

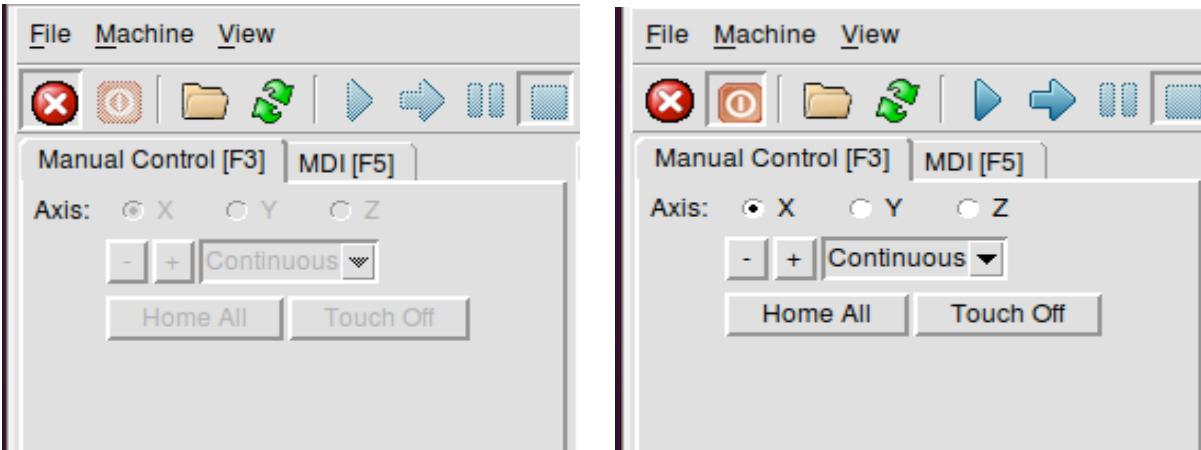
Within EMC.

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 EMC before you can do anything.

E-stop on and system powered off

E-stop off and system powered on



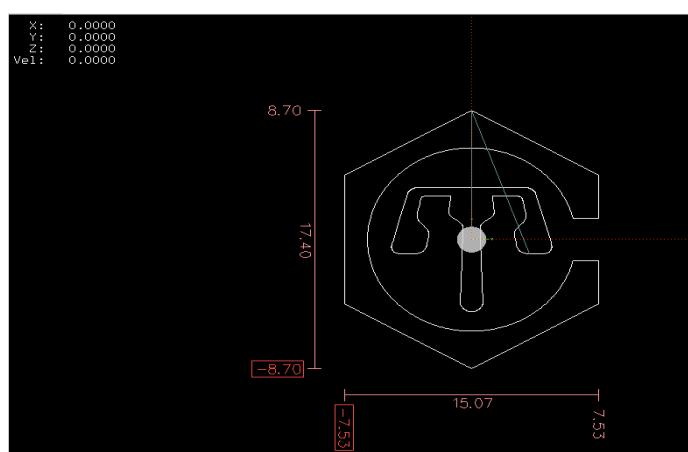
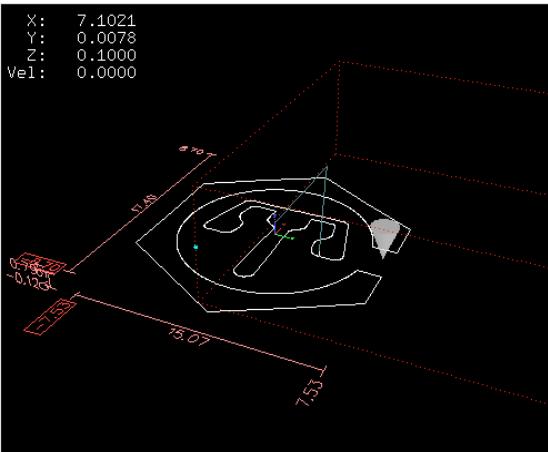
The System is now live.

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.

Orienting the machine environment:

The system now needs to orient the tool to it's physical environment. Within EMC, the working envelope of the system is the cube outlined with dashed red lines. In order for EMC 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:

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.)

