Mach4 Users Guide

v2023Q3.1

Mach4 Users Guide

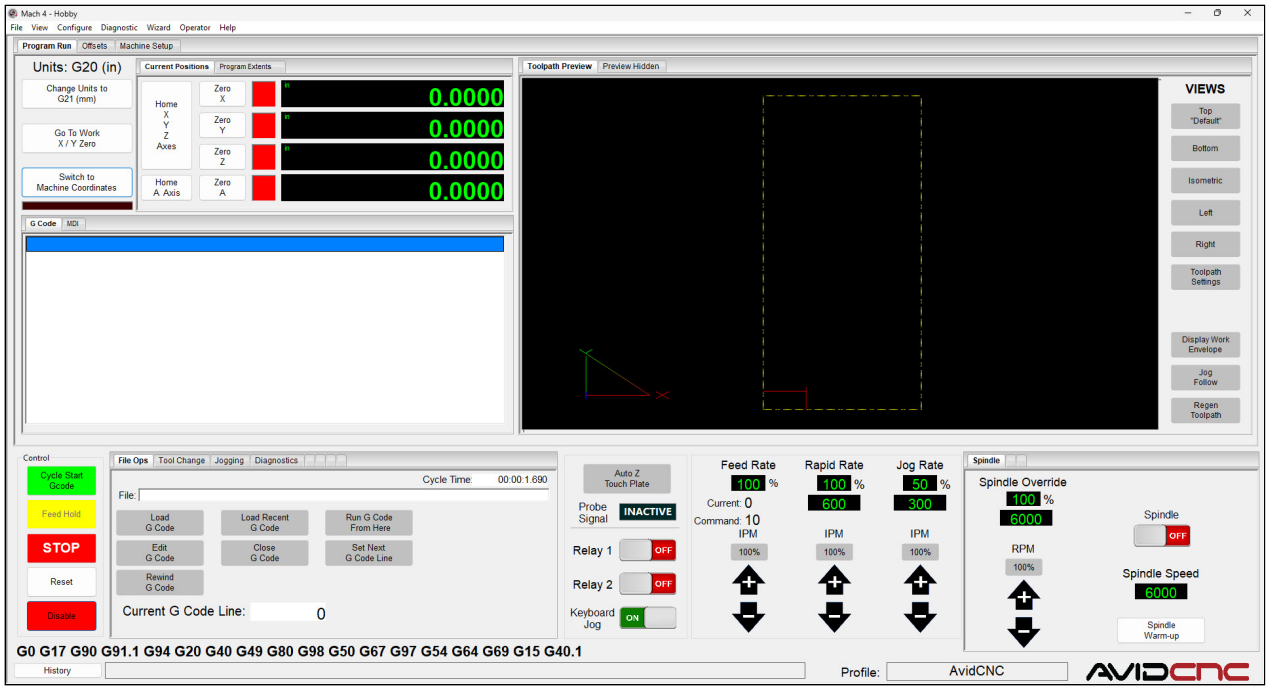
This guide will provide information about the screen elements in Avid CNC's Mach4 screen. Information on G-code programming can be found on the **PDF Downloads** page.
(https://www.avidcnc.com/support/instructions/software/downloads/documentation)

Common Features

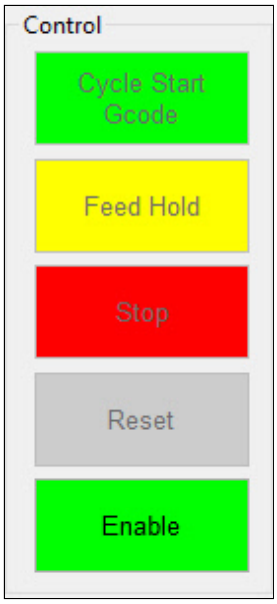
A majority of the screen shown below will be the same whether you are using a router, spindle, or plasma torch. Where there are differences, they will be noted in this guide.

For plasma users, there are additional Mach4 Plasma and SheetCam user guides.

For laser users, see the Laser Users Guide.



Control Group

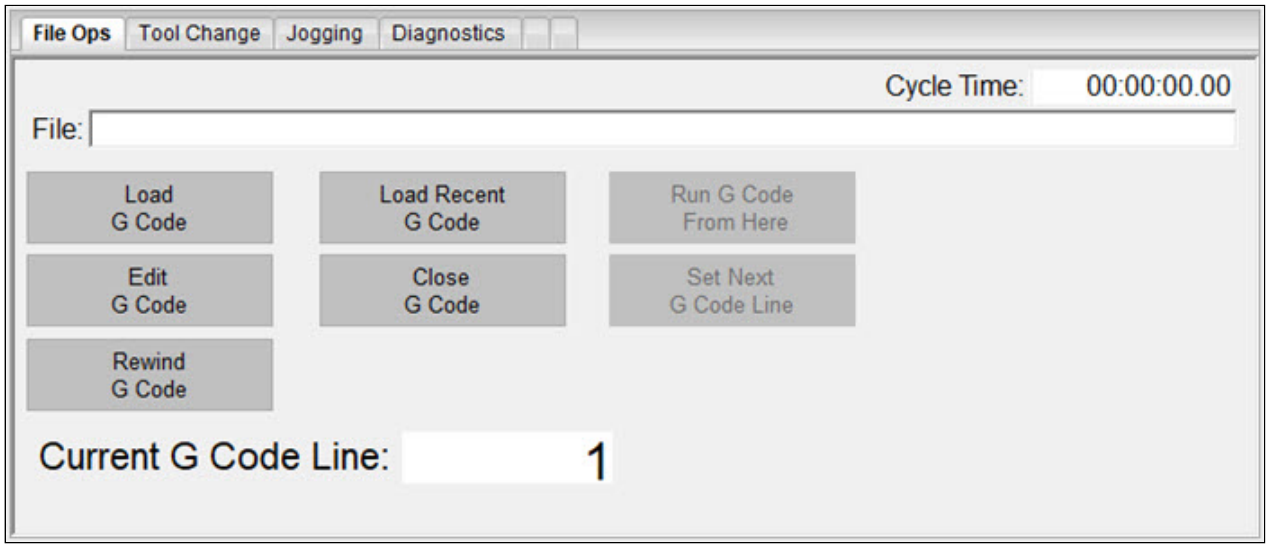


- **Cycle Start G-Code:** This button will start program execution from the current line shown in the File Ops tab.
- **Feed Hold:** This button will immediately pause the program, pausing motion of all axes. Motion will come to a stop on the programmed path at the set maximum deceleration of the motors. It will not lose motor position or require the machine to be re-homed. Feed Hold will not stop the router or spindle. When executing a feed hold in plasma mode, the torch will be turned off.
- **Stop:** This button will immediately stop the G-Code program, including the router or spindle. This is the same function as the physical Emergency Stop button. All motion will stop immediately.

> **Operation Note**
>
> Any axis in motion during a **Stop** or **Emergency Stop** will lose its position. You should re-home the machine before continuing; failure to do so will result in incorrect cuts.

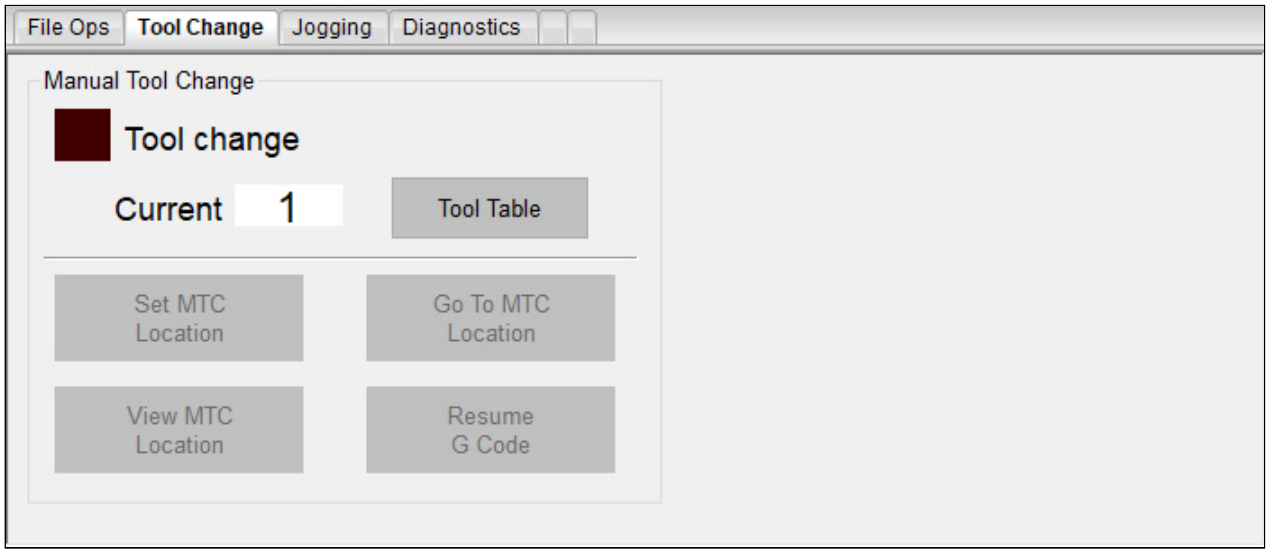
- **Reset:** This button will rewind the G-Code program back to the beginning and put Mach4 back to its default state.
- **Enable:** This button will enable (if pressed when the button is flashing green) or disable (if pressed when the button is solid red) the machine. The machine must be enabled before any operations can take place.

File Ops



- **Load G Code:** This button opens a window to allow you to navigate to your desired G-Code file and load it into Mach4.
- **Edit G Code:** This button opens the currently loaded G-Code file in a G-Code editor called GC Edit. After making any changes, save the file and close the window. Mach4 will then reload the updated G-Code file.
- **Rewind G Code:** This button rewinds the currently loaded G-Code file back to the beginning of the program.
- **Load Recent G Code:** This button opens a window displaying recently loaded G-Code files.
- **Close G Code:** This button closes the currently loaded G-Code program.
- **Run G Code From Here** and **Set Next G Code Line:** These buttons allow you to start a program somewhere other than the beginning of the program. Detailed use of these buttons are described in section 4.6.5 (page 42) of Newfangled Solution's **Mach4 CNC Controller Mill Operations Guide**(https://www.machsupport.com/wp-content/uploads/2014/05/Mach4%20Operation%20Manual.pdf)

Tool Change

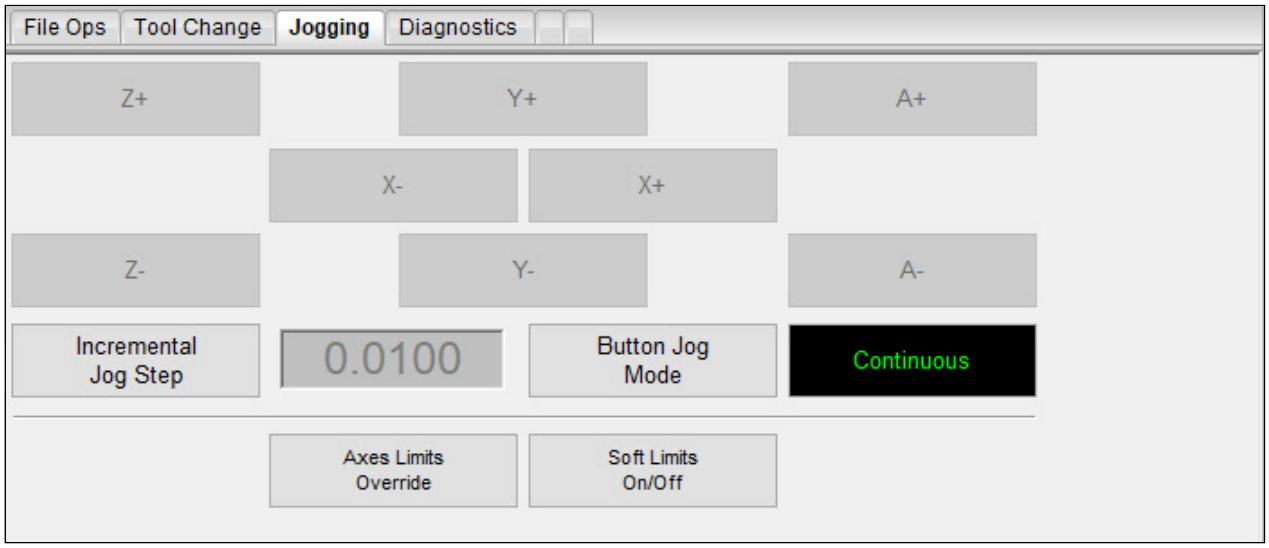


- **Tool Change LED:** Indicates when a tool change is required.
- **Current Tool:** Your current tool number.
- **Tool Table:** This button opens a new window where you can view and edit tool information. You will not need to use this unless specifying tool lengths and offsets.
- **Set MTC Location:** This button saves the current position of your machine as a "Manual Tool Change" (MTC) location. This position is saved in ***Machine Coordinates***.
- **View MTC Location:** This button displays a message box with your currently saved MTC location.
- **Go To MTC Location:** This button will move the machine back to the saved MTC location. It will first move the Z axis up to provide maximum clearance, then to the saved X and Y positions, and finally back to the saved Z position.
- **Resume G-Code:** This button is only enabled after an M06 M-Code has been called in your G-Code. It is used to resume a program after a tool change.

