

HU3910 D22 - Flex PCB

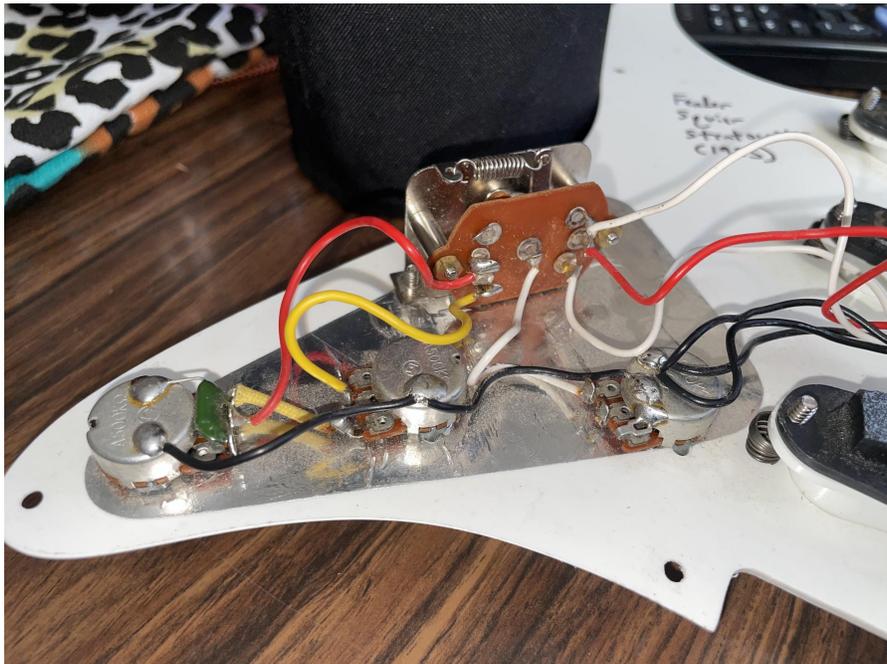
Alvaro Gonzales and Henry Livingston

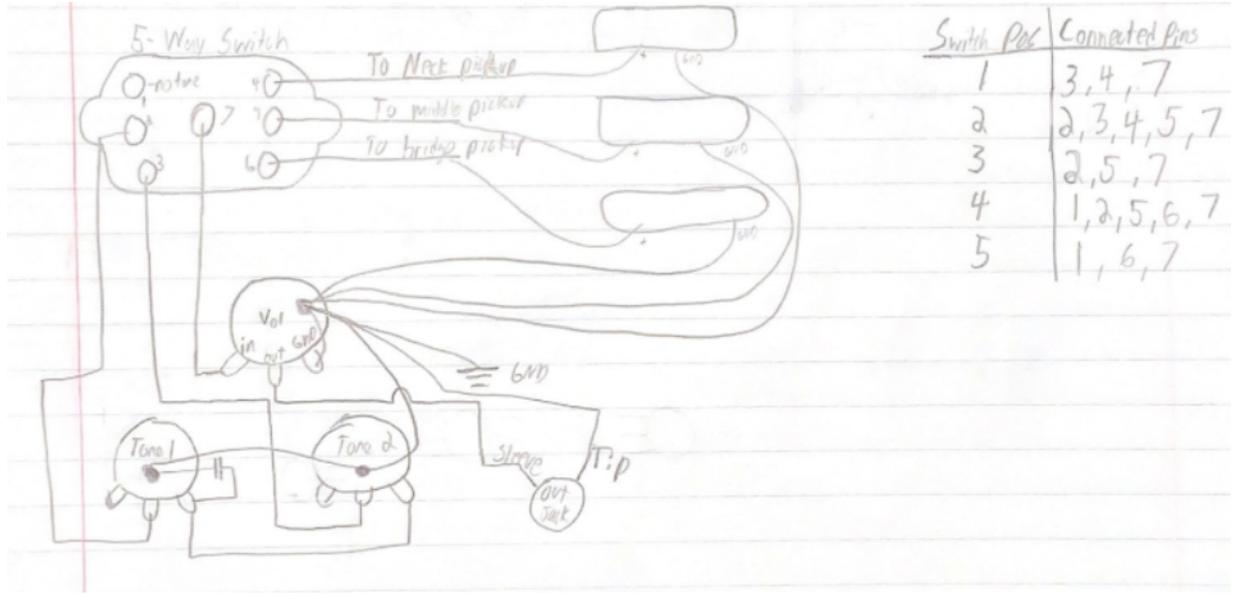
Overview

We used Fusion360 to design a flexible PCB to replace the wiring connecting the electric components together on a Fender Squier Stratocaster pickguard. Following the electronic schematic being made, 2D and 3D models of the PCB with the components were made and edited as necessary to be the right size. The 3D model could then be placed on a stratocaster 3D model, to see how it would fit and look. Once these were made, the .gbr files were sent to Anand Mitra, who printed the first flexible PCB. Once printed a continuity test was done to verify the connections. This first PCB allowed us to see what changes needed to be made in terms of size and shape for our second and final PCB that was printed.

