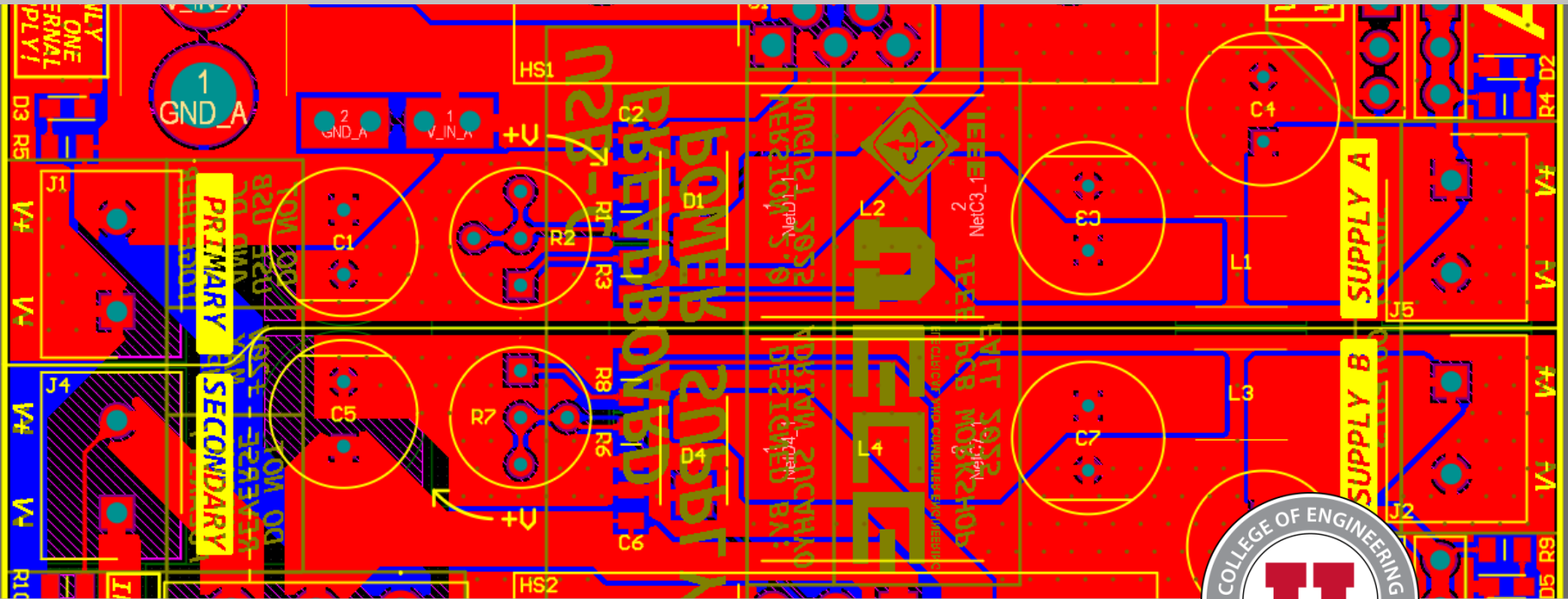# IEEE KiCAD Crash Course: (Week 03)
Introduction to PCB Layout

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

Printed Circuit Boards

- PCBs: A stack of conductive and non-conductive layers used to connect electrical components in a circuit

- Most common type of electronics assembly

- Components can be soldered to one or both sides
  - Increased Density at Increased Cost for Assembly

