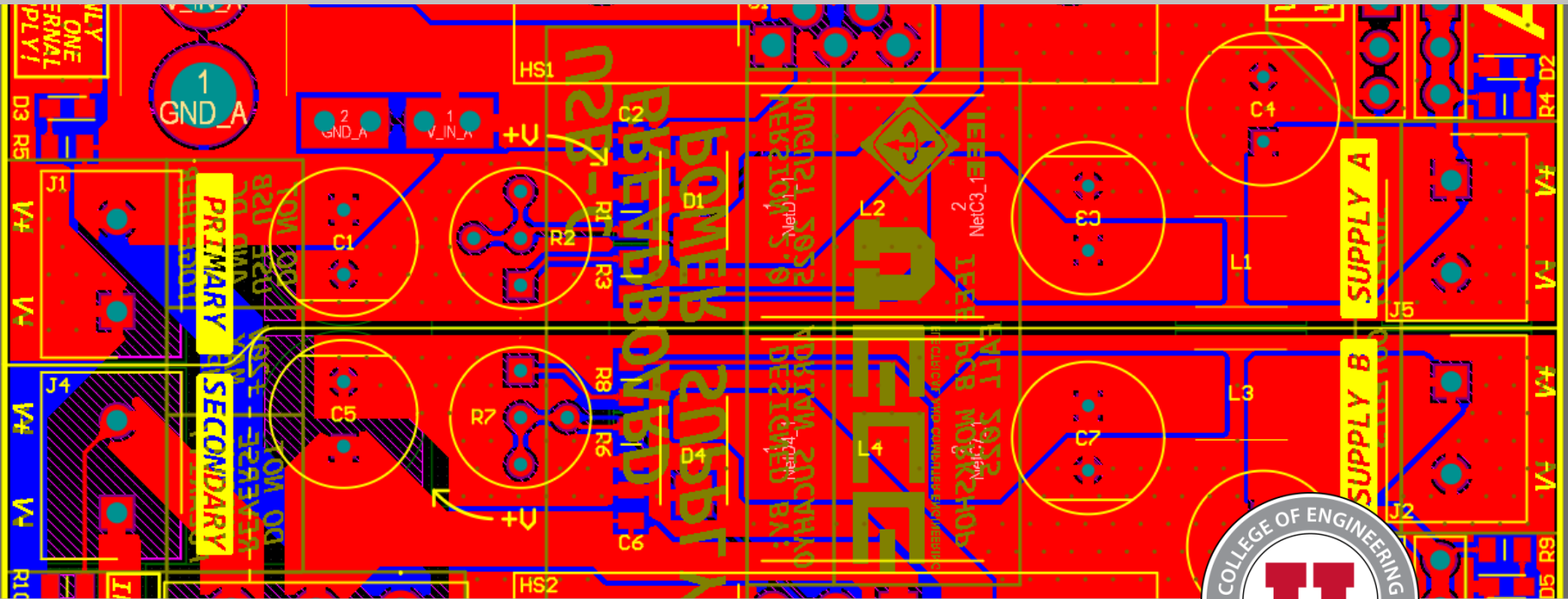


IEEE PCB Design Workshop: (Week 03) Introduction to PCB Layout



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

Workshop Outline

Tentative Schedule:

- Sept. 3 – Introduction to Schematics
- Sept. 10 – Schematics and Components
- Sept. 17 – Introduction to PCB Layout
- Sept. 24 – Layout Continued
- Oct. 1 – Open Work Session
- ** FALL BREAK **
- Oct. 22 – Soldering Week 1
- Oct. 29 – Soldering Week 2
- Nov. 5 – Soldering Week 3
- Nov. 12 – Final Notes and Next Steps

Announcements

- ASUU Budget Requests
 - We are currently still waiting for ASUU to process our budget requests.
 - We will keep you updated through email when we hear back.
- Alternative Projects
 - We will be able to get alternative project boards manufactured if submitted by the deadline.
 - Limited to the 10 cm x 10 cm dimensions outlined by JLCPCB.
 - Talk or email me if you have any questions!

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

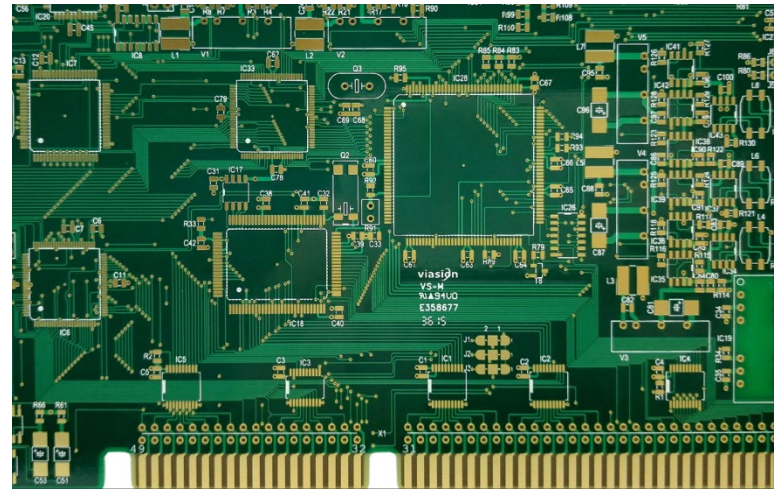
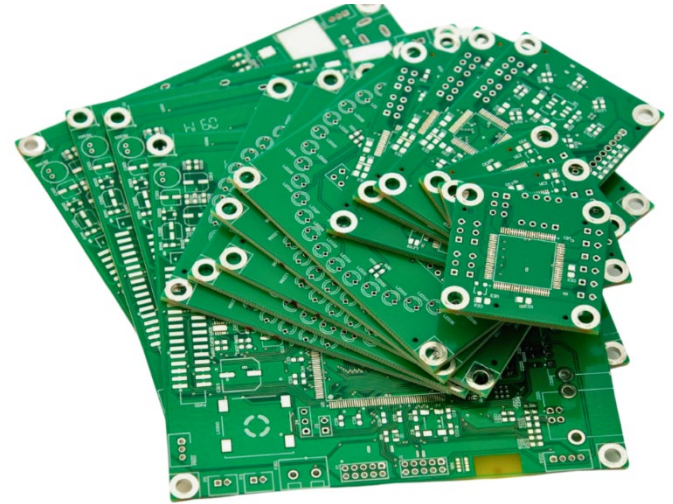


