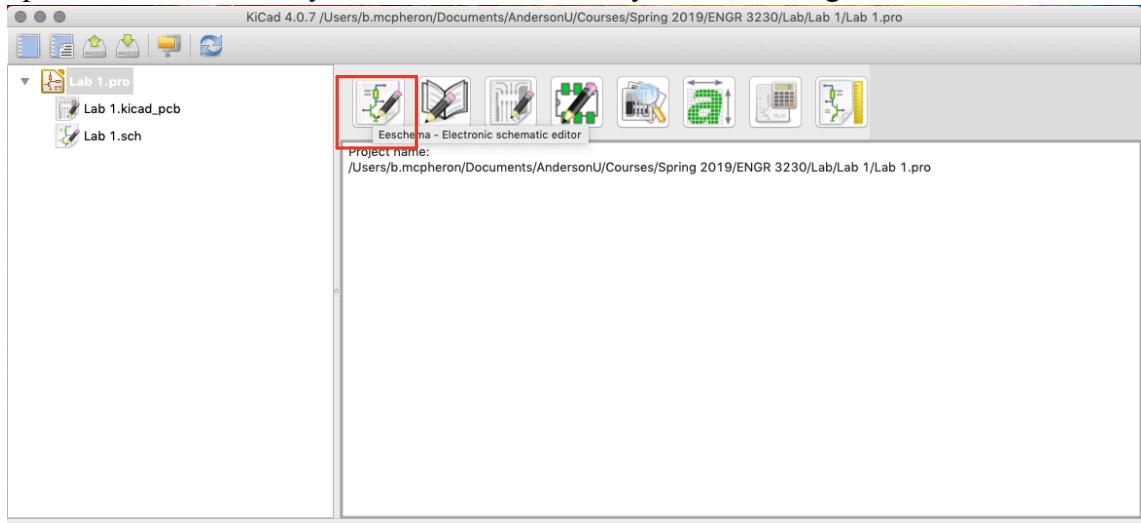
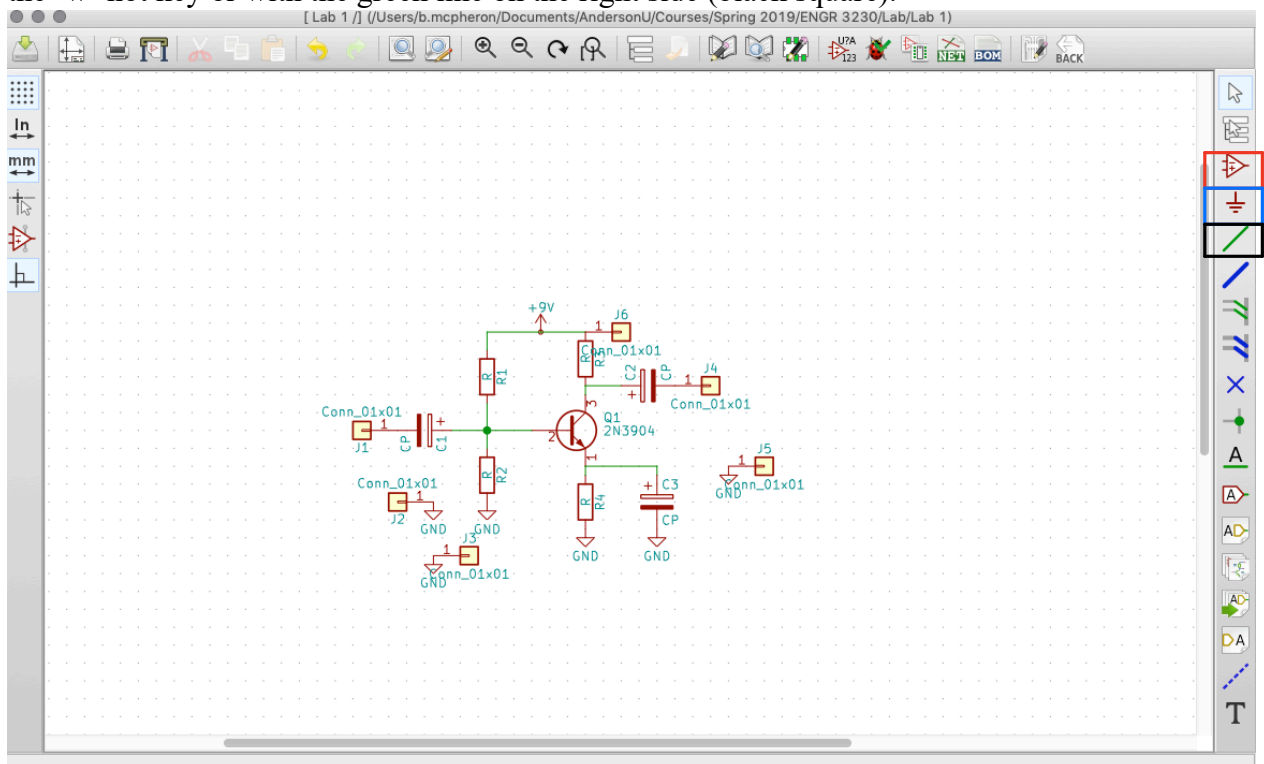


