UNIVERSITY OF
ILLINOIS
URBANA-CHAMPAIGN

PCB Review!

Electrical & Computer Engineering

Jason Jung

10-3-2025

Topics:

Basic Power Electronics
USB-C Implementation
Differential Signaling
Trace Sizing
Impedance Matching
Protection Circuits
General Power/Ground Routing Guidelines
Good Practices/Bad Mistakes/Rules of Thumb

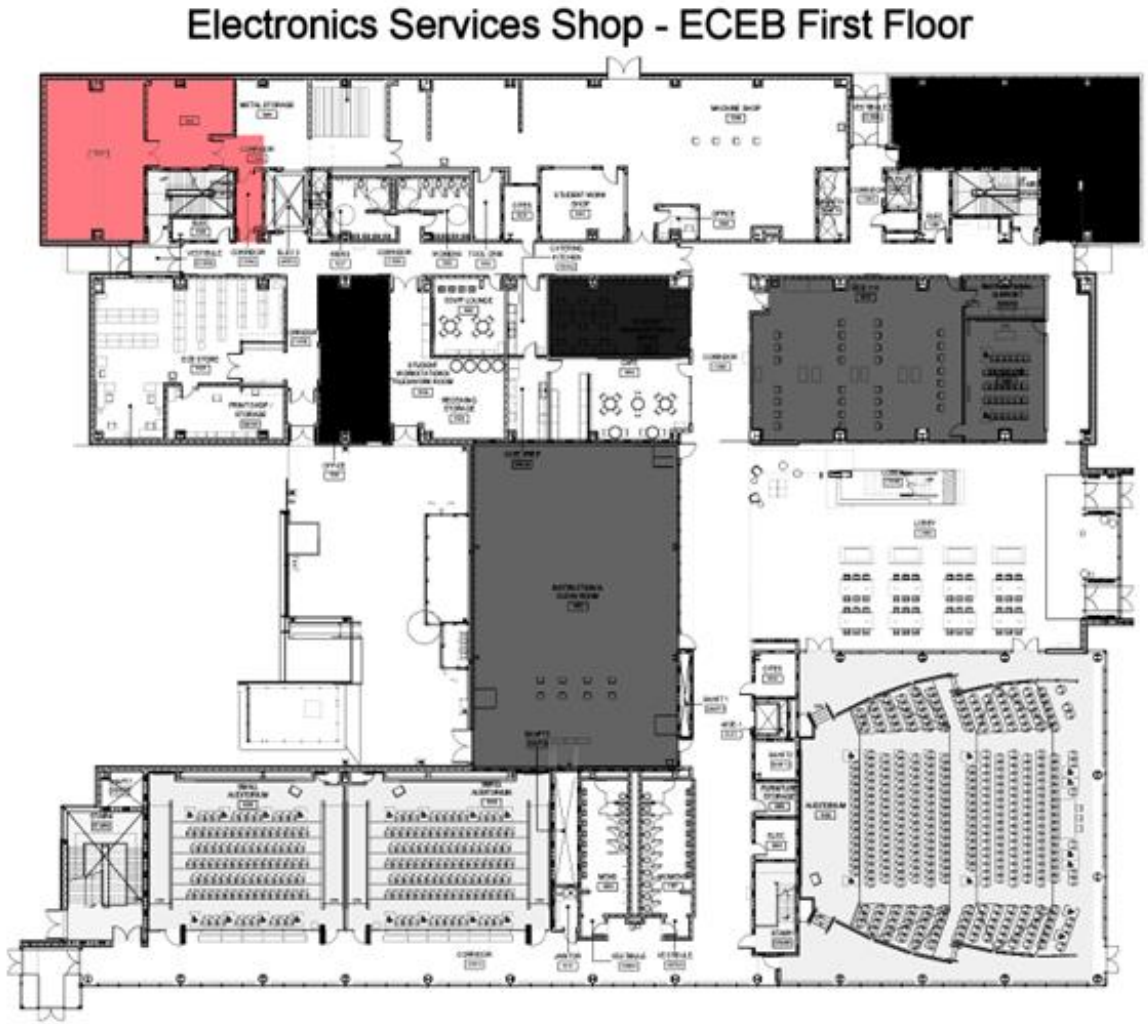
General Reminders

- Best practice is to prototype your idea before designing your PCB (**Breadboard Demos**).
- It takes about 7-10 days to go from PCB wave deadline to delivery
- Consider assembly before choosing PCB components

- Please keep the benches and the soldering stations clean
- A quick wipe-down or tidying up after you are down goes a long way