U of U IEEE Discord Link



Printed Circuit Boards

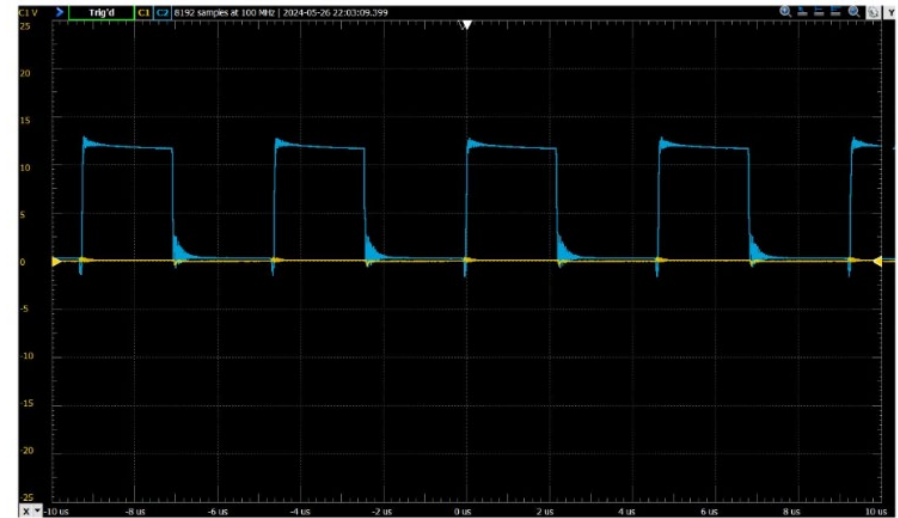
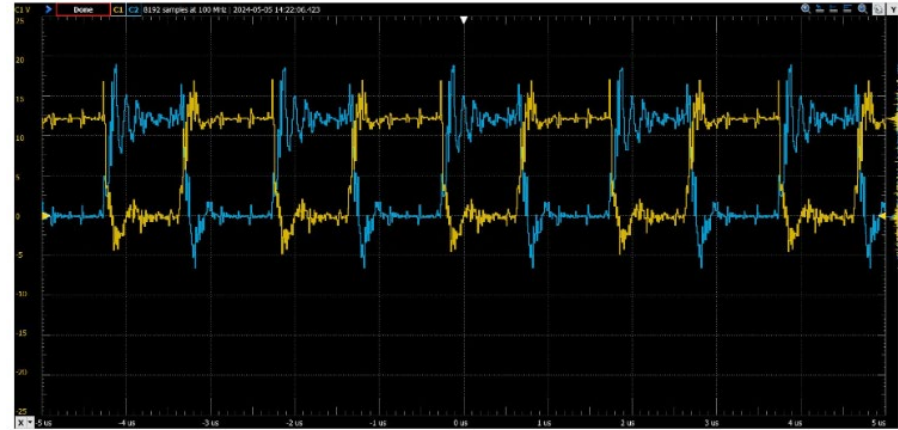
- PCBs: A stack of conductive and non-conductive layers used to connect electrical components in a circuit
- Most common type of electronics assembly
- Components can be soldered to one or both sides
 - Increased Density at Increased Cost for Assembly
- Cheap to produce and manufacture



<https://www.viasion.com/blog/pcb-layers-traces-and-pads-explained/>

Why Use PCBs?

