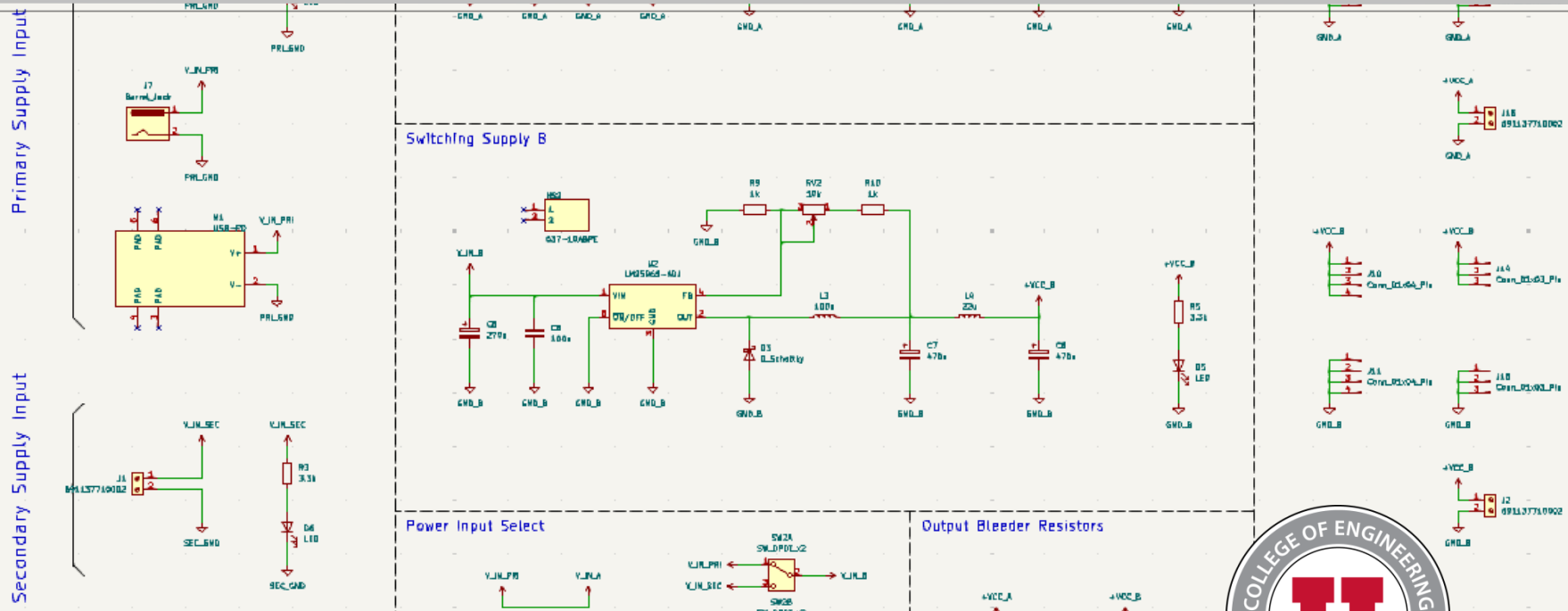
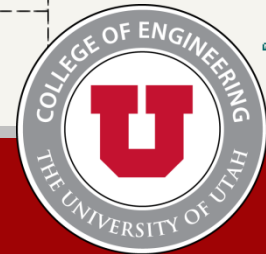


IEEE KiCAD Crash Course: (Week 01) Introduction to Schematics



Hosted By: Adrian Sucahyo and IEEE at the University of Utah
Adapted From: IEEE x FSAE Workshop SP25 with Nick Howard
and Adrian Sucahyo



Workshop Logistics

- We **highly** recommend attending all workshop sessions
 - Each session is designed to build off previously discussed materials
- By the end of the workshop, you will be able to:
 - Build an adjustable USB-C Breadboard Power Supply
 - Understand the basics of schematic layout, PCB layout, and board manufacturing
 - Have a basic understanding of manufacturing techniques and equipment used in the industry

Want more experience?

- Consider joining the FSAE tractive team!
 - The Tractive Team is currently looking for students to assist with designing and assembling the electrical system for an electric formula-style race car!
 - No experience required!