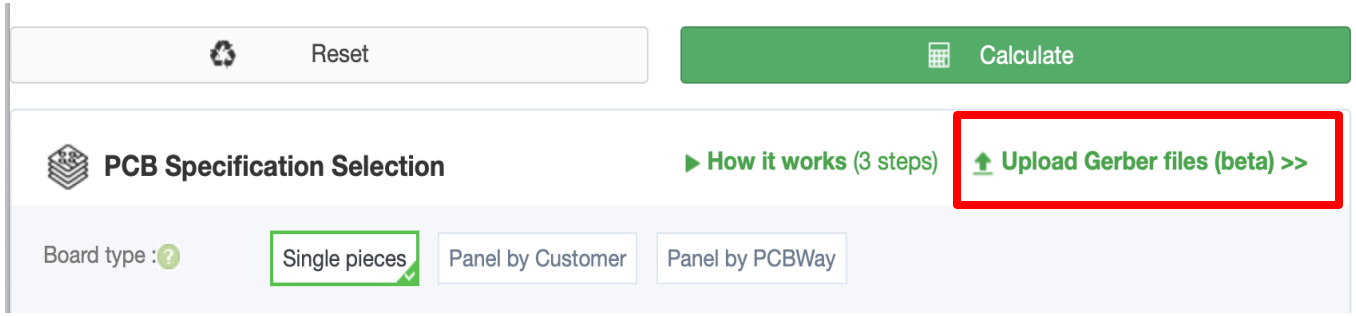
General Reminders – Getting Parts

- Parts can be procured from
  - E Shop : by TA ordering or student self service drawers
  - Online CFOP Orders
- Most required parts can be obtained via E Shop including MCUs, connectors, SMD components, etc.
- Tall White cabinet in Senior Design Lab
  - Ask a TA to checkout a part for you



PCB Ordering Guide

- Maximum Size of PCB is **100mm x 100mm**
- Like you did in the CAD assignment, export your gerber files into a zip
- Make a PCBWay account and upload your files
- Save to cart
- Make sure your board passes their auditing process
  - You will get a notification in your cart
- Once it does, send your zip file to your project TA. Modify the file name with your team number

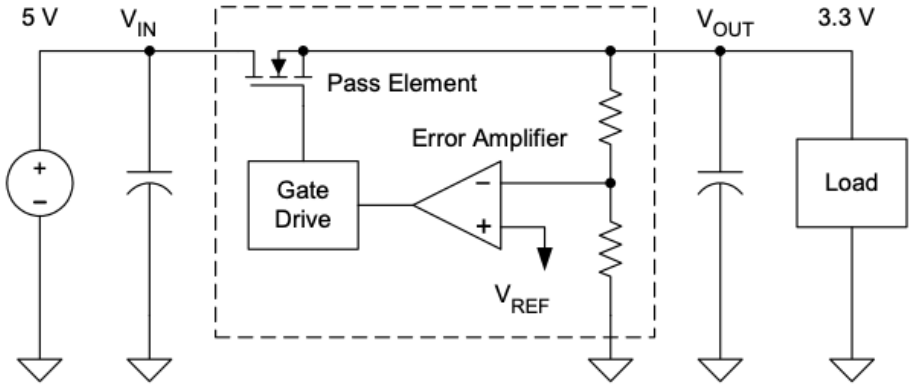


Basic Power Electronics - Regulators

- Regulators create a constant output voltage from a higher voltage input
- Require a certain amount of voltage headroom, i.e. “dropout voltage”
- At steady state conditions, it behaves as a resistor – dissipates heat
- Designed to maintain constant output voltage under dynamic loads
- Account for current rating

Two common types:

- Linear
  - Larger dropout
- Low Dropout
  - Lower drop out voltage, typically worse load handling

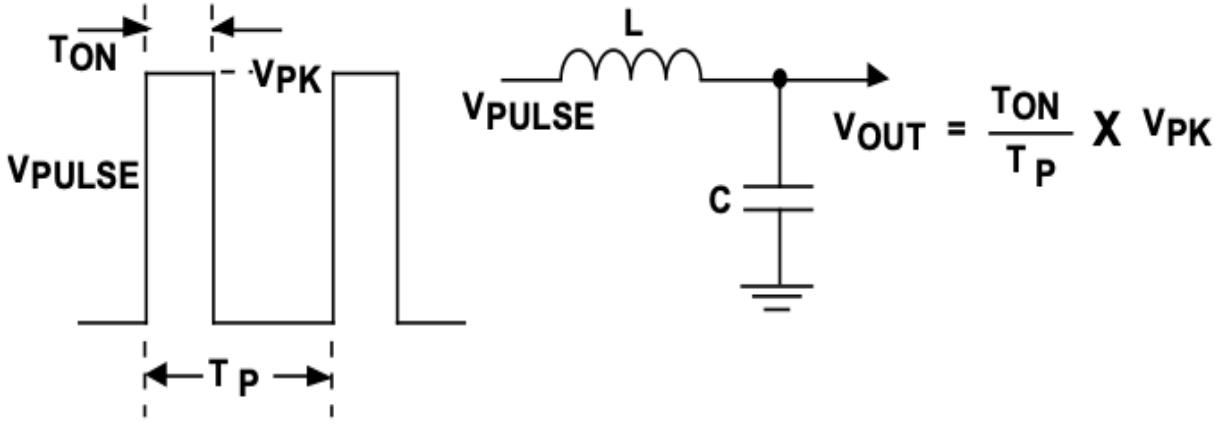


Basic Power Electronics – Switching Regulators

Higher conversion efficiency but noisier power supply
Typically greater design flexibility.
Basic principle a dynamic PWM signal that adjust its duty cycle to regulate output voltage