- Cheap to produce and manufacture

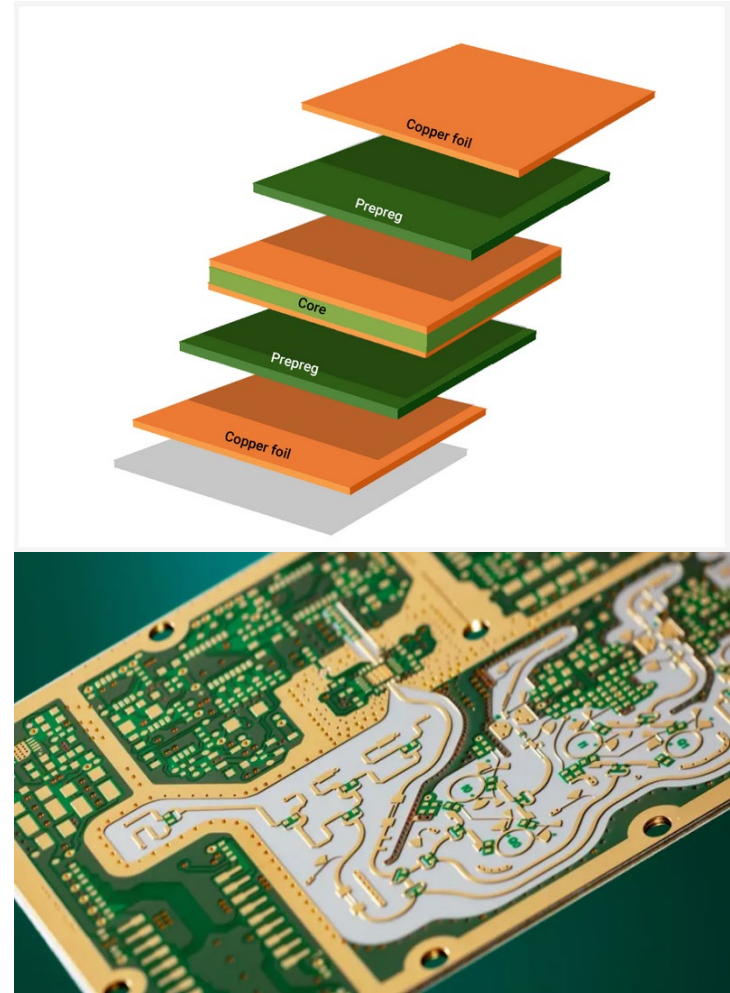https://www.viasion.com/blog/pcb-layers-traces-and-pads-explained/
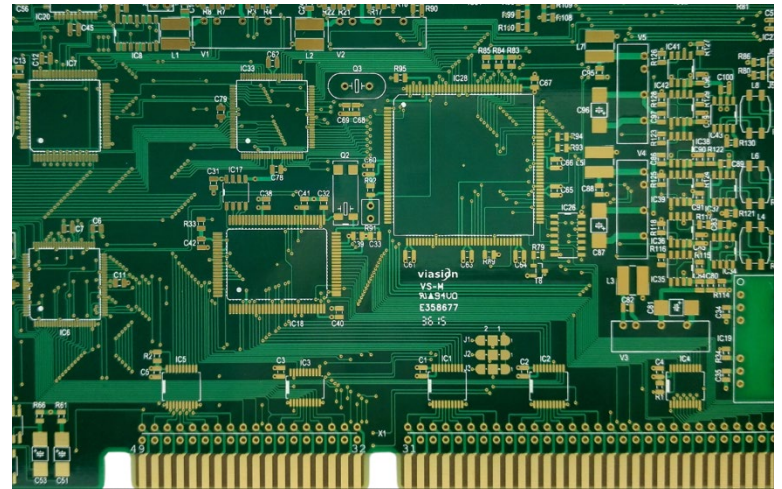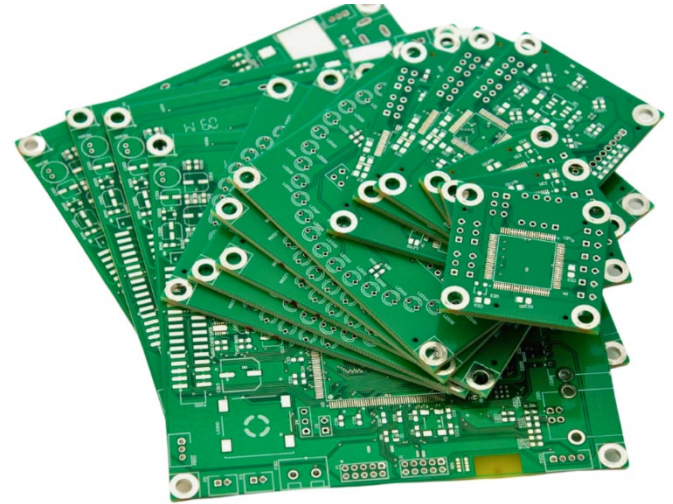
Why Use PCBs?

- Standard for most manufacturing processes
- Higher performance compared to breadboard or perfboard circuits
  - Less parasitic capacitances and inductances
- Relatively Durable
- Cheap to produce and manufacture
- High Density
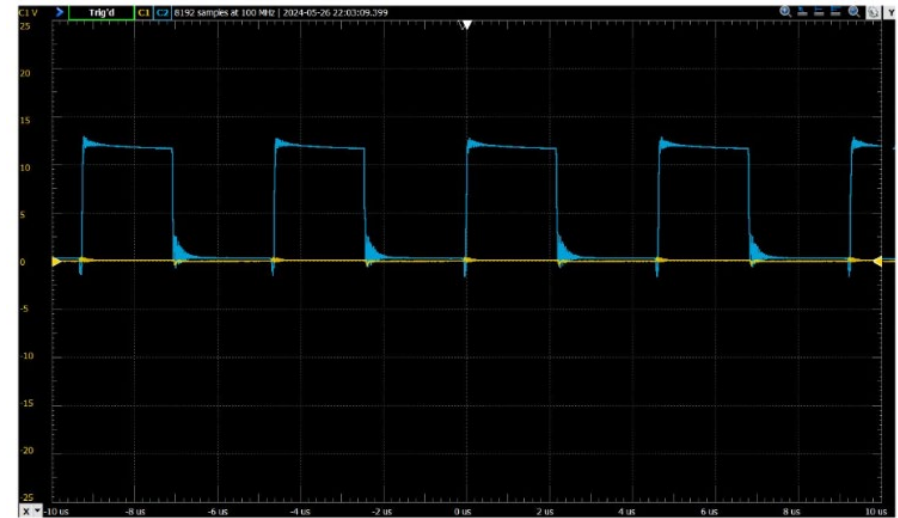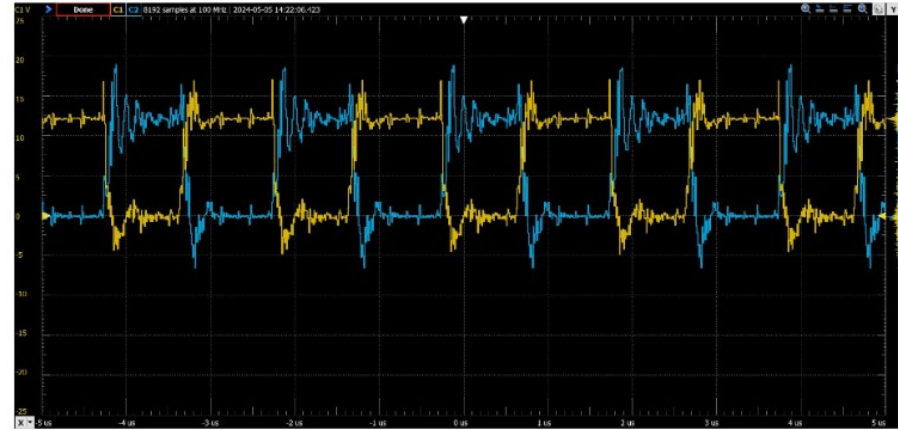
Nick Howard – Tesla Coil Ringing Demo, Adapted from SP25 Workshop

