IEEE Project Competition

Week 4: Intro to KiCad

Sign-in Form

Finalized Teams

Electric Skateboard (Mini) - Beginner	Block Moving Robot - Advanced	Audio Visualizer - Beginner (individual)	Arcade Game - Advanced (individual)	Floating Magnet - Beginner (Individual)	Batmobile - Beginner	LED Pitch Changer - Beginner	Climbing Robot - Beginner	Mobile Game - Beginner	Knightboy- Advanced
Evan Eichholz, Bree	Eric Segrest, Abdiel, Mike (maybe)	Alexander	Aldem	Joshua Guo	Amy Diaz, Liani Garcia, Jennifer	Landon Luke, Ryan Moran, Samuel Hustin	Viviana, Frankey, Rolando, Keya	Joshua, Jacob, Jason	Scott …
Airpodsred, breemars56	MrE9000, thebackdimple , _dasanii	limes1	syndric	tallasian0 281	Tedpluto, lenedinal	Landonlu ke, rmoran24	LanceEvo ,Madeem, amnotkt	Espial7, Funnmy, jasr321	.planky.

College of Engineering and Computer Science

Reminders

- Check Discord we now have created personal project group chats.
- Please make sure to have a computer and mouse on hand, trackpad is fine.
- If you have questions about team roster DM or we will talk after class.
- Workshop will be centered around one PCB.
- General structure for today:

What is KiCad -> Install -> Schematic -> Footprints -> PCB View -> Final

What is PCB design?

Printed Circuit Board

- Designing the layout of electronic circuits on a board
- Connects various components via conductive pathways, signal traces

Importance:

- Central to all electronic devices
- Determines performance, reliability, and manufacturability

Steps in PCB Design:

- Schematic Capture
- Component Placement
- Routing the Traces
- Testing and Simulation
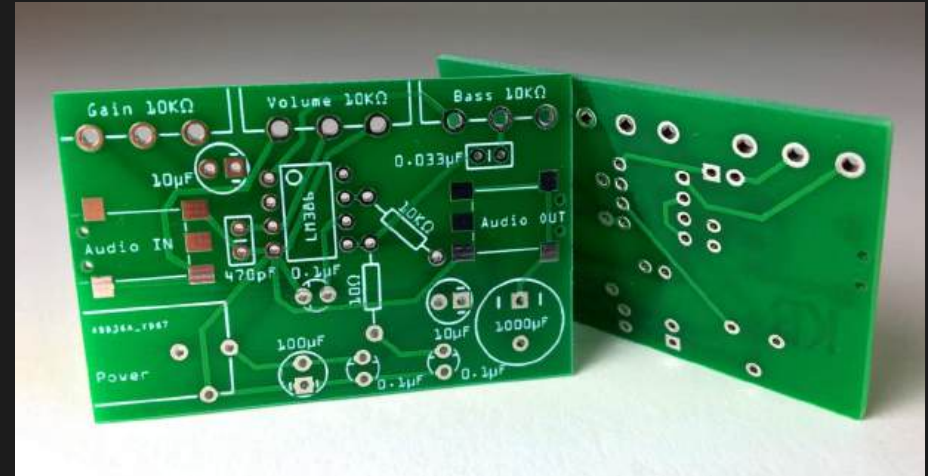
What is KiCad?

What is KiCad?

- KiCad is a full featured software suite for the production of PCBs

Key Features:

- Free and Open Source Software (FOSS)
- Works on Mac, Windows, and Linux
- Schematic Capture, PCB Layout, and 3D Viewer

Who Uses KiCad?

- Engineers
- Designers
- Educators and Students
- Hobbyists

KiCad

Main Features:

- **Schematic Editor:** Create schematics from built in parts libraries
- **PCB Editor:** Convert the schematic into a PCB
- **3D Viewer:** View preview of populated PCB

Advanced Features:

- **Symbol Editor:** Create or edit components for schematic
- **Footprint Editor:** Create or edit components for PCBs

College of Engineering
and Computer Science
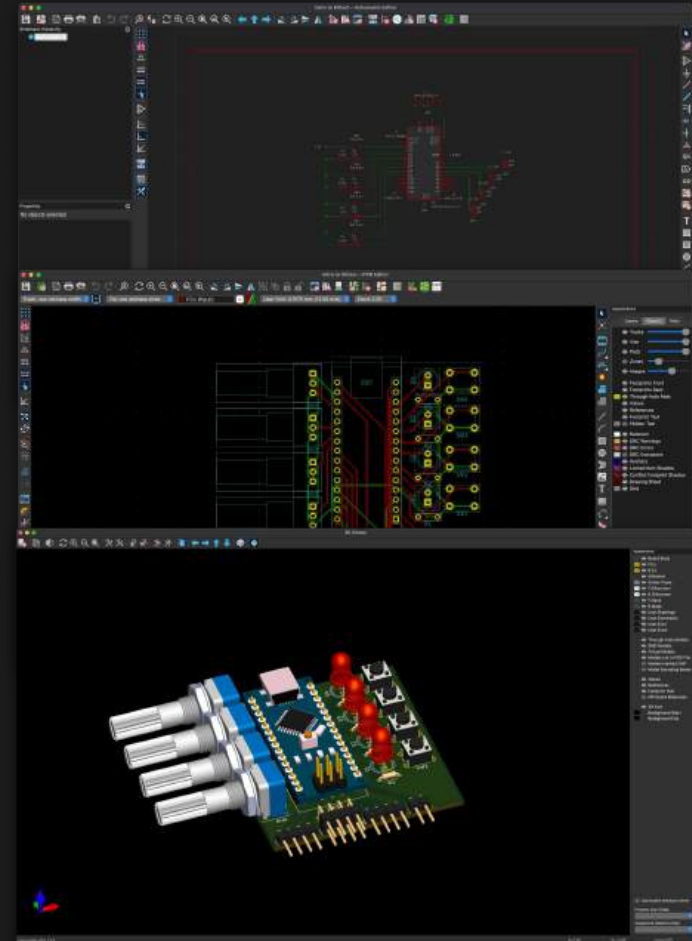
How to Download KiCad

1. **Go to** https://www.kicad.org/
2. Click Download
3. Select your operating system
4. Download from worldwide for your OS version
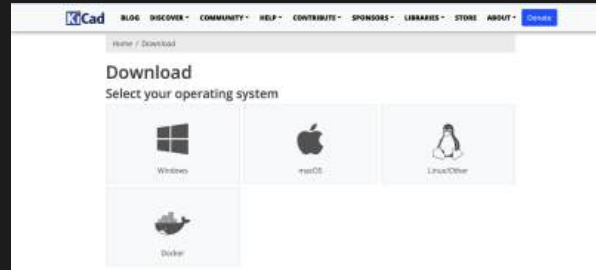
How to Install KiCad

For Mac

1. Open the DMG file
2. Drag the KiCad folder into the Applications folder shortcut
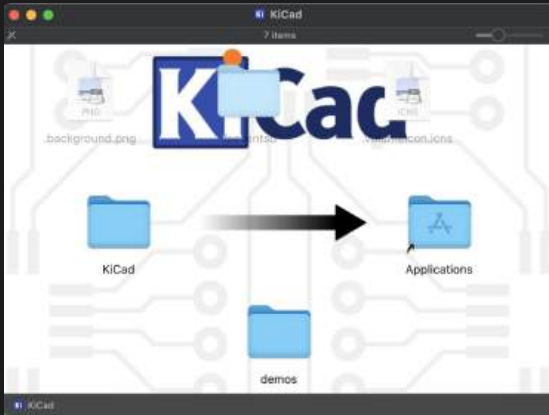
For Windows