DC/DC Converters:

- Buck (High V to low V)
- Boost (Low V to low V)
- Flyback (Multiple output configurations)
- Inverting

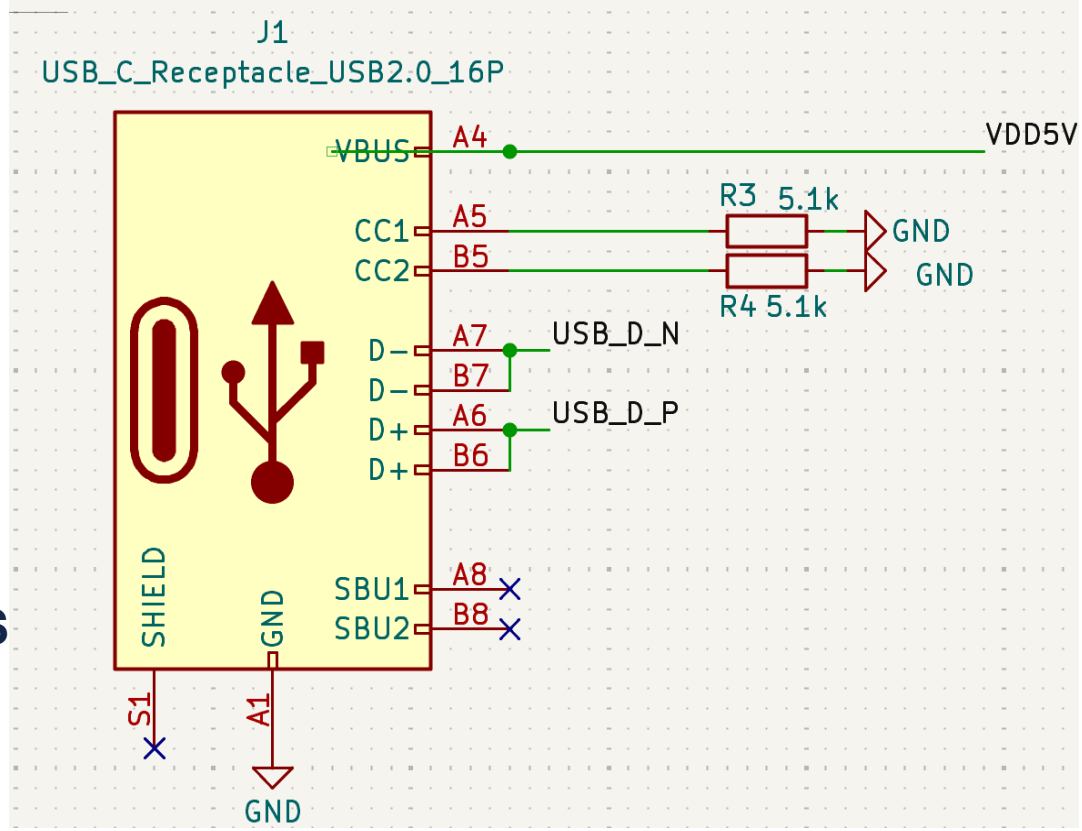


Basic rule of thumb : if your voltage drop is around greater than 5V, consider a switching regulator

USB-C Connectors

USB-C connections provide power and data. The hardware configuration determines the utility of the connector in your circuit according to the USB-C standard:

- **USB 2.0**
- **USB 3.0**
- **USB PD**
- **Others**



In its simplest configuration (right), the connector handles US 2.0 communication and can handle 15W(5V, 3A)

- **CC1, CC2 are tied down by 5.1kOhm Resistors**

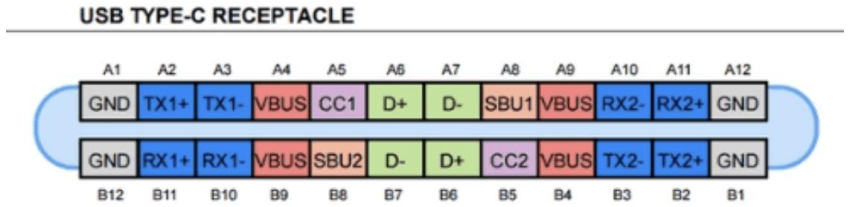
TI has a nice USB-C guideline you can refer to for more complex implementations

Choose the right USB C Connector for your purpose!

Any USB C receptacle will have 24 pins, but not all of them used and will be ignored in plenty of cases.

