

LECTURE VIII

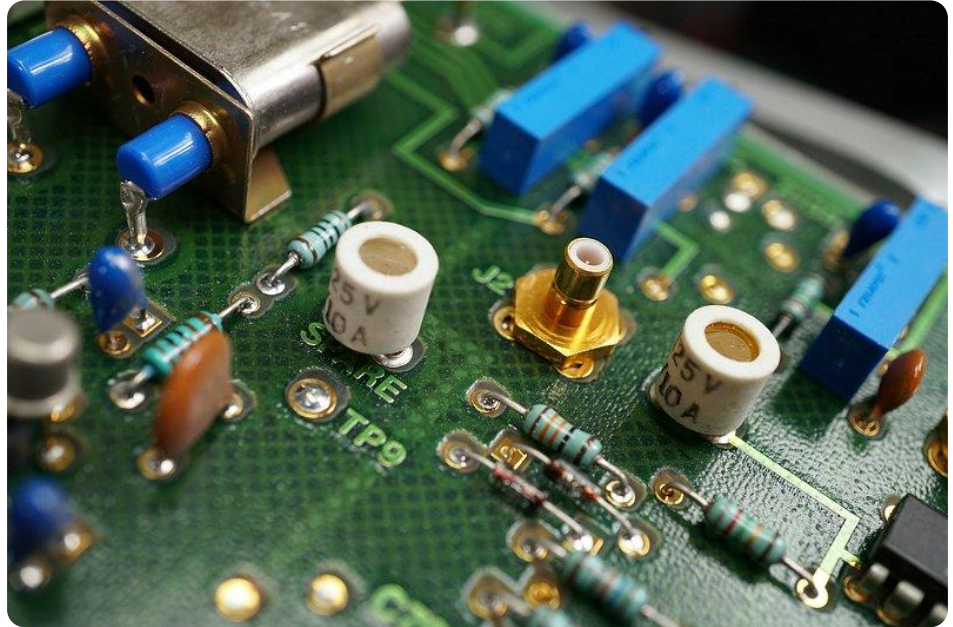
PCB Design Concepts

SECTION I

Introduction to PCBs

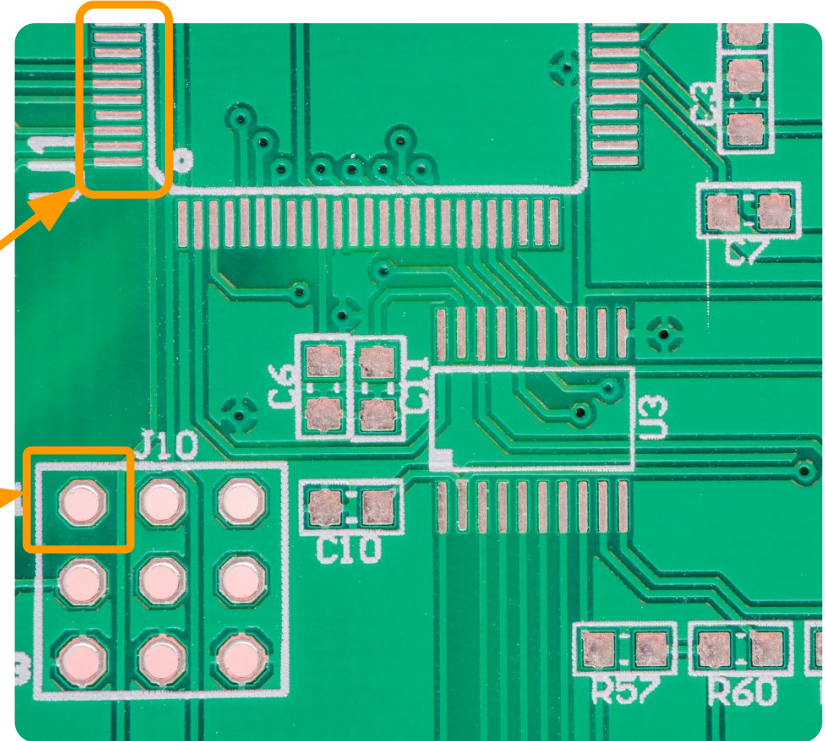
Printed Circuit Boards (PCBs)

- A **printed circuit board (PCB)** is an electronic assembly that uses copper conductors to **connect components**
 - It acts as a permanent **structural support** to electronic components
 - Made from **multiple, alternating layers of conductive** (usually copper) and **insulating material**