1. Open the EXE file
2. Follow the on screen instructions to complete install

College of Engineering and Computer Science

Creating your first project

- Click the New Project button in the top left
- Choose a location and a name to save your project under
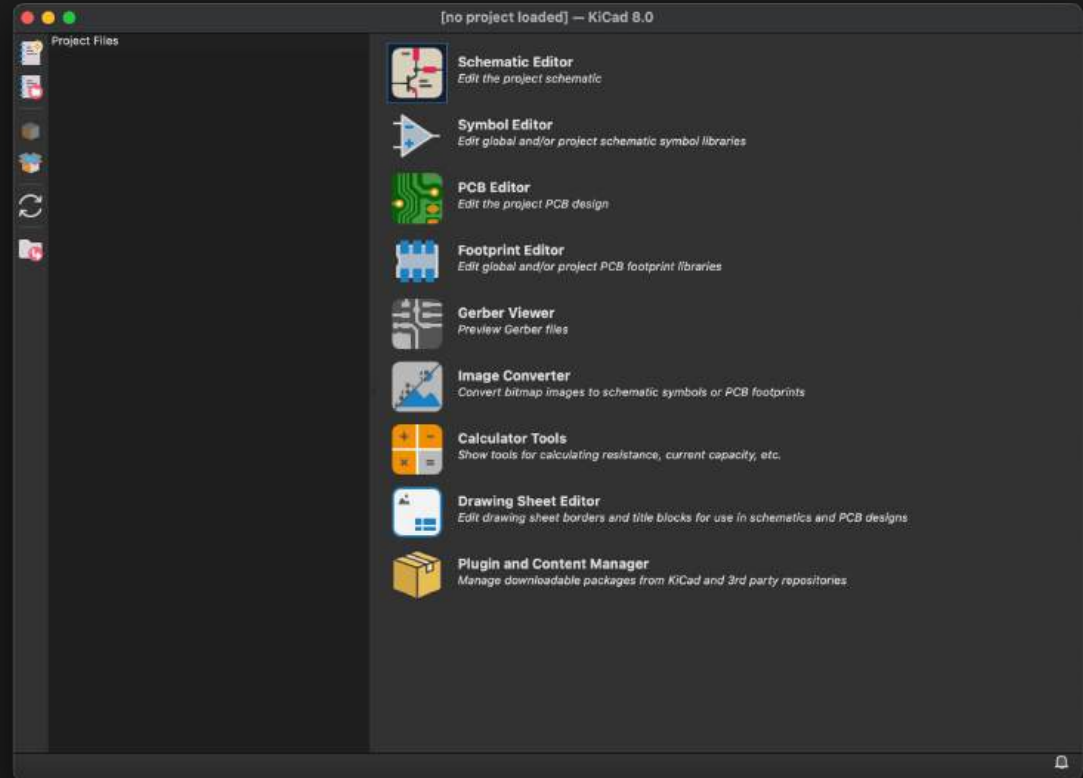
Creating your first project

- You now have a project to work under
- To get started click the Schematic Editor button
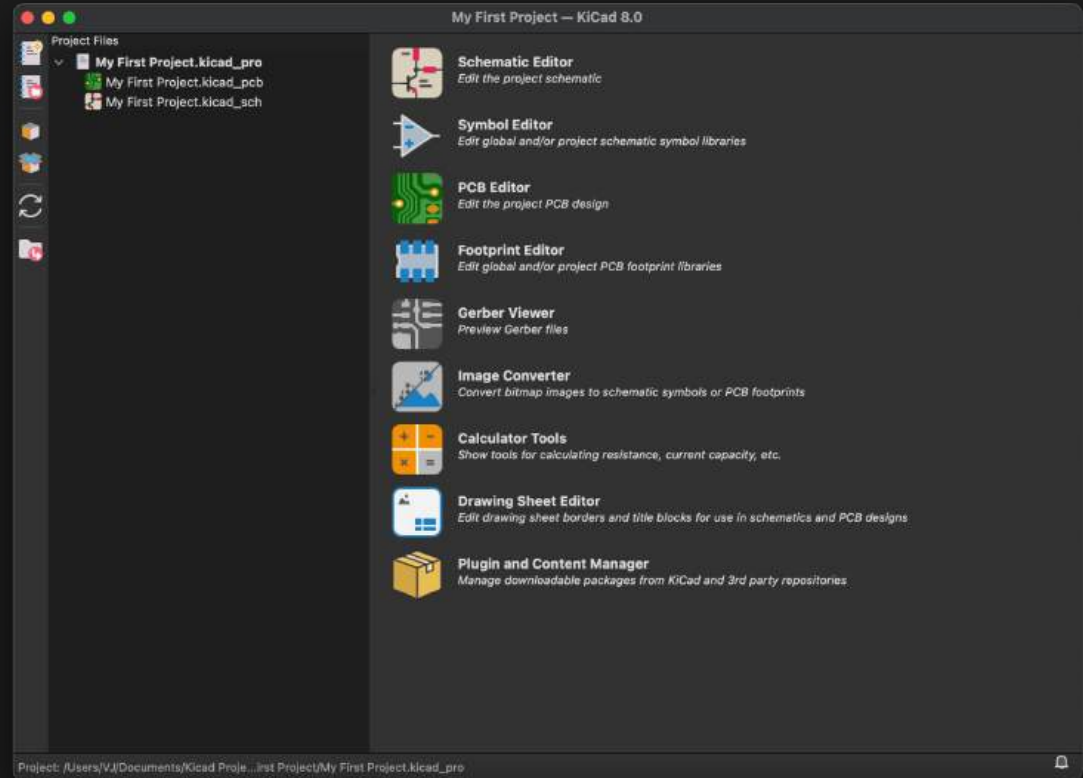
Creating your first schematic

- To get started you will use the Add Symbol tool (A) in the right toolbar
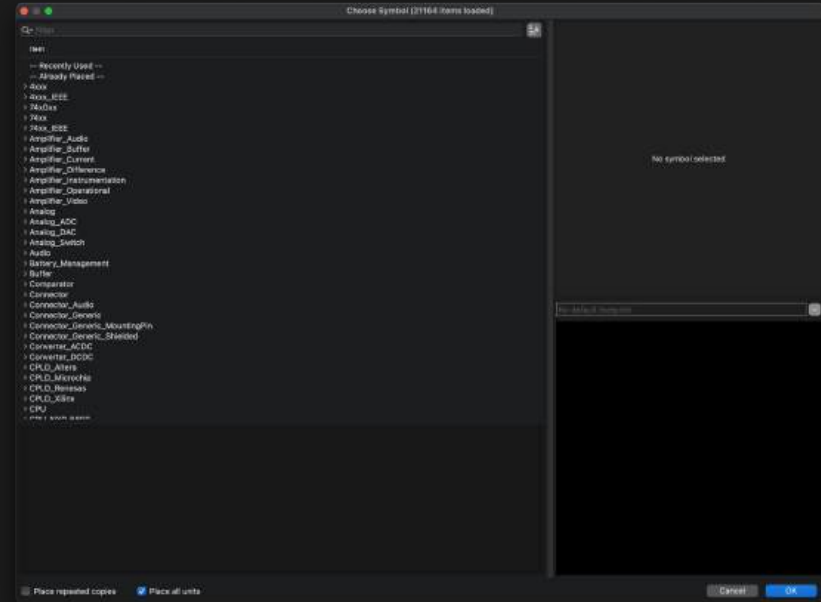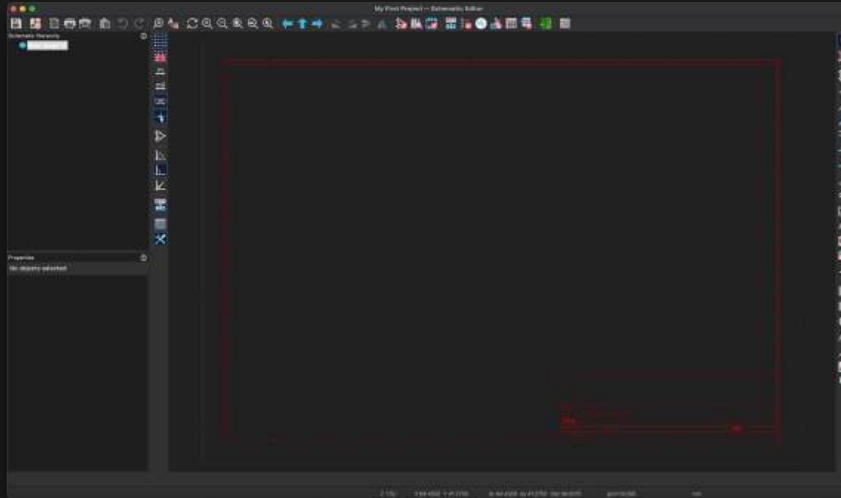
Creating your first schematic

