LECTURE VIII

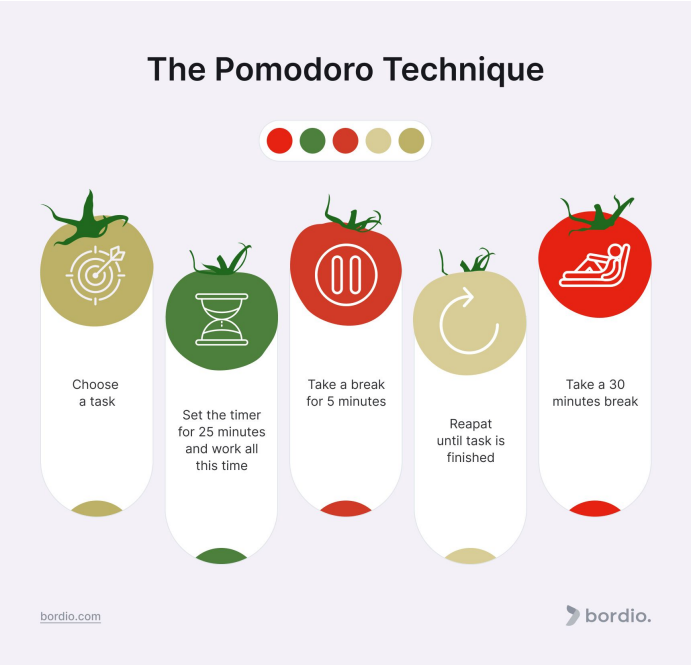
PCB Design Concepts

OPS Instructor Applications!

- OPS Instructor Applications are coming out Wednesday night! The link to apply will be in the discord.
- Being an OPS instructor means:
  - Being able to influence next year's program/curriculum
  - Helping out students new to EECS concepts
  - Being able to join IEEE as an official board member and get to know other cool people!
- Consider applying! Ask any instructor if you have any questions :)
- **OPS Students who have completed their projects have higher priority when applying!**

Challenge Projects!

- **Reminder: Project X is due March 30!**
  - This is a challenge project, so it is optional (but highly recommended!)
- **A second challenge project is on the way as well!**
  - Pomodoro Timer
  - Will also be due March 30
  - ALSO OPTIONAL!

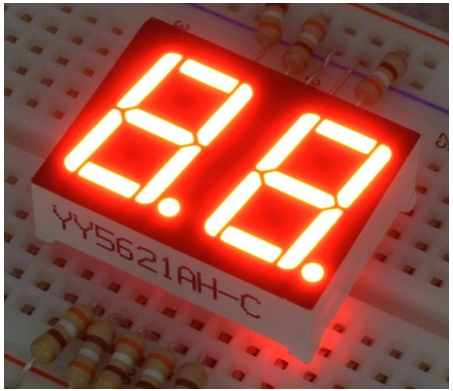


SECTION I

Project VII Review

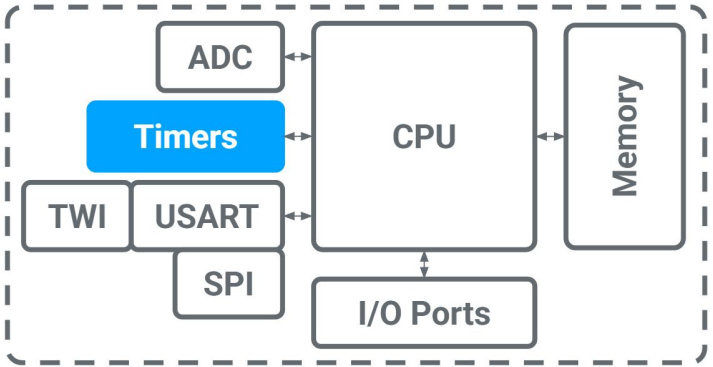
Project 7 Review

- Create a **digital stopwatch** using interrupts, timers, and a 7-segment display
- Due date: 2/23/2026 at 11:59PM



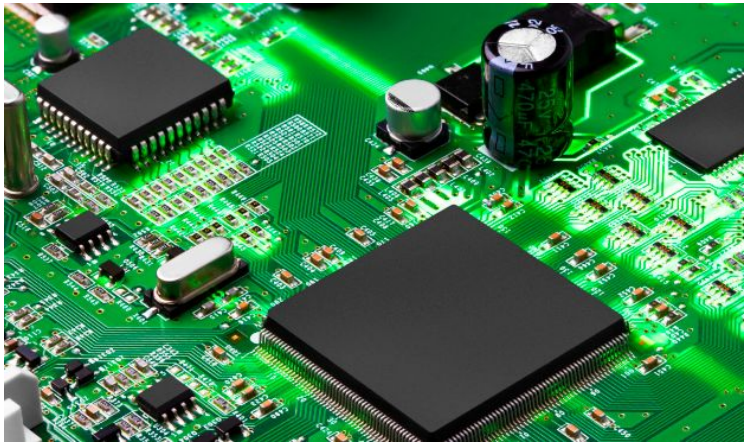
Learning Concepts:

- Interrupts
- Timers
- Arduino IDE Libraries (Continued)



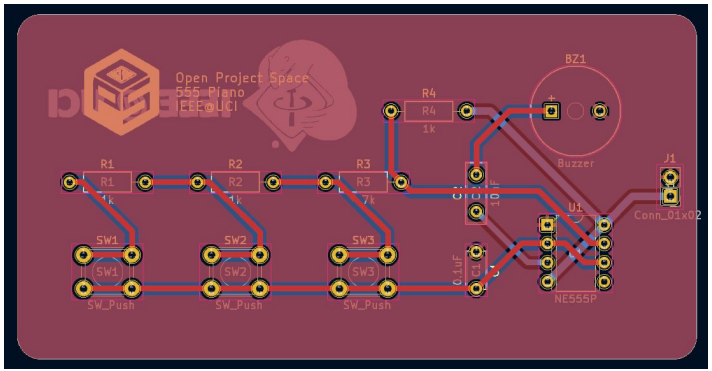
Project 8

- Design a **PCB** for the 555 Blinking LED or Piano.
- Due date: 3/13/2026 at 11:59PM
- This project is optional!



