this input is on, the software opens the chuck
- *Pause On Manual Tool Change* - If enabled, the software will pause and wait for the user to perform a tool change at each M6
- *Move to Park Position during Manual Tool Change* - If enabled, the software will move the axes to the park position before waiting for the user to change the tool

2.3.8.1 Auto Tool Changing Settings

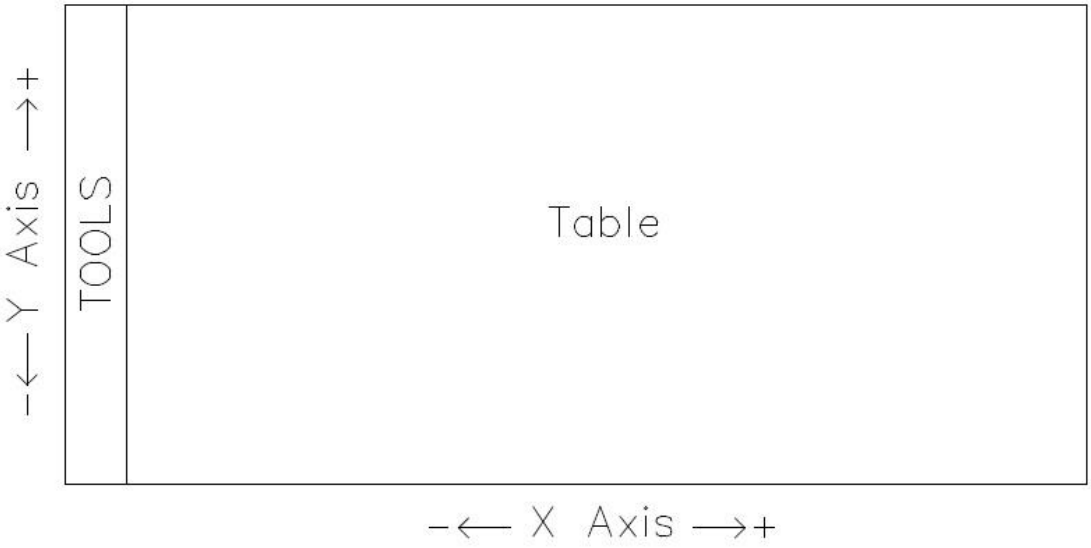


The Auto Tool Changer has the following options:

Are your tools lined up along the X axis or Y Axis? Your tool rack keeps all of the tools in a line. You should select the axis that the rack is parallel to. For example, if your table looks like this from above:



... then you should select "X Axis", and if it looks like this:

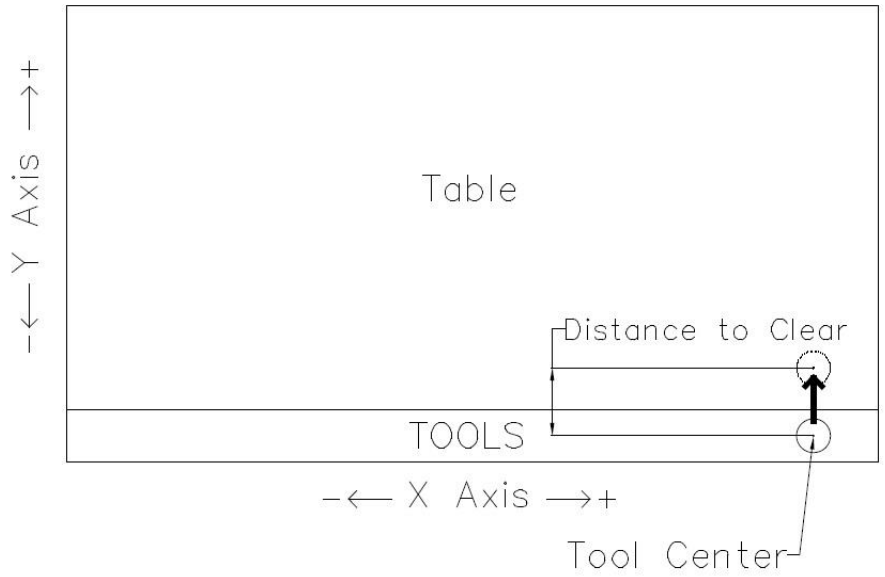


... then you should select "Y Axis".

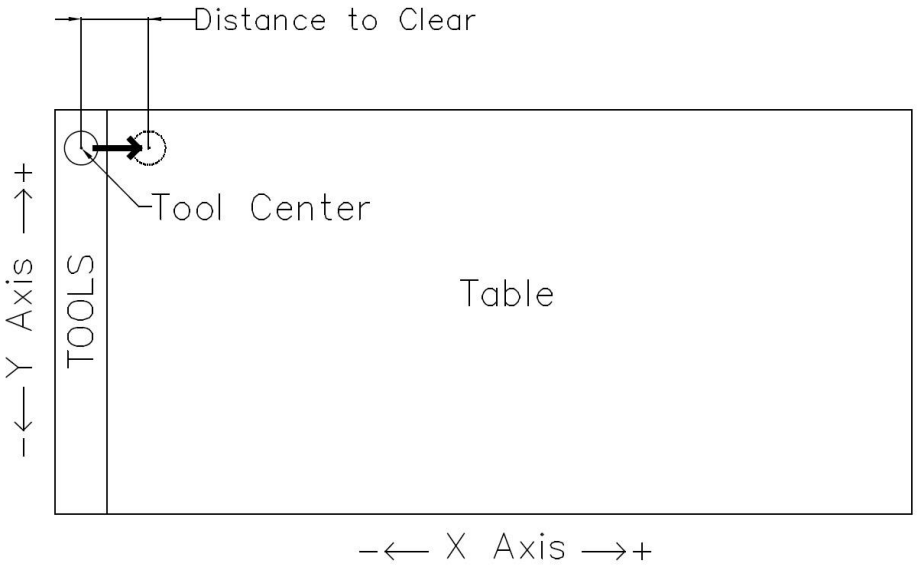
How far must you lift a tool holder to unlock it from the stand? The tool holders in each stand may be locked down, depending on your setup. In order to unlock them, they must be lifted slightly. This value is the amount to lift. Similarly, when you put a tool back, it must be pressed down this amount before releasing the tool holder to ensure it is securely placed into the stand.

How far horizontally must the tool holder be moved before it is completely out of the stand After a tool has been grabbed and lifted (if at all), it must be moved horizontally out of the stand. This number is how far horizontally it moves, and must be a large enough number to clear the stand (distance to clear in the diagrams below). The tool is always move inward towards the table as in:

Tools on X Axis:



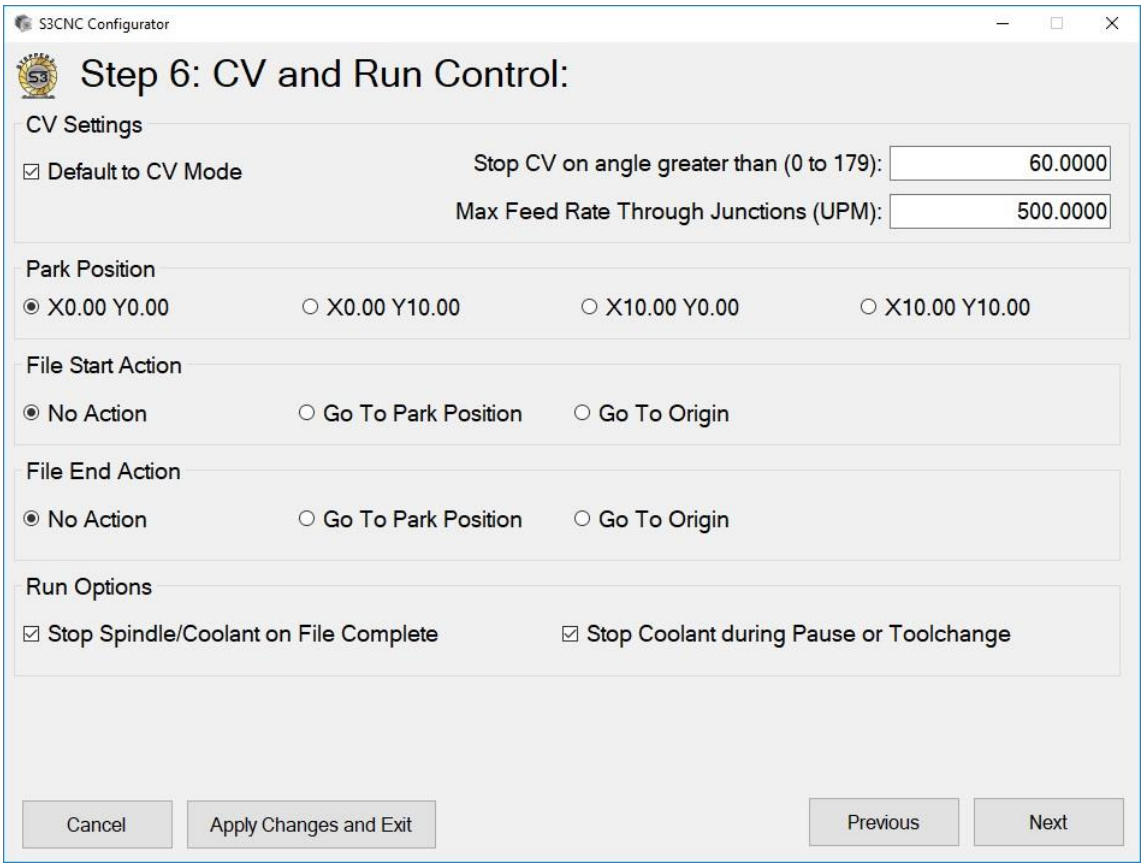
Tools on Y Axis:



Uses Chuck Fully Open Input If enabled, you must select an input pin for the chuck fully open signal. This is used to detect that the chuck has been opened during auto tool changing. Without this signal, after opening the chuck, the software will assume success. This signal provides extra safety to avoid machine malfunction in the event of a jam.

Uses Chuck Fully Closed Input If enabled, you must select an input pin for the chuck fully closed signal. This is used to detect that the chuck is closed securely around a tool. Without this signal, after grabbing a tool and closing the chuck, the software will assume success. This signal provides extra safety to avoid machine malfunction in the event of a jam.

2.3.9 Step 6: CV and Run Control

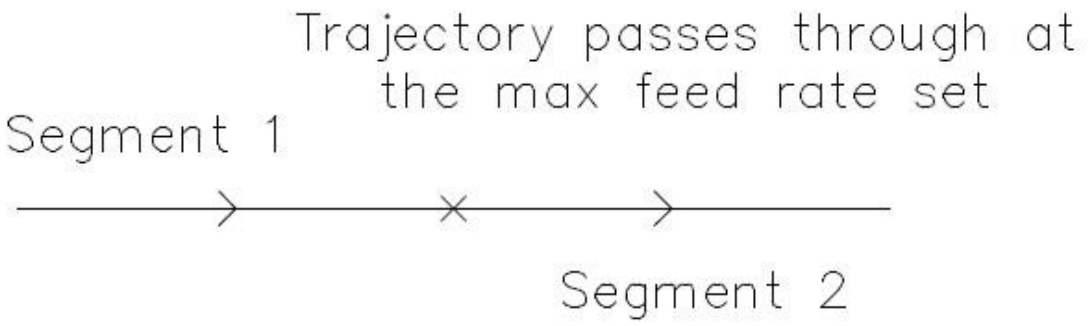


2.3.9.1 CV Settings

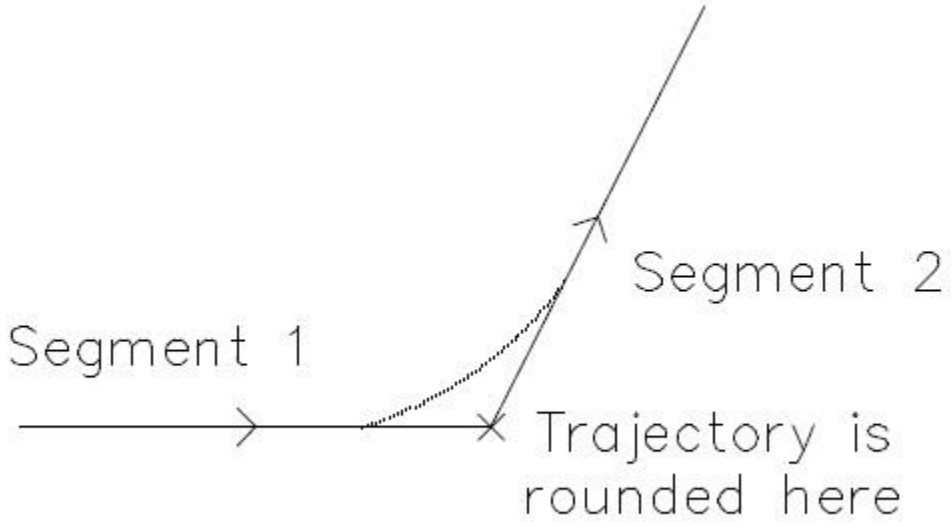
If the **Use CV Settings** option is checked, you can configure the constant velocity mode of S3CNC to ensure proper behavior with your files. The following settings are available:

Stop CV on angle greater than (0 to 179) This is an angle from between 0 and 179 that decides how sharp a turn must be in order to make the planner come to a complete stop instead of passing through with a constant velocity. By coming to a complete stop, the corner does not get rounded, which may be necessary for certain parts of some cuts.

For example, let's assume that it is set to 89 degrees. If you have two G-Code lines that look like this:

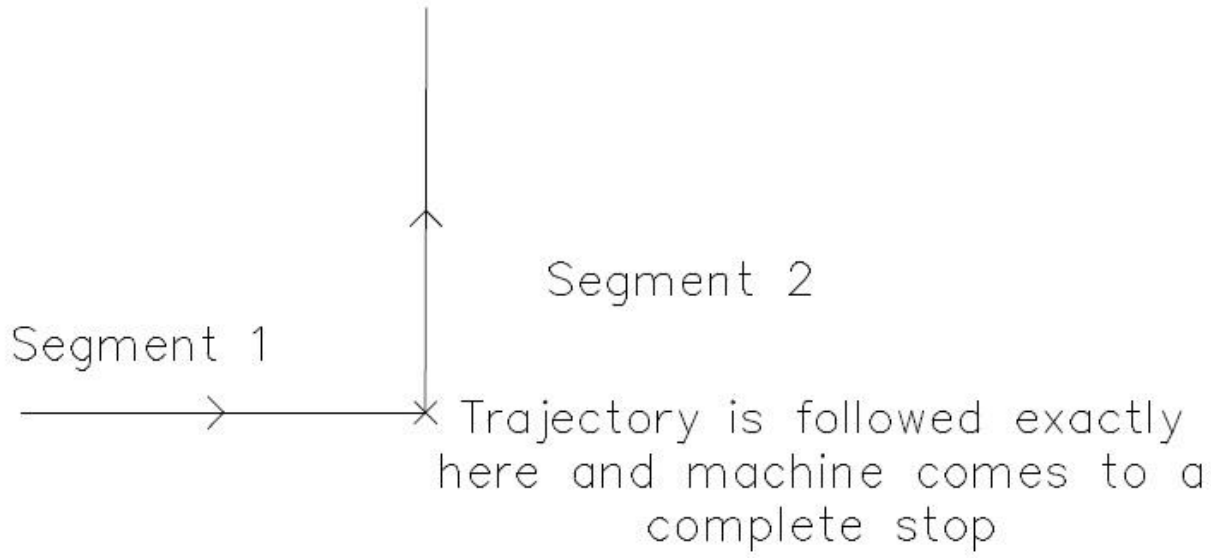


... the angle between them is 0 degrees, which is less than 89, so it will not stop at the junction between the lines. If you have:



... the angle is sharper but it is still less than 89, so it will not stop at the junction. Since it does not stop, the corner at the junction will get rounded inward based on the velocity at which it passes through.

Last, if you have:



.. then it will come to a complete stop at the junction. The angle is now 90 degrees, which is greater than 89.

Max feed rate through junctions This value decides the highest speed the software will pass through a junction at when blending junctions together for constant velocity. For example, if you set this to 0, it will behave just like exact stop mode, because it has to hit 0 units per minute at every junction between two G-Code lines. As you increase this number, the file will run faster and the constant velocity will be higher; however, the faster speeds will round corners more and so the error/deviation from the file will be greater.

2.3.9.2 Park Position

The park position is a standard position that is moved to during actions such as pausing a file, changing a tool, etc. You can select one of the four corners of your table as your park position.

