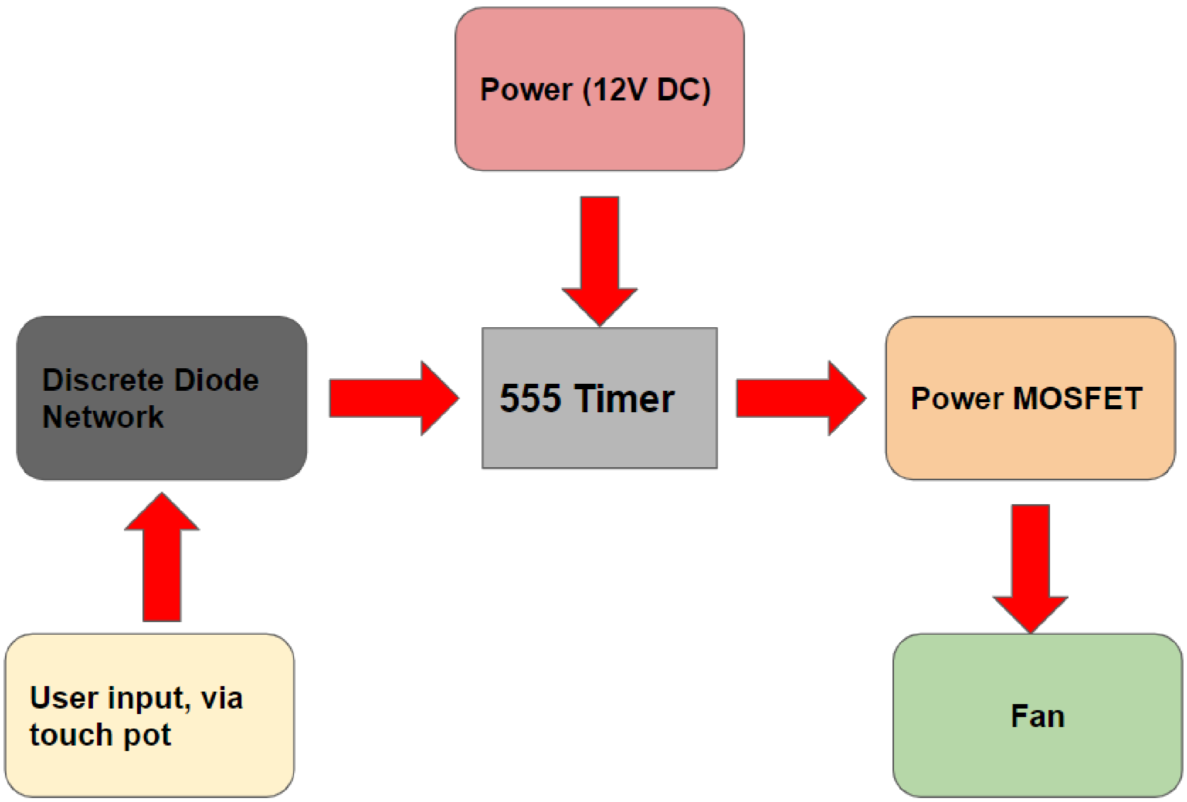
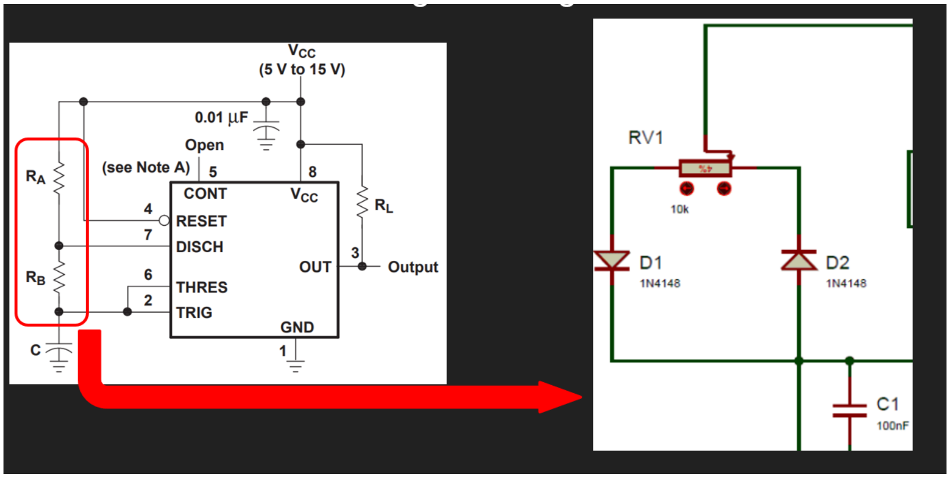
Design Overview

My project focused on using the 555 timer as a variable PWM output to drive a fan motor, while varying the motor's speed through a potentiometer input. The purpose of this design is to create a fan that can be used as a fume extractor for soldering that has a rotational speed controlled by the user. This would be useful in situations where one might want strong fume extraction when actively soldering, but only a mild extraction force when the iron is idling.

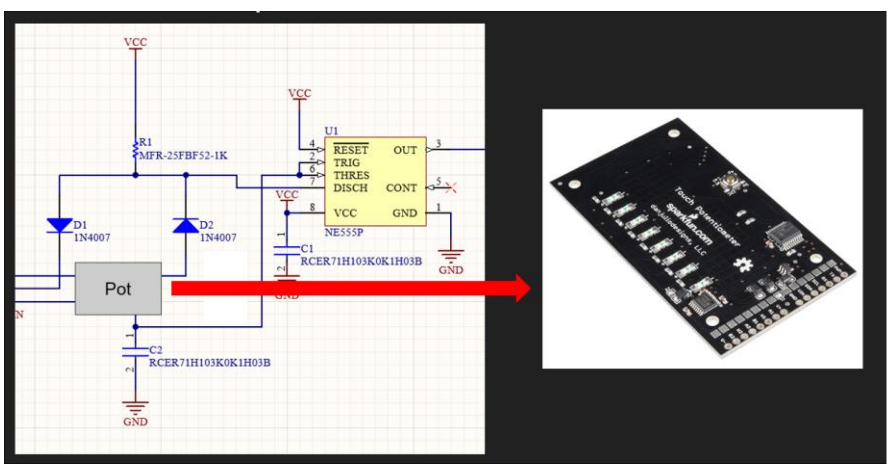
Here's a block diagram of the overall operation:



