Frequently Asked Questions

[Is the Triquetra compatible with Easel? 2](#_Toc450544995)

[What about Mach 3/4 Users? 2](#_Toc450544996)

[When I run the G-Code the bit just pushes my touch plate until it errors out. What causes this? 3](#_Toc450544997)

[I saved my g-code to a Microsoft Word document and can't load it in to my Post Processor software. What am I doing wrong? 3](#_Toc450544998)

[What file extension should I use for my g-code files? 4](#_Toc450544999)

[I changed my bit mid project. How do I re-zero just my Z-Axis? 4](#_Toc450545000)

[Can I set my X and Y axis to be in the center of my material to be carved? 4](#_Toc450545001)

[When I run the zeroing g-code my bit is not zeroed accurately. What can I do to correct this? 5](#_Toc450545002)

[I have a variety of different bits. Do I need to create a g-code file for all of them? 5](#_Toc450545003)

[Where should I position my bit before I run the zero g-code. 5](#_Toc450545004)

[What is the minimum thickness my material must be to use the Triquetra? 6](#_Toc450545005)

# Is the Triquetra compatible with Easel?

Answer: The short answer is no, however there is a work around that will allow you to use the Triquetra to achieve the same level of accuracy in setting all three axis to zero and still use Easel to carve your projects. Initially this looks like a lot of extra steps but after you have done it one time you will find that it is really quite simple. This requires the use of a Post Processor such as Universal Gcode Sender (UGS) or PicSender. Please follow these steps:

1. Load your external Post Processor such as UGS or PicSender.
2. Position your Touch Plate on the front left corner of the material to be carved, plug in the banana plug, and place the alligator clip on the bit.
3. Load the appropriate g-code file for the bit size you have installed in your spindle and run the file to zero your bit.
4. Remove the touch plate and wires.
5. In PicSender click the "Goto Zero" buttons for the X, Y, and Z axis. Be sure to click the Z axis button last. In UGS manually jog you machine until the "Work Position" readings on the "Machine Control" tab shows zero for all three axis.
6. Close or Exit your post processor software once your bit is located at X0, Y0, Z0.
7. Load Easel
8. Skip the homing sequence in Easel, and confirm that your bit is a 0 for all three axis when asked. Then continue with Easel the way you normally would.

# What about Mach 3/4 Users?

Answer: Support for Mach 3 is now included with the G-Code Generator Version 7 and later. Version 7 is the first release with Mach 3 Support. Due to the vast variety of CNC Machines, there is no way for us to guarantee compatibility. Use at your own risk.

# When I run the G-Code the bit just pushes my touch plate until it errors out. What causes this?

Answer: This can be cause by a couple of things.

1. The number one cause for new installations is a bad ground. The most common mistake is connecting the ground wire to the incorrect spot on the power supply. Please look at the wiring diagram or watch the "Touch Plate Wiring for CNC Routers" video that are available on the Triquetra User Page.
2. Check that the positive wire that goes to you touch plate is firmly plugged in to the Arduino Board at pin A5
3. Check the voltage output. With your power supply turned on and using a multi-meter, place the positive probe on the touch plate and the negative probe on the alligator clip. You should be reading approximately 4.8 volts. If not then re-check your wire connections.
4. Varnished or Burnt bit. Depending on the material you have been cutting with your bit, sometimes it can build up a layer of varnish or become dirty. This will prevent the bit from making an electrical connection with the touch plate. Clean your bit with a solvent or use another bit.
5. Improper household wiring. Your system must be connected to a properly wired power outlet that includes a ground wire. A power outlet that only has two slots and no ground pin will cause errors and is a shock hazard.

# I saved my g-code to a Microsoft Word document and can't load it in to my Post Processor software. What am I doing wrong?

Answer: Full featured word processors typically do not save files in a purely text format. They normally have lots of formatting codes that are not compatible with any CNC machine. To avoid this, always use a ASCI type editor such as Windows Notepad to save your g-code files.

# What file extension should I use for my g-code files?

Answer: Some post processors look for files that have a specific file extension such as .gcode. PicSender and Universal G-code Sender will accept the Windows default file extension used by Note Pad of .txt.

# I changed my bit mid project. How do I re-zero just my Z-Axis?

Answer: If your Post Processor allows for a mid project bit change such as the new version of Easel and then continues to run the same g-code file after the bit change then you will have to manually zero your z axis. However, if the bit change happens after the completion of a g-code file and before loading a new g-code file for the next tool path file then The G-Code Generator Spreadsheet has a tab for "Z Axis Only" option that will allow you to re-zero your Z axis only.

1. Using the G-Code Generator, click on the Z Axis Only tab. You do not need to enter any information because it is pre-populated.
2. Insure that you have the correct Operating Units selected in the blue box.
3. Copy the G-Code in the Yellow Box to a Note Pad file.
4. Place your Touch Plate upside down at any location you prefer.
5. Manually Jog your machine so that the bit is over the milled out portion of the touch plate.
6. Connect the banana plug to the touch plate and the alligator clip to the bit.
7. Load the Z axis only file you created and run it.
8. Done.

# Can I set my X and Y axis to be in the center of my material to be carved?

Answer: Yes. Starting in Version 6a5, the G-Code Generator has the ability to zero your bit and includes parameters to offset it to any location you want on the material being carved. The touch plate will still be used at the front left corner as always but after zeroing the zero positions will be offset from that corner.

# When I run the zeroing g-code my bit is not zeroed accurately. What can I do to correct this?

Answer: Your spreadsheet almost always needs an initial calibration when first setup to match your particular machine. This is due to improper stepper calibration, rough measuring of the touch plate, loose belts and other reasons. This can easily be corrected by Fine Tuning your spreadsheet. On the G-Code Generator Spreadsheet there is a "Fine Tuning" page. Please download and print the "Fine Tuning Worksheet" and watch the "Fine Tuning a Detailed Walk Through" video on the Triquetra Users Page for instructions. There is also a "Stepper Calibration" video that will guide you through using the "Stepper Calibration" page on the G-Code Generator Spreadsheet. If you change your stepper calibration you will need to fine tune your machine to match the new stepper calibration.

# I have a variety of different bits. Do I need to create a g-code file for all of them?

Answer: No. The only g-code files you need to create are files for each bit size in diameter. Bit size refers to the maximum diameter of your bit. This is normally the shank size but not always. For example, you have the following bits: 1/16" ball nose with a 1/8" shank, 1/8" endmill with a 1/8" shank, and 1/8" V-bit with a 1/8" shank. Only 1 file is needed to zero all of these bits because each of them have a 1/8" shank and it is the maximum diameter of each bit.

# Where should I position my bit before I run the zero g-code.

Answer: The g-code is written so that the maximum distance it will travel searching for the touch plate is 1 inch. This is to prevent it from continuing to search until it crashes into your limits. Your initial bit position should always be within 1/2" to the right of the touch plate, within 1/2" of the back right corner, and the tip of your bit should be as close to the work surface as possible without actually touching it. Please be aware that during the zeroing process the bit will automatically raise up 1" before it moves over the top of the touch plate to measure the Z axis. You must insure that there is sufficient travel in you Z axis available for this movement. The best way to do this is to position your bit as low as possible before starting the zeroing process.

# What is the minimum thickness my material must be to use the Triquetra?

Answer: The stock version of the Triquetra will work with material as thin as 0.22 inches. Special orders can be made for use with thinner material.