Learning Concepts:

- KiCAD
- Schematics
- PCB Design
- PCB Manufacturing

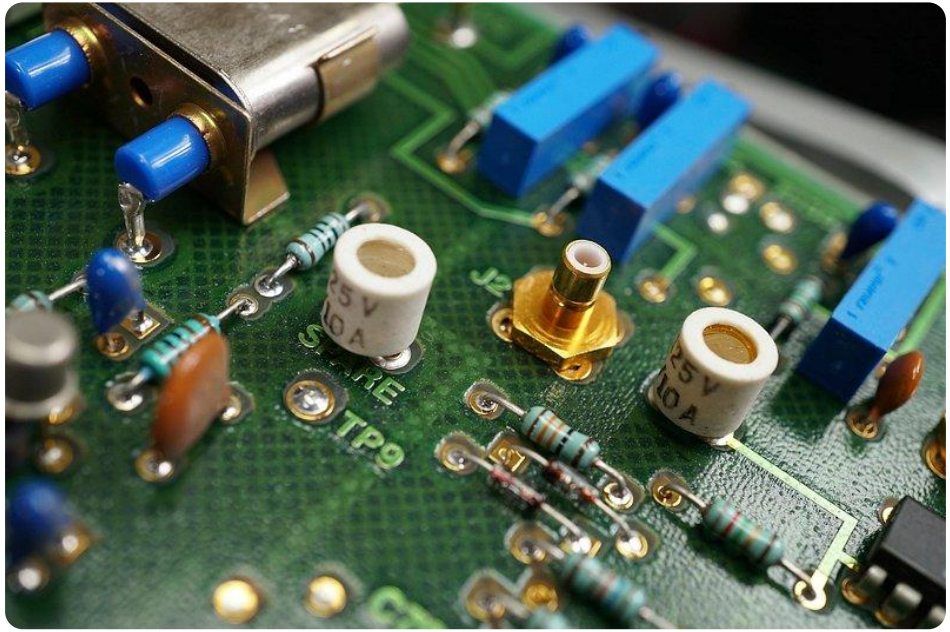


SECTION I

Introduction to PCBs

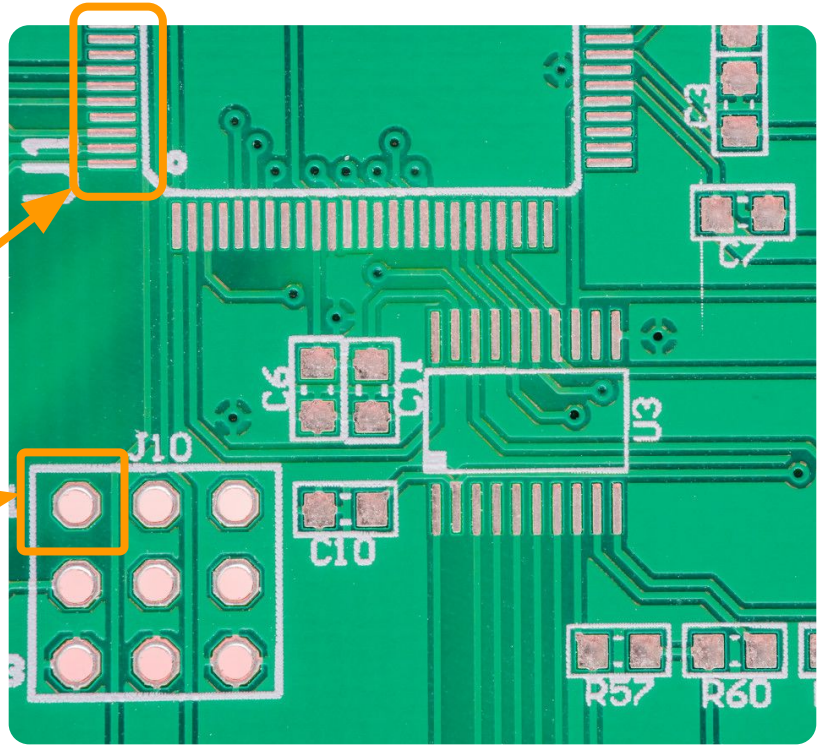
Printed Circuit Boards (PCBs)

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



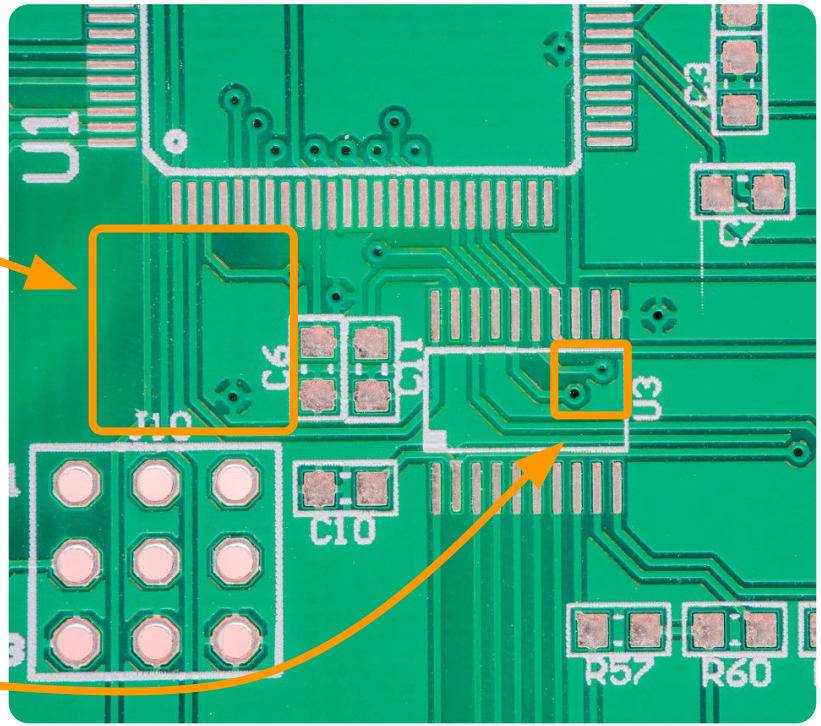
How are Components Connected?