2.3.9.3 File Start Action

This is an action that is performed every time you start running a file (when run from the beginning). The options available are:

- *No Action* - No special action, just run the file
- *Go To Park Position* - Move to the park position, then start the file
- *Go To Origin* - Move to the current work origin, then start the file
- *Go To Origin (Max Z)* - Move to the current work origin in X and Y, and the highest possible Z height

2.3.9.4 File End Action

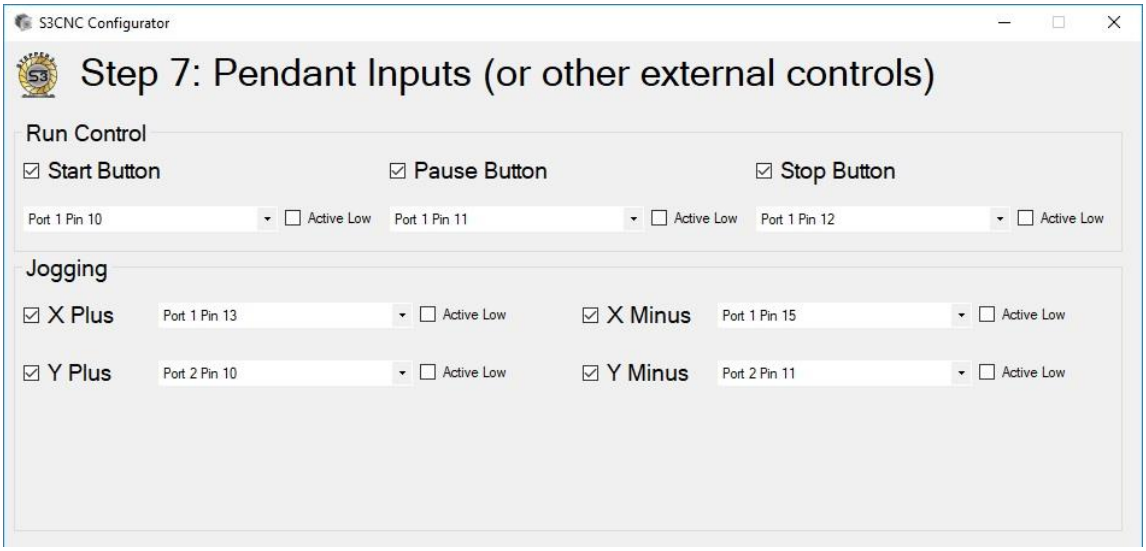
An action performed at the end when a file is completed. Has the same options available as for the start action

2.3.9.5 Run Options

There are two additional run control options:

- *Stop Spindle/Coolant On File Complete* - This is typically enabled. It makes it so that the software automatically turns the spindle and coolant off at the end of a file even if there is no explicit M5/M9. Some users may want to turn this off to keep the spindle on between multiple files, to avoid waiting for the spindle to stop and start again
- *Stop Coolant during Pause or Toolchange* - A pause or a tool change always stop the spindle. Normally, the coolant is also stopped, but you can disable this option to have them leave the coolant on

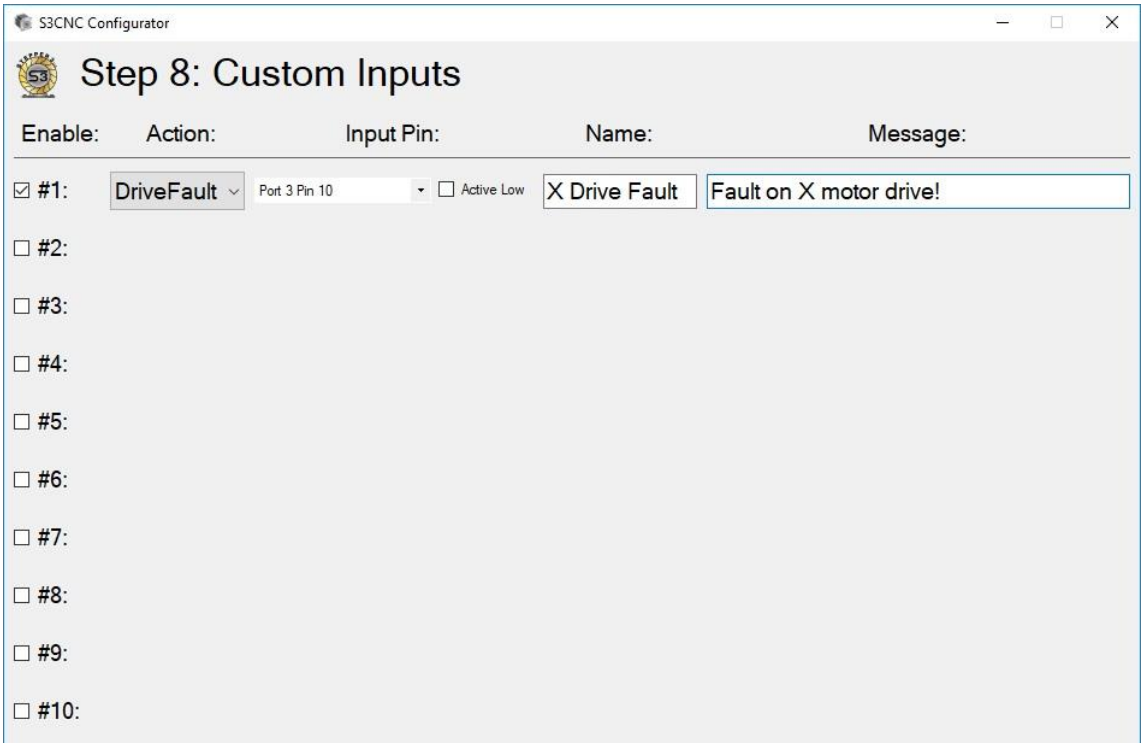
2.3.10 Step 7: Pendant Inputs



Each checkbox here enables an input for a pendant control. The following controls are available:

- *Start Button* - When this signal goes from inactive to active, the software reacts as though the start button has been pressed
- *Pause Button* - When this signal goes from inactive to active, the software reacts as though the pause button has been pressed
- *Stop Button* - When this signal goes from inactive to active, the software reacts as though the stop button has been pressed
- *Jog Plus (for each axis)* - When held active, the software will jog that axis in the positive direction
- *Jog Minus (for each axis)* - When held active, the software will jog that axis in the negative direction

2.3.11 Step 8: Custom Inputs



This section allows you to configure up to 10 custom inputs. Each custom input can be assigned 1 action as well as a message to print when it goes active. Custom inputs are used for many auxiliary features such as door alarms, drive faults, spindle faults, contact mats, etc.

Each custom input has the following options:

Enable
Check this to enable the custom input. Once enable, you must provide an input pin for the input.

Action
The action specifies what the software should do when the signal goes active. The choices are:

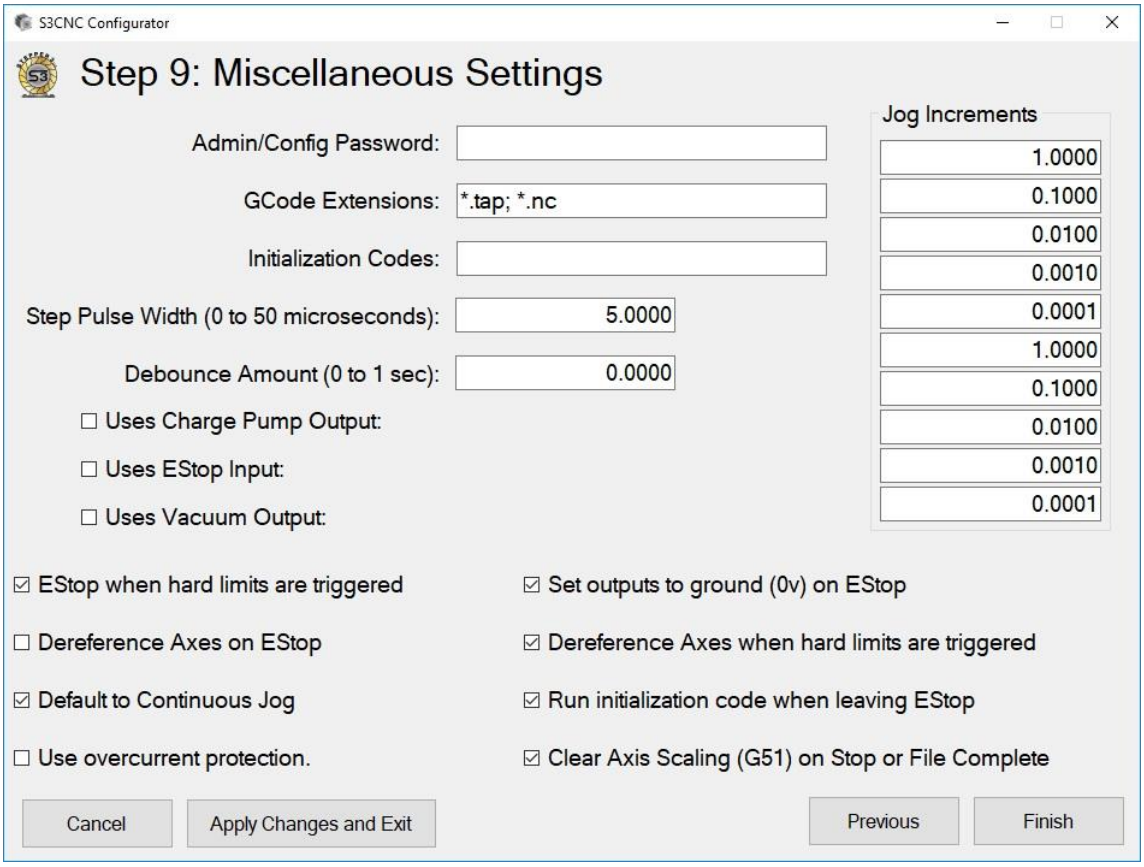
- *Message* - Just print the message to the status line
- *Stop* - Stop the currently running file and print the message
- *EStop* – Disable the software and put it in E-Stop. The system will remain in E-Stop until either the input is no longer active or the user enables the *Override Custom Inputs* mode to explicitly ignore faults from custom inputs. Prints the message
- *DriveFault* - Disable the software, put it in E-Stop, and bring up the drive recovery system. The software will reset each drive using the reset outputs in an attempt to clear the fault. Like the E-Stop action, you cannot leave E-Stop until the input is inactive or explicitly overridden

Input Pin
This input pin for the custom input.

Name
The name of the input. This is the text that is shown next to the input's diagnostic LED on the diagnostics page.

Message
This is a message that will be printed on the status line at the bottom of the software window whenever the input becomes active.

2.3.12 Step 9: Miscellaneous Settings



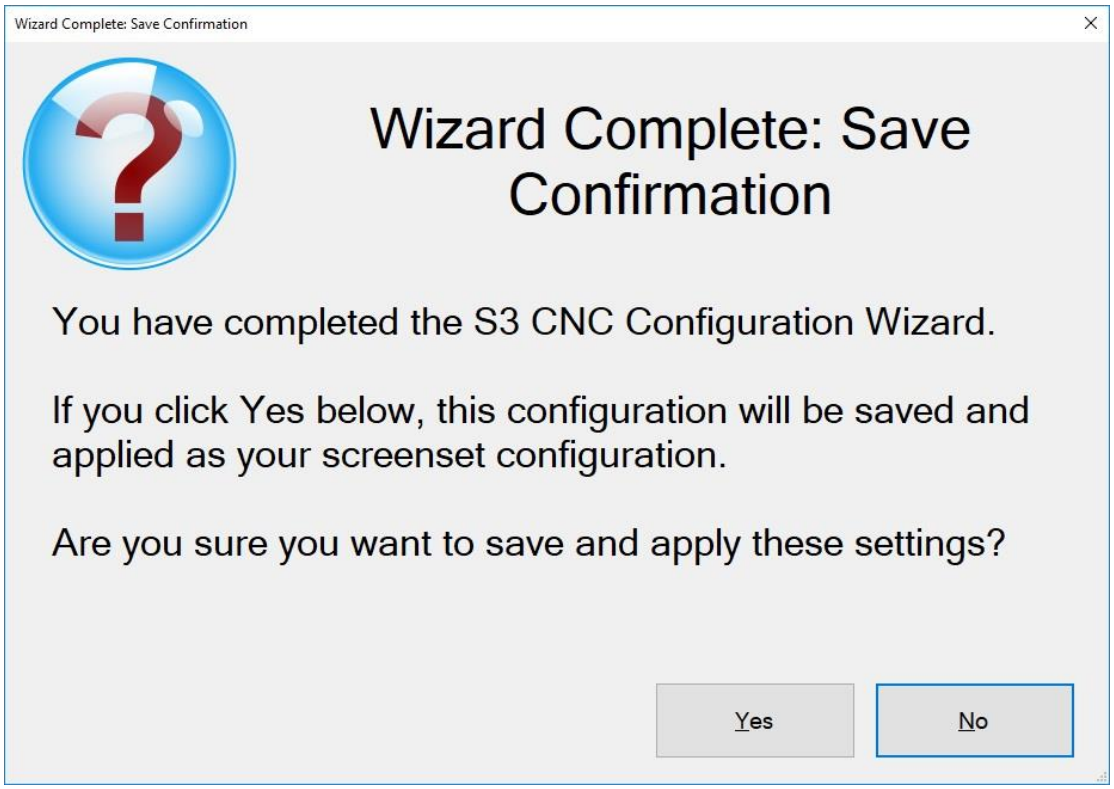
Here is where any remaining options are that did not fit into the previous categories:

- *Admin/Config Password* - This is a passphrase that must be entered on the software in order to access settings such as the configurator. This is not a secure system and is only present as a deterrent to prevent pointless tampering from unauthorized users. If you forget the password, contact Stepper3 support, it is very easy to clear it and should only take a couple minutes
- *GCode Extensions* - These are the file extensions that the File Open dialog box recognizes as G-Code files. This list is in the form: *.ext1;*.ext2;*.ext3 That is, asterisks dot extension, with

each one separated by a semicolon. There should be no spaces in this list whatsoever

- *Initialization Codes* - This line of G-Code is run whenever a new file is loaded, to establish the correct default modes for the file. The software has its own set of default modes that it applies first. Afterwards, it runs this initialization code.
- *Step Pulse Width* - The width of the step pulses, in microseconds, for all of the stepper motor axes (including stepper motor spindles) in the system.
- *Debounce Amount* - Debounce specifies a minimum amount of time that a signal must hold a particular value before the software considers it to have fully transitioned to that state. The possible range is from 0 to 1 second, but you can input very small values for this setting.
- *Uses Charge Pump Output* - If selected, you must choose an output pin on which to output the charge pump. The charge pump is a 17 kHz square wave that is used to indicate to external control systems that the software is running and enabled. This charge pump is automatically disabled in E-Stop or on software exit. Systems that use a charge pump will disable all of their outputs if it is absent
- *Uses EStop Input* - Enable this to select an input pin which triggers an E-Stop. Note that this is a software estop, not a hardware estop. A software estop is slower than the real hardware estop button because it will not stop the machine until data arrives in the software indicating that the input is active. We recommend using a hardware estop button whenever possible.
- *Uses Vacuum Output* - Enable this to select a vacuum output pin. The vacuum output is controlled by a toggle button on the main page. It can be used to control a vacuum or similar
- *EStop when hard limits are triggered* - If this is not enabled, the software will not E-Stop when hard limits are hit; it will just print a message instead. Disabling this option is not recommended
- *Set outputs to ground (0v) on EStop* - If this option is enabled, the Ether-Mach board will change all outputs to 0 during an E-Stop. If you have an external break-out board or control box that uses a charge pump, you will likely want to disable this option
- *Deference axes on EStop* - If enabled, every E-Stop will mark the axes as dereferenced and require re-homing
- *Dereference axes when hard limits are triggered* - Likewise, if this is enabled, hitting a hard limit will dereference the axis that hit the limit
- *Default to continuous jog* - If enabled, the jog mode will be continuous on startup instead of incremental
- *Run initialization code when leaving EStop* - As described
- *Use overcurrent protection* - This option is very important for servo based systems. When this is enabled, the software will monitor the hard limits and shut off the drives if the axes are ever held at the limit switches for too long. This behavior prevents servo drives from overheating from a following error at the hard stops of the table
- *Clear axis scaling (G51) on Stop or File Complete* - If this is disabled, the software will not clear axis scaling when a file stops running. This means that a file that changes the scale of the axes (ex, doubles everything) will cause that effect to persist after the file is done. If this is disabled, it is the machine operator's responsibility to ensure that the scaling is correct at all times
- *Jog increments* - A collection of 10 jog increments. The Cycle Jog Step button on the main page of the software will cycle through each of these options for incremental jog move size

2.3.13 Finish



After you click finish, the Configurator will ask you to confirm your changes and then save/apply them to the screen. The software will restart and jump back to the main page, and your changes have been applied. Remember, unless it tells you to, you do not need to restart the software to make configuration changes.