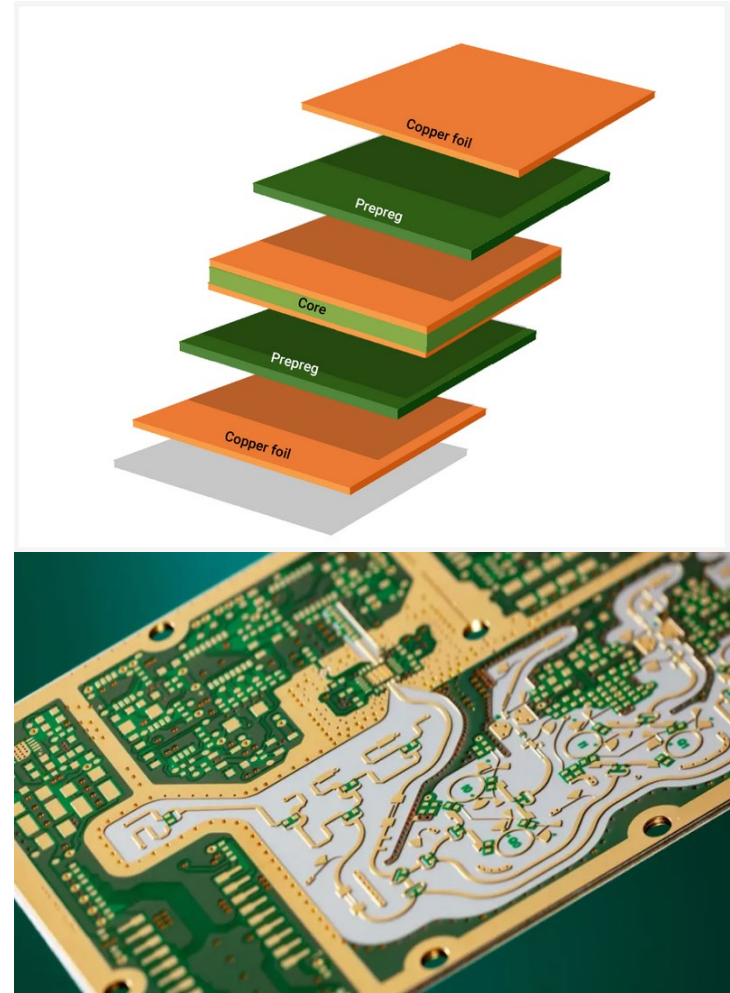
- Standard for most manufacturing processes
- Higher performance compared to breadboard or perfboard circuits
 - Less parasitic capacitances and inductances
- Relatively Durable
- Cheap to produce and manufacture
- High Density



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

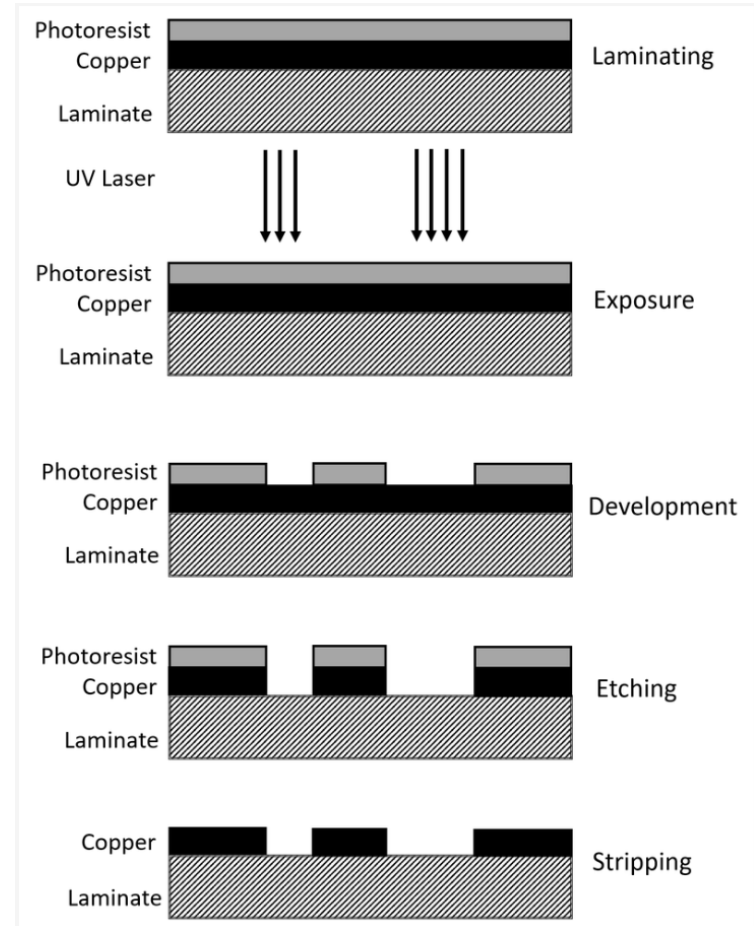
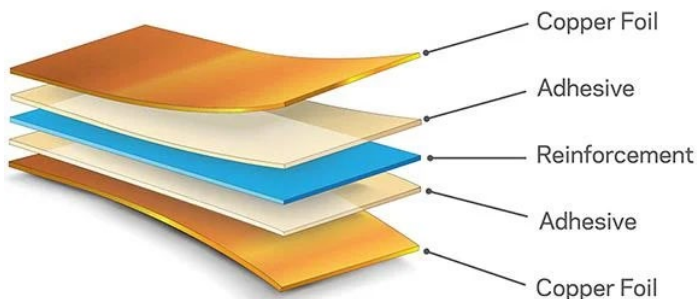


<https://www.protoexpress.com/kb/pcb-manufacturing-overview/>

<https://www.viasion.com/blog/rogers-pcb-properties-materials-advantages-and-applications/>