- **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**



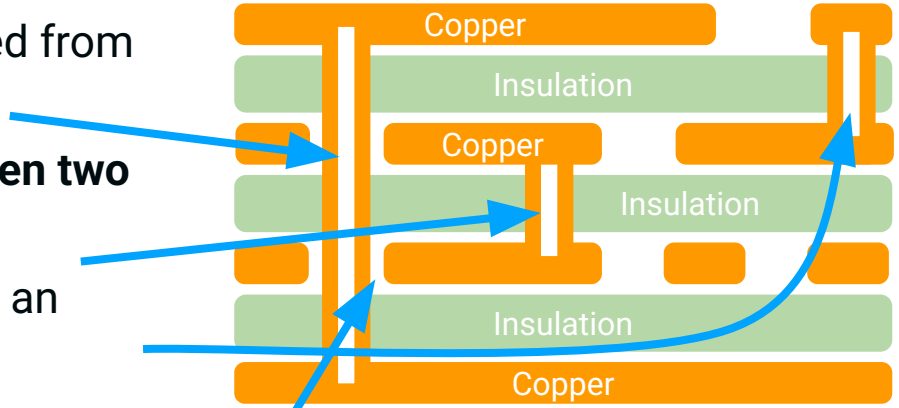
How are Components Connected? (Cont'd)

- **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

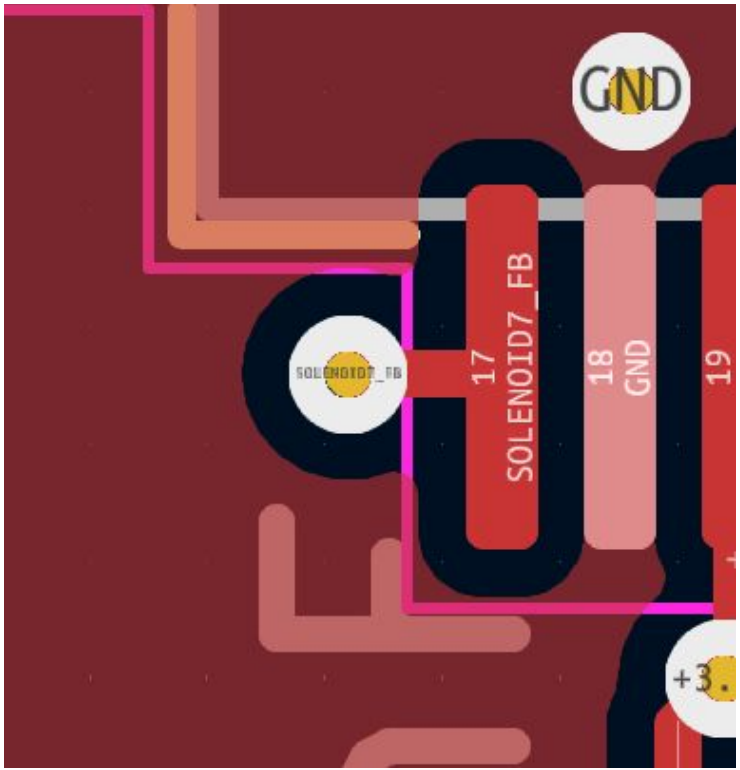
- Common via types…
  - **Through hole via** - drilled and plated from **top layer to bottom layer**
  - **Blind via** - drilled and plated **between two internal layers**
  - **Buried via** - drilled and plated from an **outside layer to an internal layer**
- A **void surrounds the via to prevent a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



Example PCB Cross Section

How are Components Connected? (Cont'd)

- A **plane** is an **inner conductive layer**
  - Used to create a ground point
- **Fills** are **large areas of copper** used for the same purpose as planes but can be integrated into the **same layer as traces**
  - The transparent shape surrounding the vias and the pads on the right is a Fill, that is used to ground the ground pad



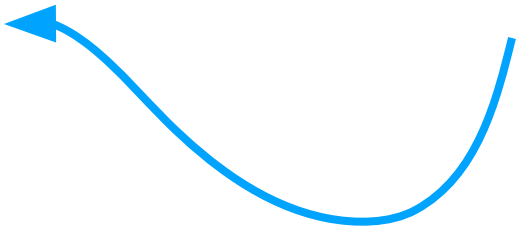
SECTION II

PCB Layers

PCB Stack-Up

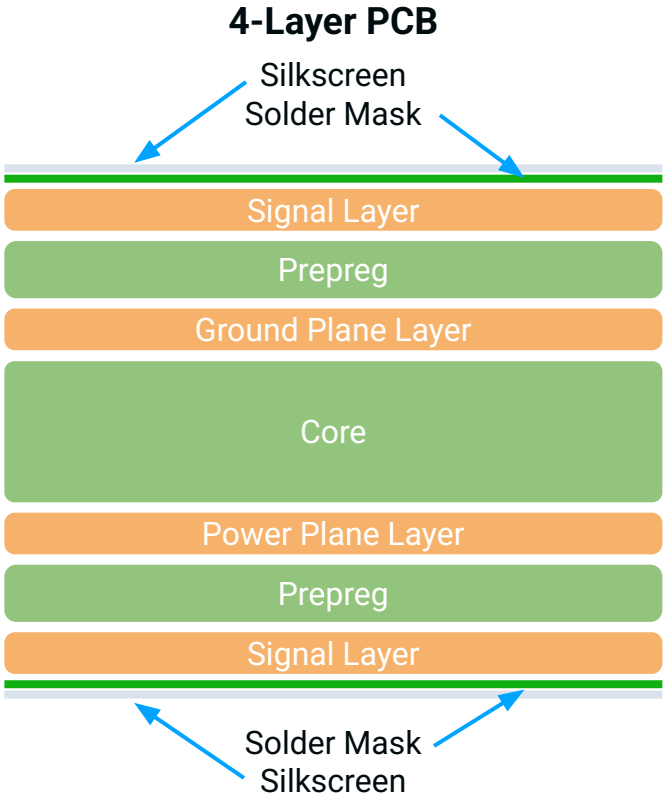
- The **PCB Stack-Up** is the **arrangement** of PCB layers
- Most PCBs have multiple of the following layers:
  - **Copper layer**
    - Signal or routing layer
    - Ground/Power plane layer
  - **Insulation layer**
    - Core
    - Prepreg
  - **Solder mask layer**
  - **Silkscreen layer**