- First symbol to place, search for arduino nano
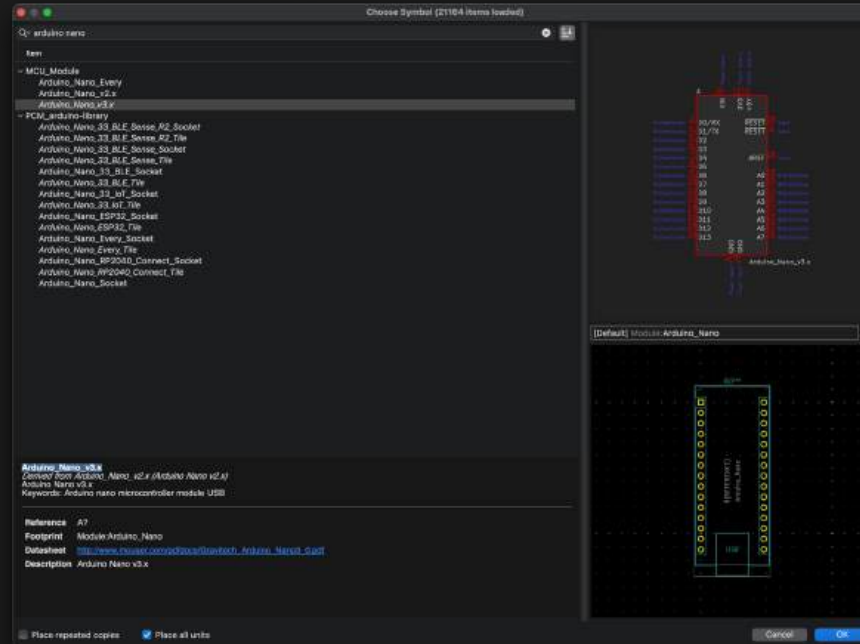
Creating your first schematic

- Keep using the Add Symbol tool (A) to add more components, along with copy and paste
- Add 4 resistors, 4 LEDs, and 4 push buttons
- Hover over a symbol and press V to Edit the Value Field

College of Engineering and Computer Science
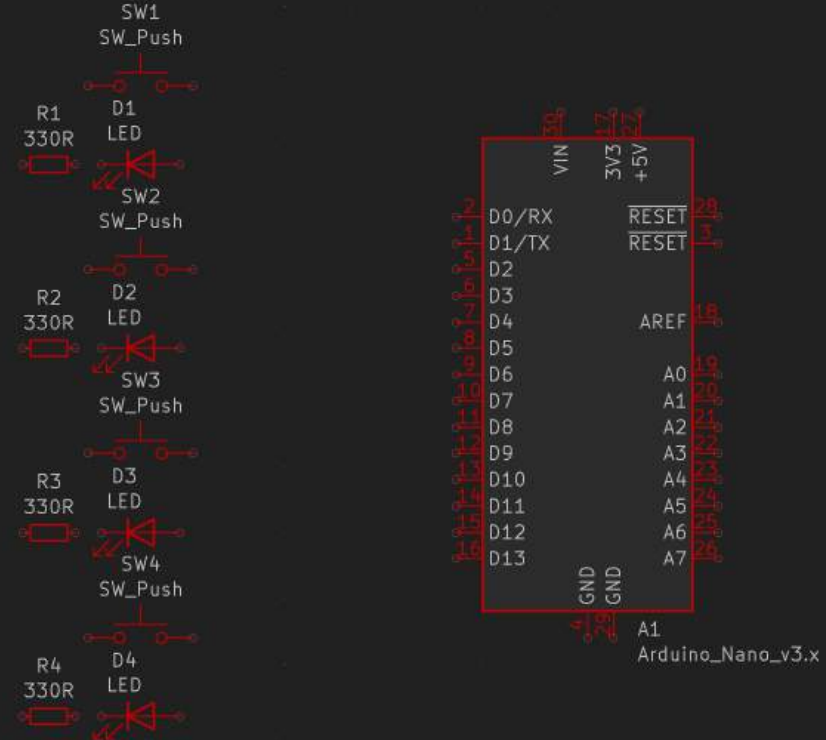
Creating your first schematic

- Use the Add Power tool (A) and search for gnd to add ground symbols
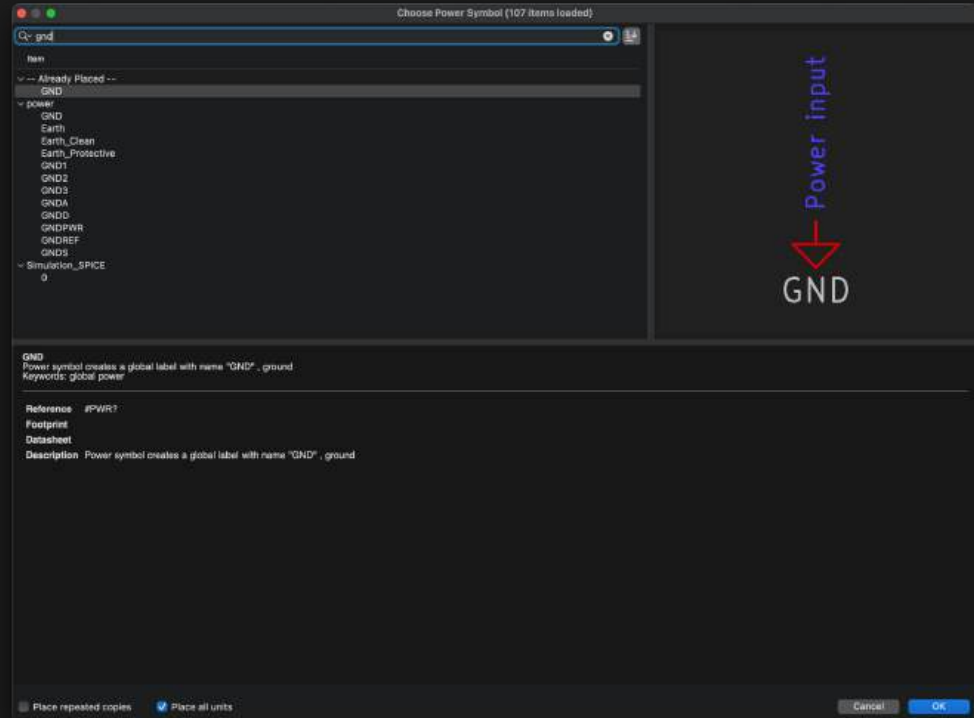
Creating your first schematic

- You can place more than one and they will automatically be connected to each other
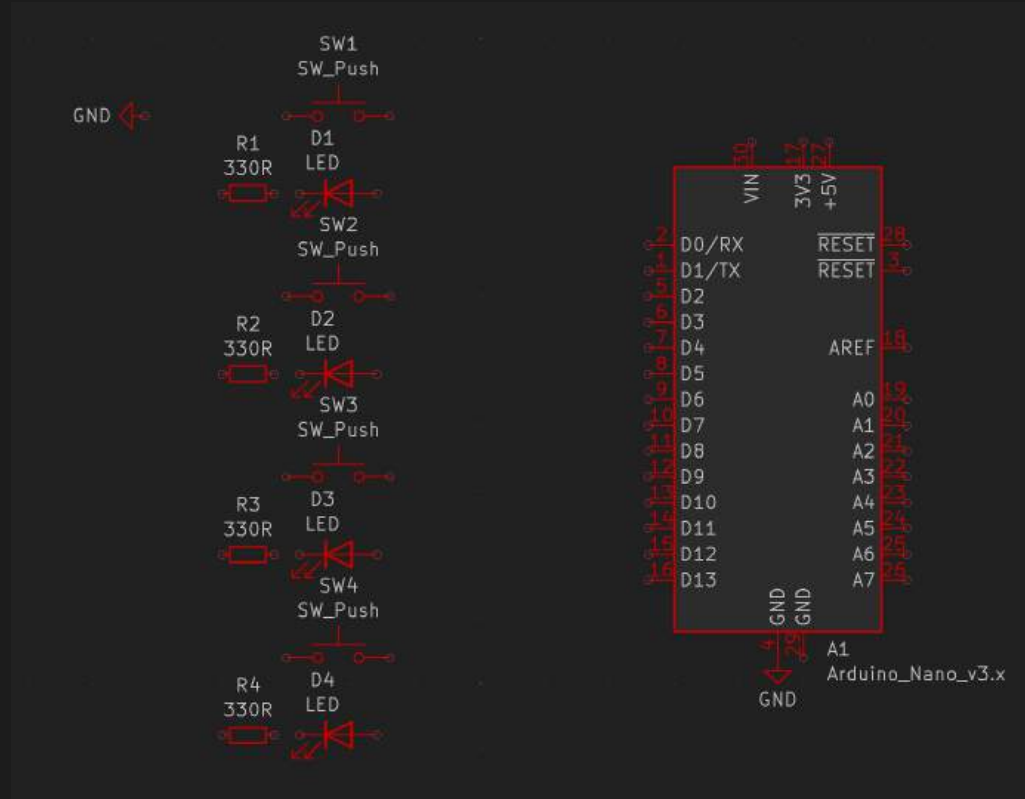
Wiring your first schematic