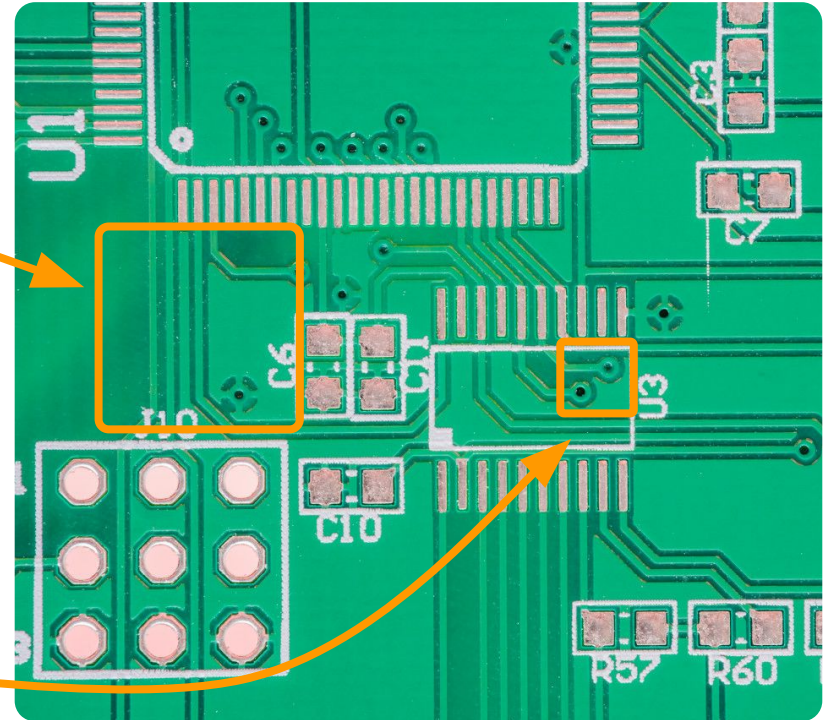
How are Components Connected?

- **Pads** are exposed copper surfaces which connect the leads of the electronic components to the board
 - Some pads are designed for **surface-mounted (SMD)** leads while others are for **through-hole (THT)** leads
 - Component **leads are soldered to the pads**



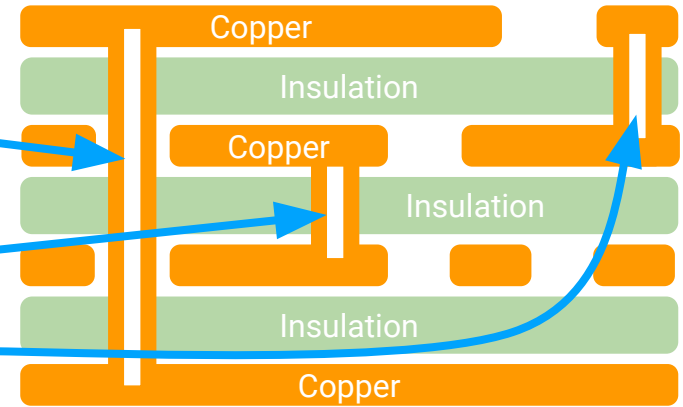
How are Components Connected? (Cont'd)

- **Traces** are **copper tracks** which connect pads
 - They are covered by a layer called the **solder mask** (colored green in the image)
- **Vias** are **conductive holes** drilled into the board to **connect different copper layers**



More on Vias

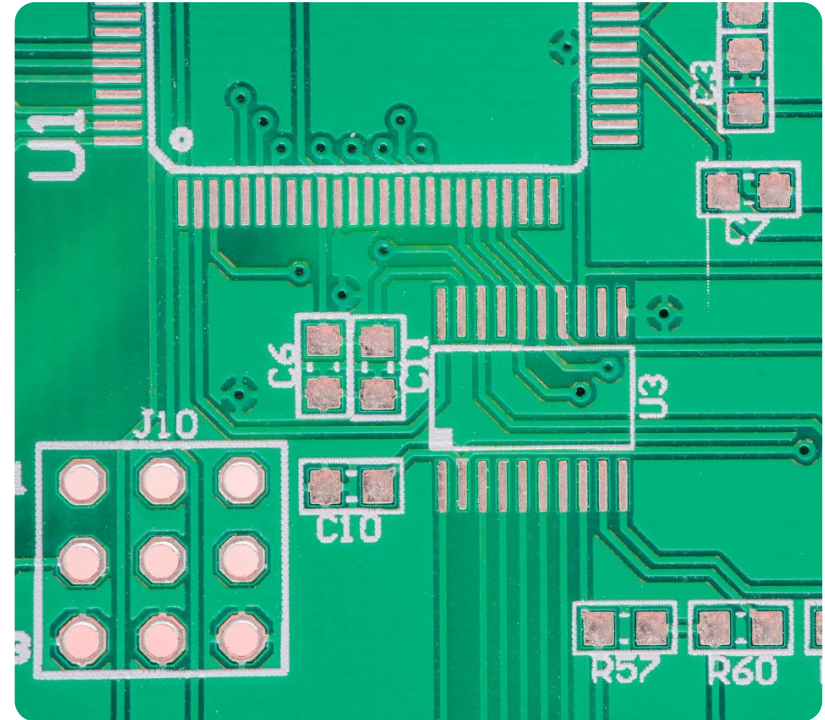
- Common via types...
 - **Through hole via** - drilled and plated from **top layer to bottom layer**
 - **Blind via** - drilled and plated **between two internal layers**
 - **Buried via** - drilled and plated from an **outside layer to an internal layer**
- A **void** surrounds the **via** to prevent a **connection** on copper layers which it should not connect



Example PCB Cross Section

How are Components Connected? (Cont'd)

- A **plane** is an **inner conductive layer** often used to create a ground point that is shared by traces and vias
 - Used as the return path for current leaving components
- **Fills** are **large areas of copper** used for the same purpose as planes but can be integrated into the **same layer as traces**