U of U FSAE Discord Link



Join the IEEE Discord

- If you haven't already, please join the IEEE Discord server for additional information and updates regarding this workshop

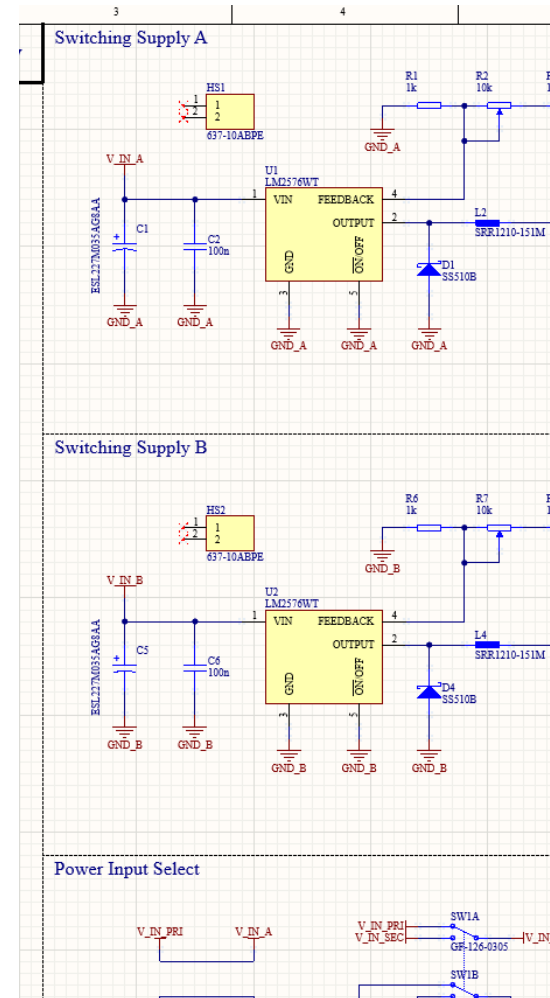


U of U IEEE Discord Link



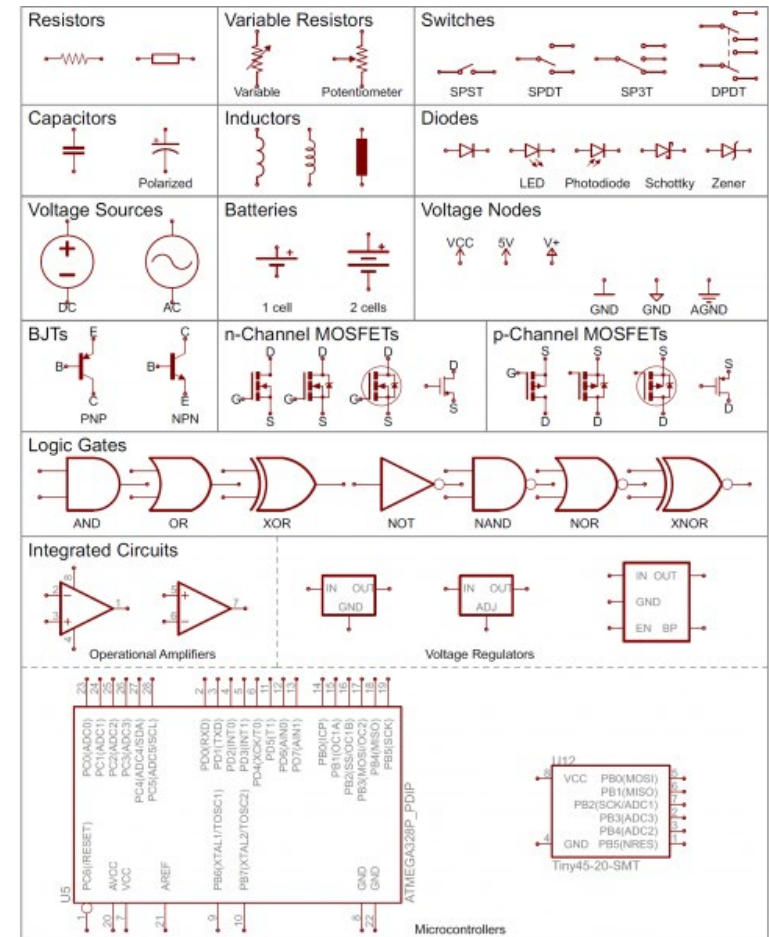
What is a schematic?