Operation Note

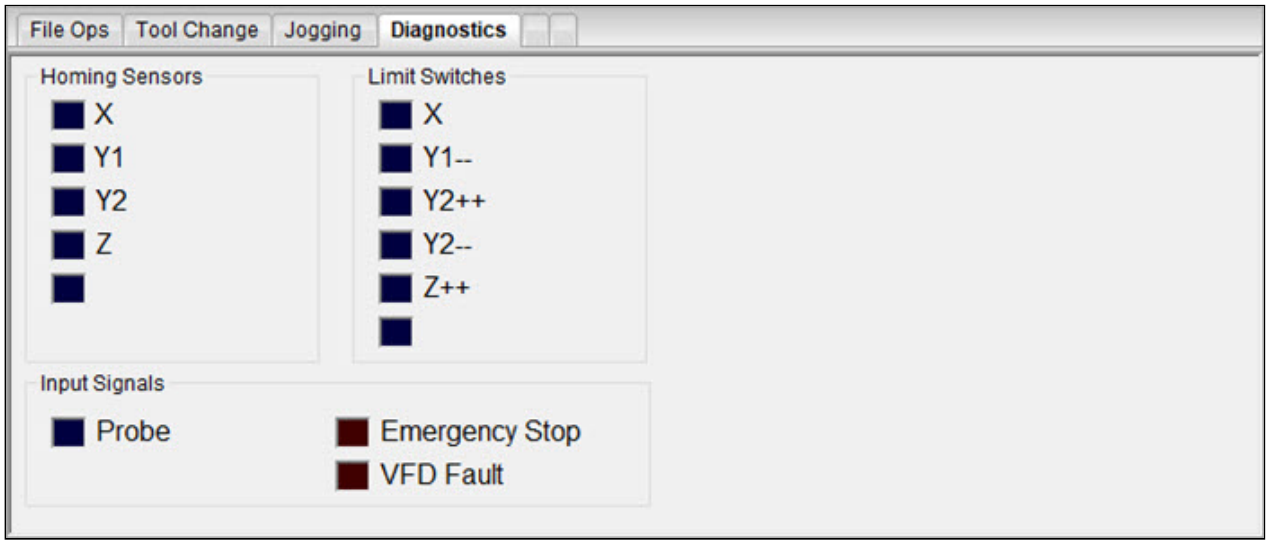
Additional instructions for using the M06 M-Code with tool changes can be found in the Manual Tool Changes section.

Jogging



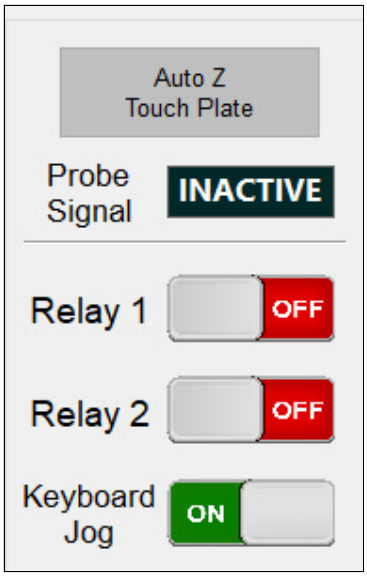
- Jogging is enabled only when the machine is enabled and a program is not currently running.
- **Axis "+" and "-" buttons:** These buttons will move the given axis in the positive or negative direction.
- **Button Jog Mode:** This will switch between incremental and continuous jog mode (displayed in the box to the right of the button). When in continuous mode, pressing and holding down one of the axis jog buttons will continuously jog the machine until the button is released. In incremental mode, pressing one of the axis jog buttons will only move the axis the specified increment (visible in the box to the left of "Button Jog Mode").
- **Incremental Jog Step:** This button changes the increment amount described above.
- **Axes Limits Override:** Under normal operation, when activating a limit switch, Mach4 will prevent the machine from moving towards the triggered limit switch. Clicking this button will override that functionality and allow movement in any direction. When "Axis Limits Override" is enabled, the button will be yellow. Use caution with this button to prevent possible damage to limit switches.
- **Soft Limits On/Off:** This button will enable or disable soft limits. When enabled, the button will be green. Soft limits (or "software limits") limit the travel of all axes based on the configured size of your machine. In the toolpath preview window, soft limits are shown as dashed yellow lines. When the machine is jogged into a soft limit, the machine is decelerated to a stop. If you load a G-Code program that has commanded moves outside of the soft limits, you will see a notification at the bottom of the screen.

Diagnostics



- These diagnostic LEDs show the state of sensors and other inputs. If the sensor or input is activated, the LED will illuminate.

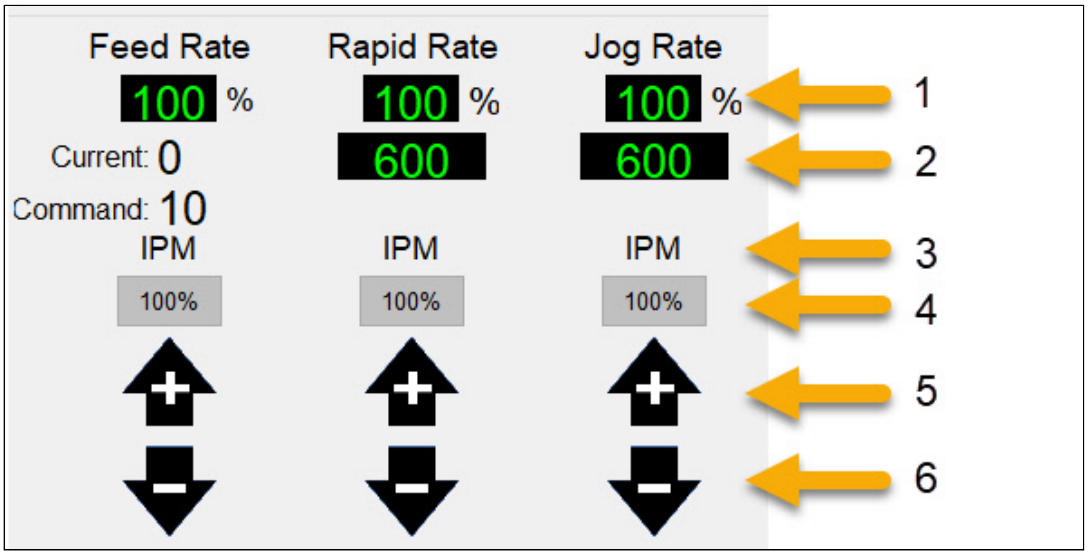
Probe and Relays



- **Auto Z Touch Plate:** This button opens the user interface for the Auto Z Touch Plate. This button is not visible when using the plasma cutting method.
- **Probe Signal:** This will display if the probe signal from the touch plate is "Active" or "Inactive". When using the plasma cutting method, this will display the probe state of the Ohmic probe.
- **Relay #1:** Toggle button for Relay 1.
  - When using a spindle or plasma cutting method, Relay 1 is mapped to Mach4's Flood Coolant output signal. You can use an M08 M-Code in your G-Code to to turn Relay 1 on. M09 will turn off both Relay 1 and 2.
  - When using a router, you will have your router connected to Relay 1 on the plug and play control box. Use the Router Control toggle button to turn the router on and off.
- **Relay #2:** Toggle button for Relay 2.
  - When using a spindle or plasma cutting method, Relay 2 is mapped to Mach4's Mist Coolant output signal. You can use an M07 M-Code in your G-Code to turn Relay 2 on. M09 will turn off both Relay 1 and 2.
  - When using a router, Relay 2 is mapped to Mach4's Flood Coolant output signal. You can use an M08 M-Code in your G-Code to to turn Relay 1 on. M09 will turn off both Relay 2.
- **Keyboard Jog:** This will enable or disable keyboard jogging. The table below shows which key will move each axis. Holding down the **SHIFT** key in addition to the respective axis key will rapid jog. Holding down the **CTRL** key in addition to the respective axis key will incremental jog.

Key	Axis	Direction
Right Arrow	X	Positive
Left Arrow	X	Negative
Up Arrow	Y	Positive
Down Arrow	Y	Negative
Page Up	Z	Positive
Page Down	Z	Negative
"["	4th Axis	Positive
"]"	4th Axis	Negative

Overrides



1. Current percentage. Jog Rate and Rapid Rate are limited to 100%, which is the maximum velocity based on motor tuning. Feed Rate is the percentage of the commanded feed rate.

2. Current speed based on the percentage. Feed Rate additionally displays the last commanded feed rate from the G-Code program.

3. Units for the displayed current speed. Either IPM (in/min) or MMPM (mm/min).

4. 100% button will change the current percentage back to 100.

5. Up arrow will increase the percentage by 25%.

6. Down arrow will decrease the percentage by 25%.

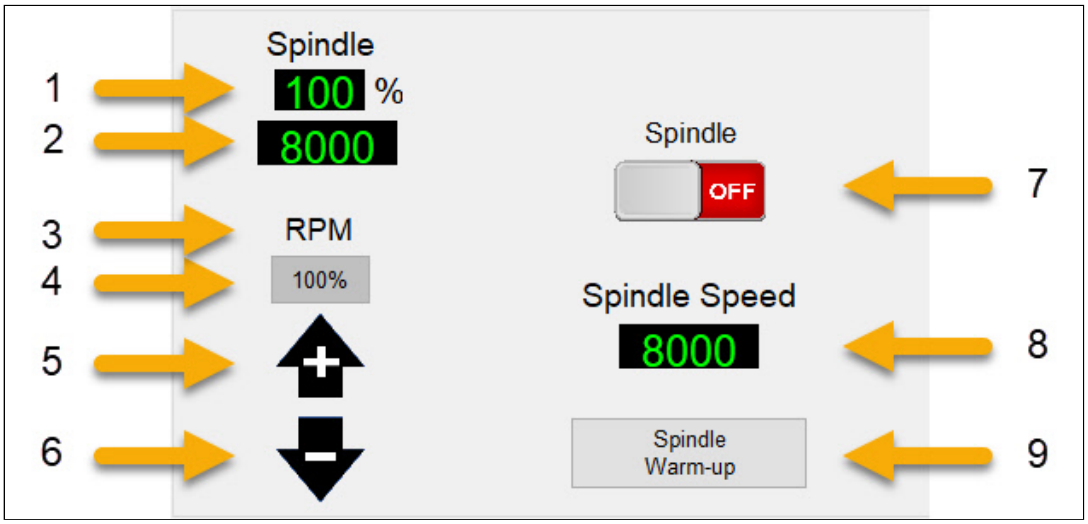
Feed Rate Note

The FRO arrows percentage can be adjusted in the Mach4 Configuration window.

Operation Note

When a DRO (digital read out) is black with green letters, you can enter numeric values directly into the DRO.

Spindle Control



- The Spindle Control group will only be visible with the spindle cutting method.

1. Current spindle override percentage.

2. Current spindle override speed. This will be the actual speed of your spindle during operation.

3. Units for spindle speed, RPM (rev/min).

4. 100% button will change the spindle override percentage back to 100%.

5. Up arrow will increase the percentage by 25%.

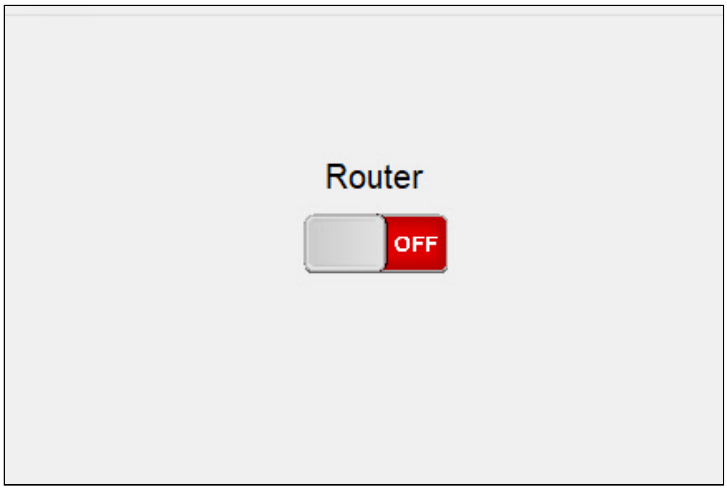
6. Down arrow will decrease the percentage by 25%.

7. Spindle On/Off toggle button. This can be used to manually turn the spindle on and off. It will also display the current state of the spindle while a G-Code program is running.

8. Spindle Speed. This is the spindle speeds 100% value. This value cannot be changed while a G-Code program is running.

9. The spindle warm-up button will start a warm-up procedure for the spindle. It is recommended to use this warm-up procedure if your spindle has not been used for more than 4-5 hours.

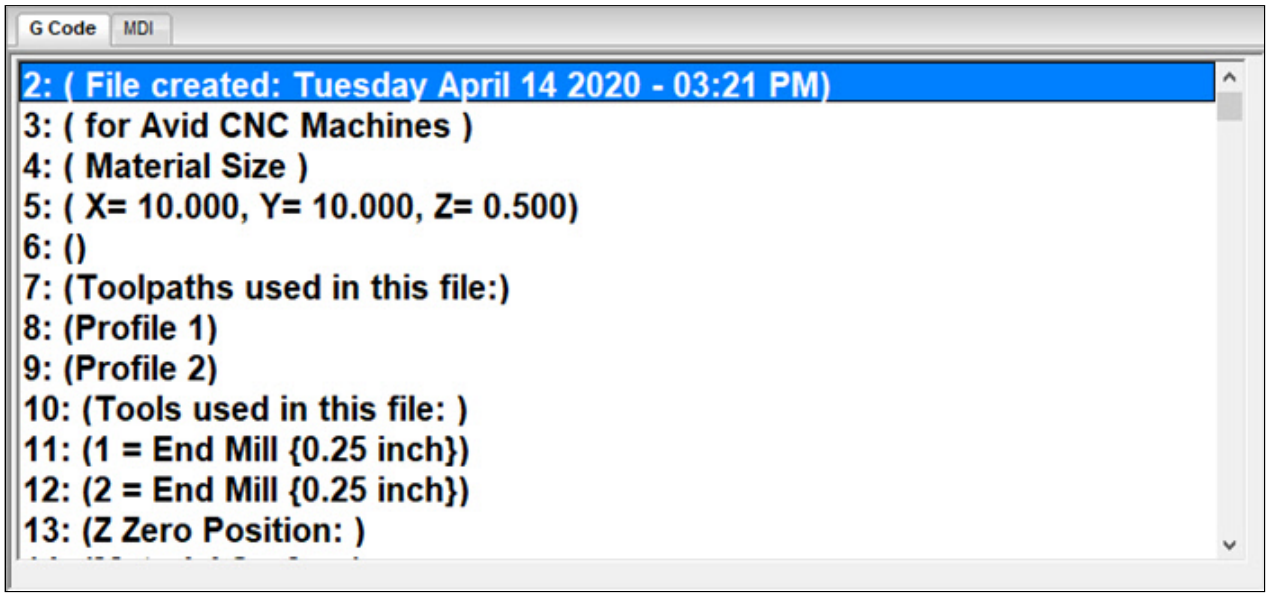
Router Control



- The Router Control group will only be visible with the router cutting method.
- **Router On/Off toggle button:** This can be used to manually turn the router on and off. It will also display the current running state of the router while a G-Code program is running.