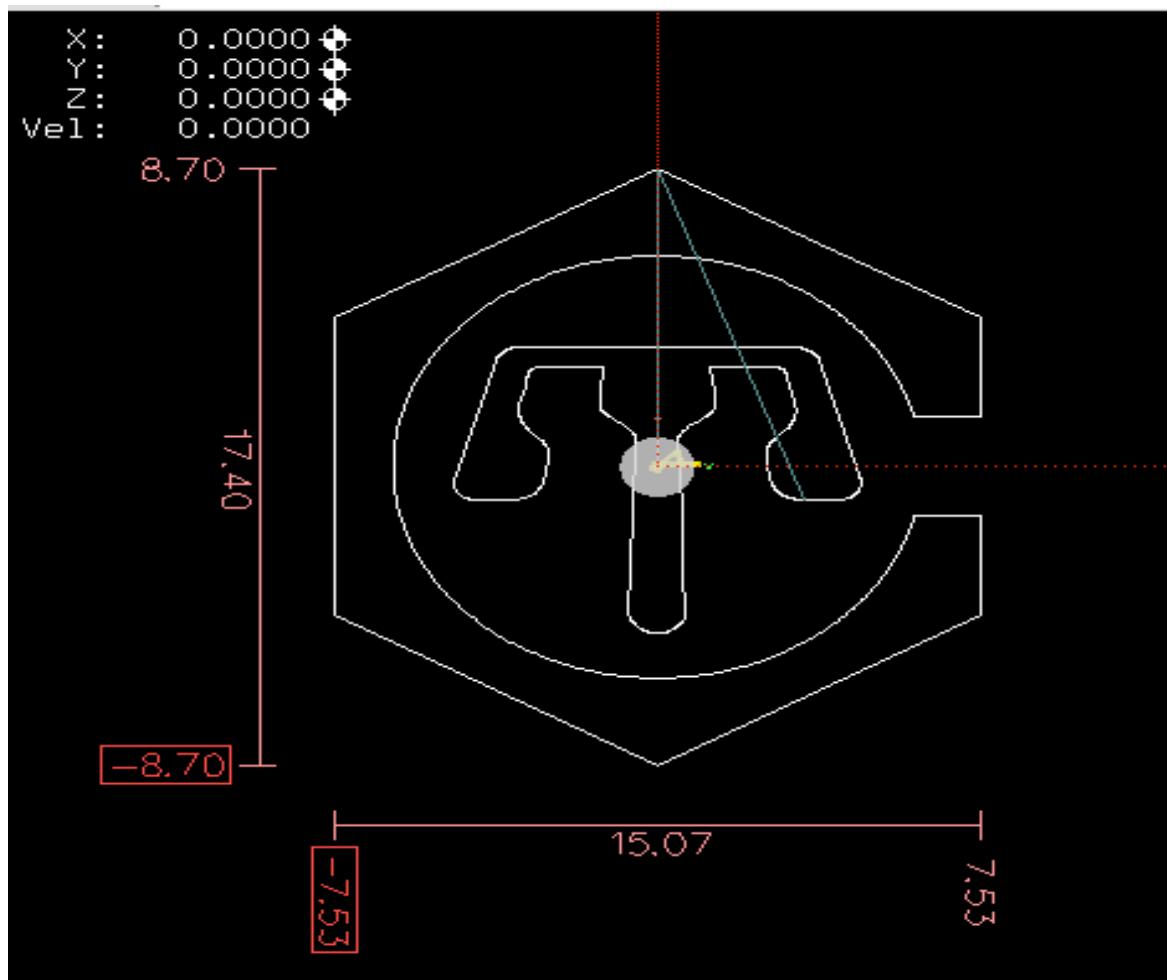
Each axis can be homed by itself as well. Either way position the router so that it can safely home and then either hit the "Home All" button or each axis individual home button, again be sure the router path is clear!

This is what the limit switches look like.