- Schematics are a graphical representation of an electrical circuit or system
 - A step up from a block diagram
 - Shows all parts and electrical connections in a circuit
 - Used to route PCB traces during layout
- Schematics are NOT a physical representation of component positions on the PCB



Components in a Schematic

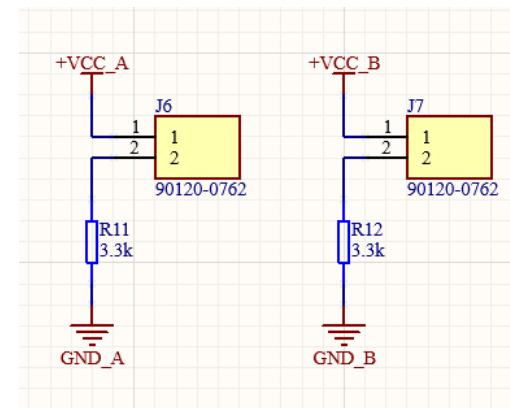
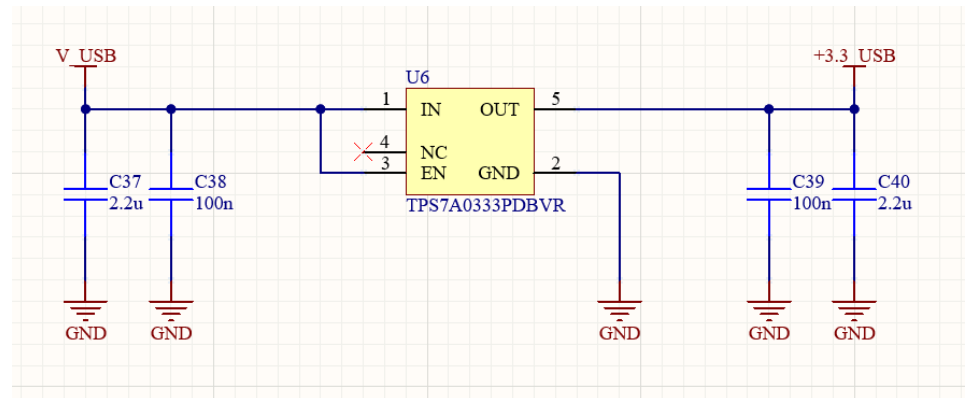
- Every electrical component will have its own symbol within the schematic
 - Contains all pins connected to the component
 - May contain additional information such as signal type
 - Relatively consistent across platforms



<https://learn.sparkfun.com/tutorials/how-to-read-a-schematic/all>

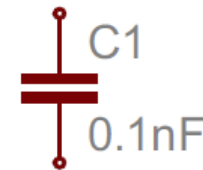
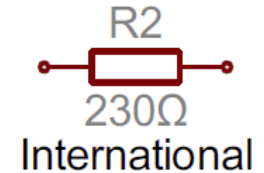
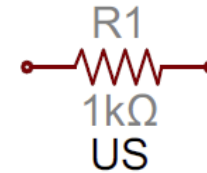
Symbol Attributes

- Symbols may have additional information regarding the component they represent
 - Designators
 - Values
 - Part Numbers
 - Pin Name
 - Pin Number

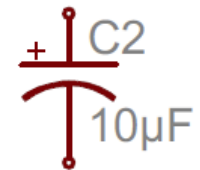


US vs International Symbols

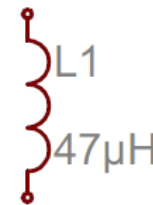
- You may see variations on certain components between US and International Schematics
 - Resistors, Capacitors, Inductors and more may have slightly different schematic symbols



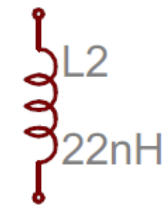
Non-polarized



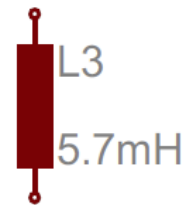
Polarized



US



22nH

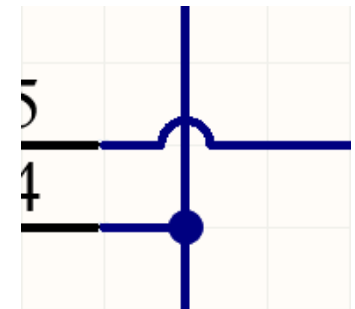
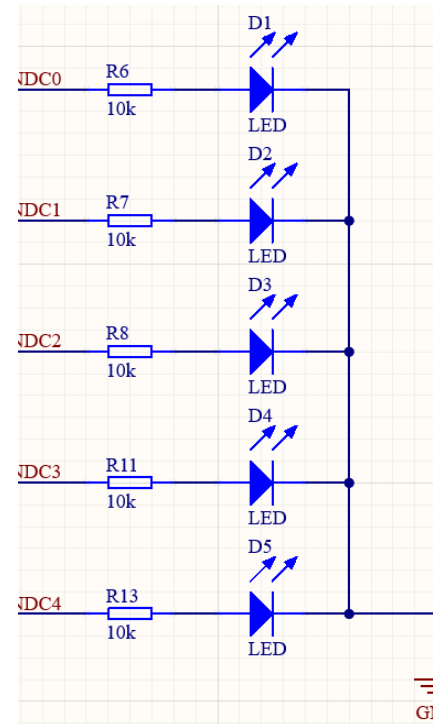