Although USB C is a standard, connectors come in different configurations:

- **6 pins, 12 pins, 18pins, etc.**
  - **Typically more pins is to allow for more data intensive applications**
- **through holes, smd**
- **Vertical Mount, Horizontal Mounts**



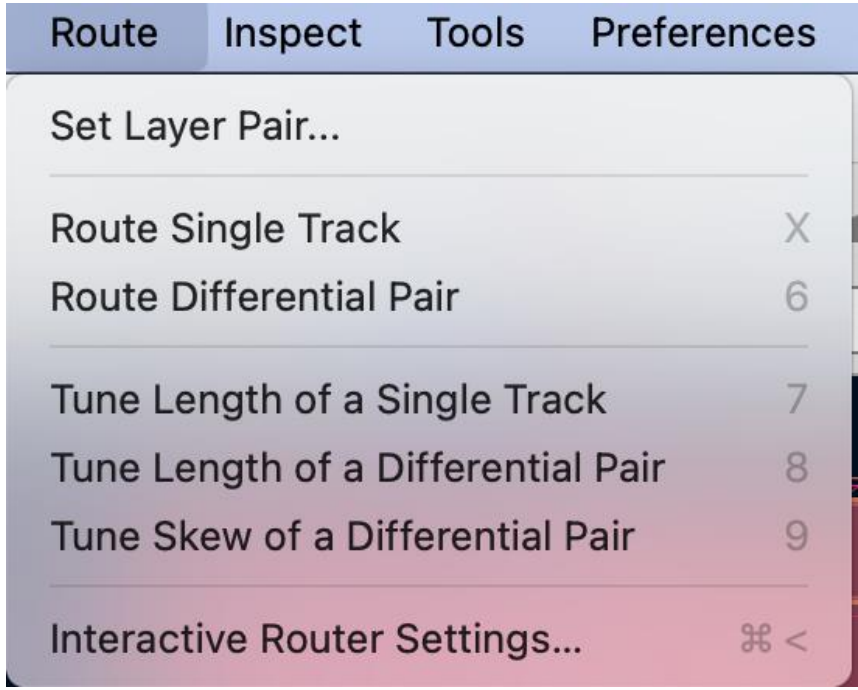
Check the physical drawings, CAD models, and datasheets for your connectors (and other parts!)

Differential Signaling (USB 2.0, USB3.0, CSI, etc.)

- **Differential signals improve noise performance by having complementary datalines**
- **In hardware, the implementation of this standard requires 2 coupled transmission lines of equivalent length**

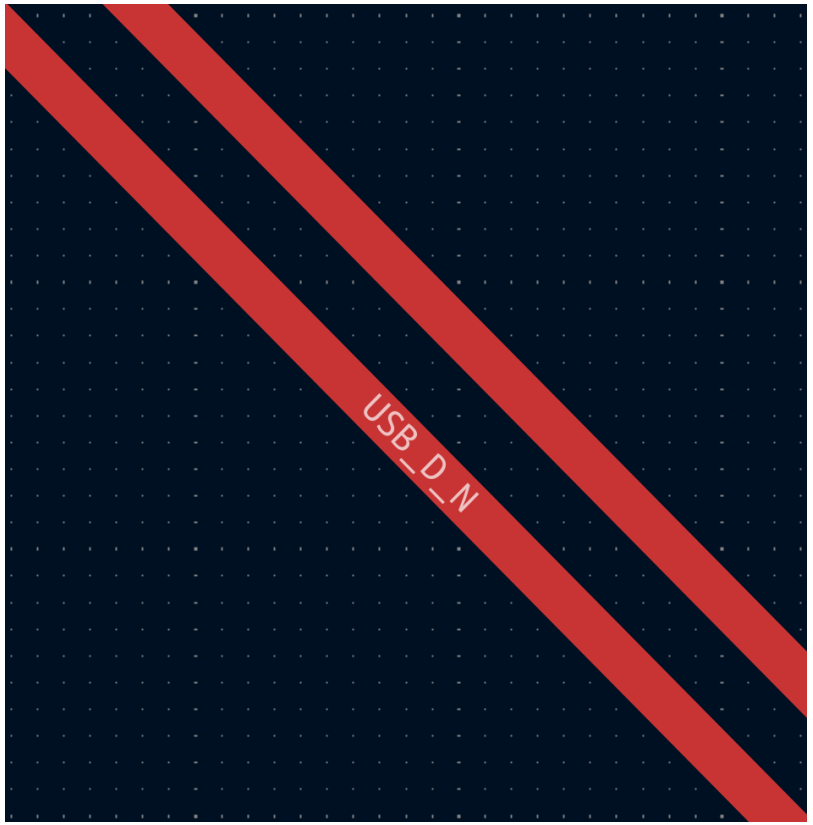
- **+/- lines in USB 2.0 for instance**
  - **USB 2.0 Signal lines require 90 Ohm Impedance with some tolerance**

- **In KiCAD, use the following tools:**
  - **"Route Differential Pair" to initate routing process**
  - **KiCAD follows a naming convention for designating differential traces**
  - **"Tune Skew of a Differential Pair" to prevent length/phase mismatches**

