# Step 4: Laying out the PCB

1. Open your schematic from step 3 and click ![SCH BRD] generate board. Accept the warning that will pop up.

2. Similarly to when we created the MX1508 package, we now have to place our components. Luckily there are only two components in our PCB that require an exact location, which are the header pin holes, as these need to line up with the ESP board that will slot into them. So let's begin by placing these accurately.

   The headers need about 2mm clearance from the center of the hole to the edge of the board, so go ahead and rotate "J1" to have the J1 label on top and move it to position (2.15, 41.15). By importing the ESP12E_DEVKIT library from our development board's Github I was able to measure the distance between the pins to be exactly 22.86mm, so place the second row of header pins that far to the right of the first: (25.01, 41.15).
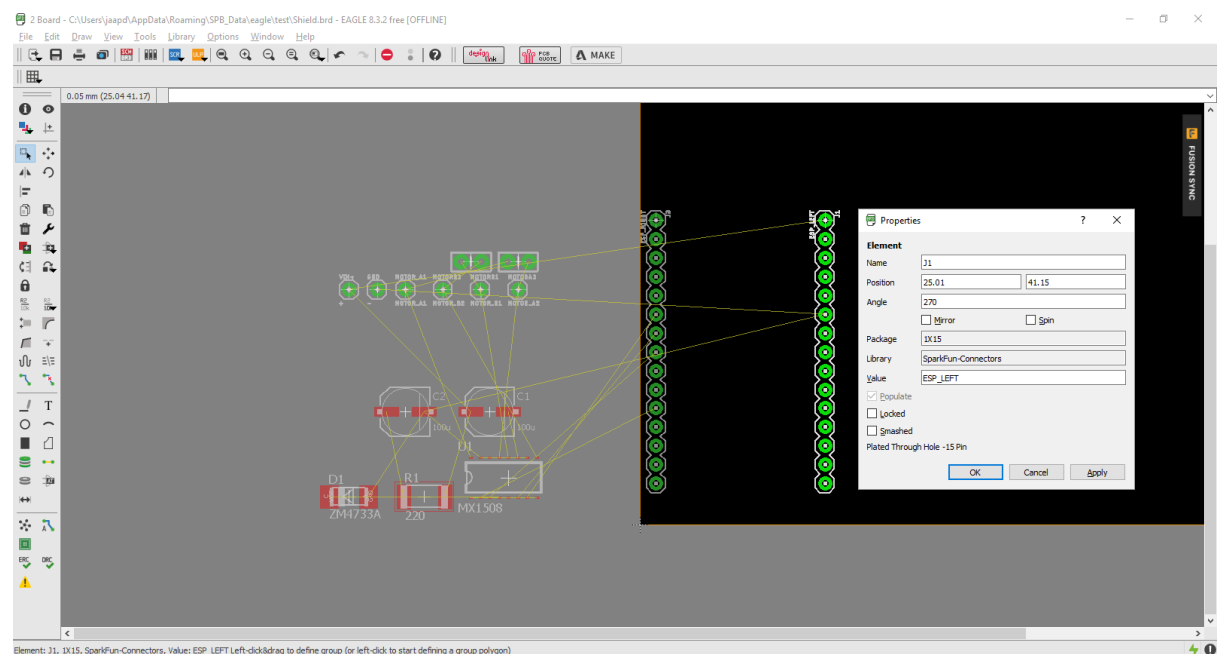
3. The orange line that divides the black from gray is the board outline. you can click and drag the corners using the move tool or position them accurately using properties. I recommend you set them accurately using the following coordinates for the corners: (0,0) (27.1, 0) (27.1, 46.84) (0, 46.84)
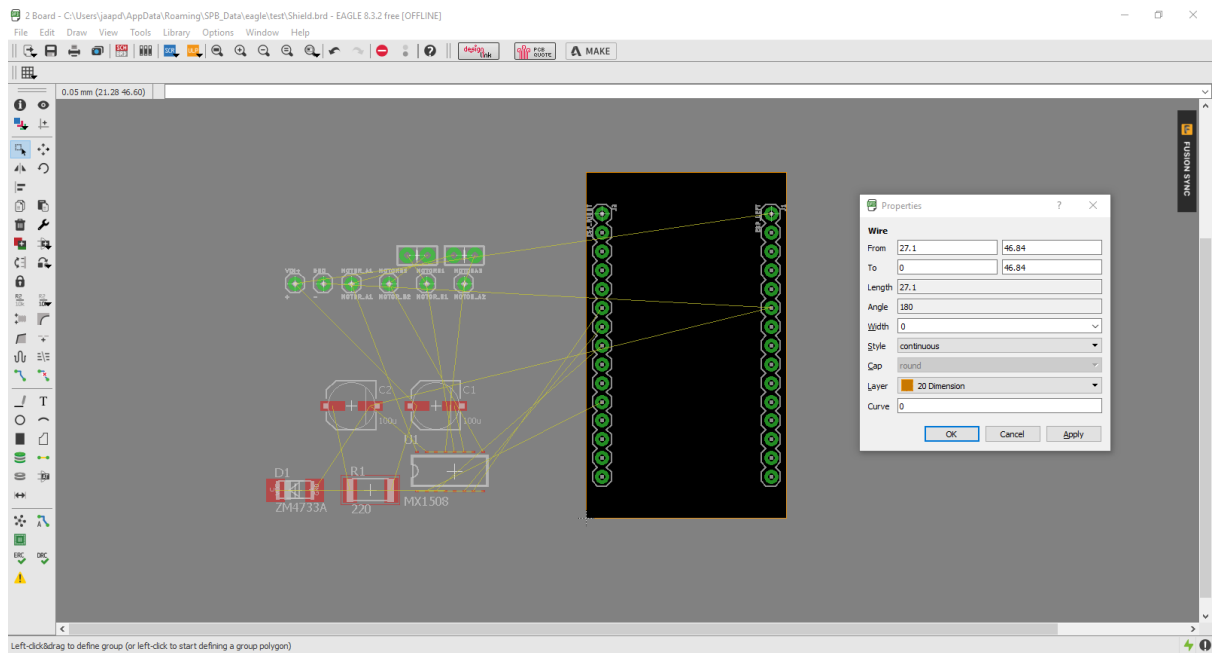
4. Now comes the hardest part: placing the components and routing the traces. How you lay out your board is left up to you. I will guide you through the process.

The yellow lines you see are called **airwires**. They are straight lines drawn between two points that need to be connected to give you a rough idea of how things are best placed. Clicking ⚙ Rat's nest will make the program rethink the airwires as there are often different ways to connect components. Click this frequently as you move things around to see how it affects the airwires.

