

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, Funnymy, jasr321	.planky.

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

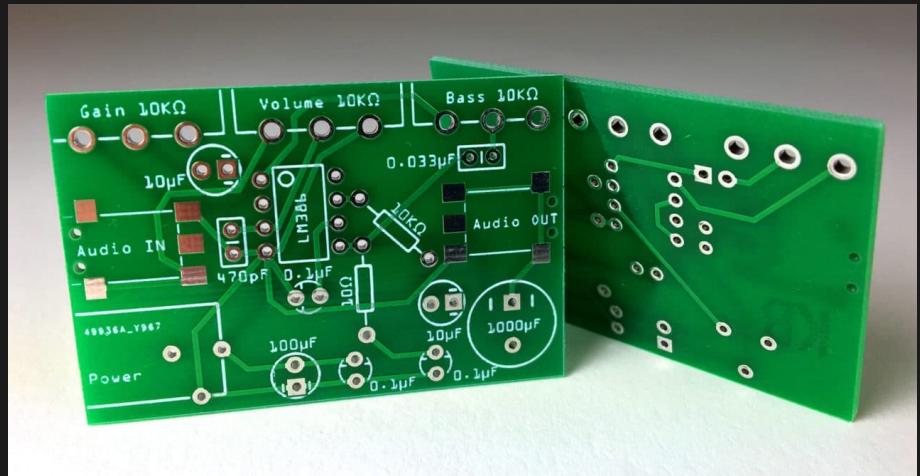
- Designing the layout of electronic circuits on a board
- Connects various components via conductive pathways, signal traces

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



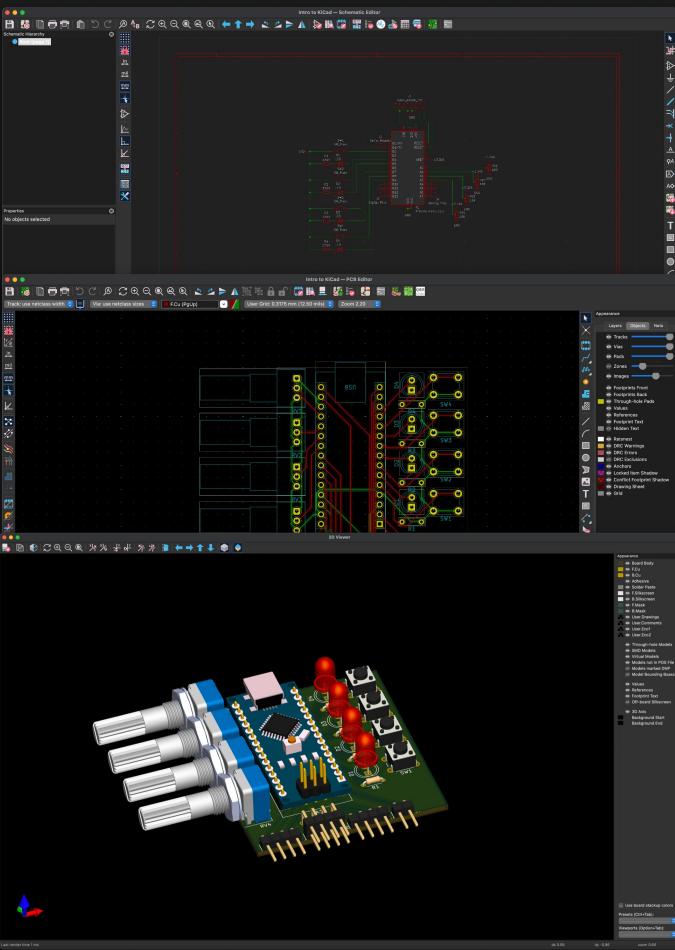
KiCad

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

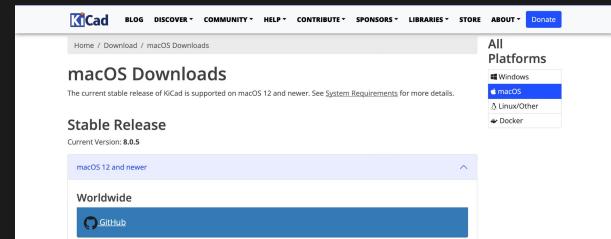
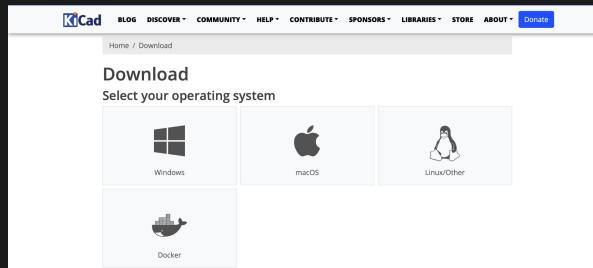
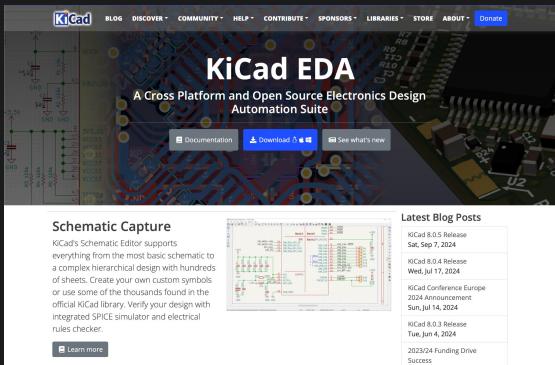
Advanced Features:

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



How to Download KiCad

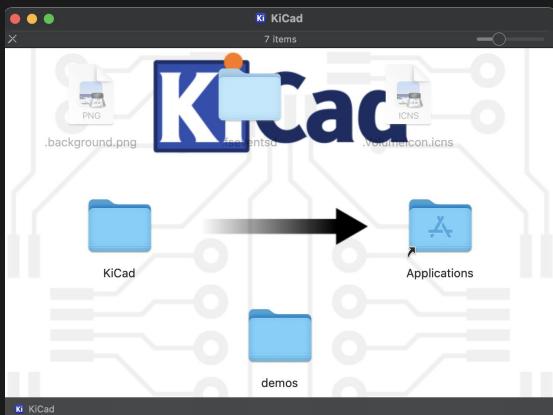
1. Go to <https://www.kicad.org/>
2. Click Download
3. Select your operating system
4. Download from worldwide for your OS version