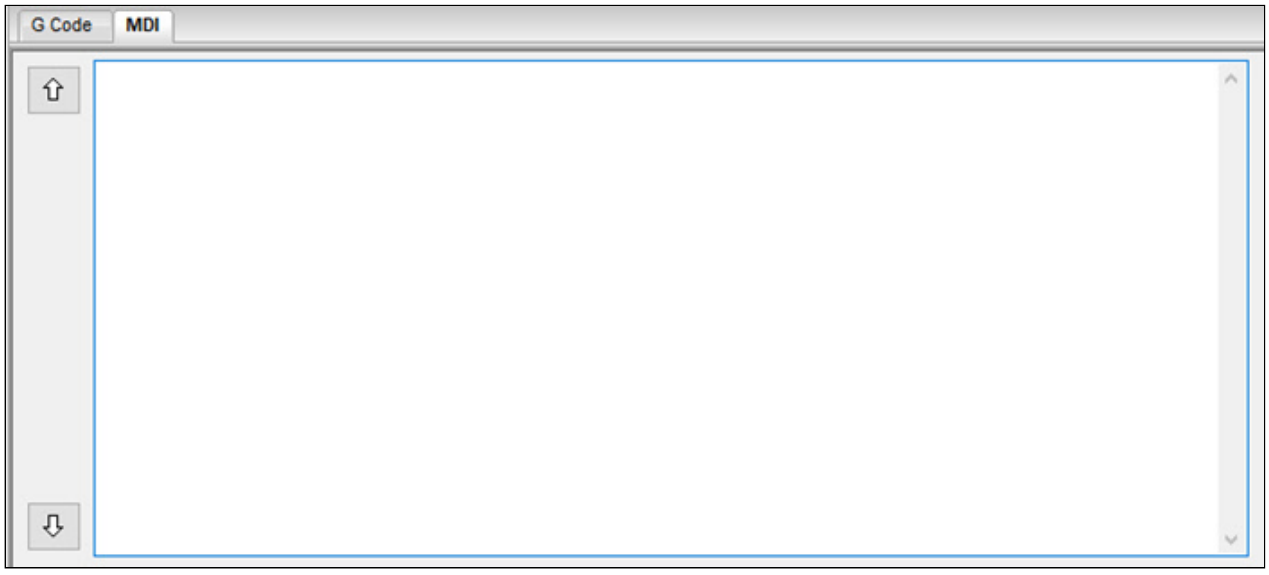
Program Run Tab

G Code Window



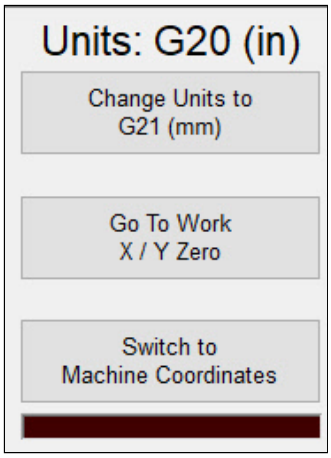
- The G Code tab will display your currently loaded G-Code program. You can double click inside the window to show line numbers.

MDI Window



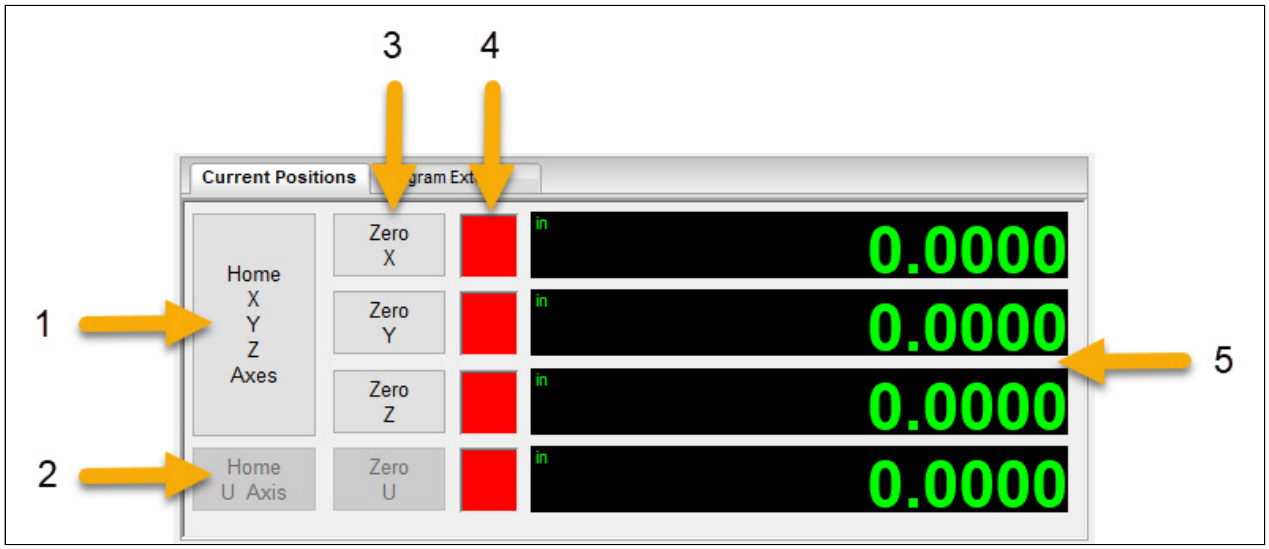
- The MDI (Manual Data Input) window allows manual entry of blocks of G-Code. These commands can then be executed by pressing the "Cycle Start" button.

Units Group



- **Units:** This displays your current units mode, either G20 (inches) or G21 (mm).
- **Change Units:** This button changes between G20 and G21 units mode. The machine must be enabled to use this button.
- **Go To Work X / Y Zero:** This button will move the machine to the work coordinate X and Y zero position.
- **Switch to Machine Coordinates:** Use this button to switch the position DROs between displaying work or machine coordinates. When displaying machine coordinates, below the button will be a blinking red LED.

Current Positions



1. Home X, Y, & Z Axes: This button will reference (home) the X, Y, and Z axes. It is recommended to home your machine each time you start Mach4, power on your electronics, or anytime the axes have been dereferenced (such as when the E-Stop has been activated). If you are using Custom Homing Settings, this button will home all axes you have selected to home by assigning their home order.

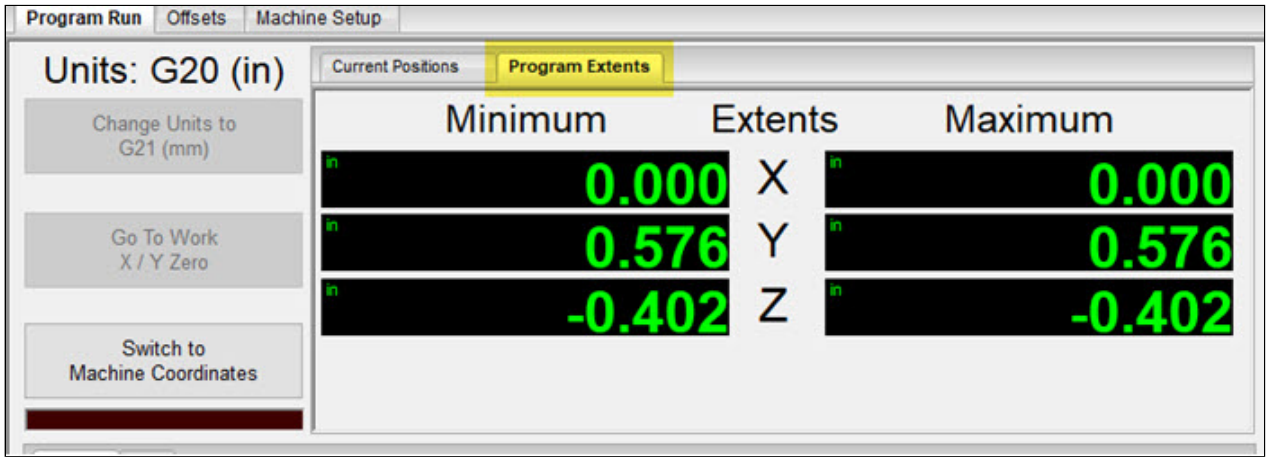
2. Home U Axis: When using a 4th axis, this button will home either the U axis (a linear 4th axis) or A axis (a rotary 4th axis).

3. Zero buttons: These buttons will set your current work coordinate position to zero for the selected axis.

4. Axis Referenced LEDs: These LEDs will display whether the respective axis is in a referenced state. After you home your machine, these LEDs will be green. If they turn red, it means a condition has occurred to dereference the axis. You will need to re-home your machine in that situation.

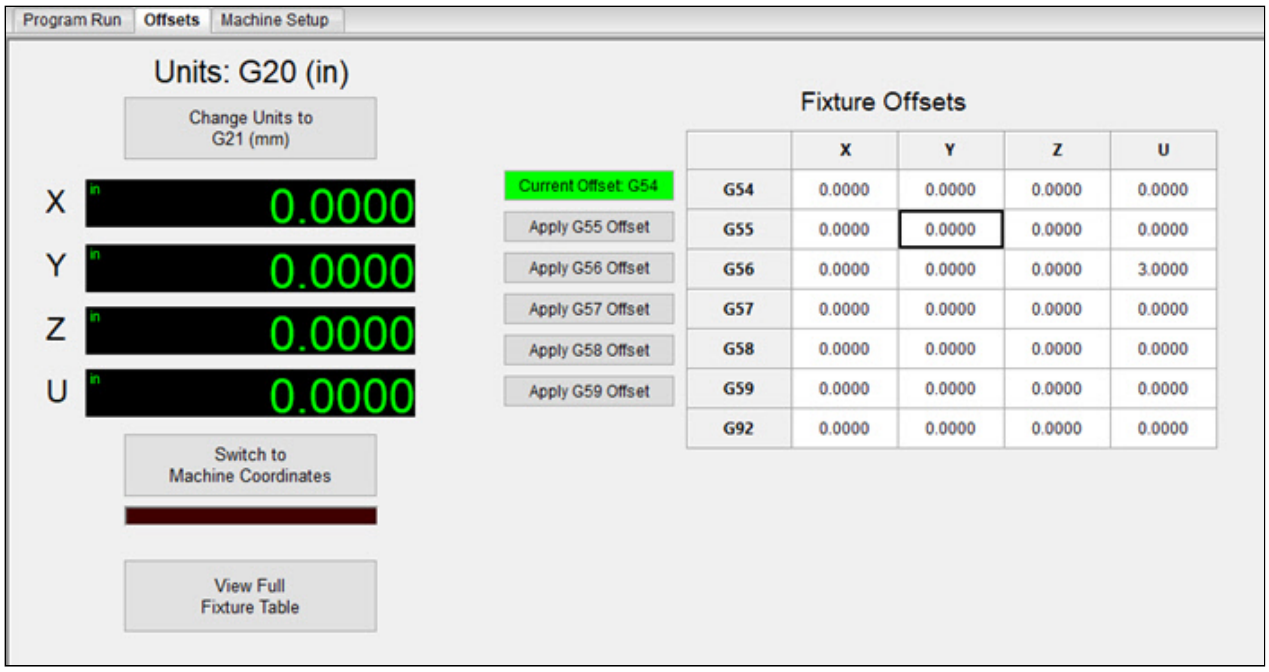
5. Position DROs: These DROs display the current axis position in either work or machine coordinates.

Program Extents



- The program extents tab shows the minimum and maximum positions (for the X, Y, and Z axes) of the currently loaded G-Code program. These can be shown in machine coordinates by using the "Switch to Machine Coordinates" button.

Offsets Tab

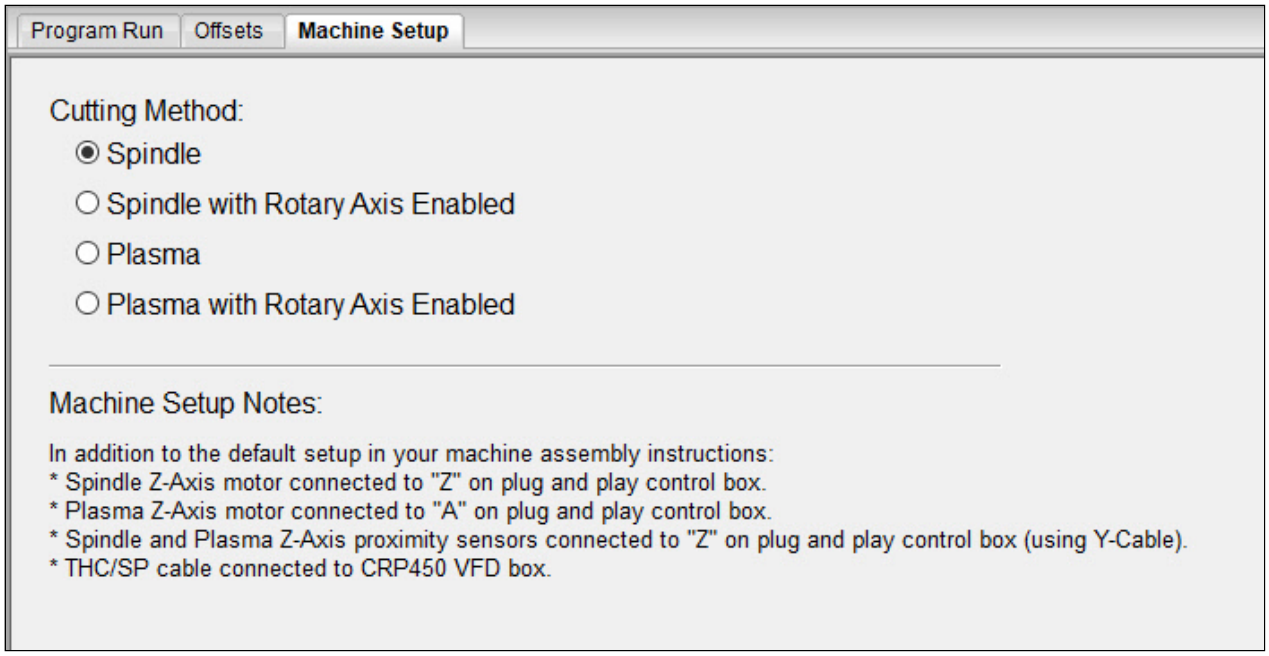


- **Units:** This displays your current units mode, either G20 (inches) or G21 (mm).
- **Change Units:** This button changes between G20 and G21 units mode. The machine must be enabled to use this button.
- **Position DROs:** These DROs display the current axis position in either work or machine coordinates. These will display the same values as the position DROs on the Current Positions tab.
- **Switch to Machine Coordinates:** Use this button to switch the position DROs between displaying work or machine coordinates. When displaying machine coordinates, below the button will be a blinking red LED.
- **View Full Fixture Table:** This will open a window that displays all of the offsets for all axes (including G54.1 P value offsets). Note, the values displayed in this table are in Machine Setup Units. The default units are inches, unless you chose the option to set your machine up in metric Machine Setup Units.
- **Apply GXX Offset:** These buttons will apply the respective offset. If the particular offset is the current offset, that button will be green and be labeled "Current Offset".
- **Fixture Offsets Table:** This table displays the current offsets for G54 - G59. You can also edit these values by clicking in the desired cell and entering any numeric value. These offset values are displayed in your current Units Mode, either G20(inches) or G21(mm).

