



Mach4 Advanced Configuration Guide

v2021Q4.1

Mach4 Advanced Configuration

These are advanced procedures that can be used to further customize Mach4 and your Avid CNC machine for specific use cases.

Swap X and Y Axes

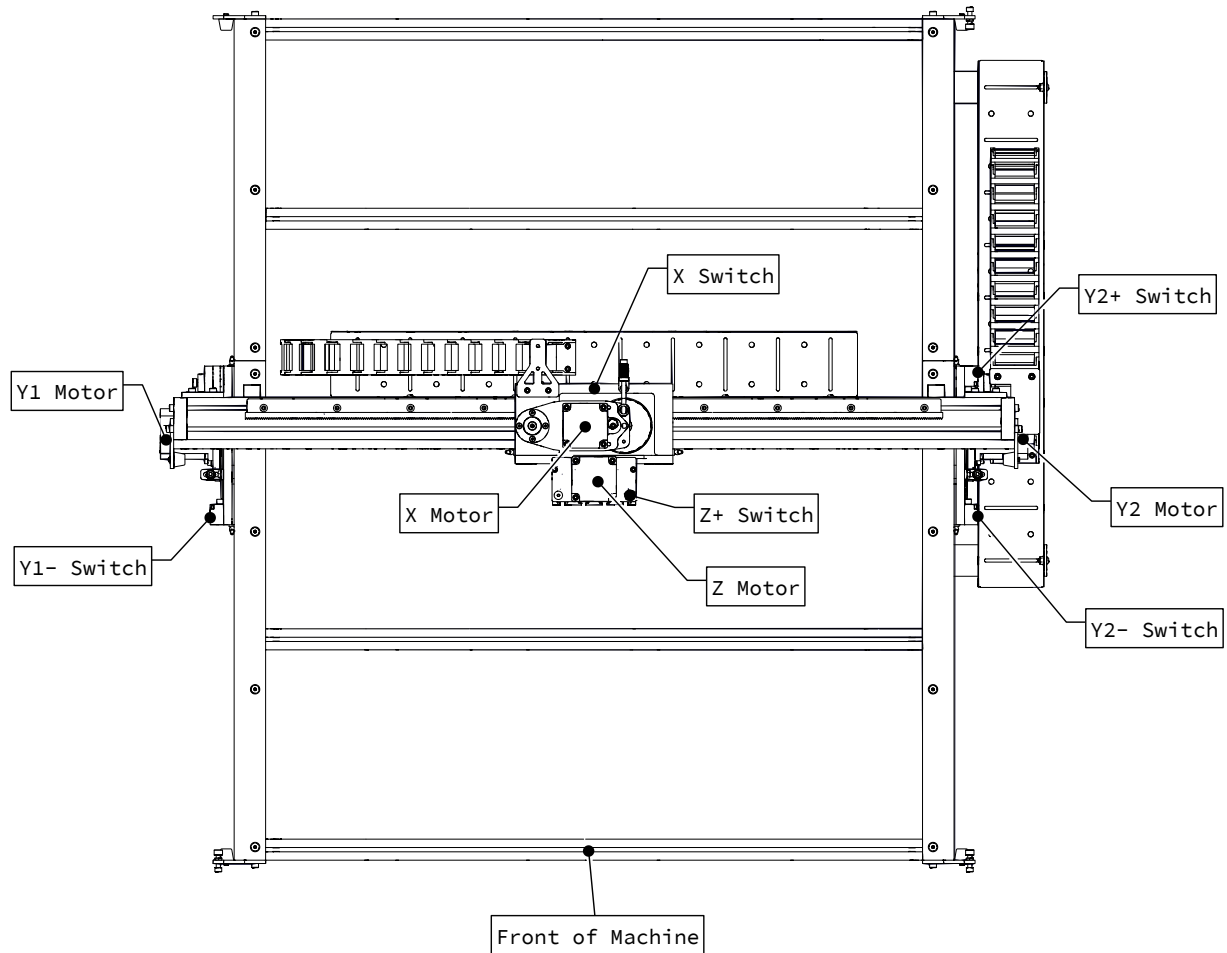
By default all Avid CNC machines are configured with the Gantry (short) axis as X and the Table (long) axis as Y. These axes follow a standard "Right Hand Rule" sign convention. The Avid CNC Mach4 profile does include an option to swap these two axes, but it is important to follow these steps to be successful.

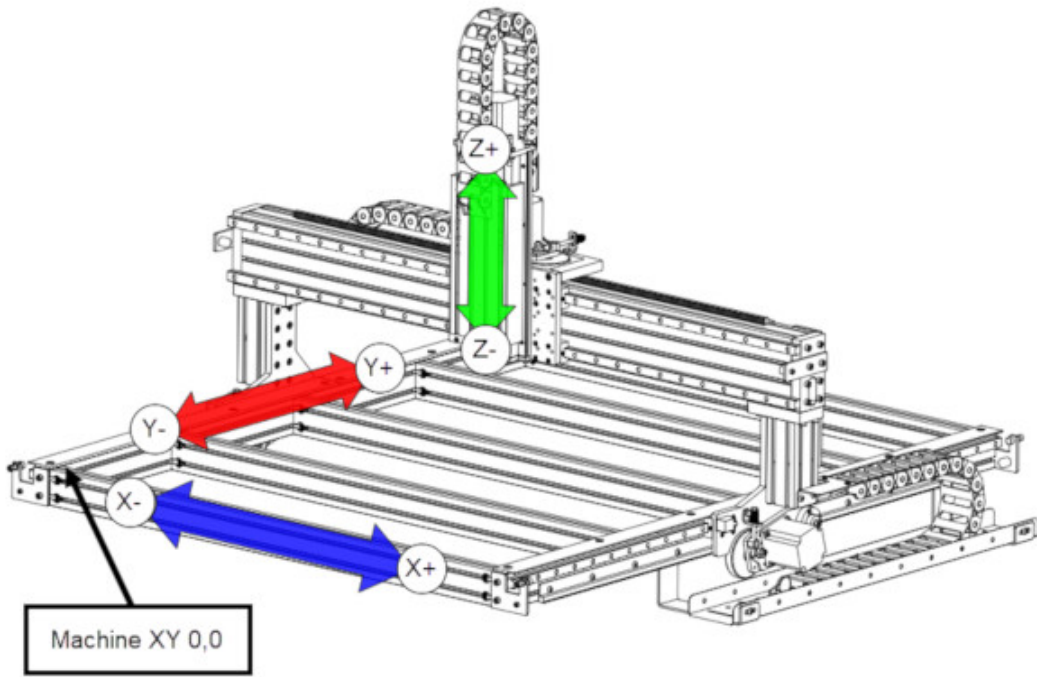
Note

This feature only works properly in Avid CNC Mach4 Profile version 2.0.6 (released 11/2019) or newer. You can check your status on the About tab (<https://www.avidcnc.com/support/instructions/software/mach4Configuration/#6-about>). To upgrade if needed, see our software Mach4 Downloads (<https://www.avidcnc.com/support/instructions/software/downloads/mach4>) page.

1. Initial Setup

Before using this feature, follow all the normal setup instructions for your machine to completion so that the machine homes and moves correctly in the default configuration. This should result in the axis motion and sensor configuration below.