3 Section 3: Software Overview

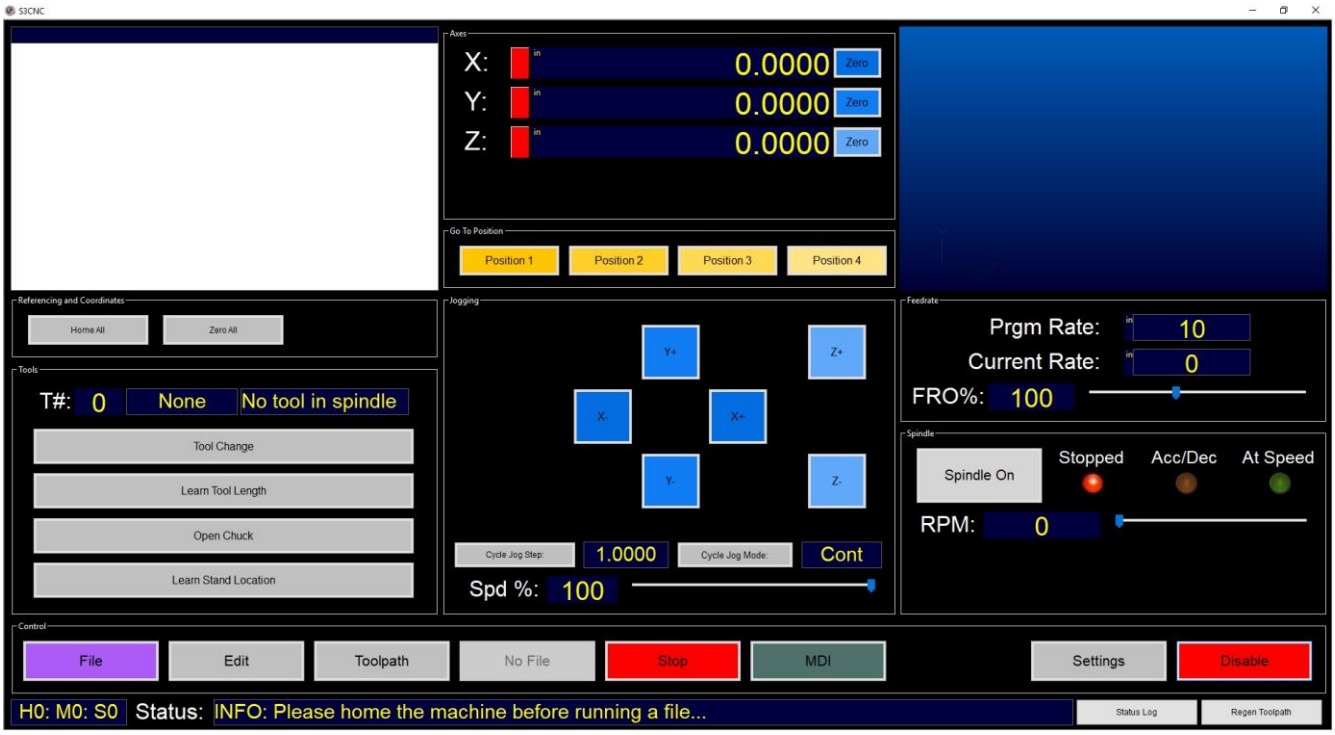
The software has three pages:

- The *Main Page*, where you spend most of your time. Controls for running files and jogging are here, along with control of your spindle or torch
- The *Toolpath Page*, which shows a larger toolpath view than the one on the main page
- The *Settings Page*, which has numerous diagnostics and settings

The settings page is divided among three tabs:

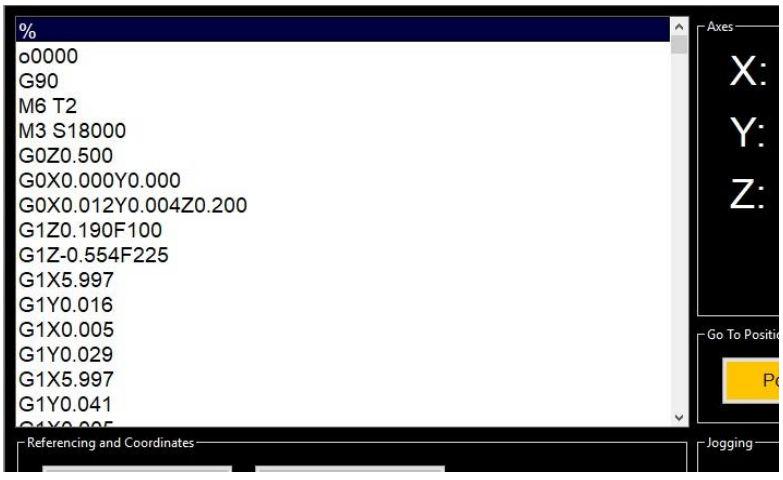
- *Diagnostics*, which shows diagnostic info on the state of inputs and outputs (IO) and the axes coordinate systems. It also has override controls for faults, limits, and relays
- *Goto Positions*, which lets you customize the Goto Position buttons on the main page
- *Settings*, which contains settings such as THC parameters and delay timings

3.1 Part 1: Main Page



The Main Page is used for daily operation and control. It is the page shown on startup. It has the following main panels:

3.1.1 G-Code Window



This windows shows the G-Code text of a currently open file. While a file is running, it highlights the current line. When a file is not running, you can click on a line to select it as the next line to run. For example, to rewind a file and start at the beginning again, you would just click on line 1.

If the first line of a G-Code file is selected, running the file will start at the beginning, so the start

button is labeled "*Start*", if any later line is selected, you would start at that point, in the middle of the file, so the button is labeled "*Run from Here*". *Run from Here* differs from just starting a file because the software performs a series of startup actions to get the table into the correct initial conditions for running the file at that point.

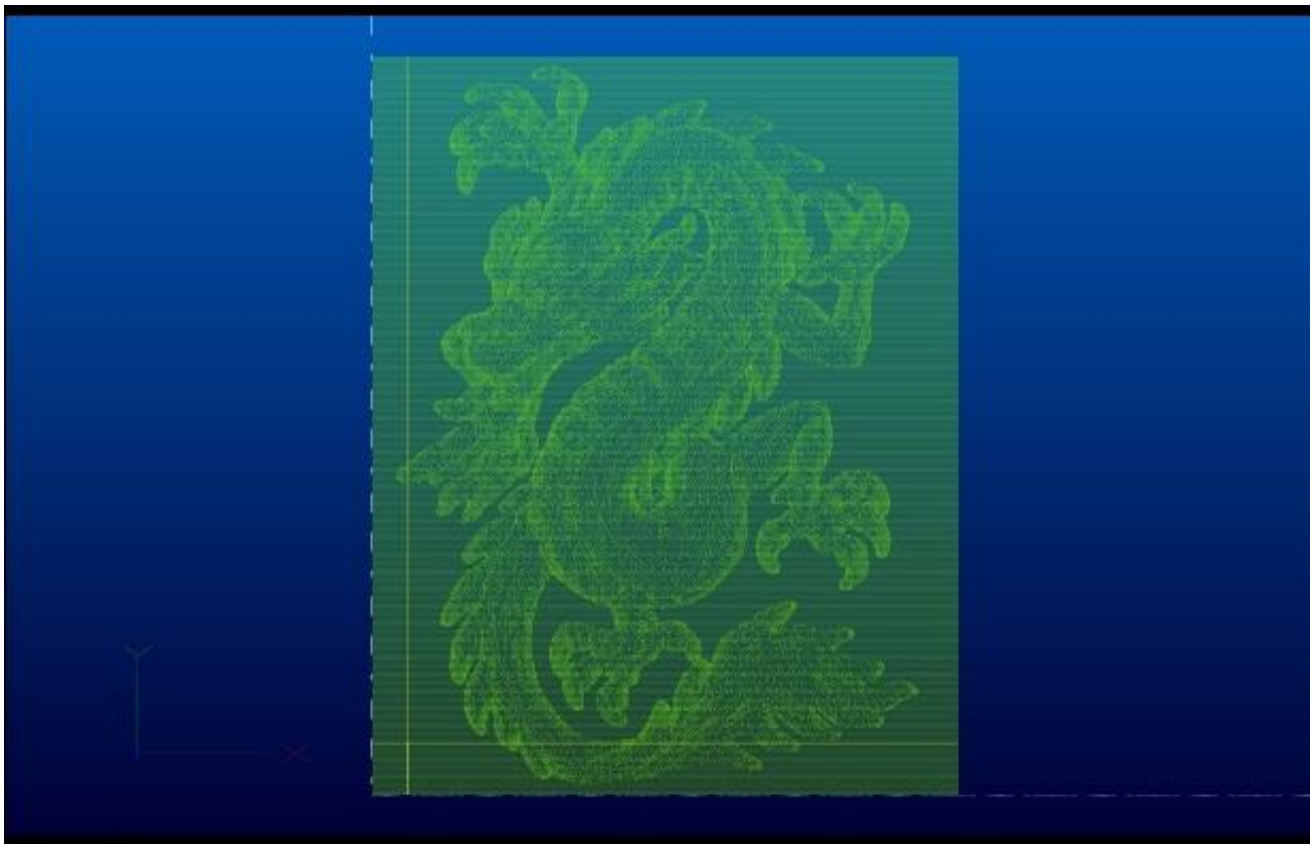
To see line numbers in the G-Code window, double click it. Line numbers will appear on the left side.

3.1.2 Axes Digital Readouts (DROs)

The axes digital readouts (DROs) show the current work coordinates for each axis. The indicator LED on the left side shows whether the axis has been homed or not. Red means un-homed and green means that the axis has been homed. On the right side, there is a zero button for each axis. These buttons change the current fixture offset to make the current position location 0 for that axis. When pressed, the axis DRO will update to read "0".

In addition to using the zero buttons, you can change the offsets by typing directly into the axis DROs. For example, if you want your current X axis position to be location $x = 3$, just type 3 into the X axis DRO and hit enter. The fixture offset will be updated such that your current position is now 3.

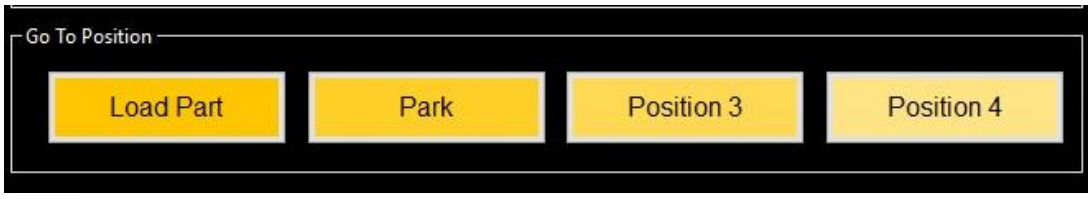
3.1.3 Toolpath



The toolpath shows a 3D rendering of the paths described by the currently loaded G-Code file. It also shows a bounding box representing the soft limits of the table.

Left click and drag to rotate the view. Right click and drag (or scroll with the scroll wheel) to zoom in and out. Left click and drag while holding control to pan (or left and right click and drag). Double left click to switch to a full top-down view and double right click for a 45 degree projection.

3.1.4 Go To Position



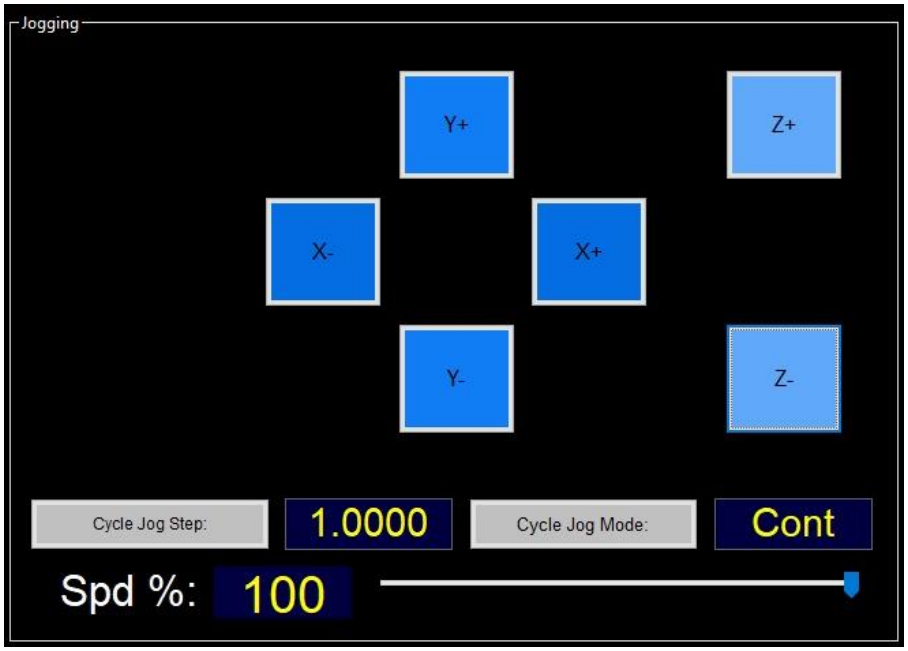
Each of these four buttons runs a Manual Data Input (MDI) to move to the position configured for that button. Each of these 4 buttons are configurable using the *Goto Positions* tab of the settings page. You can set each button to move to an origin, absolute, or incremental coordinate as well as rename them (change the text like above).

3.1.5 Referencing and Coordinates

The referencing and coordinates panel has up to three buttons used for referencing and zeroing the system:

- *Home All*, homes all axes. If any axes are not homed, the software will not allow actions other than jogging. So, you should typically home your axes first when you start up the software. The software does not home the axes automatically when the software starts for safety, and to allow the user to jog the axes most of the way first, saving time
- *Zero All*, This is equivalent to clicking the Zero button for every single axis. It sets your current work coordinates all to zero, making your current position the work origin
- *Touch Off Z*. When pressed, the Z axis will probe down until the probe input is triggered, at which point it will compute a new work offset. This button is used to zero the Z axis using a probe or touchpad

3.1.6 Jogging



On the jogging panel, there is a + and – button for each axis, to jog it in the forward or backward directions respectively. Additionally, there are the following:

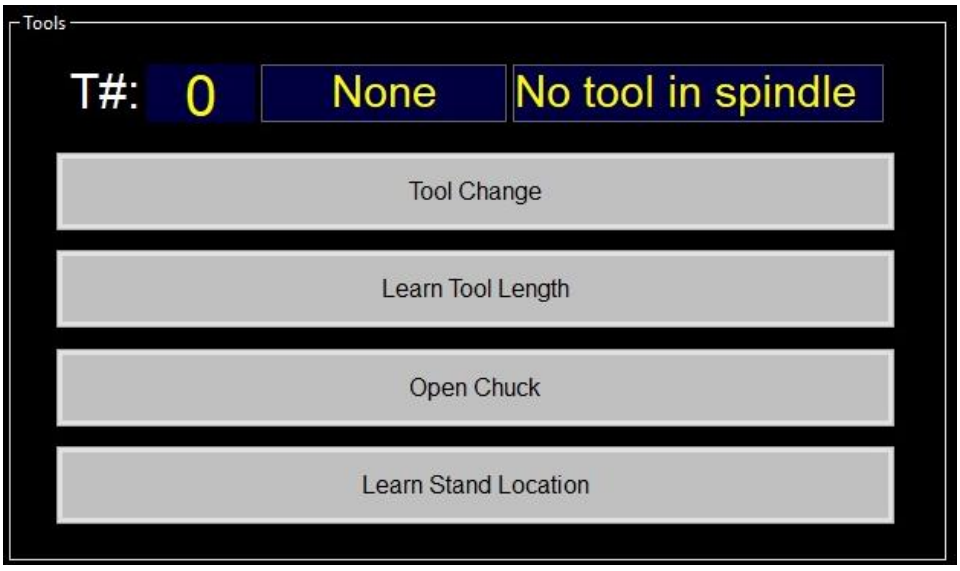
- *Cycle Jog Step* - Cycles through the available jog increments for incremental jogging. These 10 values are configured in the *Miscellaneous Settings* page of the Configurator

- *Cycle Jog Mode* - Switches between continuous and incremental jogging
- *Spd %* - This can be set by typing into the DRO or using the slider. Sets the speed of jogging as a percentage of maximum axis velocity

3.1.7 Feedrate

This panel shows the current feed rate and allows you to change the feed rate override percent. The *Prgm Rate* (program rate) is the current target feed rate as specified by the F G-Code in the program. The *Current Rate* is the instantaneous speed of the machine at that moment, which may be lower than the program rate at corners, turns, etc. The feed rate override percent (*FRO%*) is a percentage scaling of the program rate, from 1 percent to 250, which allows you to change speed while a program is running.