First, lets turn off airwires for the ground connections as we will later make an entire ground plane, making these redundant.
Right click any of the airwires coming from a ground connection, click properties and check "Airwires hidden" box.
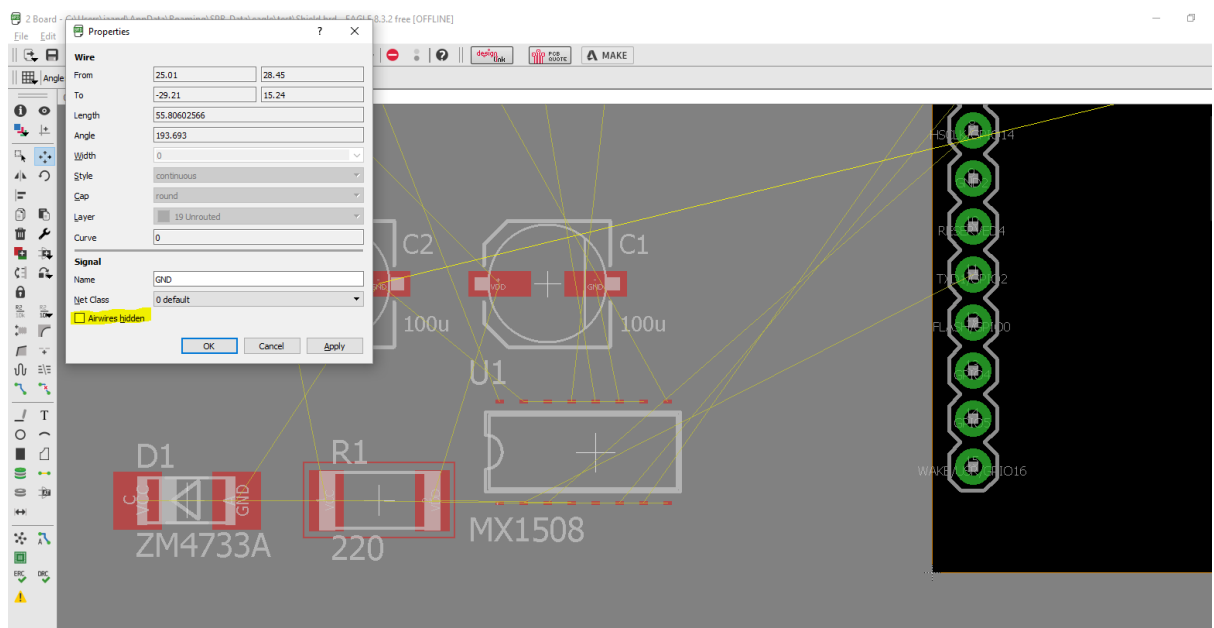
5. Start placing all the components on the board using the move tool. Try to minimize the airwire crossing and always leave a good amount of space around a chip like the MX1508 with a lot of pins that all require separate traces.
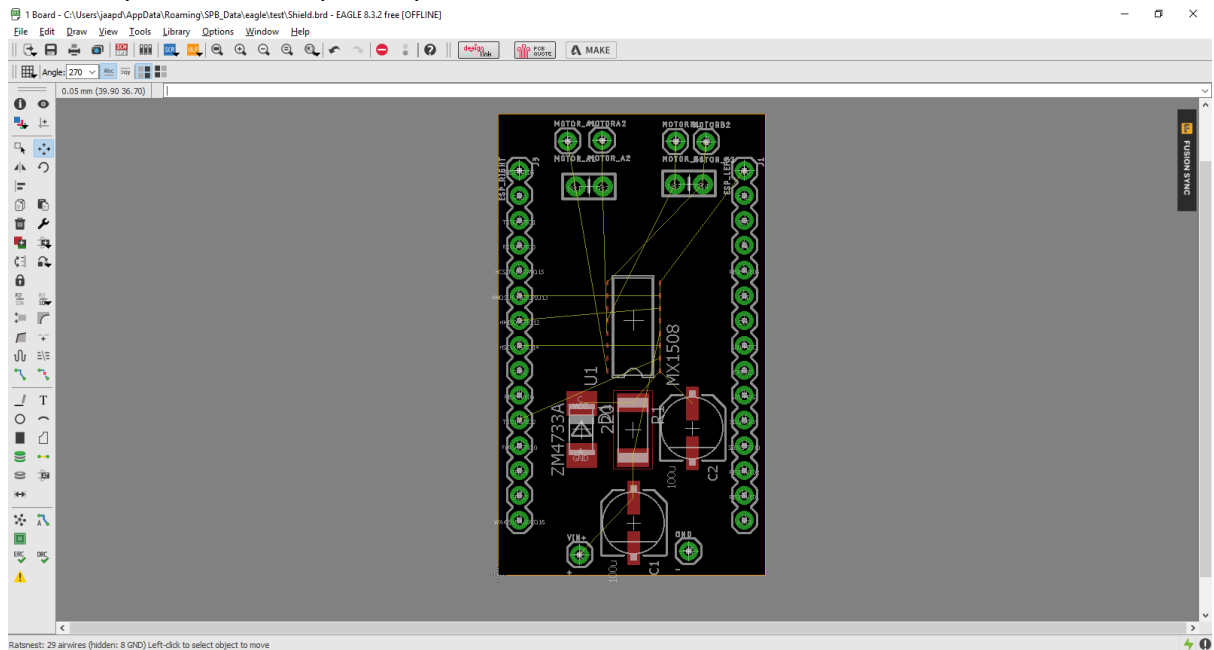

Something like this?

6. You can click autoroute and give it a try although it is not recommended.

7. Click the Route button and set the width at the top to 0.254mm. For this circuit really any trace width will work, however, very small widths are more prone to breaks.