Machine Setup Tab

Cutting Method

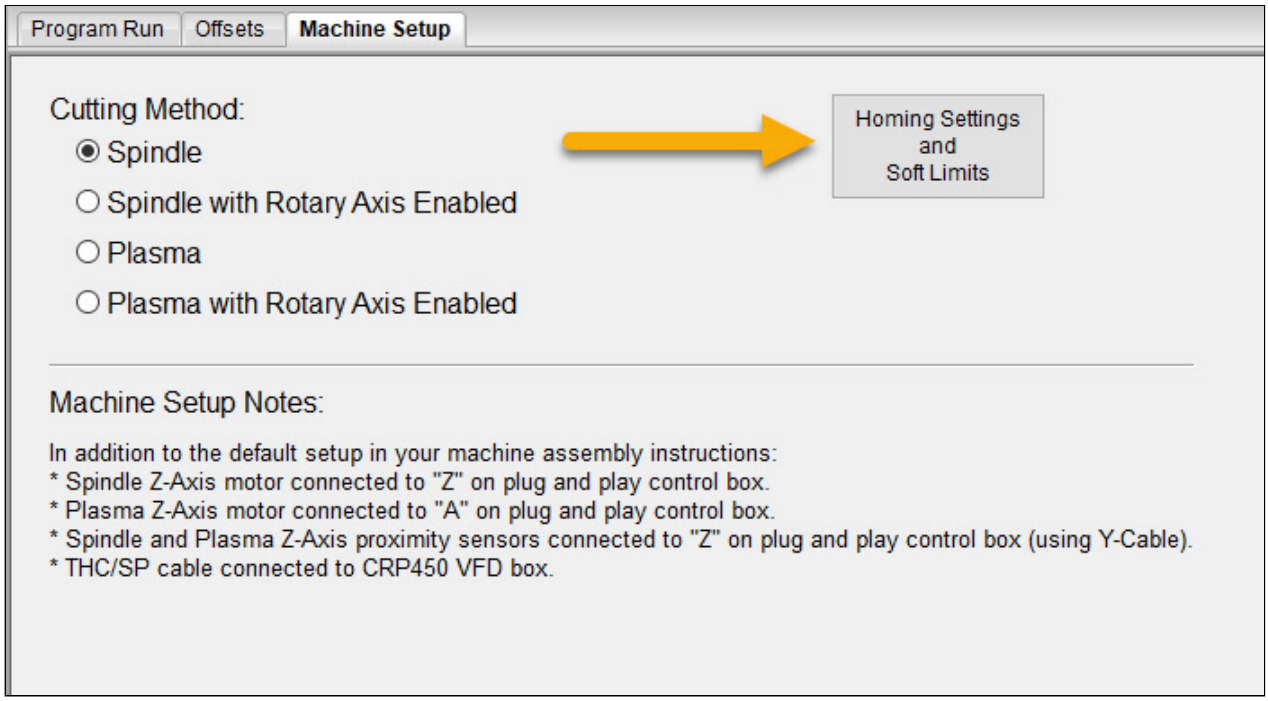
The radio buttons on this tab are used to switch between cutting methods when you have configured Mach4 with either a 4th axis, or with both milling and plasma functionality. An example screen is shown below with Mach4 configured for a machine that has a spindle, plasma torch, and Avid CNC rotary 4th axis.



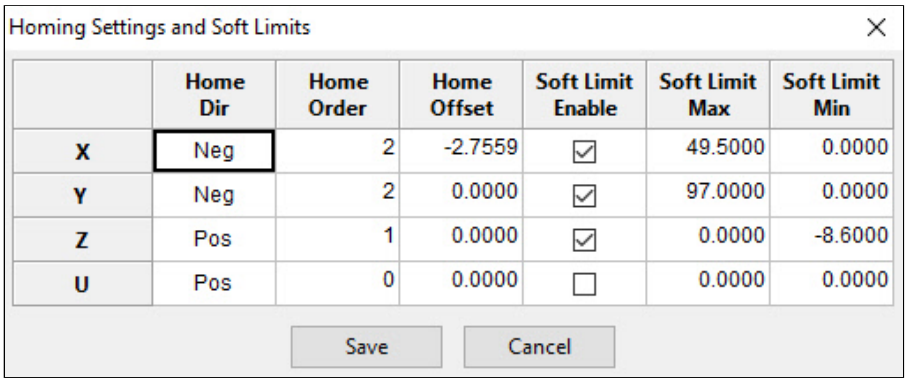
- Switching from a milling operation (either router or spindle) to plasma will display new screen elements that are specific to plasma users. These are documented in the Mach4 Plasma Users Guide.
- Enabling a cutting method which includes a 4th axis (rotary or linear) will enable that motor and the necessary buttons to control that axis. If using an Avid PRO CNC Rotary Axis, further instructions are available in the **Rotary Users Guide**.
- The **"Machine Setup Notes"** below the cutting methods will specify any connections that may need to be changed when the cutting method is changed.
- Cutting methods cannot be changed if Mach4 does not have active communications with your Avid CNC control system. A warning message will be displayed with steps to reestablish communications.

Homing and Soft Limits

If you enabled "Use custom soft limits" or "Use custom homing settings" during the Mach4 Configuration process, you will see an additional button on the Machine Setup tab. The button label will be different than the screen shown below if you only enabled one of those advanced settings.



Clicking the button shown above will open a table. The example table shown below will be seen if both the custom soft limits and custom homing settings options are enabled. If only one is being used, the table will only display the items for those settings.



	Home Dir	Home Order	Home Offset	Soft Limit Enable	Soft Limit Max	Soft Limit Min
X	Neg	2	-2.7559	☑	49.5000	0.0000
Y	Neg	2	0.0000	☑	97.0000	0.0000
Z	Pos	1	0.0000	☑	0.0000	-8.6000
U	Pos	0	0.0000	☐	0.0000	0.0000

- **Home Dir:**
  - This is the direction the respective axis will home to. The **standard sign conventions** are used for all models of Avid CNC machines. To change the home direction, click in the cell to display a drop-down menu and select either Pos or Neg.
  - Depending on machine model and electronics, you may be required to relocate proximity switches to accommodate the new home direction. In this case, you will be instructed with these changes after selecting the new home direction. If your machine is a large-format PRO CNC with CRP800 electronics, ensure you have the correct **sensor configuration** selected.
- **Home Order:** This is the order in which homing of the axes will take place. Setting the home order to "0" means that axis will not be homed using the "Home Axes" button in the Current Positions tab. The axis with a home order of "1" will home first, then the axis with a home order of "2", etc.
- **Home Offset:** This is the offset that will be applied to your current work offset once the machine homes that axis.
- **Soft Limit Enable:** This option enables soft limits for the respective axis. It does not turn soft limits on or off. When you turn soft limits on with the button in the Jogging tab, soft limits will only be used on axes which have "Soft Limits

Enabled" checked in this table.

- **Soft Limit Max:** Maximum soft limit value in machine setup units (inches for imperial setup units and mm for metric).
- **Soft Limit Min:** Minimum soft limit value in machine setup units (inches for imperial setup units and mm for metric).
- **Save:** This button will save your new homing and / or soft limit settings.
- **Cancel:** This button will close the table without saving any changes you may have made.

Auto Z Touch Plate

Auto Z and Corner Finding Touch Plate video: https://youtu.be/CnAlZiYjeiQ

This video explains how to use the Auto Z and Corner Finding Touch Plate. It currently features the use of Mach3 CNC control software, however the functions of the touch plate remain the same in Mach4. The Mach4 touch plate user interface is described in this section. Begin by clicking the Auto Z Touch Plate button to display the window shown below.

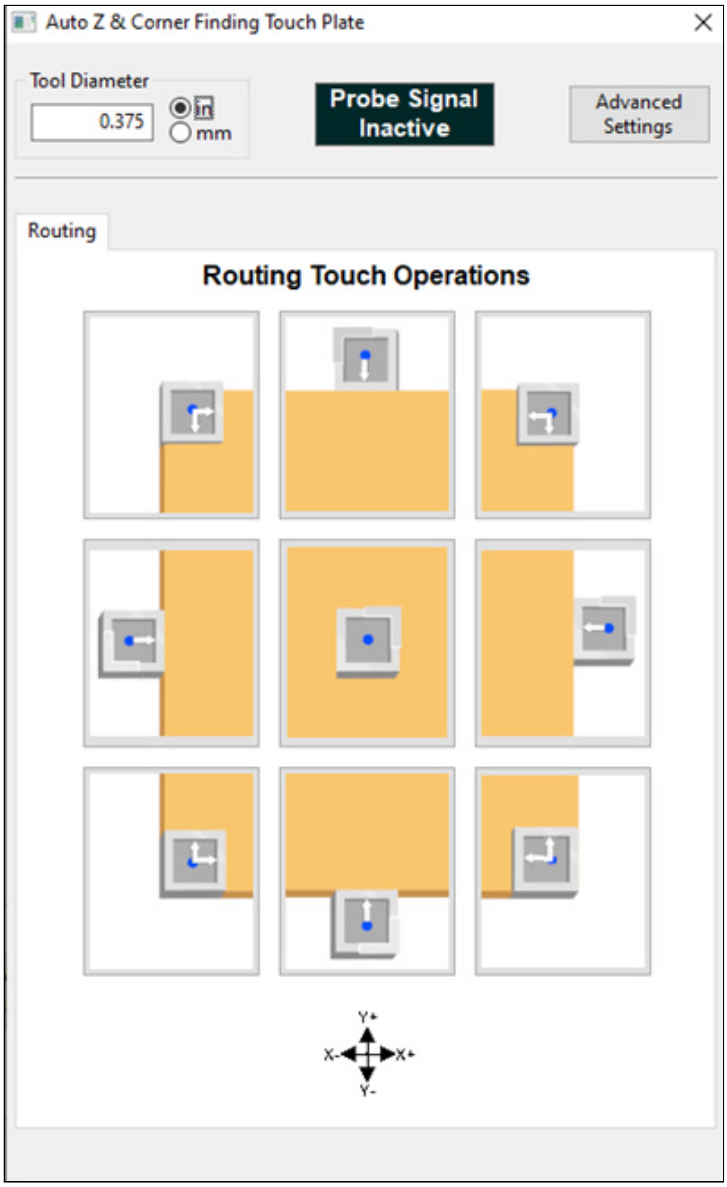
Conductivity Note

The touchplate must be electrically isolated from the machine to function as intended. It cannot be used directly on metal or workholding that has a conductive path to the machine frame.

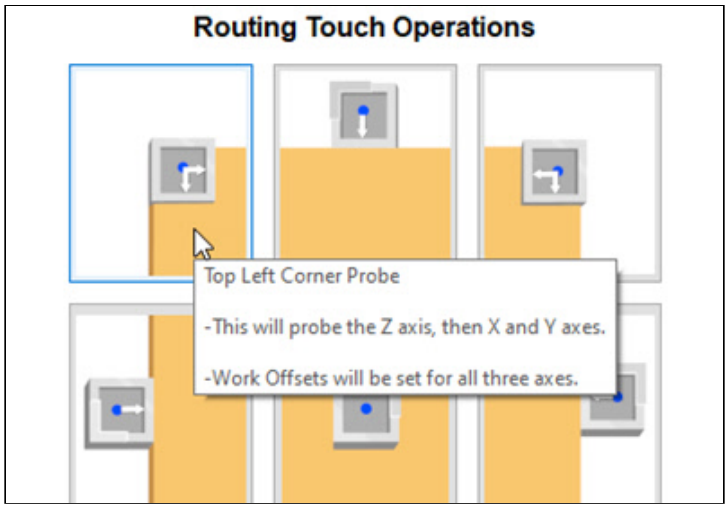
Machine Setup Note

Plug the M12 connector from the touch plate into the **"Aux 1"** sensor input on your plug and play control box.

Routing Operations

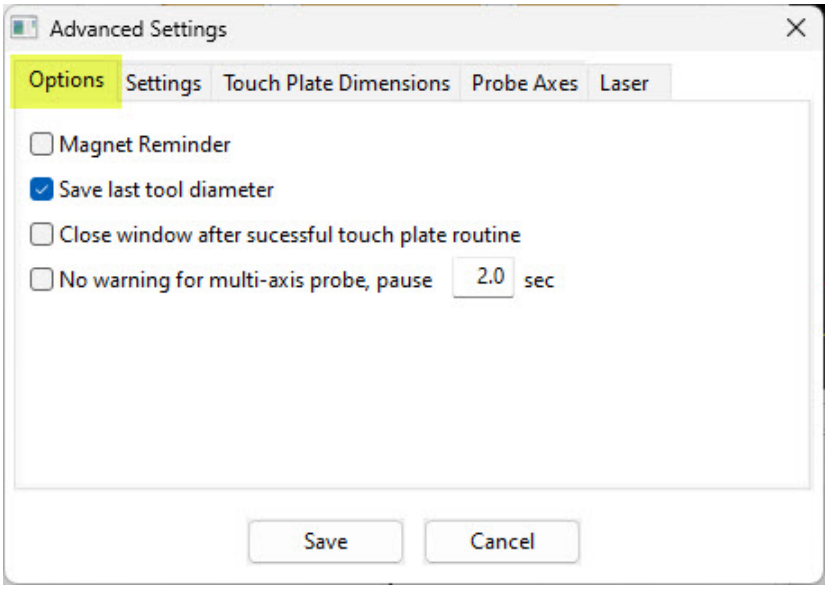


- **Tool Diameter:** Enter your tool diameter and select the correct units (inches or mm).
- **Probe is Currently Inactive:** This will tell you if your touch plate probe is currently active or inactive. You can functionally test your touch plate by touching the magnet to the touch plate. You will see this box turn blue and display "Probe is Currently Active".
- **Routing Touch Operations:** There are nine buttons corresponding to the nine possible locations of the touch plate. When you hover the cursor over a button, it will display a box explaining what that operation will do.