How to Install KiCad

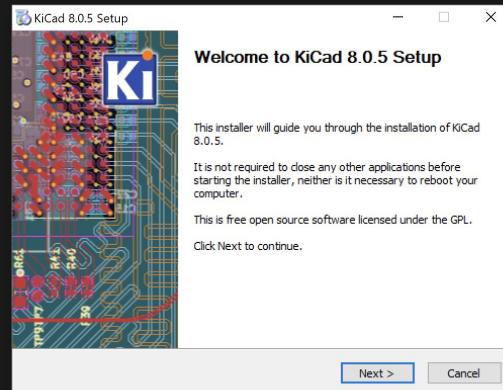
For Mac

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



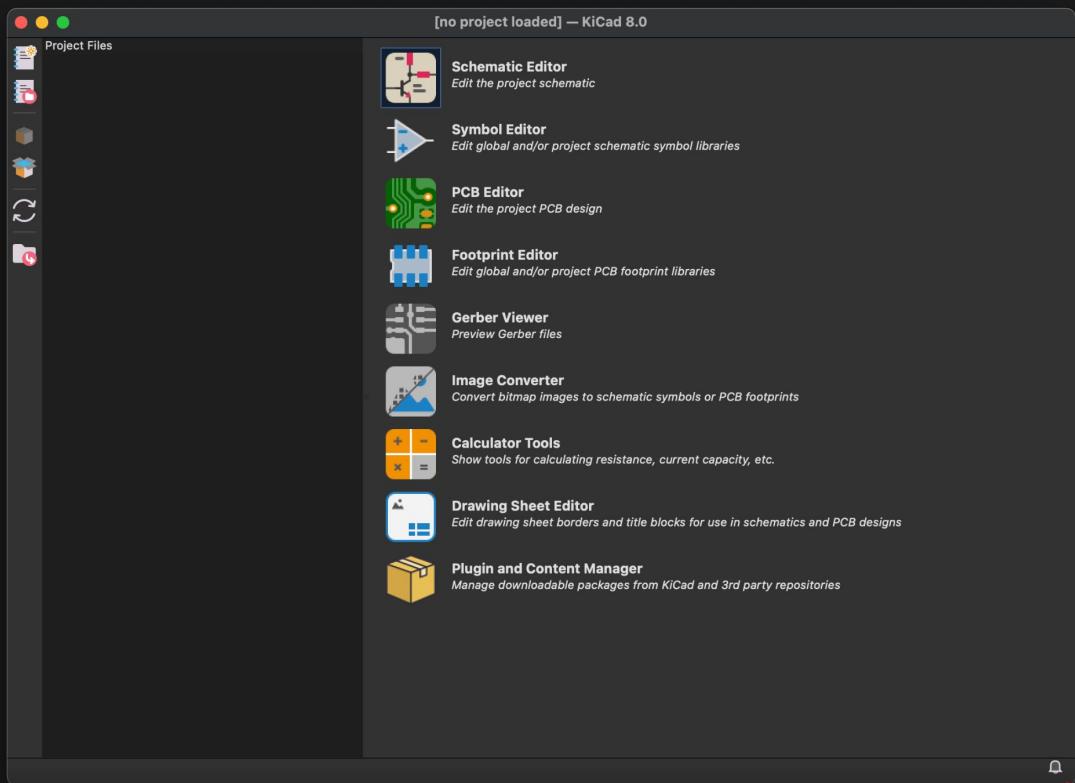
For Windows

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



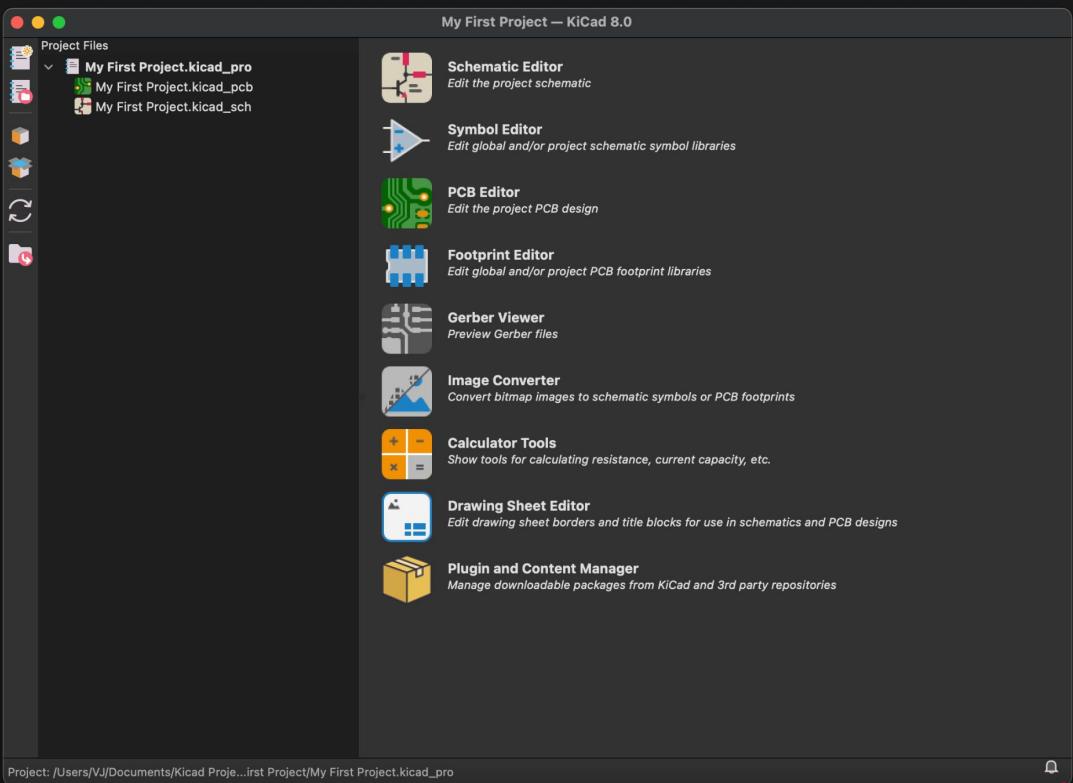
Creating your first project

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



Creating your first project

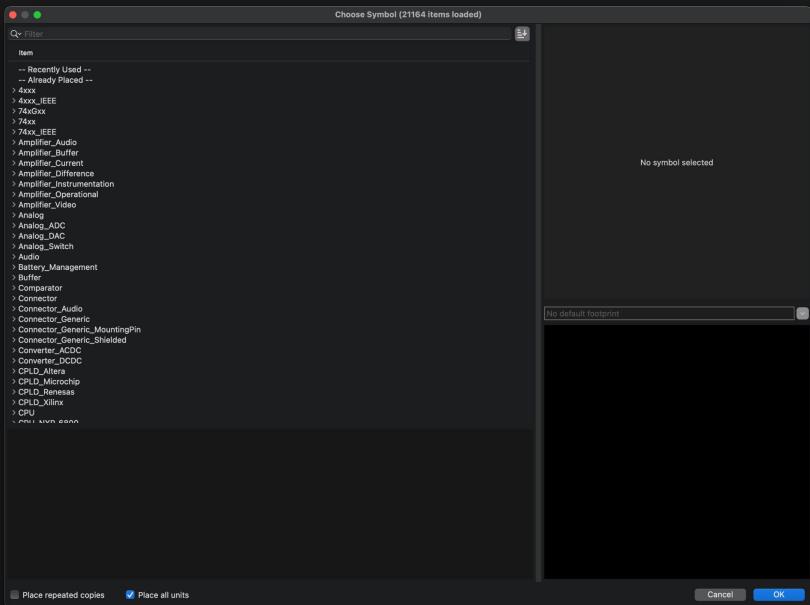
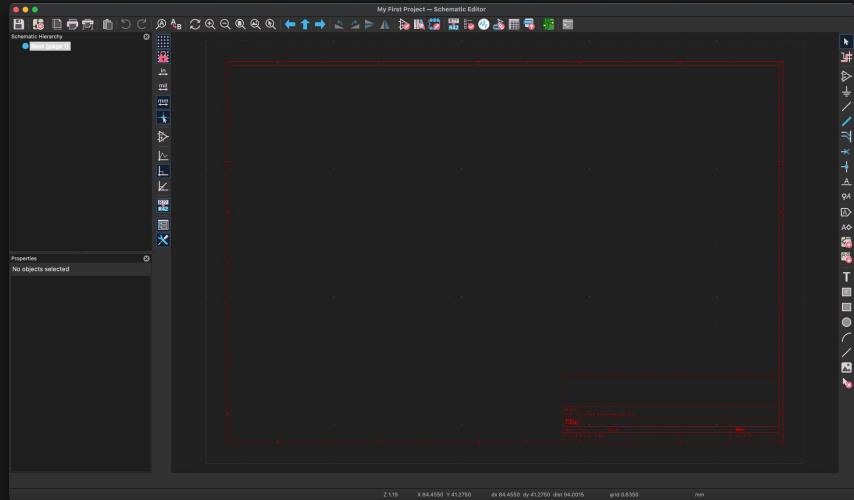
- You now have a project to work under
- To get started click the Schematic Editor button