Let's talk about each layer in greater detail



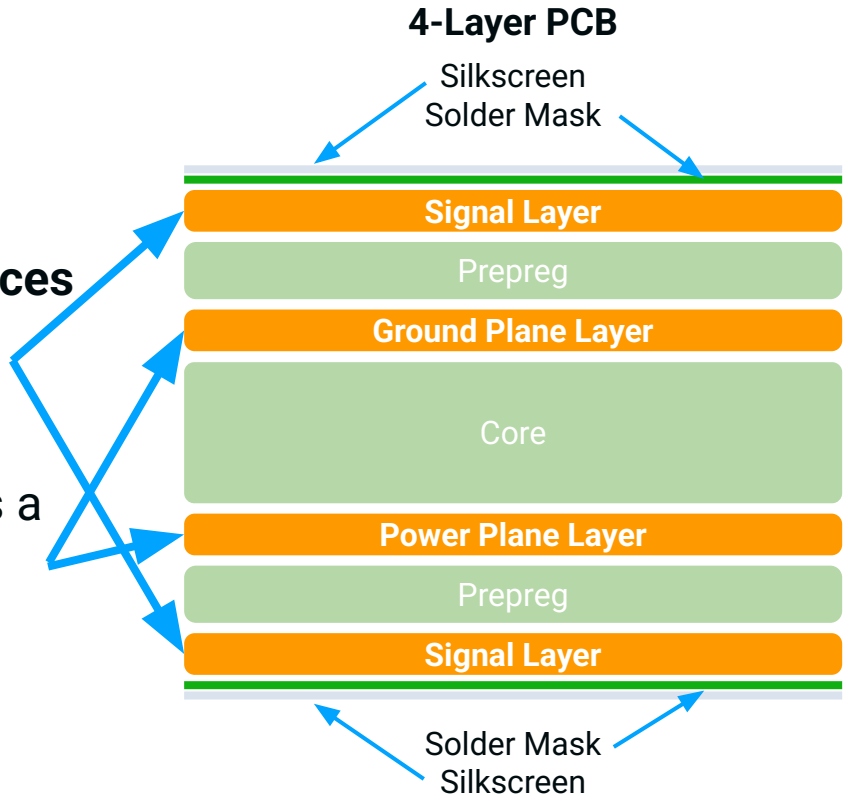
PCB Stack-Up (Cont'd)

- 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



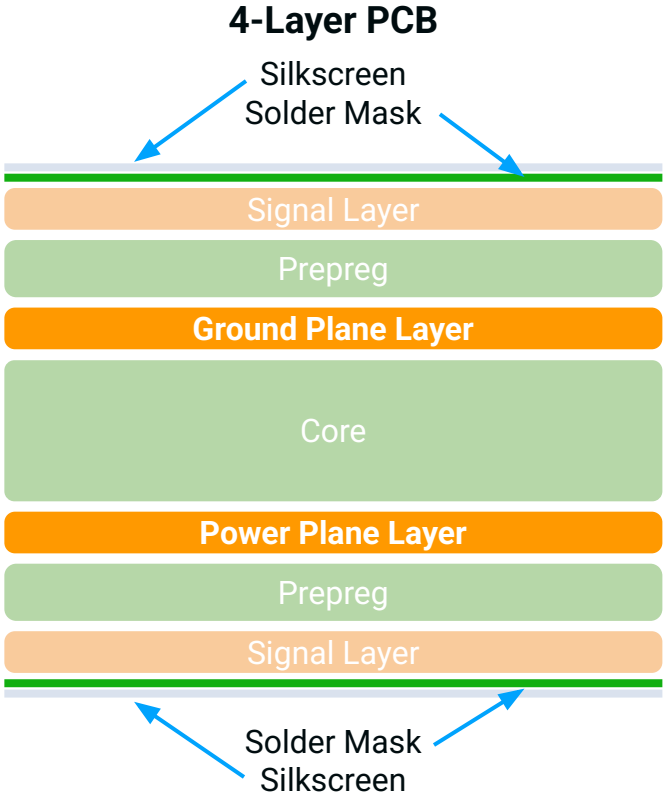
Copper Layer

- 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



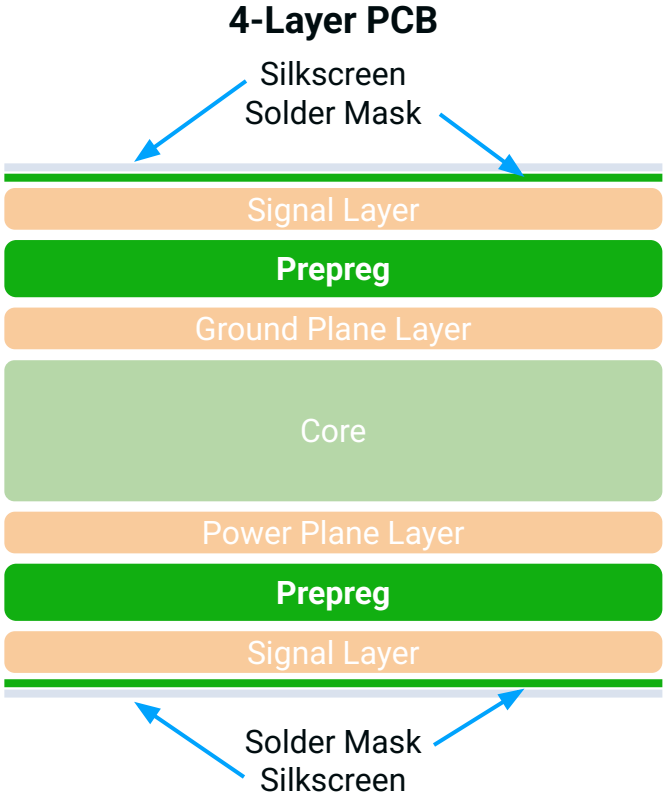
Copper Layer (Cont'd)

- **Why bother with a ground or power plane layer?**
  - Ground/power planes reduce electromagnetic interference (EMI) and improve the board's heat dissipation
  - A ground plane separating layers of high speed signals can decrease the amount of EMI leaking from one signal to the other