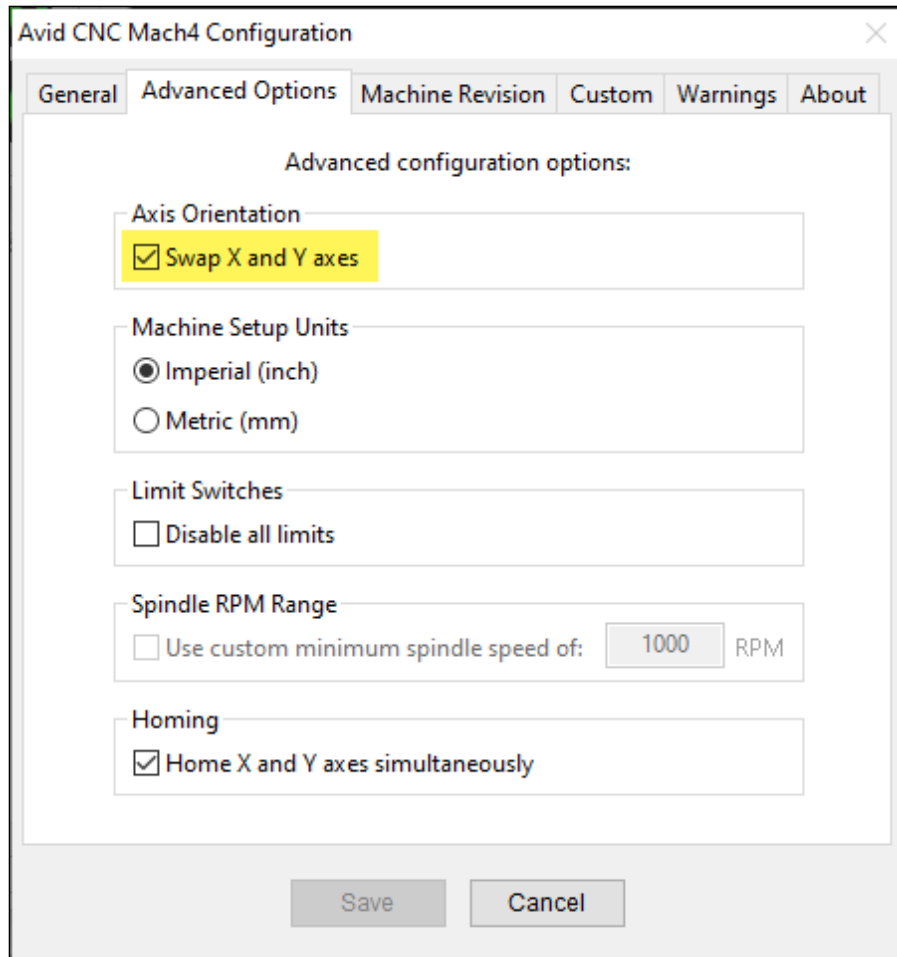




Default Machine Axis Convention

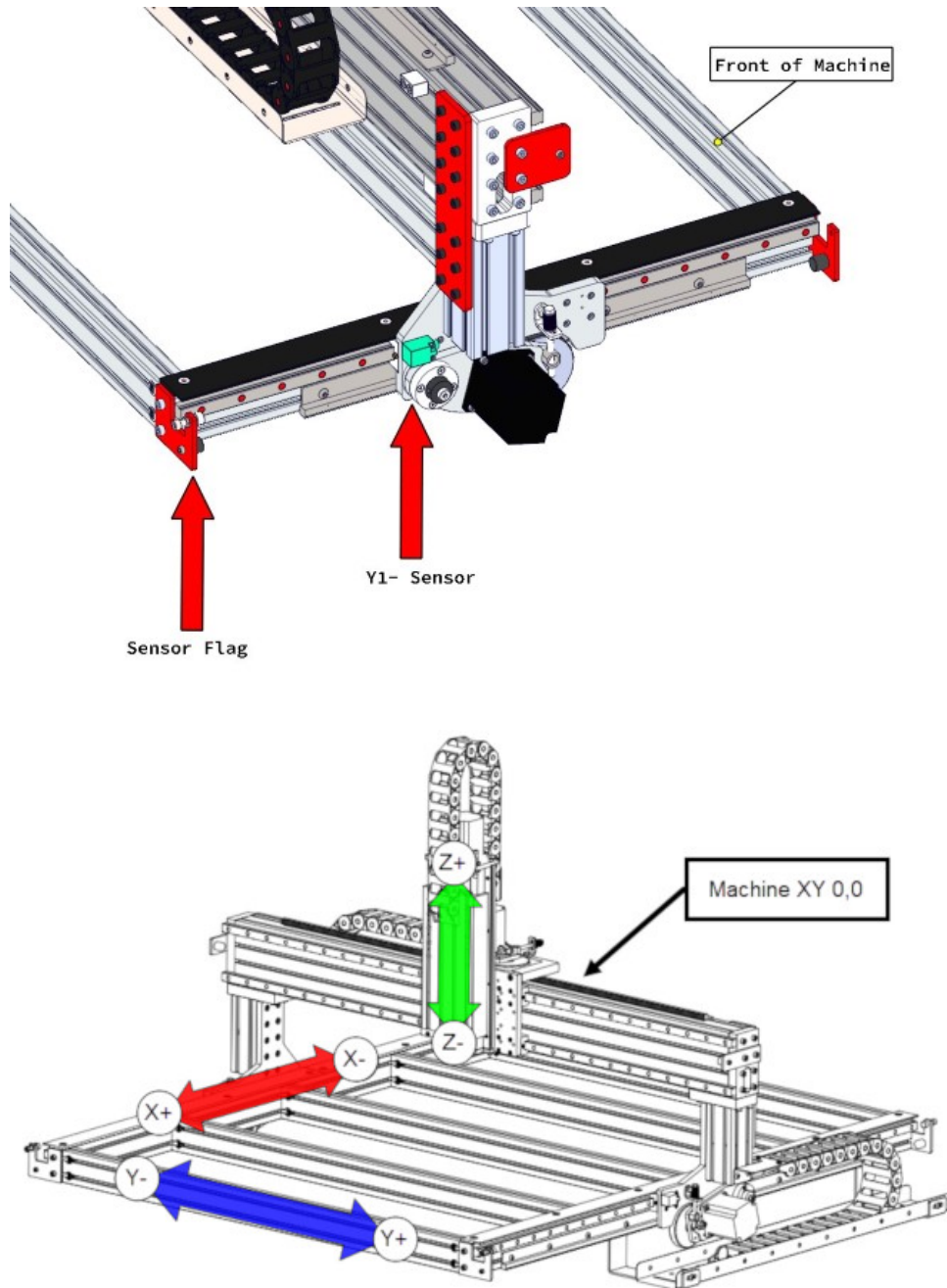
2. Software Changes

In Mach4 go to the **Configure -> Avid CNC Machine Configuration -> Advanced Options** menu to choose the “Swap X and Y axes” option. This change will switch axis assignments in Mach4 and will cause the Table axis to home to the back of the machine.



3. Machine Changes

To accommodate the software changes, you need to physically move the **Y1**- home sensor and sensor flag to the back end of the gantry riser plate and table. The new axis sign convention and necessary sensor change is shown below:



Swapped Machine Axis Convention

Configure Dual Z Machine for Routing and Drilling

The Avid CNC Mach4 profile includes support for Dual Purpose machines that use a spindle or router on one Z axis and a plasma torch on the second. For applications that require a spindle on one Z axis and a router or drill on the other, there are extra setup and operation steps required.

This guide specifically covers the setup of a Dual Z machine with an Avid CNC Plug and Play Spindle/VFD System and a typical AC powered router/drill motor. Some of the steps may apply for other tooling mounted to the second axis.