PCB Composition

- PCBs consist of many layers of insulators and conductors
  - Specific configurations is called the "Stackup"
- PCBs can be classified by the number of layers
  - 1-Layer, 2-Layer, 4-Layer, etc.
- Specific applications have tighter design tolerances
  - Ex: Rogers for RF/High Speed

https://www.protoexpress.com/kb/pcb-manufacturing-overview/
https://www.viasion.com/blog/rogers-pcbs-properties-materials-advantages-and-applications/

PCB Composition – Copper and Core

- Printed Circuit Boards (PCBs) typically begin as copper clad boards

  - Copper foil is adhered to both sides of a substrate – typically FR4 (flame-retardant fiberglass) but can be made of other materials (FR2, Polyamide, Teflon, etc.)

https://www.protoexpress.com/kb/pcb-manufacturing-overview/
https://www.protoexpress.com/kb/copper-for-pcbs/

DEPARTMENT OF
Electrical & Computer Engineering
COLLEGE OF ENGINEERING | THE UNIVERSITY OF UTAH
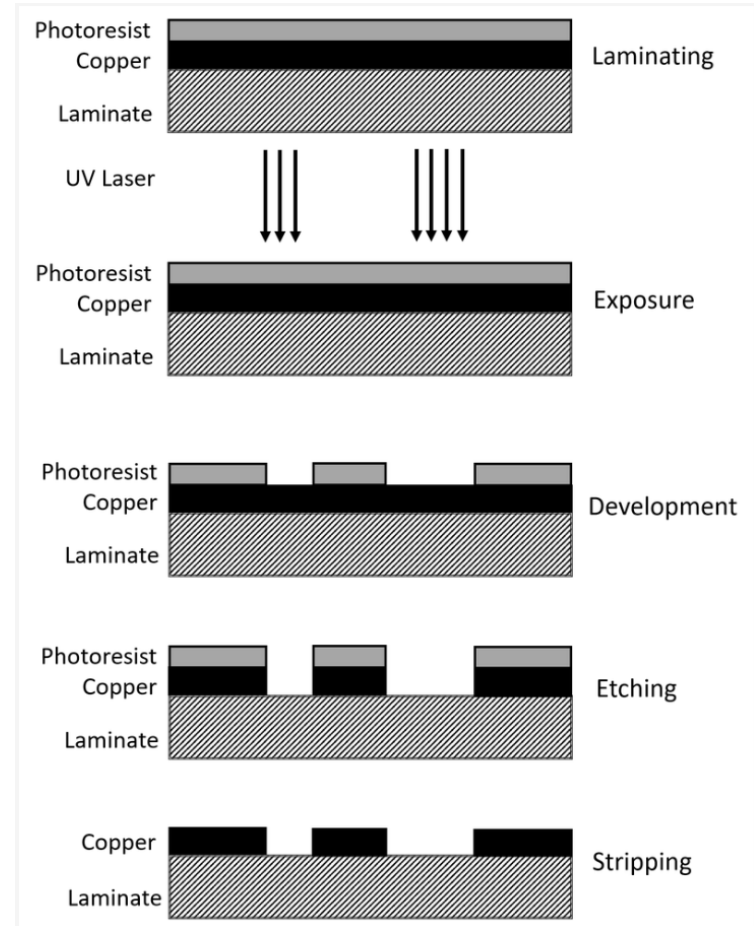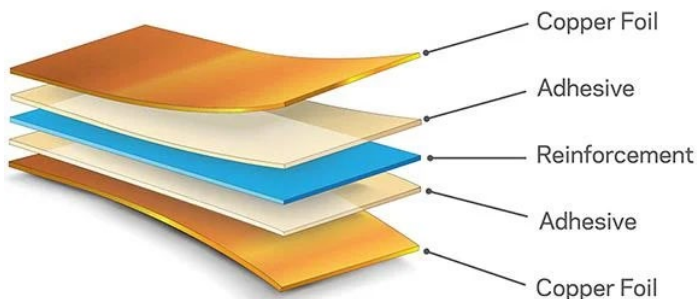
PCB Composition – Solder Mask