International

<https://learn.sparkfun.com/tutorials/how-to-read-a-schematic/all>

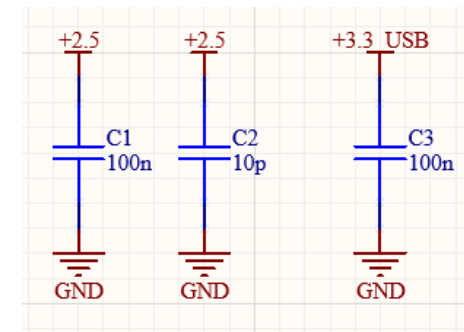
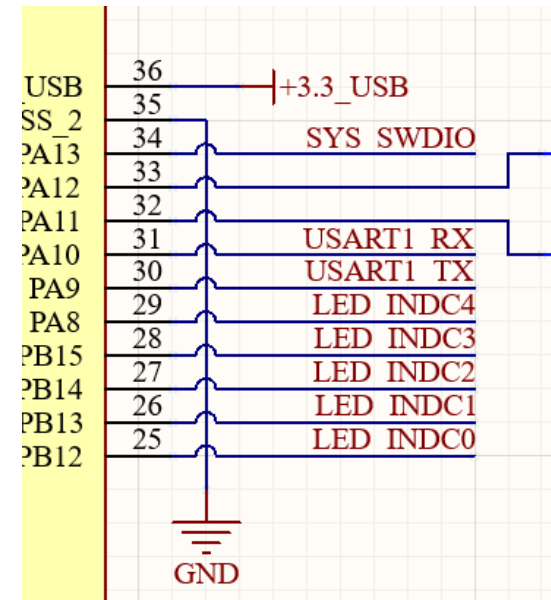
Nets and Nodes

- Schematic Nets are how components are electrically connected in a circuit
 - Represented with lines between components
- Nodes/Junctions are locations where wires split in two (or more)
 - A connection on a node is denoted by a dot