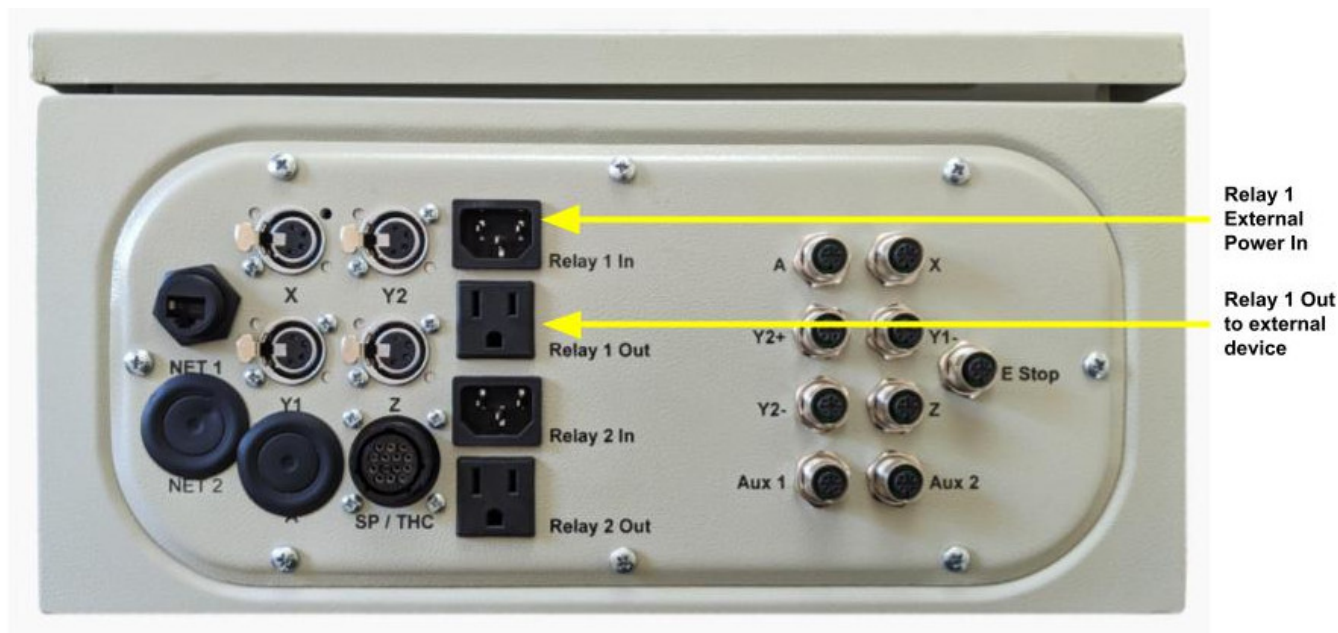
1. Physical Setup of the Cutting Tools

Install your spindle according to our standard instructions, and install your router/drill in place of the plasma torch.

The two axes will be referred to in these instructions as follows:

Cutting Tool	Axis Name	Mach4 Motor Number
Spindle	Z	2
Router/Drill	U	4

- Connect the VFD enclosure to the CRP800-00E controller with the 14-pin cable (see Plug & Play Spindle CRP800 Setup (<https://www.avidcnc.com/support/instructions/accessories/spindles/setup/CRP800setup/>) for details).
- Plug the router/drill into **Relay 1 Out** and the necessary power (110V-250V AC) into **Relay 1 In** on the CRP800 controller.

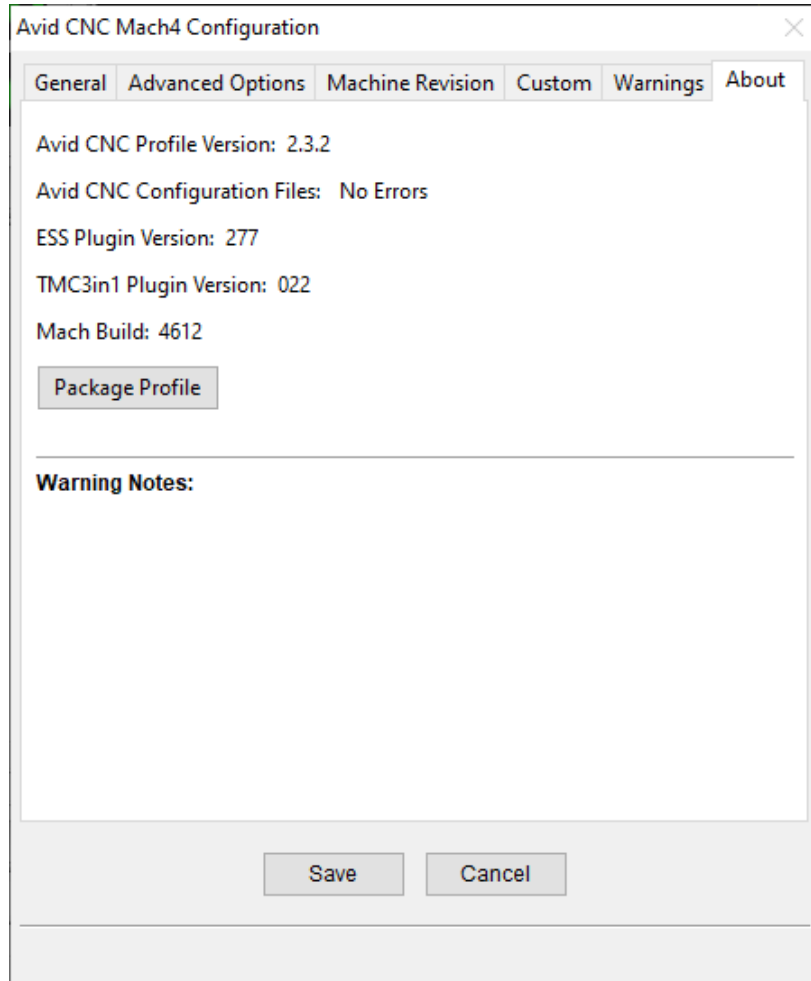


2. Mach4 Configuration

2.1 Mach4 Version

These instructions require that you have Avid CNC Mach4 Profile Version 2.1.0 or higher installed. The latest available version is recommended.

- To check your version within Mach4 go to the **Configure -> Avid CNC Mach4 Configuration -> About** tab.

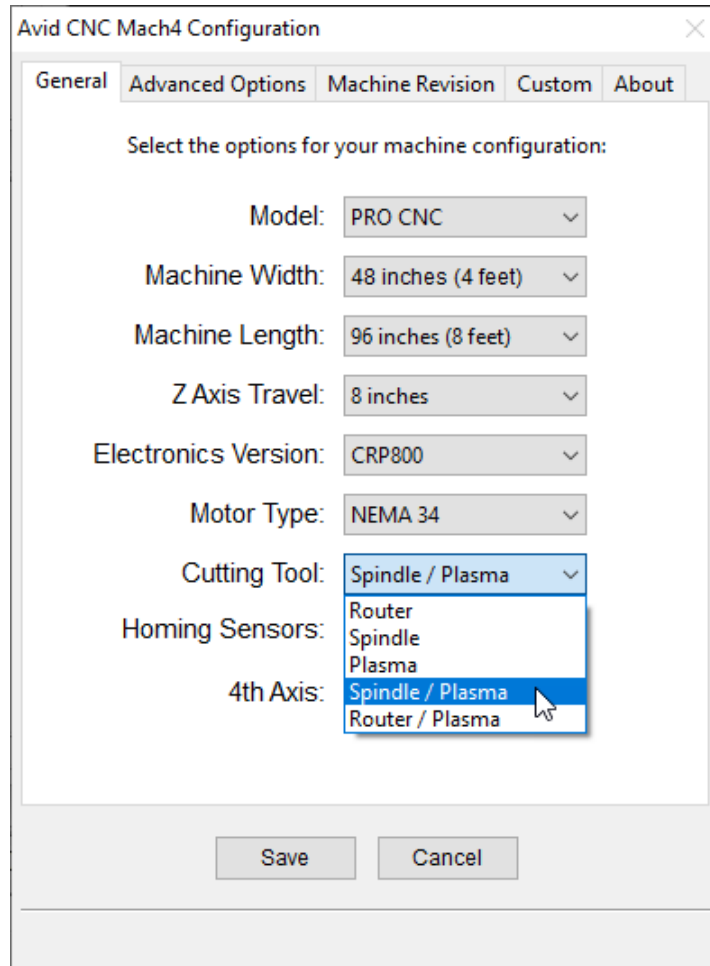


- If an update is needed, go to Mach4 for Avid CNC Machines (<https://www.avidcnc.com/support/software/downloads/mach4/>) and download/run the latest installer.