## Getting Started with KiCAD for PCB Layout and Design at AU

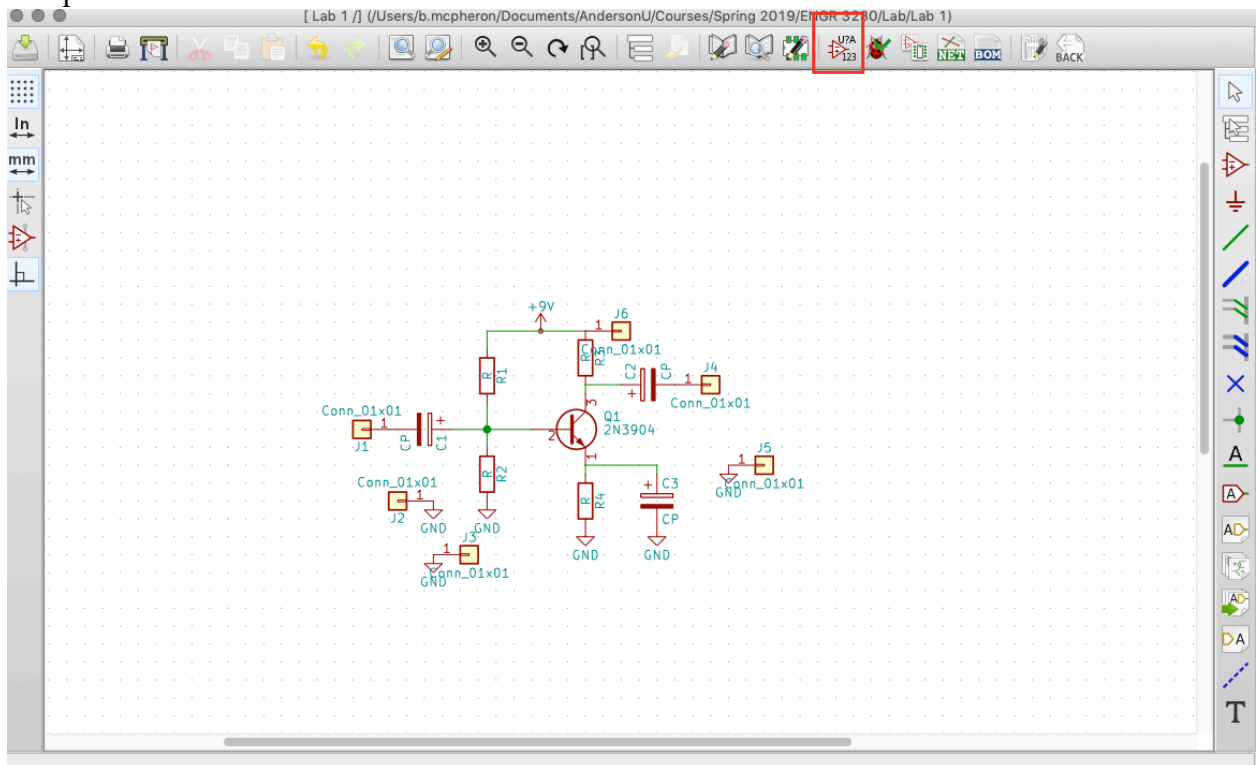
1. Download KiCAD <http://kicad-pcb.org/download/>
2. Open KiCAD
3. Click File->New Project and create a new directory for your project
4. Open Eeschema, where you can make the circuit you are working on



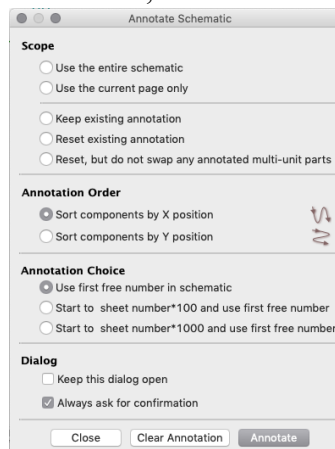
5. Build your circuit. Click the op amp symbol on the right-hand side (in the red square in the photo below), then click in the drawing area to add components like resistors, capacitors, transistors, etc. Hot keys include 'm' for move, 'c' for copy, and 'r' for rotate. An example circuit is shown below. Note that ground and power come from clicking the ground symbol to the right (inside the blue square below). Wires can be connected using the 'w' hot key or with the green line on the right side (black square).



- Click the annotate button (red square below). This will assign names and numbers to all components in the schematic.



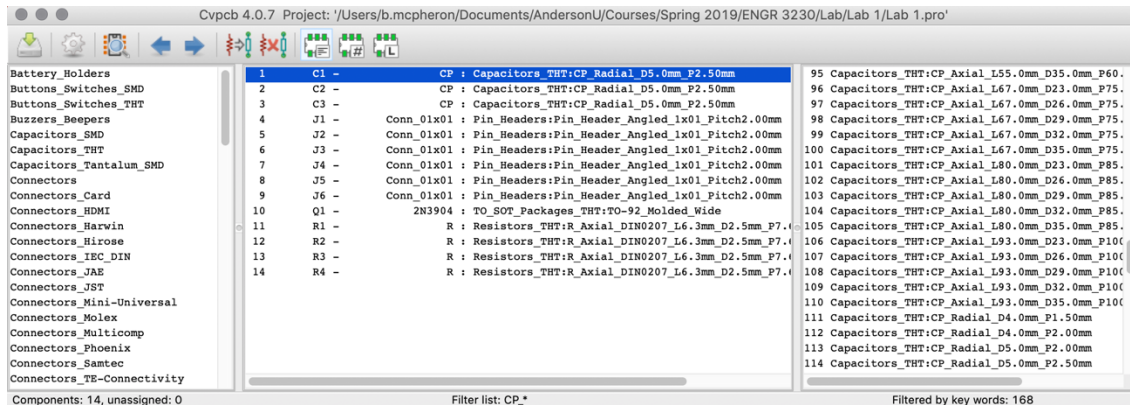
- With the following settings, click annotate, and OK in the next dialog box.



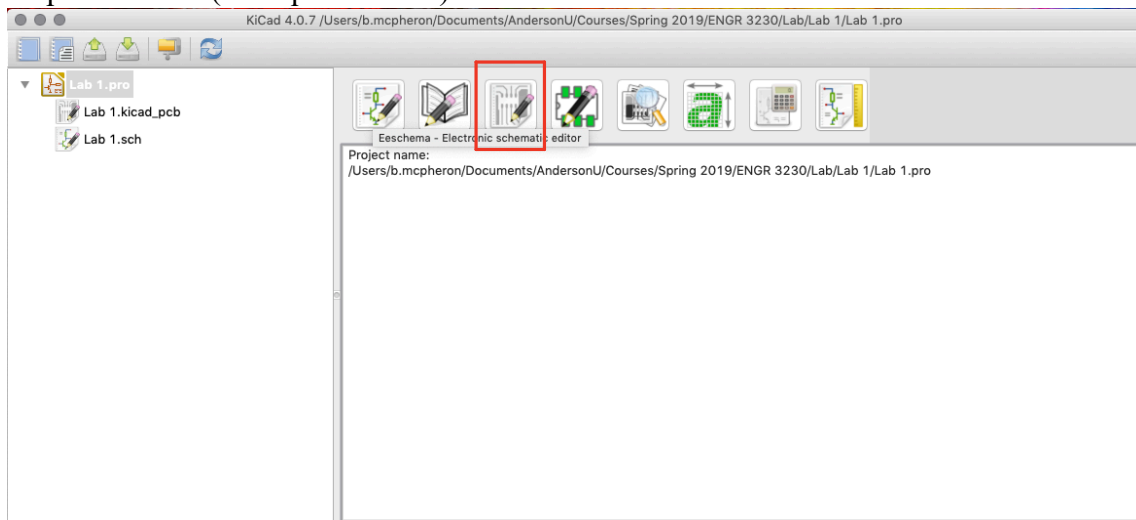
- Now the circuit is annotated. Go ahead and save the schematic project before moving on. Next, click on generate netlist (red square below). Then click generate, then save. This will make a file which defines all the nodes and connections in your circuit. We will need this to define the actual PCB layout.