- You are now ready to connect your schematic together
- Use the Add Wire tool (W) or hover over the tiny circle on a component to draw wires when this symbol appears
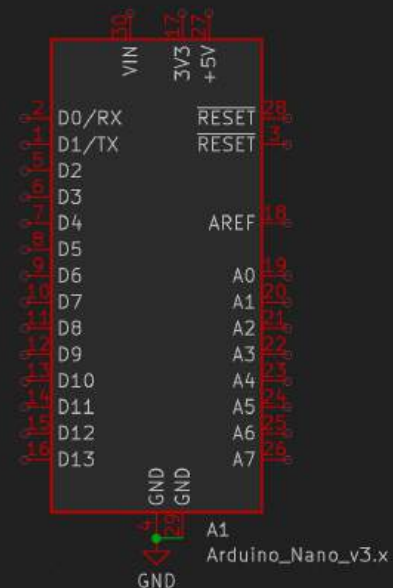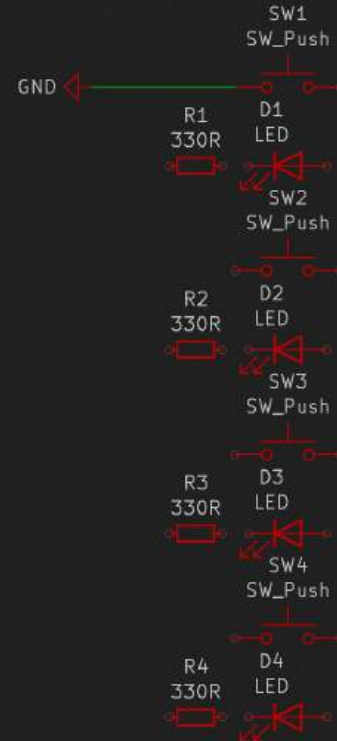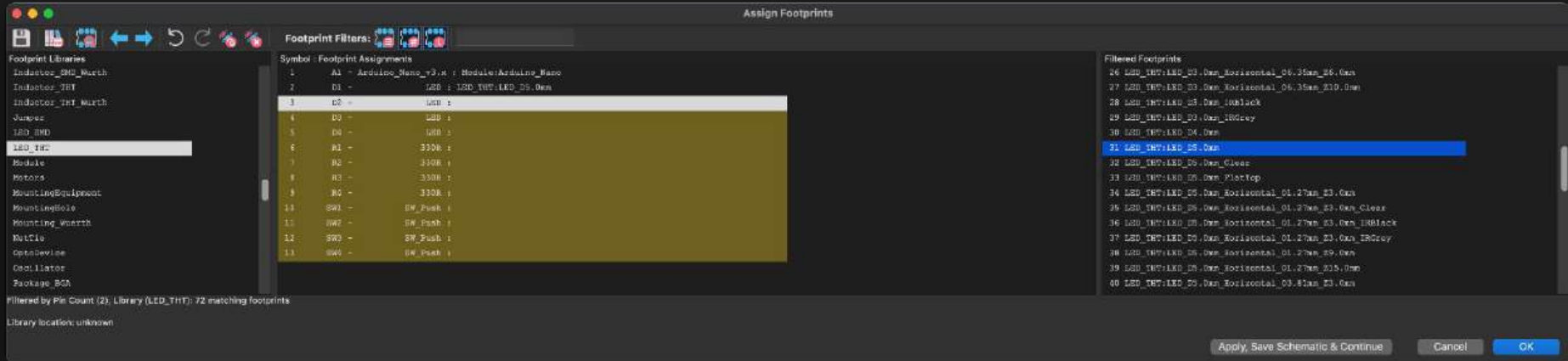
Wiring your first schematic

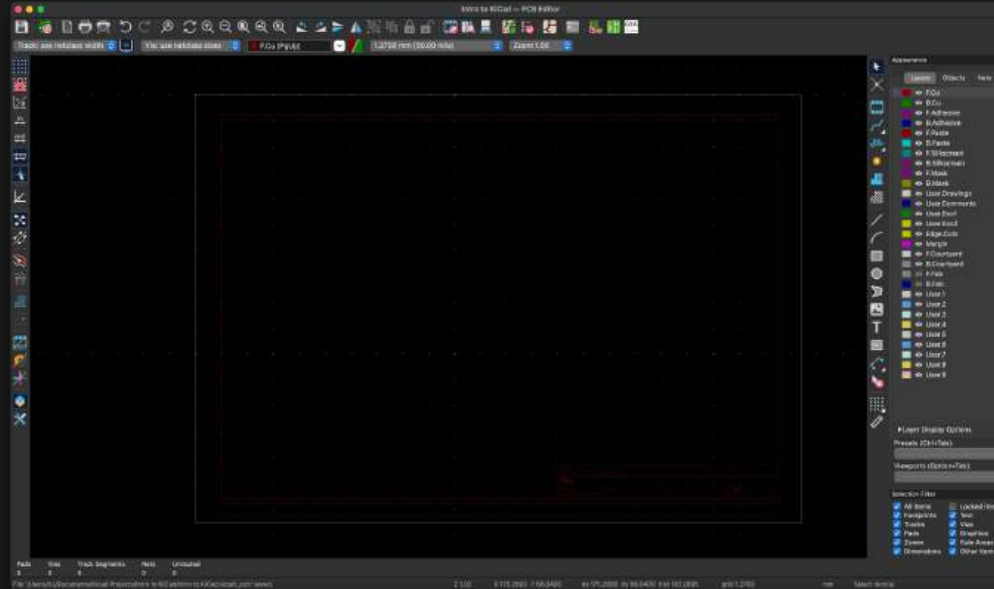- Continue to wire the entire schematic
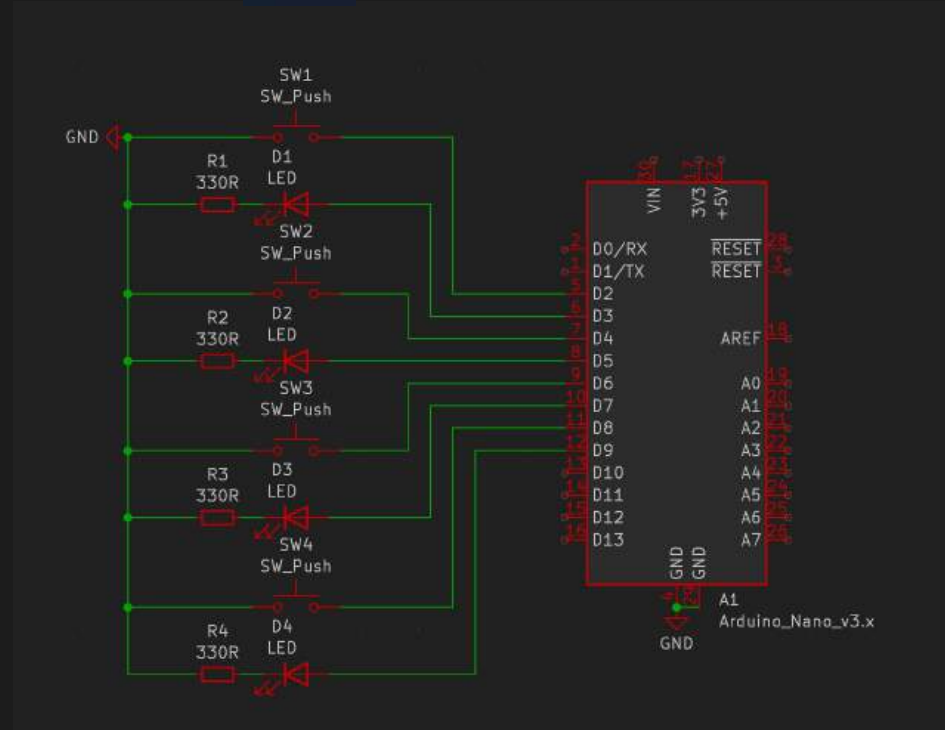
Finishing your first schematic