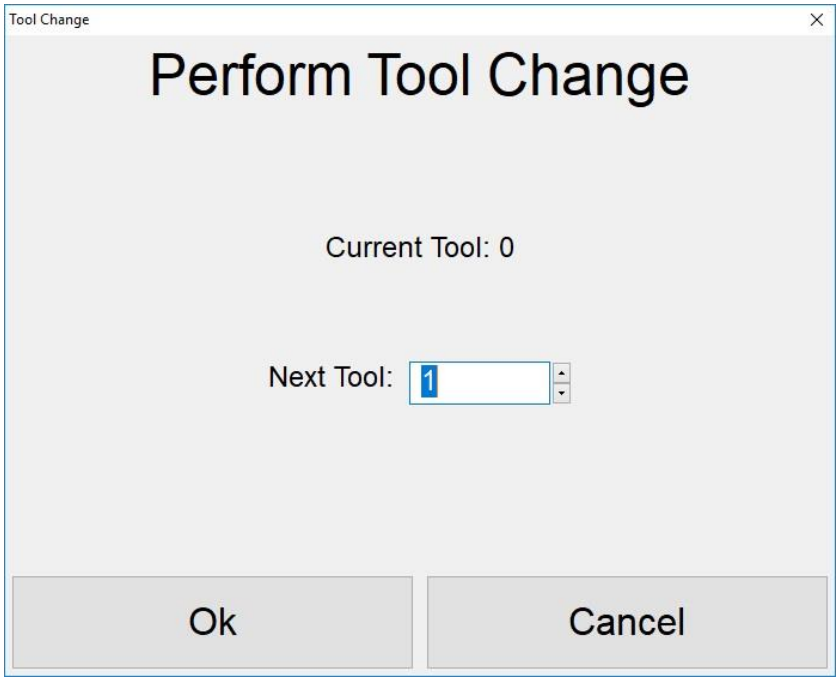
3.1.8 Tools (Only in router mode)



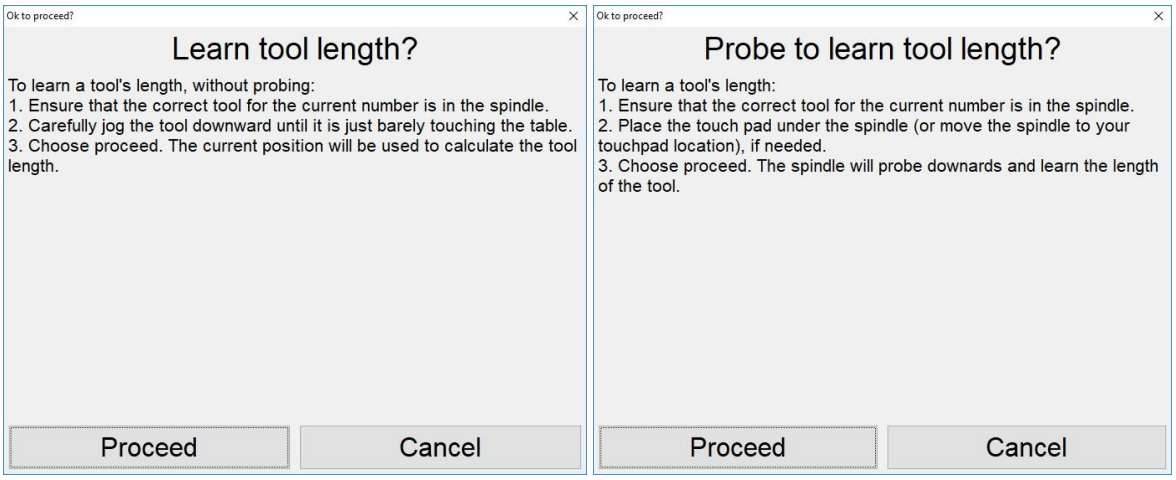
In router mode, the left side has a tools panel. At the top left of the tools panel is the *tool number DRO*, indicating the tool number of the current tool in the spindle. Changing this tool number does not initiate a tool change - it just directly changes the tool number. You can do this to correct the tool number if it does not match the tool in the spindle. To the right of the tool number are two text boxes that show the *Name* and *Parameter* fields from the *tool table*, for that tool. These fields can be filled

in to show details about the tool. Note that tool number 0 always represents the empty tool, indicating that there is no tool present. If you have no tool in the spindle, you should be on tool number 0.

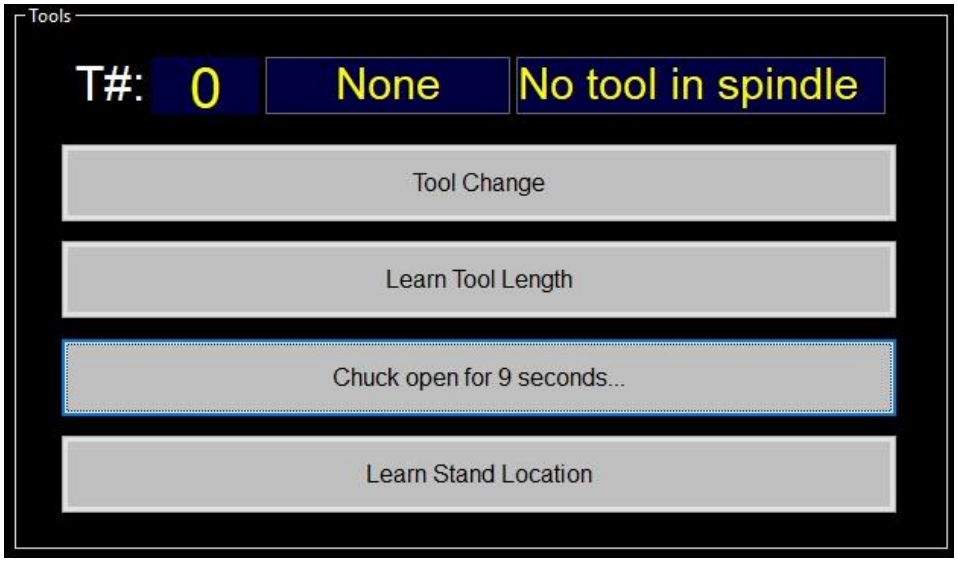
Below the tool number there are up to 4 buttons:



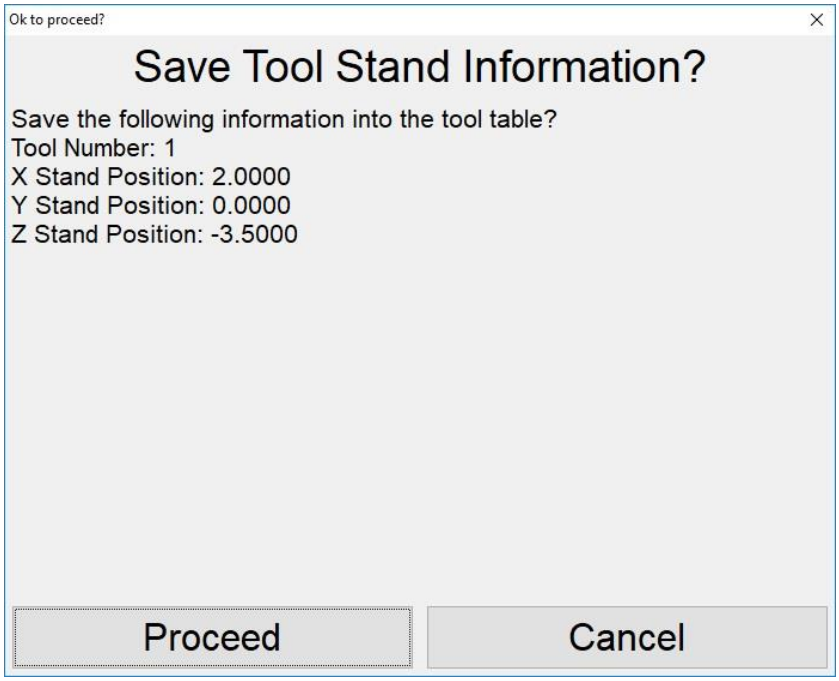
- *Tool Change* - Opens a dialog to select a new tool and perform a tool change. If auto tool changing is enabled and the current tool has auto tool changing enabled in the tool table, it will get dropped off automatically. Similarly, if the selected tool has auto tool changing enabled, it will get picked up automatically. The software will wait for a manual tool change for tools with auto tool changing turned off. If the Uses auto tool changing option is disabled in the Configurator, all tools are changed manually



- *Learn Tool Length* - Learn the length of the current tool either manually or with probing. If probing is enabled, the Z axis will probe down until the probe triggers and then compute the length. If there is no probing, the user will have to jog the Z axis down to a surface themselves. Remember to set the empty tool probe depth on the settings page - the software needs to know the depth of an empty tool in order to compute the difference and determine length

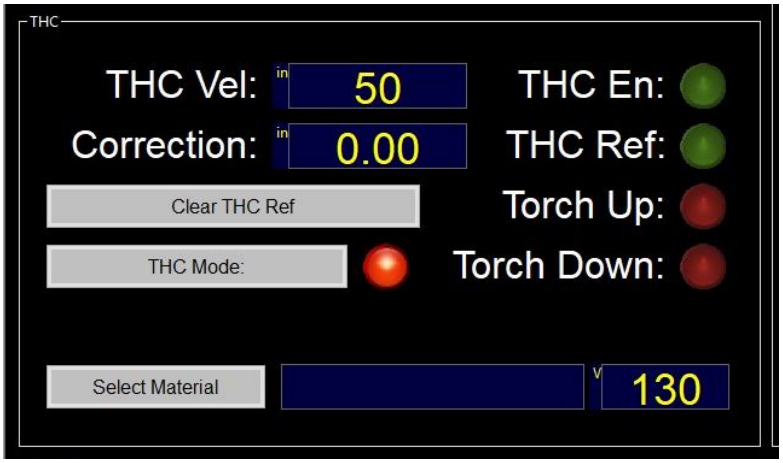


- *Open Chuck* - Opens the chuck and begins a countdown to automatically close it. Clicking again resets the counter



- *Learn Stand Location* - Saves the current absolute position as the stand location of the current tool. This should be the position of the tool holder when it is locked down into the tool stand; that is, this is the precise spot where the tool would be grabbed by the chuck when auto loading the tool

3.1.9 THC (Only in THC plasma mode)



The Torch Height Control (THC) panel has two DROs:

- *THC Vel* - The velocity of the up/down movement of the THC
- *Correction* - The current amount of offset from the original Z position applied by the THC

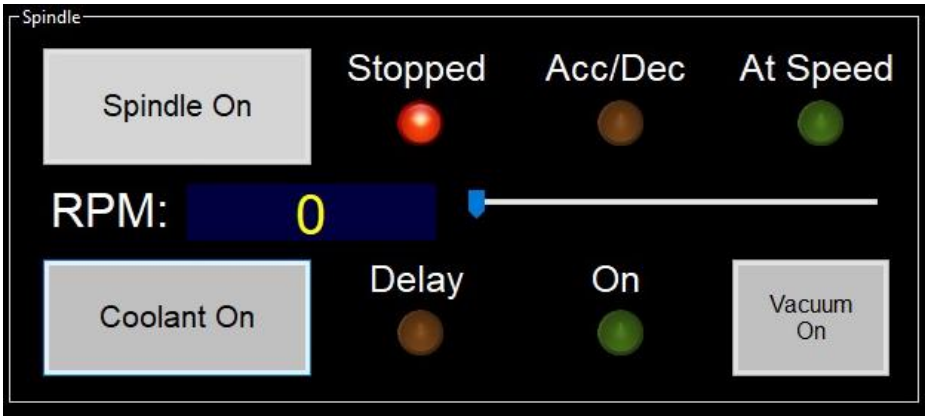
The THC panel has the following LEDs:

- *THC En* - Illuminated when the THC is active. While this is lit, G-Code Z commands are ignored and instead the Z axis is driven by the Ether-Mach THC control
- *THC Ref* - Illuminated when the Z axis has been zeroed, establishing positions for the pierce height, ignition height, etc.
- *Torch Up* - Shows the state of the Torch Up input pin
- *Torch Down* - Shows the state of the Torch Down input pin
- *Probe* - Shows the state of the probe input pin

The THC panel has the following buttons:

- *Clear THC Ref* - Clears the *THC Ref* flag. This means that you will have to re-zero the Z axis before THC can activate. The zero can be done automatically by probing when the M3 runs using probe modes once or always
- *THC Mode* - If enabled, THC activates at the end of an M3, otherwise, it is not activated
- *Probe Mode* - There are three probe modes which decide how the Z axis gets zeroed. *Never* means that the software never automatically probes. Instead the user must manually probe or zero the Z axis. *Once* means that it will probe in the M3 if the THC Ref LED is not lit. When the reference gets cleared, it will probe again. *Always* means that the software will automatically probe to zero Z in every M3
- *Select Material* - Opens a dialog to select a material from the material list and load the THC parameters set for it. Shows the material name and the target torch voltage for the current material to the right of the button

3.1.10 Spindle (Only in router mode)



The spindle panel has buttons used for controlled the spindle and related outputs.

3.1.10.1 Spindle

- *Spindle On*/*Spindle Off* - Toggles the state of the spindle. The spindle will not react immediately, as the spindle controller in the software must await the spindle delay after each state change to ensure that the target speed is attained

- *Stopped LED* - Indicates that the spindle is currently stopped
- *Acc/Dec LED* - Indicates that the spindle delay is currently active while the spindle accelerates or decelerates
- *At Speed LED* - Indicates that the spindle is currently at speed

3.1.10.2 Coolant

- *Coolant On/Coolant Off* - Toggles the state of the coolant. The coolant will not react immediately if there is a coolant delay, as the coolant controller must wait the delay after each state change
- *Delay LED* - Indicates that the coolant delay is currently active
- *On LED* - Indicates that the coolant relay is currently on

3.1.10.3 Vacuum On/Off

Lastly, there is a vacuum button that directly toggles the vacuum output pin.

3.1.11 Torch (Only in plasma mode)

The spindle panel has buttons used for controlled the spindle and related things.

3.1.11.1 Torch

- *Torch On/Off or Toggle Torch* - Without THC, the button reads Torch On/Off and can be toggled at any time. When toggled, the torch output does not change immediately as the torch controller must perform the torch down and torch delays. With THC, the button reads toggle THC. It cannot be toggled in a running file or while the torch is currently changing state. This is because the THC startup is more complex (has to move to pierce delay, etc. and does not just consists of an output toggle)
- *Delay* - Indicates that the torch delay is currently active
- *Torch On* - Indicates the state of the torch on output pin
- *Arc Ok* - Indicates the state of the Arc Ok feedback. If THC is available and THC Mode is active, this shows the state of the THC On input

3.1.11.2 Coolant

- *Coolant On/Coolant Off* - Toggles the state of the coolant. The coolant will not react immediately if there is a coolant delay, as the coolant controller must wait the delay after each state change
- *Delay LED* - Indicates that the coolant delay is currently active
- *On LED* - Indicates that the coolant relay is current on

3.1.11.3 Vacuum On/Off

There is a vacuum button that directly toggles the vacuum output pin.

3.1.11.4 Torch Mode / Trial Cut Mode

This button toggles between torch and trial cut mode. In trial cut mode, M3 and M5 codes do nothing.