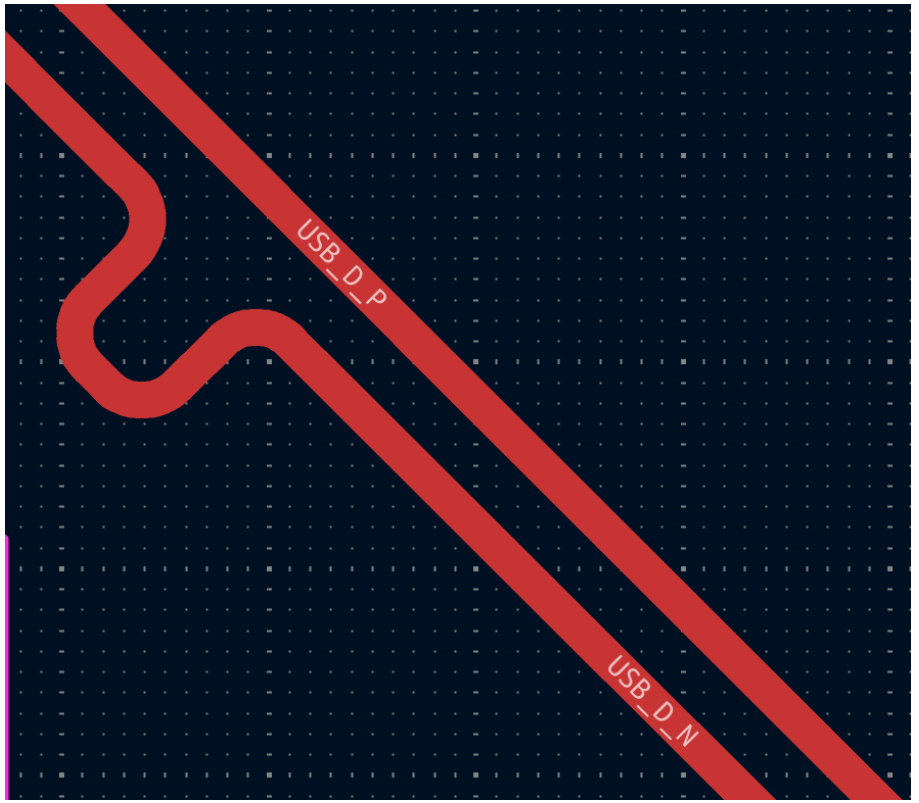


Tuning Skew of Differential Signals:

Before



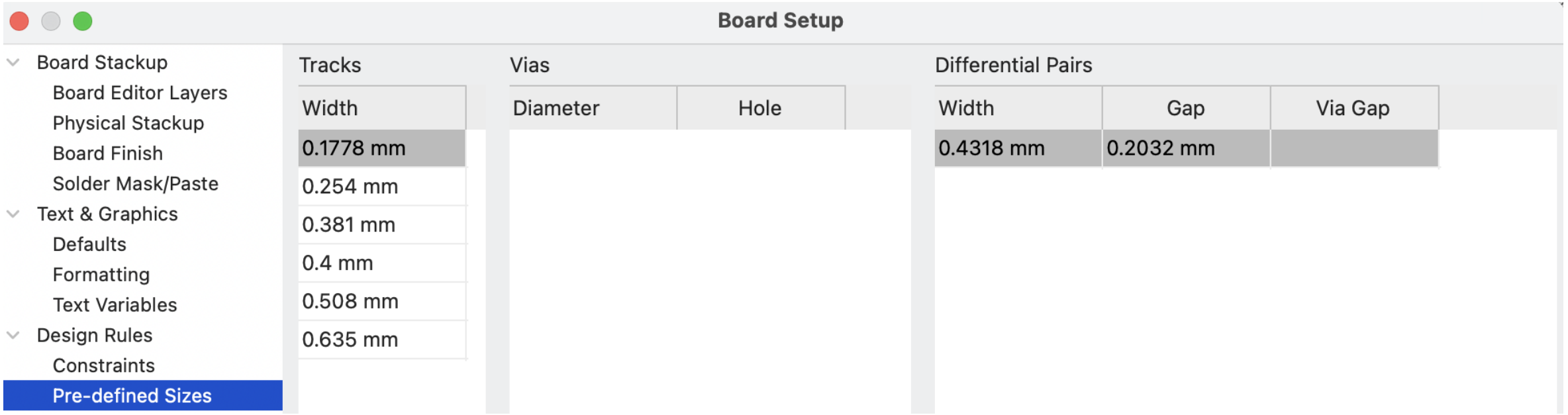
After



Custom Trace Sizing

Under "Board Setup > Design Rules > Pre-Defined Sizes" you can set trace geometry as desired