2.2 Avid CNC Mach4 Configuration Menu

In the **Avid CNC Mach4 Configuration** menu:

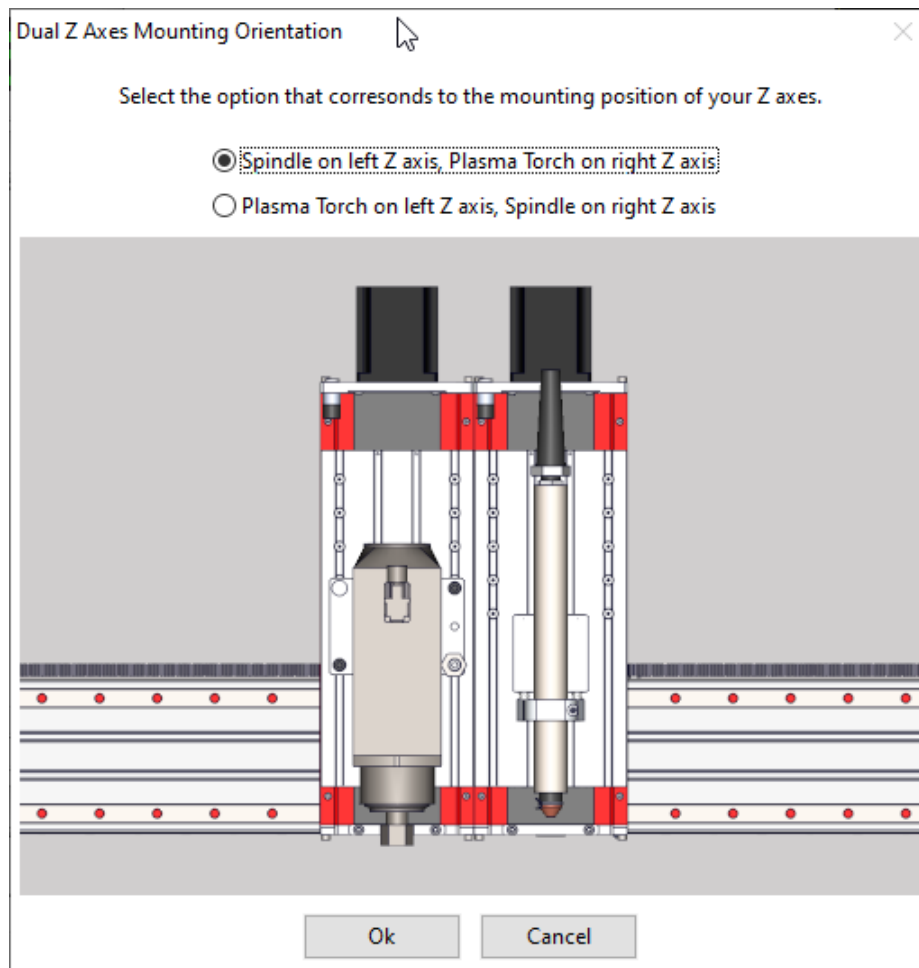
- Set the cutting tool to **Spindle/Plasma** on the *General* tab. This will enable and map the second vertical axis as the U axis motor.



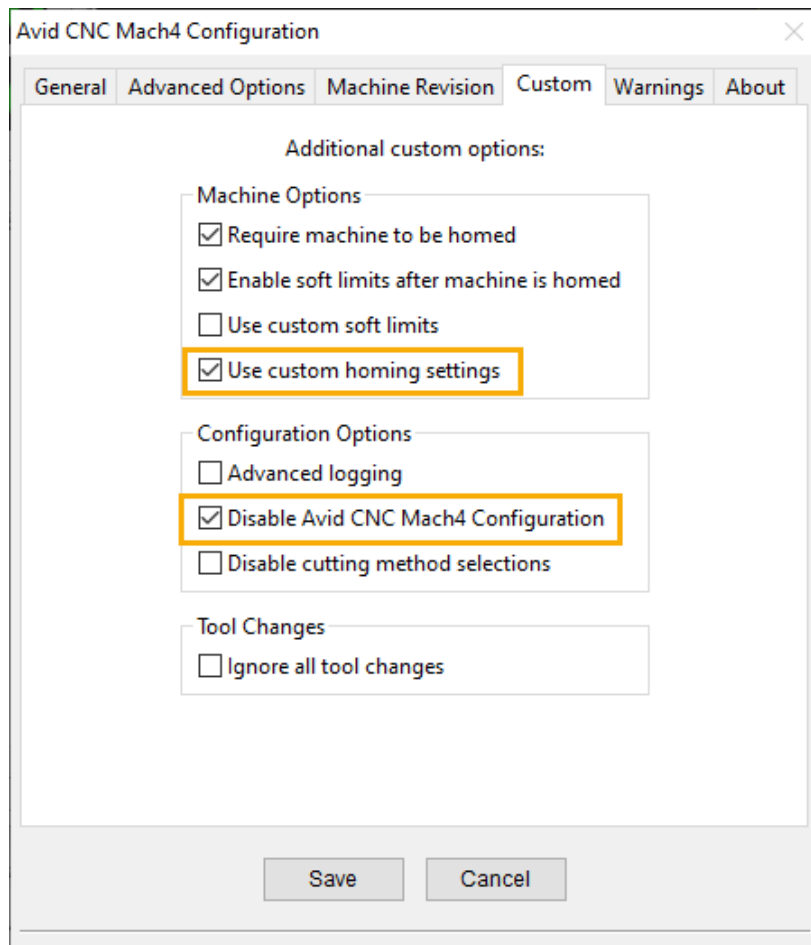
The screenshot shows the 'Avid CNC Mach4 Configuration' dialog box with the 'General' tab selected. The dialog contains several configuration options, each with a dropdown menu. The 'Cutting Tool' dropdown is currently open, showing 'Spindle / Plasma' as the selected option. The '4th Axis' dropdown is also open, showing 'Spindle / Plasma' as the selected option. The 'Save' and 'Cancel' buttons are visible at the bottom of the dialog.

Option	Value
Model	PRO CNC
Machine Width	48 inches (4 feet)
Machine Length	96 inches (8 feet)
Z Axis Travel	8 inches
Electronics Version	CRP800
Motor Type	NEMA 34
Cutting Tool	Spindle / Plasma
Homing Sensors	Router Spindle Plasma
4th Axis	Spindle / Plasma Router / Plasma

- Choose the correct side (left/right) for the spindle.

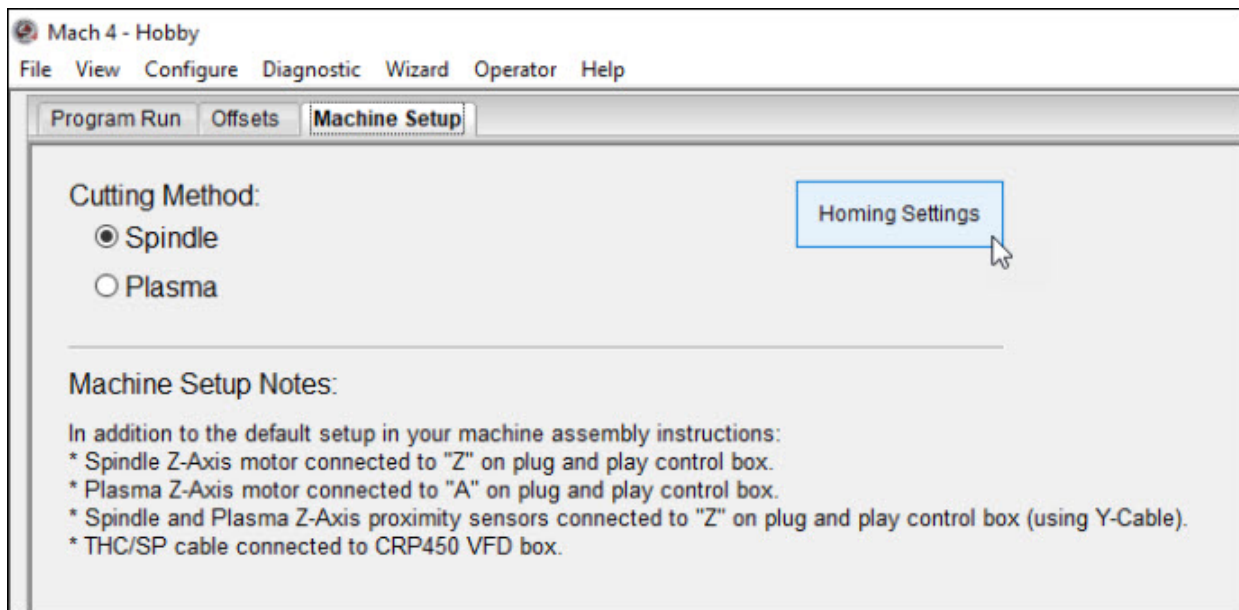


- **Save** the configuration settings at this point to lock the machine configuration.
- Navigate back to the **Avid CNC Mach4 Configuration** menu. On the *Custom* tab check the **Use Custom Homing Settings** and **Disable Avid CNC Mach4 Configuration** options.