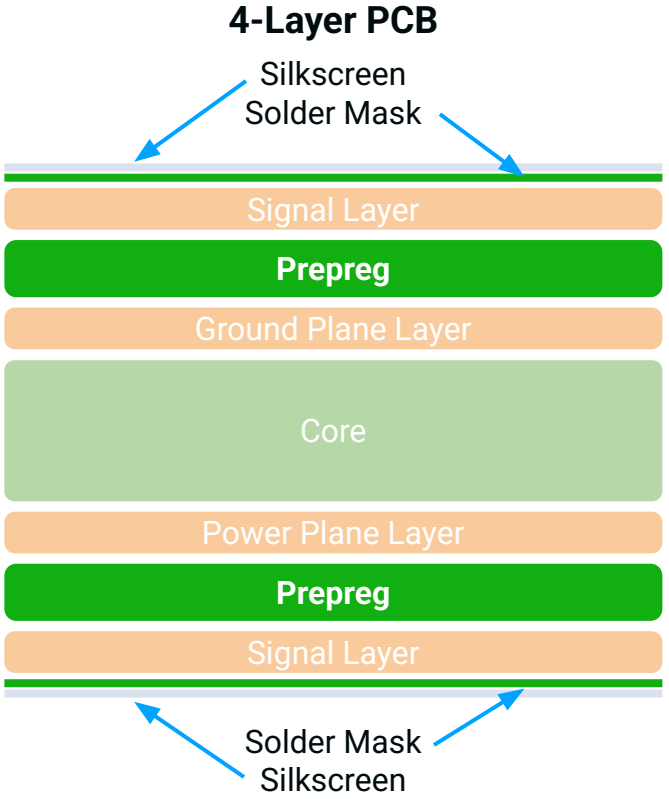
Prepreg Layer

- **Prepreg** is **an insulator layer** that separates the copper layers and 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



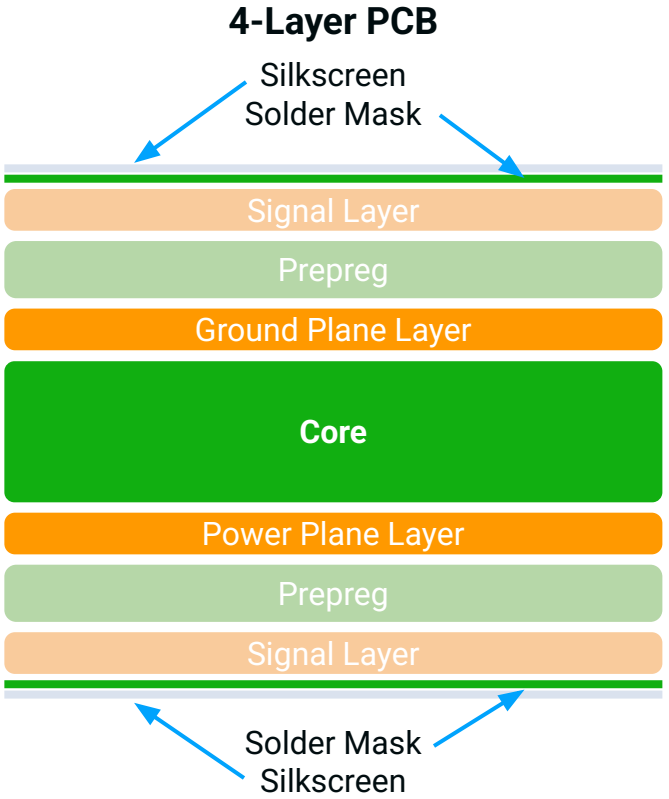
Prepreg Layer (Cont'd)

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



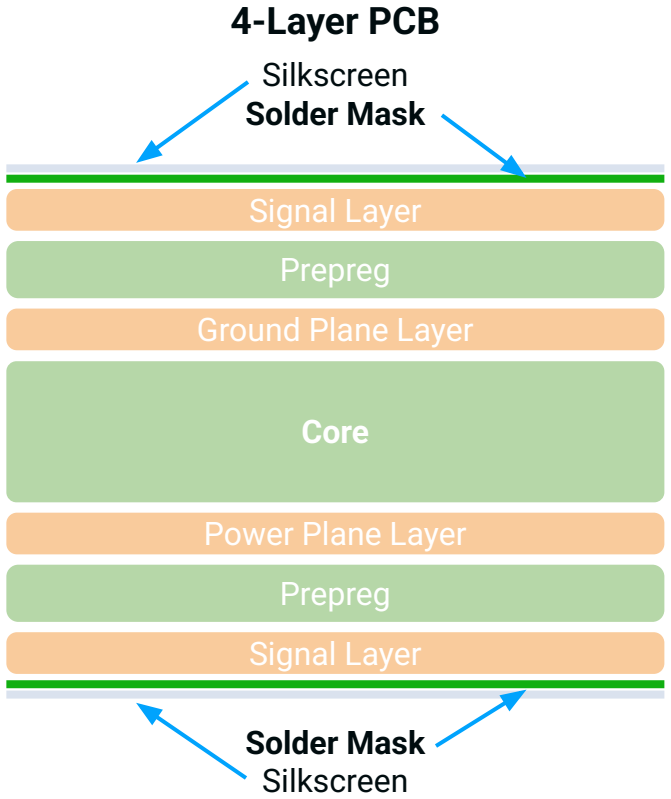
Core Layer

- **Core** is an **insulator layer** made of multiple prepreg layers pressed together, **plated with copper layers on either side**, and **cured**
  - It technically incorporates the two adjacent copper layers
- In a 2-layer PCB, **usually** only one core exists
- In a 4-layer PCB, **usually** only one core exists, separated by prepreg on either side