Process

One of the first steps taken was analyzing how the components were wired together on the Stratocaster pickguard we were given. This helped us to understand how each part worked, on its own and with the others. We then drew a rough schematic based on it's current wiring. This would serve as a starting point for making the schematic to be used for the PCB in Fusion 360.

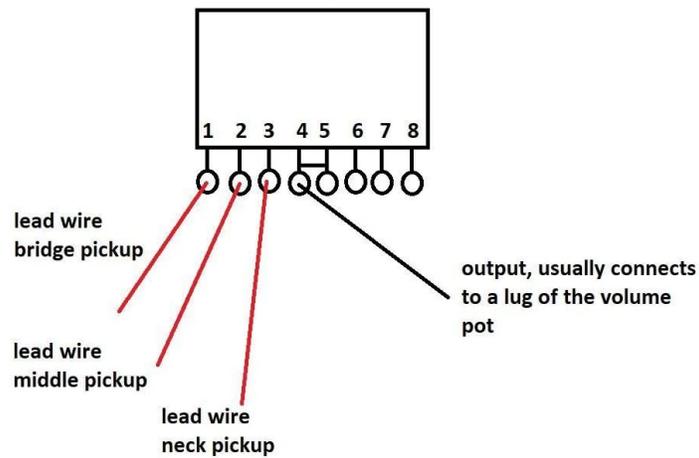


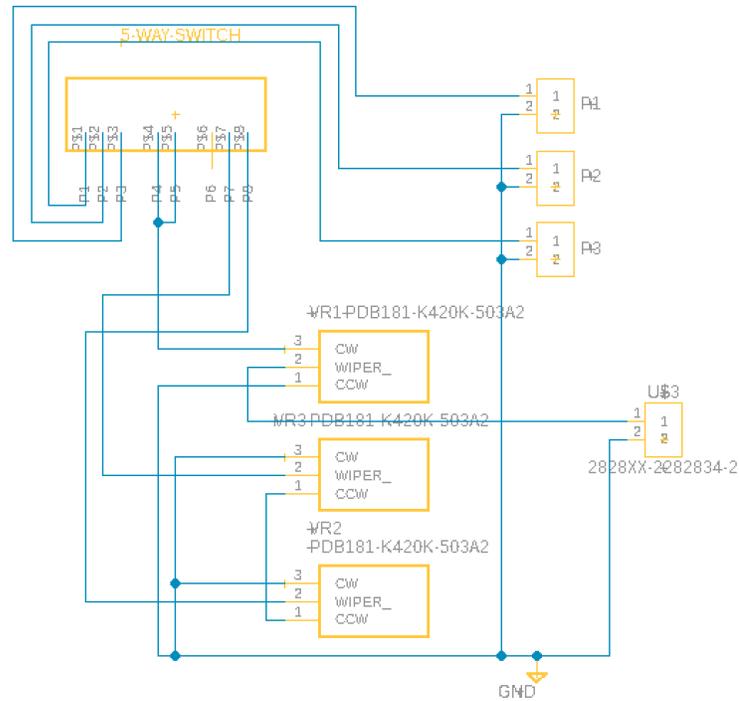


We ended up getting new potentiometers and a new switch to use. To make the schematic in Fusion 360, we made a custom part for the switch that has seven through hole pads that mirror the lugs on the new switch for easy wiring. Parts for the potentiometers were imported from mouser.