Creating your first schematic



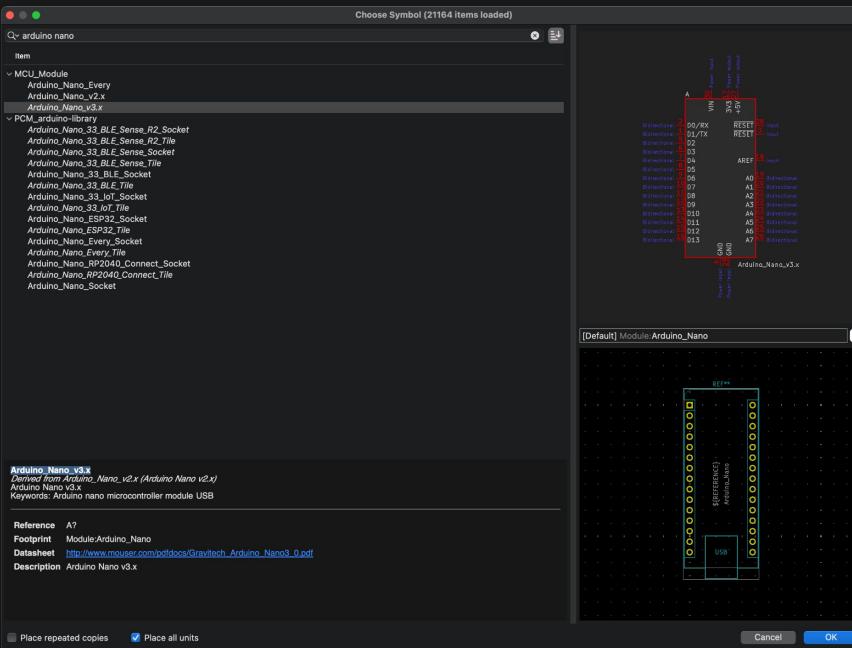
- To get started you will use the Add Symbol tool (A) in the right toolbar



Creating your first schematic



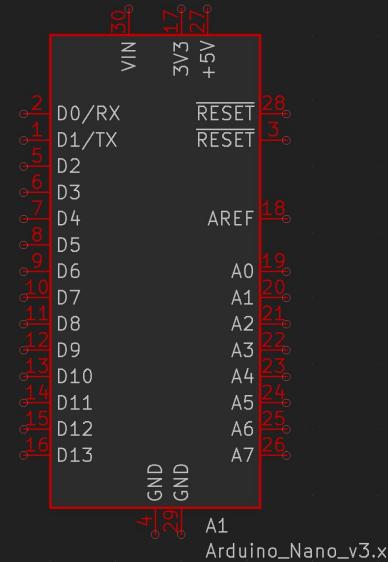
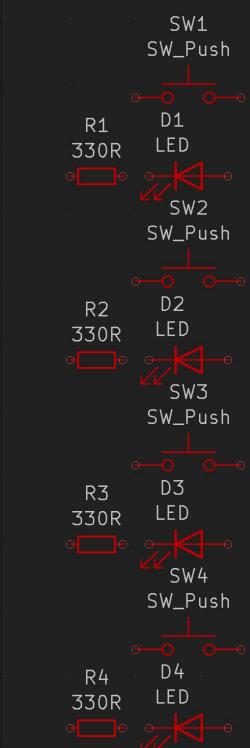
- First symbol to place, search for arduino nano



Creating your first schematic



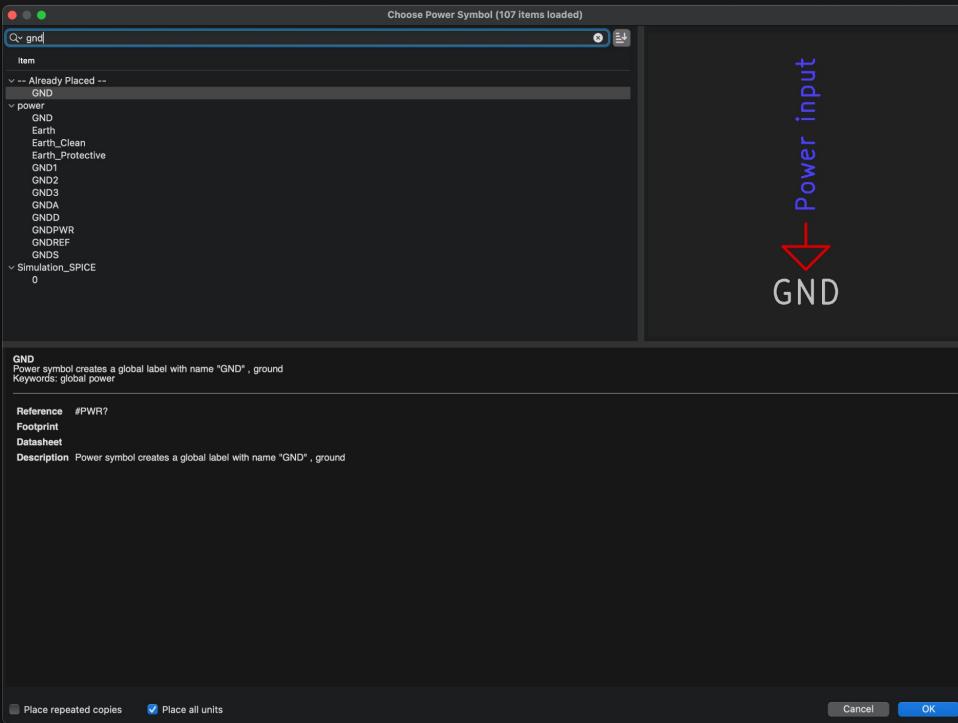
- Keep using the Add Symbol tool (A) to add more components, along with copy and paste
- Add 4 resistors, 4 LEDs, and 4 push buttons
- Hover over a symbol and press V to Edit the Value Field



Creating your first schematic



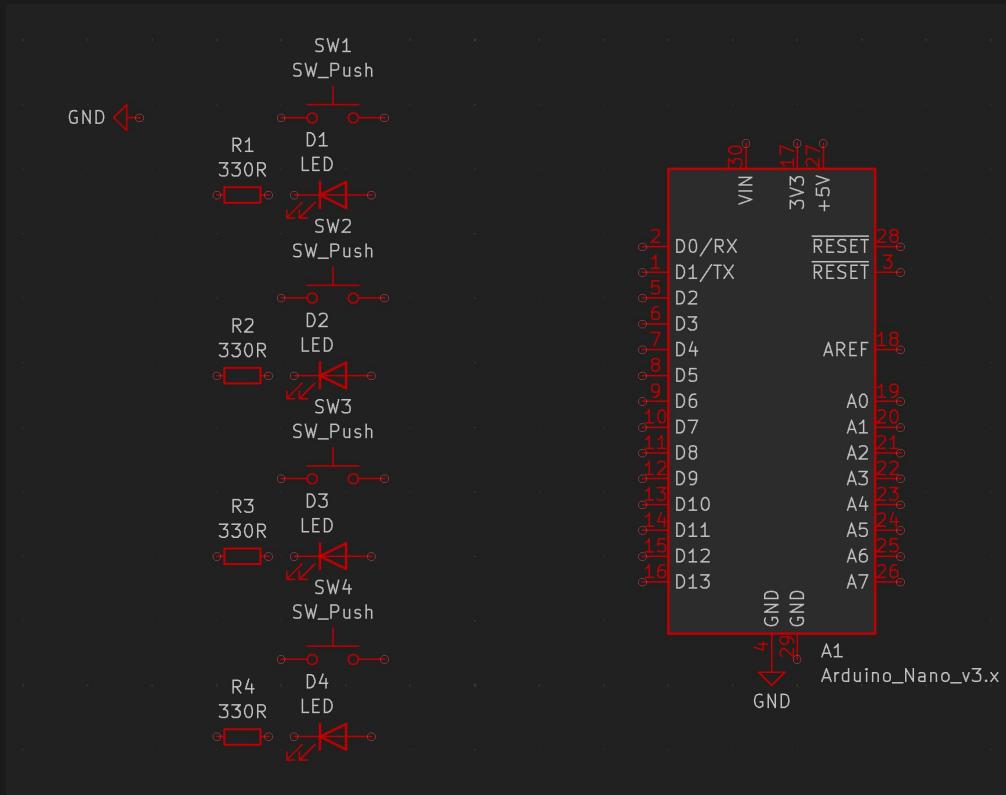
- Use the Add Power tool (A) and search for gnd to add ground symbols