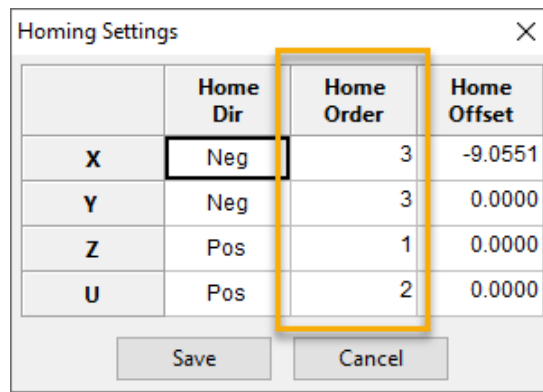


2.3 Cutting Method Configuration

- On the *Cutting Methods* tab of the main Mach4 screen, choose **Spindle** as the cutting tool.



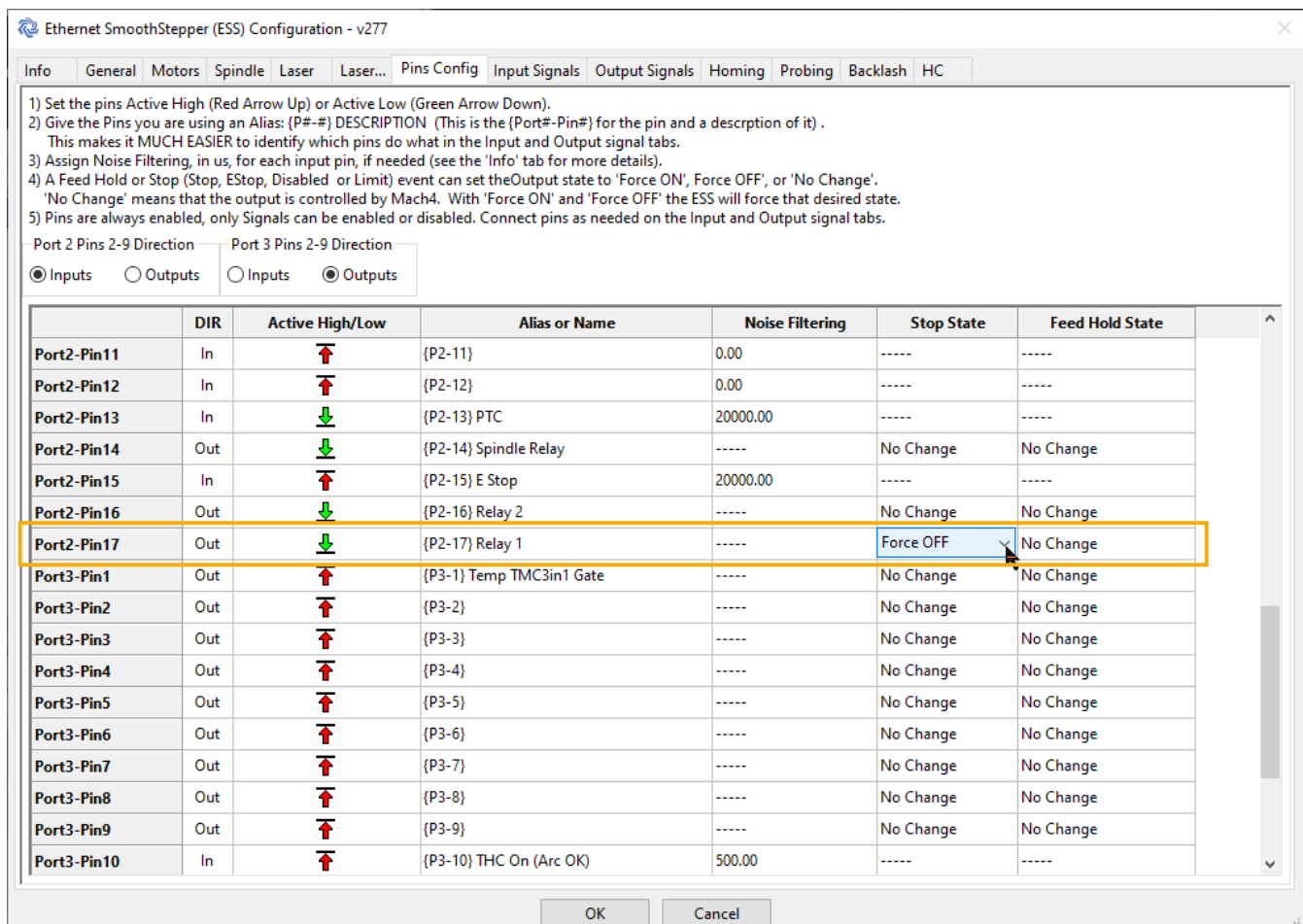
- In the **Custom Homing Settings** menu set the homing order (X=3, Y=3, Z=1, U=2) as shown below.



2.4 ESS Configuration Changes

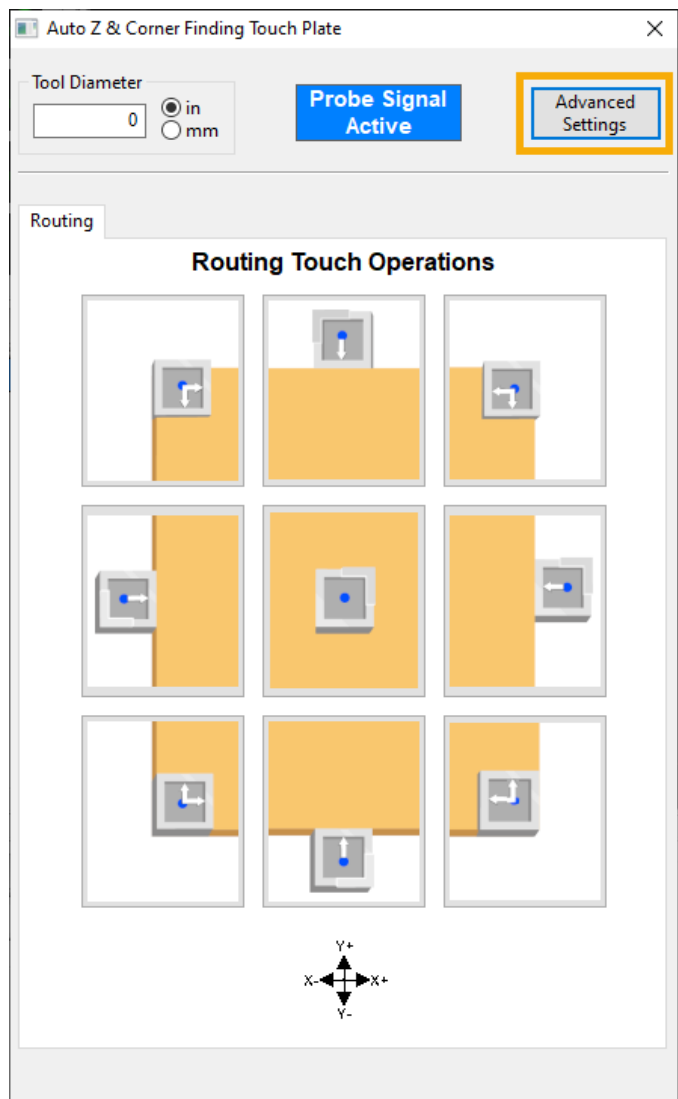
Because the Relay 1 output doesn't normally get turned off by the on-screen "Stop" button, that behavior needs to be set.

- Navigate to **Configure -> Plugins -> ESS**. On the *Pins Config* tab scroll down to find Port2-Pin17. Set the *Stop State* to **Force OFF**. Restart Mach4 for the change to take effect.

