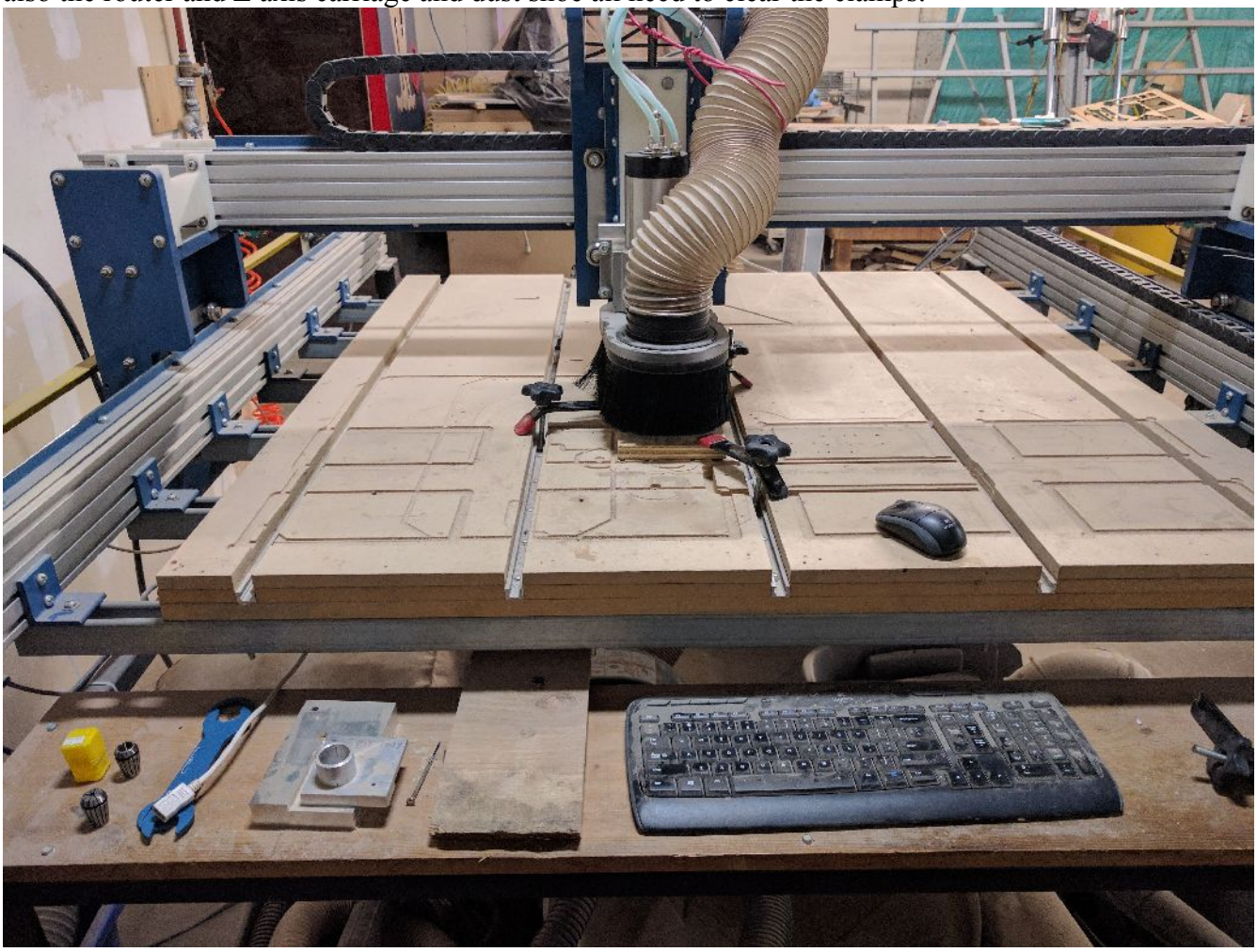
CNC Router - Operation Guide
07/01/2017

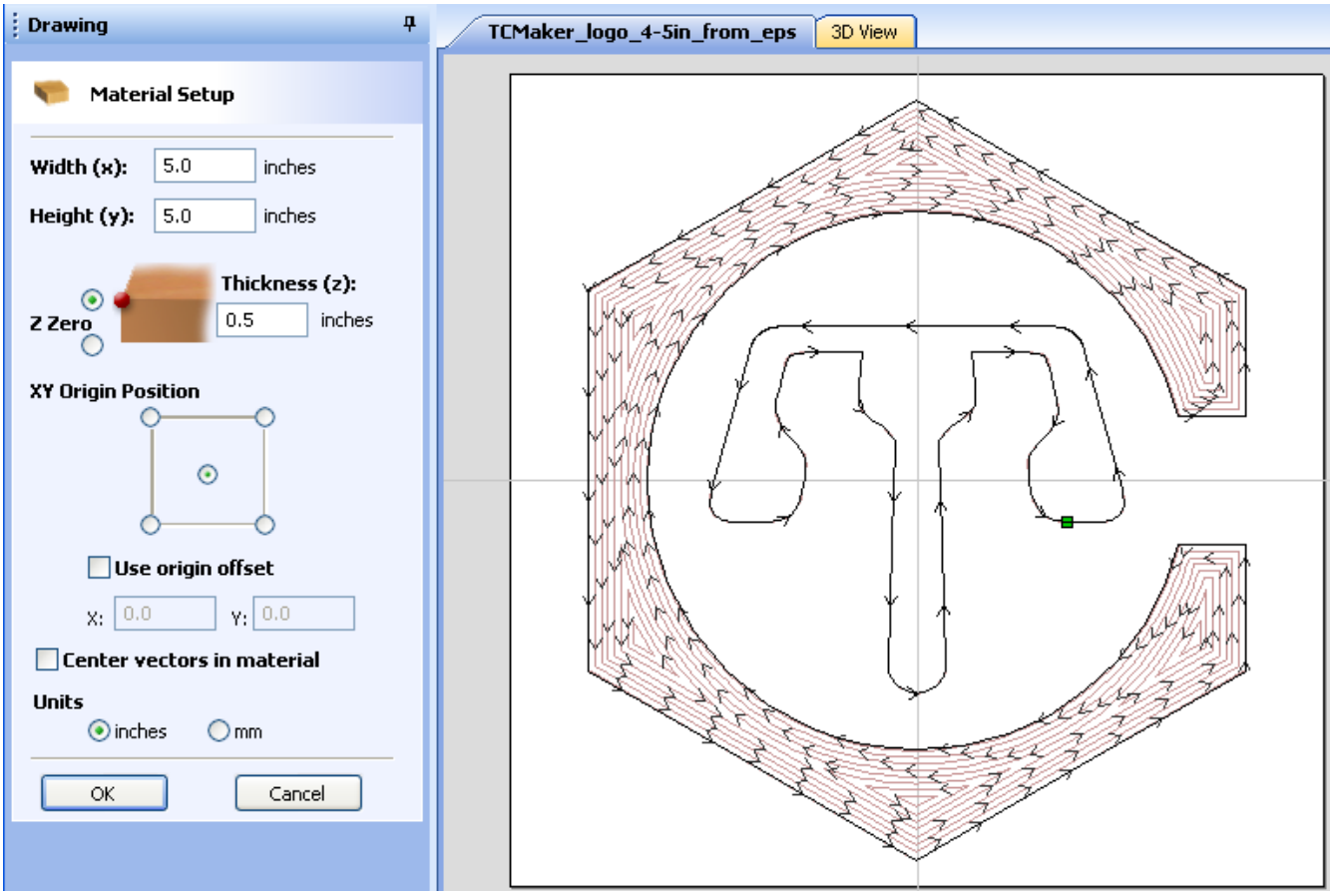
Material selection

Figure out how you will secure the work piece first, make sure you make allowances and leave sufficient clearance for clamping the work securely, remember the collet nut is much bigger than the bit also the router and Z axis carriage and dust shoe all need to clear the clamps.



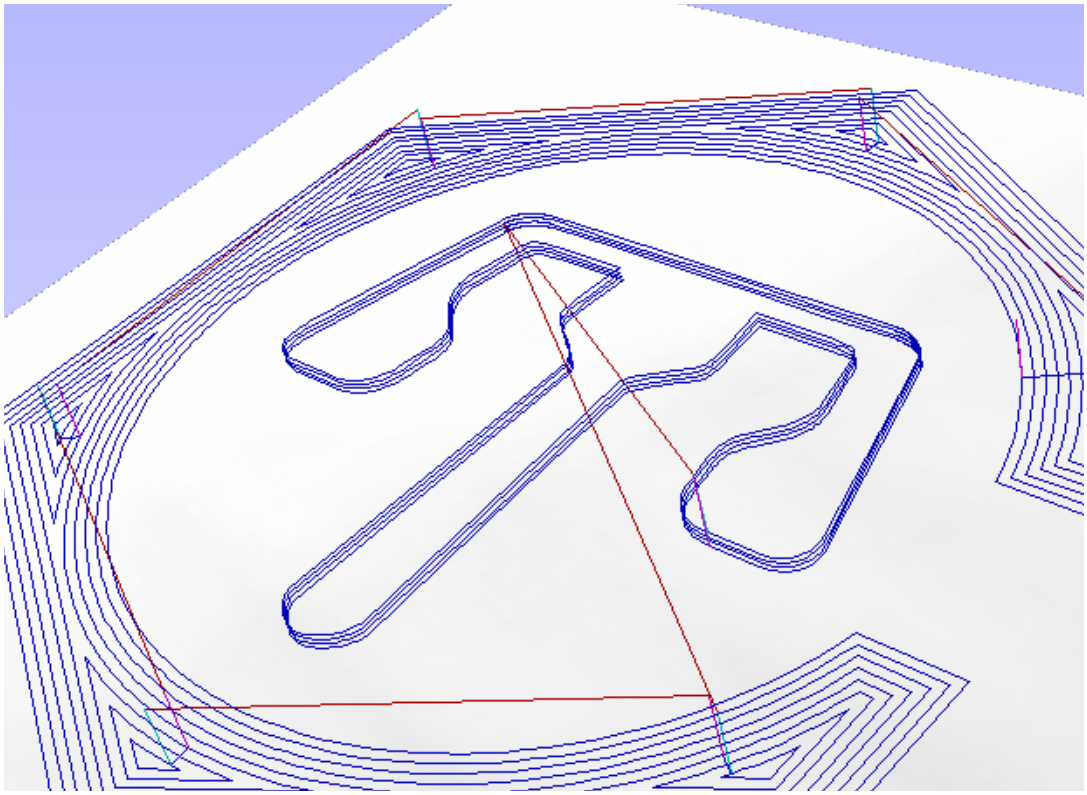
G-code creation
This is a huge topic, and I hope to put together a FAQ or something on it in the future. To support the CAM / G-code creation sections of the CNC classes we offer. The images here are from Vectric Cut2D, which along with the companion software Vectric Cut3D, are licensed and living on computers in the classroom at the Hack Factory.

Set the origin in the center, or on a known reference point.
Note: You will almost always want to set Z=0 at the surface of the material, I.e. all cuts are at a -Z.

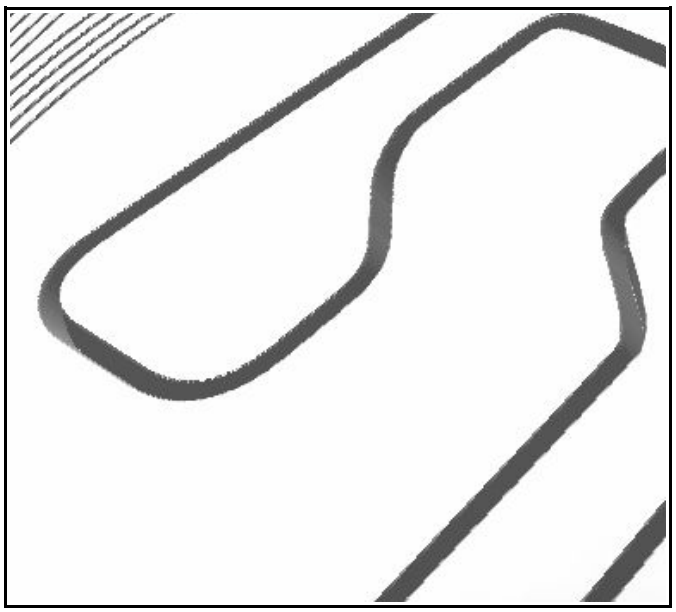


The image above is from Vectric Cut2D. The panel on the left sets the origin relative to the material you'll be using. The image on the right represents the tool paths as they relate to the material. The lines that cross in the center bisecting the material are the X and Y axis lines meeting at the origin. Though it is not toggled, the “Center vectors in material” feature was used to accomplish this. There are many means of generating g-code, most will have these basic features to allow you to set the origin in relation to the material. Also note that inches or mm must be checked, the g-code needs to be created in the correct units of measure, as the machine is set for one or the other when it is configured. (The router is in inches, if you have a drawing in metric units, convert the drawing first)

Make your initial files have slow and shallow cuts. (shallow would be ½ the bit diameter as a rule of thumb)



Above is a 3D representation of the toolpaths, note the multiple stacked lines on the “T”. These indicate multiple shallow passes will be made to obtain the desired depth. As seen in the 3D rendering of the cut below.