8. Click a point and start routing!
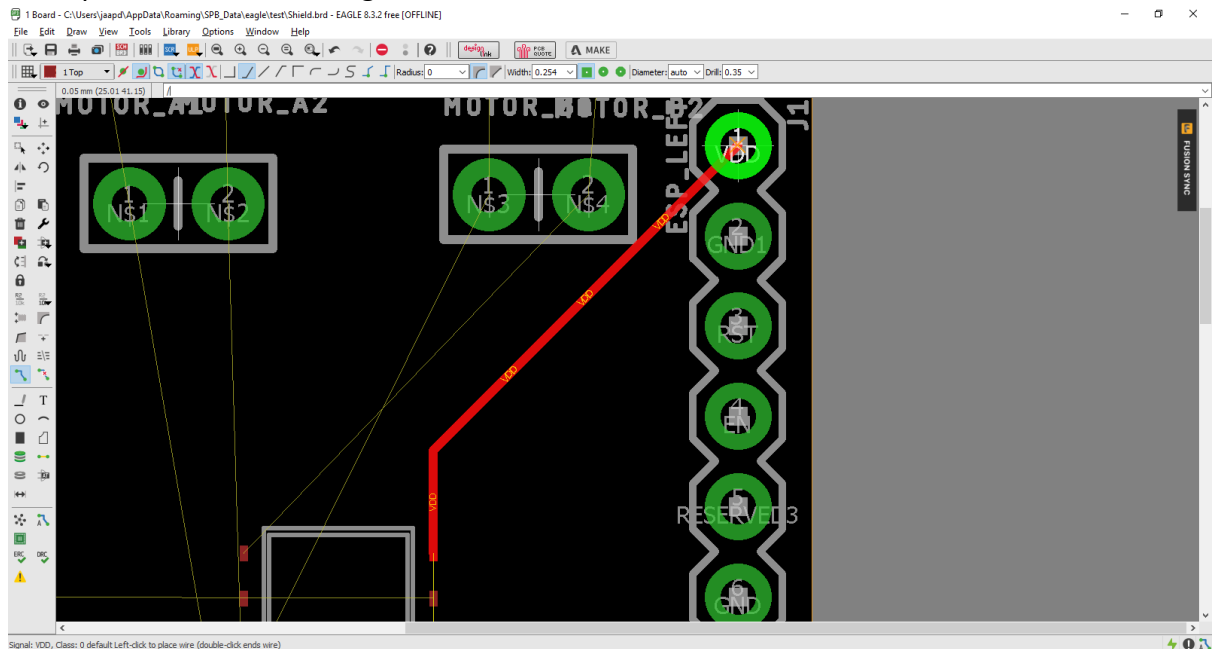
You can draw a trace on either the top or the bottom of the board. Switch between the top and bottom layers by pressing spacebar while having the route tool selected.

Pressing the spacebar while actively routing something will create a via. Vias are a small hole in the board that connect the top and bottom layer.
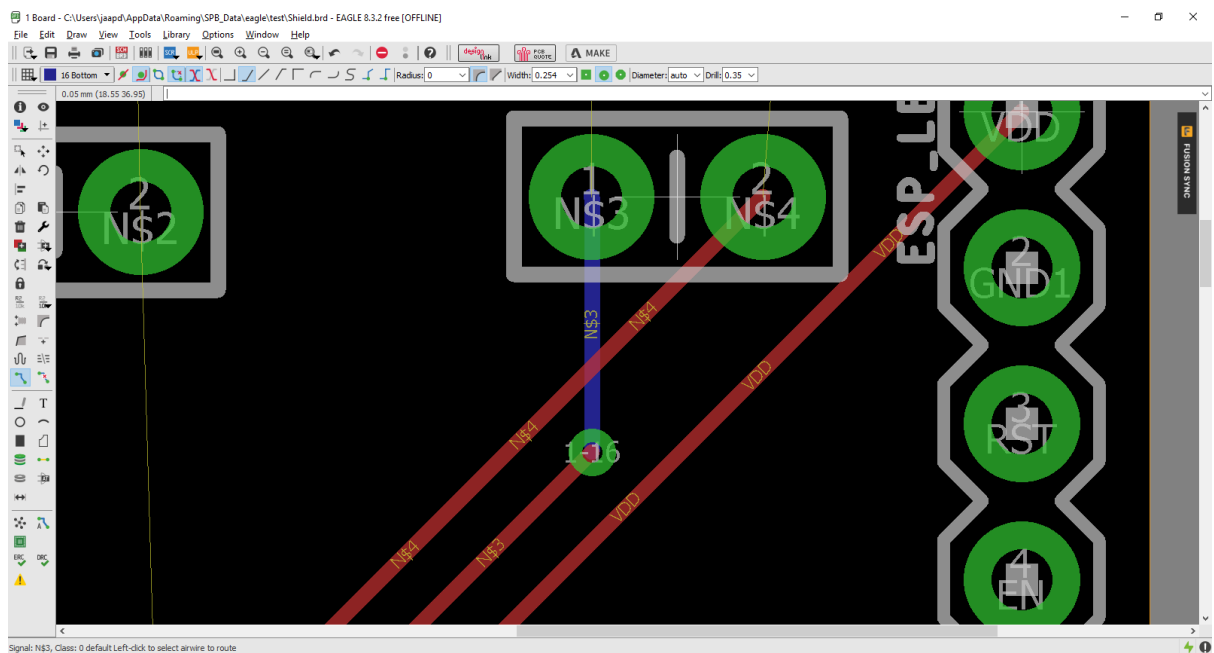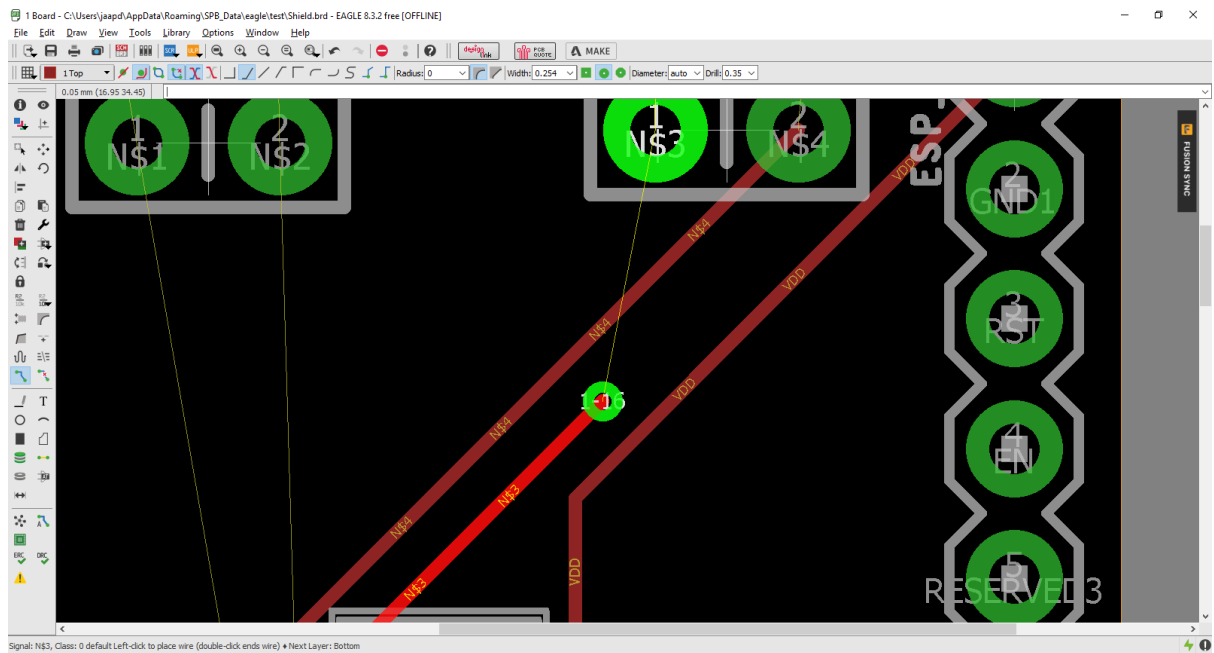
Wow smart!