- Now you will use the Assign Footprints tool in the top toolbar
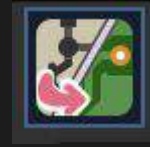- Use this to assign the actual components you will be using to their corresponding schematic symbol
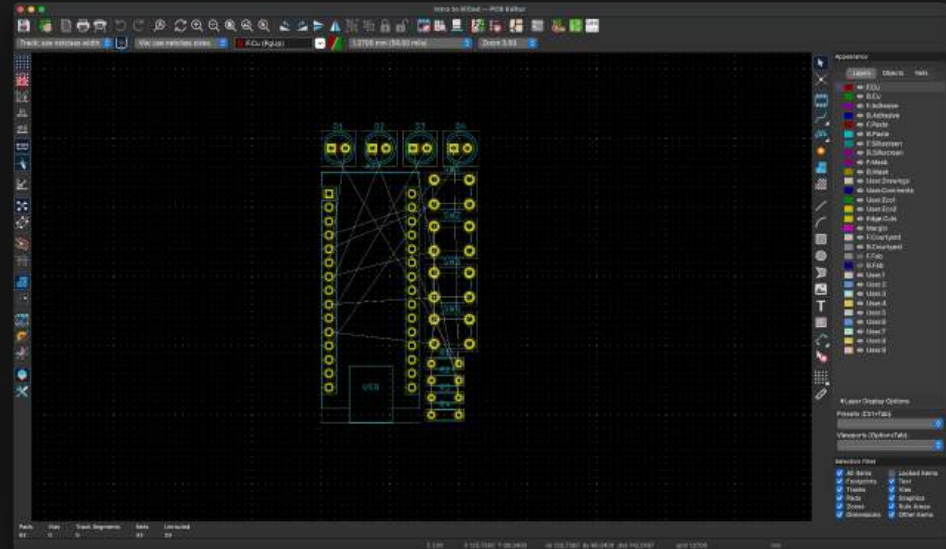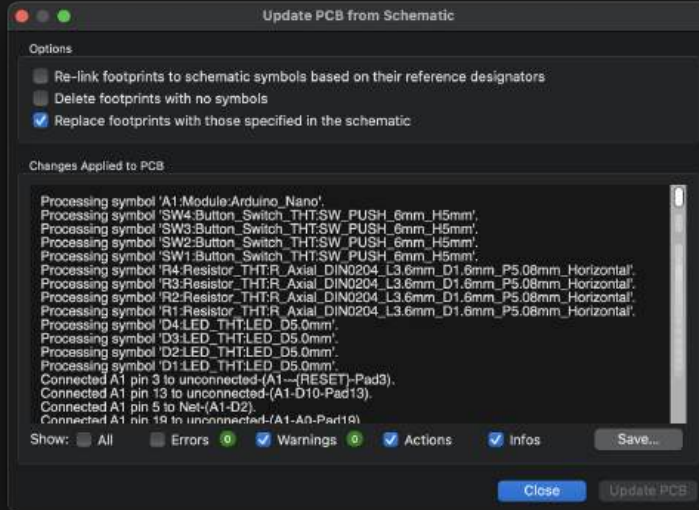
Creating your first PCB

- Click the switch to PCB Editor button in the top toolbar to open the PCB Editor
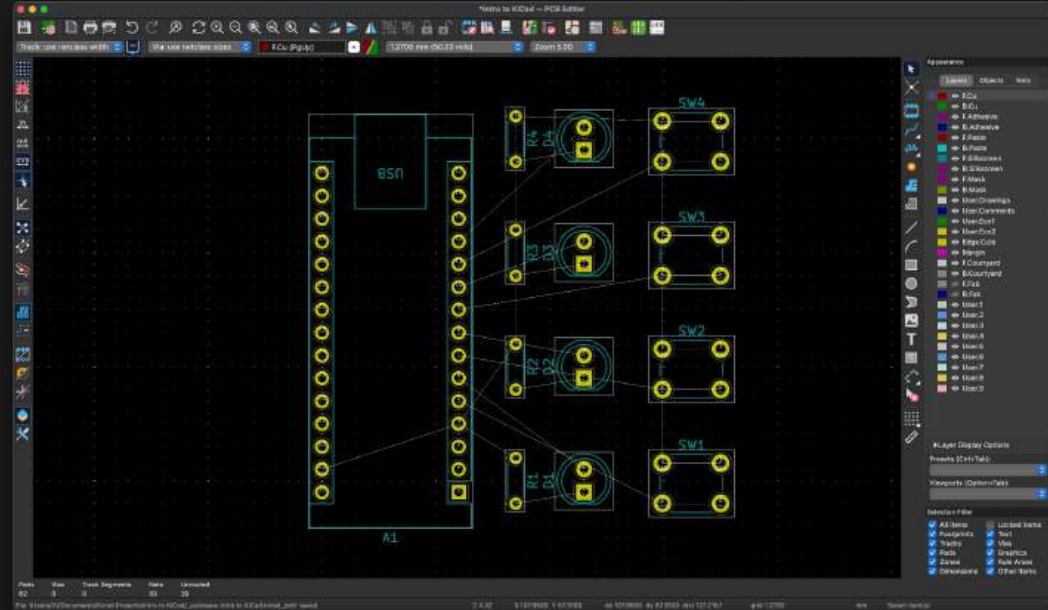
Creating your first PCB

- Click the update PCB from Schematic button in the top toolbar to add all the assigned footprints from the schematic to the PCB
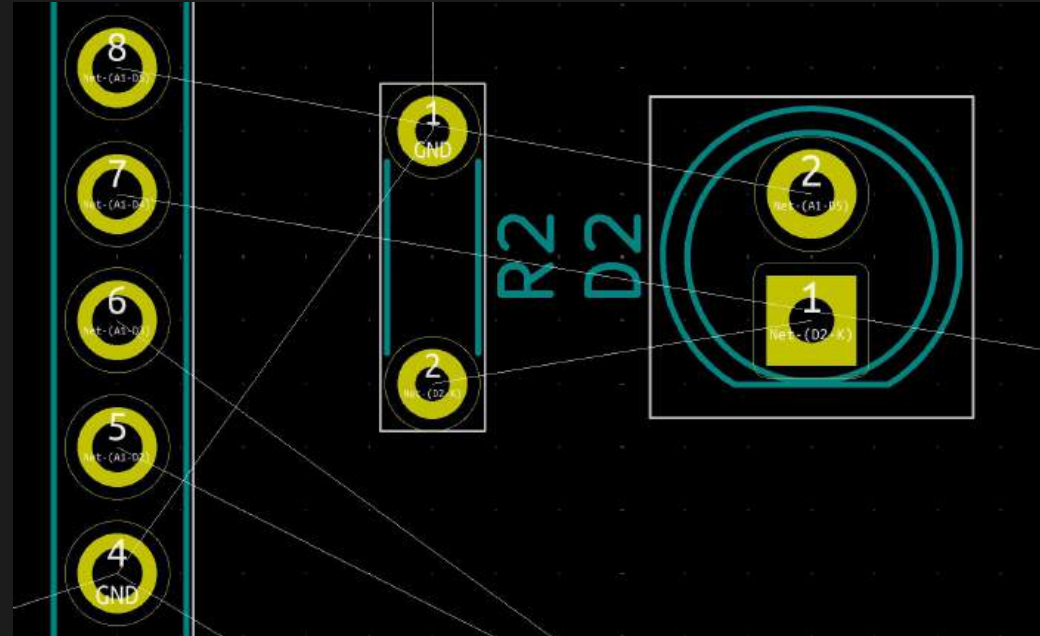
Arranging your first PCB

- Click the footprints and Use the Move command (M) along with the Rotate command (R) to arrange the components on the PCB
- If you aren't sure about placement it can help to have a working breadboard already made that you can copy from
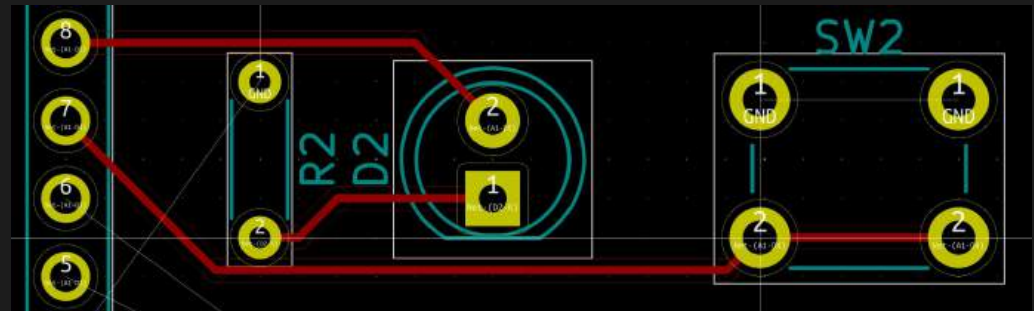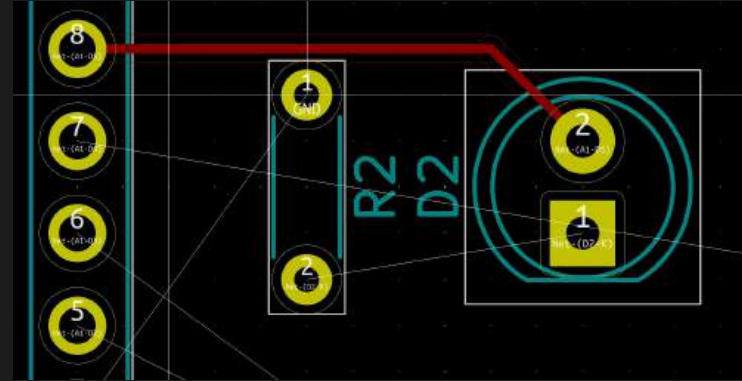
Wiring your first PCB