Creating your first schematic



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



Wiring your first schematic



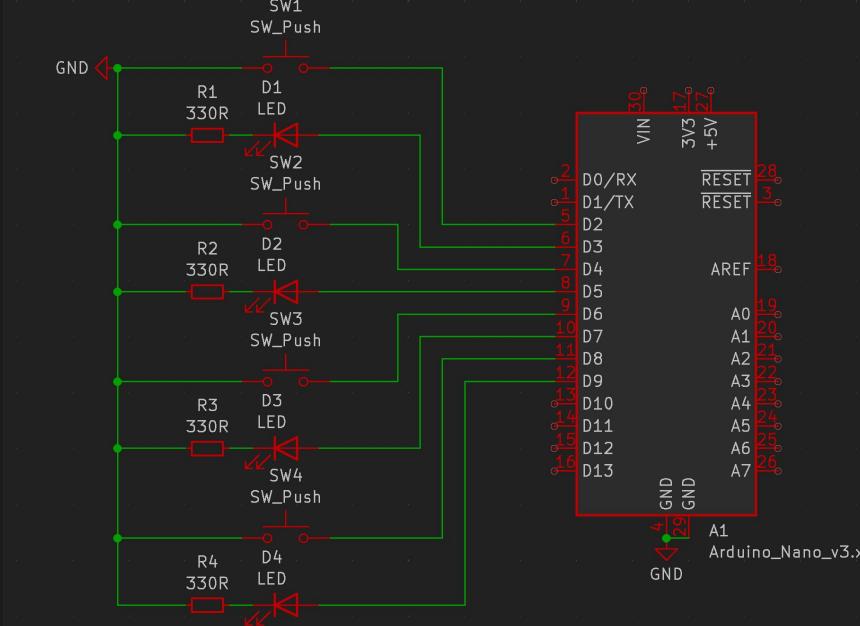
- You are now ready to connect your schematic together
 - Use the Add Wire tool (W) or hover over the tiny circle on a component to draw wires when this symbol appears



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

Assign Footprints

Footprint Libraries:

- Inductor_SMD_Wurth
- Inductor_THT
- Inductor_Wurth
- Jumper
- LED_SMD
- LED_THT**
- Module
- Motors
- MountingEquipment
- MountingHole
- Mounting_Wurth
- Nettie
- OptoDevice
- Oscillator
- Package_BGA

Footprint Filters:

Symbol	Footprint Assignments	Filtered Footprints
1 A1 -	Arduino_Nano_v3.x : Module:Arduino_Nano	26 LED_THT:LED_D3.0mm_Horizontal_06.35mm_Z6.0mm
2 D1 -	LED : LED_THT:LED_D5.0mm	27 LED_THT:LED_D3.0mm_Horizontal_06.35mm_Z10.0mm
3 D2 -	LED :	28 LED_THT:LED_D3.0mm_IRBlack
4 D3 -	LED :	29 LED_THT:LED_D3.0mm_IRGrey
5 D4 -	LED :	30 LED_THT:LED_D4.0mm
6 R1 -	330R :	31 LED_THT:LED_D5.0mm
7 R2 -	330R :	32 LED_THT:LED_D5.0mm_Clear
8 R3 -	330R :	33 LED_THT:LED_D5.0mm_FlatTop
9 R4 -	330R :	34 LED_THT:LED_D5.0mm_Horizontal_01.27mm_Z3.0mm
10 SW1 -	SW_Push :	35 LED_THT:LED_D5.0mm_Horizontal_01.27mm_Z3.0mm_Clear
11 SW2 -	SW_Push :	36 LED_THT:LED_D5.0mm_Horizontal_01.27mm_Z3.0mm_IRBlack
12 SW3 -	SW_Push :	37 LED_THT:LED_D5.0mm_Horizontal_01.27mm_Z3.0mm_IRGrey
13 SW4 -	SW_Push :	38 LED_THT:LED_D5.0mm_Horizontal_01.27mm_Z9.0mm

