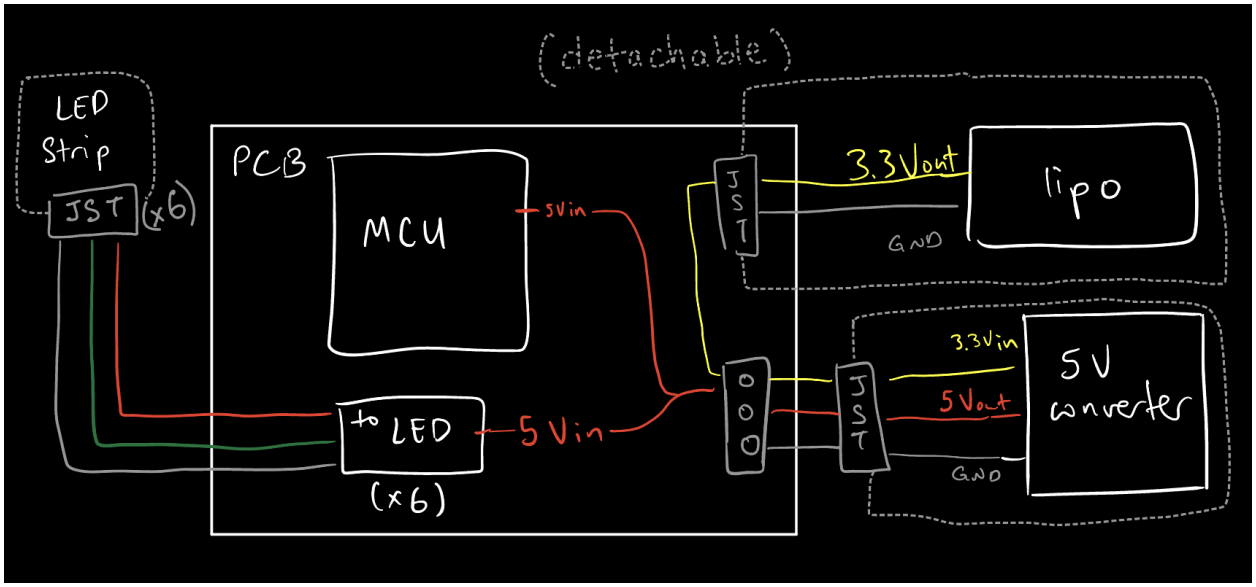
PCB Class LED Fans Project
02/03/2023

Rihn H

[LED] [Fans] [PCB Design]

Creating a small PCB to control LEDs in the fan handles



“Low Budget Flex PCB using… ***wires***”

Overview

The goal of this project was to create a PCB that could control LED strips on a custom cut aluminum fan in a way that enables further prototyping for LED fans. This would include testing different LED types, POV effects, IMU data effects, and different battery configurations.

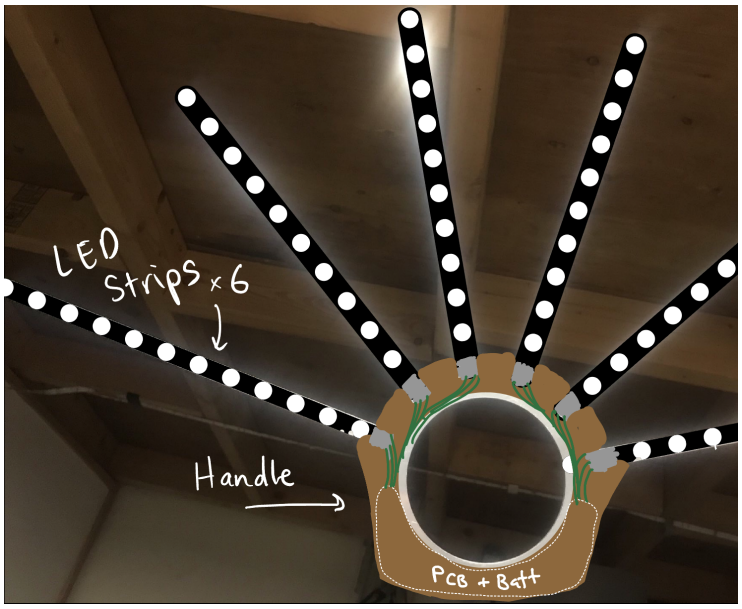
These fans can have attachable handles that would house the PCB. These handles have a roughly circular shape and should not be bulky, hence the need for a small and unobtrusive PCB footprint.

Design Requirements
Component Choice
Schematic Design
Layout Design

Design Requirements
These are meant to be the first prototype of PCB controlled LED fans on a custom frame.

- The PCB should be modular
    - Can swap out components like the battery, the power converter
    - Can test different types and configurations of LED strips (not on every fin; on either side of the fan; LED strip type and density)
- The PCB should be easily reprogrammable
- The PCB should fit in a round-ish handle and be flat

Here is a sketched out plan of the design.