- All the floating lines connecting the footprints together is known as a rats nest
- These lines correspond to your schematic so you know where to place wires
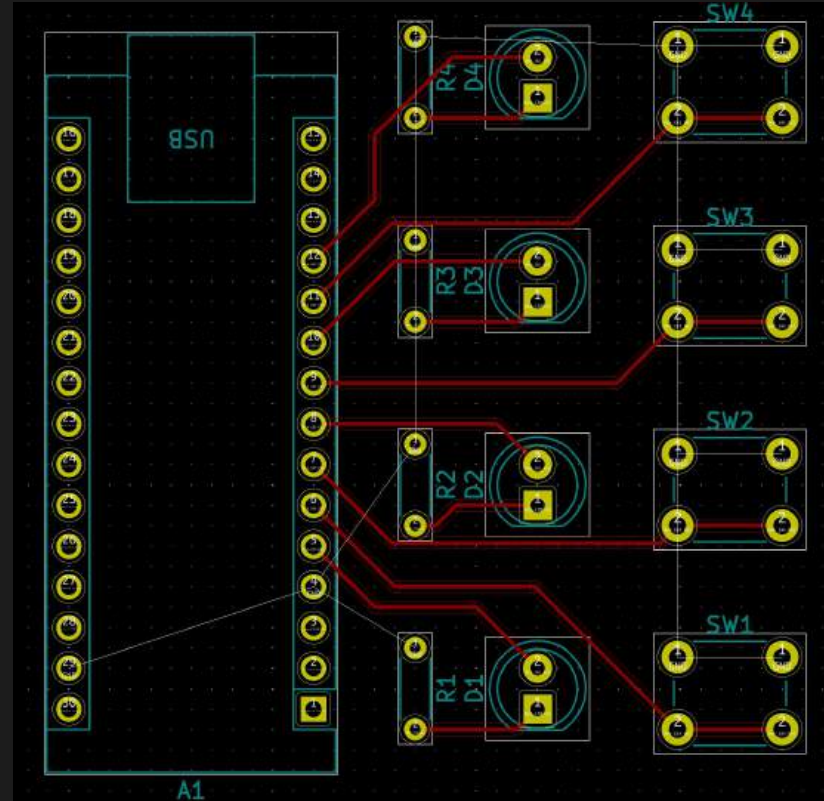
Wiring your first PCB

- Use the Route Tracks tool (X) in the right toolbar to place wires
- Click a pad with a rats nest to start routing it
- As you route tracks you can use the autoroute shortcut (F) to help speed up placement
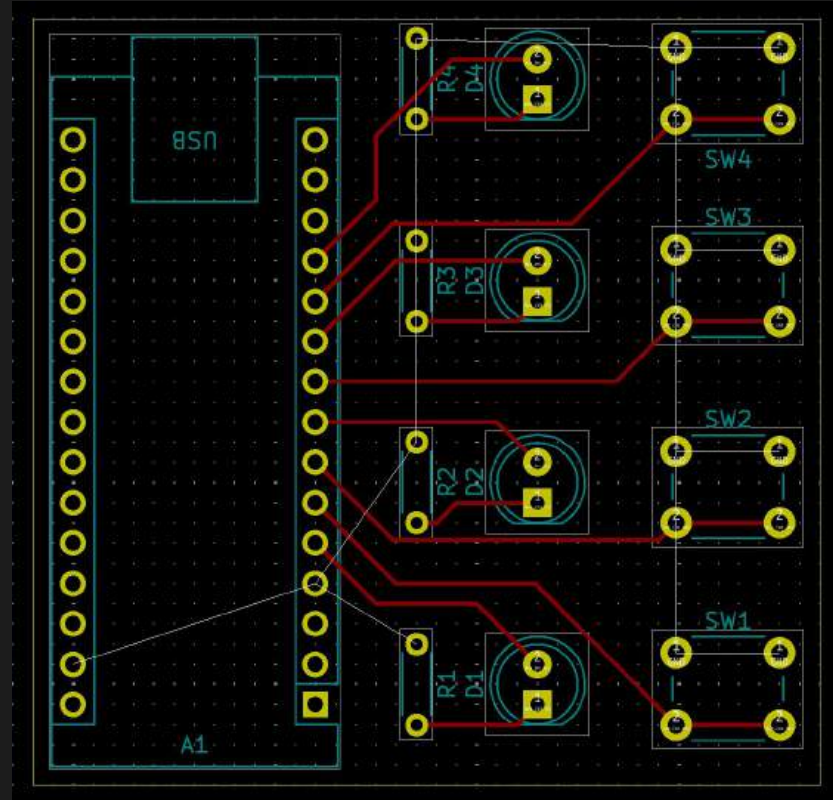- Save power and ground nets for last, you can hide them in the Nets tab on the right

Sizing your first PCB

- With most of the wiring now complete it's important to make a board outline
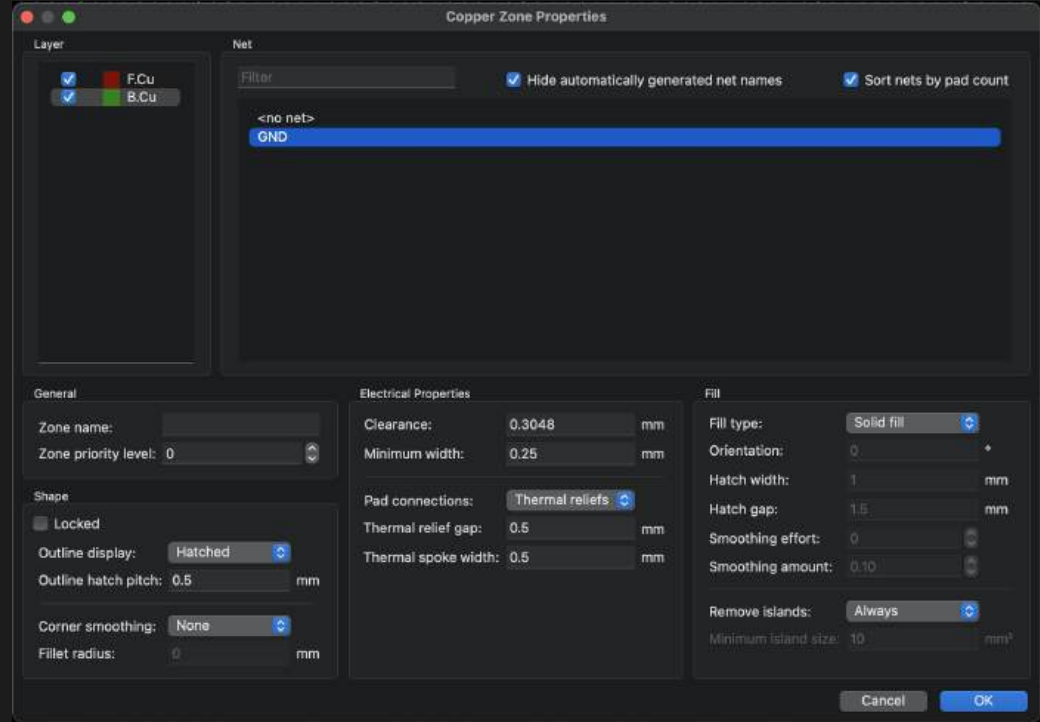
Sizing your first PCB