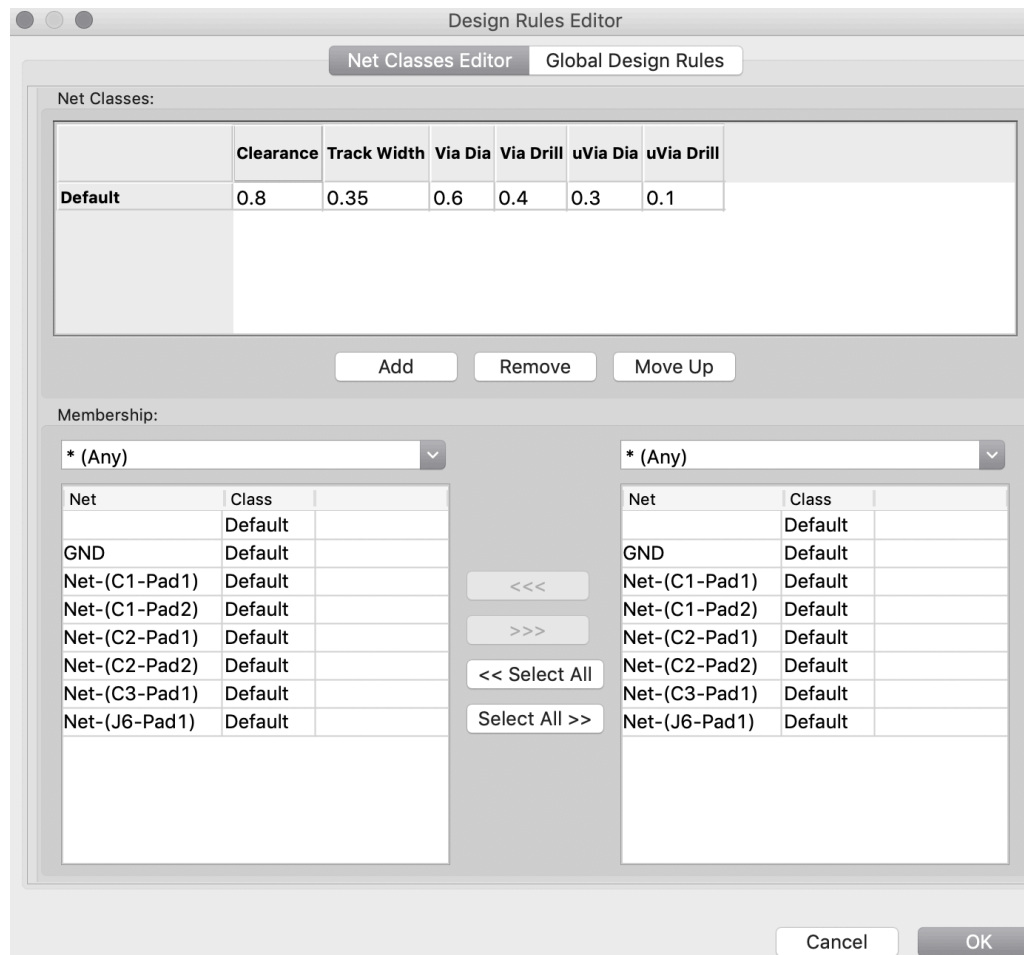
11. Choose appropriate footprints for each device. For the example circuit, this is what mine looks like



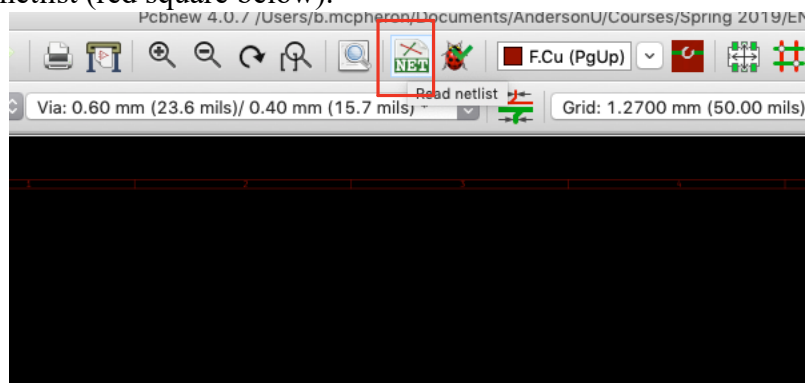
12. Close the cvpcb window. Make sure it saves as you exit. Regenerate the netlist. This associates the correct footprints with your net. Close the Eeschema window, making sure to save as you exit.
13. Open Pcbnew (red square below)



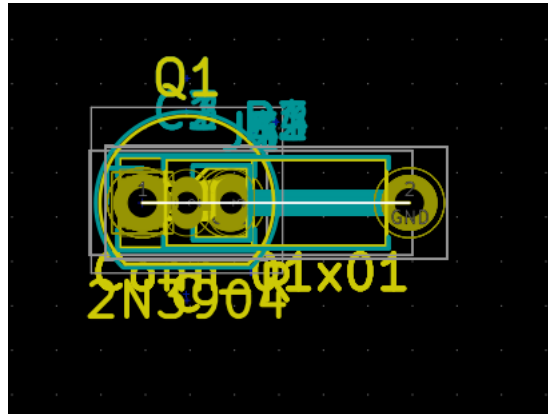
14. Change the design rules to make it possible to mill this on the OtherMill. The following settings are ok for a 1/32" end mill.