PCB Composition – Copper and Core

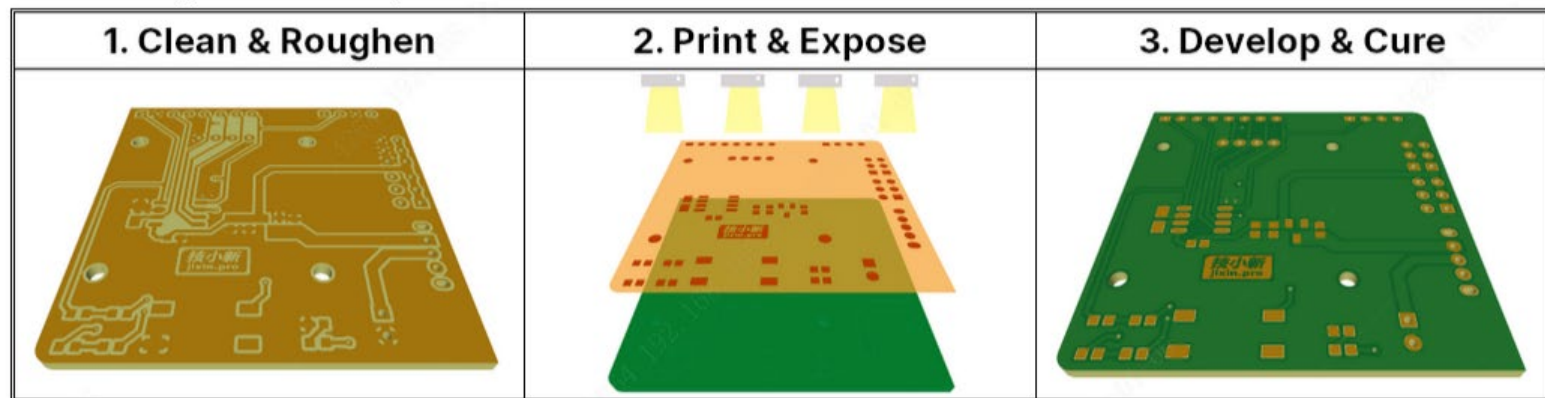
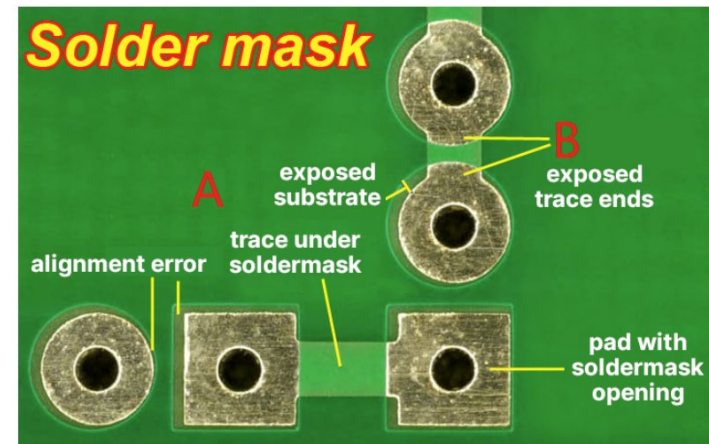
- Printed Circuit Boards (PCBs) typically begin as copper clad boards
 - Copper foil is adhered to both sides of a substrate – typically FR4 (flame-retardant fiberglass) but can be made of other materials (FR2, Polyamide, Teflon, etc.)



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

PCB Composition – Solder Mask

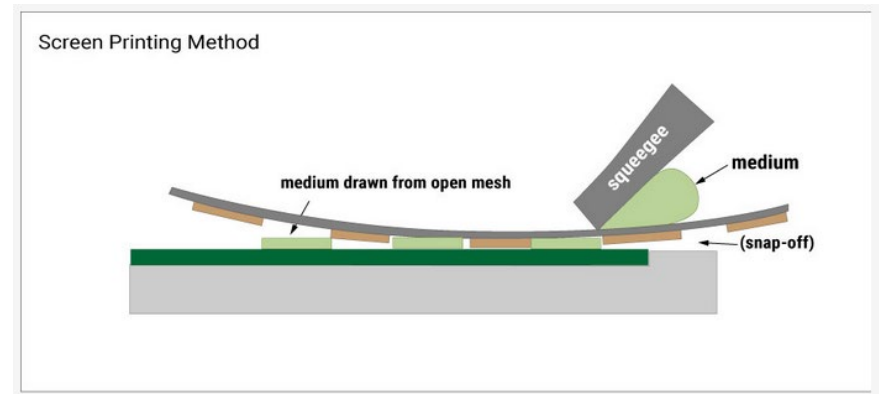
- The solder mask protects regions where solder is not meant to adhere to
- Solder masks result in a hardened layer on the board leaving intended regions exposed