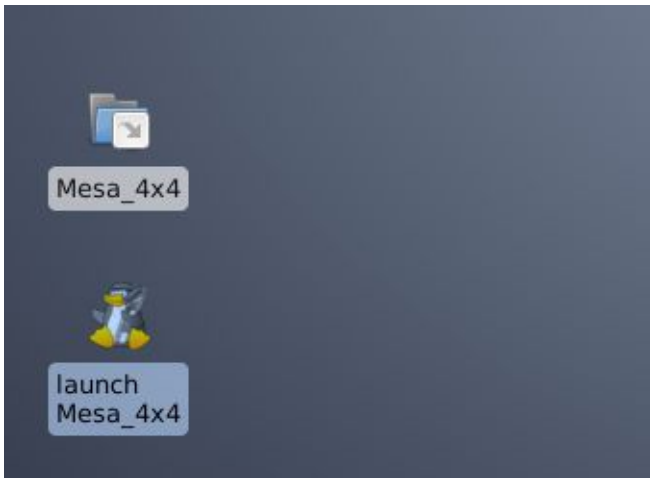
Clearance height should account for screw heads, clamps, other work piece features.

(discuss these, use of a known square edge or just in the center of a large piece, if you don't care, and everything is being cut)

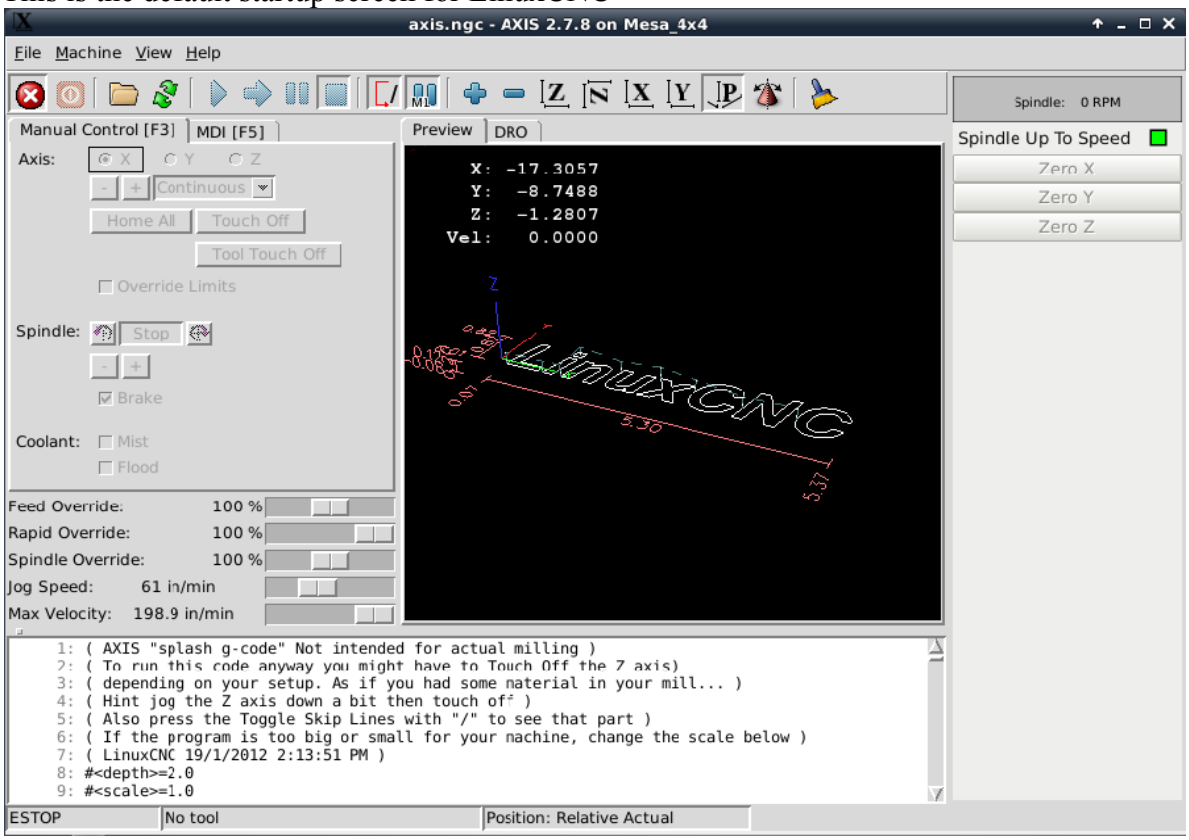
Starting the System:
The CNC Router is running LinuxCNC which is a Ubuntu build with a real time kernel. First step turn the PC and monitor on, if they are not.

The Login information is:
user tcmaker
password tcmaker

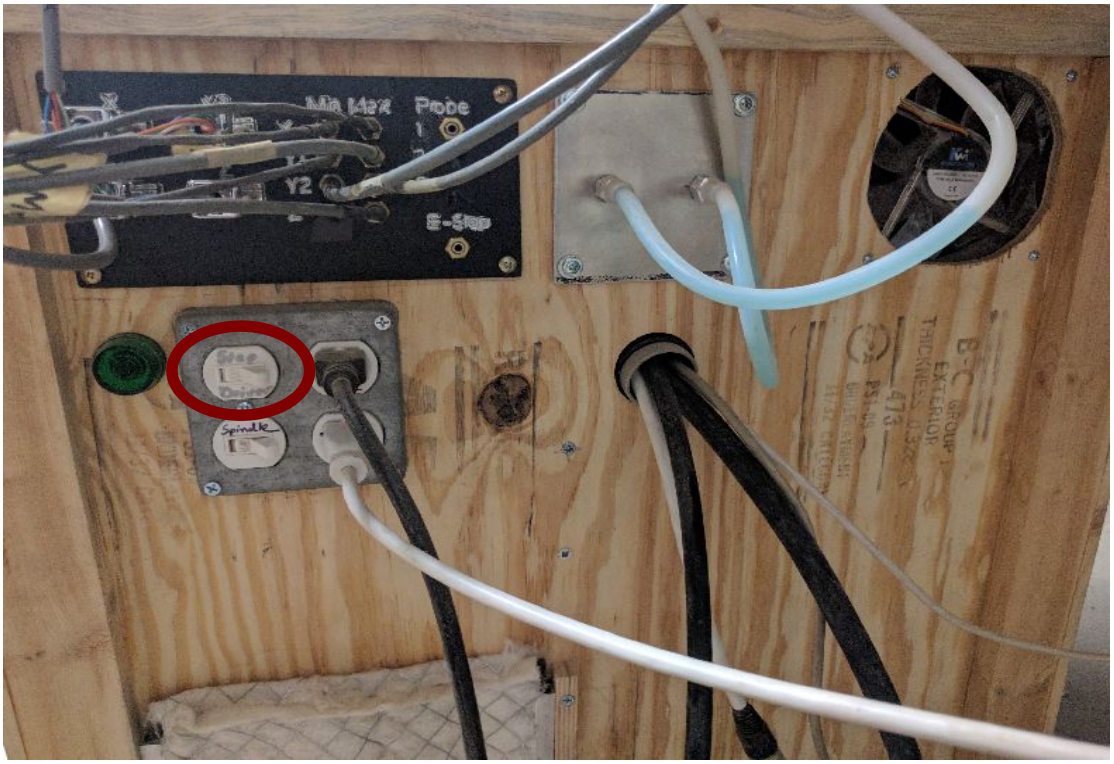
Start LinuxCNC with the “Launch Groucho” icon on the desktop