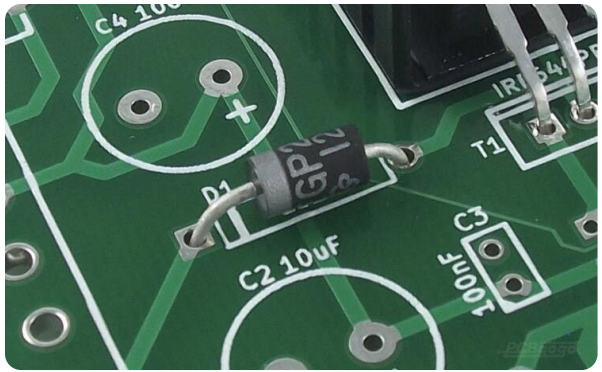
Solder Mask Layer

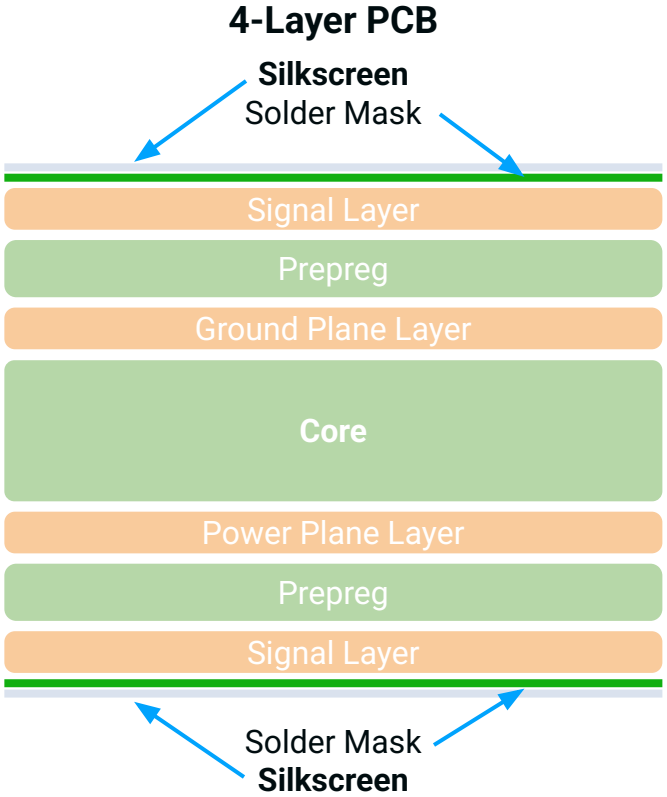
- Both outer copper layers are covered by a **solder mask layer**
  - It is a thin resin or filler that is used to **protect outer traces from oxidation** and **prevent solder bridges** between pads
  - It is the green material you see on the surface of the PCB (can be made into a different color :00)



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

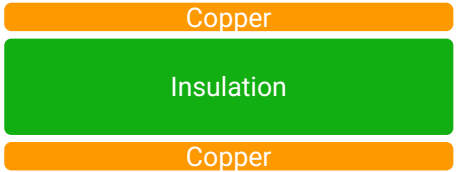
PCB Fabrication

Fabrication Process Overview

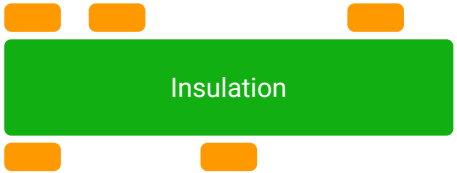
2-Layer PCB

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

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



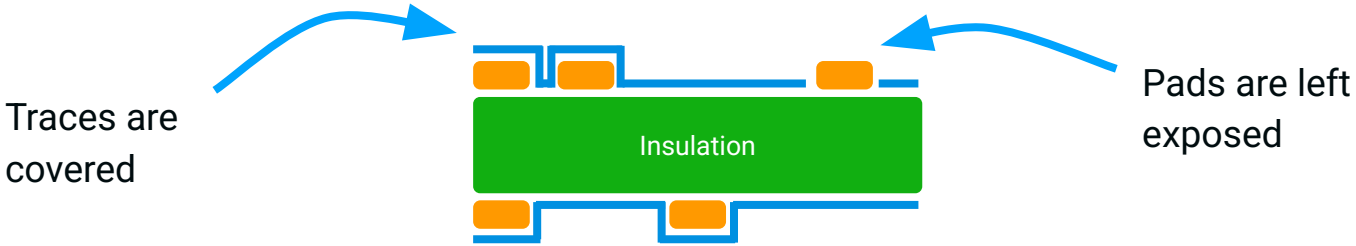
3. Copper layers are **etched** to remove excess copper, **leaving only the traces**



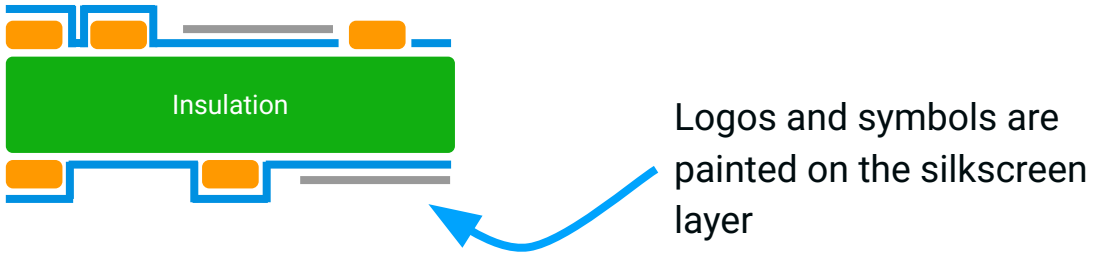
Fabrication Process Overview (Cont'd)