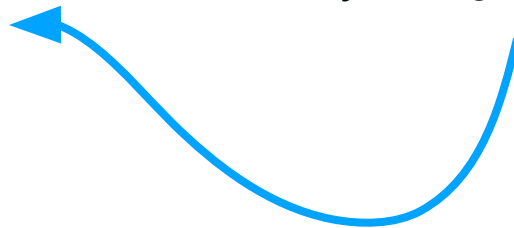
SECTION II

PCB Layers

PCB Stack-Up

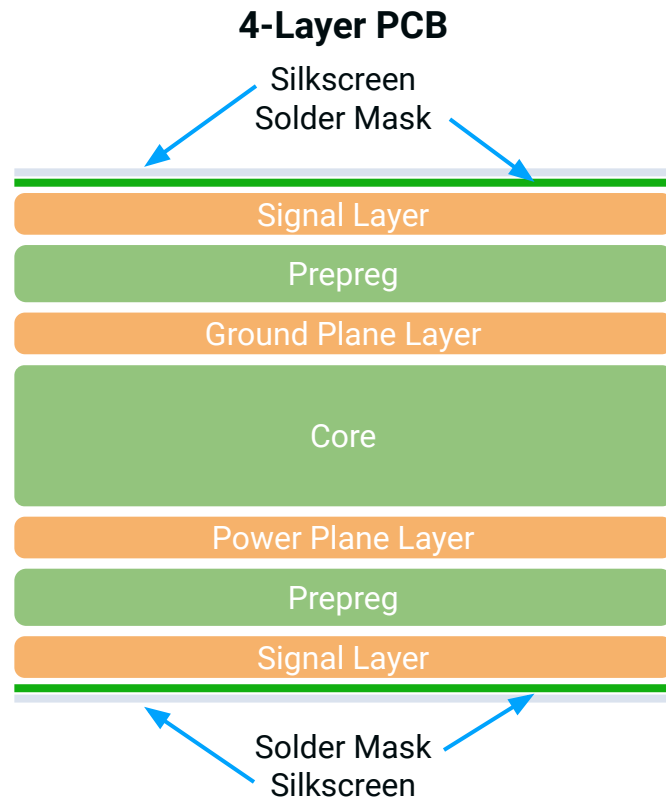
- The **PCB Stack-Up** is the **arrangement** of PCB layers
- Most PCBs have multiple of the following layers:
 - **Copper layer**
 - Signal or routing layer
 - Ground/Power plane layer
 - **Insulation layer**
 - Core
 - Prepreg
 - **Solder mask layer**
 - **Silkscreen layer**

Let's talk about each layer in greater detail



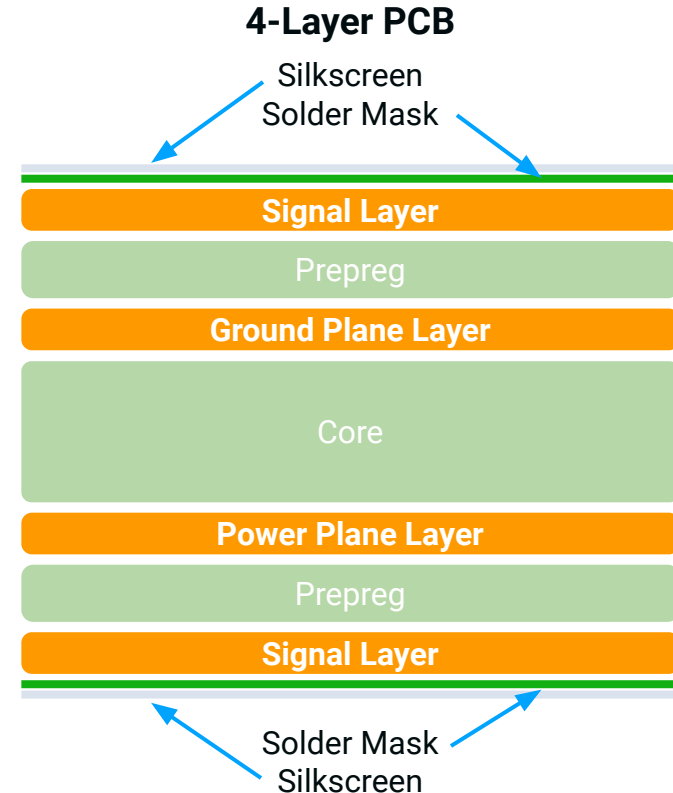
PCB Stack-Up (Cont'd)

- We will analyze an example stack-up for the **4-layer PCB**
 - An “x-layer PCB” contains x copper layers
- Note the **alternating pattern** of conductive and insulating layers