- The solder mask protects regions where solder is not meant to adhere to
- Solder masks result in a hardened layer on the board leaving intended regions exposed

DEPARTMENT OF
Electrical & Computer Engineering
COLLEGE OF ENGINEERING | THE UNIVERSITY OF UTAH
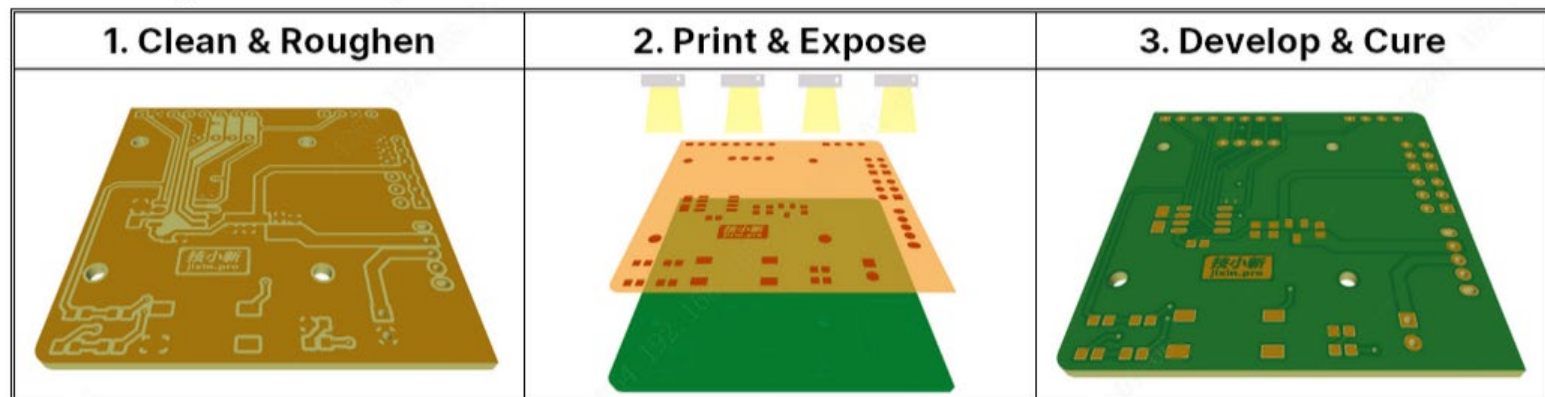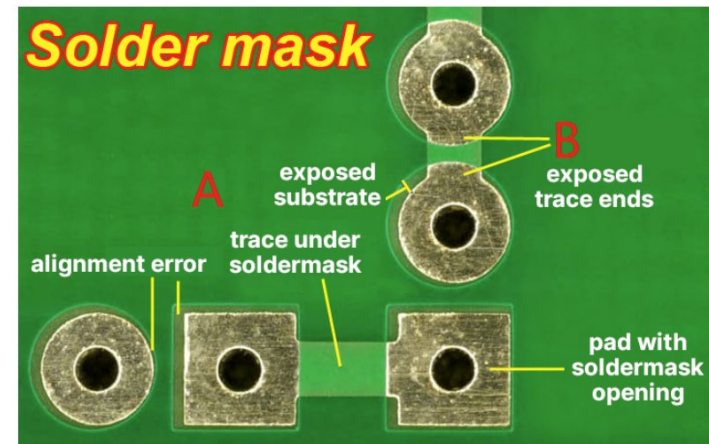
PCB Composition – Silkscreen

- Silkscreens contain all the text that you will see on the PCB
  - Typically contain component designators
  - Additional notes may be made on the silkscreen
- Silkscreens can only be applied to the solder mask
  - You can send a file to a manufacture that violates this rule, but you may end up with silkscreen on your pads

https://www.protoexpress.com/kb/silkscreen/
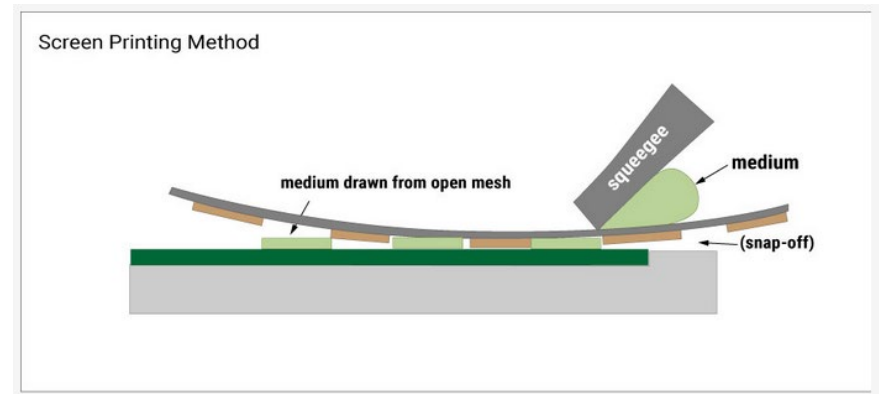
Traces

