Git Gud at PCBs #2: Time To Lock In

Last time, you developed a board solely focused on sensors however you may have been thinking where do those 3.3V/5V power flags come from physically? How do we regulate our power sources into these voltages?

Design a power management PCB

Objectives: Design a PCB with an RP2040 which regulates a 3S LiPo battery and USB-C VBUS to 3.3V at 500mA (Low noise) and 5V at 2A.

Design requirements:

- USB-C with VBUS must be included.
- 3S LiPo Battery must be regulated.
- 3.3V 500mA voltage output (must be low noise).
- 5V 2A voltage output.
- Voltage outputs should be output using connectors

Questions

- What is the specification for low noise? What's the point of it?
- What is the wattage that USB-C can output?
- What happens when only the USB-C is plugged in? When it's only LiPo?
- What happens when the USB-C and battery are both plugged in simultaneously? **BONUS**: **Did you account for this?**
- How does a buck converter work? What is the difference between synchronous and asynchronous?

- What are the benefits and drawbacks of LDOs and Buck Converters?

Goals

- Learn about different types of DC-DC voltage regulator ICs.
- Learn how to layout different voltage regulators

Hints

- Layout is a main focus for this challenge, I heavily recommended looking up Texas Instruments PCB Layout for "***", here's an example.
- Connectors come in many kinds, for simplicity look into JST connectors or screw terminals.
- **BONUS**: Struggling with handling both inputs on? Look into ORing!