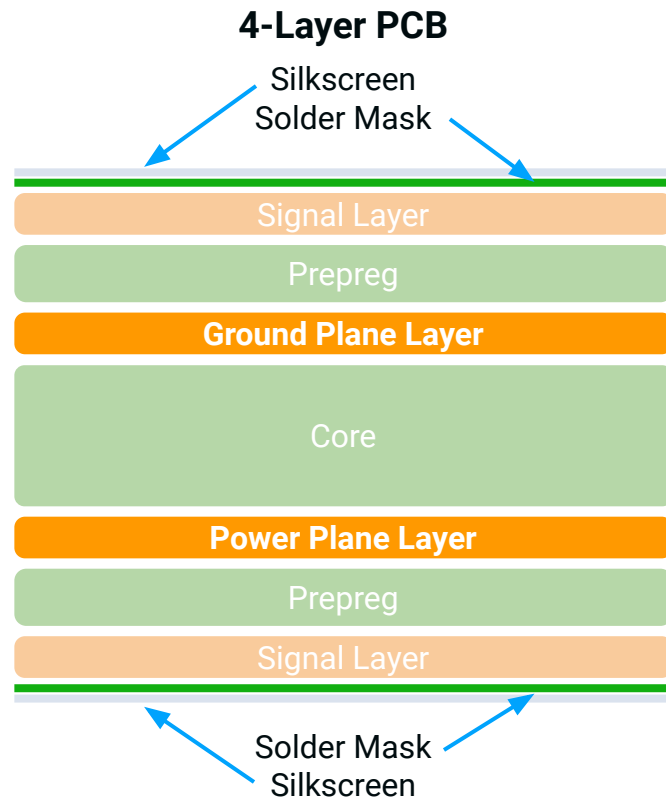
Copper Layer

- Each **copper layer** facilitates current flow between circuit components
- There are two types of copper layers:
 - A **signal/routing layer** is where the **traces and pads** are etched for connecting components
 - A **ground/power plane layer** serves as a **path to the common ground or power voltage**



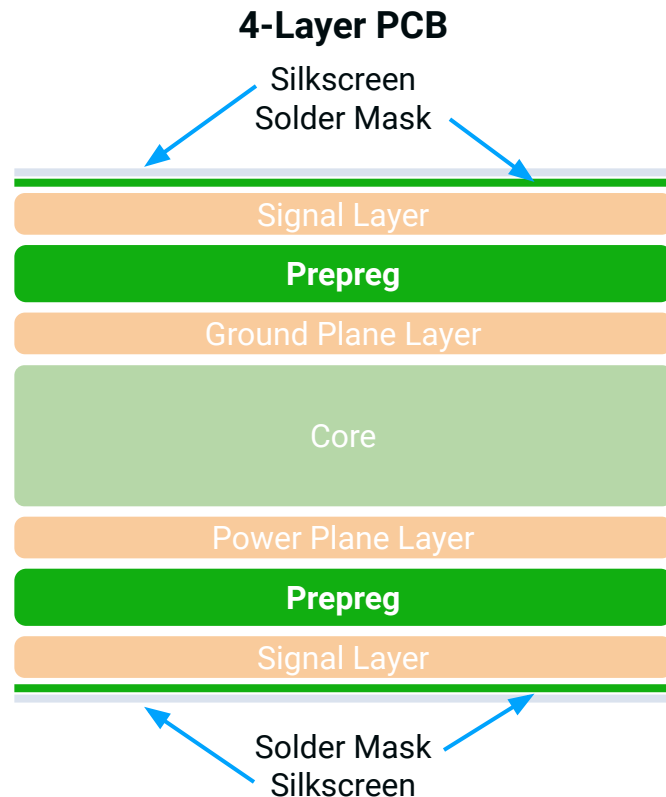
Copper Layer (Cont'd)

- **Why bother with a ground or power plane layer?**
 - Ground/power planes reduce electrical noise and improve the board's heat dissipation



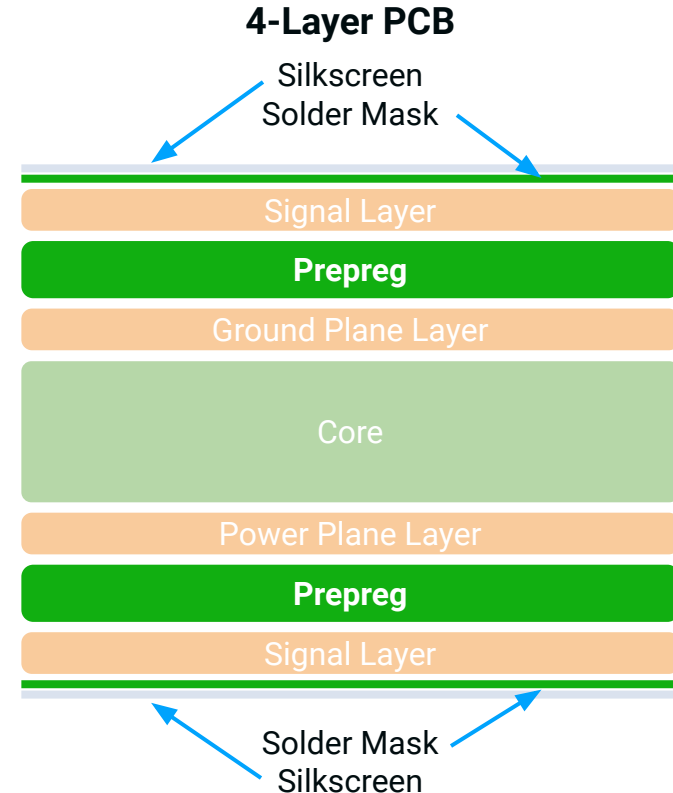
Prepreg Layer

- **Prepreg** is an **insulator layer** that acts as the “glue” to hold the core and copper layers together
 - It is **uncured** and contains resin which is responsible for its “sticky” property
 - When cured with heat and pressure, prepreg bonds all the layers together