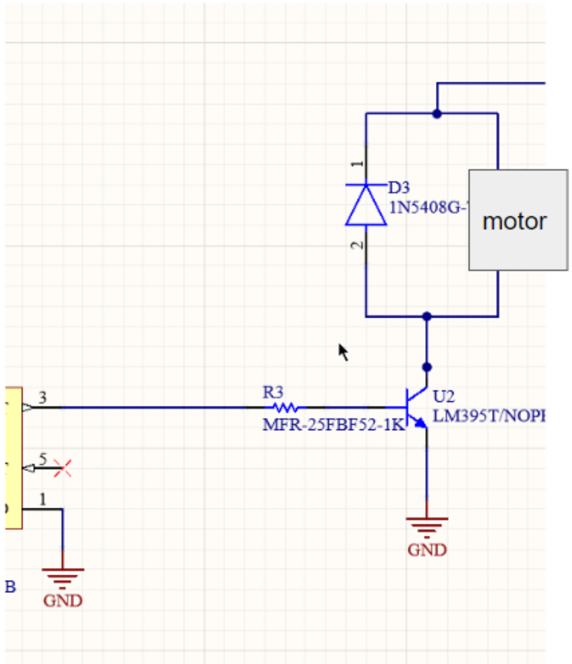
The fan driver design did not actually begin as a project found online – I only pulled the diode network used to create a PWM output from a [PWM LED driver](#) tutorial, which replaced the simple voltage divider, as shown below.



The 555 timer is configued in an astable mode, but with the diode network modification as desceibed above. I decided to use a [snazzy touch potentiometer](#) to vary the resistance in the diode network and adjust the duty cycle of the PWM output. The resistor, diodes, and capacitor 2 are used to configure the working mode of the 555 timer to be a PWM output. I also included a capacitor on the input of VCC for voltage regulation.



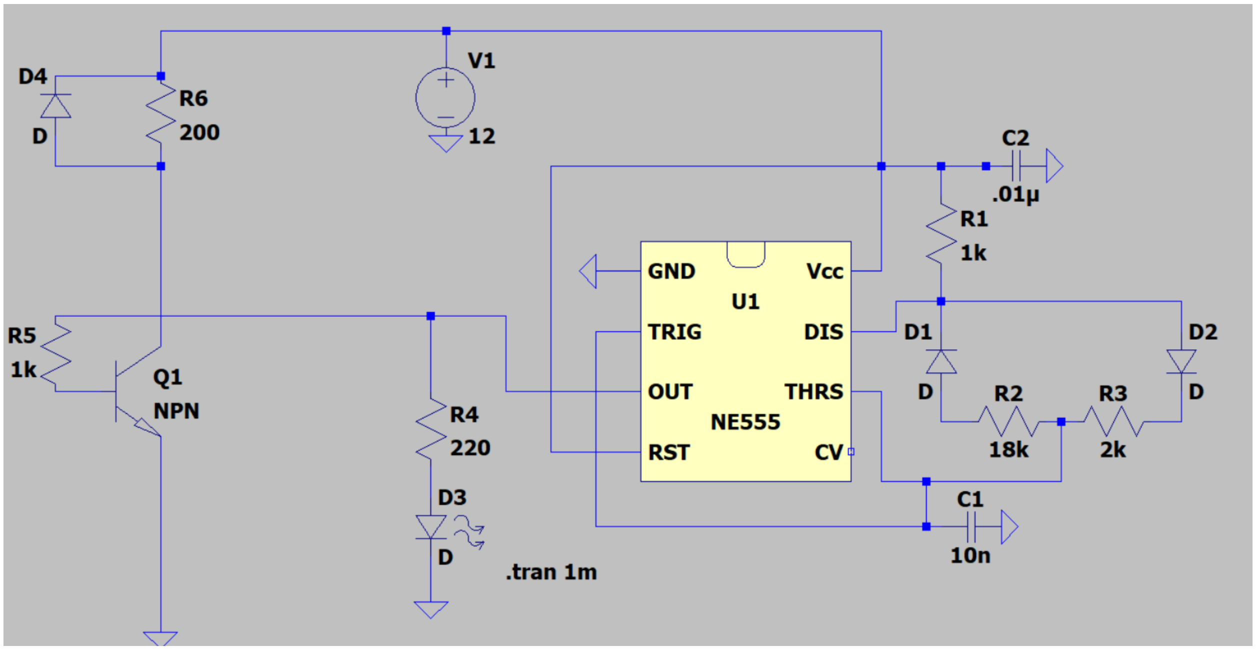
I then connected the output of the 555 timer to a MOSFET to drive the motor:



The resistor limits the current of the BJT, and the BJT acts as a switching mechanism to modulate rotational speed. The diode is a flyback diode for any inductive motor load.

Design Verification

I verified my design through LTSPICE. Below is an image of my spice schematic:

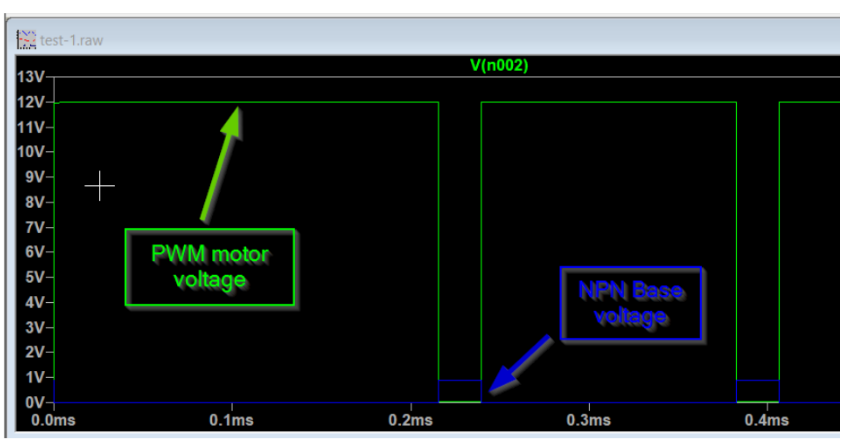


Of particular note – I simulated the fan using a 200 ohm resistor. The datasheet specfied that the fan pulls 60mA, so I figured the resistance from dividing the 12V input by the 60mA current.