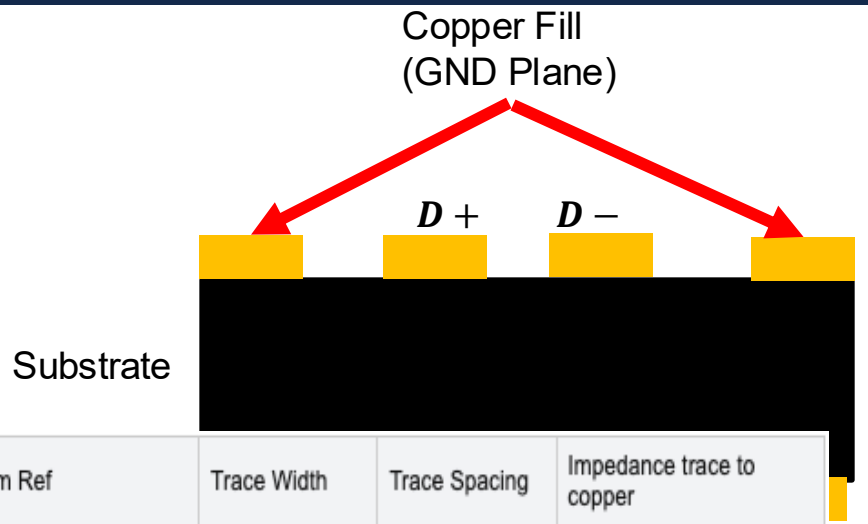
Board Setup ICON



Impedance Matching (USB 2.0) and Trace Sizing

Some parts of your system will demand certain requirements on your trace geometry
For example, USB 2.0 requires **90 Ohm** differential impedance $\pm 15\%$

If your board matches what is shown on the right, a decent approximation of a model is to use a differential coplanar waveguide model



Websites like JLCPCB, Digikey, and even KiCAD has models for these calculations you can use easily

Impedance (Ω)	Type	Signal Layer	Top Ref	Bottom Ref	Trace Width	Trace Spacing	Impedance trace to copper
90	Coplanar Differential Pair	L1	/	L2	17.2600	8.0000	8.0000

Layer	Material	Thickness (mil)	Thickness (mm)
L1	Outer Copper Weight	1.38	0.0350
Core	1.5mm 1/1OZ with copper (double-sided)	57.68	1.4650
L2	Outer Copper Weight	1.38	0.0350

Your board geometry will influence your inputs into these online calculators! Pay attention to:

- **GND Planes**
- **Distance to GND planes**
- **Substrate Thickness**
- **Trace Spacing, copper thickness, etc.**

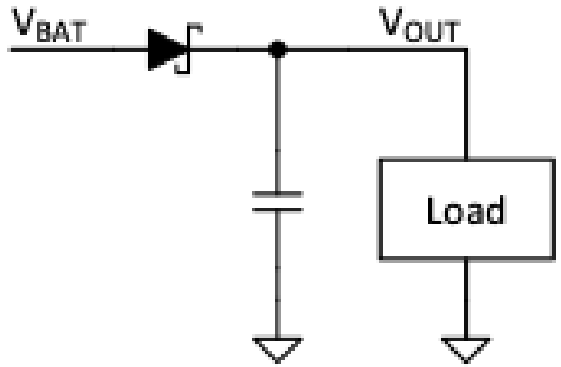
Protection Circuits

Schottkey Diodes – reverse current protection

- Diode responds quickly to changes in voltage polarity
- If multiple power supplies are present, this can be a good measure to prevent damage from miswirings
- Also good for batteries

Fuse – Overcurrent protection

- Protection from short circuiting events
- Ex. High in-rush current with motors



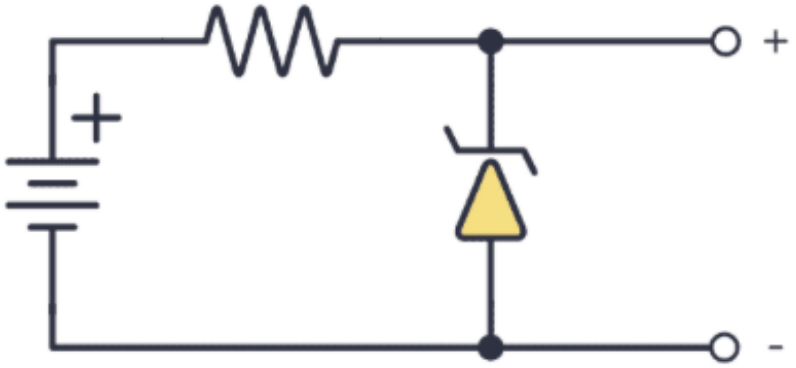
Schottkey Diode Ex.

Zener Diodes - Overvoltage protection

- Protect circuits from over voltage when in reverse bias.
- When a voltage pike goes beyond the Zener voltage, current flows through the Zener diode

Use cases:

- Can be used to clip waveforms
- Shunt regulator : surge protection, protection from transients, ESD, etc.



Zener Diode ex.

Power Routing – General Guidelines

The goal of power routing is to have a reliable source of Voltage/Current for your system needs.