<https://jlcpcb.com/blog/basic-design-of-solder-mask>

PCB Composition – Silkscreen

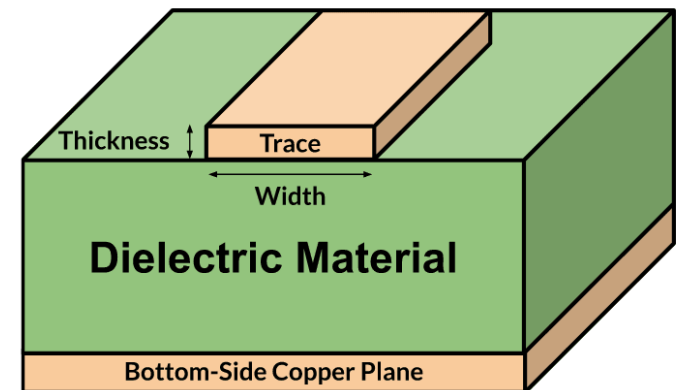
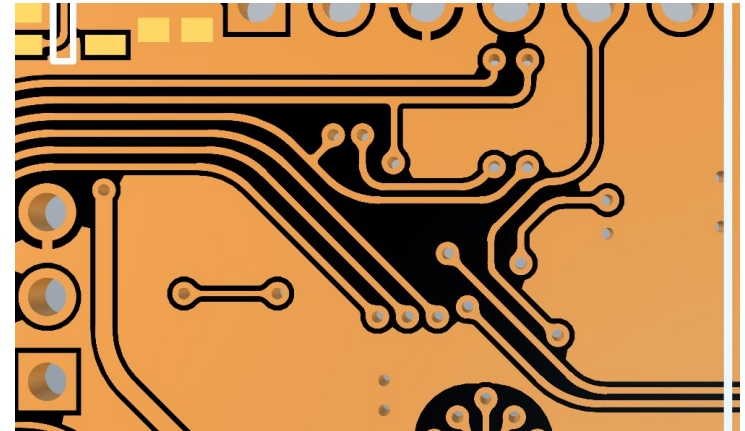
- Silkscreens contain all the text that you will see on the PCB
 - Typically contain component designators
 - Additional notes may be made on the silkscreen
- Silkscreens can only be applied to the solder mask
 - You can send a file to a manufacture that violates this rule, but you may end up with silkscreen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



<https://www.protoexpress.com/kb/silkscreen/>

Traces

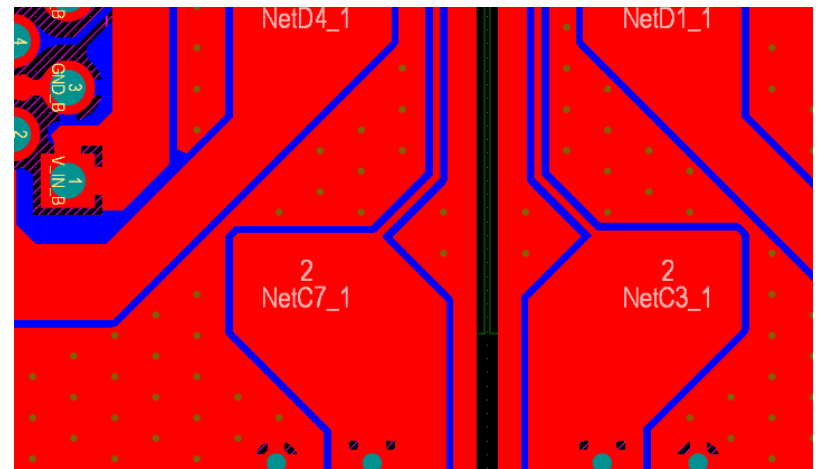
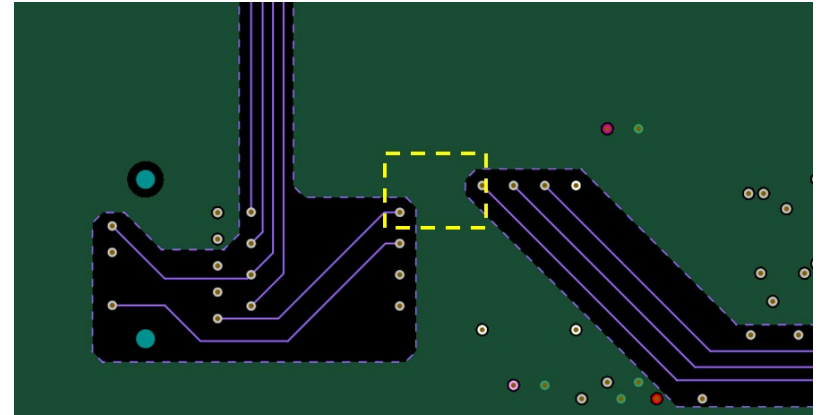
- Traces are the primary way of electrically connecting components to each other
 - Covered in solder mask to prevent short circuits
- Much like wires, traces have resistance and current capacity
 - Capacity dependent on copper weight and width
 - High Current traces must be wider than signal traces