9. Although it really won't make a difference for our little project, I should probably tell you about some design rules at this point

    I.   Avoid 90-degree angle traces. These cause EM interference as the signal reflects.

    II.   Run top and bottom traces perpendicular to reduce cross-talk.

    III.   Try to minimize vias as they also cause some EMI. One way of doing this is by using through-hole components as a via where possible.

10. If needed, use ripup to delete traces. Type ripup; into the command line at the top to delete all traces.

11. This is the time to add any extra features you may want. The example in this tutorial is for demonstration purposes only so I didn't spend much time on it. If you look at my final design you will see I added mounting holes in the corners and some extra vias to the GPIO pins in case I want to solder onto them in future.

12. When you have routed everything but the ground connections it's time to add a ground plane.
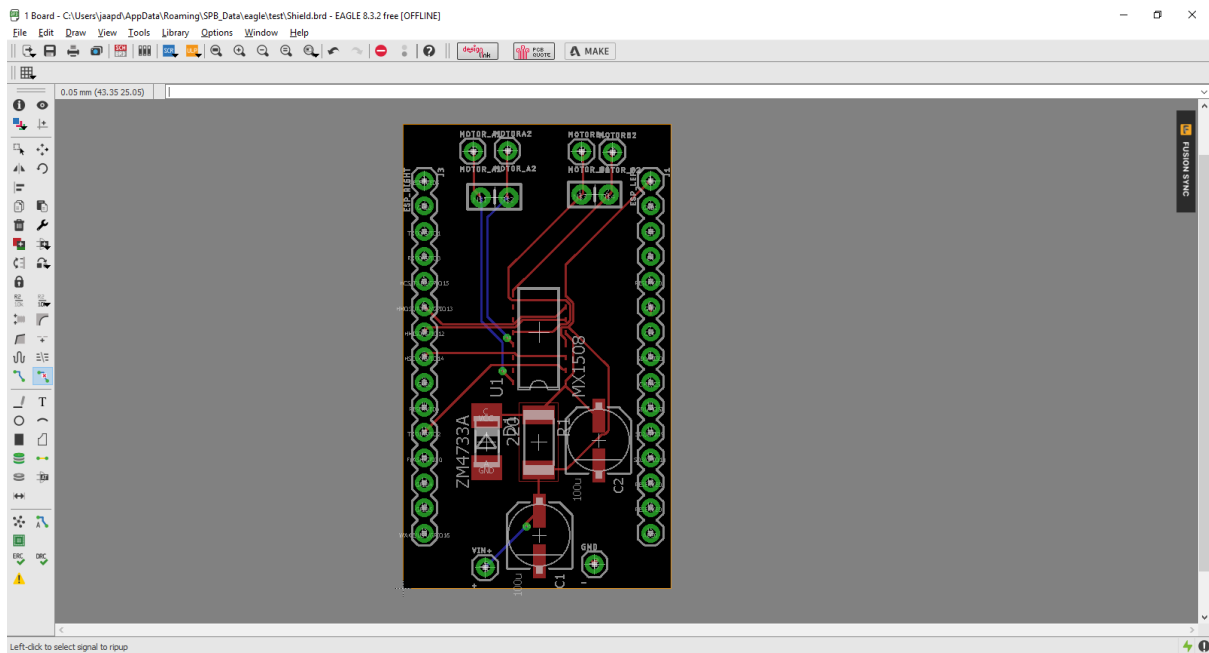
You have a few choices: often both top and bottom of the board are poured to be a ground plane, sometimes one side is used as ground, the other as power and in professional designs many planes are created on both sides for maximum coolness.
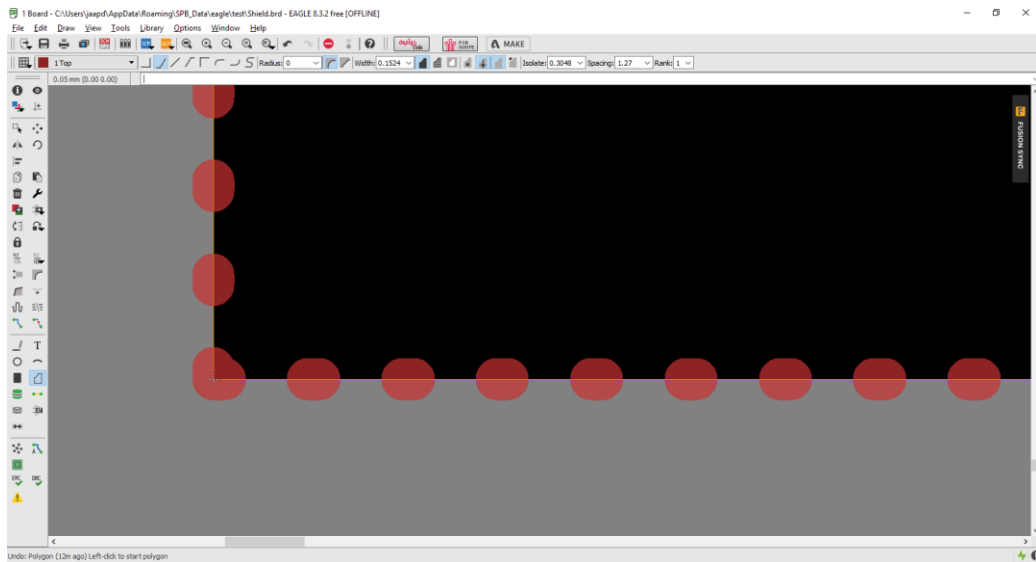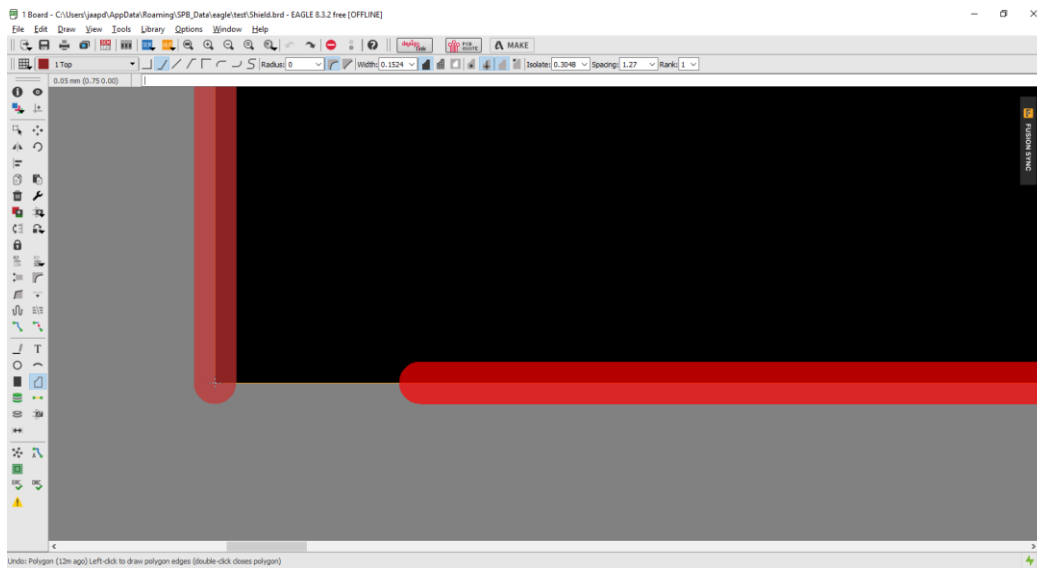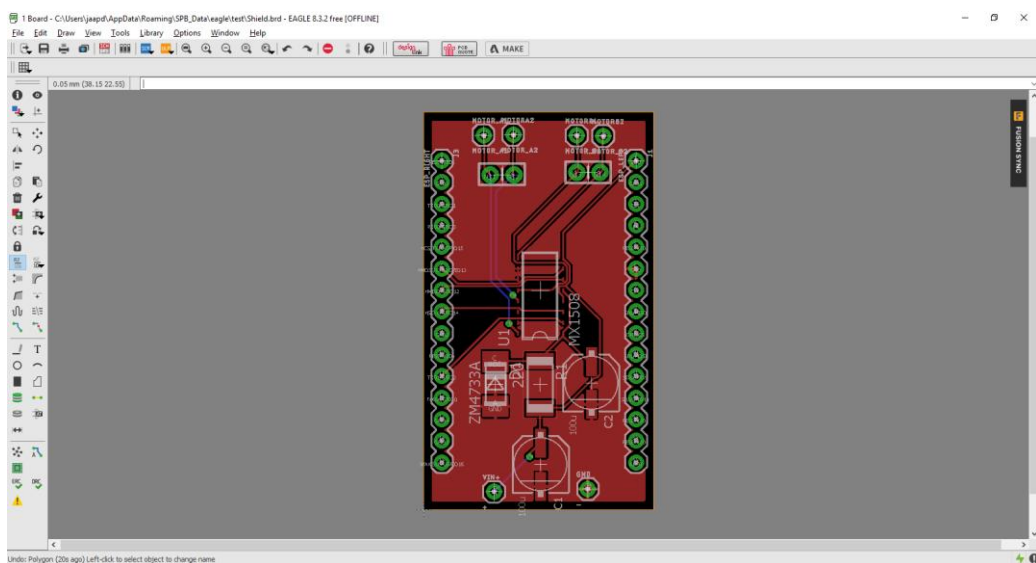