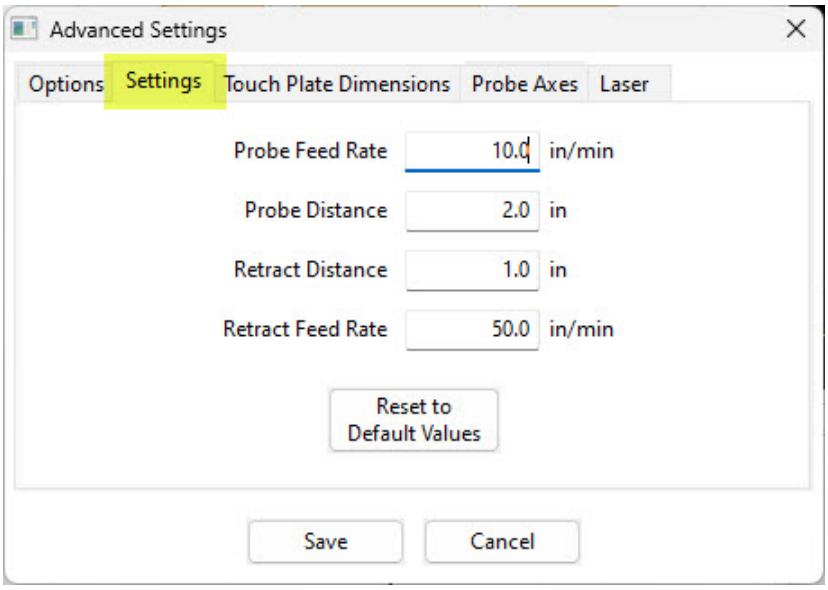


- **Advanced Settings:** This will open the Advanced Settings dialog for the touch plate, described below.

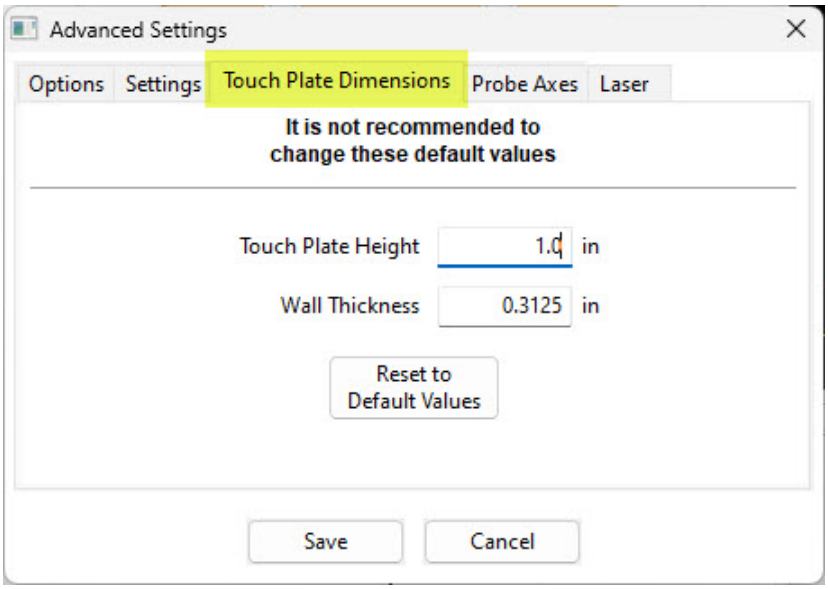
Advanced Settings



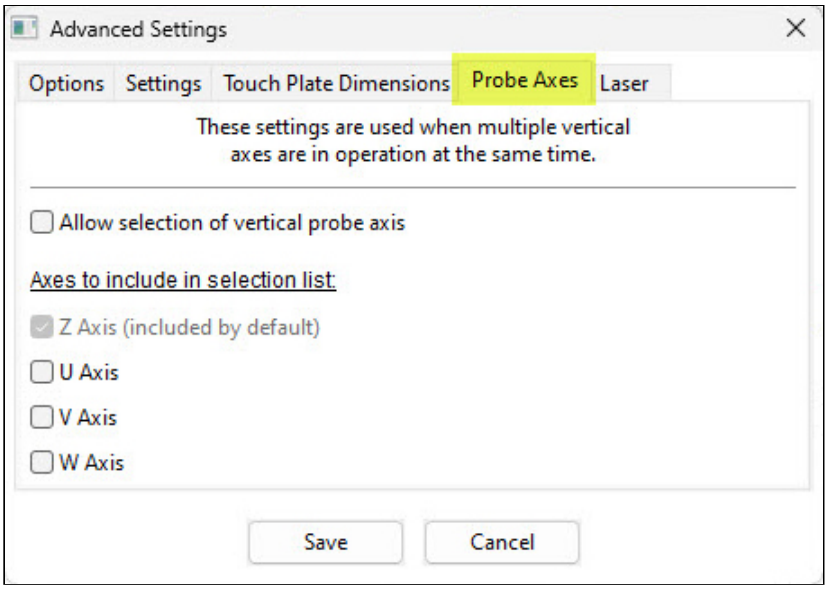
- **Options Tab:**
  - **Magnet Reminder:** With this option enabled, before each touch plate operation a warning message will be displayed with a reminder to attach the touch plate magnet.
  - **Save last tool diameter:** This option will save the last tool diameter entered. Disabling this option will set the tool diameter to 0 each time you open the touch plate window. You are then forced to enter a new tool diameter before any touch plate operation that requires a valid tool diameter.
  - **Close window after successful touch plate routine:** This option will close the touch plate window after your touch plate operation has completed and was successful.
  - **No warning for multi-axis probe:** By default, a multi-axis touch plate operation will display a message to orient the tool flutes for the correct axis. User input is then required to start probing the next axis. Enabling this option removes the user input and will pause before probing the next axis (allowing time to orient the tool flutes). The pause length can be adjusted in the text box shown above.

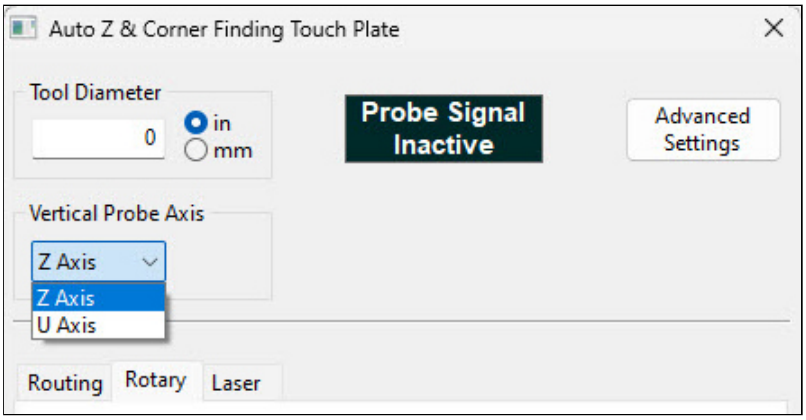


- **Settings Tab**
  - **Probe Feed Rate:** Feed rate used during a probe move.
  - **Probe Distance:** The distance a probe move is allowed to travel. If a probe strike is not detected (i.e. the tool does not make contact with the touch plate) in this distance, the probe move will stop and the current touch plate operation will terminate.
  - **Retract Distance:** The distance the tool will retract after finishing a touch plate operation.
  - **Retract Feed Rate:** The feed rate used during a final retract move.
  - **Reset to Default Values:** This button will reset all values on the "Settings" tab back to default values.

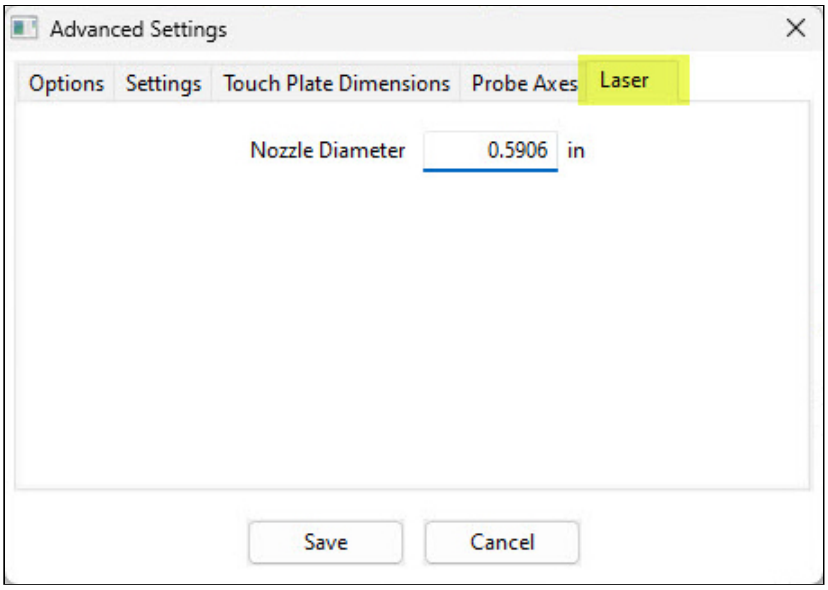


- **Touch Plate Dimensions Tab**
  - These values can be adjusted for custom touch plate applications.
  - **Touch Plate Height:** Height of the touch plate from the surface of the work piece to the brass contact surface on the top of the touch plate.
  - **Wall Thickness:** Thickness of the side wall of the touch plate.
  - **Reset to Default Values:** This button will reset all values on the "Touch Plate Dimensions" tab back to default values.



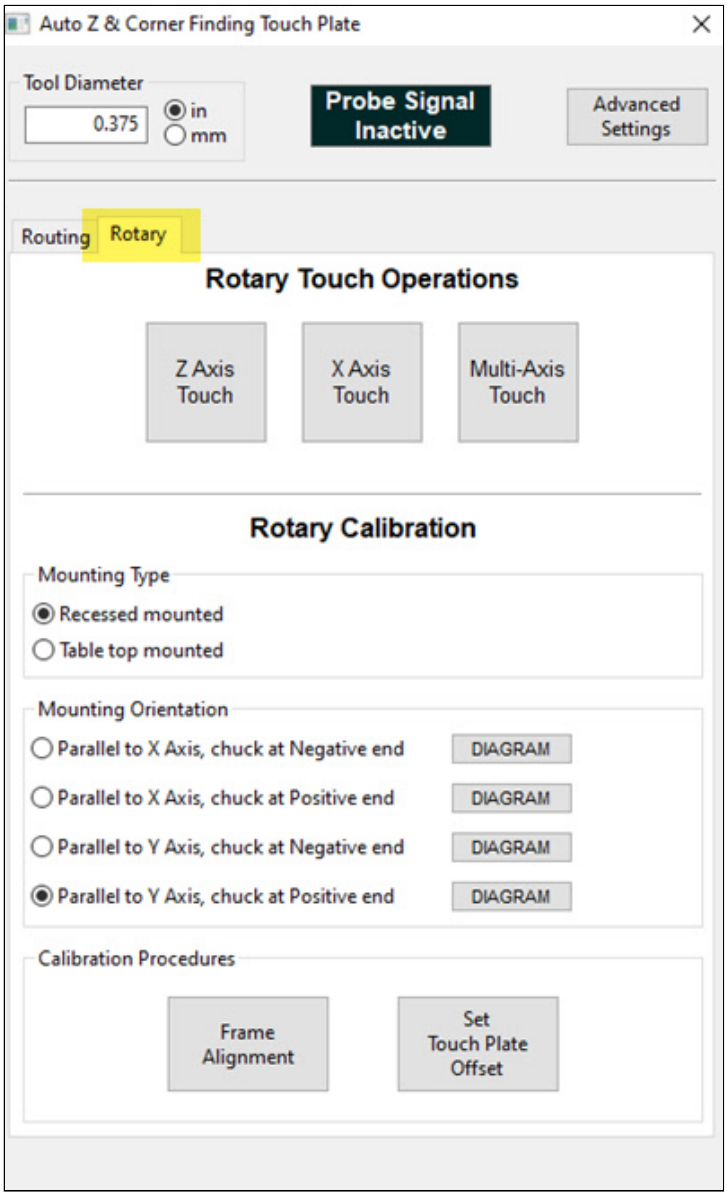


- **Probe Axes Tab**
  - **Allow selection of vertical probe axis:** For machines with multiple vertical axes. This enables a dropdown list to select the active vertical axis for touch operations.
  - **Selection list:** Populates the list of available vertical axes.