Sketch of LEDs along a fan's fins attached to a PCB & battery system via wires inside the fan's handle.

Since the handle will be added anyway to aid in manipulating the fan, its volume can be used to house the electronics.

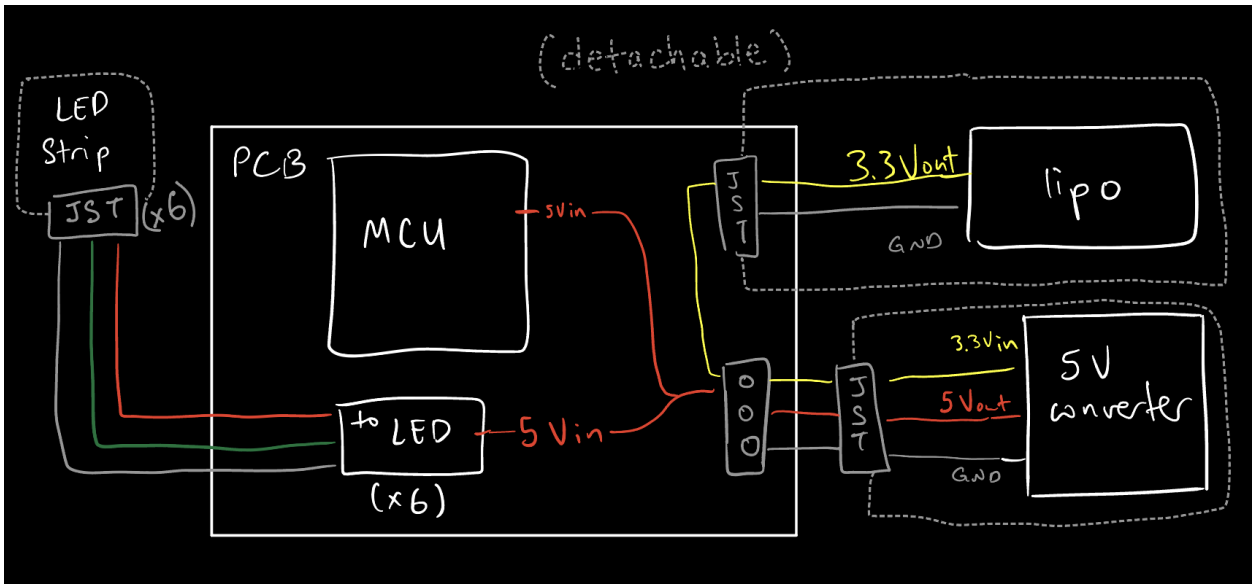
Component Choice
The design will use a Seeed Xiao Sense, since it has a built-in IMU and can be programmed via USB. It is also relatively small, approximately the size of a thumb. The USB should make adjusting the code easier/faster than having to debug the hardware or tiny solder joints.

Here are some resources for it -

https://wiki.seeedstudio.com/XIAO_BLE/

https://how2electronics.com/getting-started-with-seeed-xiao-ble-nrf52840-sense/

I've had projects before where I've spent all my time properly testing/soldering a board that had inherent design issues, and the debugging process with tiny SMD components like an IMU and processing chip took all my time away from actually getting to program it. I want something that will work and help me figure out the placement of the rest of the part. Once I figure out the overall look and feel, the goal will be to then work on iterating smaller, by switching to even smaller chips like the STM32 (https://www.st.com/en/microcontrollers-microprocessors/stm32-32-bit-arm-cortex-mcus.html#overview).



Sketch showing how LED strips, battery, and battery converter are wired to and detachable from the board.

To add other pieces, like the battery and converting system from 3.7V to 5V, I want to use a JST connector so I swap out different ones that fit with the maximum current drawn by the LEDs (since the number and type of LED strip could change).

Most of the tiny ones I have available to me have a maximum current of 500mA. I could solder multiple of these together. (https://www.pololu.com/product/2119)

Another one online I found is larger, but can do a bit over 1A. (https://www.amazon.com/AITIAO-Multi-Function-Indicator-Voltage-Lithium/dp/B0BG47N8NV/)