Very smart   [http://dangerousprototypes.com/blog/2012/07/18/eagle-polygons/]
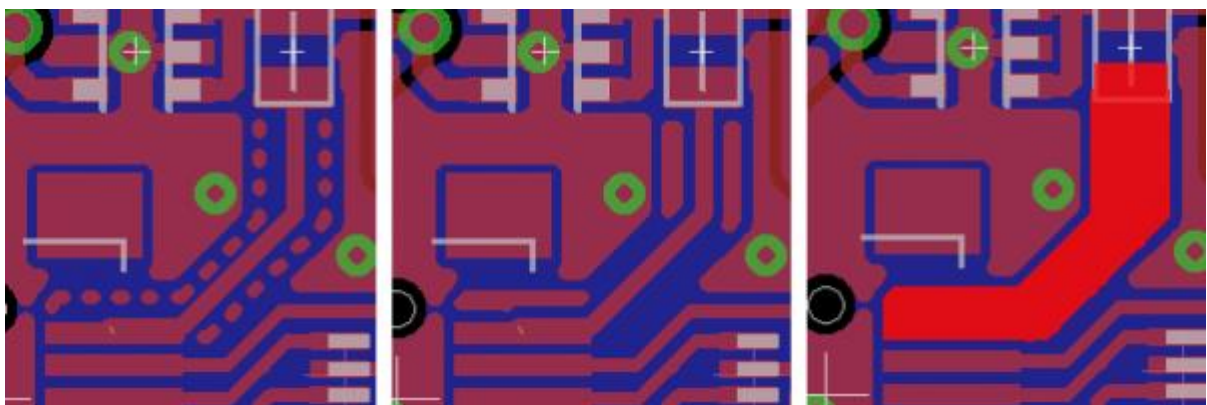
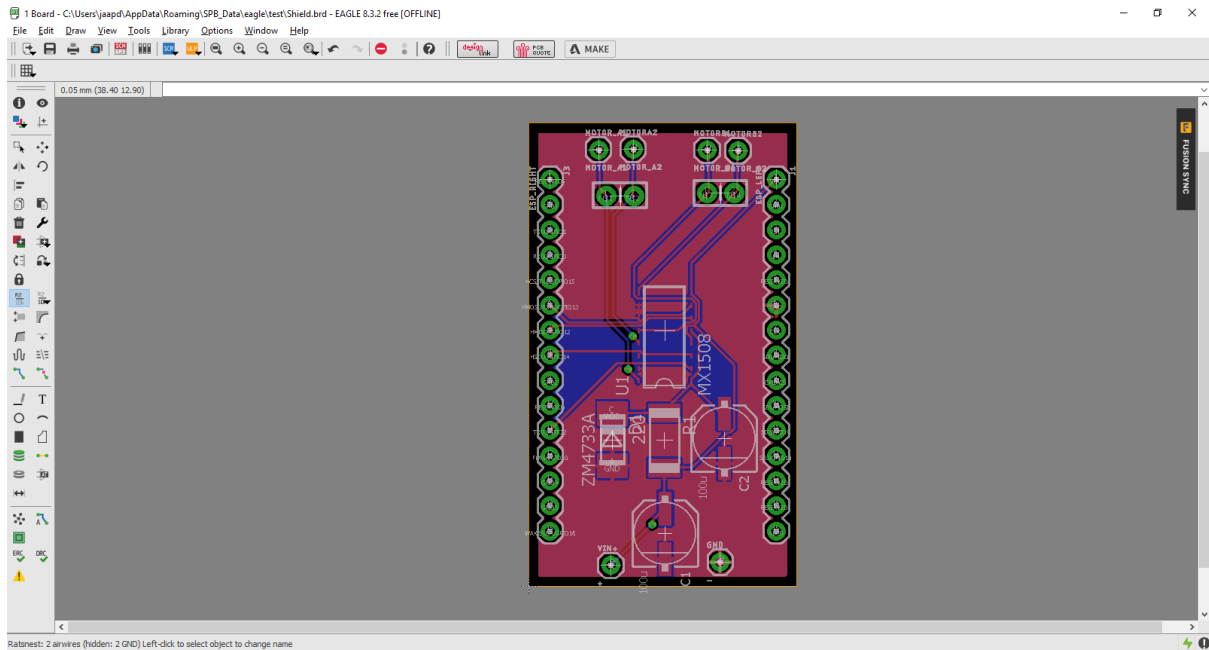Click the polygon icon, select the 1 Top layer and set Isolate to 0.3048.