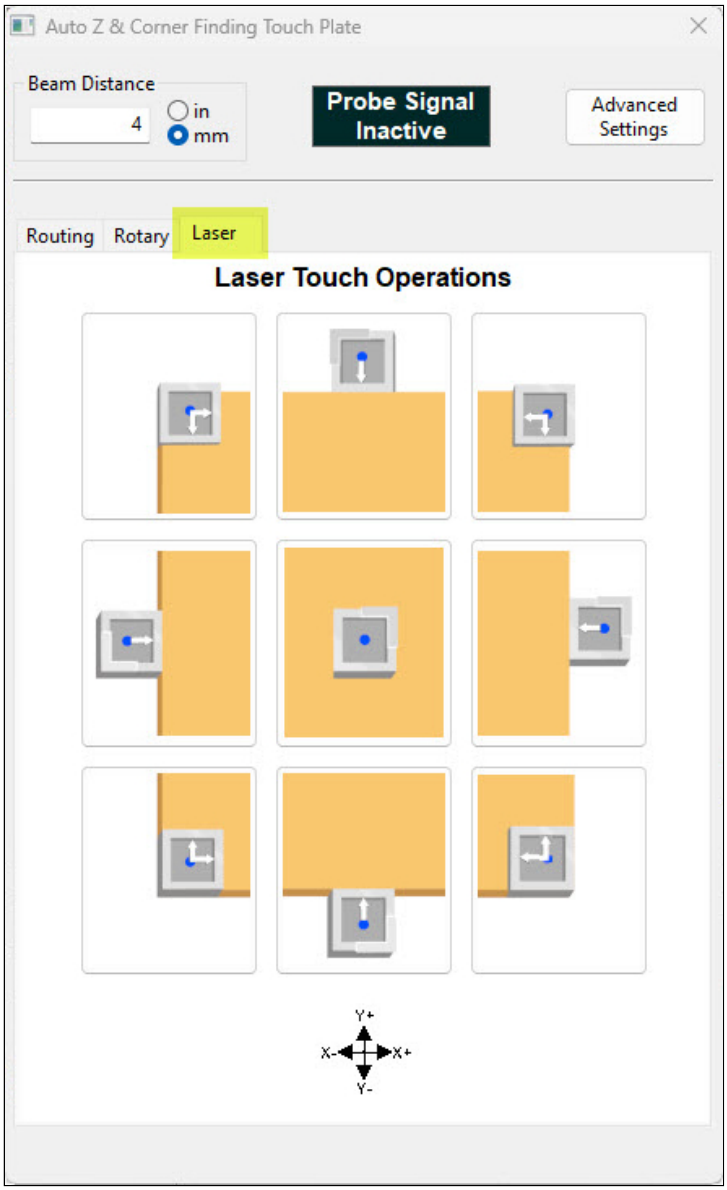
- **Laser Tab**
  - **Nozzle Diameter:** The diameter used during Laser touch operations.
  - Refer to the Laser Users Guide for more information.

Rotary Operations



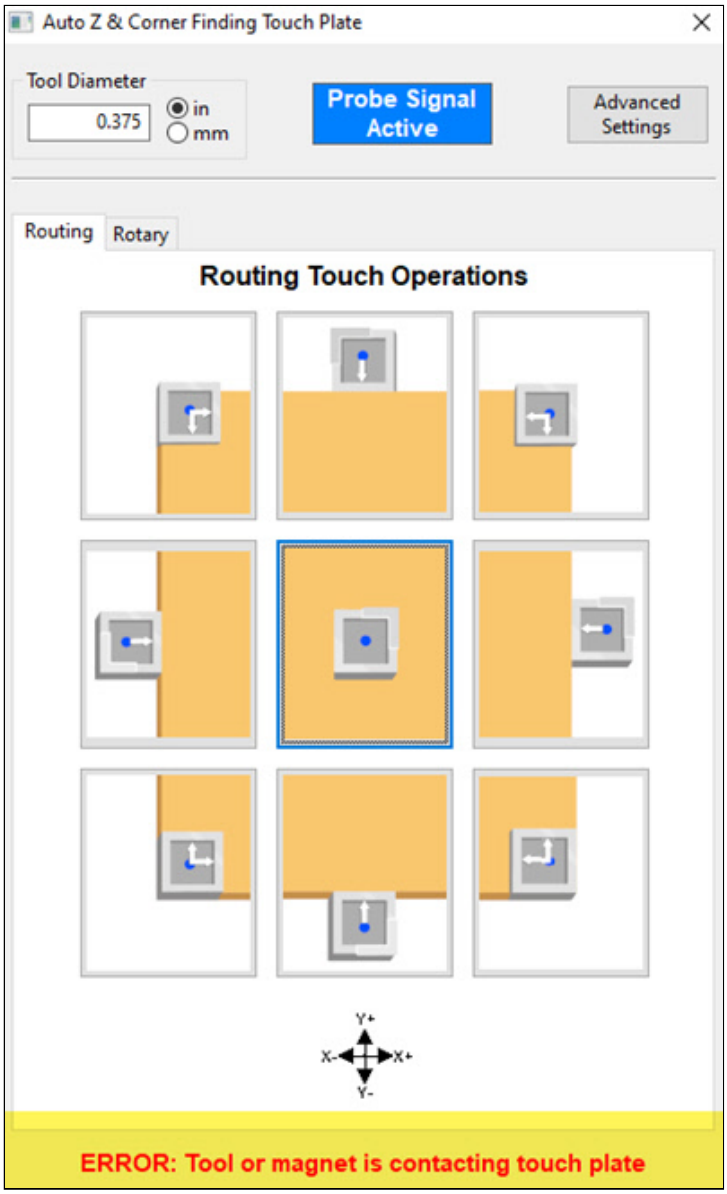
- If you are using an Avid CNC PRO Rotary Axis, there are additional rotary-specific touch plate operations on the **Rotary** tab.
- Refer to the Rotary Users Guide for more information.

Laser Operations



- If you are using an Avid CNC Laser kit, there are additional laser-specific touch plate operations on the **Laser** tab.
- Refer to the Laser Users Guide for more information.

Error Messages



- If an error occurs during a touch plate operation, an error message will be displayed at the bottom of the touch plate window.

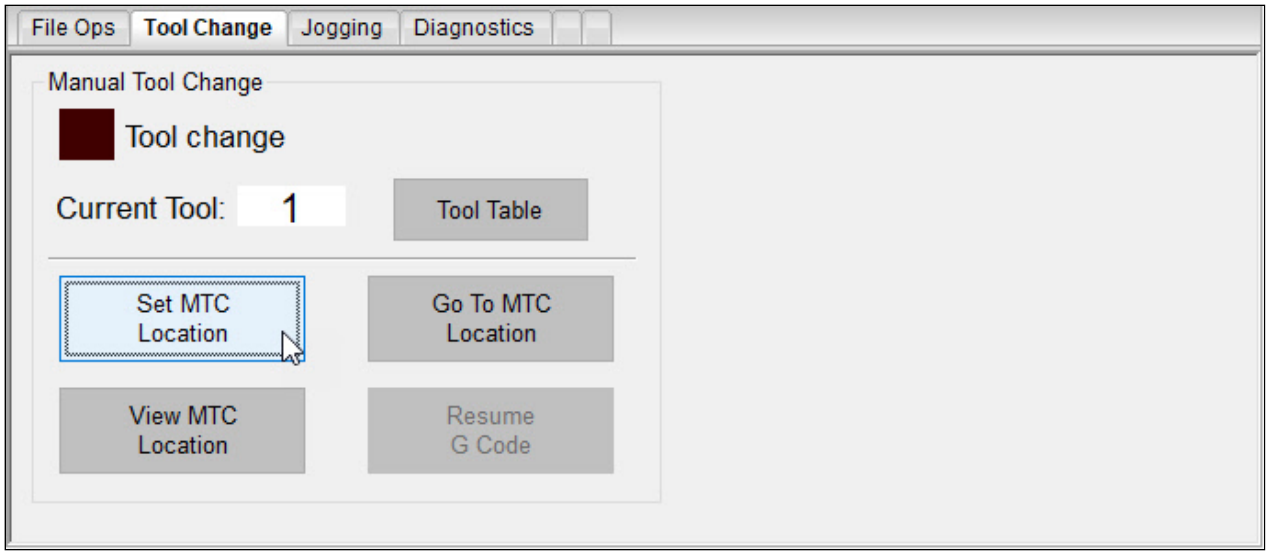
Manual Tool Changes

The M06 macro provided with the Avid CNC Mach4 profile enables manual tool changes when your CAM post processor outputs M06 tool changes. Buttons are provided in the Tool Change tab to allow you to set an optional Manual Tool Change (MTC) location. This can be used as a quick way to move the machine to a convenient location for tool changes.

The instructions below for manual tool changes include the use of the **"MTC Location"** buttons, although they are not required to be used. These buttons cannot be used if your machine does not have a Z axis home sensor. Make sure you have configured Mach4 with the correct homing sensors you have on your machine.

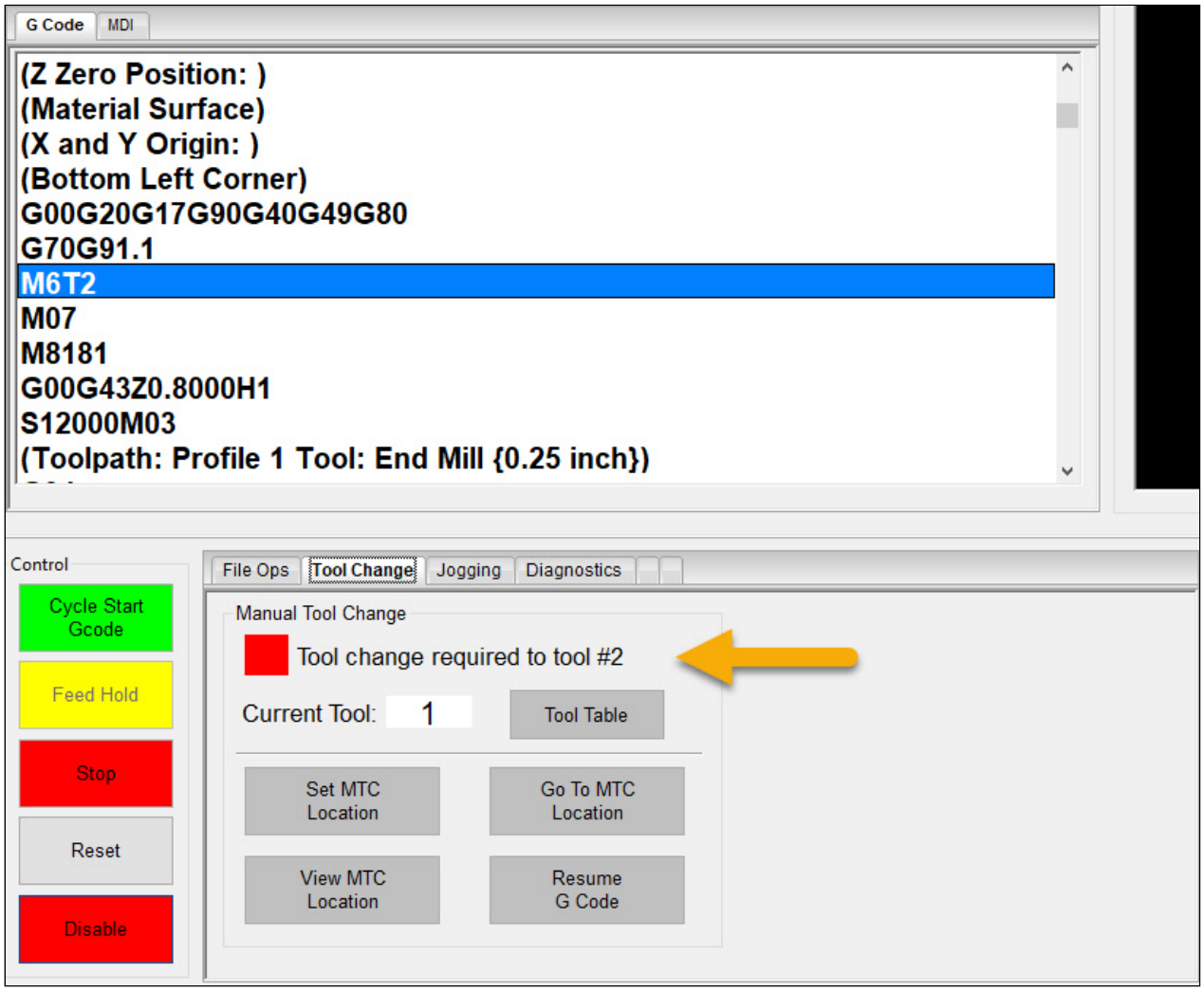
1. Set MTC Location

- Begin by homing your machine. It is **VERY IMPORTANT** to always home your machine when using the MTC location buttons.



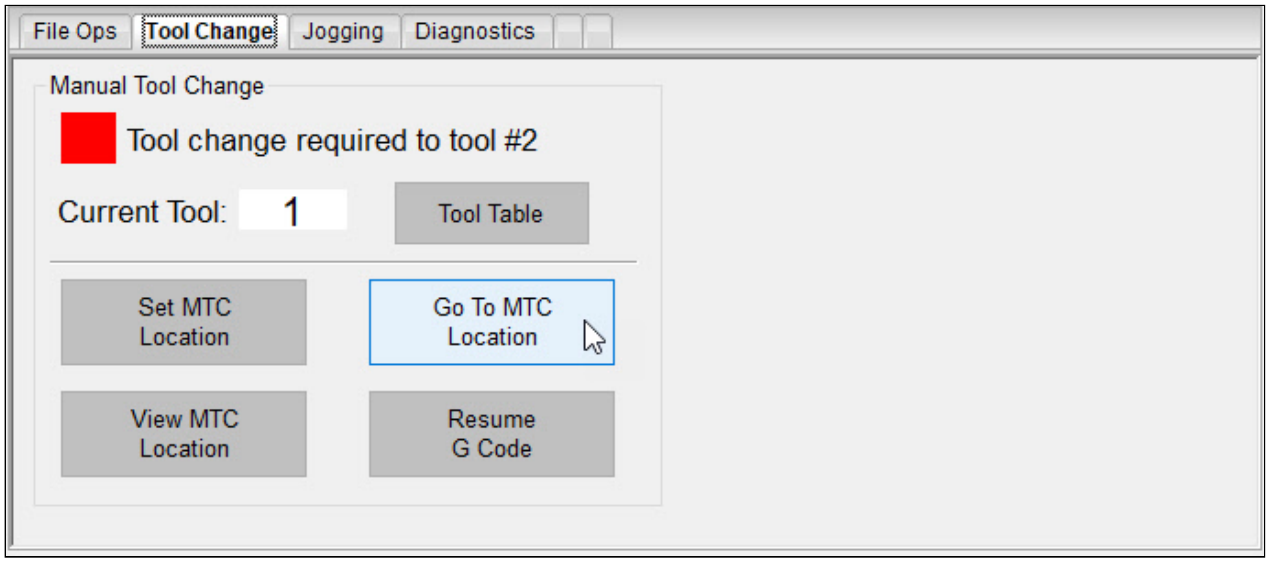
- Move your machine to the location you would like to perform your manual tool changes.
- Click **"Set MTC Location"**. A message box will display your new MTC location in Machine Coordinates.
- Unless you want to set a new MTC Location, you do not need to repeat this step.