- Traces are the primary way of electrically connecting components to each other
  - Covered in solder mask to prevent short circuits
- Much like wires, traces have resistance and current capacity
  - Capacity dependent on copper weight and width
 - High Current traces must be wider than signal traces

https://resources.pcb.cadence.com/blog/2024-optimizing-for-pcb-trace-thickness-vs-current-capacity
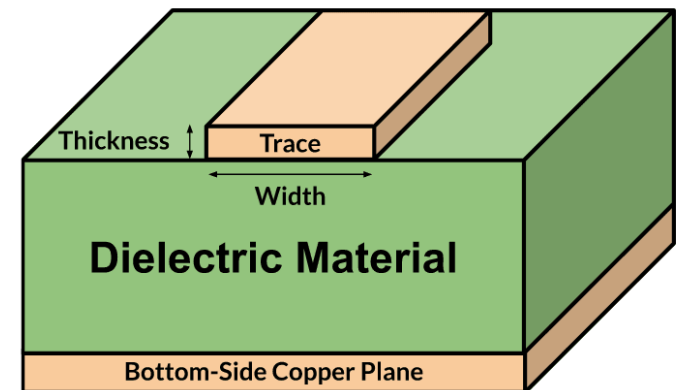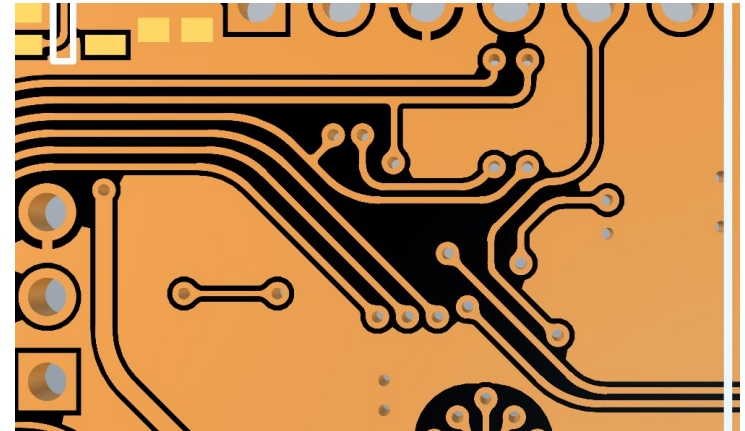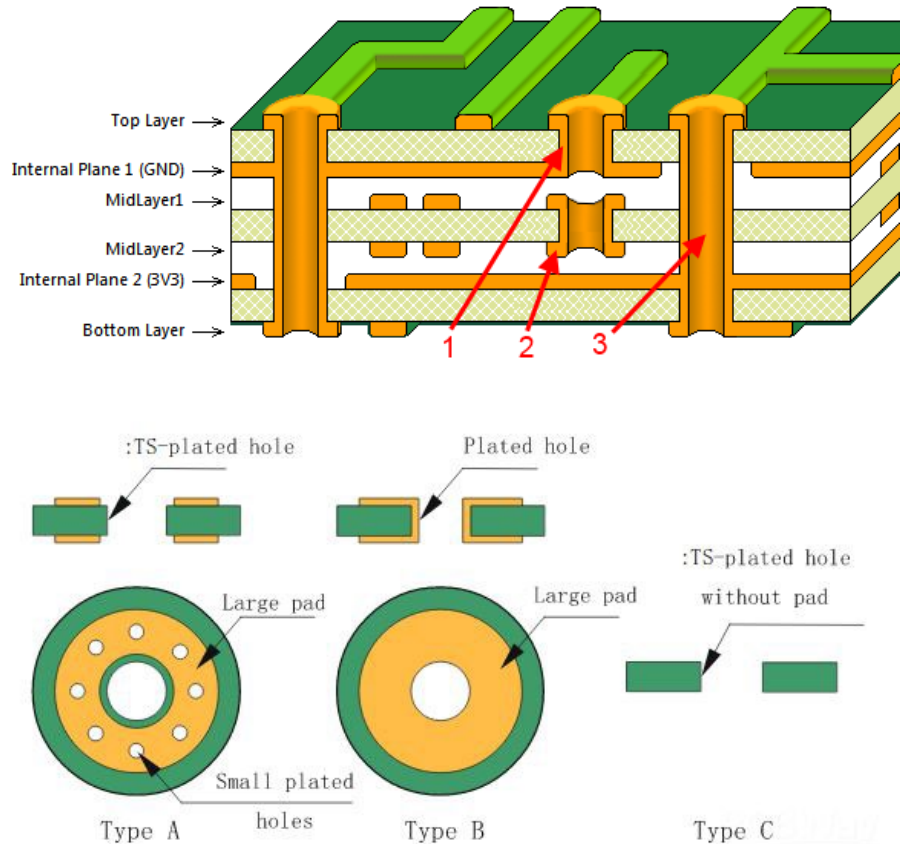
Planes

- Planes fill a specified region with copper to electrically connect them
  - Good for GND and Power distribution
  - Good current capacity (alternative to wide traces)
- Planes need special care to remove "necks" and "islands"
- Planes use "thermal reliefs" which need to be checked for capacity