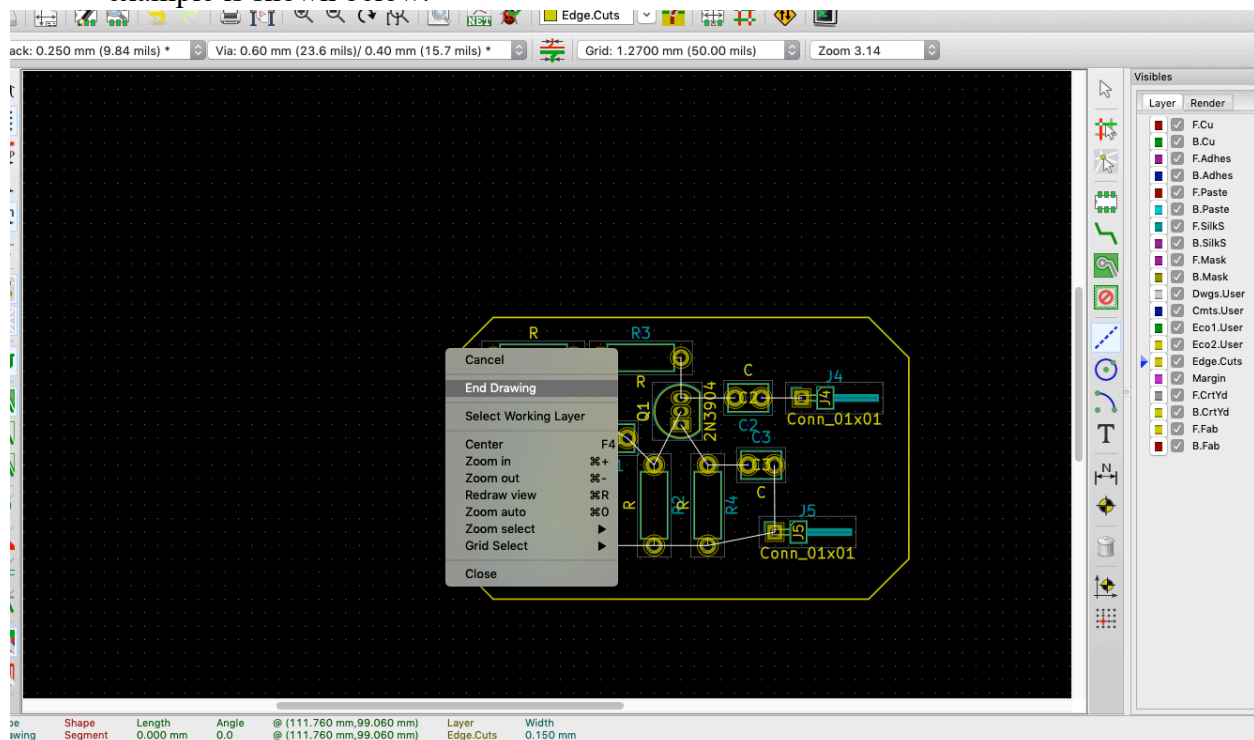
15. Read the netlist (red square below).



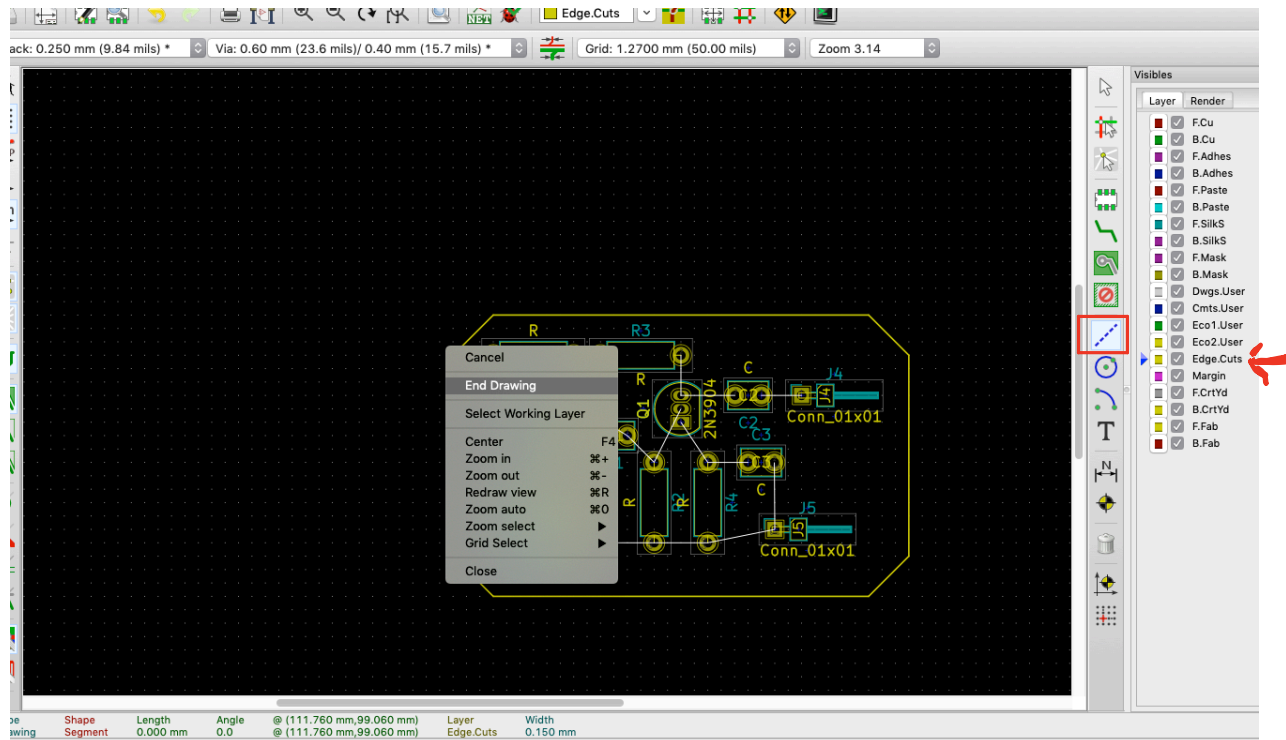
16. Click read current netlist, then close the window. This will drop all of the components into your pcb layout (all on top of each other).



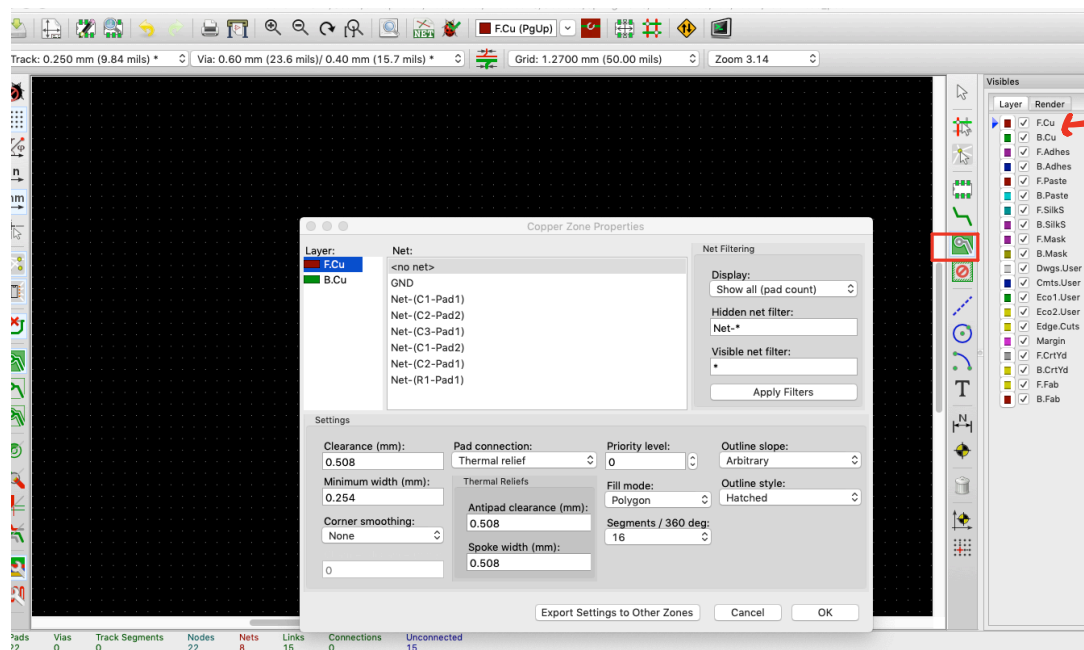
17. Use the hotkeys ‘m’ (move) and ‘r’ (rotate) to move components to the position you want. You will see thin white wires, called the “rat’s nest” which indicate which terminals are supposed to be connected based on your Eeschema schematic. Lay things out in whatever way makes the most sense to you. There are shortcut tools to automatically spread and place but the tracks that result don’t always make the most sense. My layout for this example is shown below.