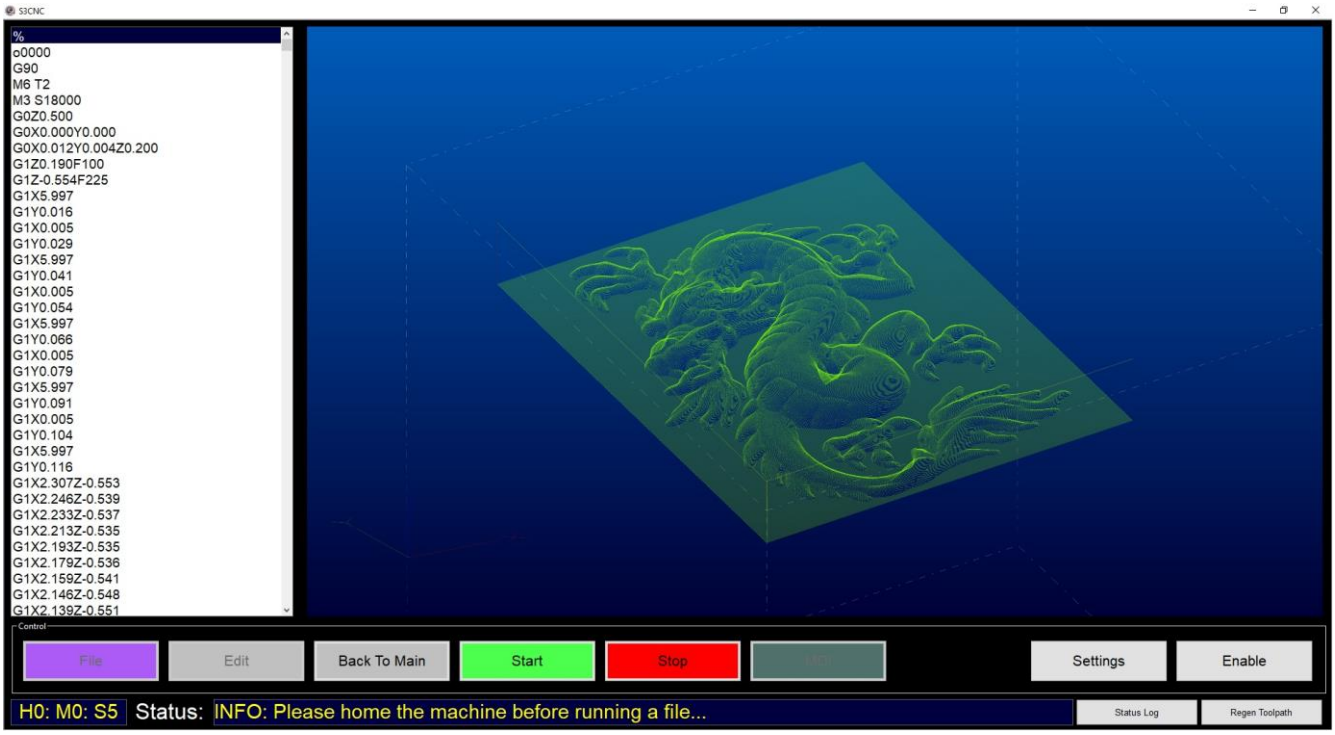
3.1.12 Control

The control panel contains the primary control buttons for the software. It has the controls used for typical operation such as for loading and running files.

Unlike all of the other panels, the control panel is present at the bottom of every page.

- *File* - Opens the file browser to select a G-Code file to load. Note that you do not need to close a currently open G-Code file to open another one. To open a new file you just hit File and select it
- *Edit* - Opens the G-Code file editor to edit the currently loaded G-Code file. After saving the file and closing the editor, the file will be reloaded, incorporating any changes
- *Toolpath* - Switches back and forth between the main page and the toolpath page
- *Start/Run From Here* - Once a file is loaded, the button will read *Start* if the top line is selected or *Run From Here* if any later line is selected. Pressing it runs the file starting at the selected line
- *Pause/Stop* - In router mode, while a file is running, this button will say *"Pause".* Pressing it will stop the current file, lift the Z axis, turn the spindle off, and move the spindle to the park position. At other times, the button reads *"Stop"*. Hitting stop cancels any currently running file or MDI and shuts off the spindle/torch and coolant
- *MDI* - Opens the MDI entry dialog. Here you can type in a G-Code line and hit enter to execute. Below the entry line is a history to load previously entered commands
- *Settings* - Switches back and forth between the settings page and the main page
- *Enable* - The E-Stop button. Switches the machine between the enabled and disabled state. An E-Stop shuts off all outputs and stops any running G-Code

3.2 Part 2: Toolpath Page



The toolpath page moves everything out of the way and shows you a large view of the G-Code Window and the Toolpath. At the bottom is the control panel with all the same buttons as the main page.

3.3 Part 3: Settings Page

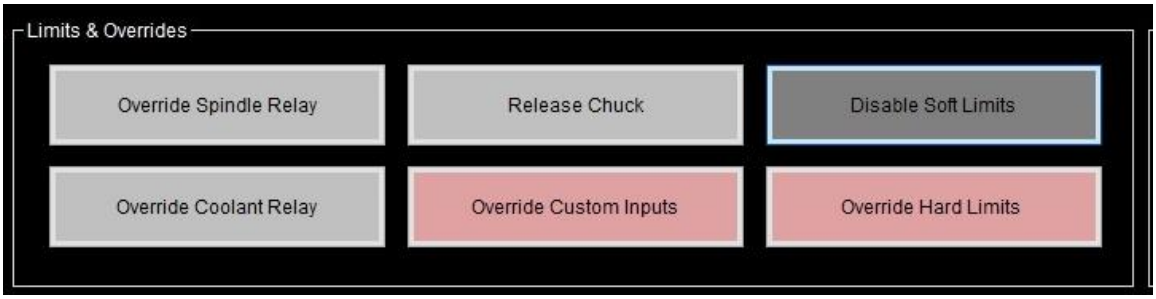
The settings page has three tabs:

- *Diagnostics* - Contains diagnostic controls and indicators
- *Goto Positions* - Allows you to customize the main page's Goto Position buttons
- *Settings* - Contains additional software settings

3.3.1 Diagnostics Tab

The following subsections go over all the panels within the Diagnostics Tab.

3.3.1.1 Limits & Overrides



These controls are used for testing and fault recovery. They should not be used during daily operation.

- *Override Spindle Relay* or *Override Torch Relay* - Direct control of the spindle/torch relay. Immediately toggles the output without waiting for the spindle delay
- *Override Coolant Relay* - Immediately toggles the coolant relay without waiting for the coolant delay
- *Release Chuck* - Immediately opens the chuck and keeps it open, without counting down the chuck close delay
- *Override Custom Inputs* - When enabled, custom inputs that have the E-Stop or Drive Fault

action are ignored (they only print their message) and do not cause an E-Stop. This allows the operator to release E-Stop and clear the condition causing it, after which they should press this button again to turn the custom inputs back on

- *Disable Soft Limits* - Turns off soft limits. Soft limits are a software feature where by the software prevents G-Code movement or jogging from moving the axes beyond the range of motion set in the configurator. If you turn this off, there is nothing stopping you from ramming your axes into your hard limits or physical stops. We do not recommend disabling soft limits
- *Override Hard Limits* - Once pressed, hard limit inputs are ignored and no longer cause an E-Stop. This allows the user to move the axes to clear the limit, at which point the button should be pressed again to re-enable hard limits. After 60 seconds from when the override button is first pressed, hard limits will automatically re-enable themselves. The operator should rarely, if ever, have to press this button, because when a hard limit occurs, the software brings up a hard limit recovery wizard that manages the process of clearing the fault for the user

3.3.1.2 Tables

- *Fixture Table* - The fixture table allows you to directly edit the values for all of the available fixture offsets. Find the row for the offset you want to edit, and just click and edit each value in the row as needed. See the Fixture Offsets Table section for information on using the Fixture Table
- *Tool Table* (Router Mode Only) - Opens the tool table. See the Tool Table section for information on using the tool table
- *Material Table* (THC Plasma Mode Only) - Opens the material table. See the Material Table section for information on using the material table

3.3.1.3 Axis Diagnostics

The axis diagnostics shows how your current work position is constructed. They are formed from the computation:

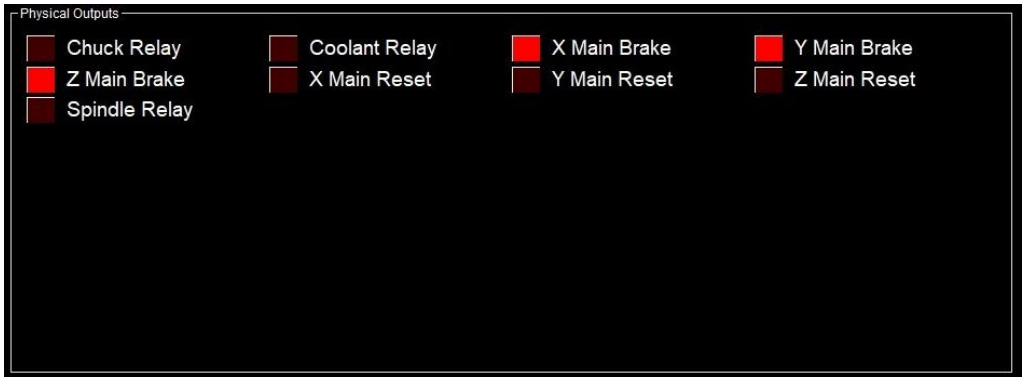
```
Work_Position = Machine_Coords - Work_Offset - G92_Offset - Head_Shift - Work_Shift - Tool_Offset
```

- *Work_Position* - This is the position shown in the axis DROs on the main page.
- *Machine_Coords* - These are also known as absolute coordinates. This coordinate is the location of the tool relative to the home switch. Machine coordinates establish a fixed, repeatable coordinate for each position on your table. Your current position comes from starting with the machine coordinates and then apply a series of offsets
- *Work_Offset* - The current fixture offset
- *G92_Offset* - An offset applied by the G92 G-Code
- *Head_Shift* - An additional offset that can be set using G-Code. This offset is normally used to select between multiple spindle heads
- *Work_Shift* - An additional offset that can be set using G-Code. This offset is normally used to select between multiple table regions
- *Tool Offset* - Offsets from tool parameters. Generally, only the Z axis will be offset

3.3.1.4 Physical Inputs

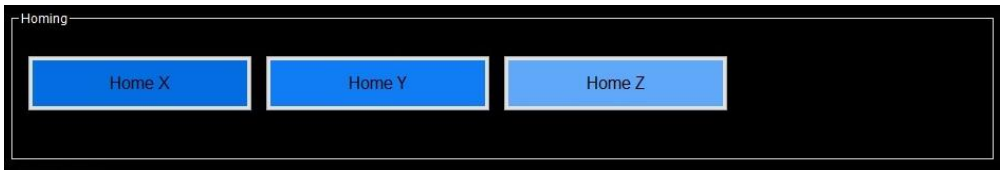
This panel shows one LED for every physical input that was set up in the Configurator.

3.3.1.5 Physical Outputs



This panel shows one LED for every physical output that was set up in the Configurator.

3.3.1.6 Homing



This panel has individual home buttons for each axis. If no axes have home inputs, this panel is not visible.

3.3.1.7 System Status

G-Code State shows the state of the system. It has many different states, the most notable being:

- *IDLE* - Doing nothing
- *JOG* - Jogging
- *FRUN* - Running a file
- *MRUN* - Running an MDI

Connectivity shows the connection status to the Ether-Mach board.

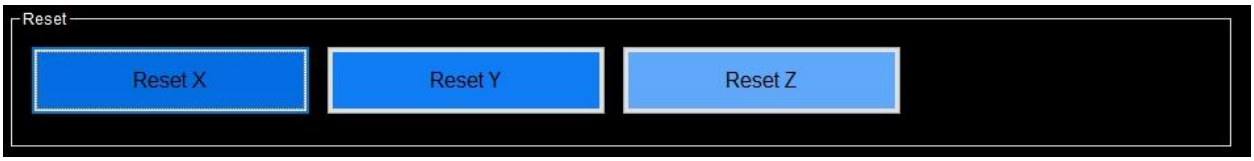
Modeline shows the currently active G-Code modals.

3.3.1.8 Spindle PWM

This panel is only visible if you are using an Analog Speed Control spindle.

It shows two read only DROs, *Duty Cycle*, which is the current duty cycle of the PWM, and *Speed*, the current spindle speed in RPM.

3.3.1.9 Reset



This panel is only visible if at least one axis has a reset output.

Pressing the reset button for an axis will:

1) Close the brakes for the axis
2) Pulse the reset output, resetting the drive
3) Wait about 4 seconds
4) Release the brake

Resetting an axis can be used to clear faults caused by servo drive overcurrent protection. You should rarely need to press these button manually, since a Drive Fault brings up an overcurrent recovery wizard that should guide you through the process of resetting the drives.

3.3.2 Goto Positions Tab

3.3.2.1 Goto Position Button Configuration

The *Goto Positions* tab has four identical panels, which allow you to configure each *Goto Position* button on the main page. For each button, you can configure:

3.3.2.1.1 Position Mode

This selects the coordinate system used for the movement. You can choose from:

- *Absolute* - Move uses absolute, or machine, coordinates. These coordinates are always relative to the home location. For example, in absolute coordinates X = 5 means 5 units in the positive direction from the home location. After such a move, your X DRO may not read 5, since there may be offsets applied in your work coordinates that make them different from the machine ones
- *Origin* - Move using the current work coordinates. This includes your fixture offset and tool offset. For example, a move to X = 0 will move until your X DRO reads 0.
- *Incremental* - Move relative to the current position. For example, a move of X = 5 will move X 5 inches in the positive direction regardless of current location

3.3.2.1.2 Name

This text box allows you to give a name to the position. This name will appear as the text on the corresponding button on the main page.

3.3.2.1.3 X, Y, Z, and A

These DROs should contain the actual coordinate values for the position.

3.3.2.1.4 Load current position

Copies the current position into the X, Y, Z, and A DROs for this Goto button. This accounts for the currently selected position mode, so pressing this with Absolute mode loads the current machine coordinates, Origin mode loads the current position, and Incremental loads all zeroes.

3.3.2.1.5 Go To Position 1

Performs the move, just like pressing the button on the main page.

3.3.2.2 Incremental move runner

This is a simple utility for performing incremental moves. Type the movement amounts into each DRO and hit Execute Incremental Move to perform the move.

3.3.2.3 Z Offset During Move To Origin

This DRO configures the move to origin performed as a file start or file end action (See the Run Control section of the Configurator). Instead of moving to `Z = 0`, a move to origin in one of these actions will move to the position specified here. Typically, users will set this to a small value like 0.1 to ensure that the Z axis comes down close to the material but does not touch it.

3.3.3 Settings Tab

There are the following settings on the left side of the tab:

- *Use M3/M5* - If set to No, M3/M5 m-codes are ignored
- *Use M7/M9* - If set to No, M7/M9 m-codes are ignored
- *Force all toolchanges as manual* - This option is visible if you have auto tool changing enabled. When set to yes, all tool changes will be manual, even for tools that have auto tool changing enabled
- *Pause for manual changes* - If set to no, the software will not stop and wait at manual tool changes, but instead will just update the tool number and continue.
- *Empty tool probe depth* - The probe depth of the spindle with no tool present (i.e. an empty collet), in machine coordinates. This can be learned by probing by pressing the learn button

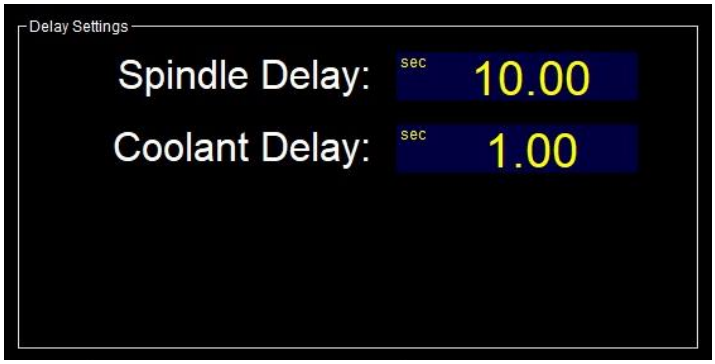
3.3.3.1 THC Settings

- *Material* - The name of the last loaded material, if any
- *Max THC Up* - The maximum number of units up from the cut height that the THC controller can move the torch
- *Max THC Down* - The maximum number of units down from the cut height that the THC controller can move the torch. This is a positive number!
- *Velocity* - The velocity of the Z axis during THC
- *Antidive Percent* - The percentage of the current feed rate below which THC stops moving the axis to prevent diving. Set this to 0 to disable antidive
- *Voltage* - The target torch voltage from the last loaded material if any
- *Pierce Height* - The height at which the torch is held during pierce. See Torch Behavior section for more details
- *Ignition Height* - The height at which the torch is activated. Can be the same as pierce height
- *Cut Height* - The height the torch moves to after the pierce delay. If THC mode is active, THC starts as soon as the torch reaches this position
- Z *Offset on Zero* - A Z offset to account for the difference between the probe point and the actual top of material. For example a Z-axis slide with a floating head will spring upwards during probing before hitting the switch. The distance traveled between the torch tip touching the material naturally and when the switch is active is the *Z Offset on Zero*

3.3.3.2 File Extents

This panel shows the extents of a file in machine coordinates.