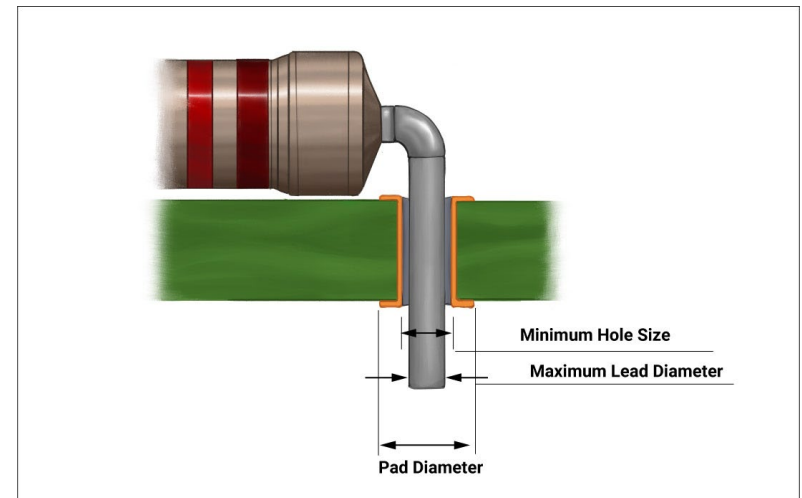https://resources.altium.com/p/should-you-route-signals-your-pcb-power-plane
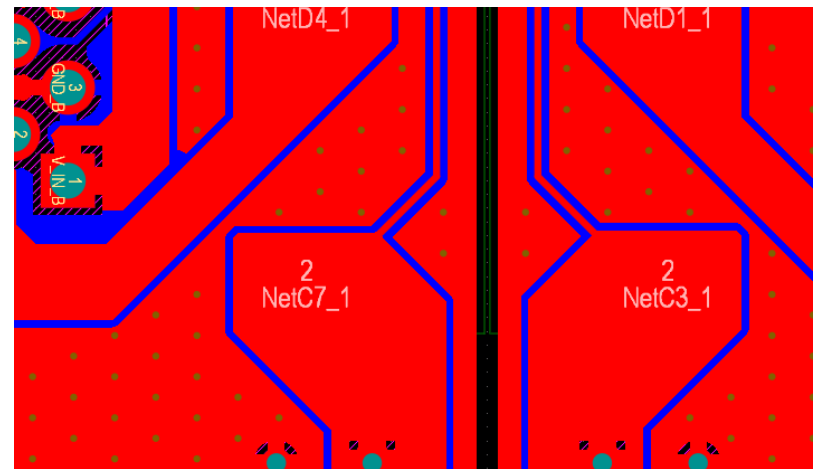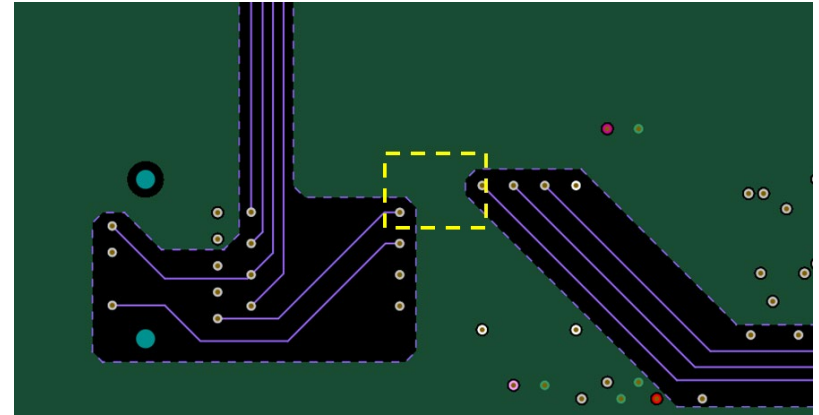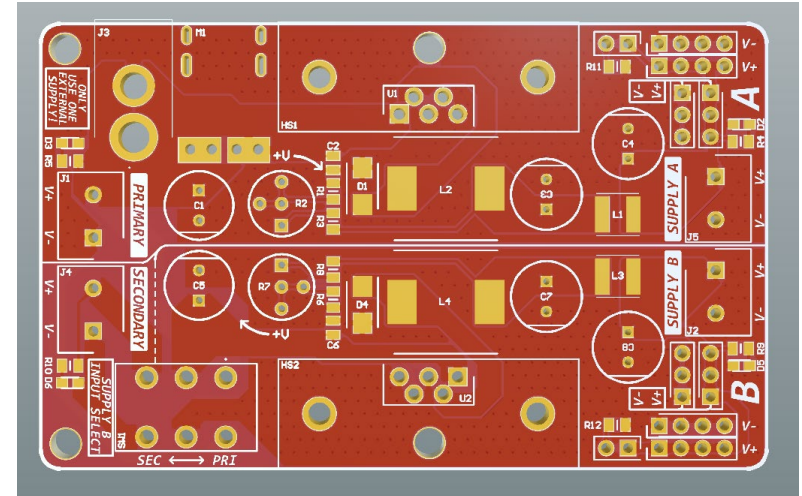
Vias and Drills

- Vias are holes used to connect between different layers of the PCB
  - Minimize the use of excessive vias when routing
  - Use multiple vias to increase current capacity

- Holes can be plated, non-plated, and have exposed copper if desired
  - Good for mounting points!