18. Click on the graphic line (red square). Then choose the edge.cuts layer. This will define the edge of your pcb. Draw a shape around your layout by clicking, and when you have completely defined the shape, right click and choose end drawing.

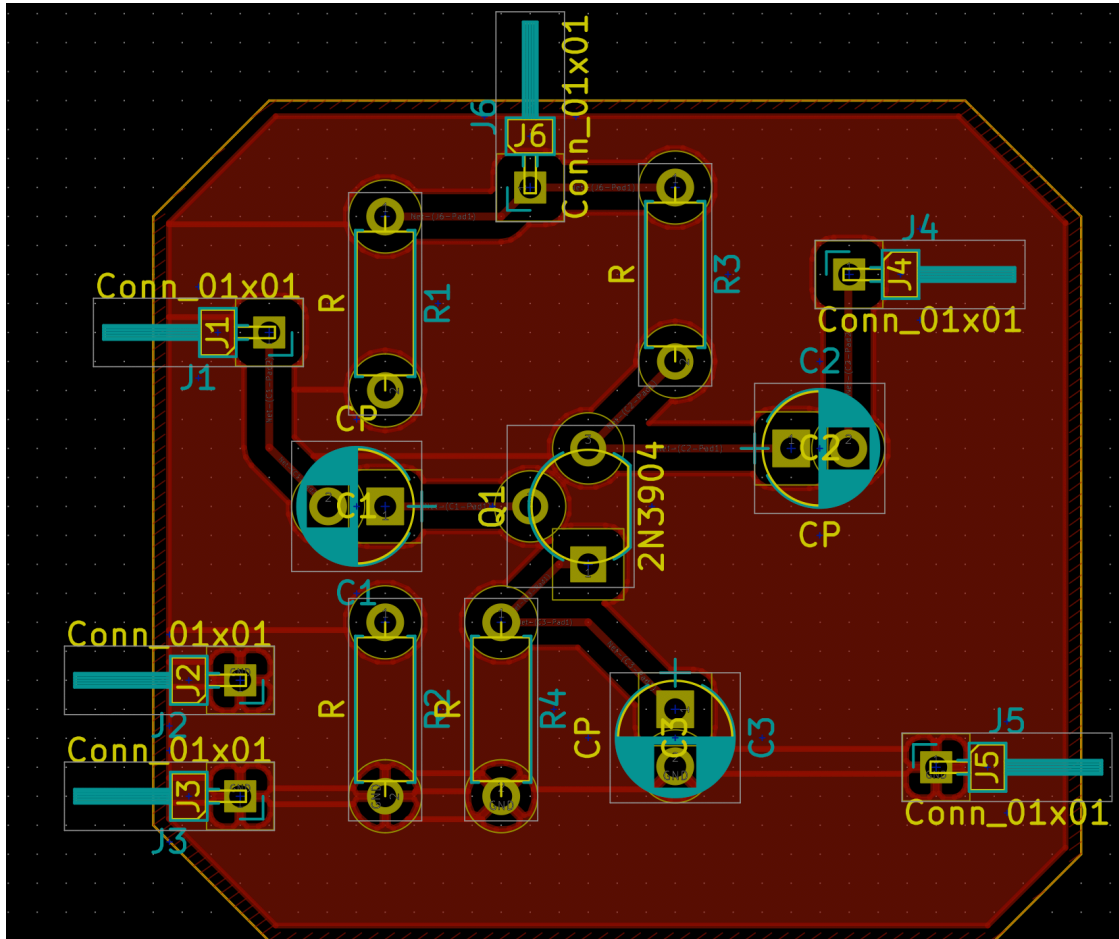


19. If you want, you can choose to make a filled zone. I am going to do so to make the filled zone be ground (since there are so many tie ins to ground in this particular circuit). Choose Add filled zones (red square), then choose F.Cu layer. Click inside your edge cut. Choosing F.Cu, and then GND, click ok. I will make this layer the full size of the PCB (I want the mill to leave this as copper). After your trace the edge cuts, right click and close zone outline.

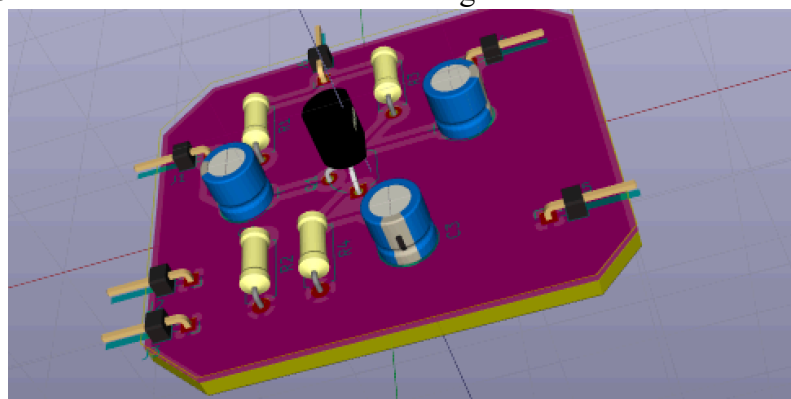




20. The edge will now be cross-hatched. Right click inside that zone and click fill or refill all zones. It should look something like this now.