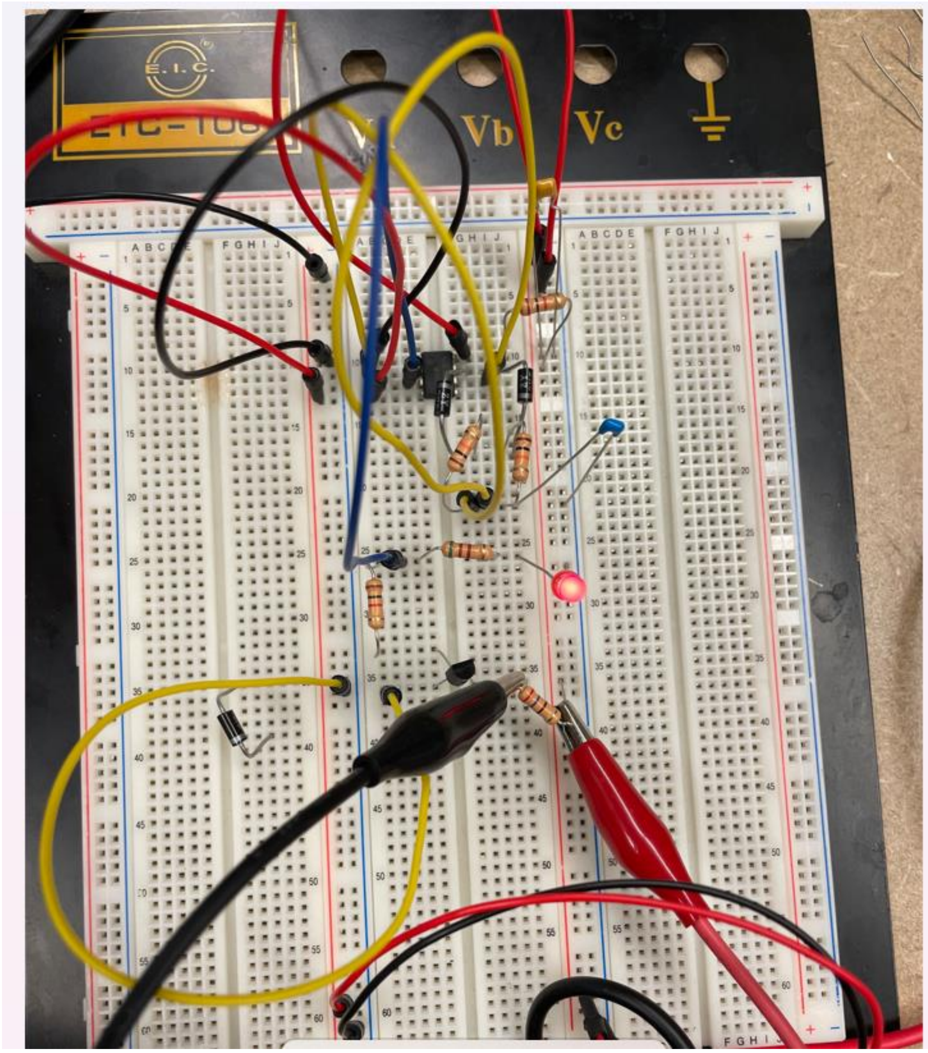
You'll also see an LED – I toyed with the idea of adding a PWM-driven LED, but I eventually abandoned the idea because I already had LEDs on the touch pot and didn't want to integrate more wires into my enclosure.

Finally, I simulated the digital pot with two simple resistors, and changed their values by hand. I used a steady-state DC analysis, as that is what I cared about – the fan's steady-state.

When simulated, SPICE gave the following waveform:

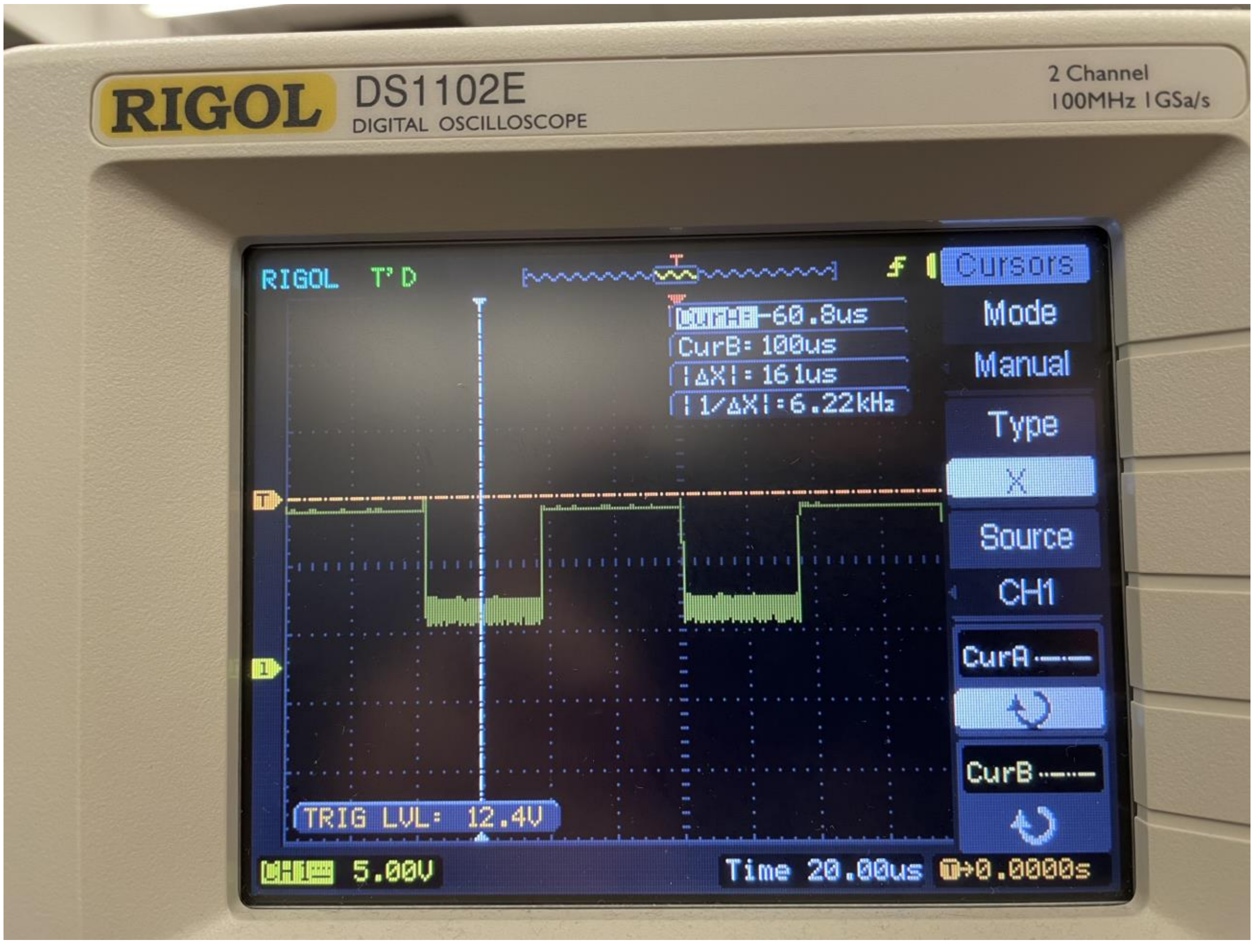


I also breadboarded the circuit, as shown below:



I used a 12V DC power supply as an input into the circuit.

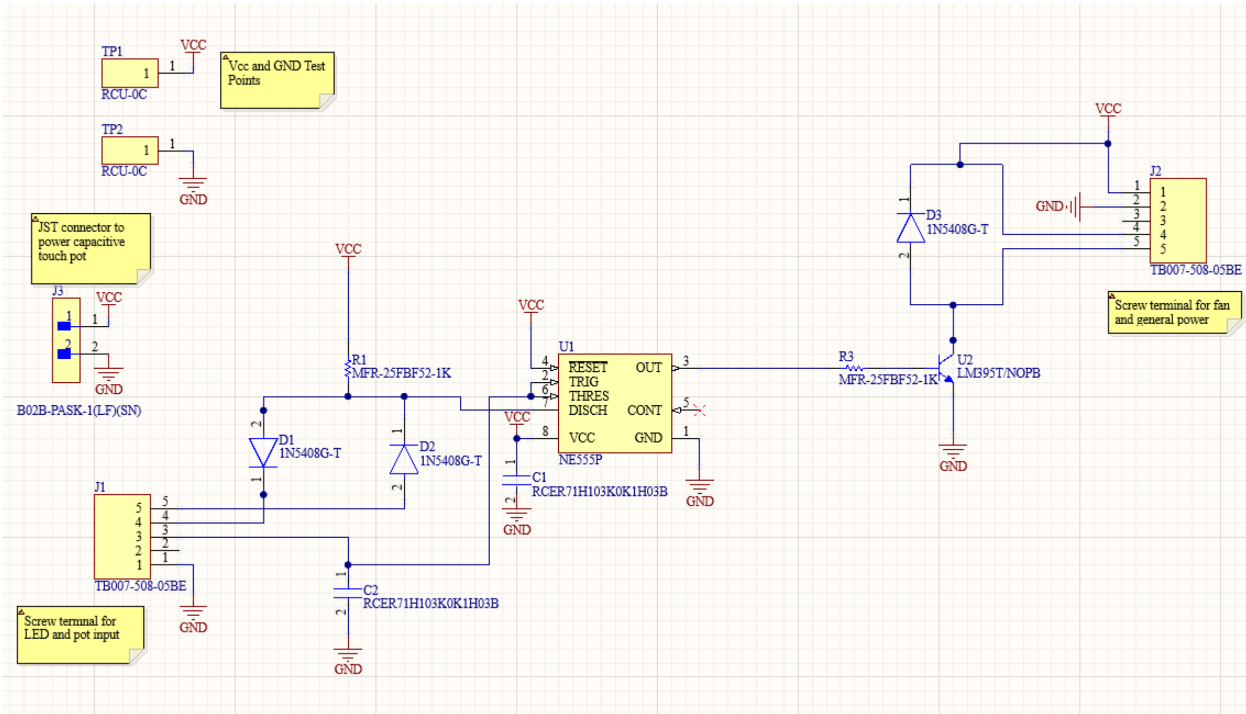
I used the same resistors to simulate the motor and digital pot as those in the spice schematic. When measured on the oscilloscope, the voltage across the fan (200 ohm resistor) was as follows:



I didn't verify the fan spinning – I was more worried about verifying the PWM output at the base of the NPN transistor. I ran tests with multiple resistor pairs (2 10k ohms, 1 15k ohm and 1 5k ohm, 1 1k ohm and 1 19k ohm, etc.) that added up to 20k ohms, which was the resistance value of the digital pot.