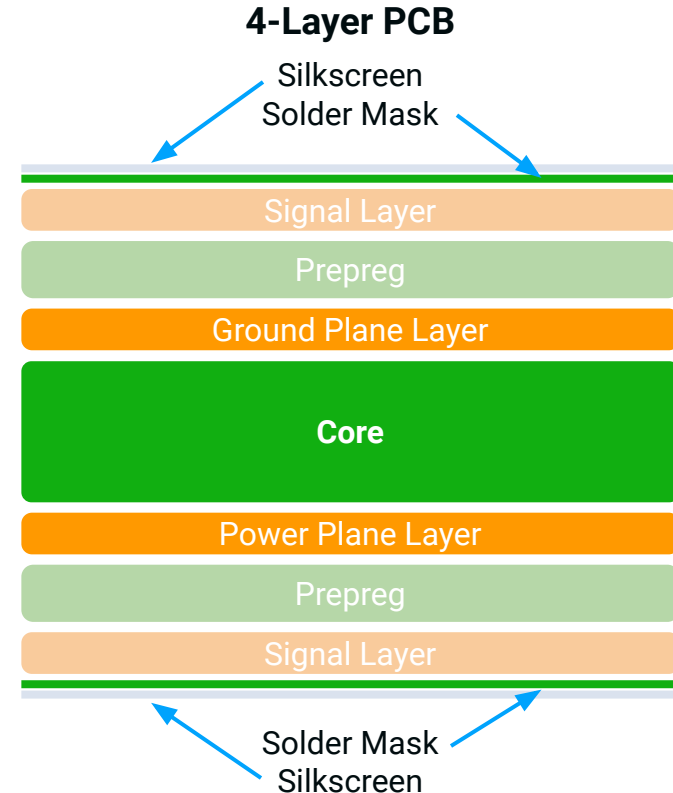
Prepreg Layer (Cont'd)

- Prepreg is made of **substrate** (insulating material)
 - Common substrates are...
 - **Fiberglass** (often FR-4)
 - Most often used
 - Cheap
 - **PTFE**
 - Better thermal stability and electrical properties than FR-4 but much more expensive



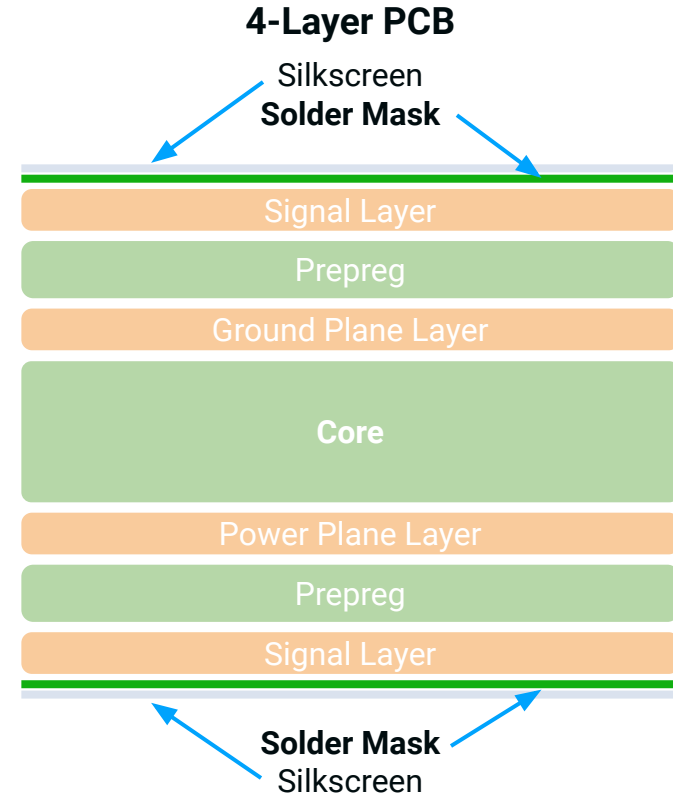
Core Layer

- **Core** is an **insulator layer** made of multiple prepreg layers pressed together, **plated with copper foil on either side**, and **cured**
 - It technically incorporates the two adjacent copper layers
- In a 4-layer PCB, only one core exists, separated by prepreg on either side
- In an 8-layer PCB, three cores are “sandwiched” by four prepreg layers