Draw over the border of the board with the polygon. Make sure to match the last point that you click with the first to see the line turn dashed.
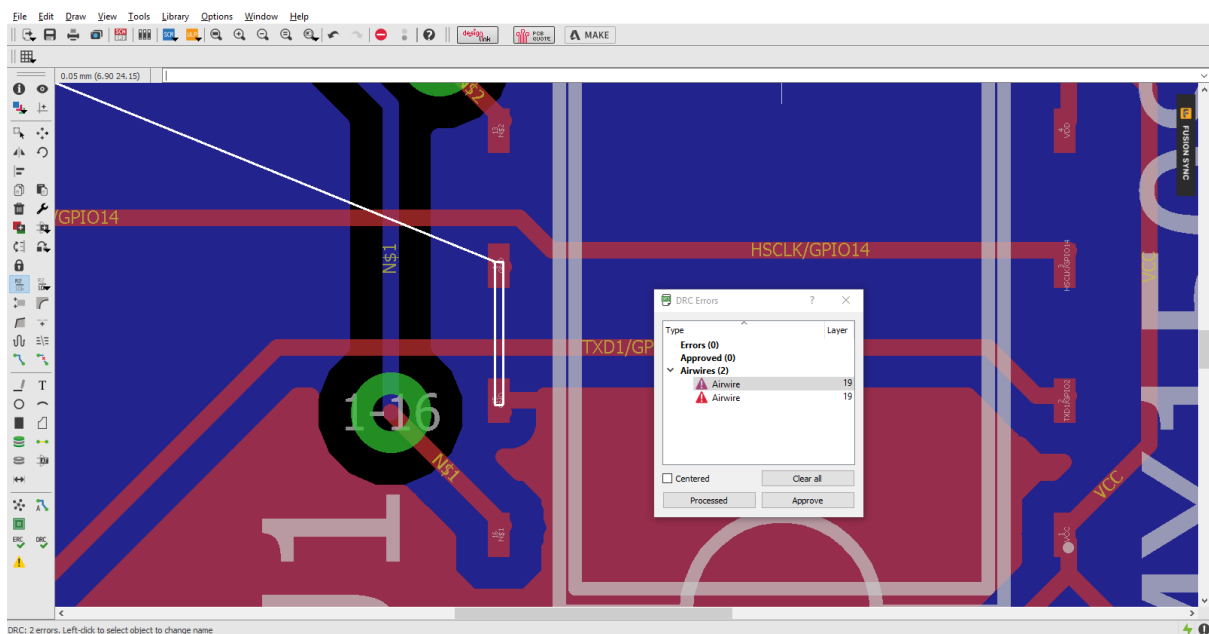
13. Name the polygon GND and click rat's nest

14. Repeat for the bottom layer.

(Note these planes will look like they disappear every time you close and open EAGLE. Click rat's nest to make them reappear.)

15. Run DRC to check for errors.

In my case, the top ground plane didn't end up covering these two ground pins, so I had to make some adjustments.

16. Now, let's "tent" our vias, meaning, having them covered up in solder mask. This is common practice as it gives the board a more professional look and leaves more room for other silkscreen labels.

Click ▣ Layers and click tStop and bStop to highlight them and select Apply to make these layers visible.
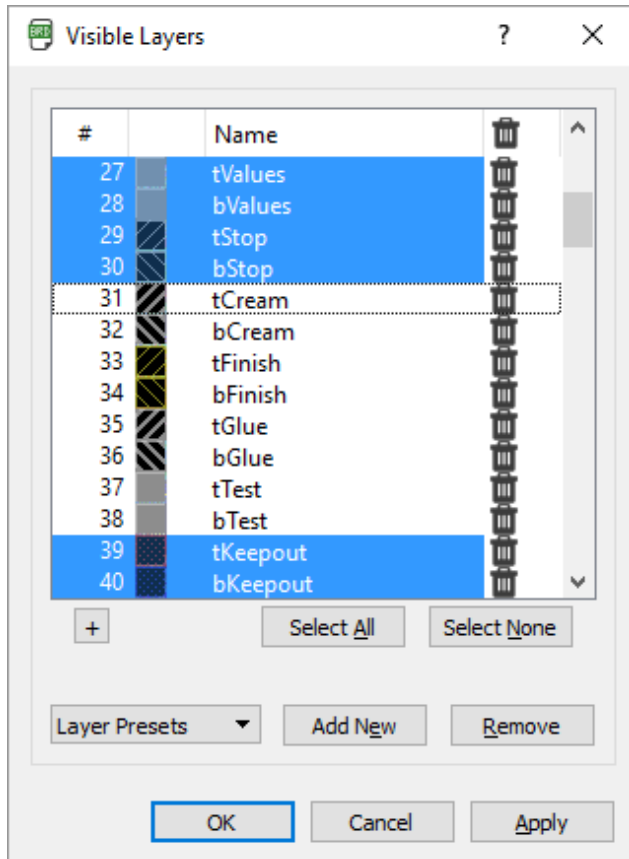

You should now see white hatching on vias and pads. This indicates where soldermask will not be poured.