DEPARTMENT OF
Electrical & Computer Engineering
COLLEGE OF ENGINEERING | THE UNIVERSITY OF UTAH

Pads

- Pads are exposed copper that can be soldered to
  - Pads will not be covered with the solder mask
  - Pads can either be surface mount or through hole
- Pads dimensions are critical to ensure components solder properly to the PCB

Minimum Hole Size
Maximum Lead Diameter
Pad Diameter

https://www.pcbasic.com/blog/pcb-pad.html

DEPARTMENT OF
Electrical & Computer Engineering
COLLEGE OF ENGINEERING | THE UNIVERSITY OF UTAH

Component Footprints

- Footprints are the arrangement of pads that are used to attach components to the PCB

- Footprints must match the component to ensure

- Each component has a unique package, however there are standard sizes.
  - Ex: 0805, 0603, 0402
  - Ex: DIP, TSSOP, QFN
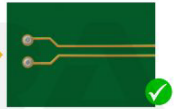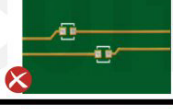
Solder Mask and Solder Paste Recommendations
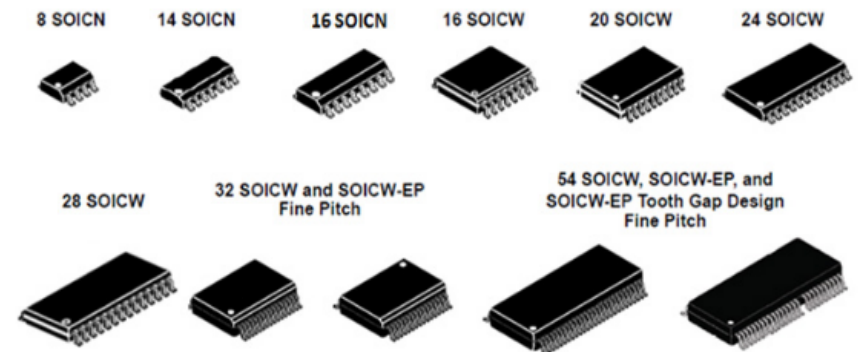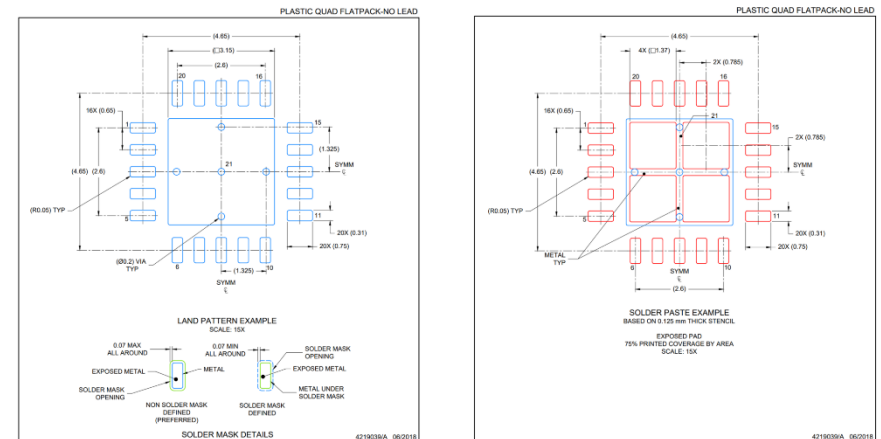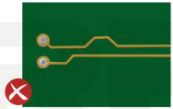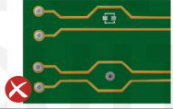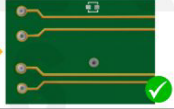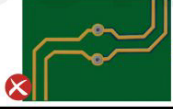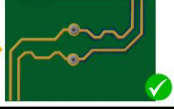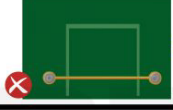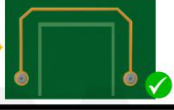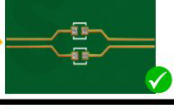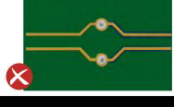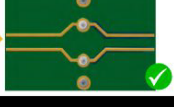
Figure 1. Standard Freescale SOIC Offerings

https://buildember.com/2024/06/24/component-footprints-what-should-you-include/
https://www.nxp.com/docs/en/application-note/AN2409.pdf

DEPARTMENT OF
Electrical & Computer Engineering
COLLEGE OF ENGINEERING | THE UNIVERSITY OF UTAH

Layout Tips

- This is the physical representation of your circuit, placement matters!