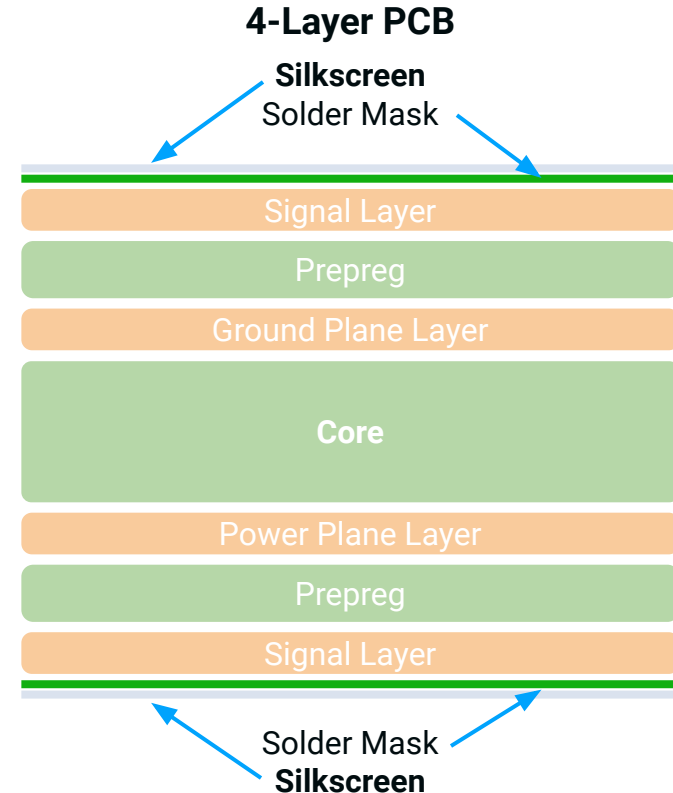
Solder Mask Layer

- Both outer copper layers are covered by a **solder mask layer**
 - It is a thin material used to **protect outer traces from oxidation** and **prevent solder bridges** between pads
 - It is the green material you see on the surface of the PCB (sometimes it is a different color)



Silkscreen Layer

- The **silkscreen** is a layer of ink traces (often white) used for symbols, logos, and other component markings
 - It is the outermost layer on either side of the board



SECTION III

PCB Fabrication

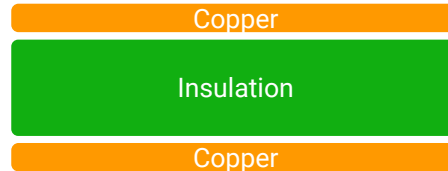
Fabrication Process Overview

2-Layer PCB

1. **Raw materials are cut** into boards and **holes are drilled**



2. Core, prepreg, and copper **layers are set and cured** (lamination)



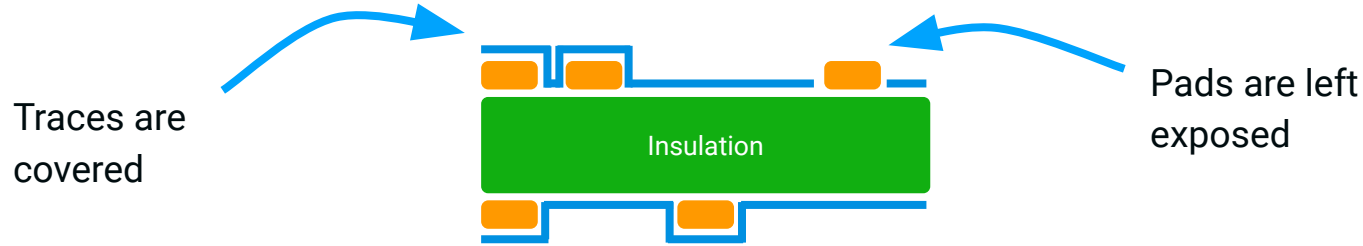
3. Copper layers are **etched** to remove excess copper, **leaving only the traces**



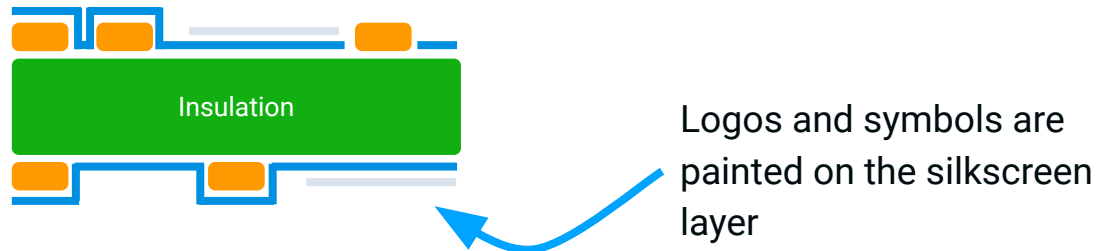
Fabrication Process Overview (Cont'd)

2-Layer PCB

4. The **solder mask is applied** to outer copper layers



5. The **silkscreen is painted** onto the solder mask



SECTION IV

PCB Design Process

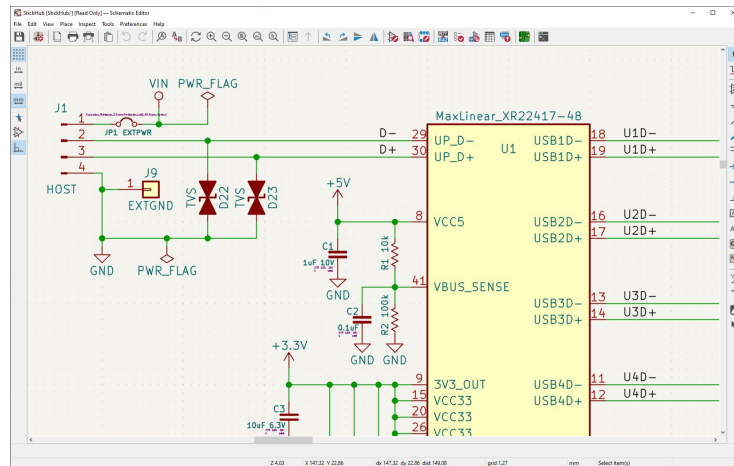
PCB Design Process

1. Identify the components and circuit diagrams you will use

- Prototype the design

2. Schematic capture

- Use CAD software to create digital schematics from the circuit diagrams
 - This software falls into a category of tools called **Electronic Design Automation (EDA)**