Filtered by Pin Count (2), Library (LED_THT): 72 matching footprints

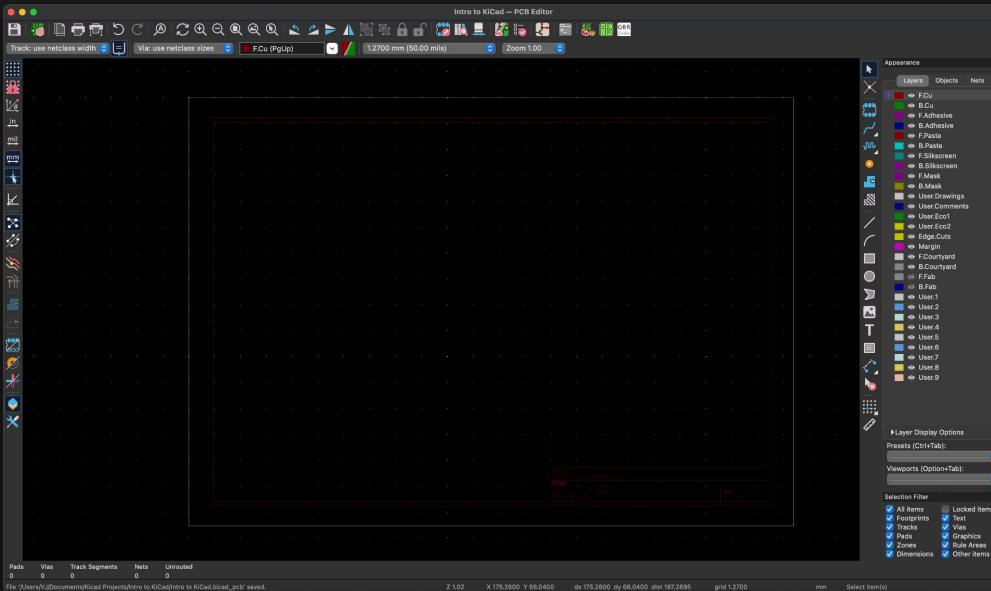
Library location: unknown

Buttons: Apply, Save Schematic & Continue | Cancel | OK

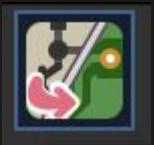
Creating your first PCB



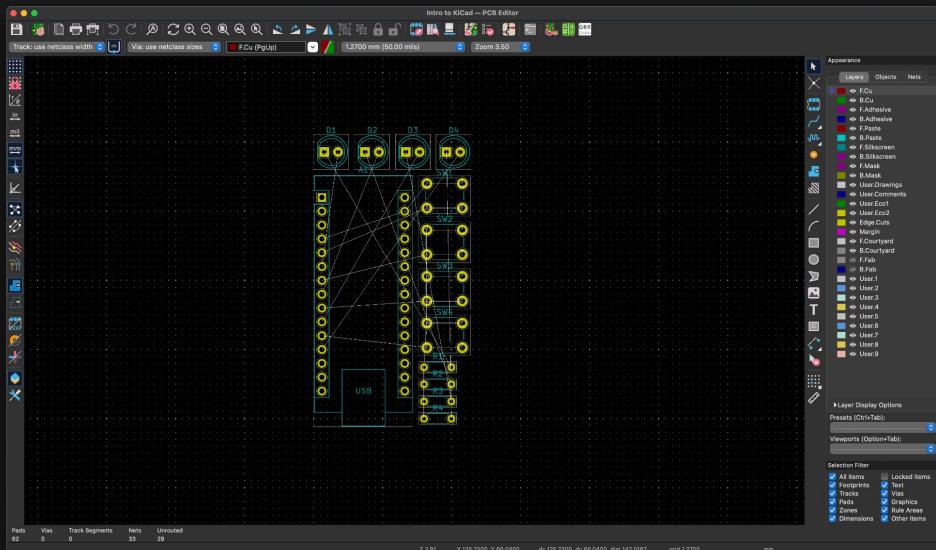
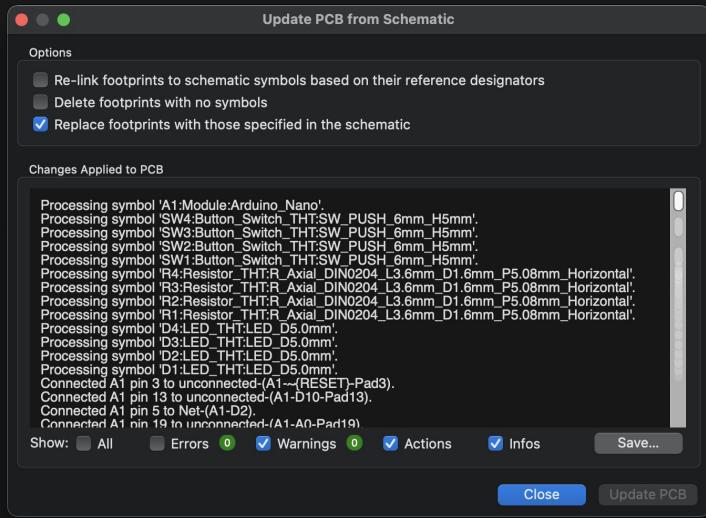
- Click the switch to PCB Editor button in the top toolbar to open the PCB Editor



Creating your first PCB

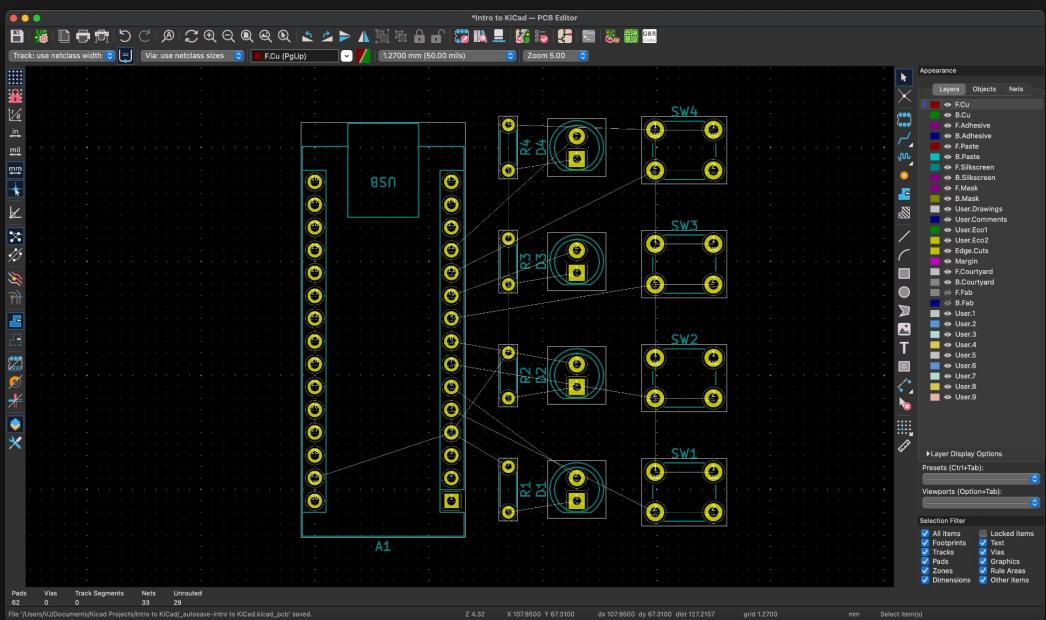


- Click the update PCB from Schematic button in the top toolbar to add all the assigned footprints from the schematic to the PCB



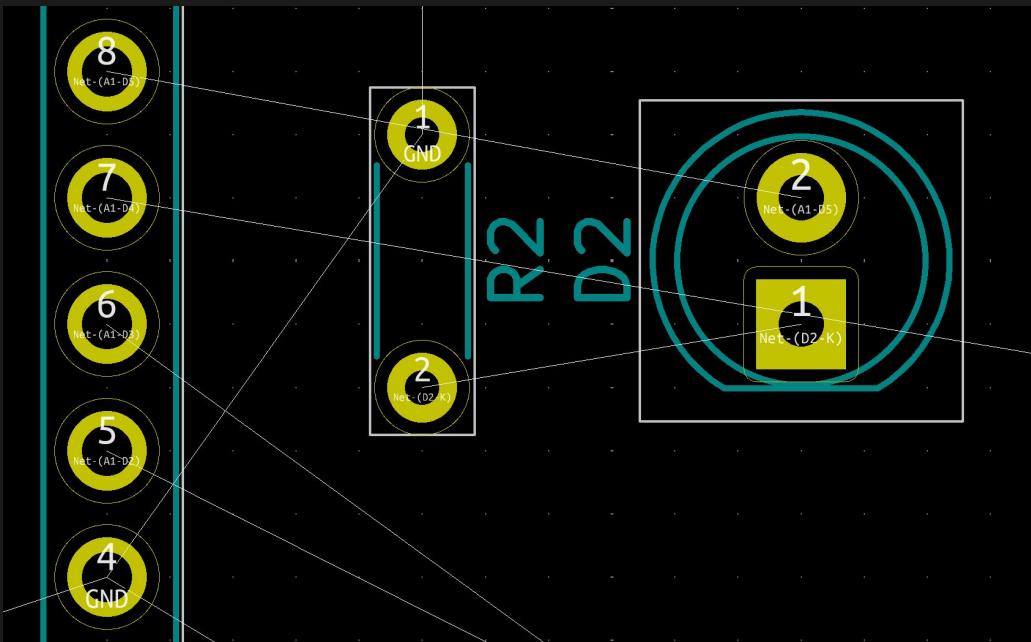
Arranging your first PCB

- 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

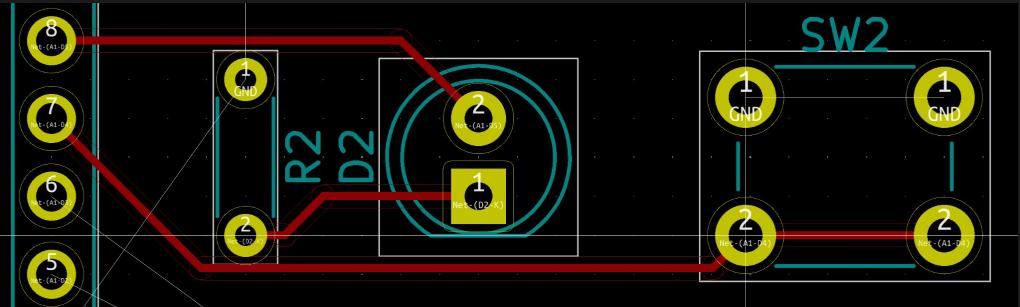
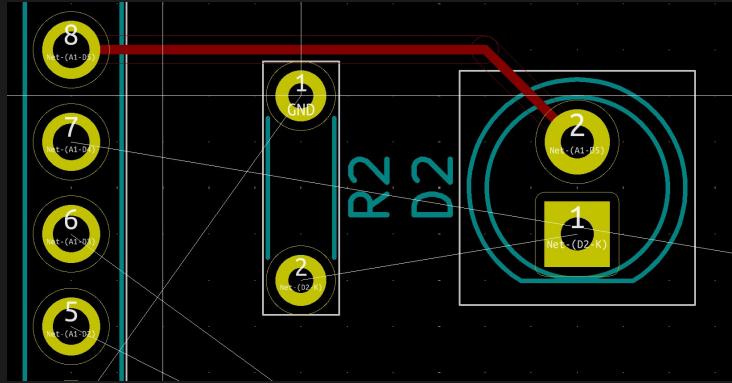
- All the floating lines connecting the footprints together is known as a rats nest
- These lines correspond to your schematic so you know where to place wires