2-Layer PCB

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



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

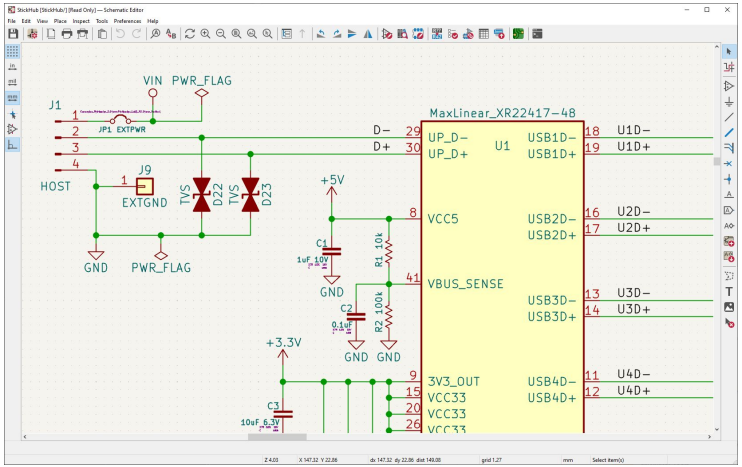


SECTION IV

PCB Design Process

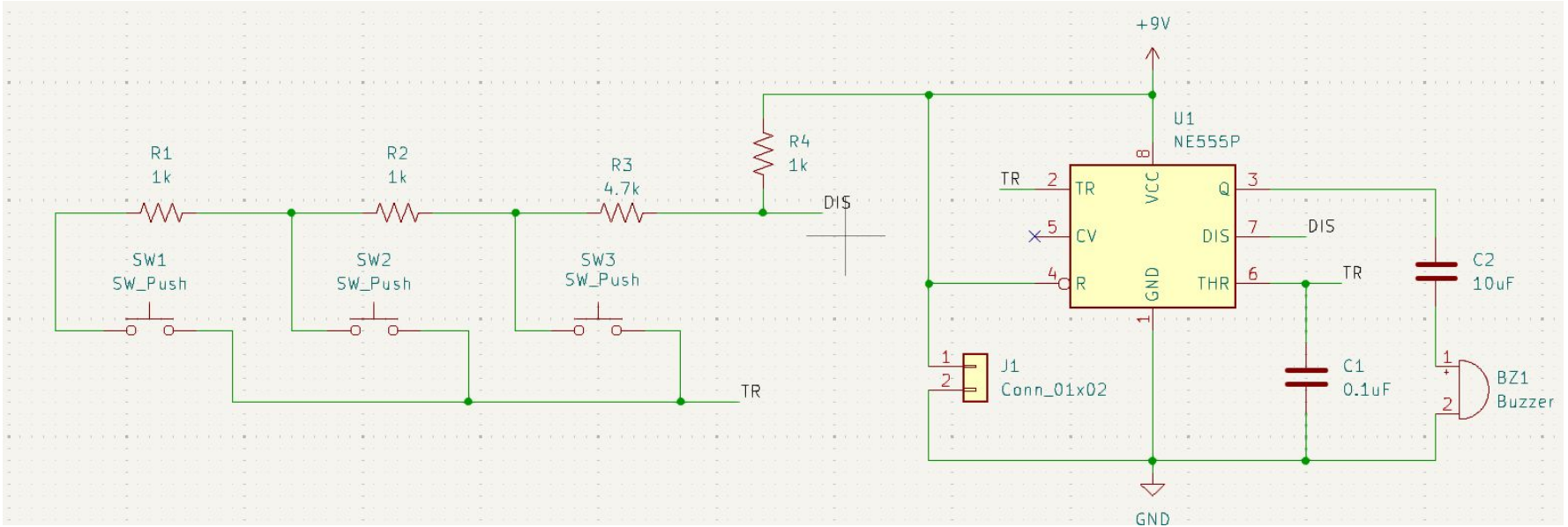
PCB Design Process

1. **Identify the components** and **circuit diagrams** you will use
    - Prototype the design on a breadboard
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

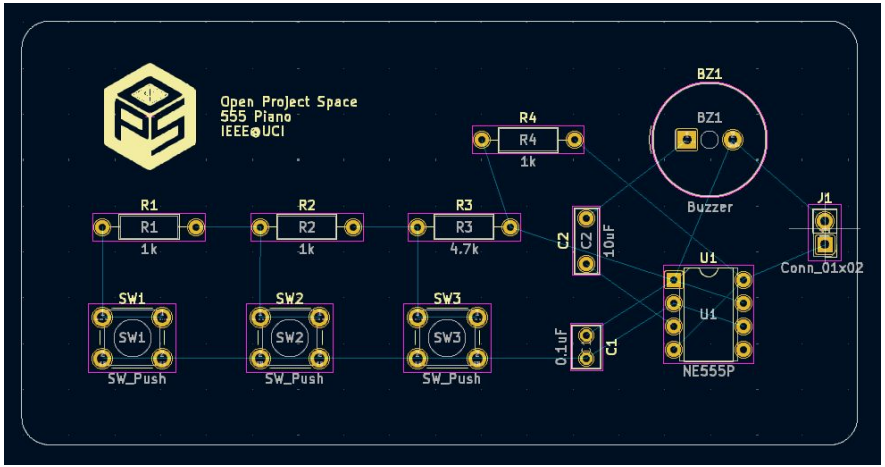
PCB schematic



This look familiar?

PCB Design Process (Cont'd)

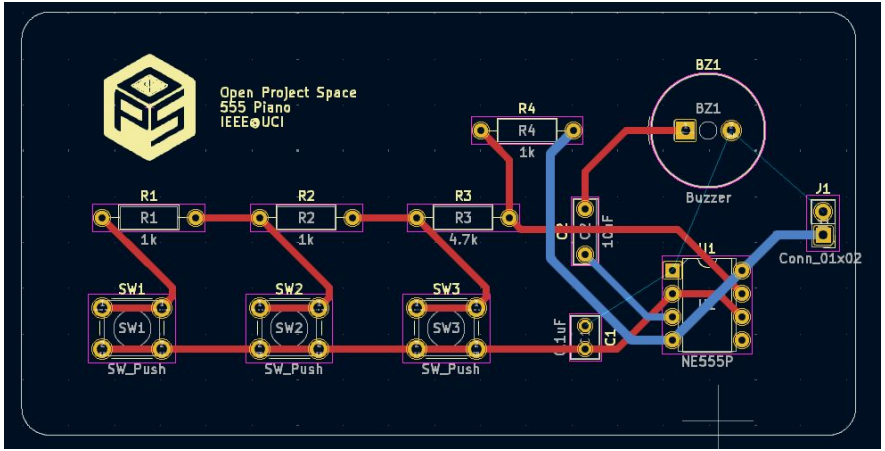
3. **Component placement** and **routing**
    - Arrange the components on the PCB layout; when placing components, consider (in order of priority)...
        - The board shape
        - Connector locations
        - Heat dissipation requirements



KiCad Component placing

PCB Design Process (Cont'd)

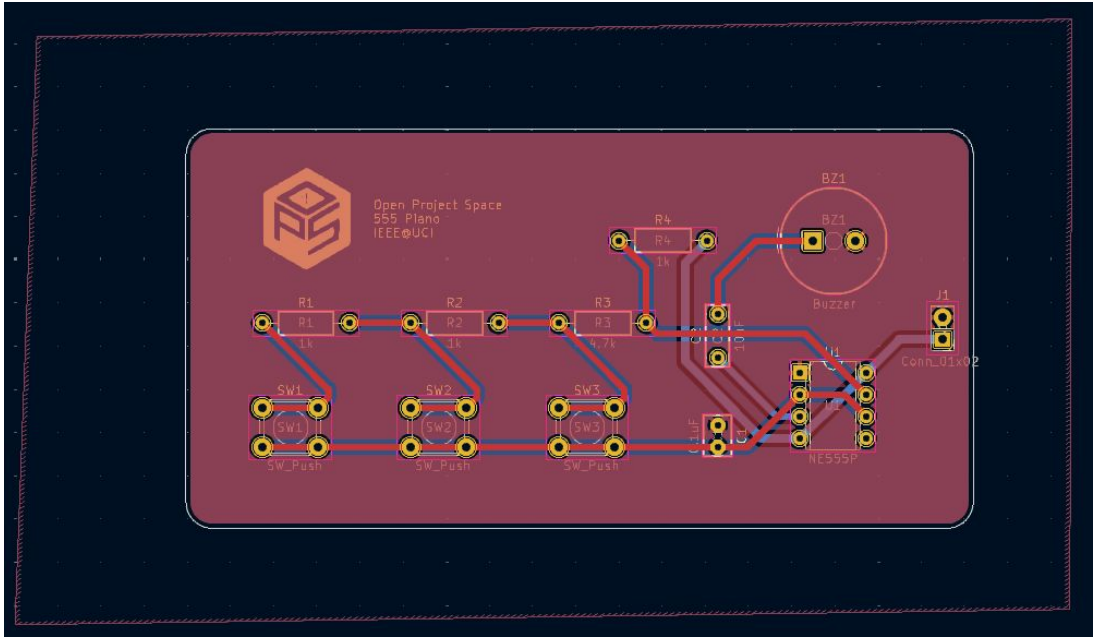
3. **Component placement** and **routing** (Cont'd)
   - Route the traces between component pads
   - EDA tools will make sure the layout conforms to the schematic you defined
   - **DESIGN RULES CHECK (DRC)**
     - Checks if there are any tolerance errors or unconnected components