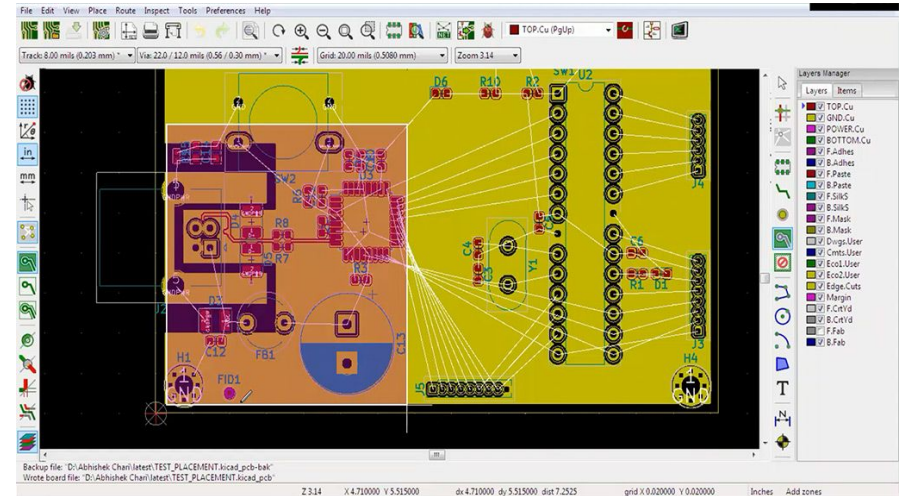


KiCad Schematic Capture

PCB Design Process (Cont'd)

3. Component placement and routing

- Arrange the components on the PCB layout; when placing components, consider (in order of priority)...
- The board shape
- Connector locations
- Heat dissipation requirements
- Decoupling capacitor locations

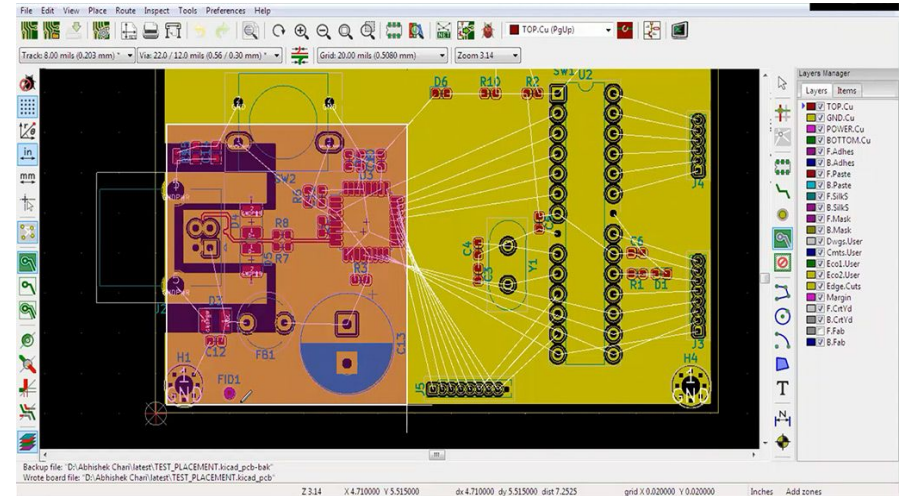


KiCad Routing

PCB Design Process (Cont'd)

3. Component placement and routing (Cont'd)

- Route the traces between component pads
- EDA tools will make sure the layout conforms to the schematic you defined



KiCad Routing

PCB Design Process (Cont'd)

4. **Verification**

- EDA software will have design rules checking tools to make sure components are properly connected and traces correctly routed

5. **Generate the manufacturing files**

- EDA software will create the files which you share with a manufacturer
- These files are used by machines for automated PCB fabrication

6. **Fabricate the PCB**

PCB EDA Software

- Popular PCB design software are...