This is the default startup screen for LinuxCNC



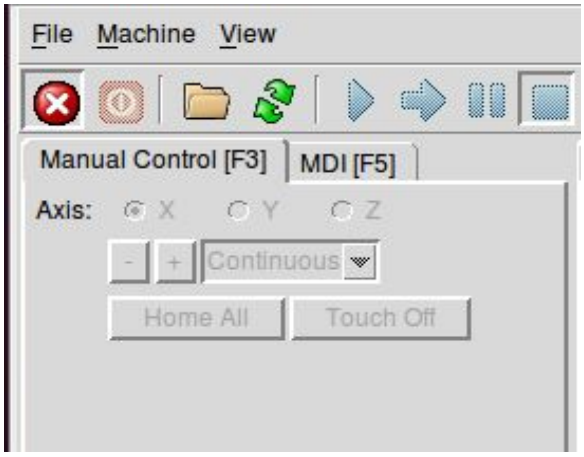
Then power up the control box which is at the back of the machine starting with the stepper driver switch.



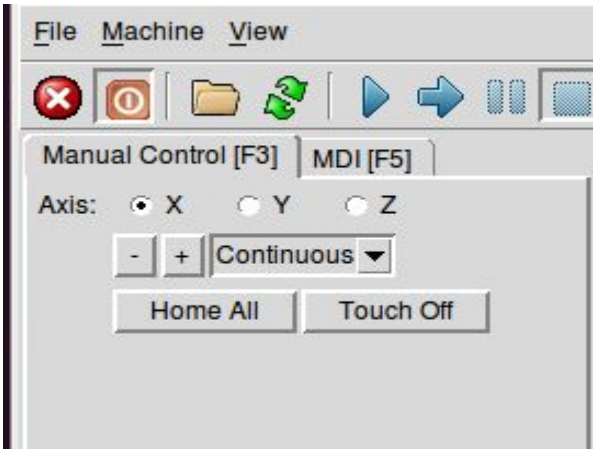
Within LinuxCNC

F1 is the E-Stop (There is also a physical E-stop which must be off before the machine will move)
F2 is Machine Power (soft)

You need to toggle both after launching LinuxCNC before you can do anything.
E-stop on and system powered off



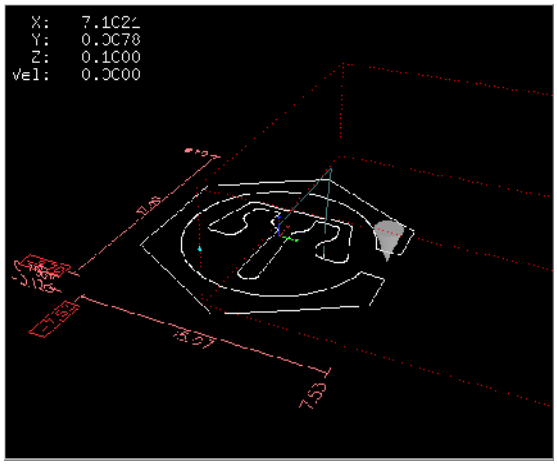
E-stop off and system powered on

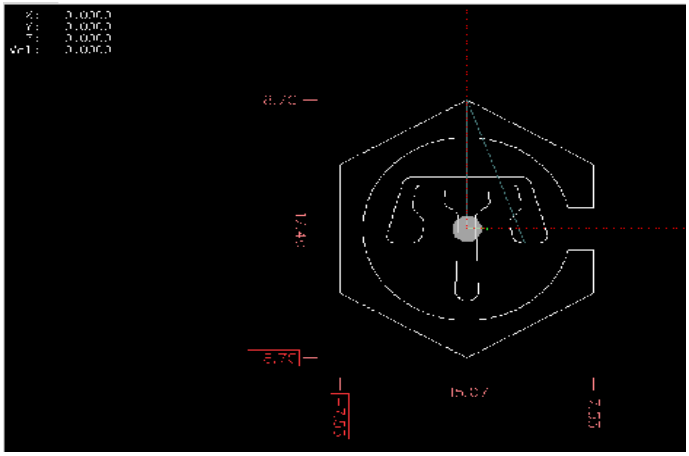


The System is now live.

Orienting the g-code file:
You can next either home the system or load your file and then home the system. I chose to load a file we'd all be familiar with next. Loading a new file is simply a matter of File - > Open (select your file) The default location for g-code files, (.gc, .ngc or tap usually) is the users home directory/linuxCNC/nc_files/. By default .tap files will not be shown, set the filter to all files and you'll see everything. (Note that the system will open raster files , .jpg, .png, .gif etc, as well and can convert them to height maps or 3D carving g-code, that will be discussed later)

The newly loaded file comes in looking like the image on the left below. After loading a file, orient it to the machine, the systems origin (X 0, Y 0) is at the front left right as you face the machine, (normal cartesian setup), To do this either use the center mouse button to rotate the image in 3D, turn the file / environment until the image corresponds to cartesian coordinates, it should be oriented the same way as it was in the CAM environment it was generated in.
Alternately you can click the Upper “Z” menu button, you may need to zoom out to see your image.





Mouse buttons:
Left button is 3D pan.
Right is Zoom.
Center is 2D pan. (currently not working !)

Orienting the machine environment:

The system now needs to orient the tool to it's physical environment. Within LinuxCNC, the working envelope of the system is the cube outlined with dashed red lines. In order for LinuxCNC to understand the tools current location the system needs to be "homed". As seen below, there is a button named "Home All". This will cause the system to run it's homing routine and orient the tool to the work envelope.