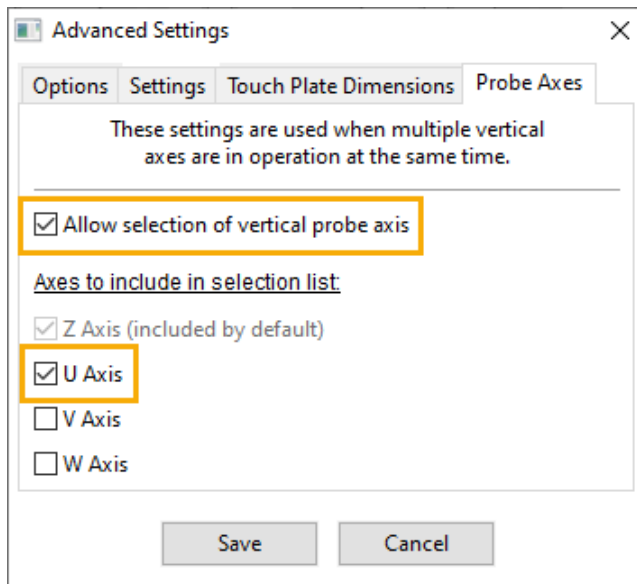


2.5 Auto Z Touch Plate Configuration

- Open the **Auto Z Touch Plate** window and navigate to *Advanced Settings*.



- On the *Probe Axes* tab check the **Allow selection of vertical probe axis** and **U Axis** options. This will allow the second vertical axis to use the touchplate for setting work offsets.



3. G-Code Modifications

We recommend using separate G-Code files for each cutting tool. This makes it easier to ensure that the correct axis and cutting tool will operate.

If you choose to combine the spindle and router/drill toolpaths in a single G-Code file, be careful to only edit the correct toolpath in the G-Code.

In order to control the second cutting tool (router/drill) correctly you will need to either make the manual changes to the G-Code outlined below, or modify a post processor to make these changes automatically.

Info

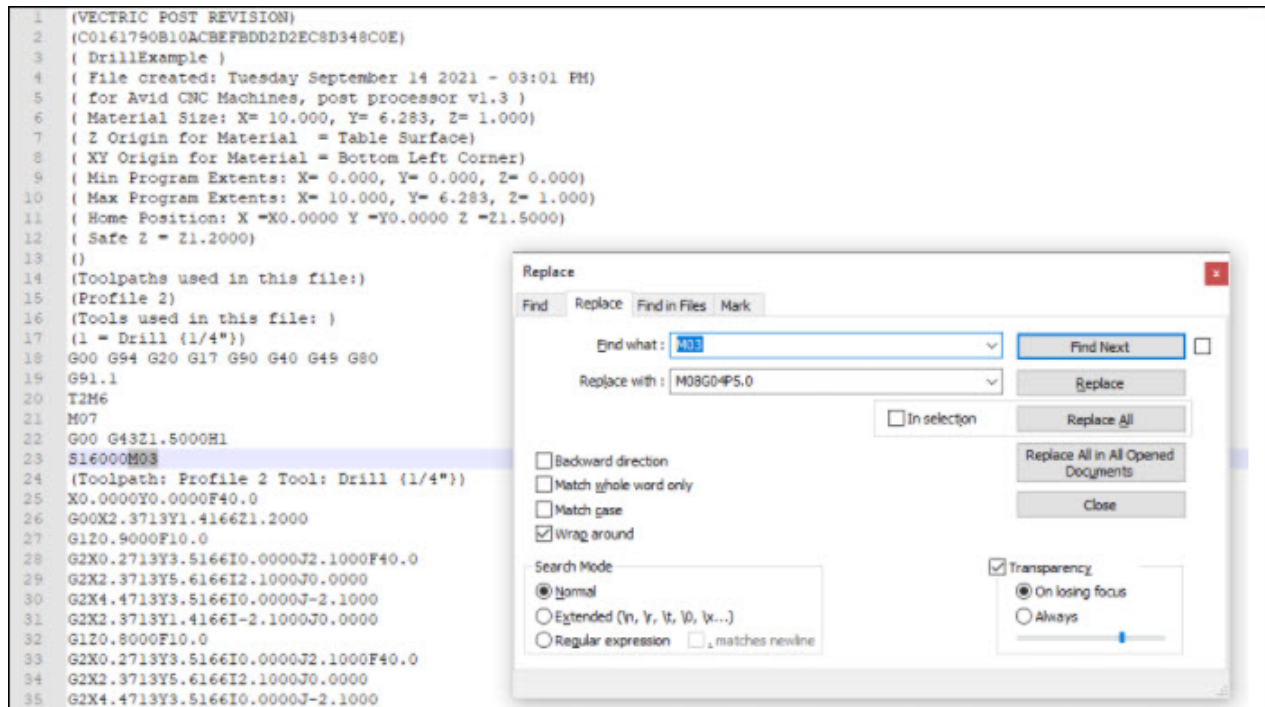
These changes can be made by post processing the toolpath using the standard Avid Mach4 post processors in your chosen CAD/CAM program, then opening the G-Code in a text editor like Notepad++ (<https://notepad-plus-plus.org/>) to make the edits using "Find/Replace".

3.1 Find/Replace

Make the Find/Replace changes below for the router/drill G-Code or toolpath only:

- **Find** *M03* (or *M3*) and **Replace** with *M08G04P5.0*

This replaces the standard *Spindle On* command with a command to turn on Relay 1 and pause for 5 seconds before continuing the program to allow for the RPM to stabilize.



The image shows a screenshot of a G-code file in a text editor. The G-code starts with a revision header and includes toolpath information for a drill. Line 23 contains the command `S16000M03`, which is highlighted. A 'Replace' dialog box is overlaid on the right side of the editor. The 'Find what' field contains 'M03' and the 'Replace with' field contains 'M08G04P5.0'. The 'Find Next' button is highlighted. The dialog also includes options for 'In selection', 'Backward direction', 'Match whole word only', 'Match case', 'Wrap around', 'Search Mode' (Normal, Extended, Regular expression), and 'Transparency' (On losing focus, Always).

- Find M05 (or M5) and Replace with M09G04P5.0

This replaces the standard *Spindle Off* command with a command to turn off Relays 1 and 2 and pause for 5 seconds before continuing the program to allow for the RPM to drop to zero.

```

104 G1Z0.4000F10.0
105 G2X4.8313Y5.3916I2.1000J0.0000F40.0
106 G2X6.9313Y3.2916I0.0000J-2.1000
107 G2X4.8313Y1.1916I-2.1000J0.0000
108 G2X2.7313Y3.2916I0.0000J2.1000
109 G1Z0.3000F10.0
110 G2X4.8313Y5.3916I2.1000J0.0000F40.0
111 G2X6.9313Y3.2916I0.0000J-2.1000
112 G2X4.8313Y1.1916I-2.1000J0.0000
113 G2X2.7313Y3.2916I0.0000J2.1000
114 G1Z0.2000F10.0
115 G2X4.8313Y5.3916I2.1000J0.0000F40.0
116 G2X6.9313Y3.2916I0.0000J-2.1000
117 G2X4.8313Y1.1916I-2.1000J0.0000
118 G2X2.7313Y3.2916I0.0000J2.1000
119 G1Z0.1000F10.0
120 G2X4.8313Y5.3916I2.1000J0.0000F40.0
121 G2X6.9313Y3.2916I0.0000J-2.1000
122 G2X4.8313Y1.1916I-2.1000J0.0000
123 G2X2.7313Y3.2916I0.0000J2.1000
124 G1Z0.0000F10.0
125 G2X4.8313Y5.3916I2.1000J0.0000F40.0
126 G2X6.9313Y3.2916I0.0000J-2.1000
127 G2X4.8313Y1.1916I-2.1000J0.0000
128 G2X2.7313Y3.2916I0.0000J2.1000
129 G00Z1.2000
130 G00Z1.5000
131 G00X0.0000Y0.0000
132 M05
133 M09
134 M30
135 %
136