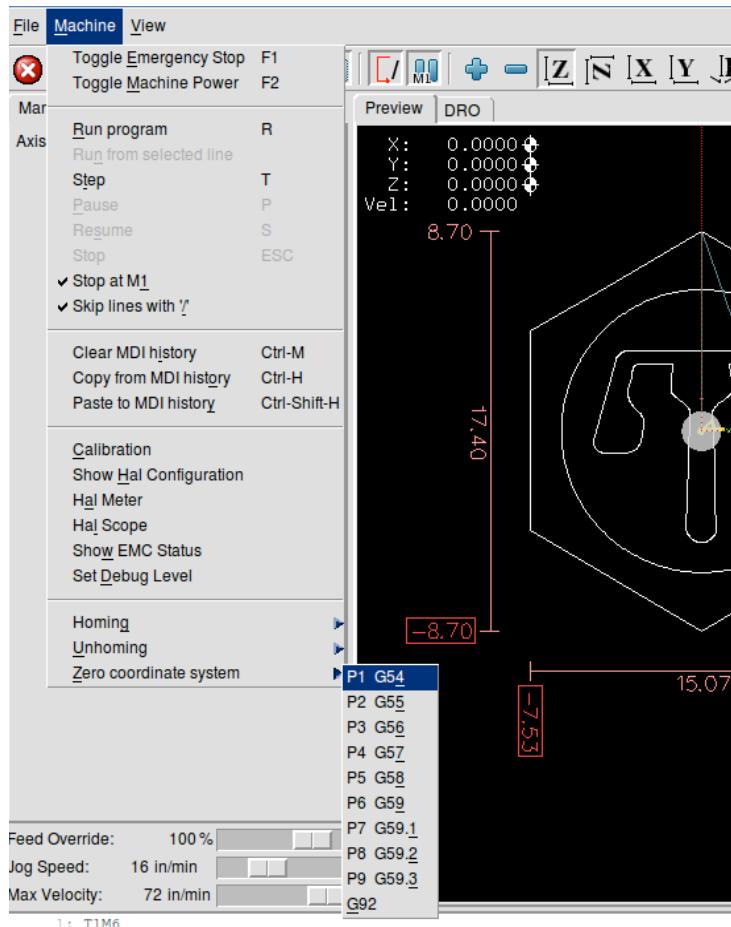


System Interface homing examples:

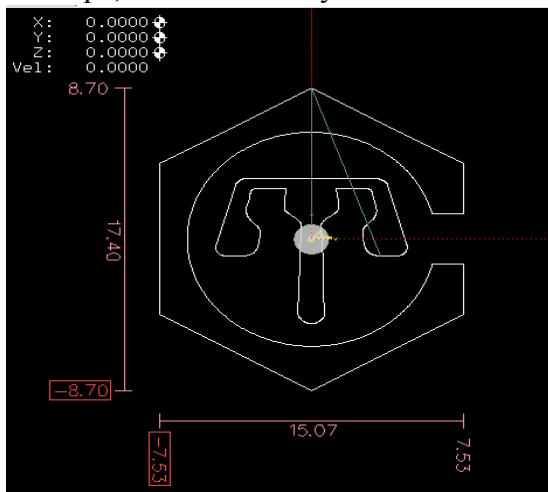
From a system perspective here's what homed looks like. Note the bulls-eye like symbols to the right of each of the 3 axis. These indicate that each axis has been homed. The zeros for each axis indicate that the router or tool is currently resting at the machines home location.



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.