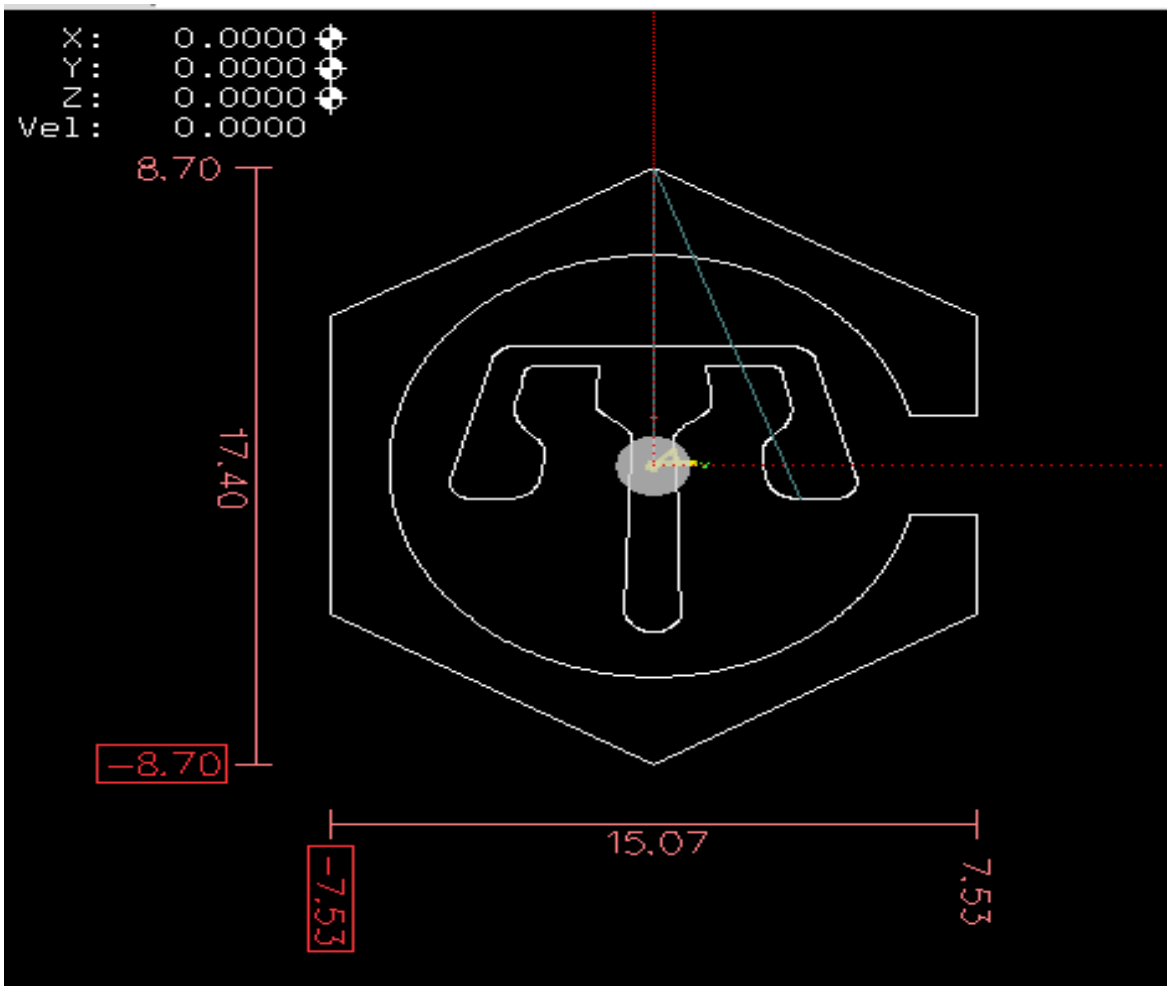
Important!:
The "Home All" button starts the homing routine immediately when pressed, and the first axis homed is the Z-axis. As Z typically refers to the surface of the workpiece, the Z-axis "0" location is highly variable, there is no fixed home switch for the Z axis to set it to "0". Instead the Z axis is homed to the top of it's travel range.
Note the machine will raise the router to the top of it's Z range and then run straight Left to home the X axis and finally it will run forward toward the operator to home the Y axis. When homing anything in the tools path will be run over, and damage to materials, clamps, the machine itself, and human bits that are in the way can occur. Verify that the homing path is clear and that all clamps and material will be cleared by the tool during it's homing sequence. To move the router see the information below.

Manually Jogging the tool to a given location: (not present yet)
Which is just a complicated way of saying manually putting the tool where you would like it. The system has three ways to do this. The quickest and easiest is to use the joy-pad controller.
The two joysticks on the joypad are proportional, and this maps to the speed with which they move the tool. The coarse speed is set by holding down the 1,2, or 3 button and pushing or pulling the stick on the axis(es) you wish to move in the direction you wish the tool to go. (It's best to just make sure the bit is above all obstructions and then just playing with the controller on the 1 or 2 speed setting.)

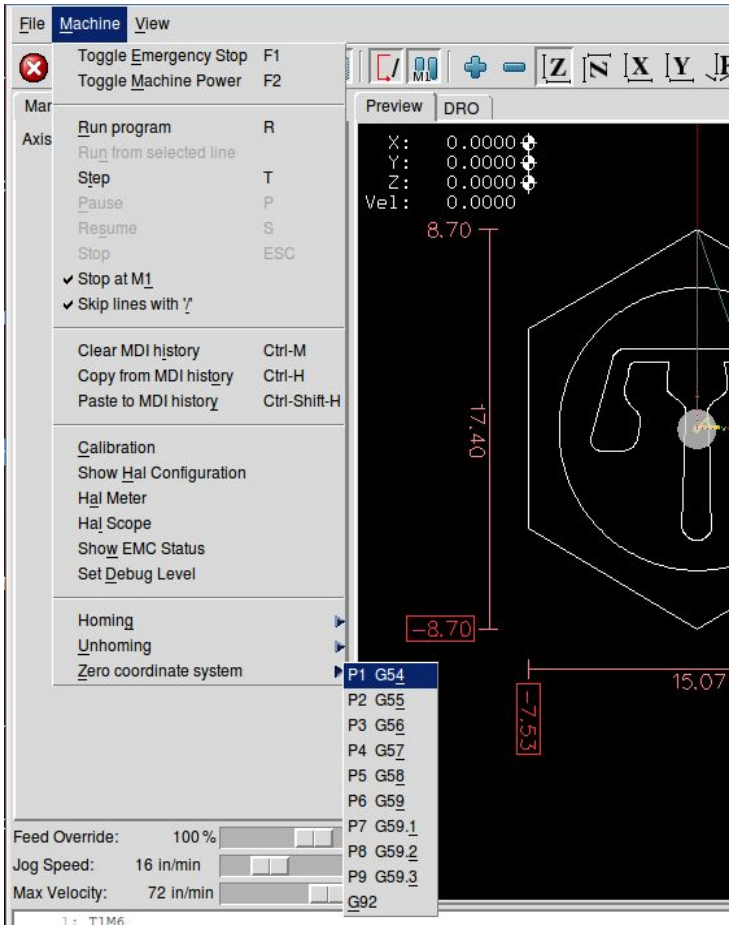
Each axis can be homed by itself as well. Either way position the router so that it can safely home and then either hit the "Home All" button or each axis individual home button, again be sure the router path is clear!

This is what the limit switches look like.