An easy way to tent all vias is to click DRC -> Masks and enter 25mil as Limit.
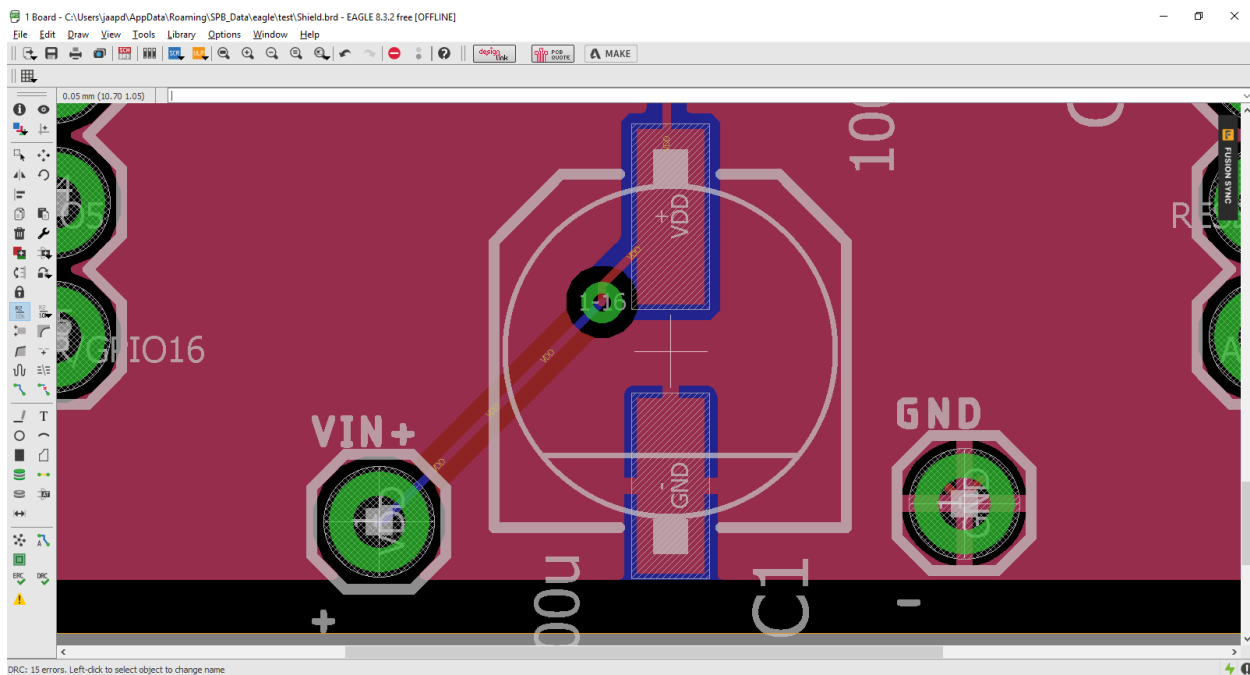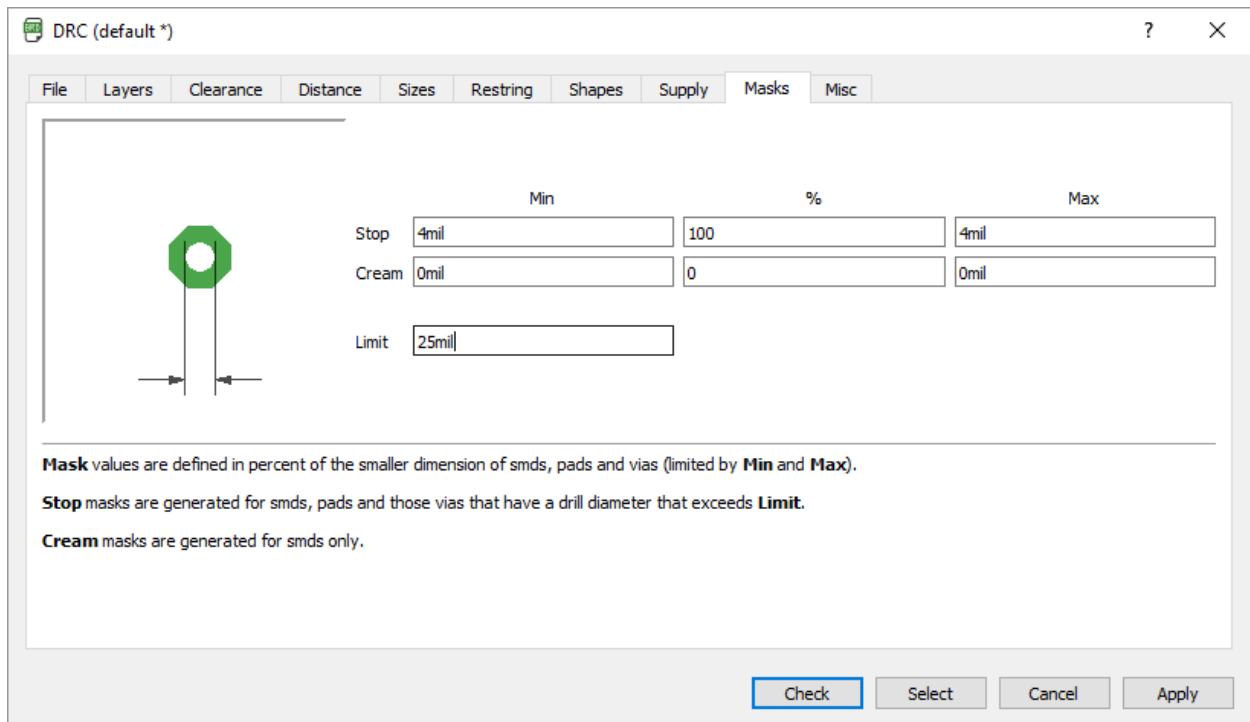
The via is no longer hatched but the pads and holes are.

17. Arrange all the silkscreen component labels to be visually appealing by using the
Smash tool and clicking a component. Now you can move around the name and value tags
using the move tool.
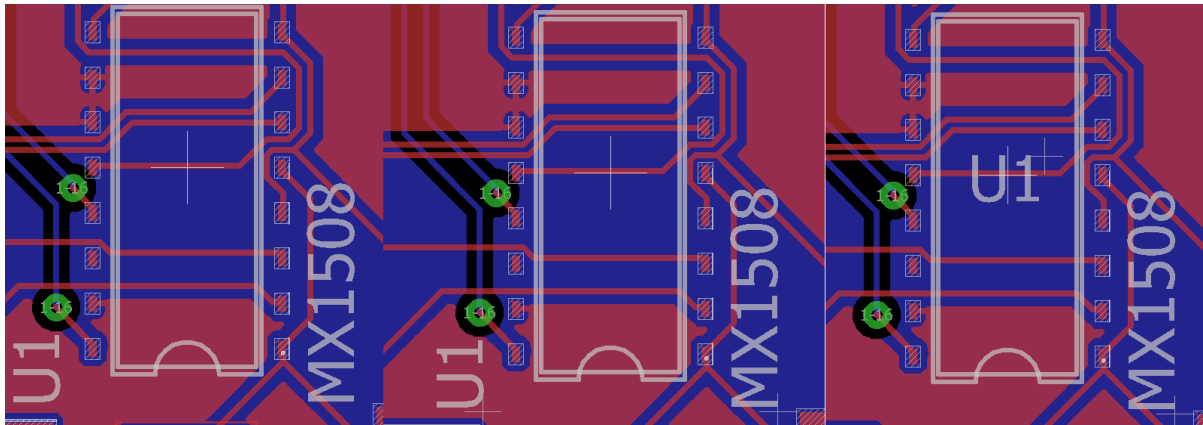
By default, value tags are not printed to the silkscreen layer, so you can ignore them.