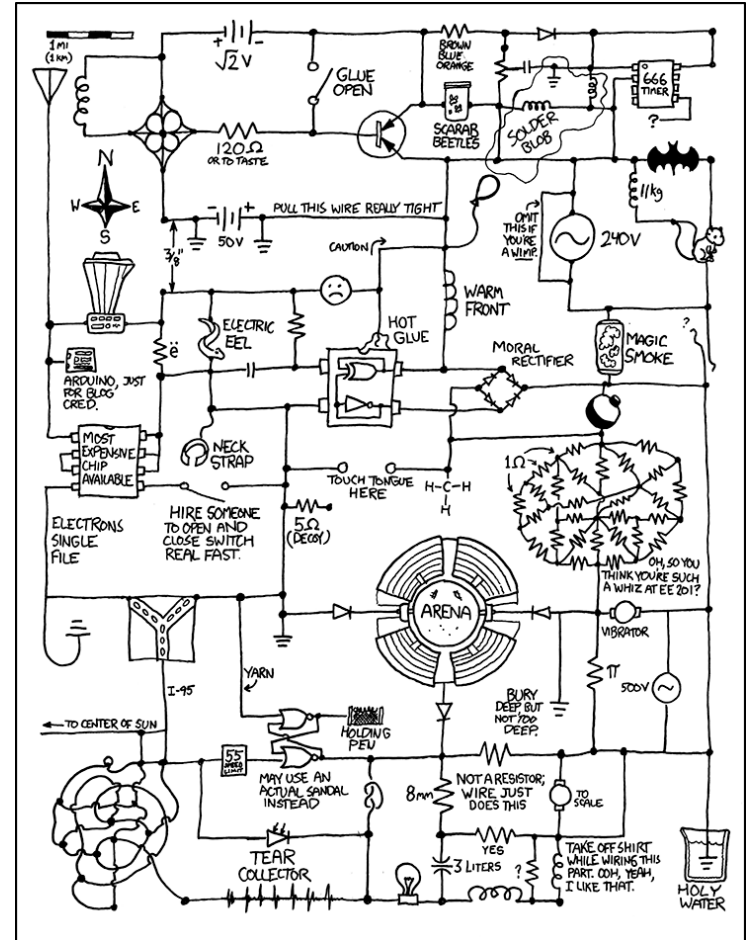
Net Labels

- Net Labels allow electrical connections to be made without having to draw a line between components
 - Net Labels tend to be local to the current schematic sheet
 - Power Labels tend to be global to the full design
- Note: Ports are similar but have a different purpose
 - Useful for hierarchal / multi-sheet design



Good Schematic Practice

- Readability is KEY!
 - The schematic needs to be read by other engineers to assist with layout and debugging
 - Readable schematics make it easier to find electrical errors which would be translated into the physical layout

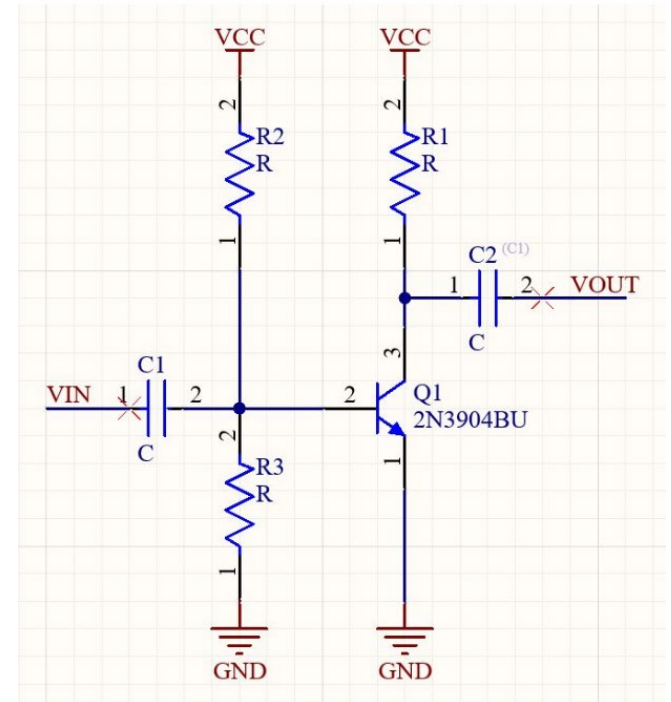
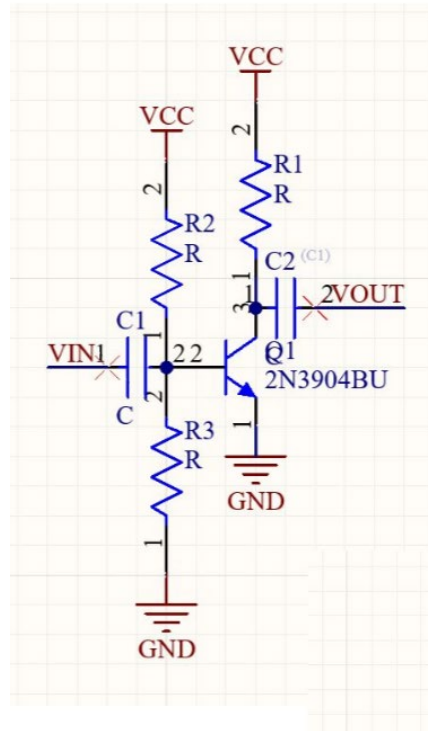
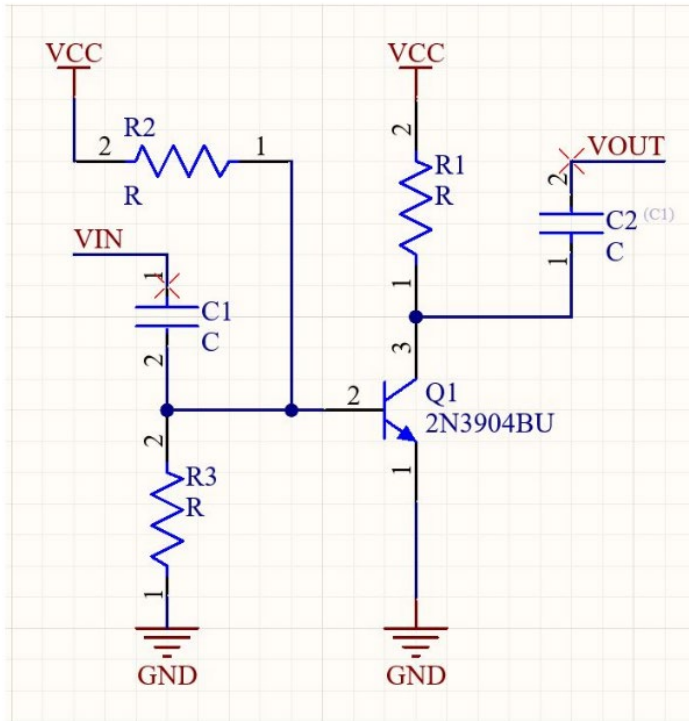