-New Potentiometers used are Bournes PDB181-K420K-504A2

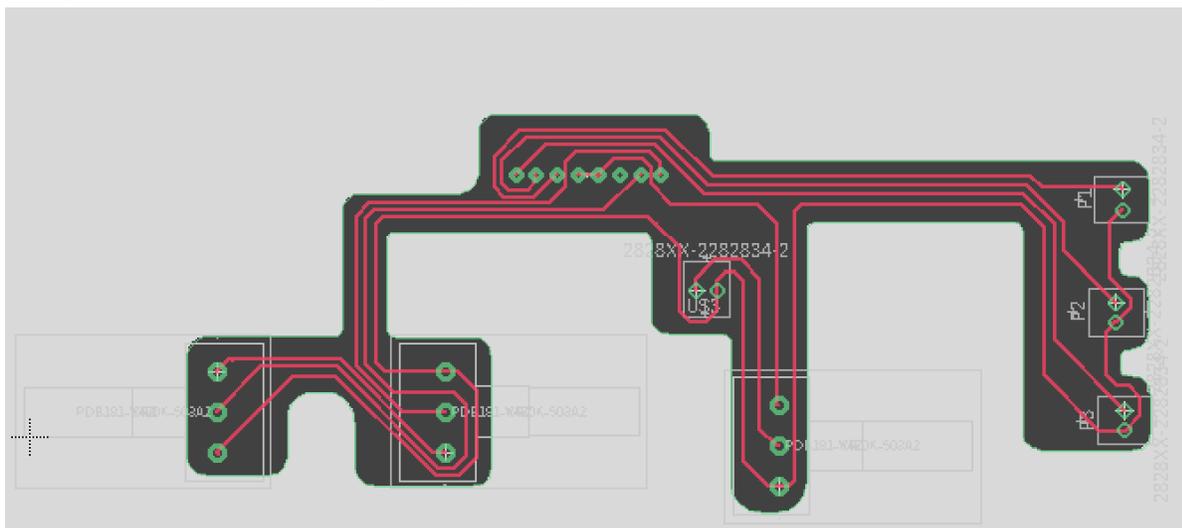
-New Switch used is [KAISH Heavy Duty 5 Way](#)