Wiring your first PCB

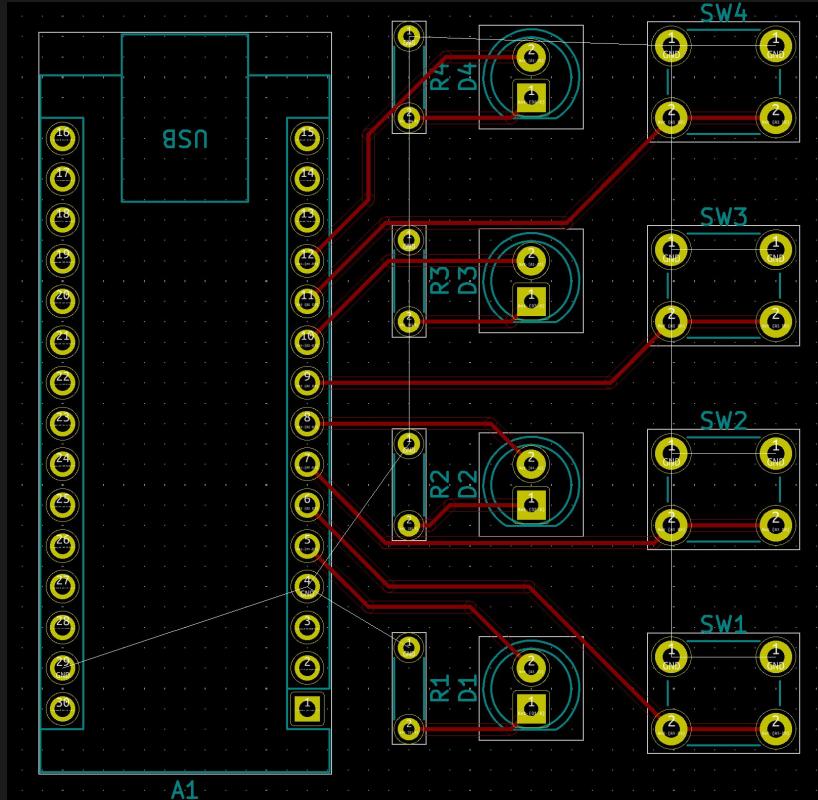


- Use the Route Tracks tool (X) in the right toolbar to place wires
- Click a pad with a rats nest to start routing it
- As you route tracks you can use the autoroute shortcut (F) to help speed up placement
- Save power and ground nets for last, you can hide them in the Nets tab on the right



Sizing your first PCB

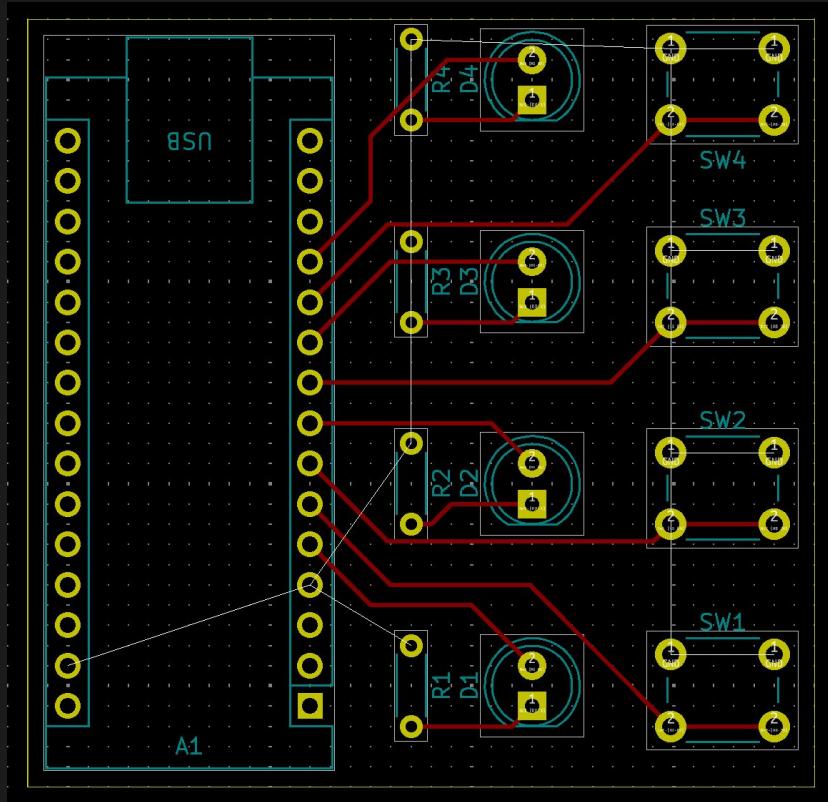
- With most of the wiring now complete it's important to make a board outline



Sizing your first PCB

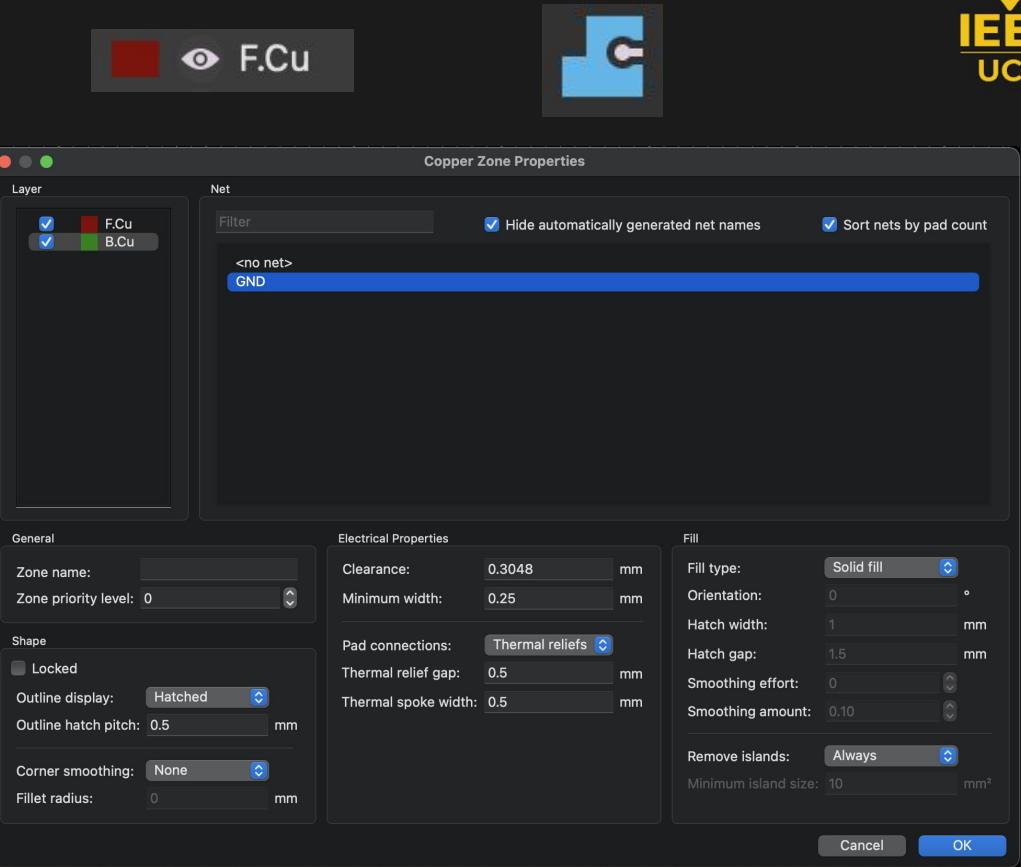


- First select the Edge.Cuts layer from the Layers tab on the right
- In that layer use the Draw Rectangle tool to draw an outline around your footprints