3.3.3.3 Delay Settings



This panel allows you to change delays using by the software for managing spindle or torch state.

- *Spindle Delay* (Router only) - The delay after turning the spindle on or off. The system delays for this amount of time to ensure that the spindle reaches its target speed or comes to a complete stop
- *Pierce Delay* (Plasma only) - With THC, this is the delay after activating the torch and moving to the pierce height. After the delay expires, the cut begins and the Z axis starts moving to the cut height
- *Coolant Delay* - The delay after turning on or off the coolant
- *Torch Off Delay* - The delay after turning the torch off. This can usually be short, as torches shut off quickly, but if you have external height control systems, this gives them time to pick the torch up
- *Pierce Timeout* - The amount of time that can pass without an Arc Ok signal before a torch failure occurs

3.3.3.4 Settings and Support

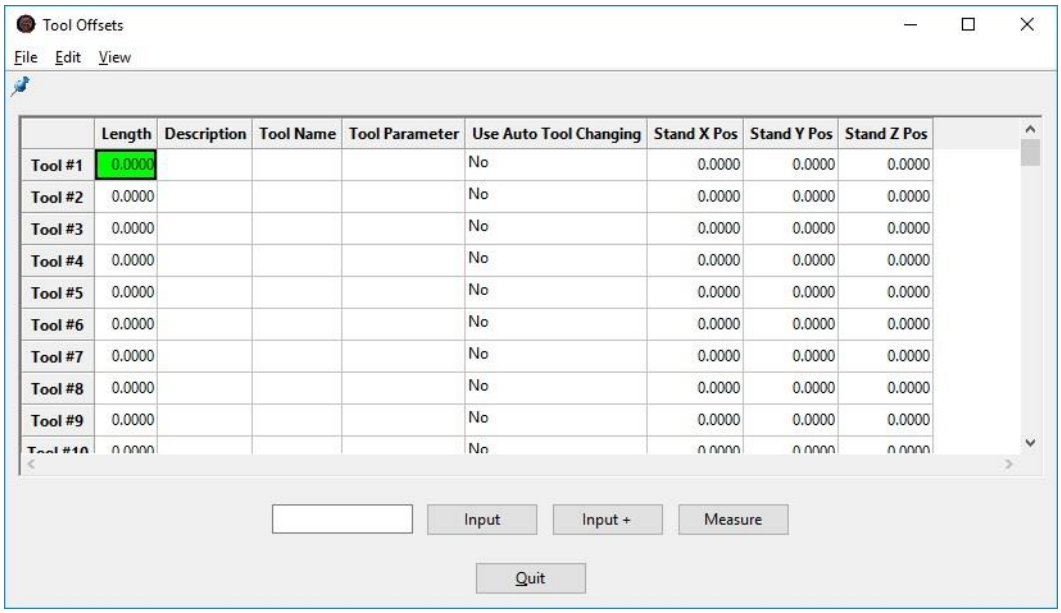
Above the *Launch Configurator* button is the version number for your software. Above that is the licensing dongle state indicator. The licensing dongle takes a long time to produce a licensing token, so initially this indicator will read "DemoMode" with a countdown of seconds left in the demo. If you have your licensing dongle installed, it should switch to saying "Licensed" after about 30-60 seconds.

At the bottom of the tab are the following buttons:

- *Launch Configurator* - Relaunch the Configurator to change software settings
- *Change Screenset Colors* - Launch the Color Picker to select colors for all of the software buttons
- *Export Configuration* - Save a copy of your software configuration. This configuration can be loaded in the configurator on this system or others to seamlessly import those settings
- *Ether-Mach Settings* - Open the Ether-Mach settings dialog
- *Support* - Launch the remote support system. This allows Stepper3 support personnel to assist with setup or trouble shooting remotely via screen sharing and/or remote control. Details on using this feature will be provided by support personnel during your support session.
- *Toggle Menu* - This button should only be used by Stepper3 support personnel. Pressing this button shows the software menu, revealing many internal options and diagnostic tools. THE S3CNC SOFTWARE WAS TESTED AGAINST THE FEATURESET PROVIDED BY THE CONFIGURATOR AND THE MENU ALLOWS PARAMETERS TO BE CONFIGURED OUTSIDE OF STABLE, TESTED RANGES. WE DO NOT RECOMMEND OR WARRANTY SHOWING THE MENU OR CHANGING ANY OF THE SETTINGS THEREIN. CHANGING SETTINGS IN THIS MENU CAN AND WILL BREAK YOUR MACHINE.

3.4 Part 4: Additional Dialogs

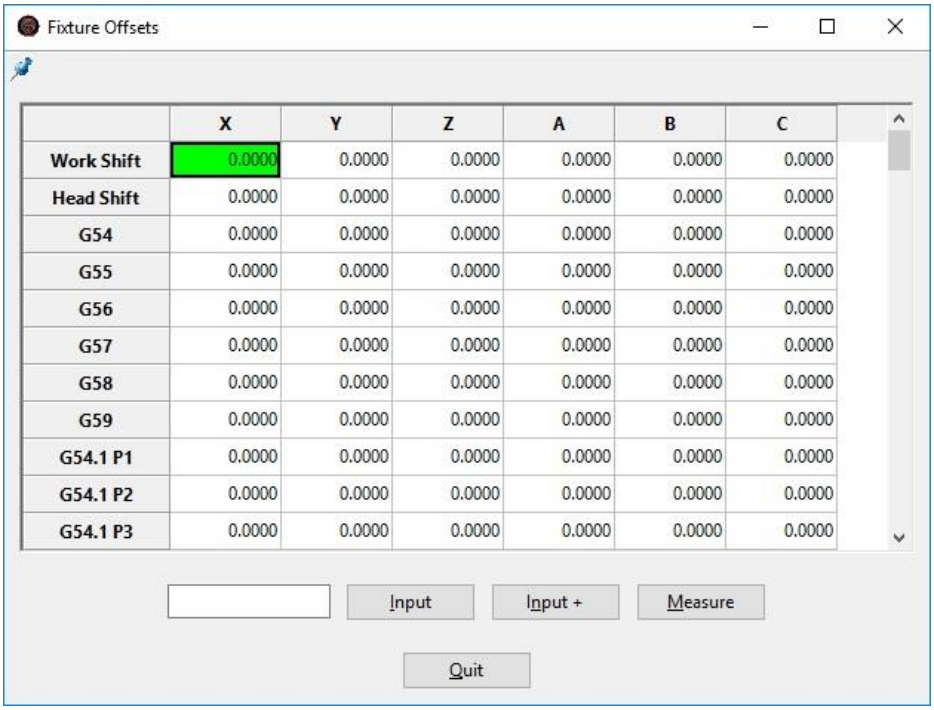
3.4.1 Tool Table



The tool table allows you to manually edit the settings for each tool. This is also where you can enable auto tool changing for a tool. There is one row in the table per tool and each tool has the following settings:

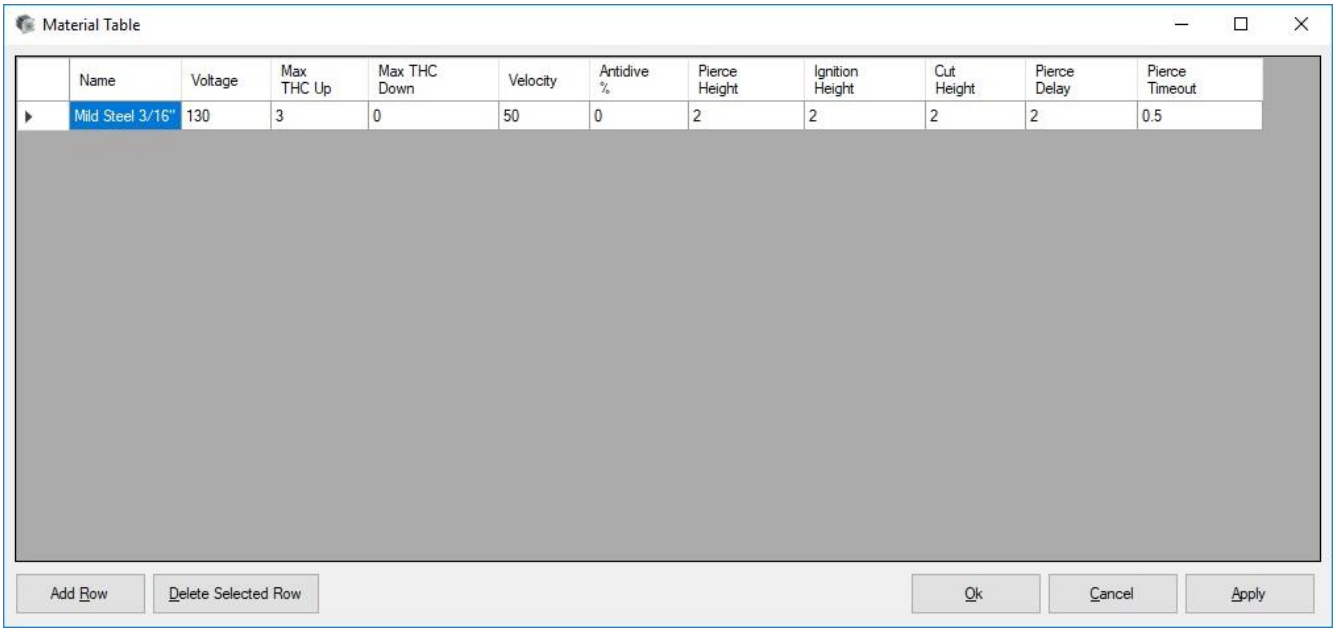
- *Length* - The length of the tool, in units
- *Description* - A description of the tool, only shown in this table
- *Tool Name* - A name for this tool. Shown on the main page when this tool is selected
- *Tool Parameter* - A text parameter for the tool. Shown on the main page when this tool is selected
- *Use Auto Tool Changing* - Set to Yes to enable auto tool changing for that tool. You must also have enabled the auto tool changing feature on the *Router Spindle Mode* page of the Configurator. Remember to learn a tool stand location for the tool before turning on auto tool changing for it!
- *Stand X Pos* - The X location of the tool stand in machine coordinates
- *Stand Y Pos* - The Y location of the tool stand in machine coordinates
- *Stand Z Pos* - The Z location of the tool stand in machine coordinates

3.4.2 Fixture Offsets Table



The fixture offsets table allows you to hand edit the each fixture offset as well as the work and head shifts. On the left, each offset is named by the G-Code used to enable it. There is a column for each axis, of which only X, Y, Z and A are used.

3.4.3 Material Table



The material table allows you to manage the THC settings for different materials cut on your plasma table.

To add a material, press Add Row in the bottom right and then edit the new row as needed. To delete a row, click one of its cells and then press Delete Selected Row in the bottom right.

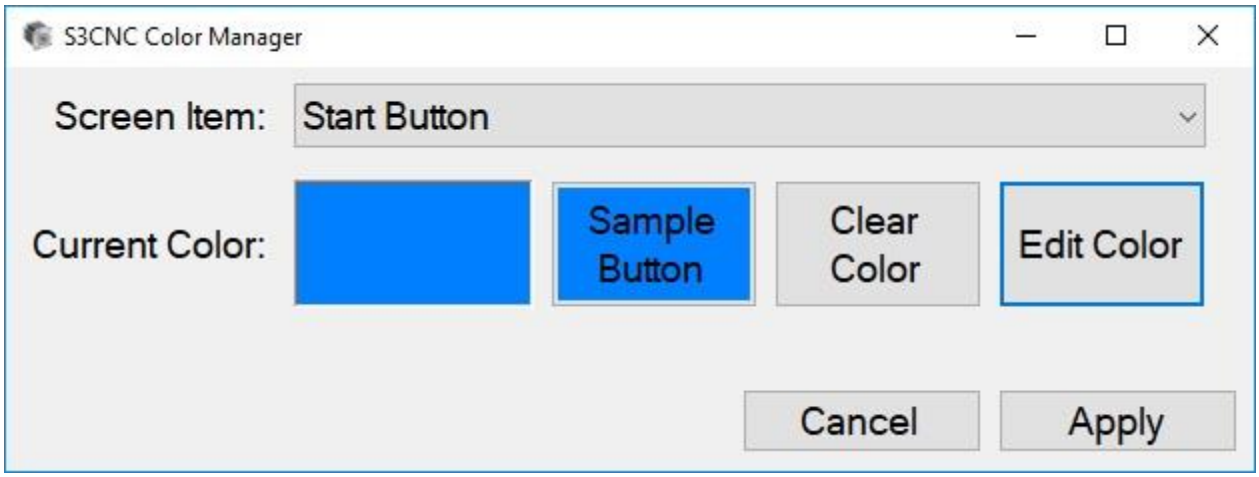
Each row has the following fields to configure:

- *Name* - The name of the last loaded material, if any
- *Voltage* - The target torch voltage from the last loaded material if any
- *Max THC Up* - The maximum number of units up from the cut height that the THC controller can move the torch
- *Max THC Down* - The maximum number of units down from the cut height that the THC controller can move the torch. This is a positive number!
- *Velocity* - The velocity of the Z axis during THC
- *Antidive Percent* - The percentage of the current feed rate below which THC stops moving the axis to prevent diving. Set this to 0 to disable antidive
- *Pierce Height* - The height at which the torch is held during pierce. See Torch Behavior section for more details
- *Ignition Height* - The height at which the torch is activated. Can be the same as pierce height
- *Cut Height* - The height the torch moves to after the pierce delay. If THC mode is active, THC starts as soon as the torch reaches this position
- *Pierce Delay*- With THC, this is the delay after activating the torch and moving to the pierce height. After the delay expires, the cut begins and the Z axis starts moving to the cut height
- Pierce Timeout - The amount of time that can pass without an Arc Ok signal before a torch

failure occurs

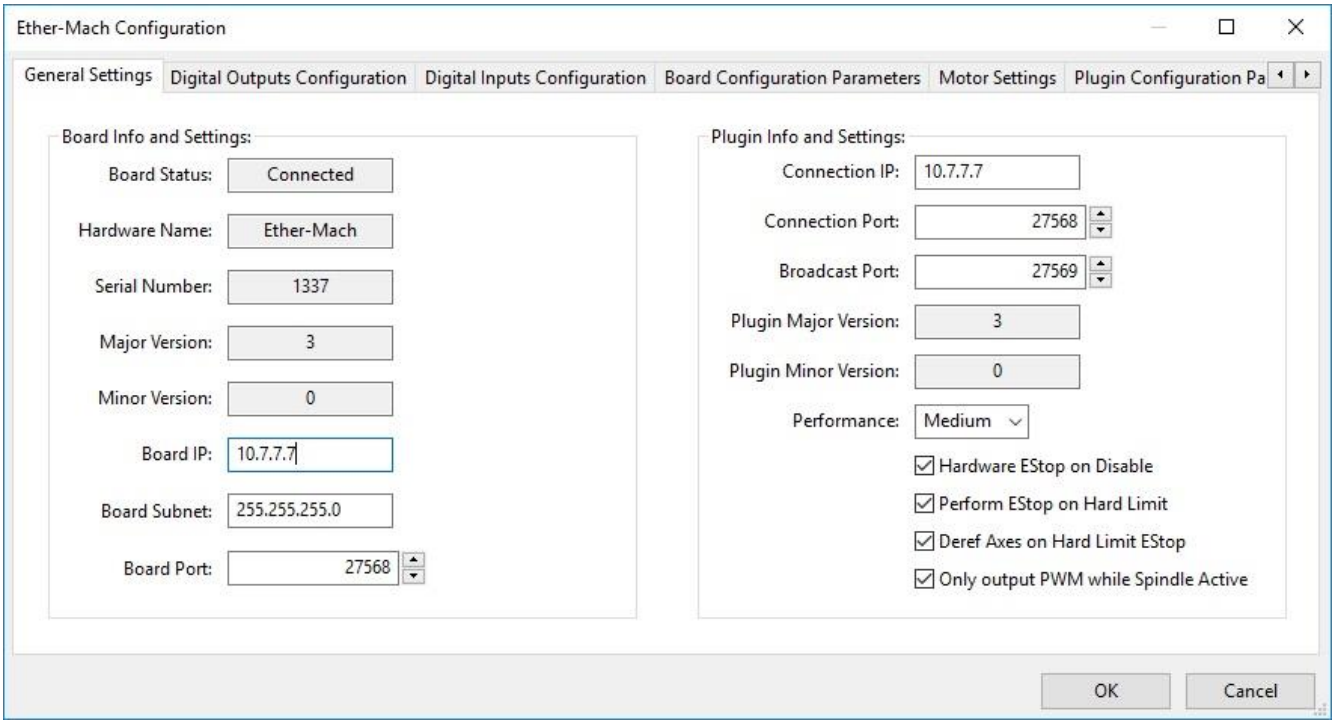
Once you have added materials to the table, they can be selected using the Select Material button on the main page. Selecting a material copies all of its data to the corresponding DROs on the settings page. This allows you to quickly switch between the settings for different materials without having to perform excessive manual data entry.