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

Planes

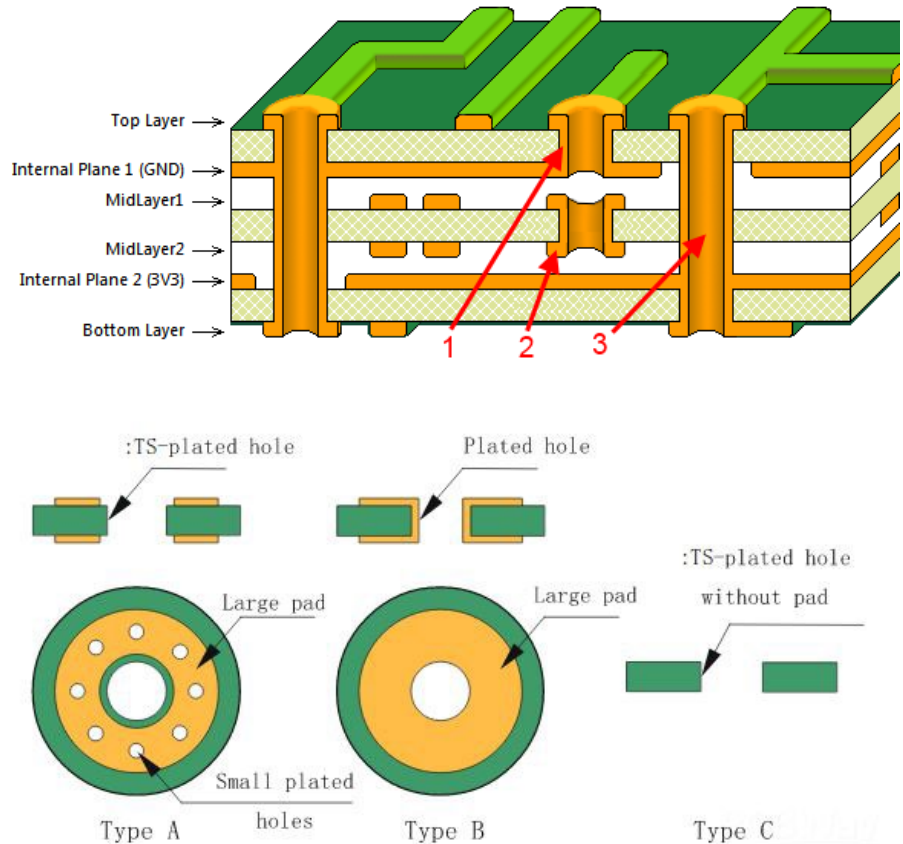
- Planes fill a specified region with copper to electrically connect them
 - Good for GND and Power distribution
 - Good current capacity (alternative to wide traces)
- Planes need special care to remove “necks” and “islands”
- Planes use “thermal reliefs” which need to be checked for capacity



<https://resources.altium.com/p/should-you-route-signals-your-pcb-power-plane>

Vias and Drills

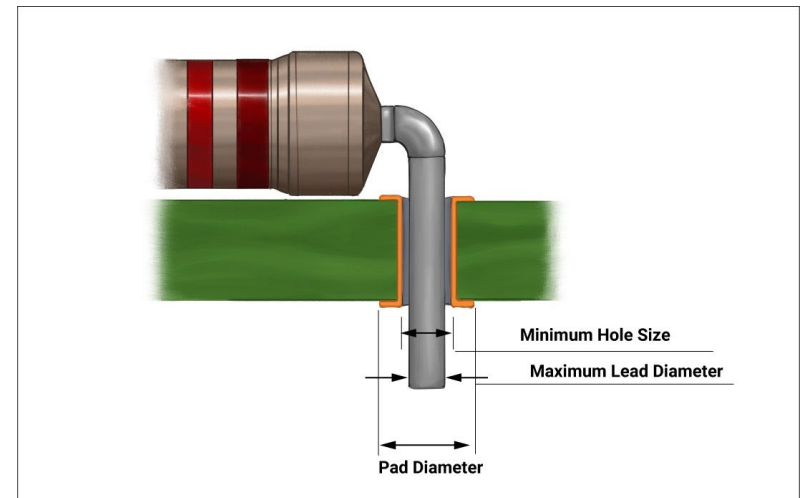
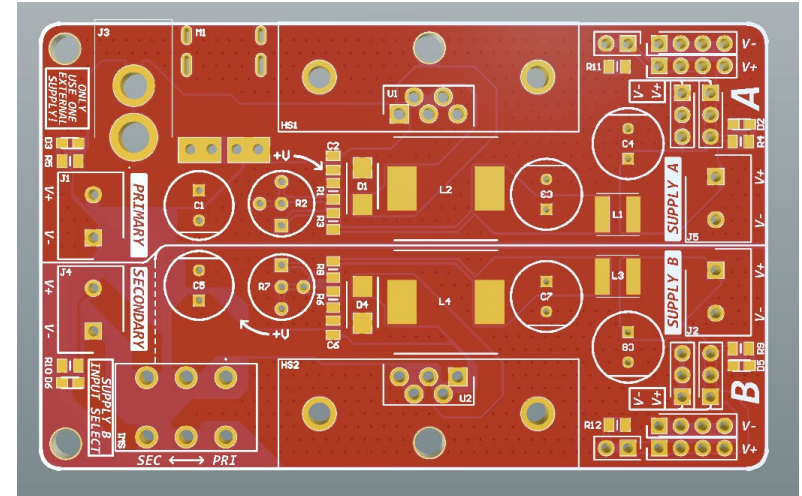
- Vias are holes used to connect between different layers of the PCB
 - Minimize the use of excessive vias when routing
 - Use multiple vias to increase current capacity
- Holes can be plated, non-plated, and have exposed copper if desired
 - Good for mounting points!