```

- Find Z and Replace with U

```

1 (VECTRIC POST REVISION)
2 (C0161790B10ACBEFDD2D2EC8D348C0E)
3 ( DrillExample )
4 ( File created: Tuesday September 14 2021 - 03:01 PM)
5 ( For Avid CNC Machines, post processor v1.3 )
6 ( Material SiDe: X= 10.000, Y= 6.283, U= 1.000)
7 ( U Origin for Material = Table Surface)
8 ( XY Origin for Material = Bottom Left Corner)
9 ( Min Program Extents: X= 0.000, Y= 0.000, U= 0.000)
10 ( Max Program Extents: X= 10.000, Y= 6.283, U= 1.000)
11 ( Home Position: X=X0.0000 Y=Y0.0000 U=U1.5000)
12 ( Safe U = U1.2000)
13 ()
14 (Toolpaths used in this file:)
15 (Profile 2)
16 (Tools used in this file: )
17 (I = Drill (1/4"))
18 G00 G94 G20 G17 G90 G40 G49 G80
19 G91.1
20 T2M6
21 M07
22 G00 G43U1.5000H1
23 S1600M08G04P5.0
24 (Toolpath: Profile 2 Tool: Drill (1/4"))
25 X0.0000Y0.0000F40.0
26 G00X2.3713Y1.4166I1.2000
27 G1D0.9000F10.0
28 G2X0.2713Y3.5166I0.0000J2.1000F40.0
29 G2X2.3713Y5.6166I2.1000J0.0000
30 G2X4.4713Y3.5166I0.0000J-2.1000
31 G2X2.3713Y1.4166I-2.1000J0.0000
32 G1D0.8000F10.0
33 G2X0.2713Y3.5166I0.0000J2.1000F40.0
34 G2X2.3713Y5.6166I2.1000J0.0000
35 G2X4.4713Y3.5166I0.0000J-2.1000
36 G2X2.3713Y1.4166I-2.1000J0.0000
37 G1D0.7000F10.0

```


3.2 Spindle Work Offset

- Find the *Safe Start* line(s) at the beginning of the spindle G-Code. This line is a series of modal codes that set the proper state of the controller.
- Add **G54** to this line.

This sets the active work offset for this G-Code to G54, which will get set using the Auto Z Touch Plate and the Z axis.

```
1 (VECTRIC POST REVISION)
2 (C0161790B10ACBEFBDD2D2EC8D348C0E)
3 ( SpindleExample )
4 ( File created: Tuesday September 14 2021 - 03:01 PM)
5 ( for Avid CNC Machines, post processor v1.3 )
6 ( Material Size: X= 10.000, Y= 6.283, Z= 1.000)
7 ( Z Origin for Material = Table Surface)
8 ( XY Origin for Material = Bottom Left Corner)
9 ( Min Program Extents: X= 0.000, Y= 0.000, Z= 0.000)
10 ( Max Program Extents: X= 10.000, Y= 6.283, Z= 1.000)
11 ( Home Position: X =X0.0000 Y =Y0.0000 Z =Z1.5000)
12 ( Safe Z = Z1.2000)
13 ( )
14 (Toolpaths used in this file:)
15 (Profile 1)
16 (Tools used in this file: )
17 (1 = End Mill {1/4"})
18 G00 G94 G20 G17 G90 G40 G49 G80 G54
19 G91.1
20 T1M6
21 M07
22 G00 G43Z1.5000H1
23 S16000M03
24 (Toolpath: Profile 1 Tool: End Mill {1/4"})
25 X0.0000Y0.0000F40.0
26 G00X2.3713Y1.4166Z1.2000
27 G1Z0.9000F10.0
28 G2X0.2713Y3.5166I0.0000J2.1000F40.0
29 G3Y2.3713X5.6166I2.1000J0.0000
```

3.3 Router/Drill Work Offset

- Find the *Safe Start* line(s) at the beginning of the router/drill G-Code. This line is a series of modal codes that set the proper state of the controller.
- Add **G55** to this line.

This sets the active work offset for this G-Code to G55, which will get set using the Auto Z Touch Plate and the U axis.

```
1 (VECTRIC POST REVISION)
2 (C0161790B10ACBEFBDD2D2EC8D348C0E)
3 ( DrillExample )
4 ( File created: Tuesday September 14 2021 - 03:01 PM)
5 ( for Avid CNC Machines, post processor v1.3 )
6 ( Material SiUe: X= 10.000, Y= 6.283, U= 1.000)
7 ( U Origin for Material = Table Surface)
8 ( XY Origin for Material = Bottom Left Corner)
9 ( Min Program Extents: X= 0.000, Y= 0.000, U= 0.000)
10 ( Max Program Extents: X= 10.000, Y= 6.283, U= 1.000)
11 ( Home Position: X =X0.0000 Y =Y0.0000 U =U1.5000)
12 ( Safe U = U1.2000)
13 ( )
14 (Toolpaths used in this file:)
15 (Profile 2)
16 (Tools used in this file: )
17 (1 = Drill {1/4"})
18 G00 G94 G20 G17 G90 G40 G49 G80 G55
19 G91.1
20 T2M6
21 M07
22 G00 G43U1.5000H1
23 S1600M08G04P5.0
24 (Toolpath: Profile 2 Tool: Drill {1/4"})
25 X0.0000Y0.0000F40.0
26 G00X2.3713Y1.4166U1.2000
27 G1U0.9000F10.0
28 G2X0.2713Y3.5166I0.0000J2.1000F40.0
29 G2X2.3713Y5.6166I2.1000J0.0000
```

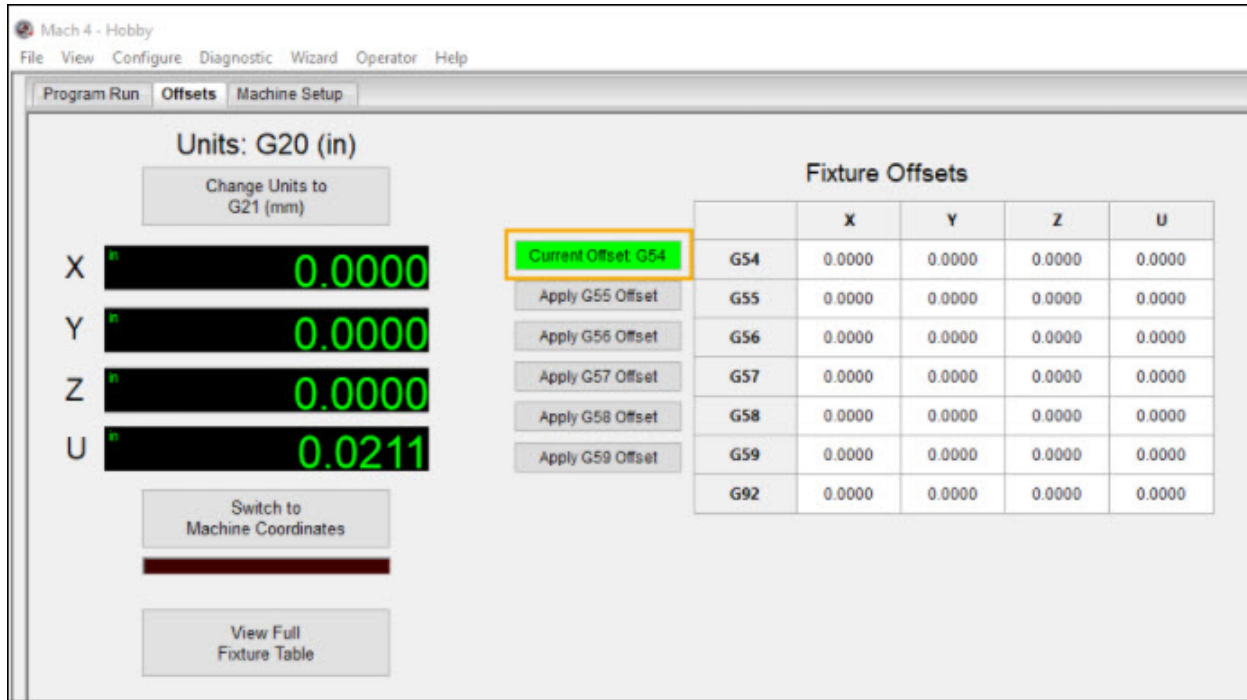
Warning

If you post processed the spindle and router/drill toolpaths in a single file you will need to add **G54** at the start of each spindle toolpath and **G55** at the start of each router/drill toolpath.