This is made challenging due to parasitic effects:

- Ground Bounce – ground is not really ground
- EMI Issues - cross talk, lack of shielding
- Thermal Issues – your ICs may not be able to dissipate heat properly in any configuration

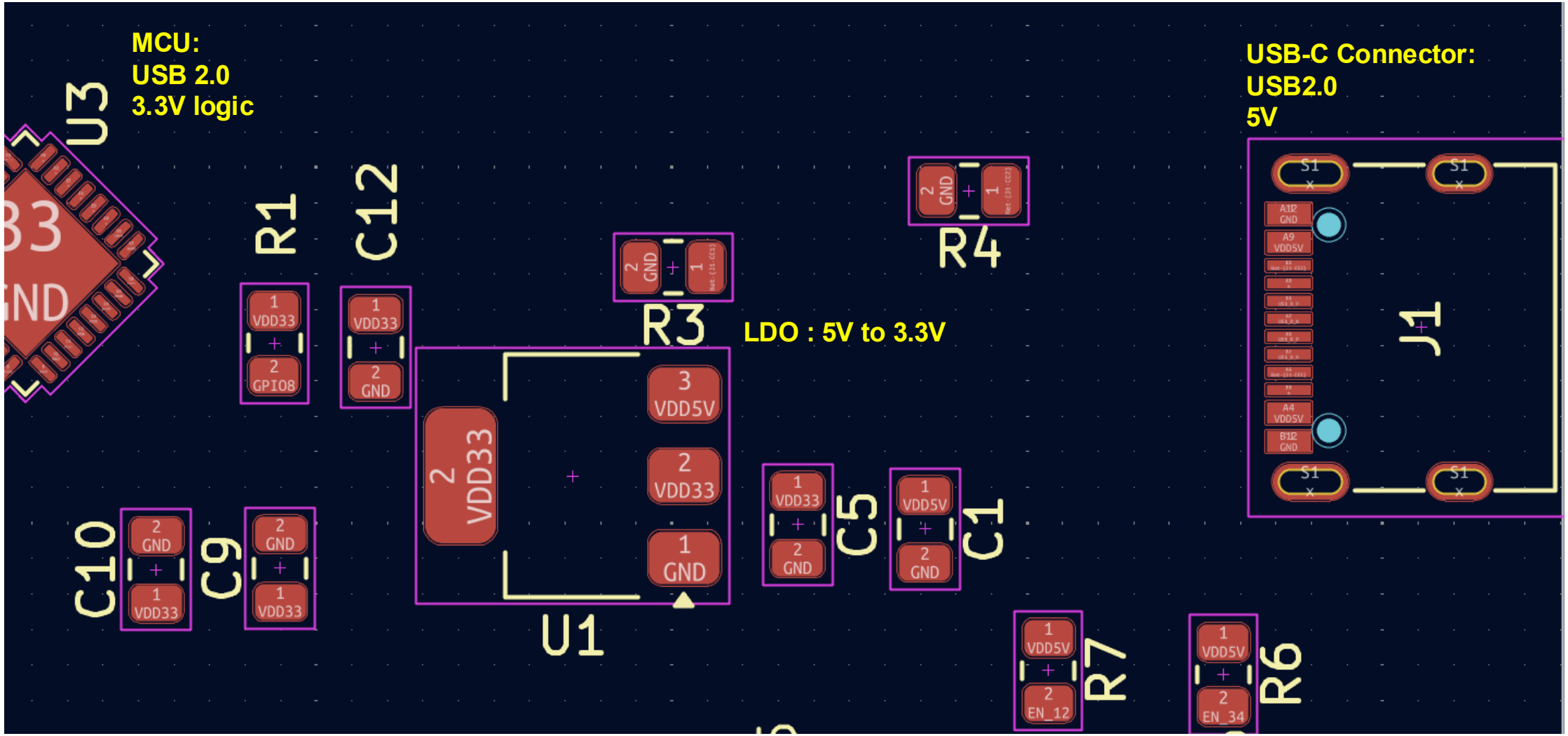
Unless you are doing a multi-layer board, there should not be a need for a dedicated power plane

- A wide trace dedicated to power will help
- 2-layer boards should suffice for majority of projects
- Determine trace width with a trace calculator
- Try to minimize trace length to reduce parasitic inductances

Position high power components with intention

- Aim for good airflow/heat dissipation
- Place these components close to one another

Example of Spacing Components



General Grounding Guidelines

Return paths are important to consider. Without one, current will find a random way back to the source, causing issues.

General Techniques/Guidelines:

Fill unused PCB areas with a copper ground

Try to have a ground path below/close to your signals

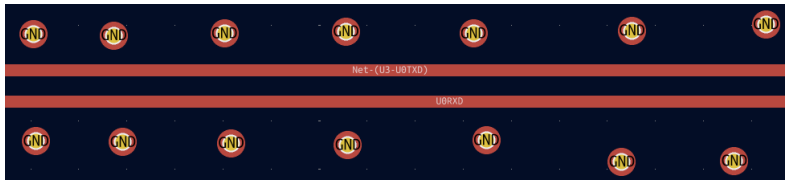
Use shielding vias for more sensitive signals