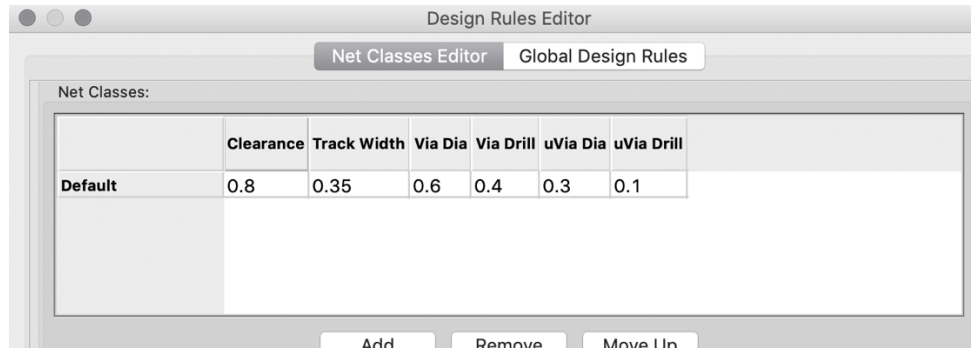


21. You can use 3D viewer to see what the circuit might look like when built!

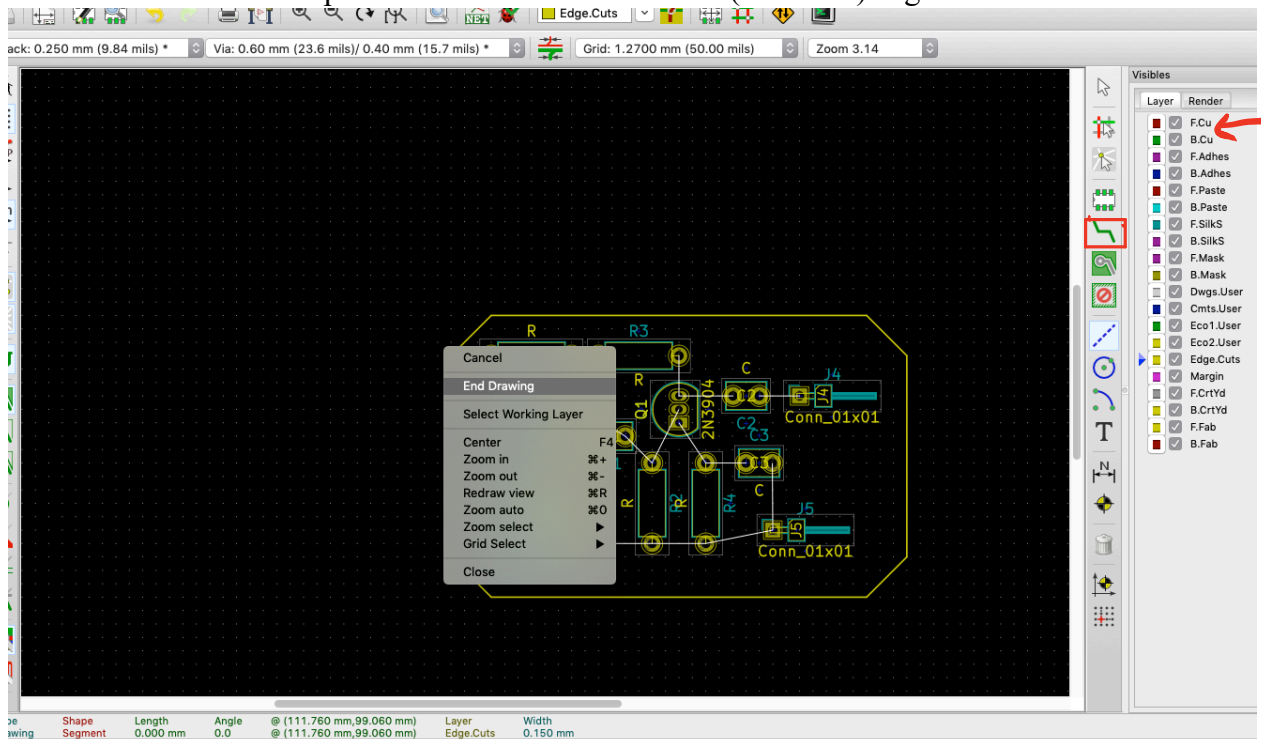


22. Now it's time to lay tracks. **BEFORE LAYING TRACKS:** change your design rules. Click Design Rules->Design Rules. These rules let you set clearance (important for our mills) and track size (also important). The clearance and track sizes are determined by our head sizes for the mill. The following will work for the 1/32 head on the Othermill. (Note we don't do vias).

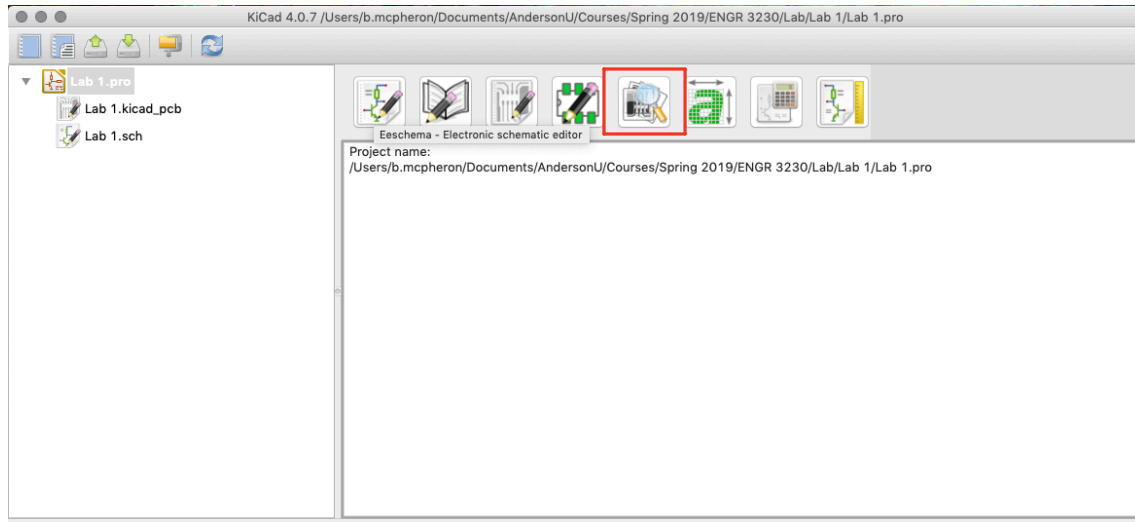




23. Click on add tracks and vias (red square on right), and layer F.Cu (front copper). Note that you will start a track by left clicking but end a track by right clicking and choosing end track. Continue this process until all the white lines (rat's nest) is gone.