<https://xkcd.com/730/>

Tips:

- Read Left to Right
- Don't Overcrowd
 - Use a larger sheet or use multi-sheet drawings
- Use Logical Blocks to Divide Circuit Elements
- Reference Manufacturer Datasheets
 - Follow established layouts
- Label Signals
 - Especially those which are not immediately obvious
- Keep Power Sources at the top and GND at the Bottom

Examples:



ECAD Software

- ECAD (Electronic Computer-Aided Design) are software used to design and create diagrams and layouts for PCBs
- There are many different vendors for ECAD software:
 - Altium
 - KiCAD
 - Eagle
 - OrCAD
 - EasyEDA

Altium
Designer®

KiCad

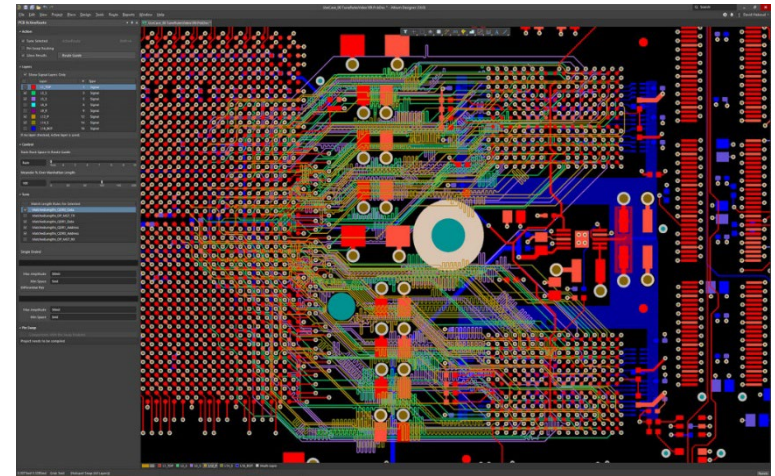
E AUTODESK®
EAGLE

OrCAD™
CADENCE PCB SOLUTIONS

 **EasyEDA**

Why Altium?

- Altium is very commonly used in the industry for PCB and circuit assembly
- Not many opportunities to get access to the software outside of university
 - Please get your student license ASAP if you haven't already
- Great addition to your resume!