Use stitching vias to connect front and back GND planes

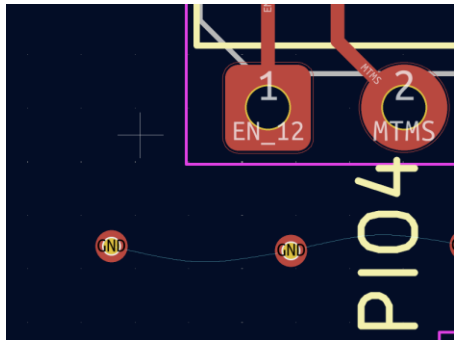
Do your best to keep components on 1 side first, try to avoid excessive use of backside components

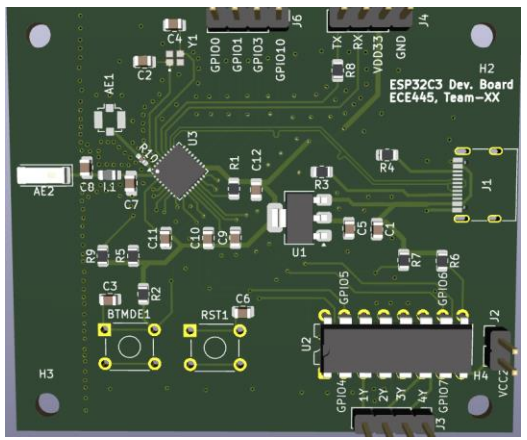
Grounding is a cool + complex topic! Especially as board designs get more demanding.

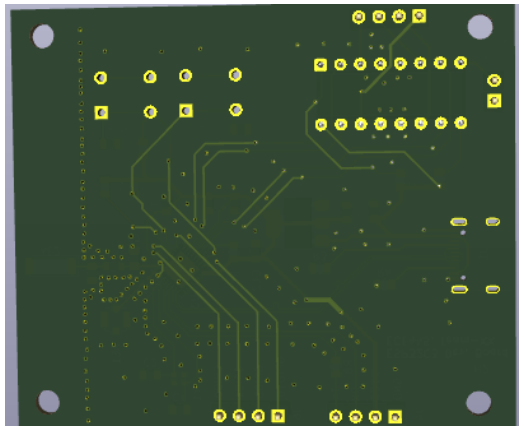
Shielding vias



Stitching vias







Do not ignore MCU Bootmodes

MCUs have boot modes that are defined by HW

This should NOT be wrong on your PCB or else there is 0 chance to program

Give yourself options, buttons for your for boot-modes are a good choice to have

4.1 Chip Boot Mode Control

GPIO0 and GPIO46 control the boot mode after the reset is released. See Table 4-3 *Chip Boot Mode Control*.

Table 4-3. Chip Boot Mode Control

Boot Mode	GPIO0	GPIO46
SPI Boot	**1**	Any value
Joint Download Boot [2]	0	0

[1] **Bold** marks the default value and configuration.

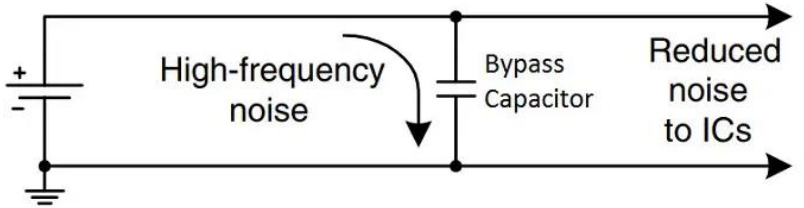
[2] Joint Download Boot mode supports the following download methods:

- USB Download Boot:
  - USB-Serial-JTAG Download Boot
  - USB-OTG Download Boot
- UART Download Boot

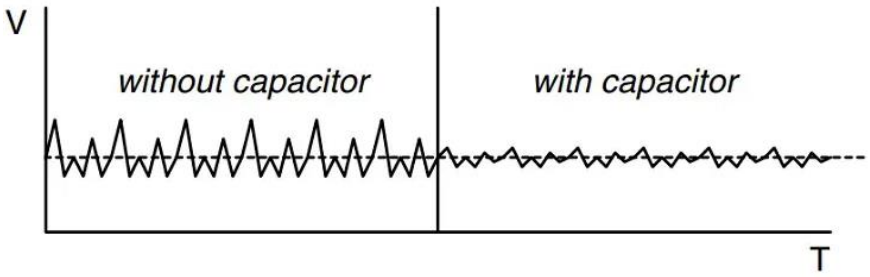
Good Practices and Common Mistakes

- **Test Points –** Everyone should have them!
  - Pads, Holes, are all basically free to use
- **Space Components apart appropriately** – make soldering easy
- **PCBWay DRC File**
- **Lack of Mounting Holes**
- **Connector Placement**
- **No right-angle traces**

- **Decoupling Capacitors**
  - Place near ICs Pins
  - Shunts noise to GND
  - 0.1uF, 1uF, nFs range are common
  - Your IC datasheets may have suggestions



- **Traces**
  - Make power traces wide (~20mils)
  - Signal traces width (~10 mils) typically good
  - Size traces to match impedance as necessary
  - Keep traces adequately spaced apart to reduce capacitive coupling/crosstalk
  - ~2-3x trace width spacing is usually good



Questions?

We can start the PCB review session