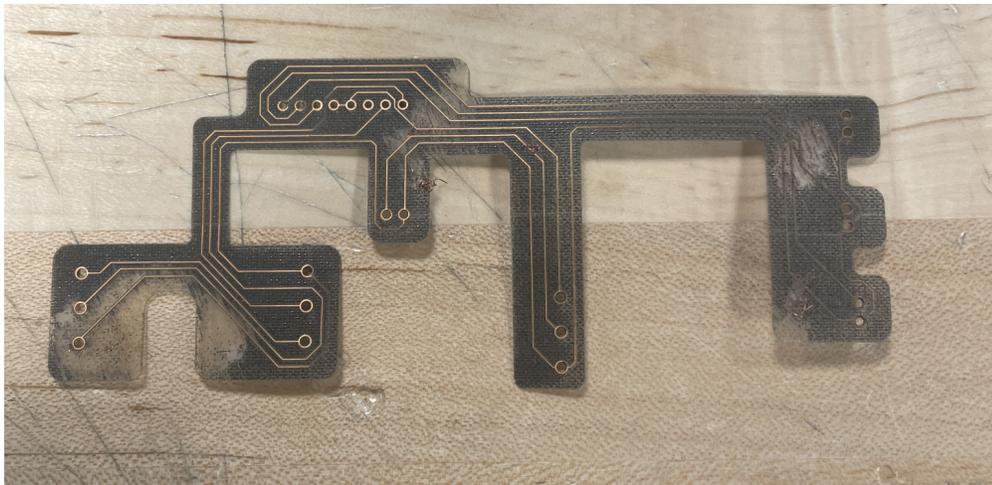
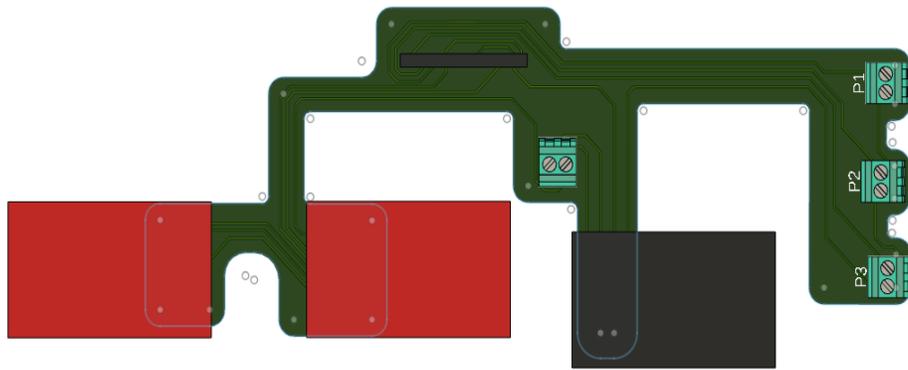
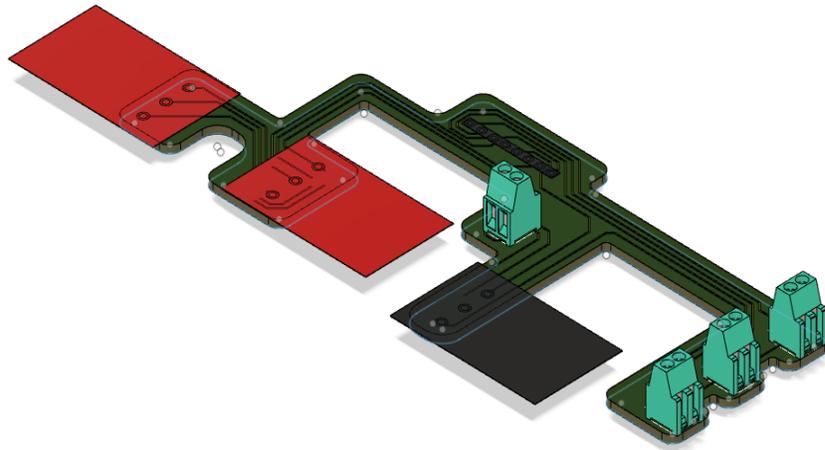




Once the schematic was made in Fusion 360, with some slight modifications to the design, we set out to make the shape of the PCB. This involved measuring where each of the parts were mounted to the pickguard, relative to each other, to see what size and shape the PCB needed to be to connect all of them together. The shape of the PCB can be determined by making a 2D shape of the PCB. Once the shape is done the 2D model of the PCB can be linked with the designed shape and altered to the user's convenience.

Finally, after the shape was made and the part footprints were in place, we routed the traces using only one layer to make printing the PCB easier and more reliable.





Future recommendations:

- Ensure the method of grounding the potentiometers through the 3rd previously unused pin on each work. The previous method was grounding them using the body of the potentiometer. If there's any issues, jumping a wire from the ground pin to the body of the potentiometer may be worth trying.
- Make through hole pads used to connect the pickups and switch larger so it's easier to solder wires to them
- Modify PCB to connect directly to switch, rather than soldering wires between them. It is currently to be connected with wires because the current switch contains solder lugs rather than pins.
- Further refine PCB shape.

Other important things to know:

- In Fusion 360 actual connector parts were used for the pickup inputs and the output. This was mostly for convenience and the plan was to just solder the wires right to the PCB.
- Where Fusion 360 shows the side profile of the potentiometers is not very important as the PCB will flex to reach the pins and not lay flat. The location of the through hole pads for the potentiometer pins is most important.
- Capacitor is currently not included in the electronics schematic or PCB design as we were planning to simply solder it between the pins on the potentiometer.

Where are the resources:

- <https://www.youtube.com/watch?v=8-tJZHFzWXo> -> Create your own electronic parts in Fusion360
- <https://www.youtube.com/watch?v=VZZBEocoYDA> -> PCB Layout Tutorial Walkthrough
- <https://www.youtube.com/watch?v=OE9dYO1o-uM> -> Creating a Custom PCB in Fusion 360
- The resources can be found in the project Wiki page.
- Project Files can be found in the Git Repository