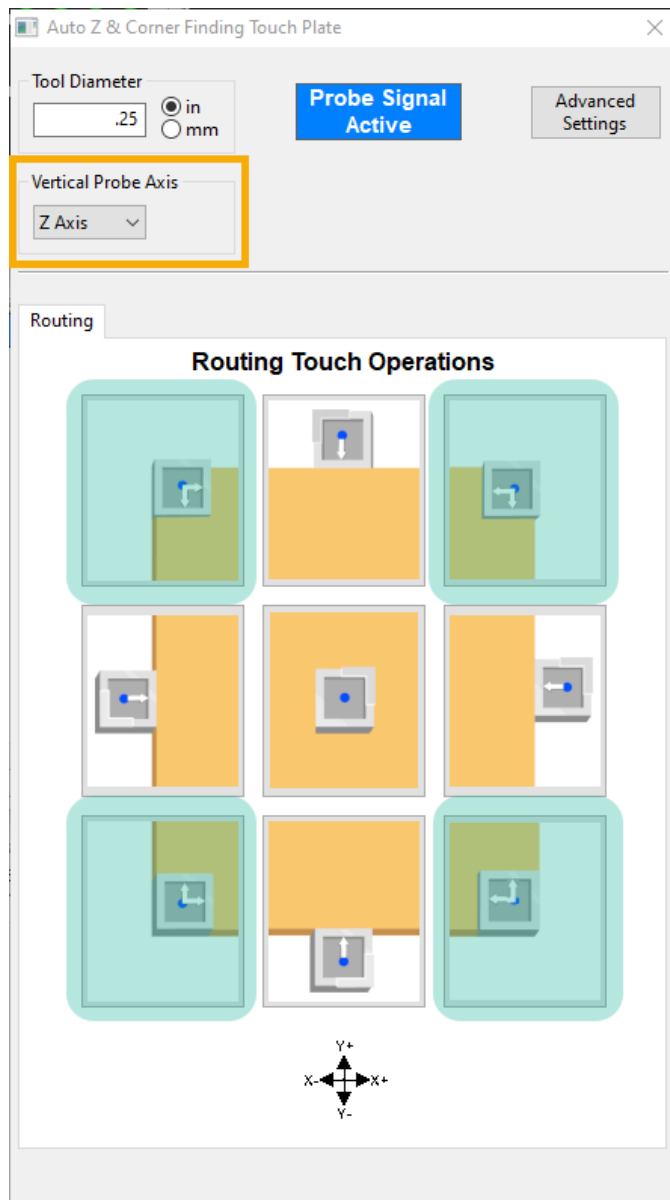
4. Running the Programs

4.1 Probe the Spindle Tool (Z axis)

- On the *Offsets* tab in Mach4, select **G54** as the active work offset.



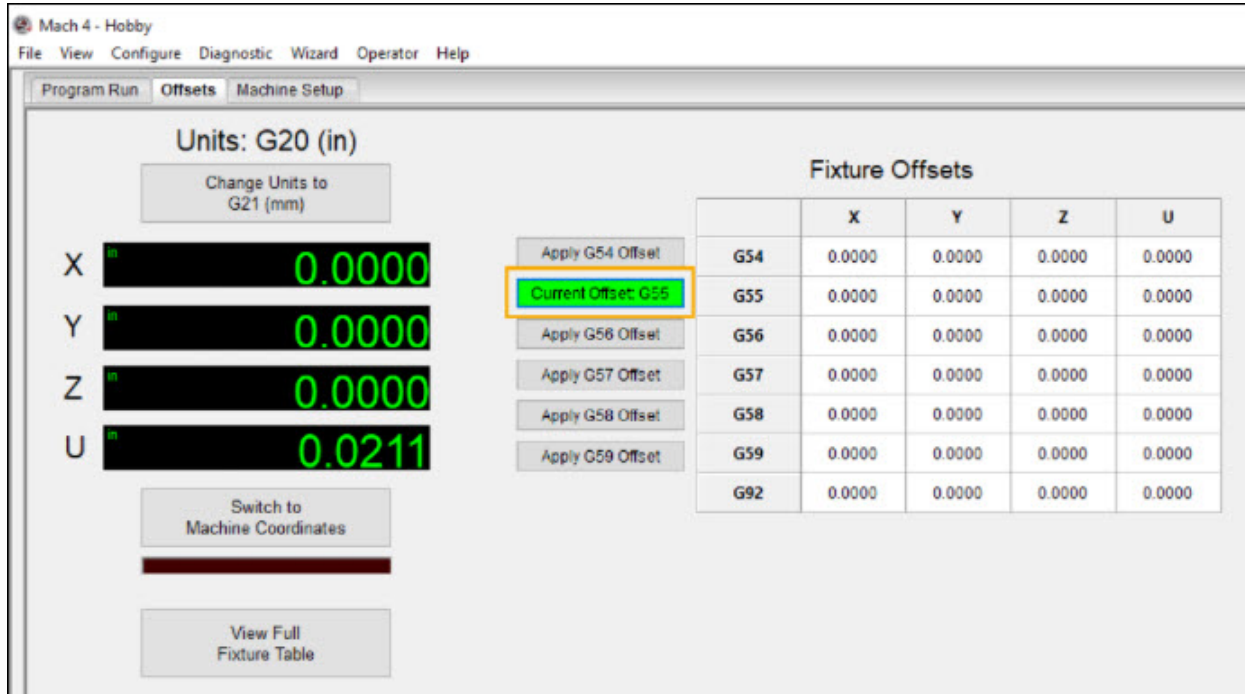
- Open the Auto-Z Touch Plate window. Choose the **Z** as the axis to probe.



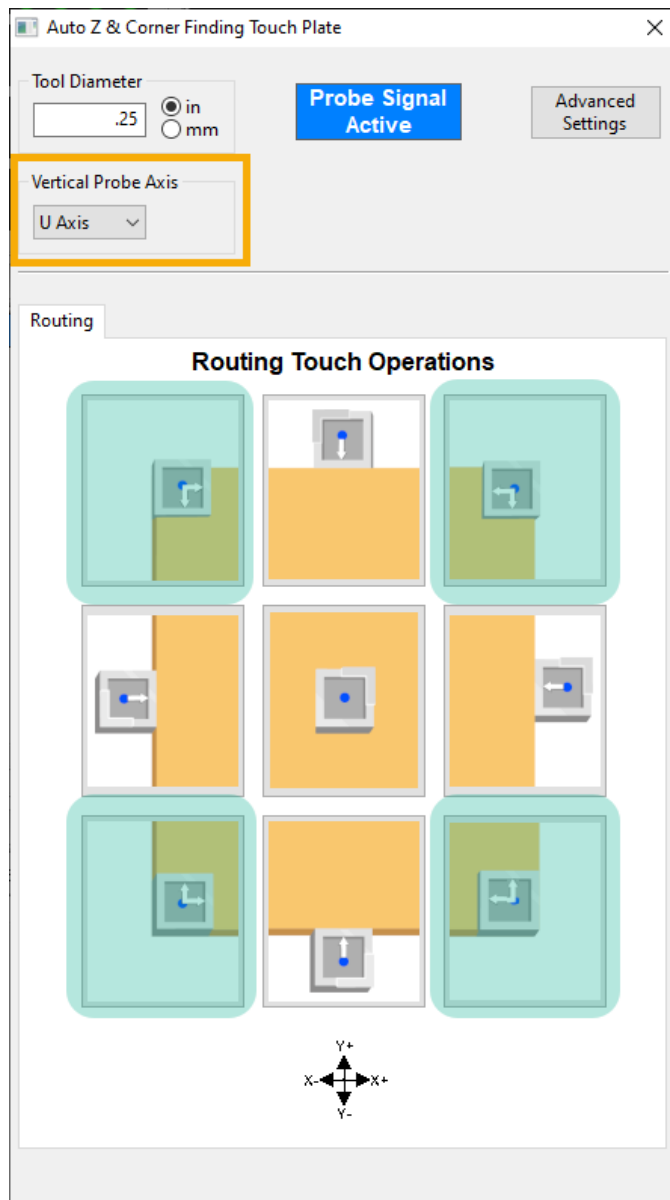
- Jog the spindle tool over the touch plate use it as normal. Probe one of the corners to set the G54 XYZ offsets.

4.2 Probe the Router/Drill Tool (U Axis)

- On the *Offsets* tab in Mach4, select **G55** as the active work offset.



- Open the Auto-Z Touch Plate window. Choose the **U** as the axis to probe.



- Jog the router/drill tool over the touch plate and use it as normal. Probe one of the corners to set the G55 XYU offsets.

i Info

The **U** axis can be jogged using the bracket keys, [(up) and] (down).

4.3 Program Run

The spindle and router/drill G-Code programs can now be run properly. The G-Code changes made in these instructions will make sure that the correct work offsets and axes are used.