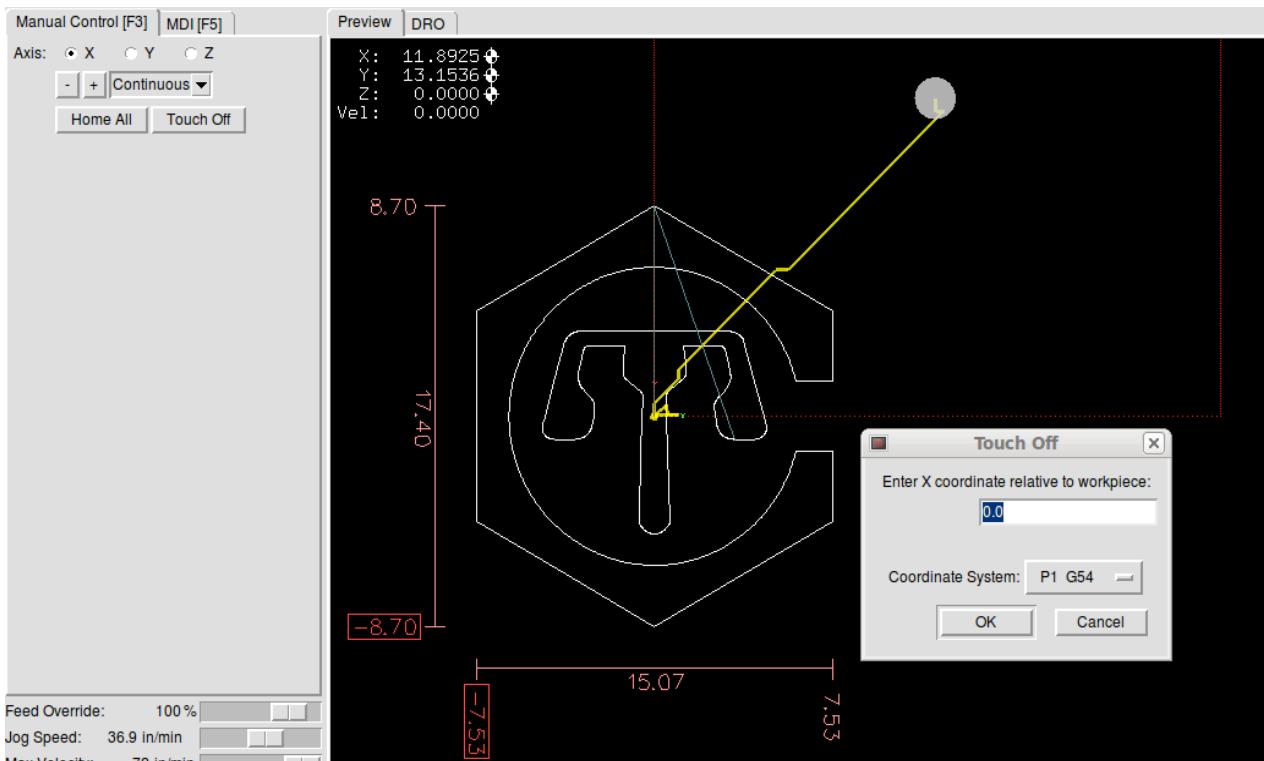
You'll note that the image of the file is centered on the tool, (the gray cone, or gray circle from this perspective) but the file protrudes beyond the limits of the machine, (beyond the red lines) this is because although the system is now homed, and knows where the tool is within the machine work envelope, 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:

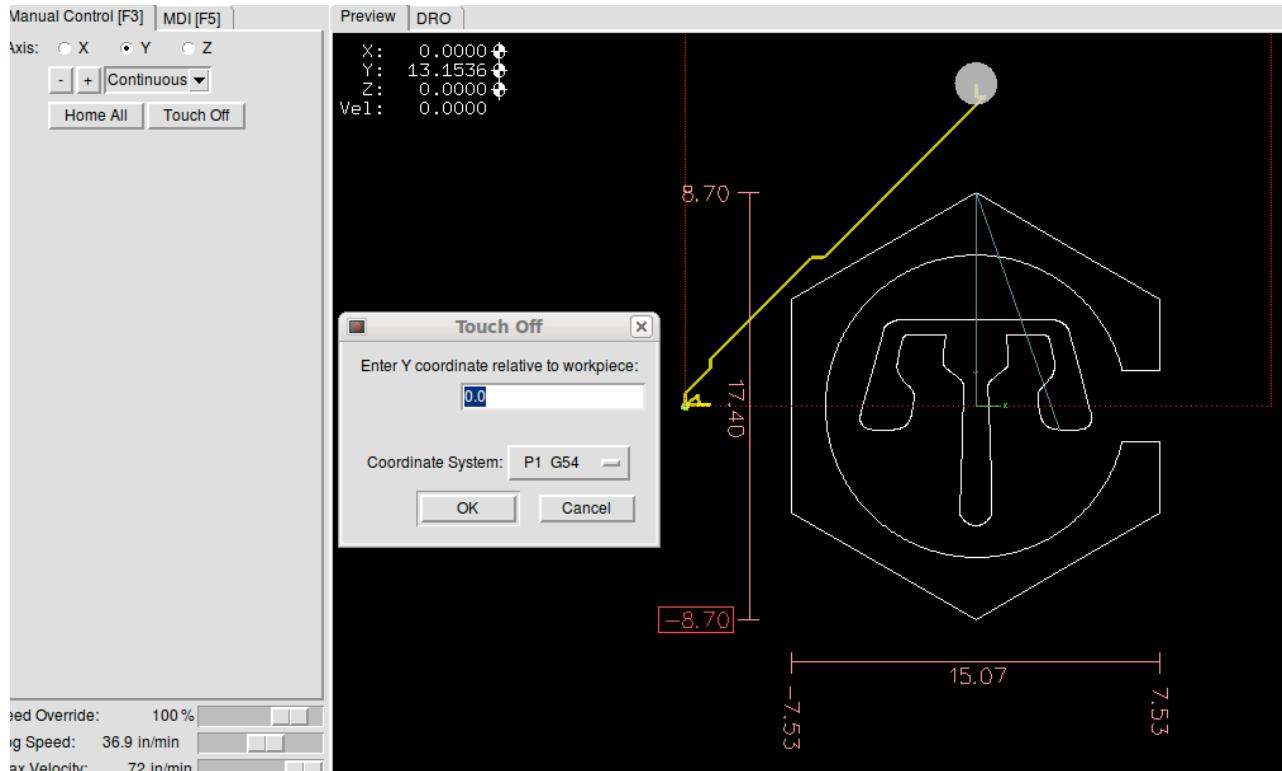


(Note the new X and Y axis readings which indicate how far the tool has been jogged to the center of the table, the yellow line is the path the tool traveled while being jogged, the pink dimensions to the left and below the TC Maker symbol tool paths indicate the size of the object being cut, and the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

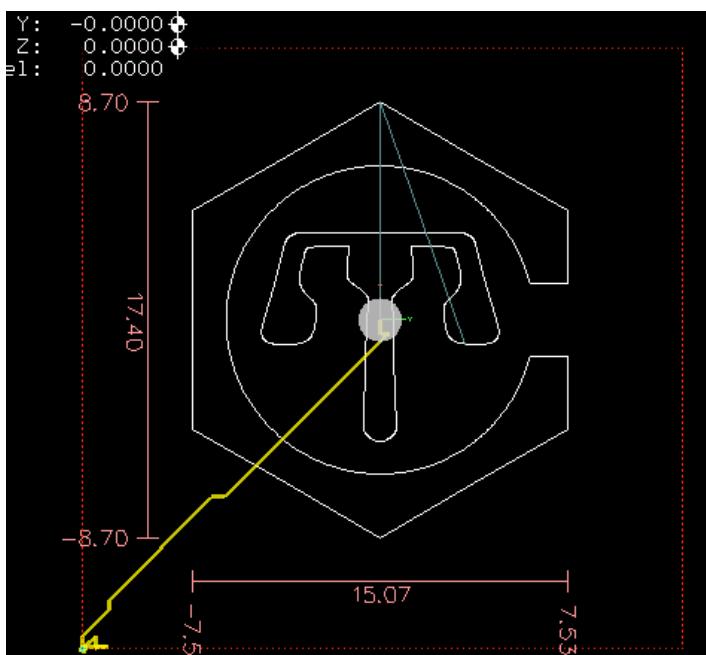
If you have the tool centered over your work pieces origin then “0.0” is correct, if you are at an edge that is a known distance from the center then you add or subtract that distance, plus the radius of the tool (half of the diameter) you currently have in the router. I.e if you have a 1/4” bit touching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



Please see the “Securing the work piece:” documentation next!