EMPLOYMENT / JOB APPLICATION

PERSONAL INFORMATION

FULL NAME: _____ DATE: _____
First Middle Last

ADDRESS: _____
Street Address Apt/Suite

City State Zip Code

E-MAIL: _____ PHONE: _____

SOCIAL SECURITY NUMBER (SSN): _____

DATE AVAILABLE: _____ DESIRED PAY: \$ _____ ☐ HOUR ☐ SALARY

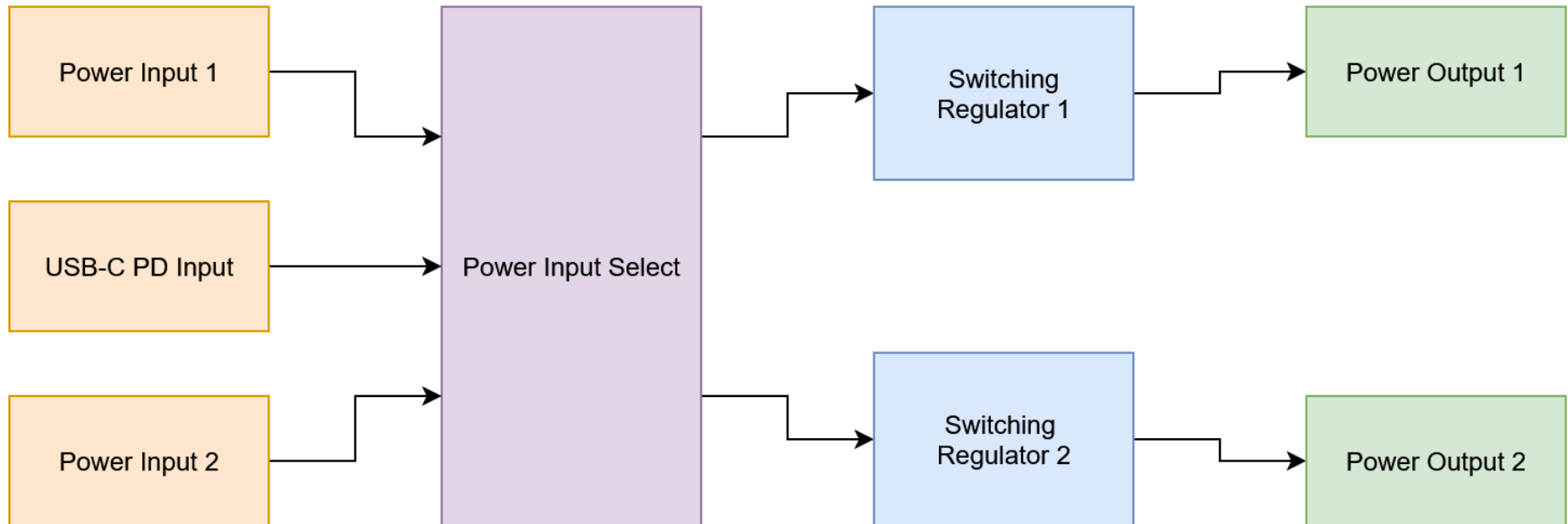
POSITION APPLIED FOR: _____

EMPLOYMENT DESIRED: ☐ FULL-TIME ☐ PART-TIME ☐ SEASONAL

Project Introduction

- **USB-C Breadboard Power Supply**
 - Dual Switching Supplies
 - Independent Voltage Outputs
 - Switch between isolated / tandem supplies
 - Dual Input Supplies
 - Primary and Secondary Supplies
 - Terminal Block Input
 - USB-C PD trigger on Primary Supply
 - Additional Barrel Jack DC input
 - Configurable output bleeder resistors