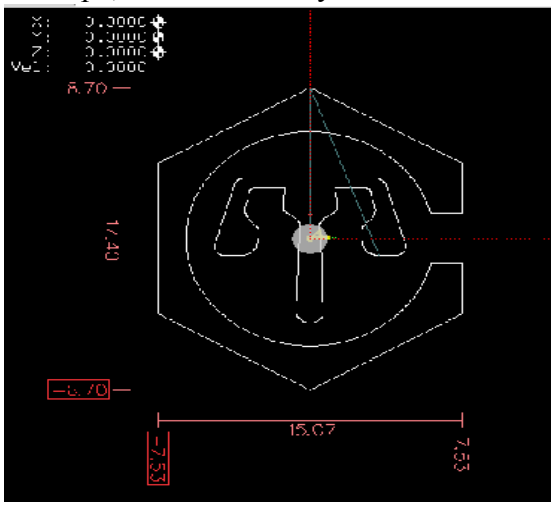
System Interface homing examples:
From a system perspective here's what homed looks like. Note the bulls-eye like symbols to the right of each of the 3 axis. These indicate that each axis has been homed. The zeros for each axis indicate that the router or tool is currently resting at the machines home location.



I recommend that you also zero the coordinate system after homing, as the machine can remember offsets even if it has been restarted. To do this select :
Machine → Zero Coordinate System → P1 G54.
G54 is the default coordinate system, use of additional coordinate systems is out of scope for this document and class.



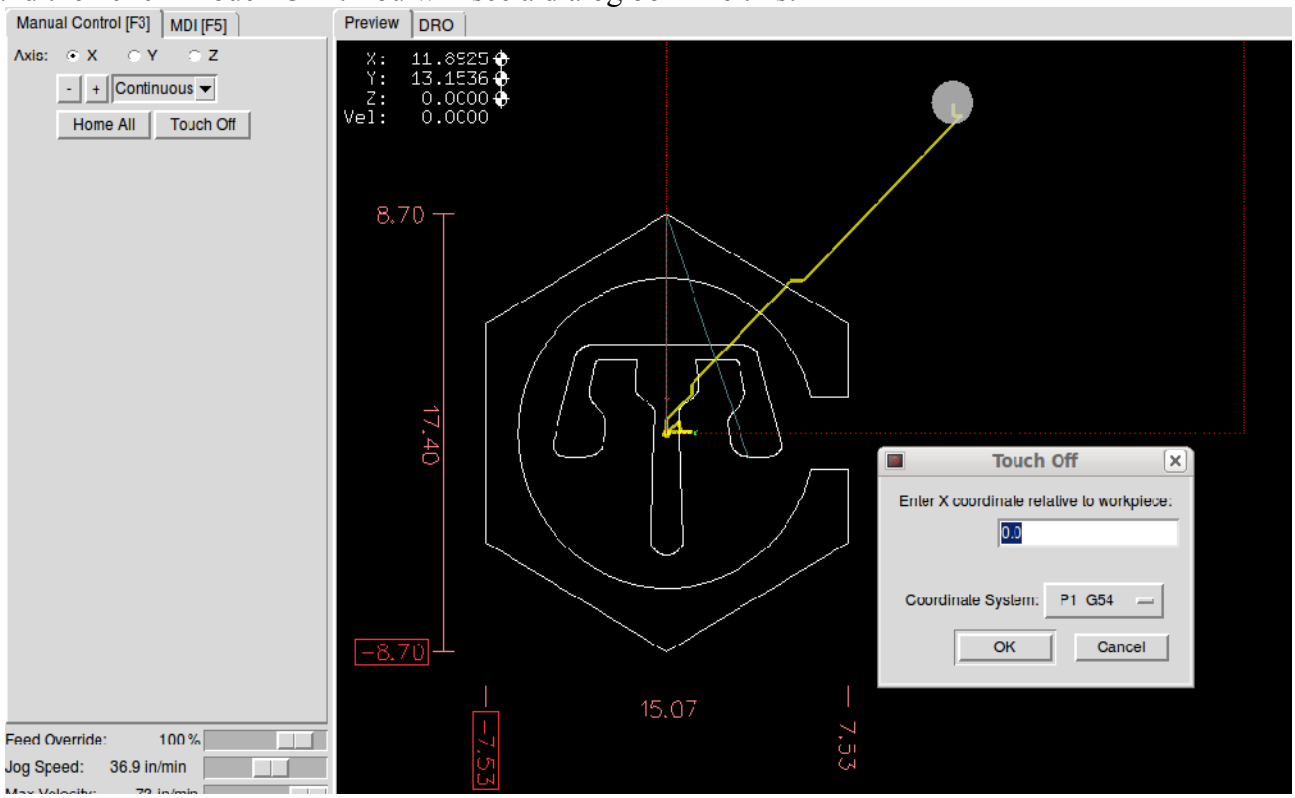
You'll note that the image of the file is centered on the tool, (the gray cone, or gray circle from this perspective) but the file protrudes beyond the limits of the machine, (beyond the red lines) this is because although the system is now homed, and knows where the tool is within the machine work envelope, but it does not yet know where the workpiece has been placed. So this is what we to do next.



Orienting the work piece to the machine environment (touching off):

The next step is to inform the system where the work piece is. This is known as “touching off”. It is OK if your piece is not secured at this point, it's best to orient the system, file and physical work piece first. You will then have a visual and verifiable representation of all tool travel, which is very helpful in setting up clamps, etc so that they will not interfere with the cutting process. Drive the router to where ever you have set the origin on your material, as described above in **Manually Jogging** above. In our example file the origin is set in the center of the piece, so we can center our material on the machine bed, and jog the router near to the materials center, and begin the “touching off” process.

We will begin by “touching off” the X-axis. To do so select the X-axis radio button (upper left below) and then click “Touch Off”. You will see a dialog box like this:

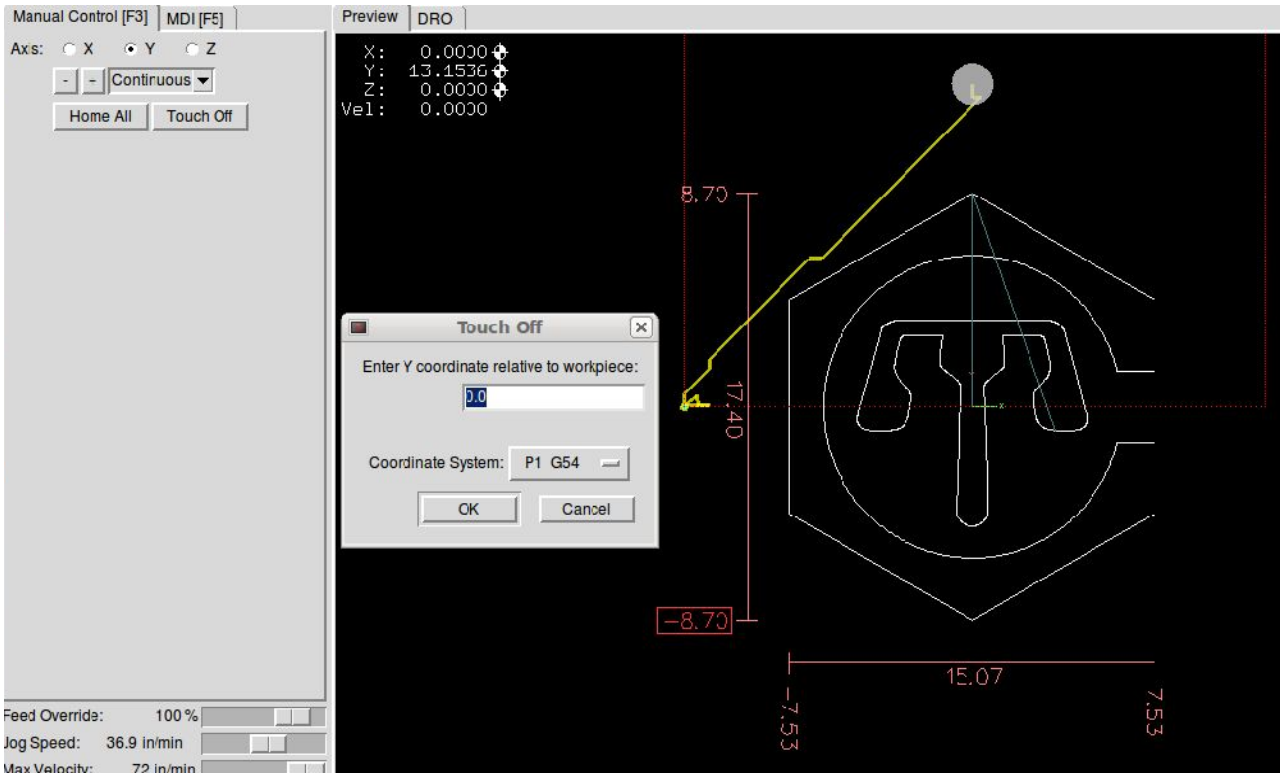


(Note the new X and Y axis readings which indicate how far the tool has been jogged to the center of the table, the yellow line is the path the tool traveled while being jogged, the pink dimensions to the left and below the TC Maker symbol tool paths indicate the size of the object being cut, and the distance from the files origin, which is in the center of the TC Maker symbol.)

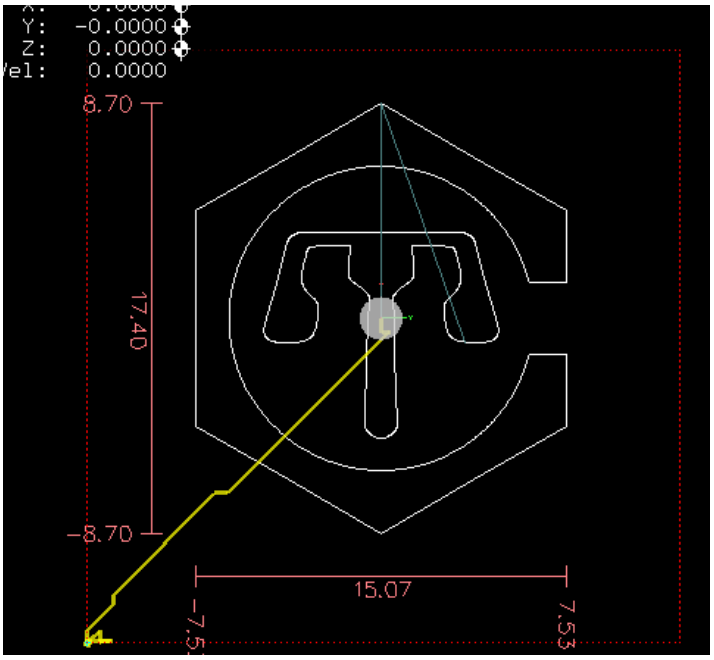
If you have the tool centered over your work pieces origin then “0.0” is correct, if you are at an edge that is a known distance from the center then you add or subtract that distance, plus the radius of the tool (half of the diameter) you currently have in the router. I.e if you have a 1/4” bit touching the right edge of the piece which is 2” from the center, you would use 2.125 as your touch off value.

Next we will touch of the Y-axis in much the same way. Select the Y-axis radio button and then click “Touch Off”. For the Y-axis negative numbers are to the front of the machine, positive numbers are to the back of the machine.

You can see in the image below that X-axis is now at 0 and the image is centered left to right on the tool. The tool is represented by the white cone, straight below it is the small red, green and blue origin marker.



Once the Y-axis is touched off the system will look something like this.



Securing the work piece:

Now that you can see the extents of the piece you will be cutting out, and the tool paths that will be traveled, you can clamp the work piece to the table. Use the t-slot clamps underneath the router, they slide into the t-slots on the table and then screw down. You may wish to block up the back of the clamp so you have a down angle on the clamp as it holds the work piece.

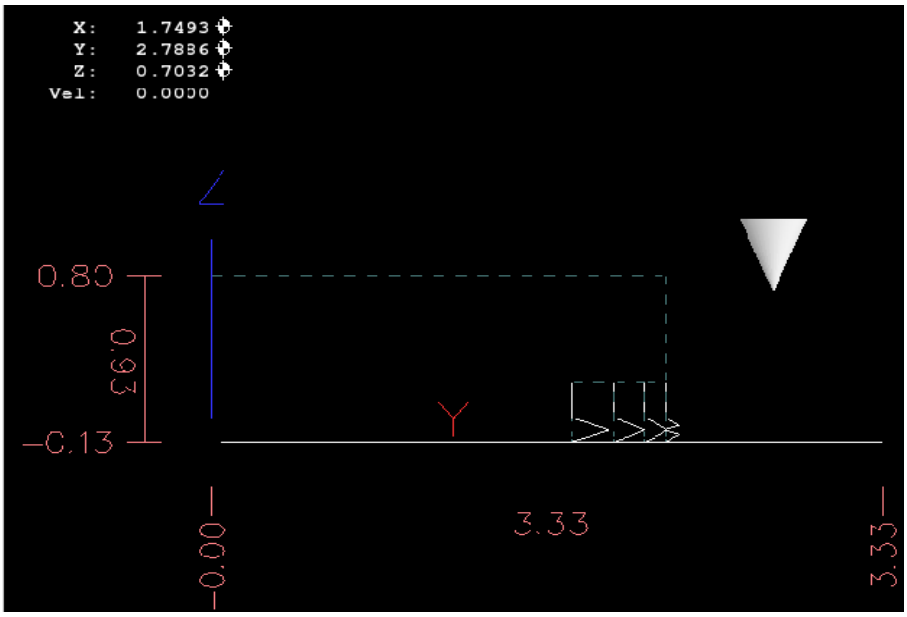
With the work piece secured, you can drive to the extents of the tool paths with the keyboard and verify that the router body, collet nut, dust shoe and bit will not run over any clamps etc. You may need to pan the image to keep a straight down perspective on the work piece as you go.