2. Run G-Code Program



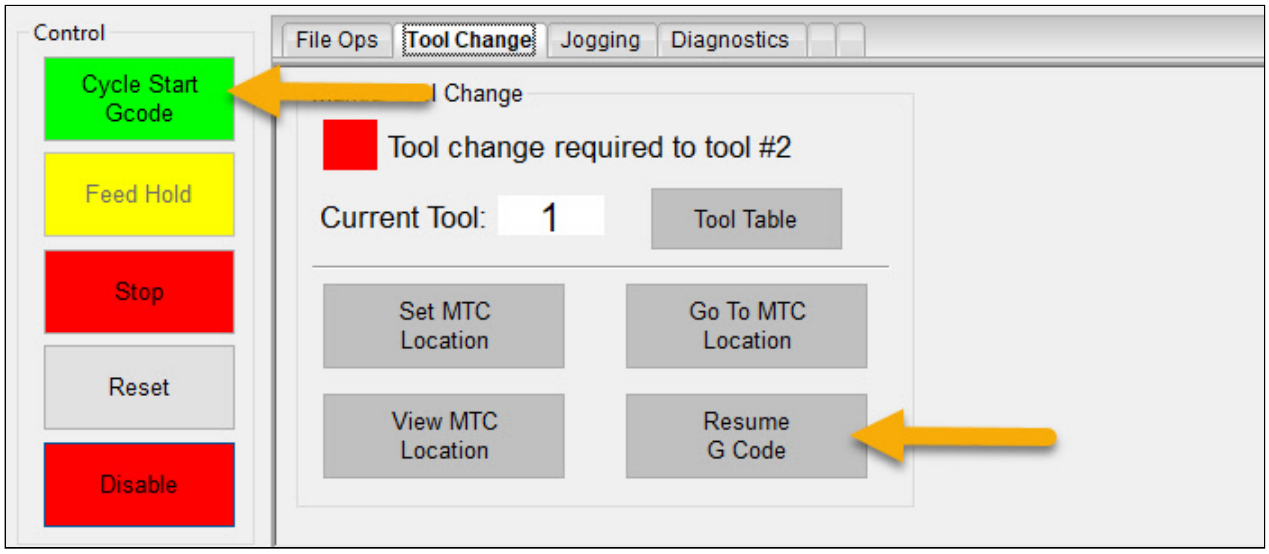
- When you encounter a tool change command in your program that specifies a tool number different than your current tool, the **Tool Change** tab will display the required tool number for the tool change.
- In this example, the G-Code program was started with tool "1" as the current tool. When the program reached the command "M6T2", it is requesting a tool change to number "2".

3. Change Tool



- You can now use the **"Go To MTC Location"** button to move the machine to your previously saved MTC location. Alternatively, you can also jog the machine to a desired spot for the tool change.
- Take caution to ensure the machine has a clear path to move to this location. Beginning at the maximum Z Axis clearance height, the machine will execute X and Y moves before moving to the MTC location Z position.
- Perform your tool change. You will most likely need to use the Auto Z Touch Plate to zero the Z Axis only, with your new tool.

4. Resume G-Code



- There are two ways to resume the G-Code program after completing a tool change:
  1. **Resume G-Code Button:** This button is enabled after the program encounters a tool change command. When using this to resume G-Code, the Z axis will move to a safe clearance height before continuing with the program.
  2. **Cycle Start G-Code Button** Using the cycle start button will resume program executing without performing a preceding move to safe Z height.

Mach4 Troubleshooting

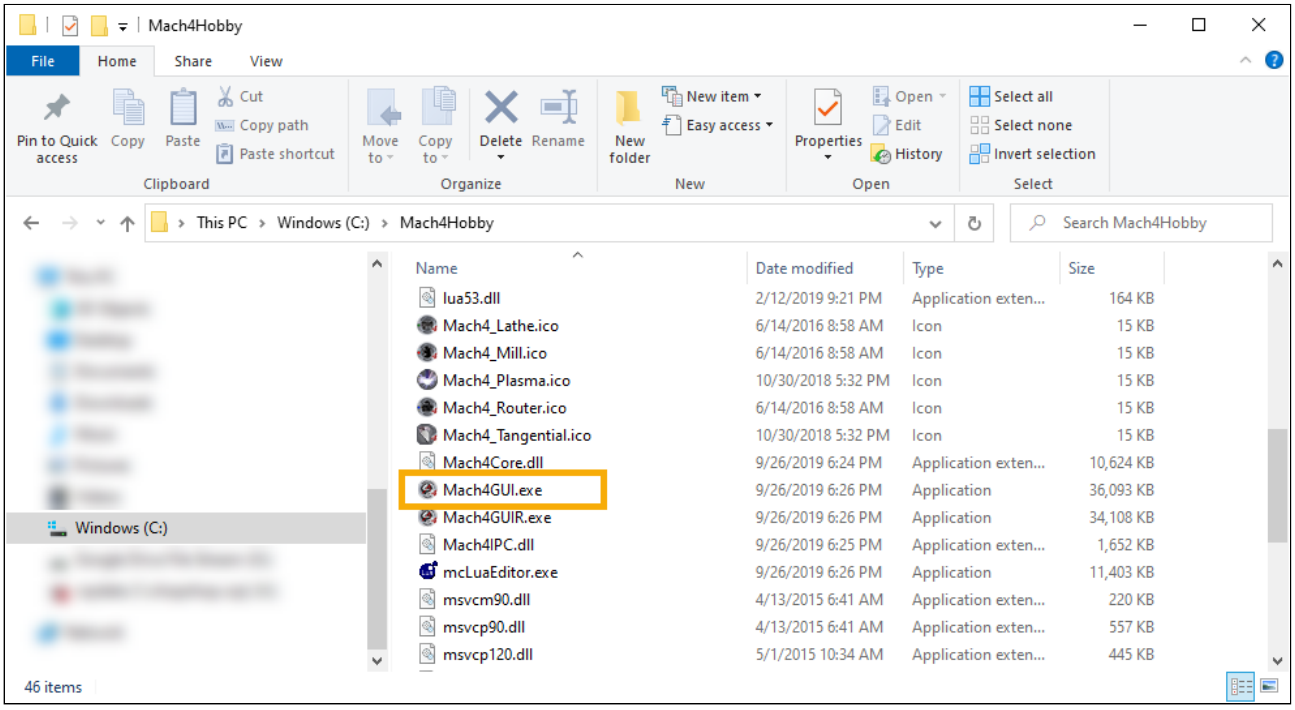
Restore Mach4 Profile

Mach4 profiles can be corrupted when the program fails to close properly. During use settings are transferred constantly between Mach4 and a .ini configuration file. When this transfer is interrupted settings can be unintentionally changed. Common symptoms are:

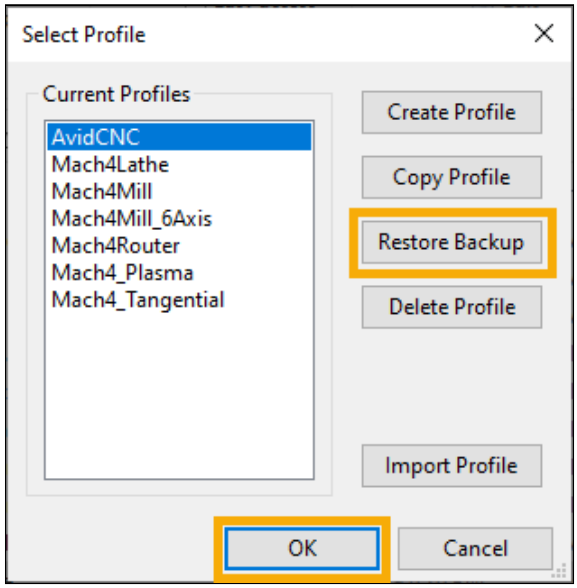
- Some axes moving backwards
- Some limit switches stuck on
- Plasma related error messages in a Router/spindle profile
- Generally inexplicable changes to Mach4 settings

A default configuration file is included with the Mach4 installation. This can be used to restore your Mach4 profile back to default settings by following the process below. If you have applied custom settings to your Mach4 profile, please Contact Us before continuing with this process.

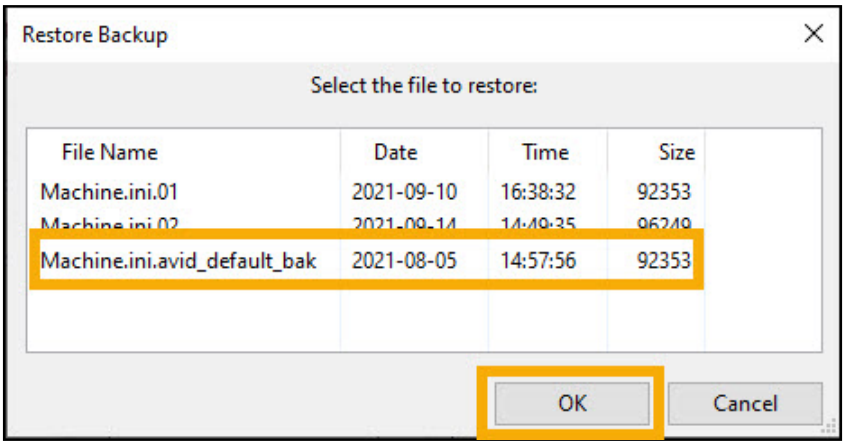
1. Close Mach4
2. Go to the **C:\Mach4Hobby** directory on your PC. Find the **Mach4GUI.exe** file and open it.



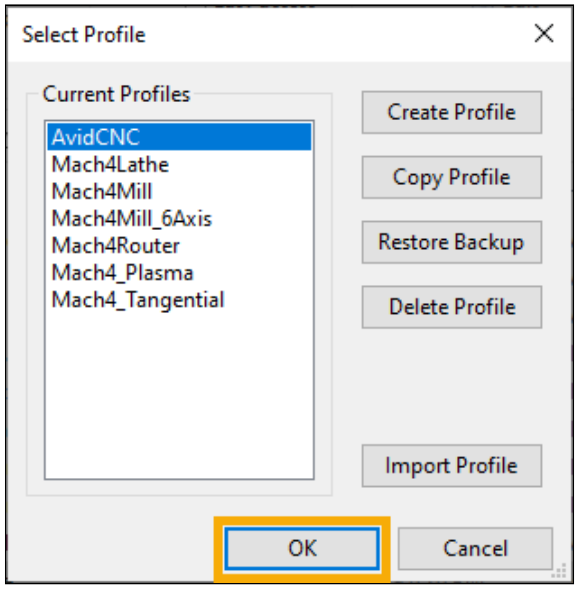
3. This will bring up the Mach4 load screen. Choose the profile you normally use (most likely AvidCNC) and select Restore Backup.