KiCad Routing

PCB Design Process (Cont'd)

- Fill zones
  - Fills in the rest of the empty pcb with copper
  - Can connect component together
  - We normally used the top and bottom fills with ground



KiCad Fill zones

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 (free for UCI students!)
  - Autodesk EAGLE
  - KiCAD EDA (Free)
- We will use KiCAD for this course
  - It is an open source schematic capture and PCB design tool

PCB Manufacturers

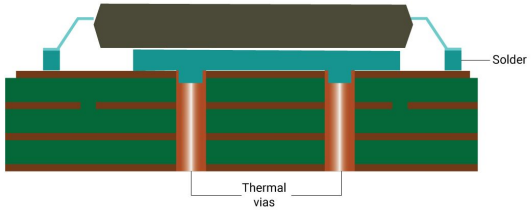
- JLCPCB
  - Inexpensive 2-layer FR-4 boards
  - Fast manufacturing and shipping
  - Used by hobbyists
- PCBWay
  - Better for more precise design requirements (small traces, vias, etc.)
  - Advanced manufacturing options

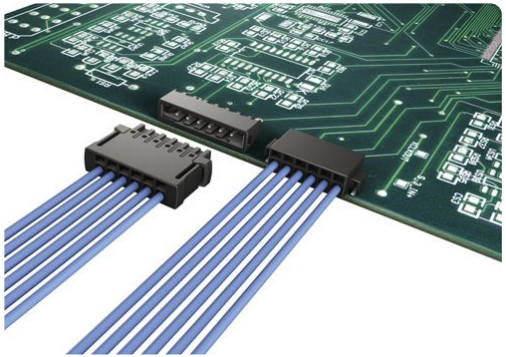
SECTION V

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)

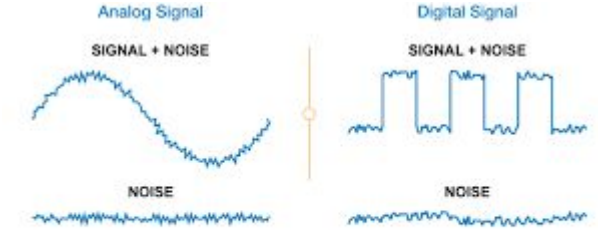
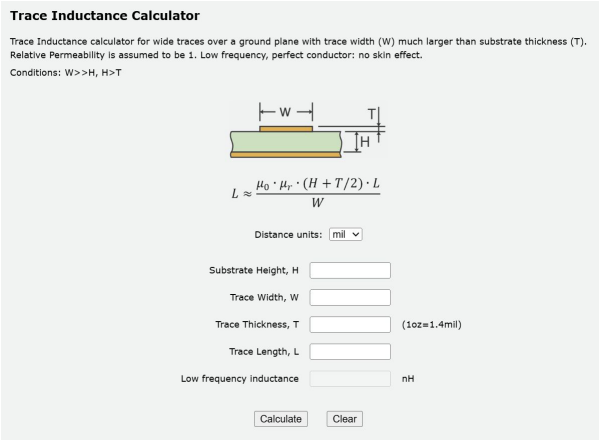
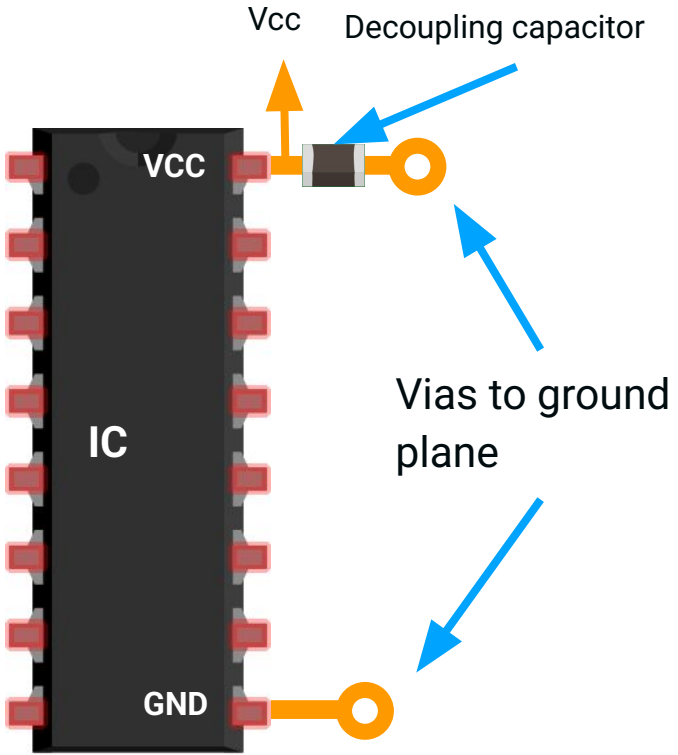


FIGURE 1. Noise in Analog and Digital Signals



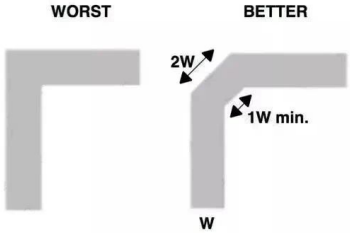
PCB Layout Tips

- **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 135° angles instead
  - The corners of 90° angles are narrower than the standard trace width; traces should be consistent widths
  - 90 degree are harder to etch as a trace
- **Make the power and ground planes big as possible**
  - Reduces heat buildup from high current
  - Improves signal integrity
  - Signal traces, which are low current, may be narrower



Setting up KiCad

- **Get out your laptops!**
- We will be setting up KiCad
    - This tutorial will be available on the project page when it releases

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.

CC BY-NC-SA 4.0