Design Implementation

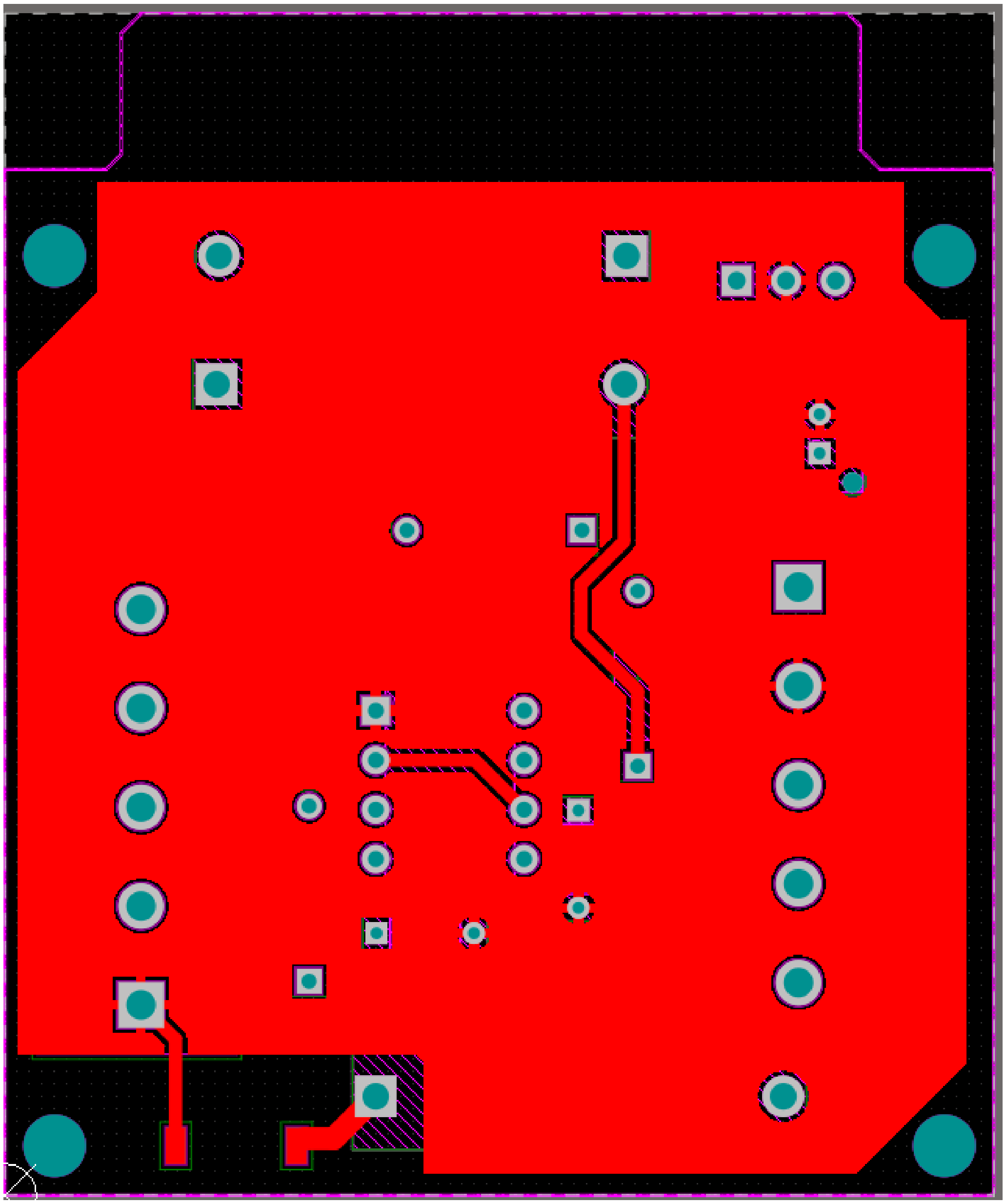
Below is the entire Altium schematic:



This looks similar to my spice schematic – but of particular note here is the absence of fan or pot. This is because I hooked up the input/outputs of the potentiometer to a screw terminal schematic diagram. I did the same for the inputs/outputs of the fan. I included a JST connector connected to VCC and GND in case I needed them for my digital pot, as I wasn't 100% certain of its operation and I hadn't received it yet. I didn't need this, but it can't hurt my design. I also included TEST POINTS because I've learned that testing a PCB without them can be a nightmare.

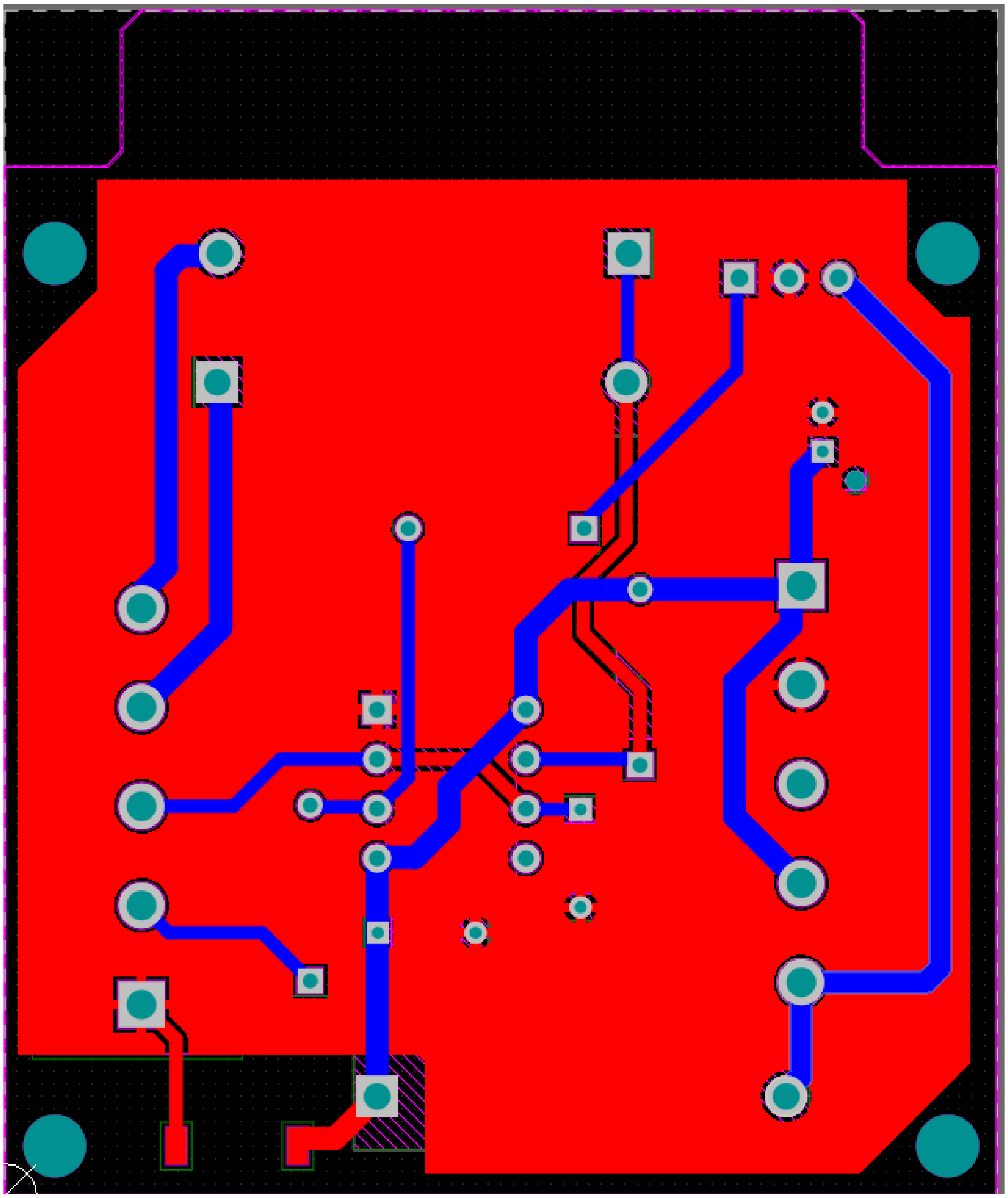
I used Screw terminals because they don't require crimping and are reasonably reliable, as my design shouldn't have any rigorous vibration requirements.