3.4.4 Color Picker



If you press the Change Screenset Colors button on the settings tab of the settings page, you will open the Color Picker. The color picker allows you to customize the colors of all of the buttons in the software as well as the colors in the toolpath display. At the top is a combo box to select the item you want to color. Once selected, the current color for the item, if any, is shown in the box to the bottom left as well as on the *Sample Button*. You can select a color with *Edit Color* or clear any selected color for that item with *Clear Color*. After setting the color for all the buttons that you wanted to, press apply to save the changes.

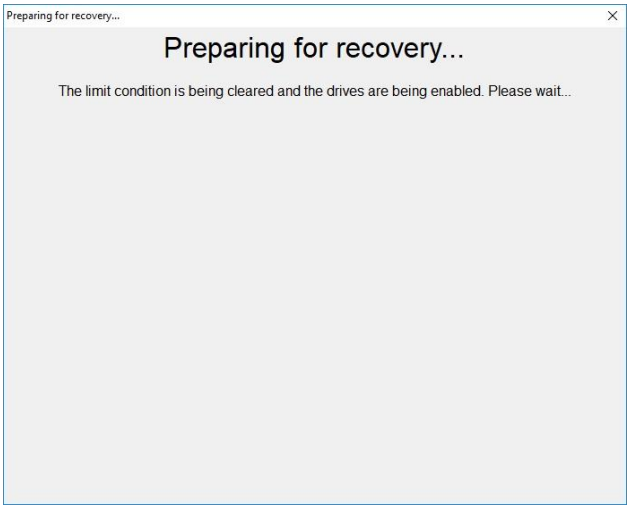
3.4.5 Ether-Mach Settings

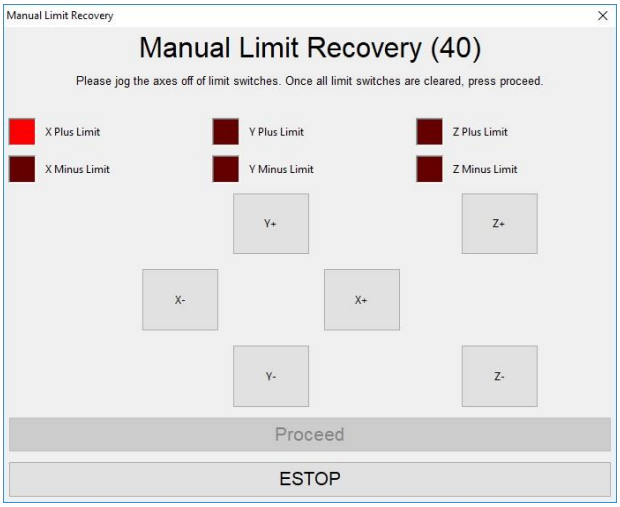


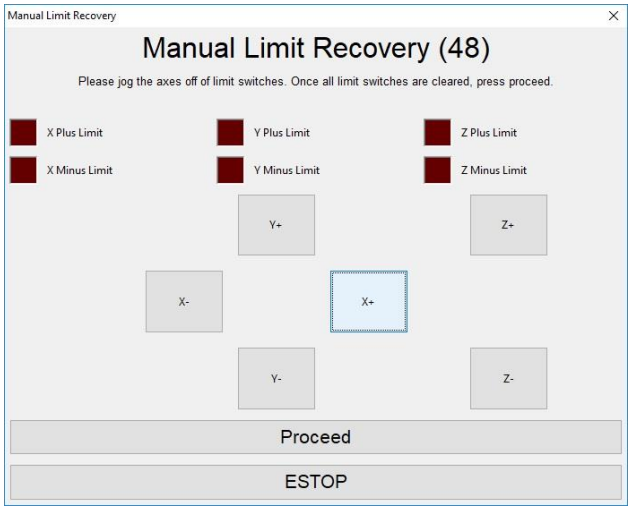
The Ether-Mach settings panel consists primarily of settings that are set up automatically by the Configurator. Still, there are some settings that you may need to access and change directly, which are all in the *General Settings* tab.

- *Performance* - Set this from Medium to Low to improve stability on old or slow computers. If you are having issues with connection loss, setting this to low is likely to resolve the issue at the cost of higher latency when jogging
- *Connection IP*, *Board IP*, *Board Subnet* - These set the IP address the software connects to at startup, the IP address the board listens on and the subnet mask the board listens to for incoming connections. Changing these allows you to switch the IP address used by the board to get around conflicts with existing network configurations. In general, the connection IP and board IP should always be the same, and the board subnet should be 255.255.255.0. For help with these settings and network configuration, contact your local system administrator and/or Stepper3 support

3.4.6 Hard Limit Recovery







If you hit a hard limit while operating the machine, the software will bring up a recovery dialog. First, it will ask you to press the *Begin Recovery* button and it will proceed to clear any drive faults that may have been caused. Once the drives are all up and running, it will switch to the manual recovery dialog. Here, there are a set of LEDs at the top showing the state of each hard limit, and a set of jog buttons. You must jog the axes until all of the hard limit inputs are cleared (i.e., all the LEDs are off) and then hit *Proceed*. After recovery, you may need to re-home your axes.

4 Section 4: Setup and Usage Guides

In this section, we present a series of guides on setting up and operating the software to complete various tasks. The first guide is a general guide on basic CNC topics and each guide that follows is for a specific machine type.

4.1 Guide 1: CNC Software Terminology

In this guide, we will cover some basic CNC topics. This is not a comprehensive tutorial on CNC but is instead design to focus on subjects that repeatedly come up in support sessions.

The CNC software is responsible for operating your CNC machine. By operating the machine, we mean that it controls motors, spindle, torches, etc. in a coordinated way in order to cut, mill, or manipulate material for manufacturing purposes.

The particular path that is cut is described by G-Code files. These files contain a simple command language called G-Code that is used to describe a path as a series of waypoints, or junctions. In addition to the path, other control information such as when to turn your spindle/torch on or off are included in the file.

The G-Code files are generated by the G-Code post of your CAD/CAM software. The G-Code post converts the drawing or design in the software into G-Code based on configurable rules and a set of scripts implemented to produce G-Code compliant with a particular CNC software.

The goal of the CNC software is to create a workflow that allows you to load and run files seamlessly throughout a manufacturing shift. In addition to loading and performing the commands from the G-Code file, this process is assisted by features such as the *Goto Position* buttons or automatic tool changing, which can be used to optimize or customize your workflow.

To begin this section, we will describe a number of disparate CNC concepts. At the end, we will tie them all together with an example of hypothetical machine operation.

4.1.1 Homing and Limits

The first challenge when having computer software move a torch or spindle (henceforth, the tool) along a trajectory is determining where exactly the tool is. On startup, the CNC software knows nothing about the position and orientation of the tool or the state of the motors. So, at startup, you can jog the axis blindly in any direction but the software cannot do anything else.

Before G-Code can be run, the software must determine its location on the table through a process called **Homing**. Homing involves moving each axis slowly in a direction toward a *home switch*. When it strikes the home switch, it backs off a small amount and then establishes that location as *machine coordinate 0*, *absolute coordinate 0* or the *home location*.

From the information you provided in the Configurator, the software knows where that home switch is as well as how far it can travel away from that switch (the range of travel). Putting this together for every axis means that, once homed, any position on your table can be described in units moved away from the home location for each axis. These units are known as **Machine Coordinates** or **Absolute Coordinates**. These coordinates allow the software to describe exact locations on the table in a reliable, repeatable way.

Since the software knows where it is, it can prevent you from jogging out of the table's range of travel or from running a G-Code file that would strike a hard limit. This feature is called software limits, and it is enabled automatically as soon as every axis has been homed. With soft limits on and set up correctly, hitting a hard limit is very difficult.

Still, as an extra layer of protection, the extents of each axis will typically have limit switches. Often the limit switch at one end also serves as the home switch and it is also not uncommon for them to all share a single input pin. Hard limit switches are placed so that they will be hit before an axis runs off its track or into a physical hard stop. When a hard limit switch actives, the software enters E-Stop and brings up the recovery dialog to help you get your axes back in the right place.

4.1.2 Coordinates and Zeroing

So, once you have finished homing, the software knows where the tool is. Afterwards, you load a G-Code file that cuts out a square. This leads to the next question: where does it cut out the square?

Let's say the G-Code file looks like this (for a router):

```
G0 X0 Y0 Z1
M3
G1 Z0
G1 X5 Y0
G1 X5 Y5
G1 X0 Y5
G1 X0 Y0
G1 Z1
M5
G0 X0 Y0
```

This file describes the process of cutting out a 5 inch square. However, if you look at this file, the coordinates are fixed - the square goes from `X = 0 to 5` and `Y = 0 to 5` and Z goes up to 1 and down to 0. This leads to a number of problems:

1) The top of our Z travel is 0, so we cannot go up to 1
2) The material is much lower, down at say, Z = -6
3) The square is always at location 0

Clearly, we cannot run this file using machine coordinates and expect this to work. To solve this,

instead of running our files in machine coordinates, we will use another, shifted coordinate system, which are called the **work coordinates**. The work coordinates are the ones shown to you on the axis DROs on the main page. They are created by applying an offset to each axis' machine coordinate. Applying this offset moves the origin position from the home location over to a new location known as the **work origin**.

For example, let's say you have a two axis machine at machine coordinate `X = 5, Y = 5`. If you apply an offset of `X = 5, Y = 4`, then the work coordinates are `X = 0, Y = 1`. Your work origin is `X = 5, Y = 4`; that means that if you move your tool 5 inches from the X home location and 4 from the Y home location, your position on the X and Y DROs will read `X = 0, Y = 0`.

So how do you set this offset? The work coordinates are formed using fixture offsets. Each fixture offset is a set of offsets for each axis that can be selected to change your work origin. Your CNC machine has a number of fixture offsets and they can be selected using G-Code. You can edit the values of these offsets in the Fixture Table. By default, fixture offset G54 is active.

So how do these offsets solve the problem with the G-Code above? Well, before running the file, you can change your current fixture offset to:

```
X = 5
Y = 5
Z = -6
```

Now, when the file moves to Z = 0 in work coordinates, it will move to machine coordinate -6, which is the top of the material, and it has plenty of clearance to move up to work coordinate 1 (machine coordinate -5). Now, instead of making the square from 0 to 5, the square is from 5 to 10 (in machine coordinates). By changing the fixture offset, we can run the same file in different places and at different material heights without changing the file.

So how do you change the offsets? There are three ways:

- By editing the fixture table directly.
- By typing new positions into the axis DROs
- By clicking the Zero buttons or the Zero All button

By far, the most common way is using the Zero All button. To set your work origin, you just jog to where you want the origin to be. You move X and Y to where you want to cut the part, bring Z down to just touch the material, and click Zero All. Now you have set that position at the top of the material as your work origin, and that is where the part will cut.

4.1.3 Step By Step CNC workflow

1) Design parts to cut using CAD/CAM software
2) Use a G-Code post to create G-Code files
3) Start S3CNC
4) Hit "*Home All*" to home every axis
5) Load the first file
6) Place your material on the table
7) Jog to the location on the material where the part should be cut
8) Jog Z down to the top of the material
9) *Zero All*
10) Hit *Start*
11) Repeat for each file

If you have a touchpad, instead of jogging Z down, you can hit Touch Off Z to probe for the touchpad and then just hit Zero for X and Y.

4.2 Guide 2: 2-Axis Plasma Table

4.2.1 Setup

When starting a new configuration for 2-Axis plasma you will want to select a *plasma machine type* and *relay only* torch mode on the first two pages of the Configurator wizard. The Configurator will guarantee that all required IO is configured. At a minimum you will need to configure two axes, a torch on output, and an arc OK input. The torch on output is used to turn the torch on and off, and the arc OK input is the feedback from torch to confirm that the arc is stable. You will also want to configure the *Pierce Delay* and *Torch Off Delay* on the software settings tab.

There are two ways the 2 axis plasma can be configured - to use only a torch on output or to use both torch down and torch on.

If your torch can be lowered and activated with just a single output relay, you would just use the torch on output. However, some torches receive separate down and on signals. The first signal (torch down), commands the torch to move down onto the material surface, and the second (torch on) fires the torch.

To use the torch down output, check the *Use Torch Down Signal* option on the Torch Settings page of the Configurator. You will have to select an output pin for the torch down signal. You will also want to set the torch down delay on the Torch Settings section of the Configurator.

You will likely want to enable the *Use CV Settings* option in the Run Control section. This will allow you to configure constant velocity movement appropriately for the shapes you are cutting. In particular, set the angle limit to prevent sharp corners from being rounded in your files. You may also wish to lower the maximum velocity through junctions to get better accuracy when they are rounded.

4.2.2 Torch Behavior

When the torch is activated, such as by an M3, the software performs the following sequence:

1) Enable the Torch Down output
2) Wait for the Torch Down delay
3) Enable the Torch On output
4) Wait for the Pierce Delay
5) Begin running the cut and start monitoring Arc OK

If there is no torch down output, steps 1 and 2 are skipped.

If Arc OK shuts off and remains off for a duration longer than the Pierce Timeout, the software will stop the running file and bring up a Torch Recovery dialog. You can recover with torch off (Trial Cut), recover with torch on, or cancel to stop and give up on the run.

When the torch turns off, it performs the following sequence:

1) Disable the torch on output
2) Wait for the Torch Off Delay
3) Disable the torch down output
4) Wait for the Torch Down Delay

If there is no torch down output, steps 3 and 4 are skipped.

4.2.3 Operation

Once set up as two axis plasma, the following will be the sequence of daily operations:

4.2.3.1 On Startup

1) Start the S3CNC software
2) (Optional) Jog your axes close to their home locations
3) Press *Home All*

4.2.3.2 Per file

1) (Optional) Move to the starting location of the cut and press *Zero All*
2) Press *File* and select the file to run
3) Press *Start*
4) If a torch failure occurs, select to recover with torch on, recover with torch off, or cancel, as needed
5) Repeat for each file

4.3 Guide 3: 3-Axis/THC Plasma Table

4.3.1 Setup

4.3.1.1 Configurator

When starting a new configuration for 3-Axis/THC Plasma, you will want to select a *plasma machine type* and *Torch Height Control (3 Axis* torch mode on the first two pages of the Configurator wizard. The Configurator will guarantee that all required IO is configured. At a minimum you will need to configure three axes, a torch on output, an arc OK input, and the three THC inputs (Up, Down, On). The setup page can be found in the previous section: Torch Height Control Settings.

The IO are as follows:

- *Arc OK Input* - This is a feedback from the torch indicating that the arc is stable
- *Torch On Output* - This output is used to turn the torch on and off
- *THC Up Input* - This is an input from the THC controller (ex, the Proma 150). It indicates to the software that the Z axis must move up in order to correct the arc voltage
- *THC Down Input* - This is an input from the THC controller (ex, the Proma 150). It indicates to the software that the Z axis must move down in order to correct the arc voltage
- *THC On* - This indicates to the software that the arc is active and stable and that THC should be active. When running in THC mode

If your torch can probe to find the material surface, enable the *Uses Probing To Touch Off Option* and provide a probe input pin (and probe feed rate).

You will likely want to enable the *Use CV Settings* option in the Run Control section. This will allow you to configure constant velocity movement appropriately for the shapes you are cutting. In particular, set the angle limit to prevent sharp corners from being rounded in your files. You may also wish to lower the maximum velocity through junctions to get better accuracy when they are rounded.

4.3.1.2 Software Settings

The first thing you will want to do is open the materials table and fill in the details for each of the materials you will be cutting. Go to the diagnostics tab on the settings page and click material table. See the Material Table section for details on each material parameter.

You will also want to set the torch off delay and pierce timeout on the settings tab.