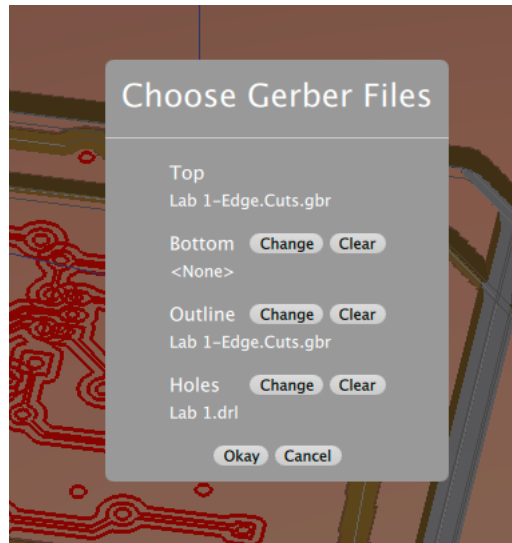
24. Click Tools->DRC. This will check for errors and violations of design rules. Choose 'Start DRC'. If there are no errors, great! Click ok. If there are errors, fix them and repeat this step.
25. **NOTE:** if you are planning on fabricating using our mills, you may need to change pad/hole sizes for drill outs before moving on. The tutorial for this follows the main tutorial in this document.
26. Go to File->Plot
27. Make sure the plot format is Gerber and choose the layers you want. In this example, you want F.Cu and edge cuts selected. Click Plot. This will provide files that will tell the Mill which places to mill out.
28. Click Generate Drill File. Choose Gerber file format, then click Drill File, then close.
29. Click close on the plot.
30. Click close on cypcb. Make sure you save as you exit.
31. To check your Gerber files, you can use GerbView (red square).



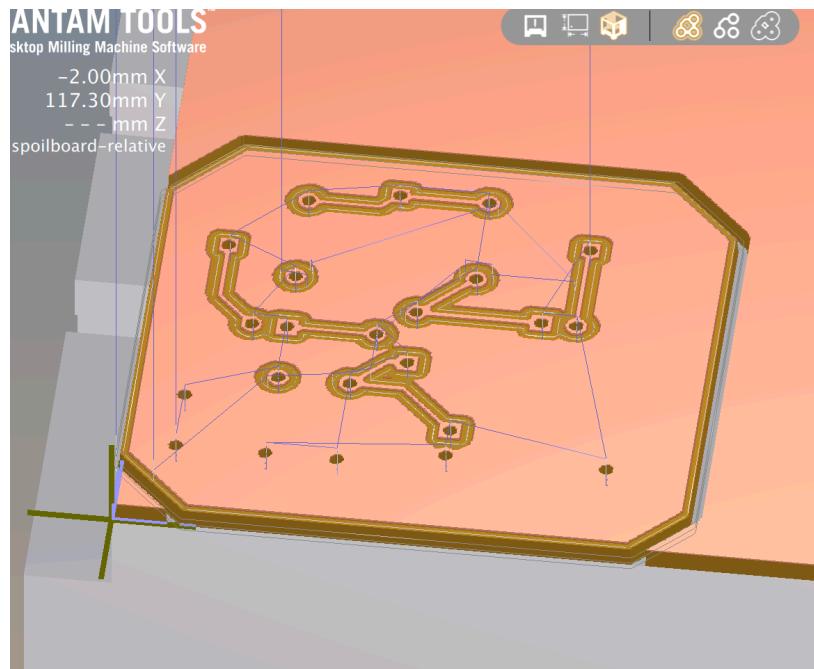
32. Click File-> open Gerber file, and choose the Gerber files you just made. You can check that things look ok, but usually you are ok at this step.

## OtherMill Software

1. If you haven't yet, download OtherPlan (now called Bantam Tools). This will let you see how things will look for the mill. [https://resources.bantamtools.com/software-download?\\_ga=2.244515973.1203592963.1542040477-356876006.1542040477](https://resources.bantamtools.com/software-download?_ga=2.244515973.1203592963.1542040477-356876006.1542040477)
2. Open Bantam Tools.
3. File->Open File. Choose the F.Cu Gerber file for the top. Choose Edge.Cuts for the outline, and the .drl file for the holes. Click ok.



4. Note, depending on your layout, you might need to further spread pieces out, so the end mill can actually manufacture the PCB. Anything in red is not possible to mill with your current bit. You can change the bit, but we would prefer to use 1/16 (very difficult to make this possible) or 1/32. Here is what a successful (no red) part looks like.

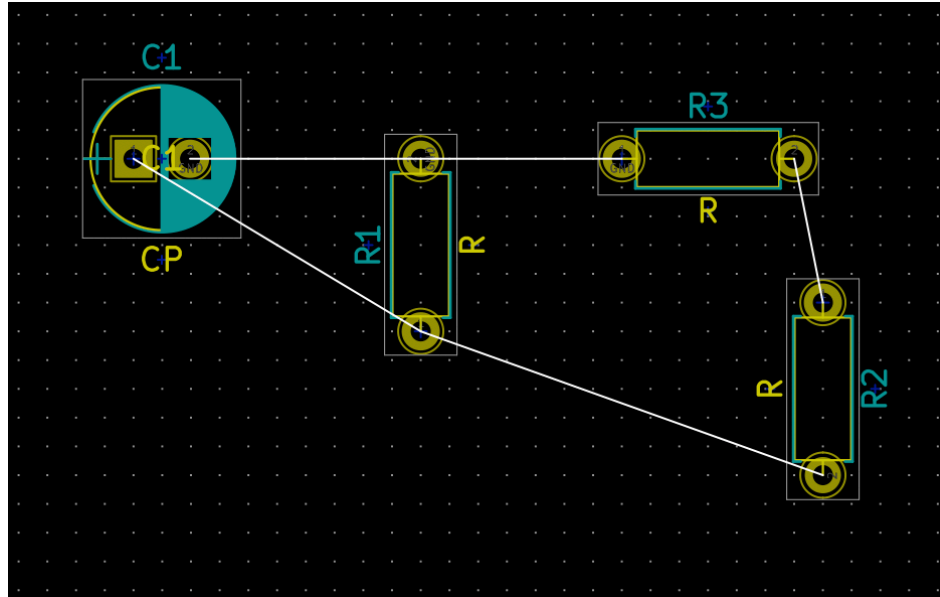


5. Once you have a good part, mill! The PCB should have two-sided tape used to secure the board to the bed should cover the entire bottom of the board. Please watch the mill while