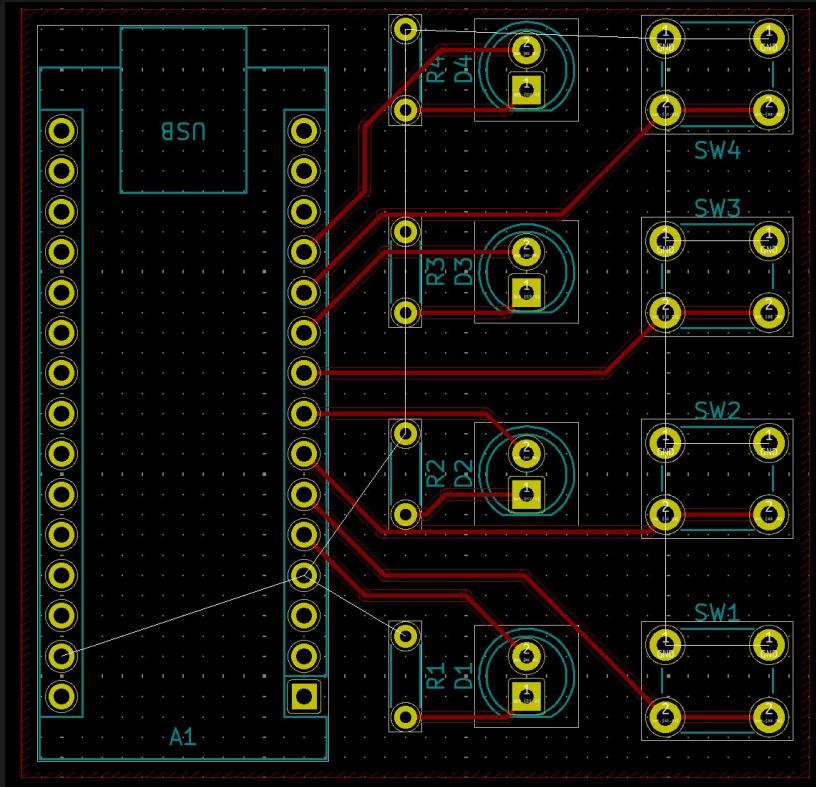
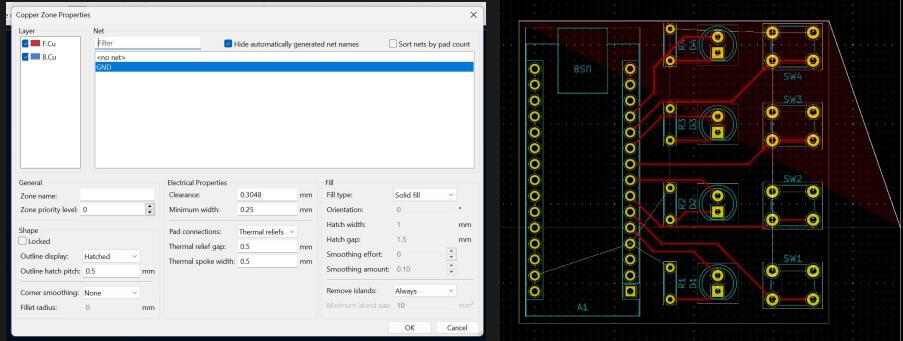
Wiring your first PCB