- [Altium Designer](#)
- [Autodesk EAGLE](#)
- [KiCAD EDA](#) (Free)

Altium
Designer

E EAGLE

- We will use KiCAD for this course

- It is an open source schematic capture and PCB design tool

KiCad

PCB Manufacturers

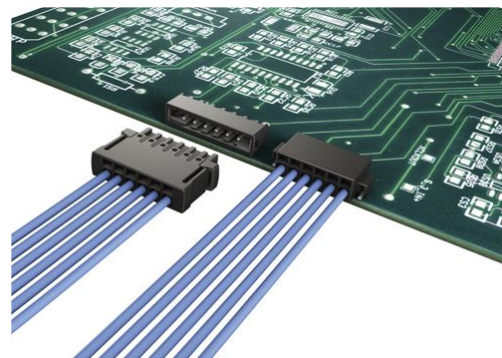
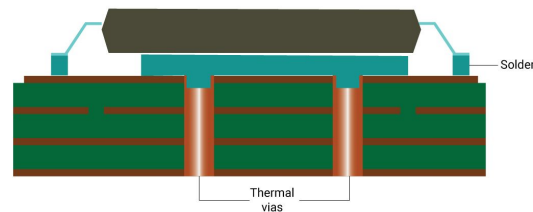
- [JLCPCB](#)
 - Inexpensive 2-layer FR-4 boards
 - Fast manufacturing and shipping
 - Used by hobbyists
- [PCBWay](#)
 - Better for more precise design requirements (small traces, vias, etc.)
 - Advanced manufacturing options

SECTION V

PCB Layout Tips

PCB Layout Tips

- **Use thermal vias to help cool components**
 - **Thermal vias** (unconnected vias) move heat away from components through the board layers
- **Place board-to-wire connectors near the edge of the PCB**
 - It's harder to connect wires in the middle of the PCB

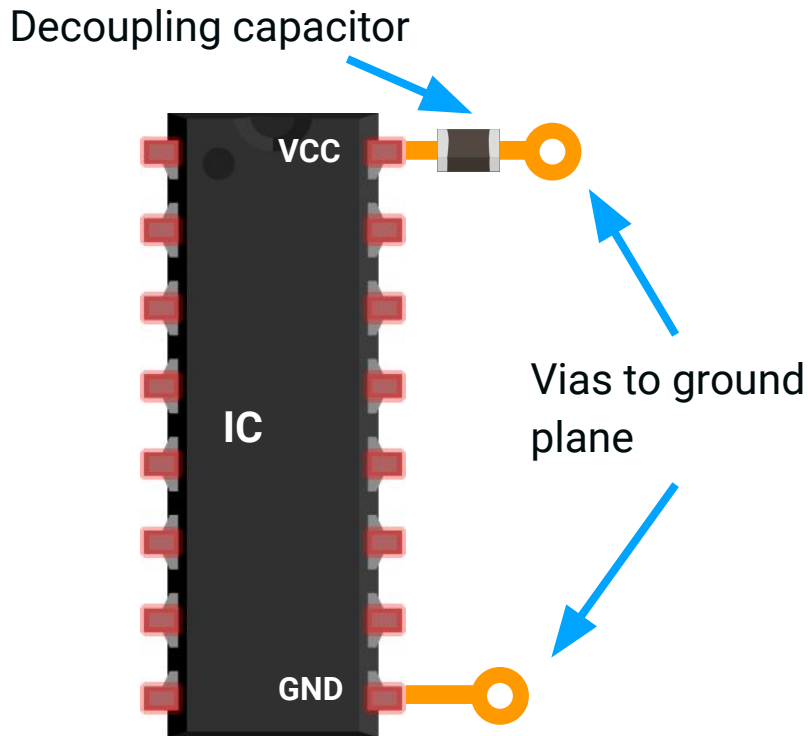


PCB Layout Tips (Cont'd)

- **Use ground fills or planes** to reduce electrical noise and improve signal integrity
 - **Electrical noise** - random variations in voltage and current, which affects sensitive components
 - **Signal integrity** - how well a signal maintains its original characteristics (strength, shape, timing) from sender to receiver
- **Leave space between** pads and traces
 - Make sure that the pad and trace spacing **adheres to design constraints** (devices packaging, manufacturer requirements)

PCB Layout Tips

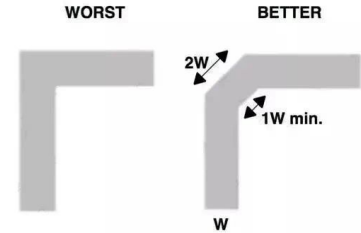
- **Use decoupling capacitors on ICs**
 - **Decoupling capacitors** help **reduce electrical noise** in power supply signals
 - **Ensures a clean and stable power supply** for the IC, preventing unexpected behavior and malfunctions
 - Place the capacitor **between the VCC and GND pins** of the IC



PCB Layout Tips (Cont'd)

- **Avoid 90° trace angles**

- Use 45° angles instead
- The corners of 90° angles are narrower than the standard trace width; traces should be consistent widths



- **Widen the power and ground traces**

- Reduces heat buildup from high current
- Improves signal integrity
- Signal traces, which are low current, may be narrower

“Darling, **PCBs are like Hollywood backlots** – a whole lot of drama, some **unexpected connections**, and you better know where to put your power or you'll end up with a **whole mess of smoke and burned out stars.**”

Betty White (circa 1997)

Famous Misquotes



FAIR USE DISCLAIMER

Copyright Disclaimer under section 107 of the Copyright Act 1976, allowance is made for “fair use” for purposes such as criticism, comment, news reporting, teaching, scholarship, education and research.

Fair use is a use permitted by copyright statute that might otherwise be infringing.

Non-profit, educational or personal use tips the balance in favor of fair use.