18. If you want to add some flair to your board you can use any of the drawing tools and the

T text tool to add silkscreen drawing or text to layers 21 tPlace for the top of the board and 22 bPlace for the bottom.

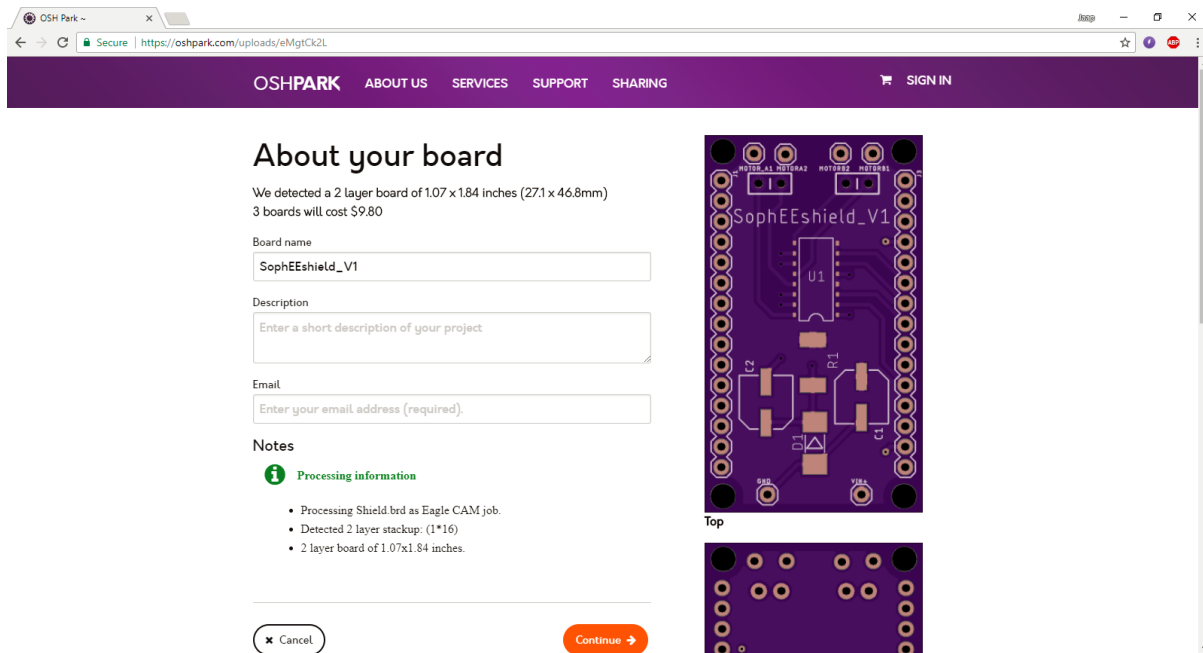Let's add some text to the board, for example:
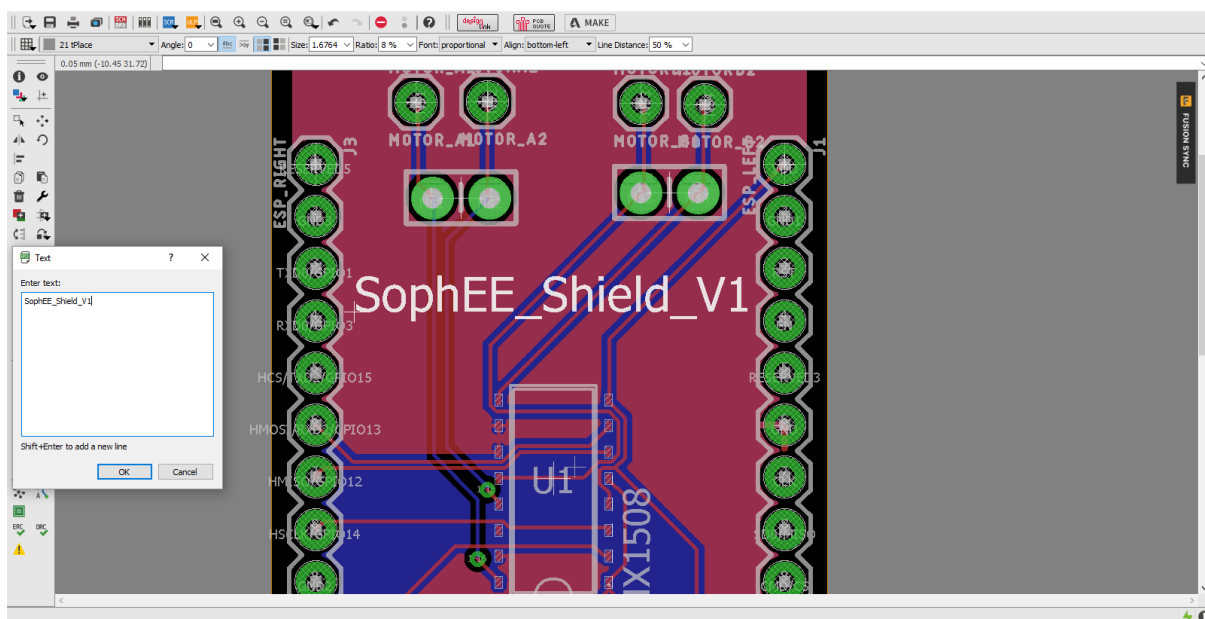
or anything you may want to write. Just make sure it's on the correct layer!

19. Is the board ready to manufacture? Let's find out. Run DRC one last time to check there are no errors, double check you didn't reverse any pins, there's no overlapping silkscreen and no spacing issues that may arise when trying to solder the components.

20. Go to https://oshpark.com/ and upload your .brd file

Oshpark is great as a preview of your printed board is immediately generated.

This is very useful to see if everything looks the way you want it to. If not, go back and fix it!

That's it for now. I hope to see some creative and/or professional designs!

References/Further reading

http://web.csulb.edu/~hill/ee400d/Technical%20Training%20Series/11%20PCB%20Layout%20with%20Eagle%20CAD.pdf

Autodesk "Layout Basics" series
https://www.autodesk.com/products/eagle/blog/pcb-layout-basics-component-placement/

https://www.autodesk.com/products/eagle/blog/routing-autorouting-pcb-layout-basics-2/

https://www.autodesk.com/products/eagle/blog/design-rule-check-pcb-layout-basics-3/

Sparkfun "Better PCBs in Eagle"
https://www.sparkfun.com/tutorials/115