- First select the Edge.Cuts layer from the Layers tab on the right
- In that layer use the Draw Rectangle tool to draw an outline around your footprints
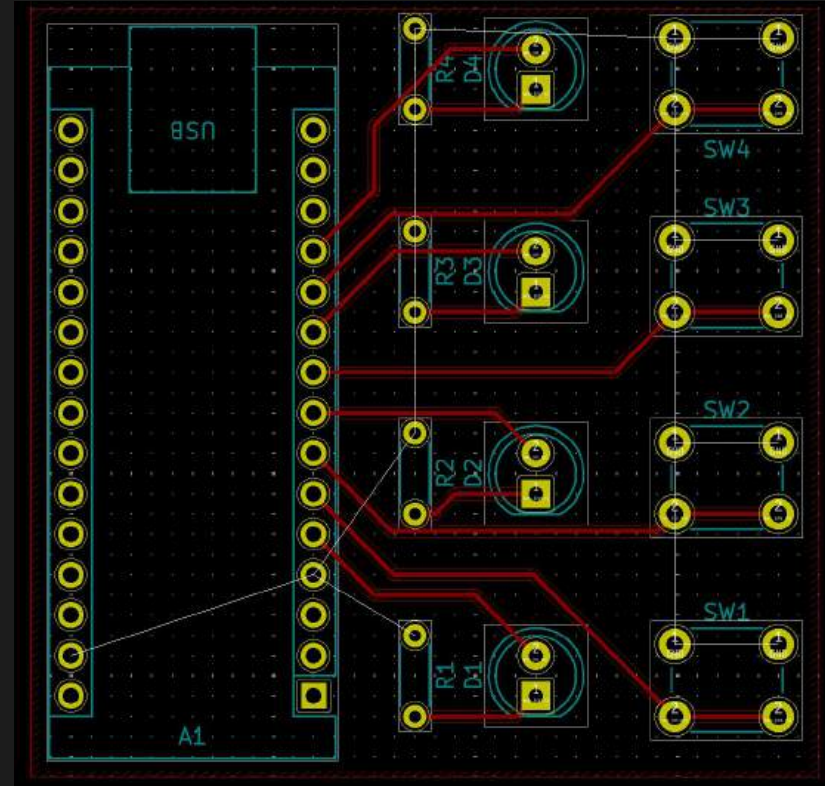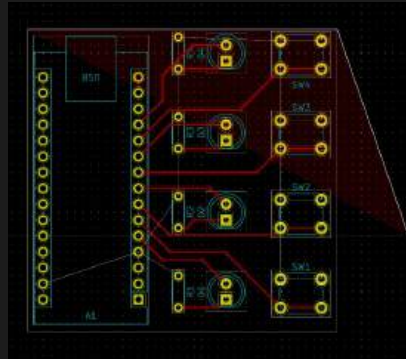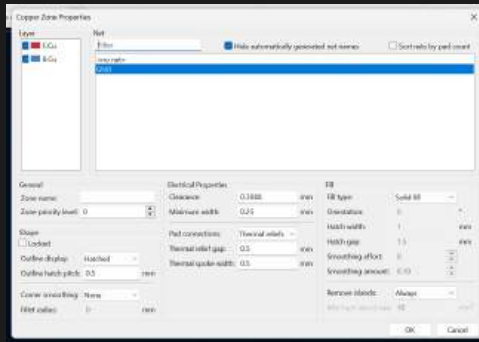
Wiring your first PCB

- All we have left is ground so we can use a ground plane
- Go back to the Front layer
- Use the Add Filled Zone tool and place it in the corner of the board outline
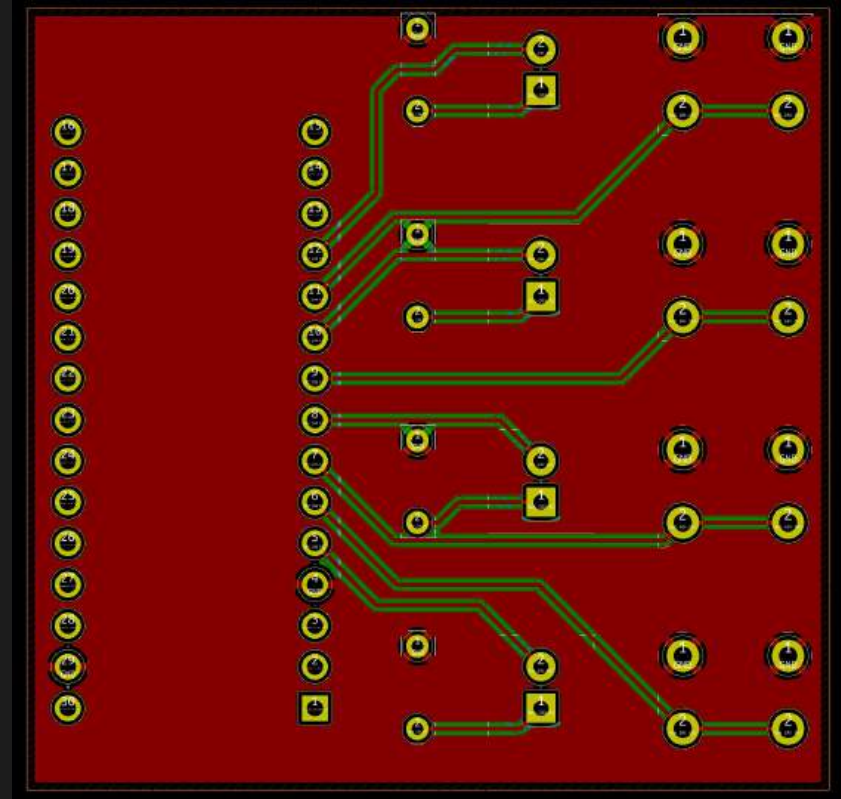- Checkmark both Layers and select the GND net

College of Engineering and Computer Science

Wiring your first PCB

- Once you click OK you can now place the outline of the ground plane, put it in the same place as the board outline
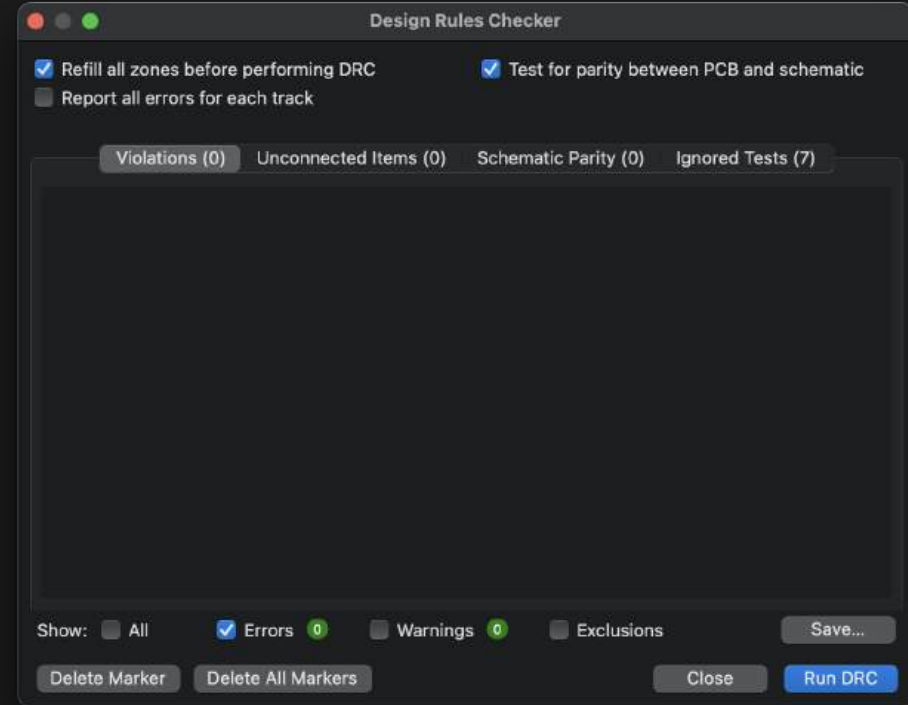
Wiring your first PCB

- Now you use the Fill All Zones tool (B) to update your ground plane
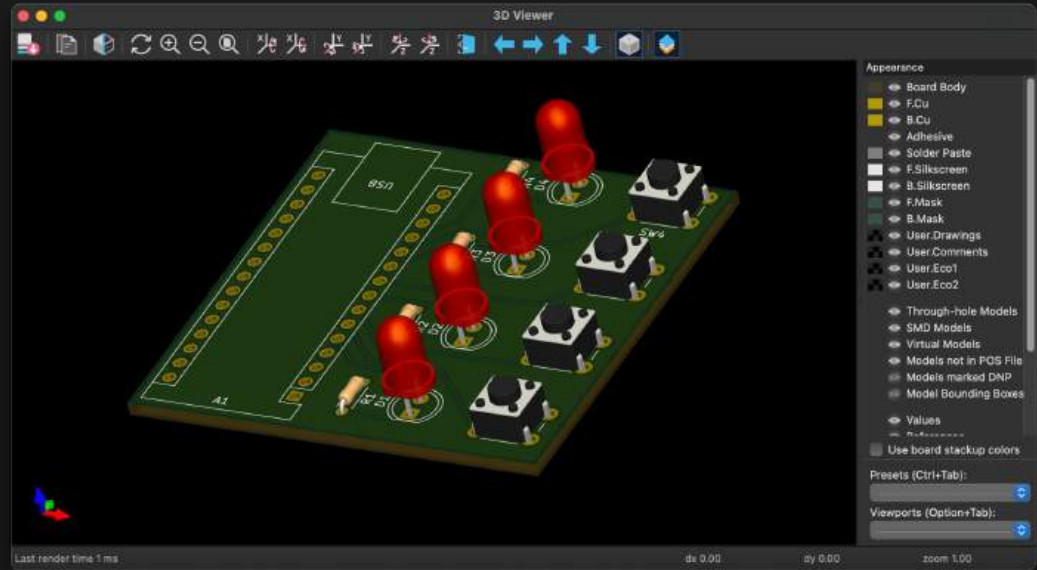
Verifying your first PCB

- Use the Design Rules Checker in the top toolbar to check for any errors in your design
- If no errors are present then your first PCB is complete

Design Rules Checker

☑ Refill all zones before performing DRC   ☑ Test for parity between PCB and schematic
☐ Report all errors for each track

| Violations (0) | Unconnected Items (0) | Schematic Parity (0) | Ignored Tests (7) |

Show: ☐ All   ☑ Errors 0   ☐ Warnings 0   ☐ Exclusions   Save...