I laid out my PCB my starting with a ground plane on the top layer. I've isolated that layer below:



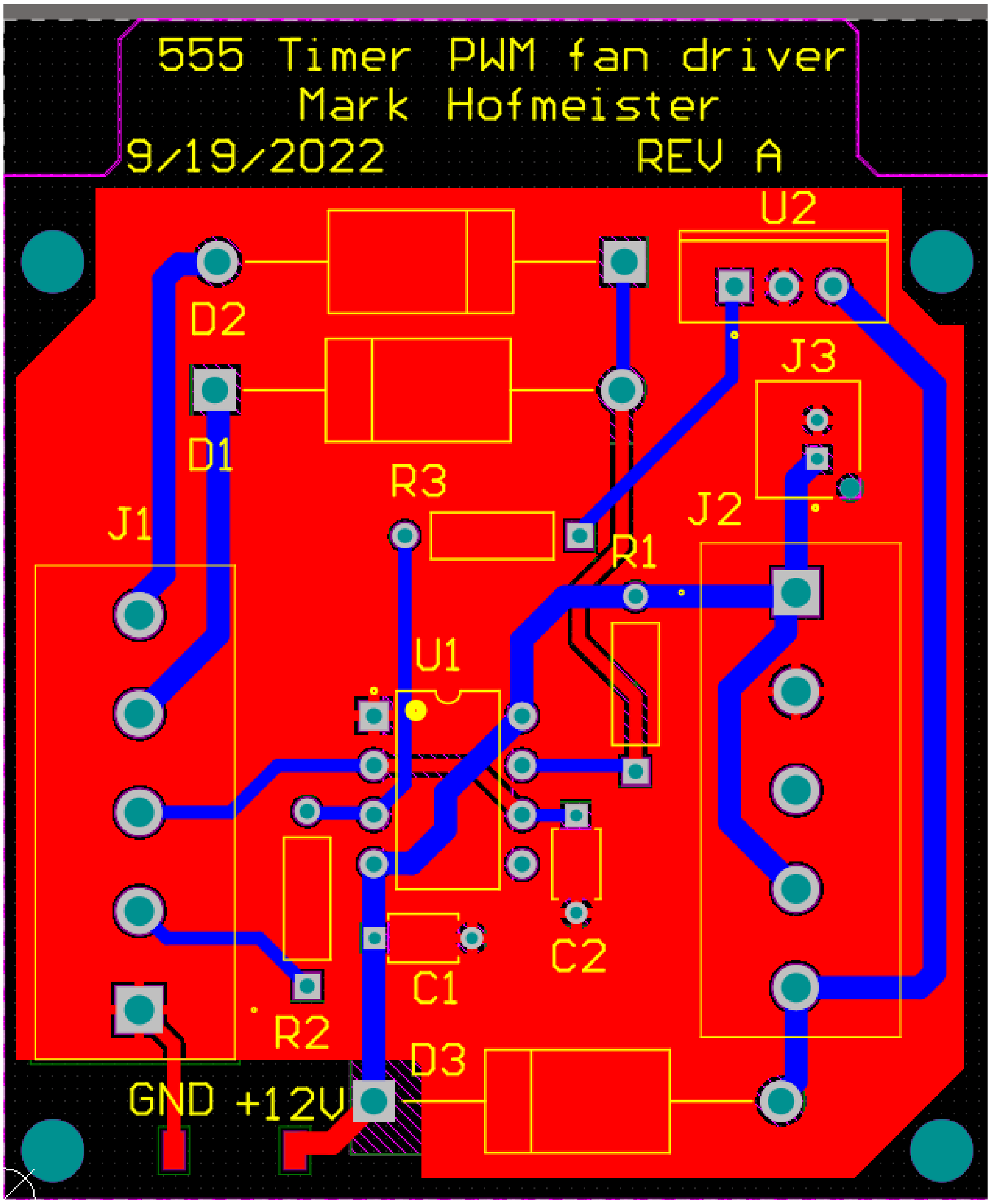
After my time working with high-speed/RF signal PCB design over the summer, I've developed a crush on Ground planes. I love them. I included thermal relief on the vias for ease of soldering. You can see the test points on the top as well in the lower-left corner.

I then routed the bottom layer, as shown below:

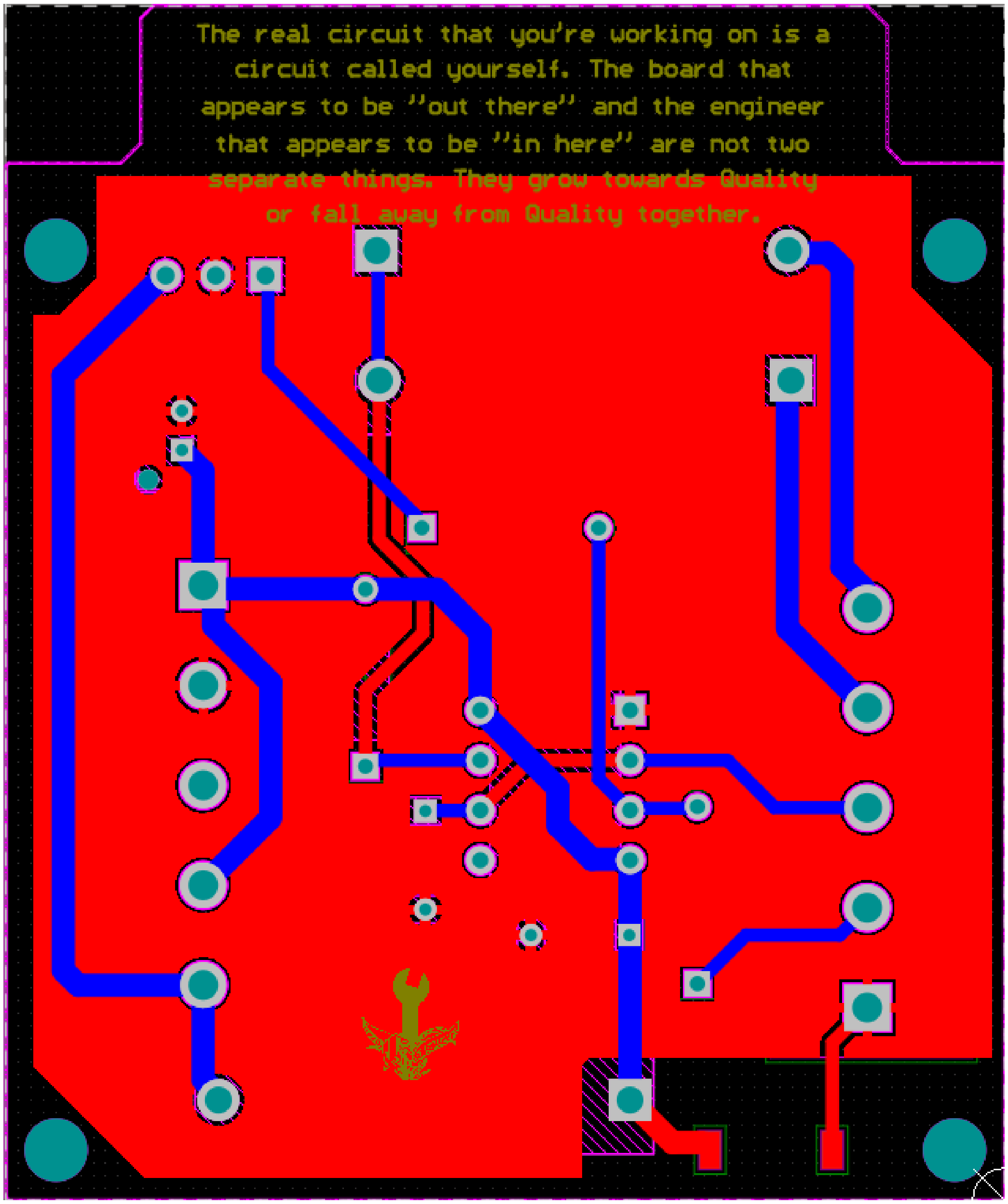


You'll notice that all components that can only be soldered from the bottom layer (screw terminals, JST connector) are routed completely on the bottom layer. In case I had to mill the PCB in-house, the vias would not be plated through. This is alright for a TH resistor, as one can solder both lead sides, but not for a JST connector. I learned that the hard way over the summer.

Finally, I put finishing touches on the top silk screen and board cutout:

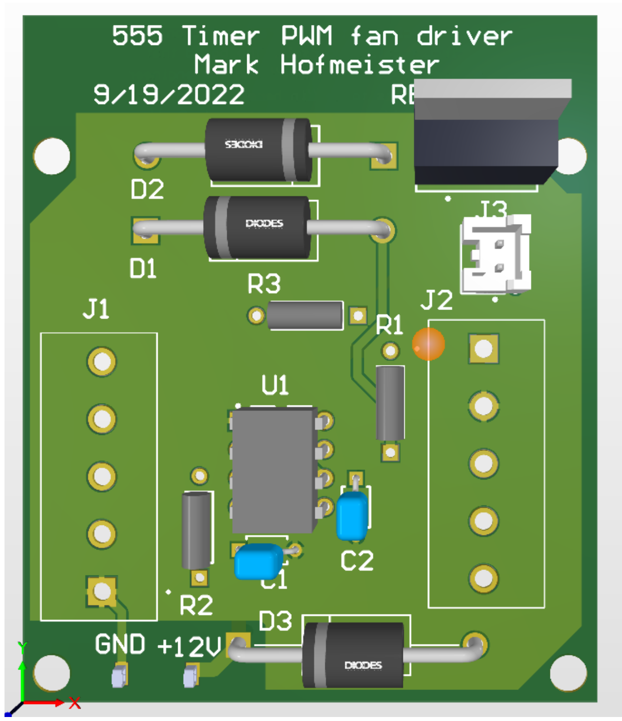


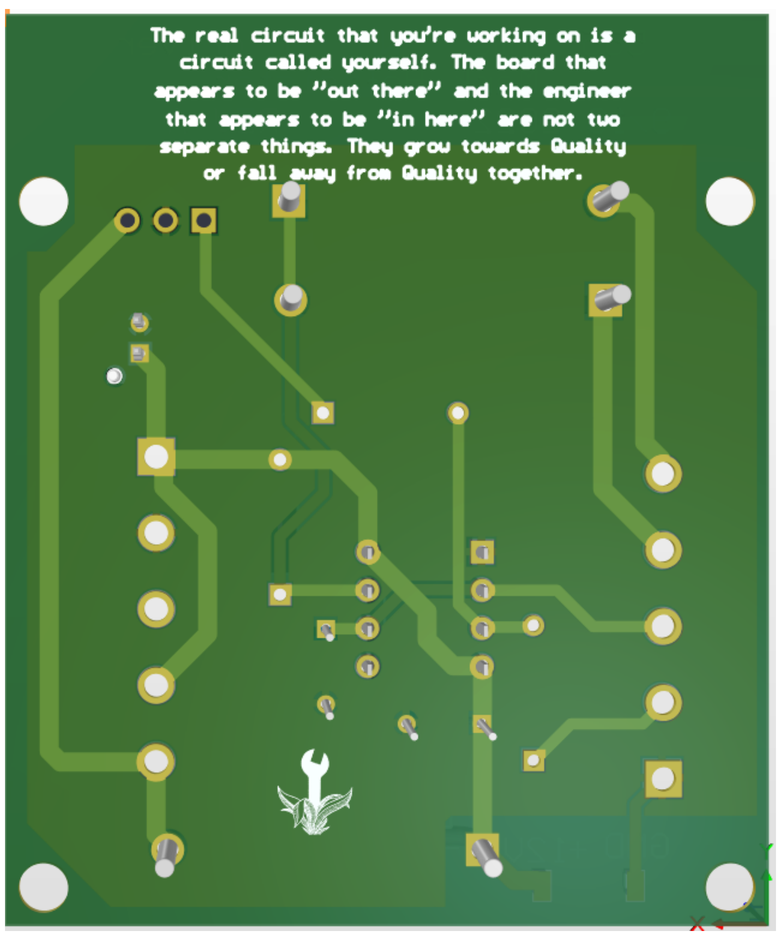
As well as the bottom silkscreen:



In addition to the ground plane, test points, and mounting holes, I avoided acid traps/sharp andgles and made sure to use wide trace widths. I also used TH components, because there's no way I'm messing with SMT for this if it's not necessary.

Here's a pretty 3D model:





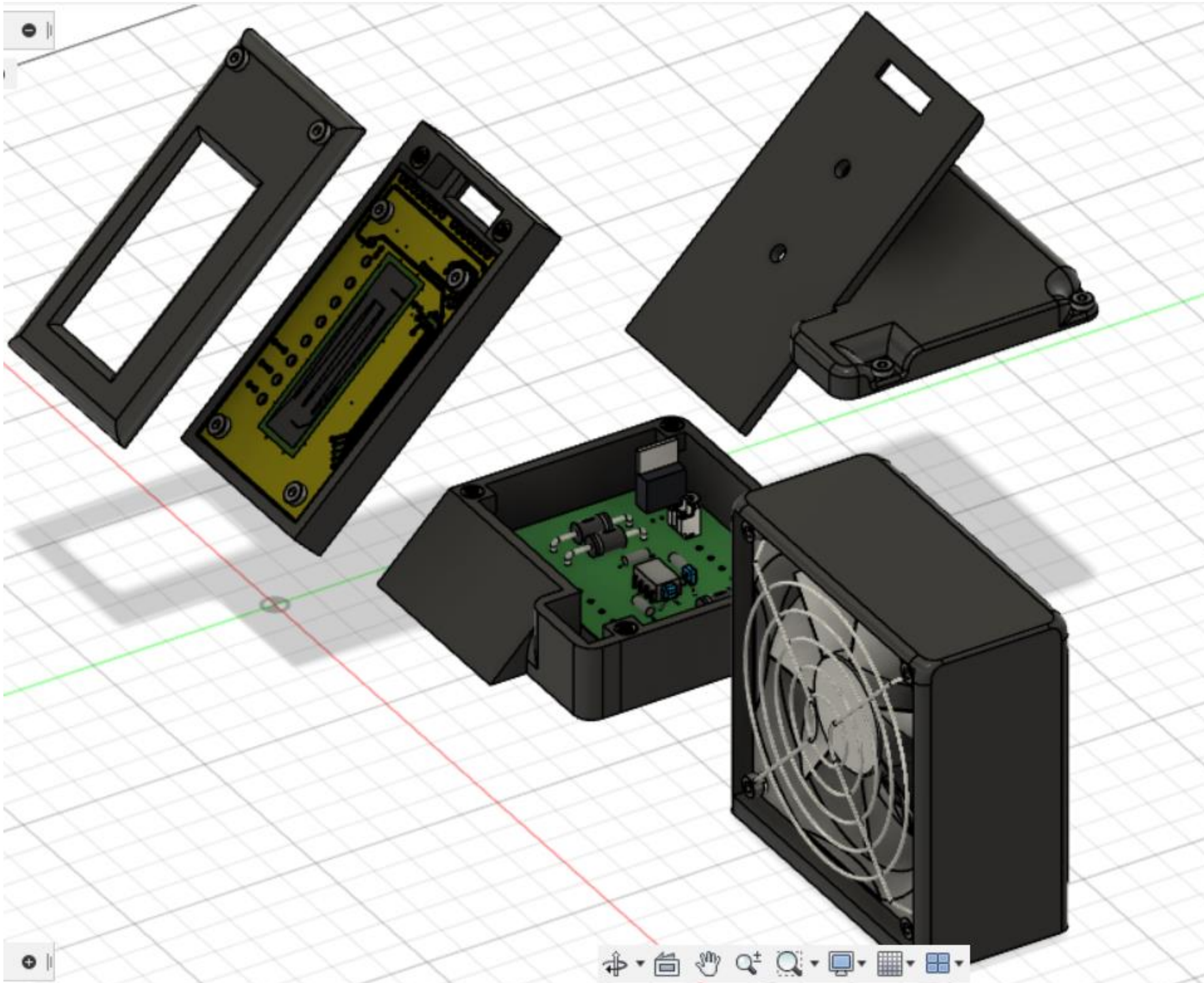
The bottom quote and decal are odes to a favorite novel of mine, “Zen and the Art of Motorcycle Maintenance” by Robert Pirsig.

Design Manufacturing and Assembly

I soldered up my board easily, due to TH components. I used an 8-pin DIP socket to ensure that if I burned out the 555 I wouldn’t hate myself. The solder flowed easily – I wasn’t sure if they’d do an ENIPEG coating.

I also made a very fancy enclosure, of which a 3D rendering is shown below:

I unfortunately didn't have time to make a nice exploded view, but a rough example is below:



The PCB sits in the bottom enclosure, mounted on standoffs, and the top screws on through the use of threaded inserts. The touch pot is also mounted on standoffs and mounted on the inclined plane of the PCB enclosure top. The fan is just connected through wires.

I 3D printed all of the parts, installed the heat-set threaded inserts, and attached everything together. It looks quite pleasant, as you'll see in the video. It was finicky to get the wires in the enclosure, and the supply chain issues prevented me from receiving some ordered screws, but it's robust.

Design Testing

I burned out my first digital pot. I forgot to break two bridges on the PCB to allow a +12VDC input, and fried it. After I replaced it, however, everything worked perfectly. I was shocked. I think that this is the first time that this has ever happened to me.

See my video submitted to Canvas to “See it in action!”