4. Choose the Machine.ini.avid_default_bak file and hit OK.

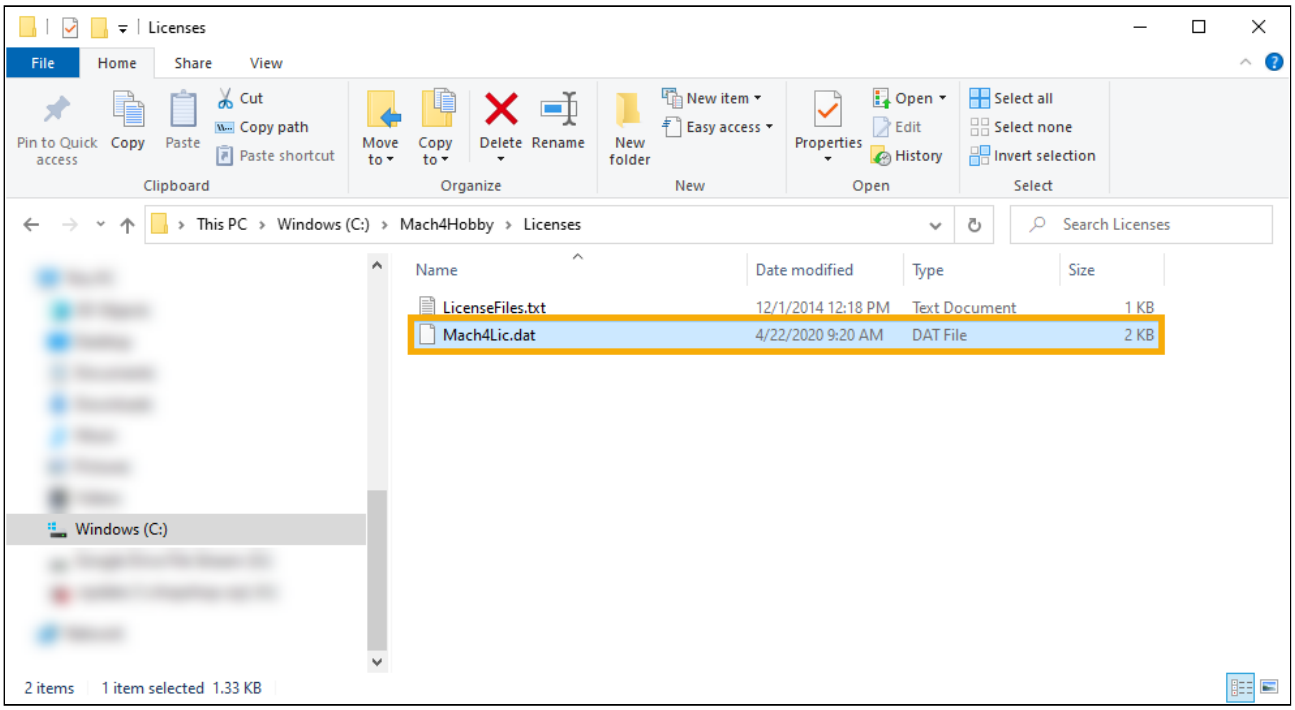


5. Back on the load screen hit OK and this will open up the Mach4 profile with default settings.

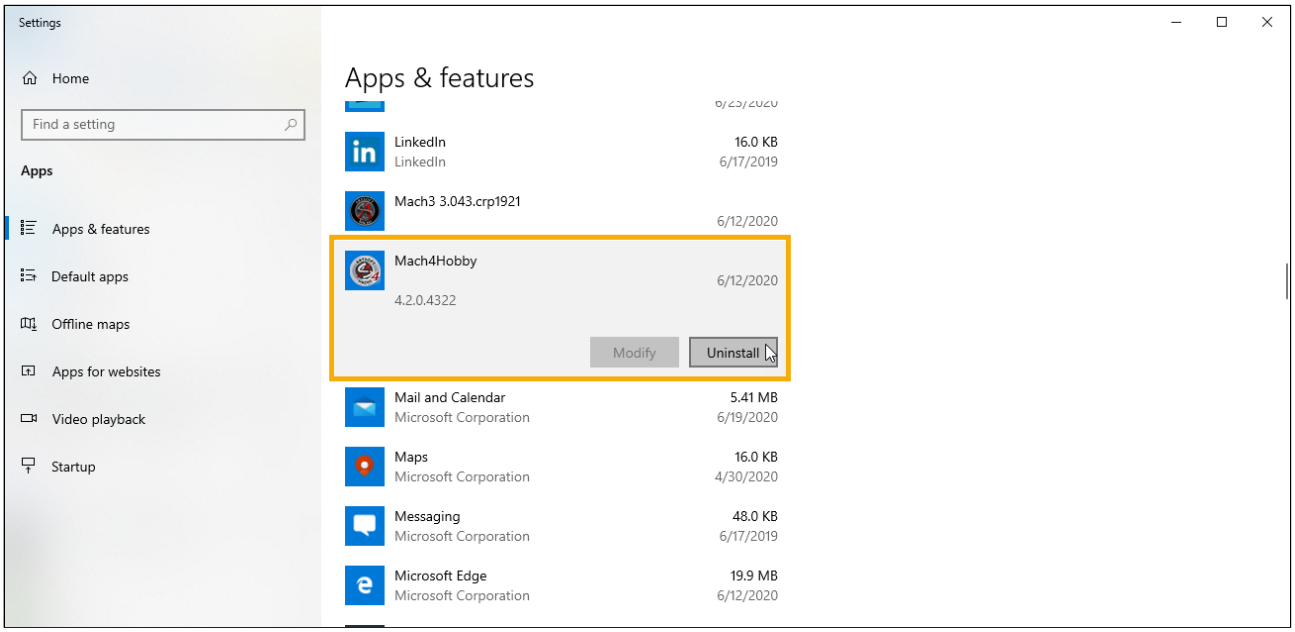


Reset Mach4 Installation

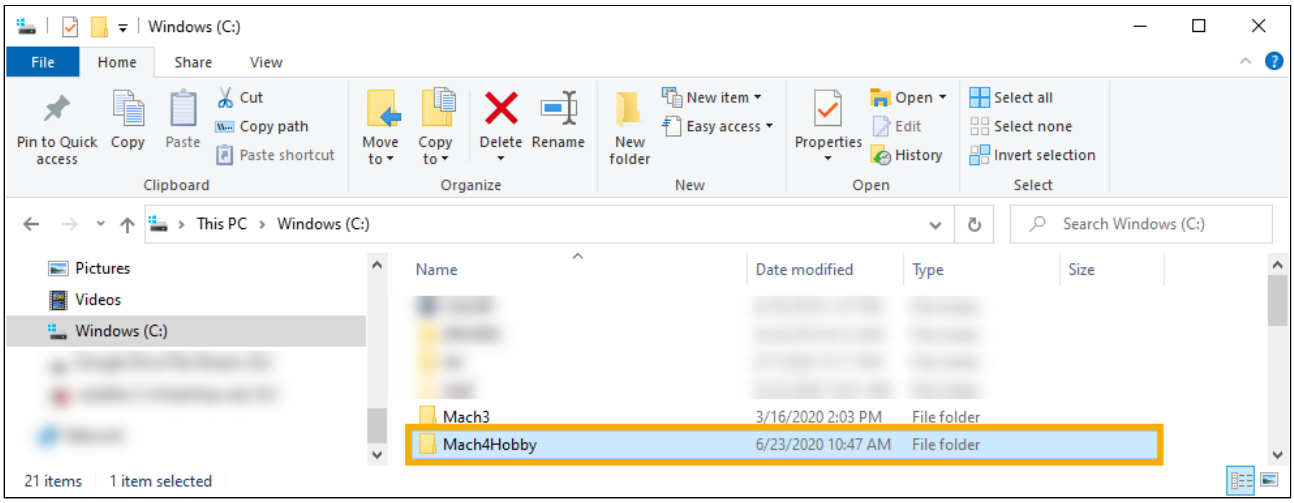
1. Navigate to **C:\Mach4hobby\licenses** and copy your Mach4lic.dat file to your desktop to save for later.



2. Navigate to Windows Apps and Features (or equivalent) and uninstall Mach4 Hobby.

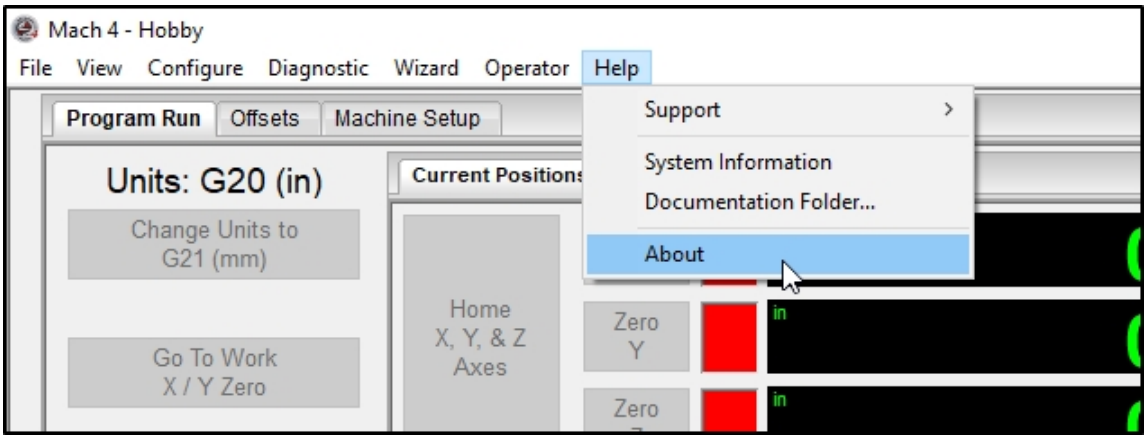


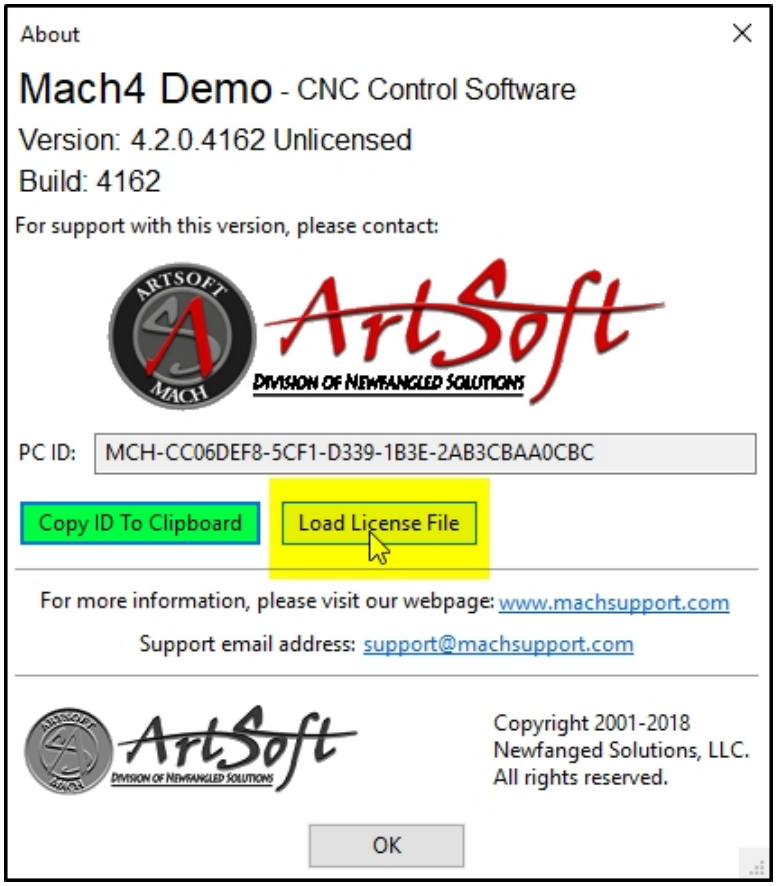
3. Navigate to **C:** and delete the Mach4 Hobby folder.



4. Use this link to download and install Mach4 from our website: **Mach4 Download Page**

5. Navigate to **Help > About** in Mach4 and reload your license file

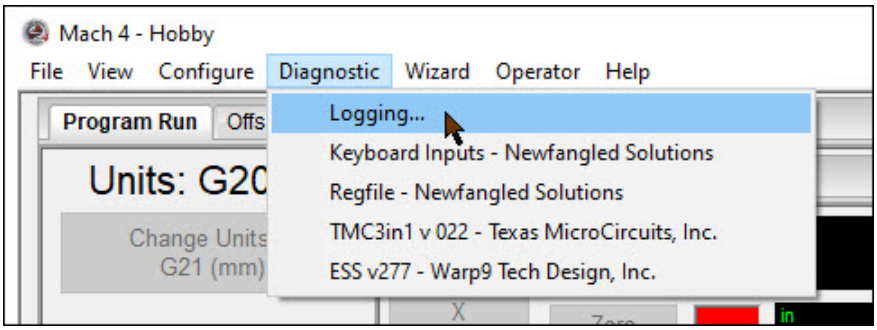




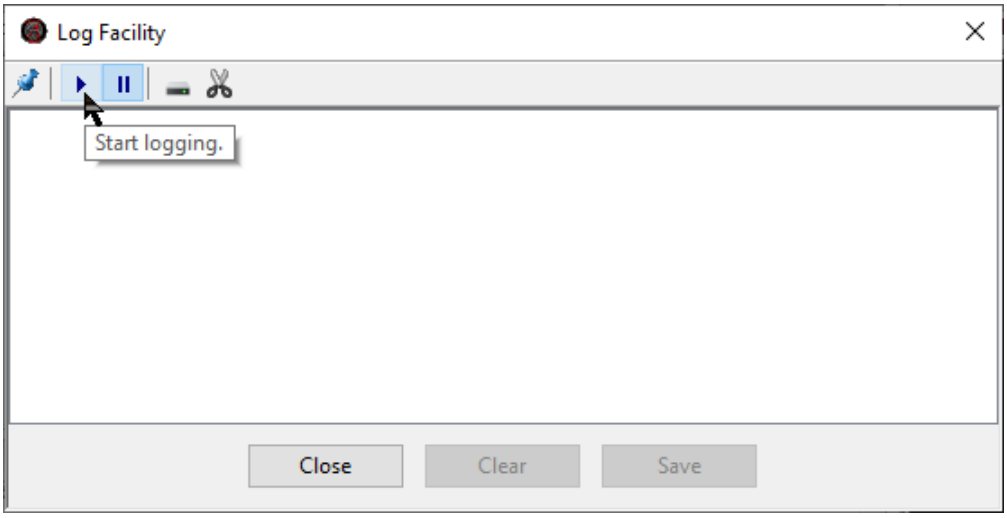
6. In Mach4 navigate to **Configure > Avid CNC Machine Configuration** and configure your machine. See our setup instructions for more information: Mach4 Configuration (https://www.avidcnc.com/support/instructions/software/mach4Configuration)
7. Verify you can jog by using the arrow keys. Verify you can Home by pressing Home X Y Z Axes.
8. If you find you are not able to move the machine, re-run the Warp9 System Configuration Utility that will have installed onto your desktop when installing Mach4. See our Mach4 setup instructions for more information: Configure PC Settings (https://www.avidcnc.com/support/instructions/software/mach4SoftwareSetup/#3-configure-pc-settings)

Record Mach4 Diagnostic Log

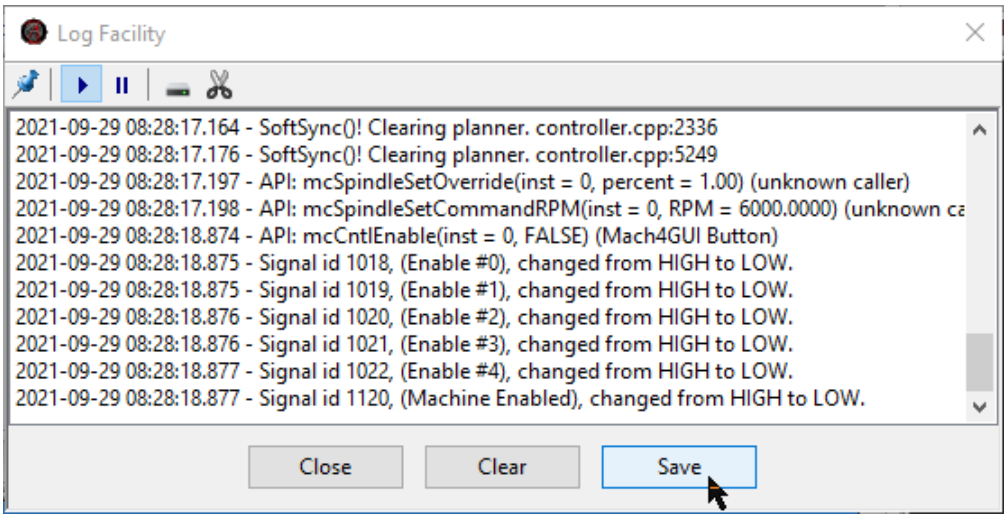
1. In Mach4 go to the **Diagnostic -> Logging** menu.



2. Click the Play button to start logging.



3. Click Save after capturing the relevant events to save the log as a text file. **Note:** Mach4 logs capture many events and can be very long. Try to focus the log on a specific event for easier diagnostics.



Package Mach4 Profile

1. In Mach4 go to the **Configure -> Avid CNC Mach4 Configuration -> About** menu.
2. Choose the "Package Profile" option. Save the resulting .zip file somewhere you can easily find it and attach in an email to **support@avidcnc.com**.