<https://www.teachmemicro.com/understanding-pcb-via/>
<https://statics3.seeedstudio.com/fusion/ebook/PCB+DFM+V1.0+.pdf>

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



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

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

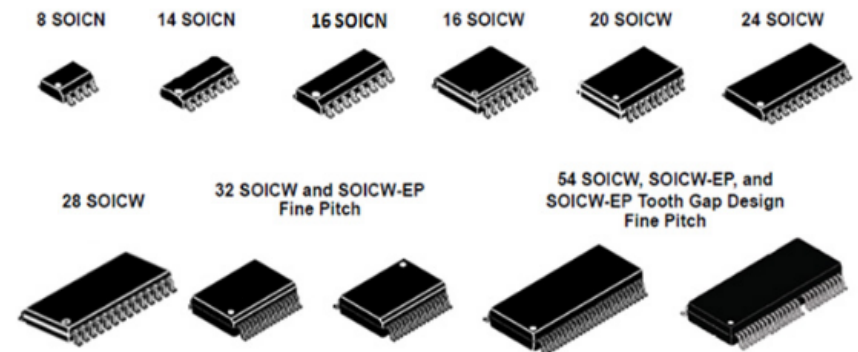
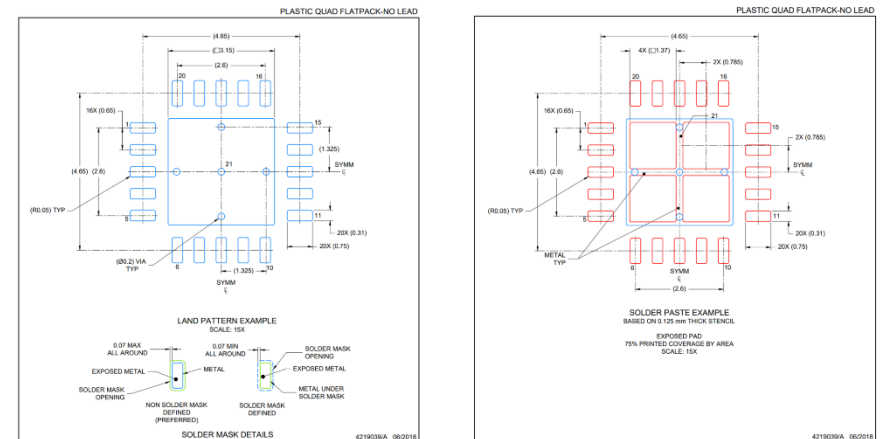
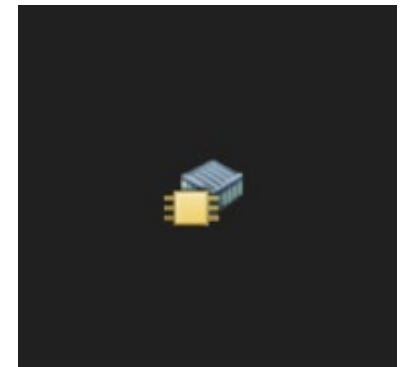
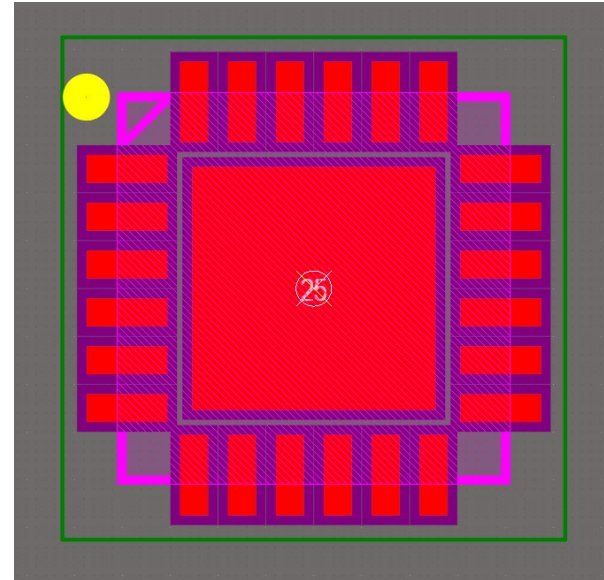


Figure 1. Standard Freescale SOIC Offerings

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

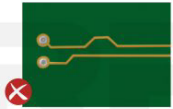
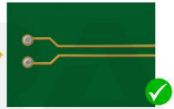
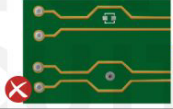
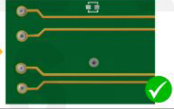
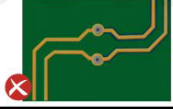
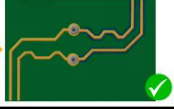
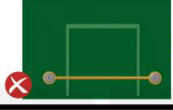
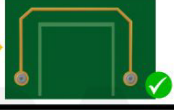


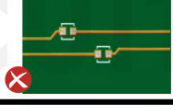
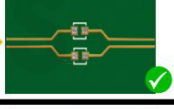
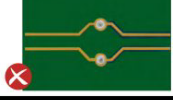
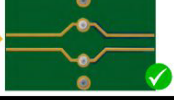
Footprint Libraries

- Very similar to component libraries, although only containing footprints
- Can be utilized across multiple projects
 - Managing a library of very common footprints can save a lot of time
 - SamacSys is also a good tool to help generate footprints quickly
- Altium Footprint Libraries are “.PcbLib”