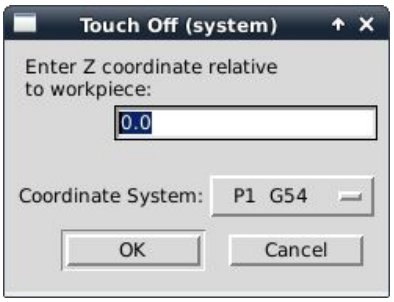
Mouse buttons:
Left button is 3D pan.
Right is Zoom.
Center is 2D pan.

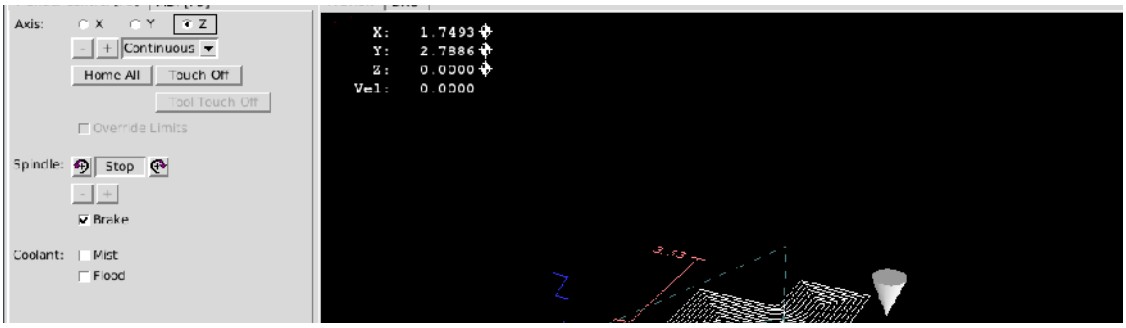
Note: Cutting a part out from material that is the same or nearly the same size as the finished part is not an easy task, as room to hold the material with clamps does not exist, in such a case you may need to cut internal features such as holes first and then screw the part down before cutting it out, plan carefully, and review your plan before you start cutting! Buy extra material whenever possible.

Preparing to cut:
Cut Air first, before touching off Z on the work surface, it's a good idea to:
A: drive to the extremes of the tool paths and check clamp placement as mentioned above.
B: Cut air, which means

> Place the end of the tool about an inch above the work piece surface, and touch off the Z axis at 0, you can then run the program with the spindle off and verify that no tool paths with run into clamps, off the stock, etc.







Once you are satisfied that the piece is setup correctly, you can set the tool on top of the workpiece and touch off Z at 0 again, before begining the actual cut.

Powering up the spindle motor.
The spindle is a water cooled 3 phase spindle, its speed is controlled by the software, and is typically set as part of the tool definition in your CAM software. Turning the spindle on, which also turns on it's coolant pump is done by turning on the lower switch on the control box.

Some materials, especially those that melt. (Foam, plastics, etc) will benefit from running at less then full speed. If cuts in wood are burning, it is likely that the tool is moving too slow, and increasing the feed rate is the best remedy, you may need to have more shallow cuts to accomplish the cuts!

The dust collector is enabled by enabling coolant on the computer control panel, you will hear the dust collector start when you enable this setting, and be able to feel the draft of the suction inside the dust shoe, if not it is not on and you need to investigate before cutting. The dust collection unit plugs into the upper right outlet in the image above, make sure it is plugged in there.

First cuts:

You and everyone near you have your safety glasses on right?
Hearing protection is also highly recommended!
You may want to hear the router and bit make it's first few contacts, but the prolonged whine is hard on your hearing. The spindle is about 3 HP router, and can easily snap 1/8” and 1/4” bits, cut through aluminum clamps, and generally ruin stuff in a hurry, human appendages included, do not become complacent while running the CNC router, and never leave the machine while it is cutting, if you need to leave the area, pause the job, power the router off and do the reverse when you return, please be safe!

You can control the speed of the tools movement in real time by sliding the “Feed Override” slider. Also the Max Velocity can be set as well and overrides the Feed Override settings (If Max is 20 in/min then 20 is as fast as any move will go)

Start slow, which I consider 20-50 IPM much below that and wood will burn and the bits take excessive heat and wear rapidly. An exception to this would be very small bits, they are prone to snapping. You'll need to experiment depending on the material. In general the depth of cut, which is the amount that the tool takes in each pass should be the diameter of the bit at most and closer to the radius for small bits or when in doubt.

Remember to triple check for clearance especially if you had removed the dust shoe in previous checks.

Appendix of sorts:

How to change a bit:

The collet wrenches etc currently live in the left side of the operator area near the monitor.

On the right top to bottom is the collet nut, 1/2” and 13mm collet, the 1/4” collet, and the 1/8” collet. You are unlikely to need other sizes especially for wood working router bits.
To the right in blue are the collet wrenches. The spindle uses an ER 20 system, so always use the ER 20 size of these wrenches. You will notices the wrenches are Aluminum, this reinforces, that the amount of force required to properly tension a collet nut is low, excessive force will no improve the collet grip and will ruin expensive components. DON'T OVER TIGHTEN THE COLLET NUT!

It's hard to see, running the Z-axis up helps to see. As does removing the dust shoe, which snaps into place with magnets and can be popped on and off easily. Also bringing the router to the front of the table helps.

Start by inserting the correct sized collet into the nut. In this example I'm using a 1/4” shank tool.

Snap in gently, one finger pushing in will work just fine.

Then insert the tool into the collet, and hand tighten the nut onto the spindle nose.

The small wrench goes up above the nut on the shaft of the spindle. Remember use the ER 20 side.

A quarter turn is all the tightening you will need, the wrenches move away from each other to tighten.

To change tools reverse the process, line up the tools so that you can squeeze them together with one hand so you aren't tempted to put your fingers between the tools, which will gash your knuckles when the tension is released and pinches your fingers!

Collets and Bits:

Bits will be supplied for class, when you want to cut your own parts, you will be expected to provide your own bits, or may use the community router bits in the tool chest in the wood shop. (Check on availability of the size(s) and type(s) you need.

I have found that the roto zip bits are good bits to learn on in soft materials like foam and coroplast.. The Saber Cut version has a flatter bottom, more like an end mill. The standard points are more drill bit shaped. They need the 1/8” collet adaptor as pictured below. (that's a Saber cut)
The other collet adaptors are for other less than 1/4” shank bits, there should be 3 not sure where the third one has gone. The live in the little envelope with the collet wrenches.

Here's a few more bit types that are commonly used in CNC projects..

In the center are End Mills (these are High Helix Aluminum) ones, they work well on plastic too, not so much on wood. On the right are wood bits, the top one being a shallow, maybe 90 degree Vee bit.

Comments, corrections requests for further details gladly accepted, the guide like the router is a work in progress.

Pete