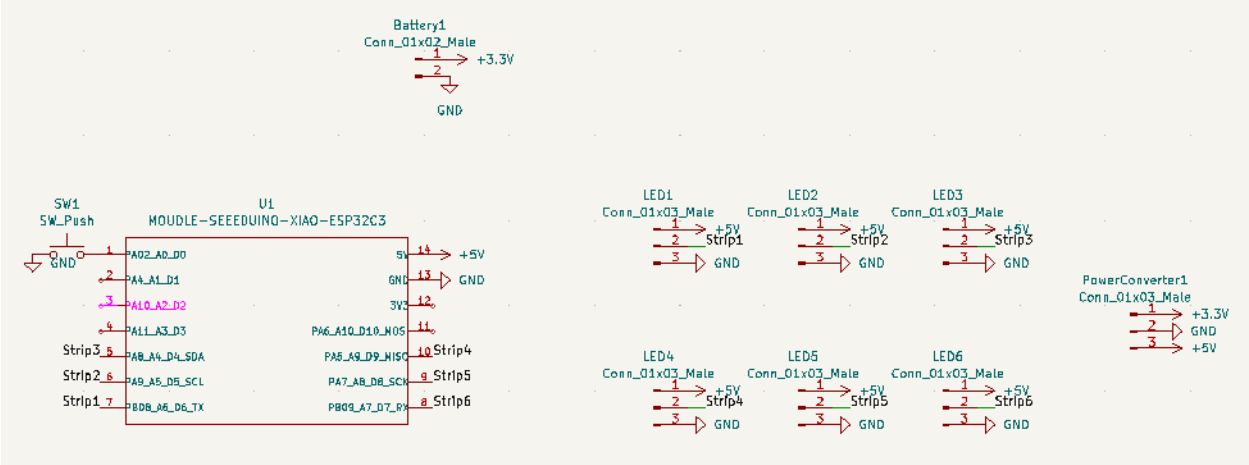
The battery will also be attached via JST, and will be a tiny lithium polymer battery that can be replaced.

The wires allow for some flexibility in the handle design and for the components to move in 3D space as needed.

Schematic Design

I created this in KiCAD since my current primary OS is Linux, and I don't have to worry about VMs and Altium.
It was relatively simple. I also added a button for interfacing. This could be made a bit more complex, like a power switch plus a button, but just keep things simple until other functionalities are needed, since this board does have wireless capabilities.

The detachable components are all represented by connectors for wires.



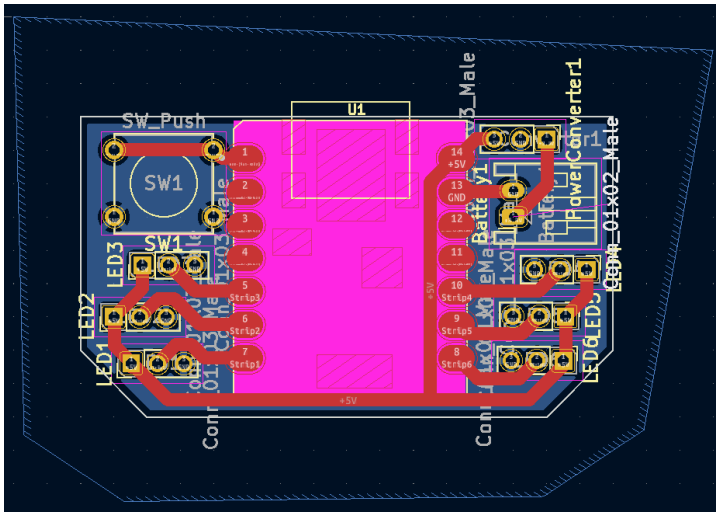
An image of the PCB schematic.

Layout Design

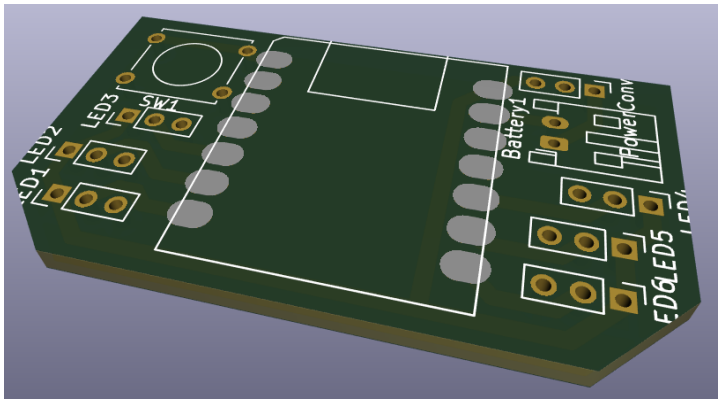
To be fair, I wasn't sure if I wanted separate cables for the power and ground of each LED strip, since that requires 9 wires running along each side of the handle upwards, when only 5 are really needed if there is a splitter further along.

However, for the handle wiring, 3 ribbon cables with 3 wires each might be okay.

The traces lean on the larger side, 1mm, since this PCB is intended to be milled on copper clad board and tiny traces breaking may pose an issue.



An image of the PCB layout.



An image of the PCB board in 3D without components.

Stay tuned for the fabrication and testing of this PCB!

Acknowledgements

Thank you to Oliver for pulling an all-nighter and Josh for the second of his two almost all-nighters with me the day before the projects are due and helping with occasional questions and camaraderie.

Thank you to Fischer and Adi for their work on the Art and Science of PCB Design course (https://pcb.mit.edu/lectures/) and their design/ideation recommendations.