If you are using probing, you will want to make sure the THC probe mode on the main page is set to either *Once* or *Always*. *Always* is recommended, since it will account for fluctuations in material height by re-probing for each cut. However, you can cut more efficiently on genuinely flat material by setting to once, in which case it will only probe if there is no reference for Z. If you do not have probing, the *Never* probe mode must be used, which requires you to manually zero Z at the material surface.

4.3.2 Operation

Once set up as THC plasma, the following will be the sequence of daily operations:

4.3.3 On Startup

1) Start the S3CNC software
2) (Optional) Jog your axes close to their home locations
3) Press *Home All*

4.3.4 Per File

1) (Optional) Press *select material* and select the material used
2) (Optional) Tune the THC controller (ex, Proma 150) to match the voltage settings for the material
3) Move the starting location for the cut and press *Zero* for the X and Y axes
4) If you do not have probing enabled, jog Z down to the material surface and click the *Z Zero* button. The *THC Ref LED* should now be lit
5) Press *File* and select the file to run
6) Press *Start*
7) If a torch failure occurs, select to recover with torch on, recover with torch off, or cancel, as needed
8) Repeat for each file

4.4 Guide 4: 2-Axis Router Table

4.4.1 Setup

When starting a new configuration for 2-Axis router you will want to select a *router machine type*. On the Router Spindle Mode page, answer *No* to the question *Do you have a Z axis?* You can choose any of the four spindle speed control methods: relay-only, analog, digital, or stepper motor. The Configurator will guarantee that all required IO is configured. At a minimum, you will need to configure two axes and a spindle output. Depending on your spindle speed control selection, there may be additional required settings.

You will want to configure the spindle delay to be greater than the amount of time it takes for your spindle to start up. If you are using something with negligible start-up time, you can set it to 0.

If you are using analog spindle speed control, you will need to set a spindle speed PWM output and a PWM period. If you are using digital control, you will need to set the up to three digital speed control outputs. If you are using stepper motor speed control, you will need to set up a step and direction pin as well as a motion profile (steps per rev, velocity, acceleration) for the motor. See Router Step 4: Spindle Settings for more detail on spindle speed control.

You will likely want to enable the *Use CV Settings* option in the Run Control section. This will allow you to configure constant velocity movement appropriately for the shapes you are cutting. In particular,

set the angle limit to prevent sharp corners from being rounded in your files. You may also wish to lower the maximum velocity through junctions to get better accuracy when they are rounded.

Note that auto tool changing is not supported on two axis router systems.

4.4.2 Spindle Behavior

When the spindle is activated, such as by an M3, the software performs the following sequence:

1) Enable the spindle output
2) Wait for the spindle delay
3) If available, check the spindle at speed signal. If not active, stop and report an error
4) Begin running the cut

When the spindle is deactivated, such as by an M5, the software performs the following sequence:

1) Disable the spindle output
2) Wait for the spindle delay
3) If available, check the spindle stopped signal, If not active, stop and report an error
4) Begin running the cut

4.4.3 Operation

Once set up for two axis router, the following will be the sequence of operations:

4.4.3.1 On Startup

1) Start the S3CNC software
2) (Optional) Jog your axes close to their home locations
3) Press *Home All*

4.4.3.2 Per File

1) (Optional) Move to the starting location of the file and press *Zero All*
2) Press F*ile* and select the file to run
3) Press *Start*
4) Repeat for each file

If a problem occurs, you can press the Stop button mid run. The Stop button will cause the software to:

1) Stop the current file using a feed hold (slows to a stop)
2) Turn off the spindle

The current line number in the G-Code window will not change - it will stay at where you stopped. So, the start button will read "Run From Here" and can be pressed to resume the file at that line. If you

need to start earlier in order to properly recover the cut, just select the earlier G-Code line manually and press "Run From Here".

An M6 line will initiate a manual tool change. During a manual tool change, depending on your settings, the spindle will either remain in place or move to the park position. Once moved, it will bring up a dialog with the number of the current tool and the next tool. While this dialog is up, you should switch the tools. Once the tool has been changed, press *Proceed* and the file will continue. If, for some reason, you cannot do the tool change or you do not want to continue the file, hit *Cancel* and the file run will stop.

4.5 Guide 5: 3-Axis Router Table

4.5.1 Setup

When starting a new configuration for 3-Axis router, you will want to select a *router machine type*. On the Router Spindle Mode page, answer *Yes* to the question *Do you have a Z axis?* You can choose any of the four spindle speed control methods: relay only, analog, digital, or stepper motor. The Configurator will guarantee that all required IO is configured. At a minimum, you will need to configure three axes and a spindle output. Depending on your spindle speed control selection, and whether or not you enable auto tool changing, there may be additional required settings.

You will want to configure the spindle delay to be greater than the amount of time it takes for your spindle to start up. If you are using something with negligible start-up time, you can set it to 0.

If you are using analog spindle speed control, you will need to set a spindle speed PWM output and a PWM period. If you are using digital control, you will need to set the up to three digital speed control outputs. If you are using stepper motor speed control, you will need to set up a step and direction pin as well as a motion profile (steps per rev, velocity, acceleration) for the motor.

If you have a probe or a touchpad used for zeroing the Z axis, enable the *Uses Probe/Touchpad to touch off* option on the Spindle Settings page. You will need to provide a probe input and probe feed rate. If you are using a touchpad, enable the *Probe input is from a Touchpad* option and provide the thickness of the touchpad.

You will likely want to enable the *Use CV Settings* option in the Run Control section. This will allow you to configure constant velocity movement appropriately for the shapes you are cutting. In particular, set the angle limit to prevent sharp corners from being rounded in your files. You may also wish to lower the maximum velocity through junctions to get better accuracy when they are rounded.

If you answer *Yes* to *Do you use an automatic tool changer?*, you will be required to provide a chuck open output, which is needed to open and close the chuck during auto tool changing. Other auto tool changing IO are optional, but highly recommended in order to prevent damage to your table, spindle, or tools.

More details on auto tool changing and its setup will be provided in the next section Guide 6: Auto Tool Changing for 3 Axis Router Table.

4.5.2 Spindle Behavior

When the spindle is activated, such as by an M3, the software performs the following sequence:

1) Enable the spindle output
2) Wait for the spindle delay
3) If available, check the spindle at speed signal. If not active, stop and report an error
4) Begin running the cut

When the spindle is deactivated, such as by an M5, the software performs the following sequence:

1) Disable the spindle output
2) Wait for the spindle delay
3) If available, check the spindle stopped signal, If not active, stop and report an error
4) Begin running the cut

4.5.3 Operation

Once set up for three axis router, the following will be the sequence of operations:

4.5.3.1 On Startup

1) Start the S3CNC software
2) (Optional) Jog your axes close to their home locations
3) Press *Home All*

4.5.3.2 Per File

1) (Optional) Move to the starting location of the file and press *Zero All*
2) If you have a probe/touchpad, press *Touch Off* Z to zero the Z axis. Otherwise, manually jog Z down to the material and zero it
3) Press File and select the file to run
4) Press Start
5) Repeat for each file

If a problem occurs, you can press the Pause button mid run. The pause button will cause the software to:

1) Stop the current file using a feed hold (slows to a stop)
2) Lifts the spindle to the max Z height
3) Turns off the spindle
4) Moves to the park position

The current line number in the G-Code window will not change - it will stay at where you paused; so, the start button will read "Run From Here" and can be pressed to resume the file at that line. If you need to start earlier in order to properly recover the cut, just select the earlier G-Code line manually and press "Run From Here".

An M6 line will initiate a tool change.

During a manual tool change, depending on your settings, the spindle will either remain in place or move to the park position. Once moved, it will bring up a dialog with the number of the current tool and the next tool. While this dialog is up, you should switch the tools. Once the tool has been changed, press *Proceed* and the file will continue. If, for some reason, you cannot do the tool change or you do not want to continue the file, hit *Cancel* and the file run will stop.

During an automatic tool change, the software will automatically drop off or pick up a tool. You can mix automatic and manually changed tools. If you mix them, then the software will either ask you to manually remove the existing tool, after which it will pick up the new one, or it will drop off the old one and then ask you to manually add the new tool.

4.6 Guide 6: Auto Tool Changing for 3 Axis Router Table

4.6.1 Setup

4.6.1.1 Configuration

To enable the auto tool changing feature, you will need to answer *Yes* to the question *Do you use an automatic toolchanger?* on the Router Spindle Mode page of the Configurator. If you do, the Tool Changing page will have many more settings, and the chuck open output will be required.
We will now cover all of the settings in the Configurator for auto tool changing:

Chuck Open Output
This is the required output used to control (open/close) the chuck.

Are your tools line up along the X axis or Y axis?
Your tool rack keeps all of the tools in a line. You should select the axis that the rack is parallel to (i.e., lined up with). You can see diagrams in Auto Tool Changing Settings.

How far must you lift a tool holder to unlock it from the stand? The tool holders in each stand are locked down. In order to unlock them so they can be moved horizontally out of the stand, they must be lifted slightly. This value is the amount to lift. Similarly, when you put a tool back, it must be pressed down this amount before releasing the tool to ensure it is securely placed into the stand.

How far horizontally must the tool holder be moved before it is completely out of the stand? After a tool has been grabbed and lifted, it must be moved horizontally out of the stand. This number is how far horizontally it moves, and must be a large enough number to clear the stand. The

tool is always move inward towards the table. You can see diagrams in Auto Tool Changing Settings.

Uses Chuck Fully Open Input (Optional but highly recommended) If enabled, you must select an input pin for the chuck fully open signal. This is used to detect that the chuck has been opened during auto tool changing. Without this signal, after opening the chuck, the software will assume success. This signal provides extra safety to avoid machine malfunction in the event of a jam.

Uses Chuck Fully Closed Input (Optional but highly recommended) If enabled, you must select an input pin for the chuck fully closed signal. This is used to detect that the chuck is closed securely around a tool. Without this signal, after grabbing a tool and closing the chuck, the software will assume success. This signal provides extra safety to avoid machine malfunction in the event of a jam.

4.6.1.2 Learning Tool Locations

Before you can use auto tool changing with a tool, you must learn the stand location for it. To learn a stand location for a tool, perform the following steps:

1) Put the tool you want to learn into the spindle
2) Manually select the correct tool number for the tool using the tool number DRO on the tool panel of the main page
3) Jog your axes until you are close to the tool's stand
4) Lower your jog speed to < 10%
5) Carefully move the tool into the stand and down into its locked position. You need to be very accurate with this, so do not hesitate to lower the jog speed more (or use a small incremental jog) and make finer and finer adjustments
6) Once you have the tool in the proper position in its stand, press the Learn Stand Location button on the tool panel of the main page (see Tools (Only in router mode) for more detail)

4.6.1.3 Enabling auto tool changing for the tool

Now that the tool's stand location is known, you can safely enable auto tool changing for the tool. By default, tools do not use auto tool changing because it cannot know whether your stand locations are valid. Once you have added a valid stand location, you can go to the tool table and manually turn on auto tool changing. To do so:

1) Go to the settings page, and then the diagnostics tab
2) Open the tool table and find the row for the tool for which you want to enable auto tool changing (see Tool Table for more detail)
3) Change the Use auto tool changing option from No to Yes
4) Close the tool table

4.6.2 Auto Tool Changing Behavior

For the duration of the tool change, the spindle and coolant will be disabled. If they were on when the tool change began, the software will turn them back on after the tool change completes.

When the software uses auto tool changing to drop off a tool, it performs the following sequence:

1) Moves X and Y to the side of the tool stand and Z to max height
2) Drops Z down to the height needed to slide in the tool (defined by stand location plus life amount)
3) Moves into the tool stand (moves to stand X and Y location)
4) Moves tool down to lock it in
5) Open the chuck
6) If available, checks the chuck fully open input and fails if not active
7) Lifts the spindle up to max Z height
8) If available, checks the chuck fully open input again
9) Closes the chuck

When the software uses auto tool changing to pick up a tool, it performs the following sequence:

1) Moves to max Z and moves X and Y above the tool
2) Opens the chuck
3) If available, checks the chuck fully open input and fails if not active
4) Moves Z down, bringing the chuck on to the tool
5) Closes the chuck
6) If available, checks the chuck fully closed input and fails if not active
7) Lift the tool slightly to unlock it from the stand
8) Move the tool sideways out of the stand
9) If available, checks the chuck fully closed input again
10) Lift Z to max height
11) If available, checks the chuck fully closed input again

5 Section 5: Post Guidelines and Examples

The S3CNC software is permissive in what G-Code it will accept, so many different posts will work correctly with S3CNC. The biggest cause of problems with G-code posts are when they are written to perform many tasks themselves. The S3CNC software handles most of the machine complexity and expects very simple G-Code. If it is given G-Code that uses complex features (i.e., changing offsets, performing probes, etc.), the two may conflict with one another.

So, here are collection of guidelines on post output. Following these guidelines are example files for each machine type.

5.1 Post Guidelines

5.1.1 G and M codes

With the exception of the mode line at the beginning of the file, G and M codes should be one per line.

5.1.2 M30 and blank line at end

The file should end with an M30 followed by a blank line. The blank line is necessary because a valid G-Code line must end with a newline. Without the blank line, the line with the M30 will not have a newline at the end and will not be a full line. This will cause it to be truncated.

5.1.3 Mode lines

Start each G-Code file with a mode line. This mode line should set the units, positioning mode, etc. An example mode line:

```
G00 G20 G17 G90 G90.1 G40 G80
```

Each G-Code is as follows:

- G00 - Switch to rapid mode
- G20 - Switch to inches
- G17 - Switch to XY Plane for circular moves
- G90 - Use non-relative coordinates (use work coordinates)
- G90.1 - Use absolute IJK coordinates
- G40 - Disable cutter compensation
- G80 - Disable canned cycles

5.1.4 Velocity and Spindle Speed

Following the mode line, you should set an initial velocity and spindle speed. You can change these later, but an initial value should be set to avoid mixing values from previous files. Example:

```
F50 S2000
```

5.1.5 M3/M5 pairs

In general, M3 and M5 should come in pairs. Each M3 should have a corresponding M5. Example:

```
G0 X0 Y0 Z1
M3
G1 Z0
G1 X1 Y1
G1 Z1
M5
G0 X5 Y5 Z1
M3
G1 Z0
G1 X6 Y6
G1 Z1
M5
```

5.1.6 Tool Length Offsets

If you answered *No* to the question *Does you G-Code post apply the tool offsets using G43/G49?*, you should not use G43, G44 or G49 in the post. The software will automatically apply the required tool offsets in the M6. If you answer *Yes* to the question, your post will be responsible for enabling the tool length offset using G43/G44 and cancelling is later with G49. You should add a G49 to your mode line.

5.1.7 Tool changes

The tool number parameter (TX) should be one the same line as the M6, ex:

```
M6 T2
```

5.1.8 Position Words

Position words are optional. So, for example, this G-Code…

```
G0 X1 Z1
G0 X2
G0 Z2
```

…produces the same motion as…

```
G0 X1 Z1
G0 X2 Z1
G0 X2 Z2
```

In our examples, unchanged axes positions are omitted, but in most posts the positions for all axes are repeated on every line for simplicity in the G-Code generator.

5.2 Example Files

5.2.1 Three Axis Router

```
G00 G20 G17 G90 G90.1 G40 G80
F75 S2000
M6 T3
G00 Z0.5
G00 X5 Y5
F30 S12000
M3
G01 Z-0.1
G01 X7.5 Y5
G01 X7.5 Y7.5
G01 X5 Y7.5
G01 X5 Y5
G01 Z0.5
M5
M6 T1
G00 Z0.5
G00 X5 Y5
F50
S18000
M3
G01 Z-0.2
G02 X10 I7.5 J5
G02 X5 I7.5 J5
M5
G00 Z0.5
M30
```

5.2.2 Two Axis Router

```
G00 G20 G17 G90 G90.1 G40 G80
F75 S2000
M6 T3
G00 X5 Y5
F30 S12000
M3
G01 X7.5 Y5
G01 X7.5 Y7.5
G01 X5 Y7.5
G01 X5 Y5
M5
M6 T1
G00 X5 Y5
F50
S18000
M3
G02 X10 I7.5 J5
G02 X5 I7.5 J5
M5
M30
```

5.2.3 Plasma (Two Axis or THC)

```
G00 G20 G17 G90 G90.1 G40 G80
F75
G00 X5 Y5
F30
M3
G01 X7.5 Y5
G01 X7.5 Y7.5
G01 X5 Y7.5
G01 X5 Y5
M5
G00 X5 Y5
F50
M3
G02 X10 I7.5 J5
G02 X5 I7.5 J5
M5
M30
```