- Avoid "weaving" signals with vias

- Avoid running digital traces next to analog signals

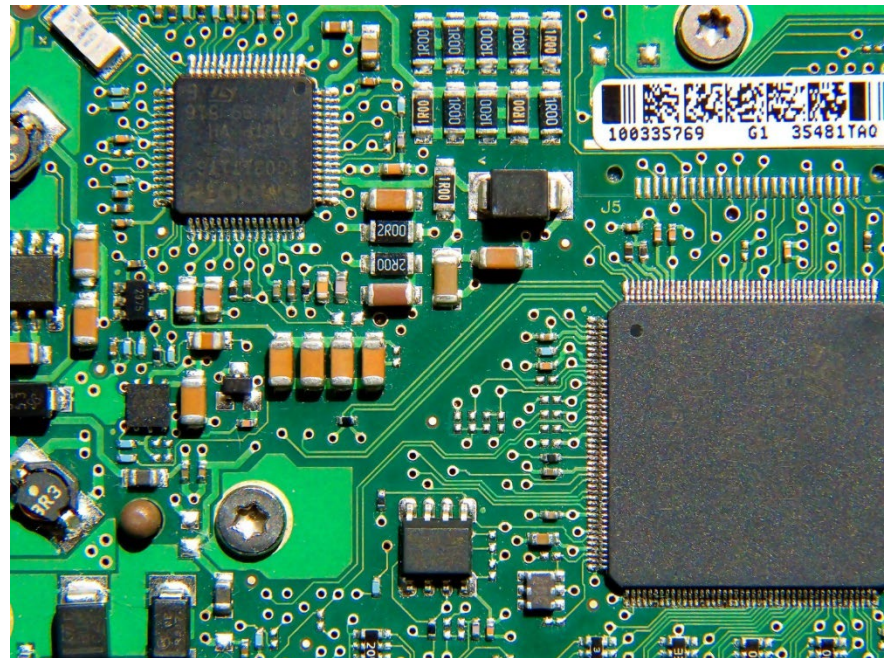- Ensure that traces will be able to handle the current it will carry

https://www.protoexpress.com/blog/best-high-speed-pcb-routing-practices/

Layout Tips

- Take time to place your components "nicely"
    - Align components
    - Establish a logical flow
- Label components and leave helpful notes for users
    - Ex: Voltage Range, Polarity
- Avoid 90 degree turns in traces
- Place decoupling capacitor close to target devices

https://predictabledesigns.com/7-ways-to-quickly-judge-the-quality-of-your-printed-circuit-board-pcb-design/

Project Information

- We will be going over the majority of the setup for PCB layout

- If you have not completed the schematic, you may use the reference to proceed

- We highly recommend you follow the reference board dimensions, especially the one highlighted

- Feel free to add more components, it's your own custom board!
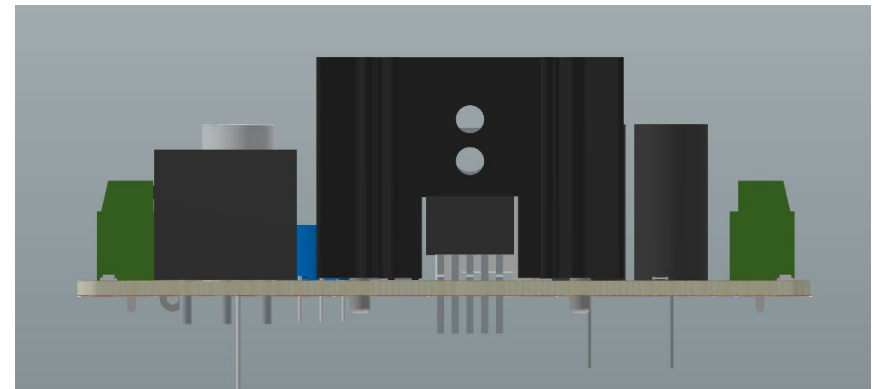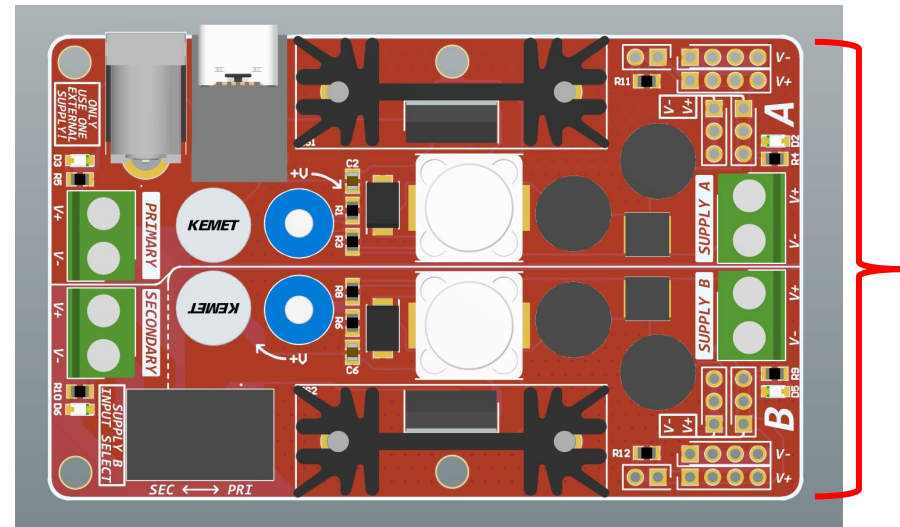
Questions?

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