Layout Tips

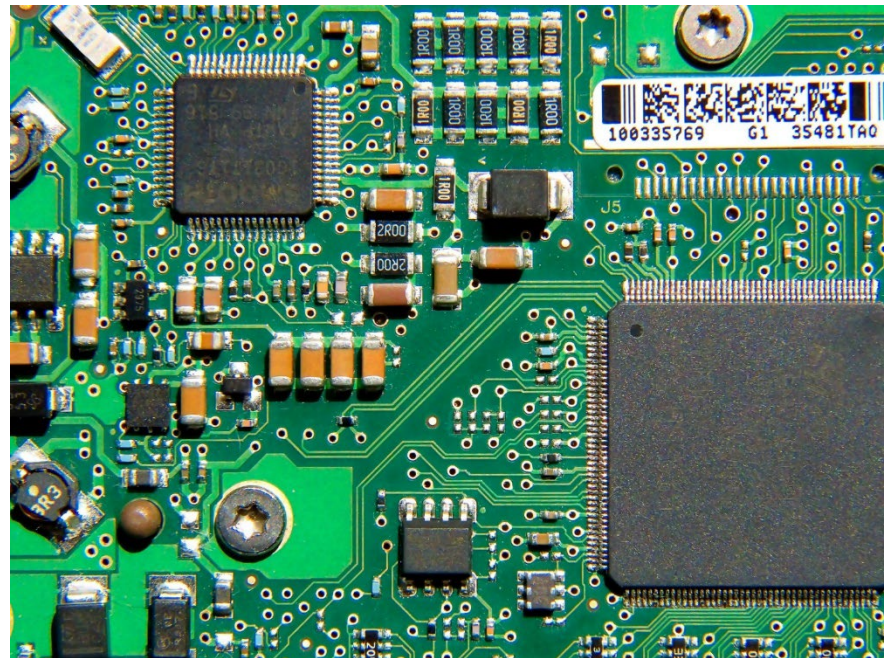
- This is the physical representation of your circuit, placement matters!
- Avoid “weaving” signals with vias
- Avoid running digital traces next to analog signals
- Ensure that traces will be able to handle the current it will carry

SIERRA CIRCUITS		7 PCB Routing Tips to Achieve Controlled Impedance	
S.No	Tips	Layout Dos and Don'ts	
01.	» Route differential pairs symmetrically and keep the signals parallel		
02.	» Avoid placing components and vias between differential lines		
03.	» Add serpentine traces to ensure length matching		
04.	» Do not route high-speed signals over a split plane		
05.	» Incorporate 45° trace bends instead of right-angled bends		
06.	» Maintain symmetry while placing coupling capacitors		
07.	» Place transition vias close to the signal vias		

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

Layout Tips

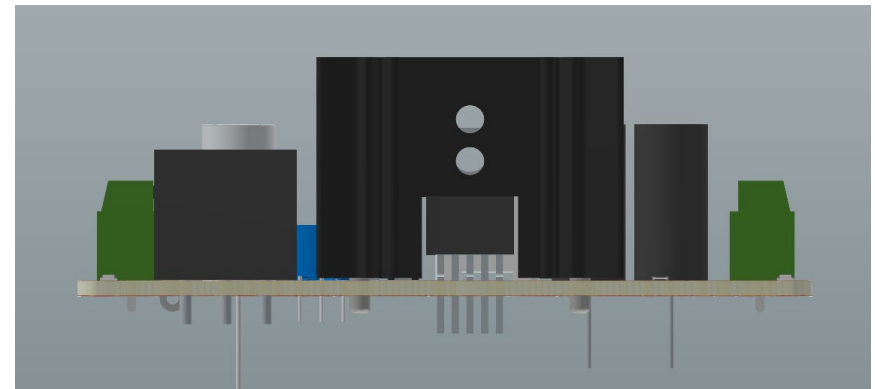
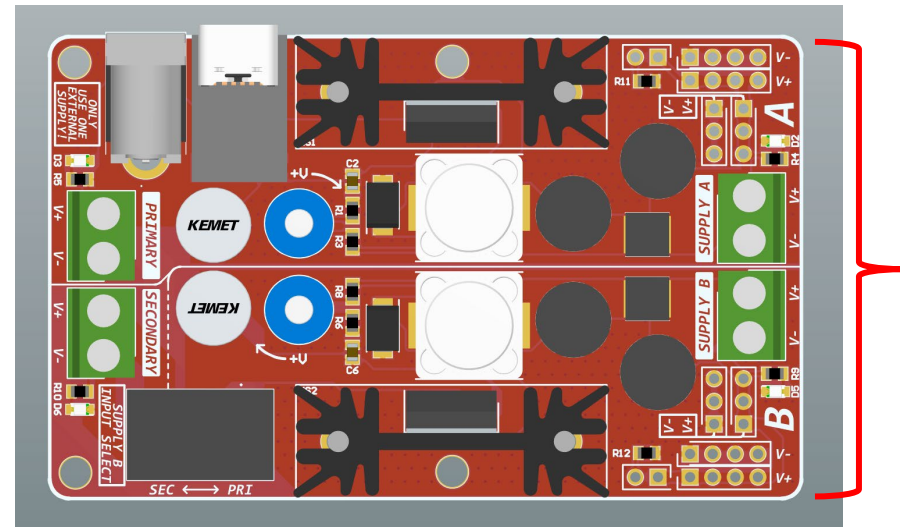
- Take time to place your components “nicely”
 - Align components
 - Establish a logical flow
- Label components and leave helpful notes for users
 - Ex: Voltage Range, Polarity
- Avoid 90 degree turns in traces
- Place decoupling capacitor close to target devices



<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?

Questions?

Download Today's Project Files

Navigate to the workshop GitHub and
download today's files

<https://github.com/IEEE-U-of-U/IEEE-PCB-Workshop-Fall-2025>