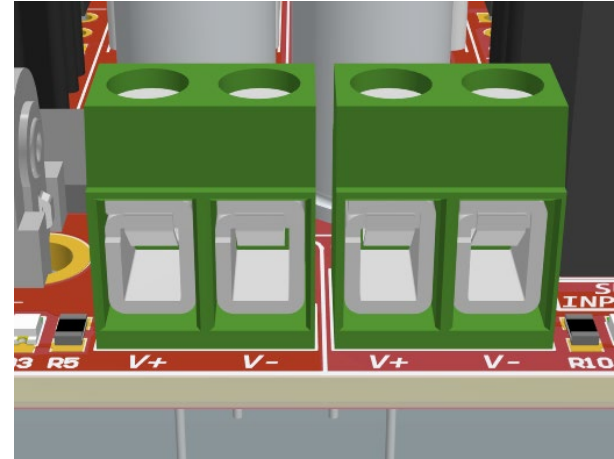
Block Diagram



Power Inputs and Input Selection

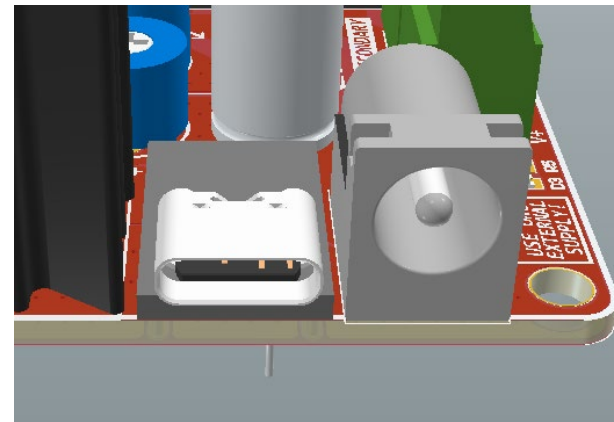
- **Power Inputs**

- Terminal Block Input
- Barrel Jack Input
- USB-C PD Breakout
- 20 VDC Max



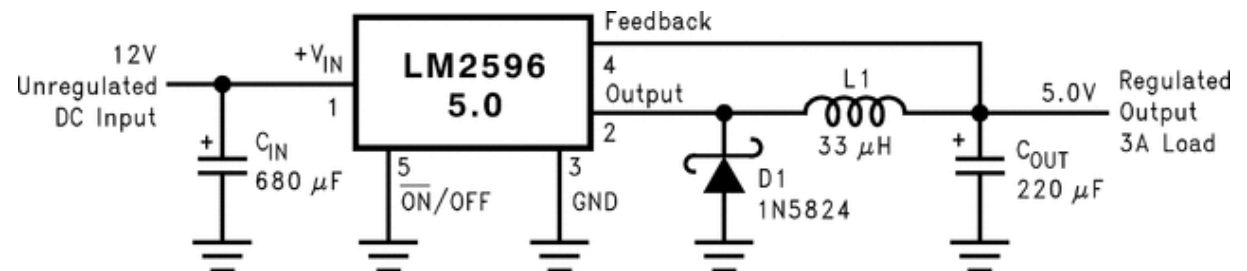
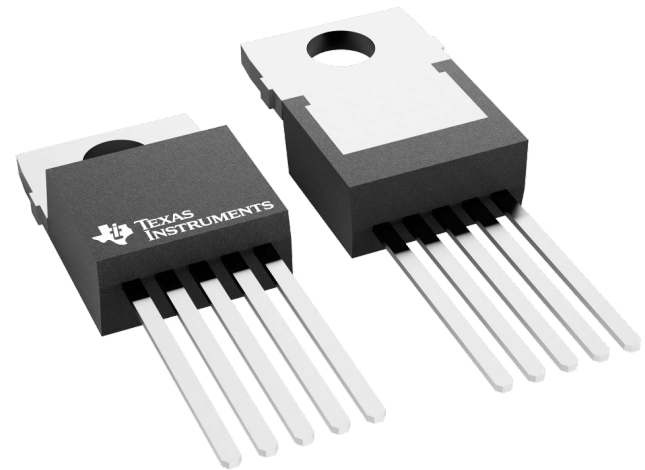
- **Input Selection**

- Select between a primary and secondary supply
- Isolate supply A and B for independent operation



Switching Supplies

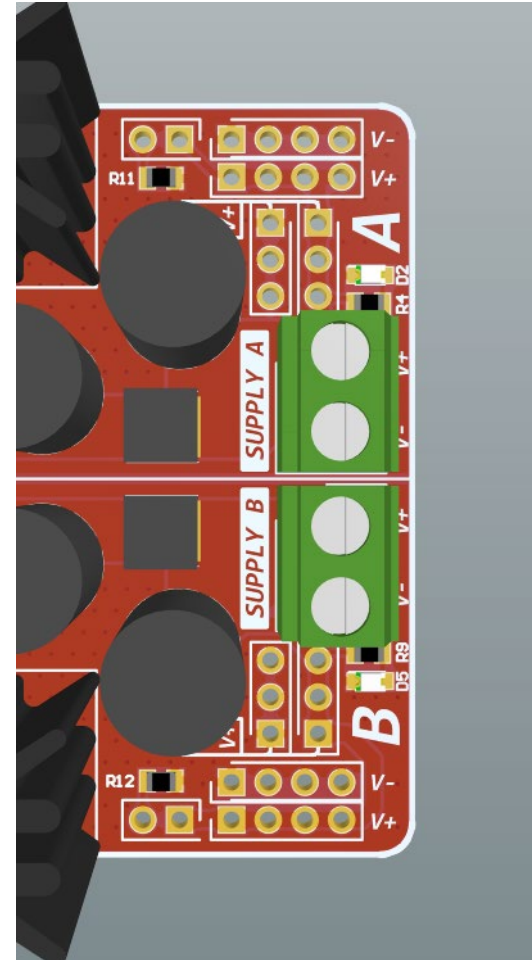
- **Texas Instruments Simple Switchers – LM2596**
 - Buck Converter
 - Adjustable Output
 - 3 Amp Output Load Capacity
- Reference Datasheet for more information and reference circuit



<https://www.ti.com/product/LM2596>

Power Outputs

- **Power Outputs**
 - Terminal Block Output
 - Pin Headers
 - Breadboard width for easy utilization
- Output Bleeder Resistors



Questions?

Questions?

Download Today's Project Files

Navigate to the workshop GitHub and
download today's files

<https://github.com/IEEE-U-of-U/IEEE-KiCAD-Crash-Course>