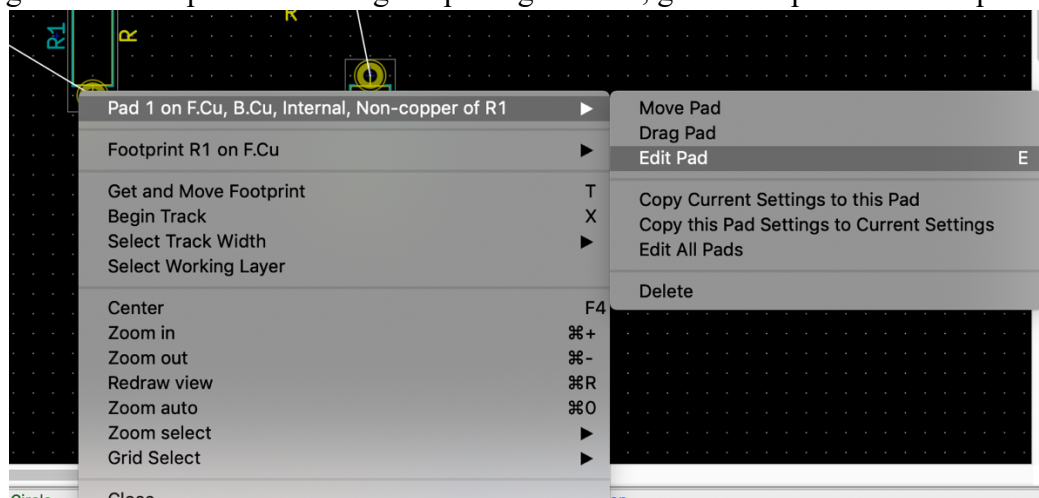
operating to make sure nothing unexpected happens such as breaking a tool or the board coming off the table.

## Changing Pad/Hole Sizes

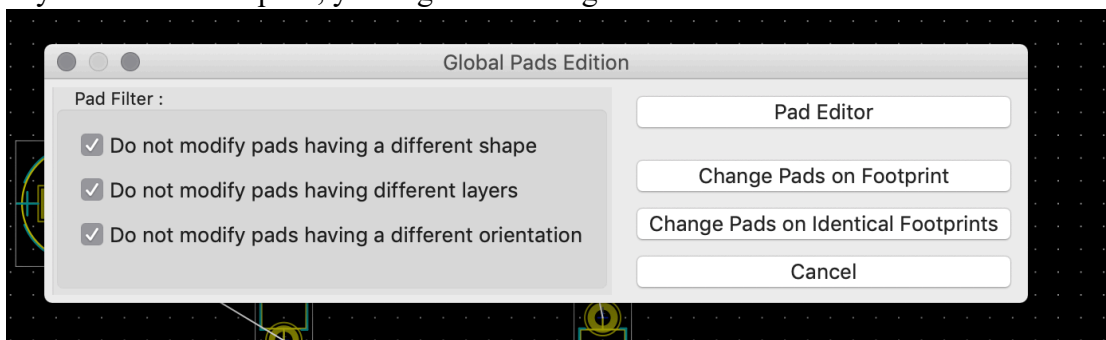
1. Look at your circuit in Pcbnew



2. Right click on a pad. Following the pad right arrow, go to Edit pad or edit all pads.

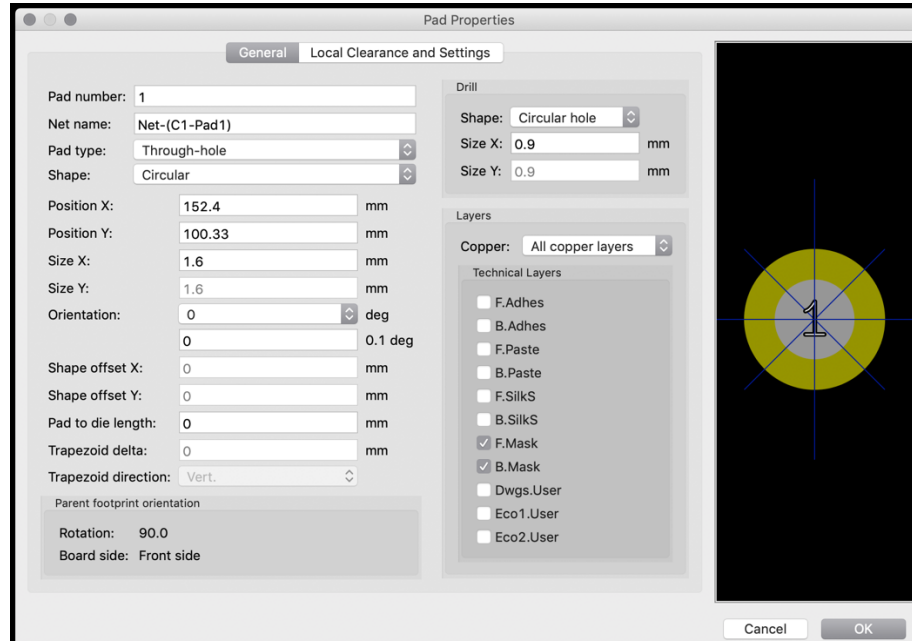


3. If you click edit all pads, you'll get something that looks like this

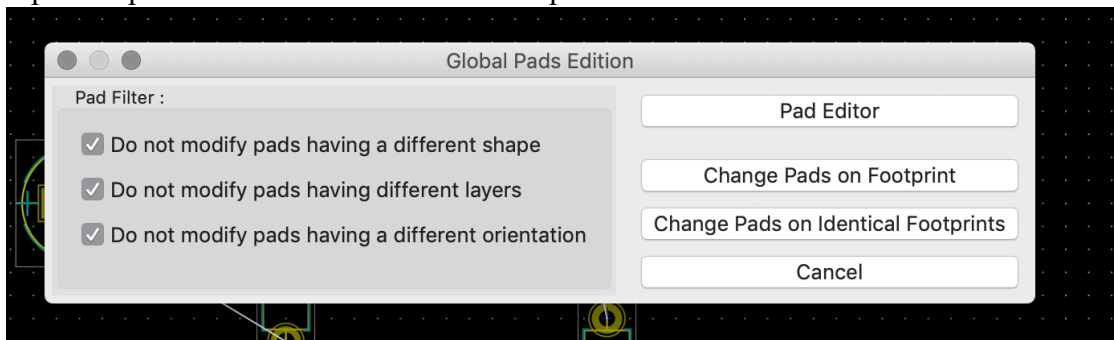


4. Click on Pad Editor. If you just chose edit pad in step 2, this is where it will take you.
5. You can now change the pad size and hole size. For the 1/32 head on the Othermill, I've found that the holes need to be  $>0.8$  (I've found 0.9 works). This is found in the section

called 'Drill' on the right-hand side. If you want to increase the pad size, you will change 'size x' on the left-hand side. Click ok when done.



6. If you originally clicked Edit All Pads, it will return you to the following menu. Otherwise, repeat steps one and two and click edit all pads.



7. Click Change Pads on Identical Footprints. This will change all of 'the same pads' on each identical footprint.
8. Repeat for all unique pads on all unique footprints.