Delete Marker   Delete All Markers   Close   Run DRC

Viewing your first PCB

- In the PCB Editor use the 3D Viewer tool (Alt+3) in the top toolbar to see a preview of your PCB

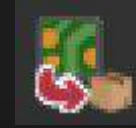College of Engineering and Computer Science
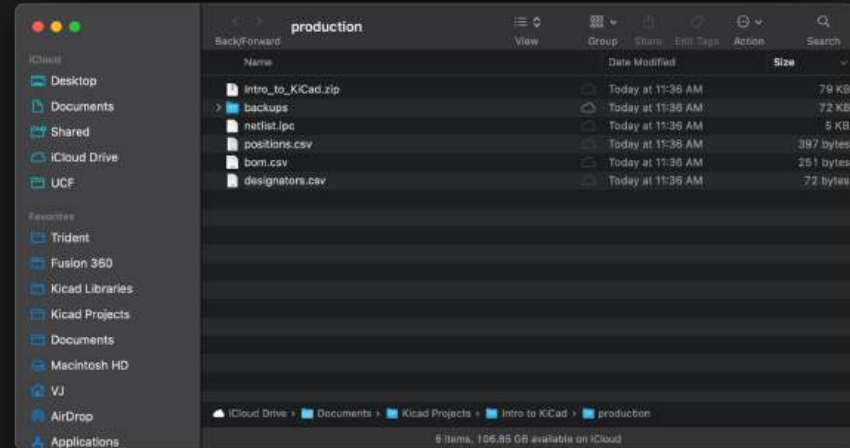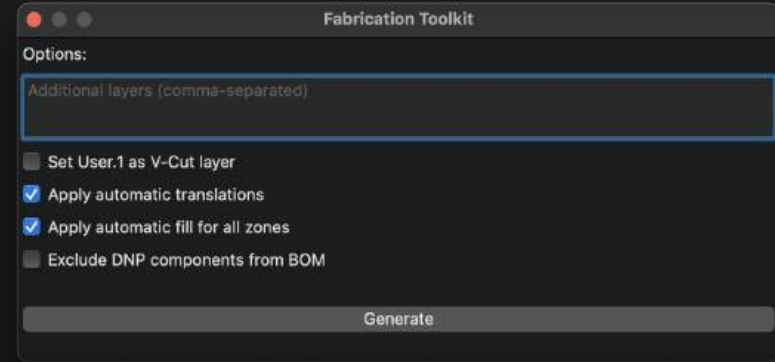
Exporting your first PCB

- In the Project Manager window click Plugin and Content Manager

- Search for jlc and install Fabrication Toolkit
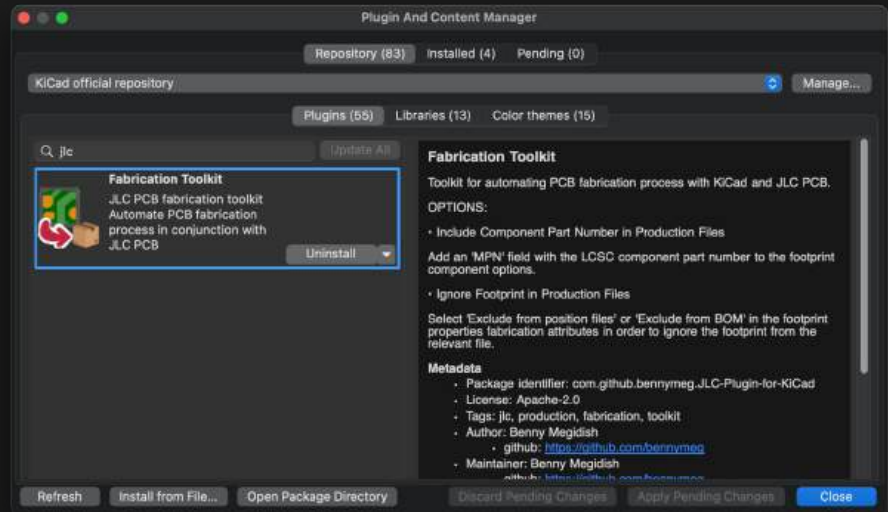
Exporting your first PCB

- In the PCB Editor click the Fabrication Toolkit icon in the top toolbar and click generate
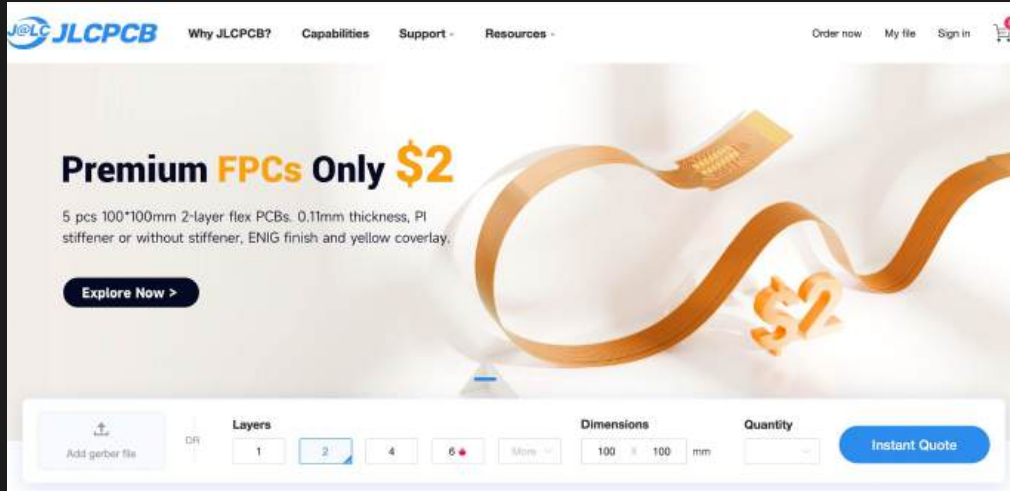- Once complete the folder with the production files will open automatically
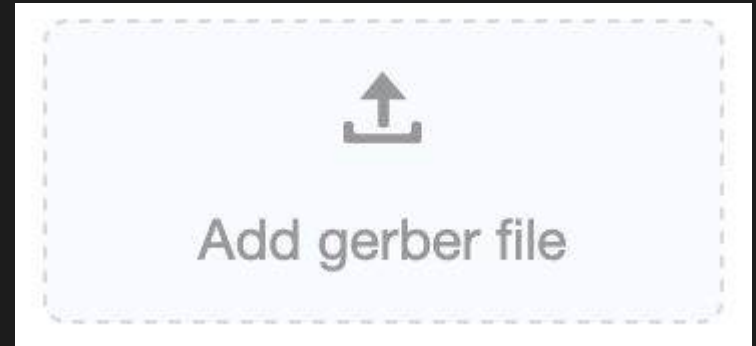
Ordering your first PCB

Intro_to_KiCad.zip

- Navigate to
  https://jlcpcb.com

- Drag & Drop the zip file in the production folder onto the Add gerber file box in the website

College of Engineering and Computer Science

Ordering your first PCB

- Congratulations! You can now order your first PCB

Timeline and Important Dates

Weekly Meetings - Estimated Timeline

~~Week 1~~	~~Week 2~~	~~Week 3~~	~~Week 4~~	Week 5	Week 6	WEEK 7	WEEK 8
~~Introduction~~	~~Microcontrollers - Erik~~	~~Fusion 360/Solidworks - Matias~~	~~KiCAD - Tino~~	Soldering/ Prototyping /General Project Management Skills/ Github	BOM Development Workshop	BOM DUE - Technical Assistance Workshop	Technical Assistance Workshop - Progress Checks

Reminders

- Please check discord channels for your respective project.
- Recruit if you want! The more the more you may learn and get out of the project!
- Next week soldering workshop/prototype development/project development/github. Applying our skills we learned!