- 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



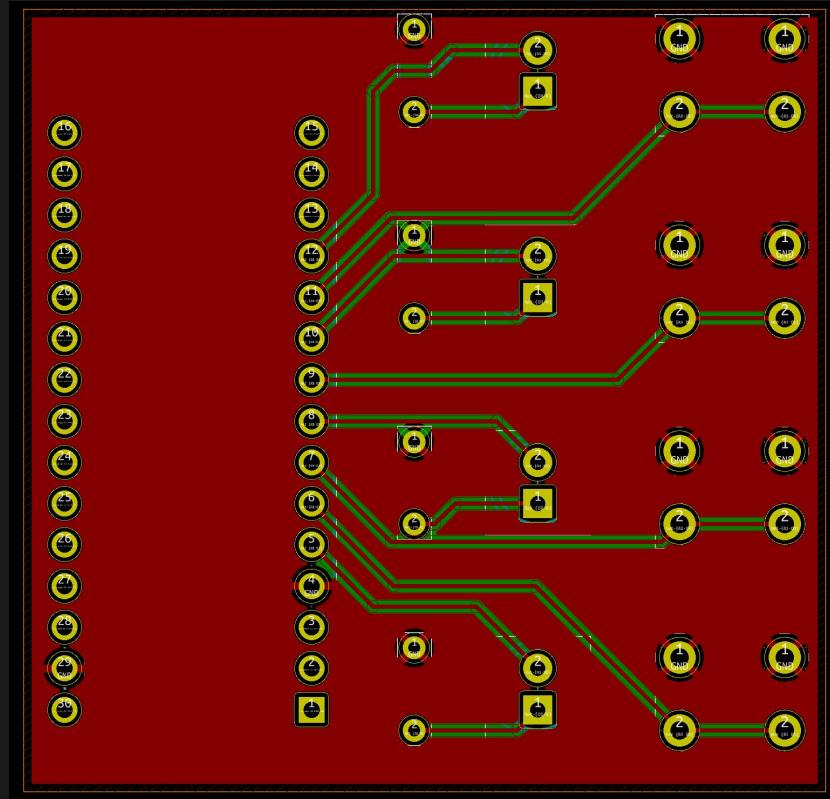
Wiring your first PCB

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



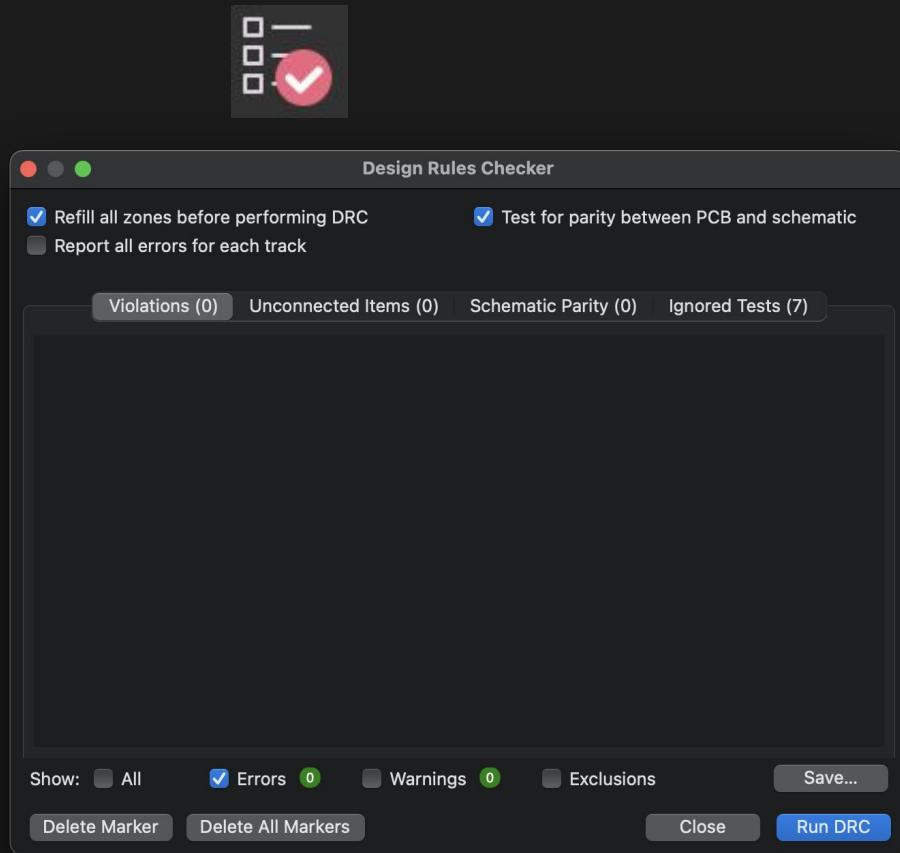
Wiring your first PCB

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



Verifying your first PCB

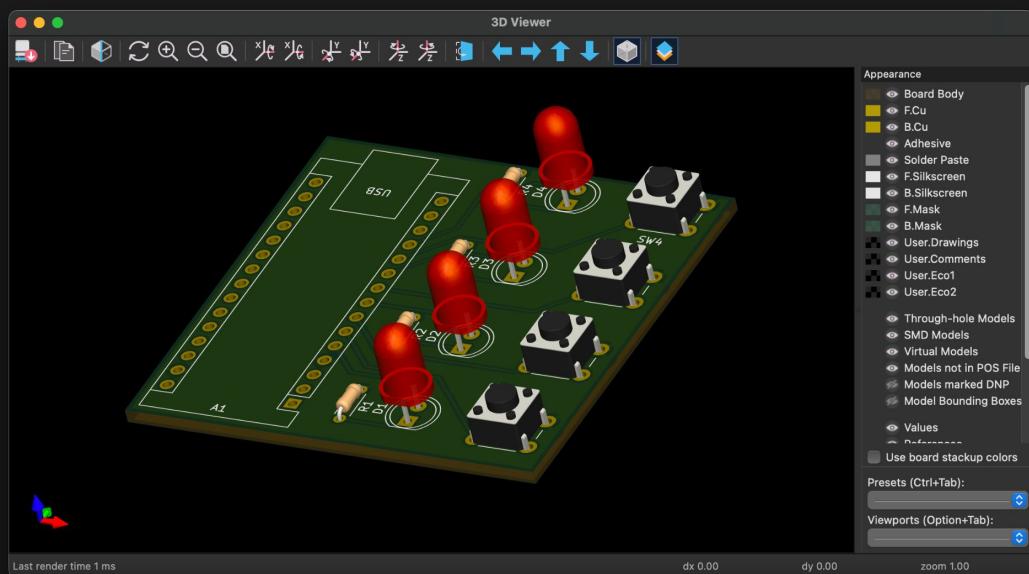
- Use the Design Rules Checker in the top toolbar to check for any errors in your design
- If no errors are present then your first PCB is complete



Viewing your first PCB

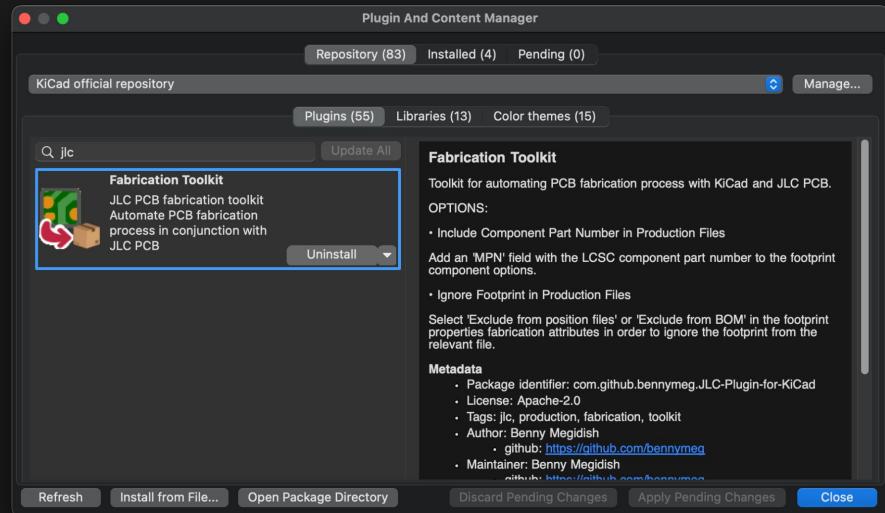
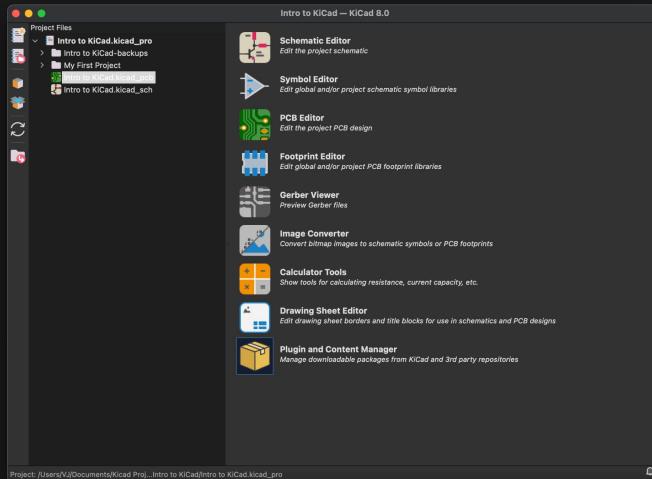


- In the PCB Editor use the 3D Viewer tool (Alt+3) in the top toolbar to see a preview of your PCB



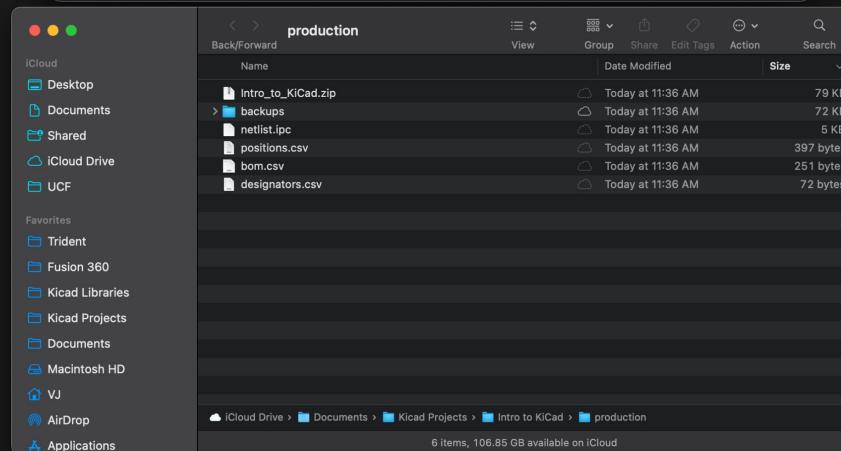
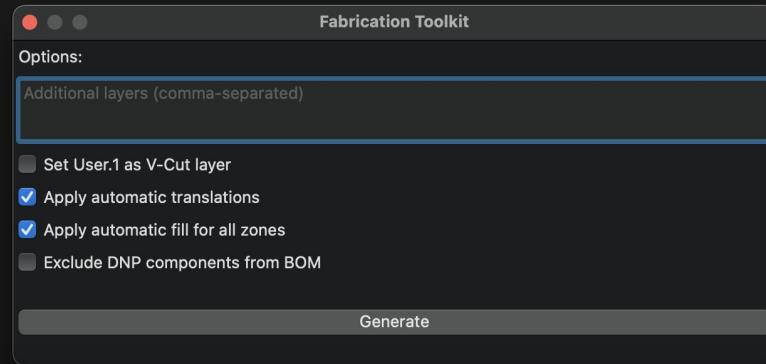
Exporting your first PCB

- In the Project Manager window click Plugin and Content Manager
- Search for `jlc` and install Fabrication Toolkit



Exporting your first PCB

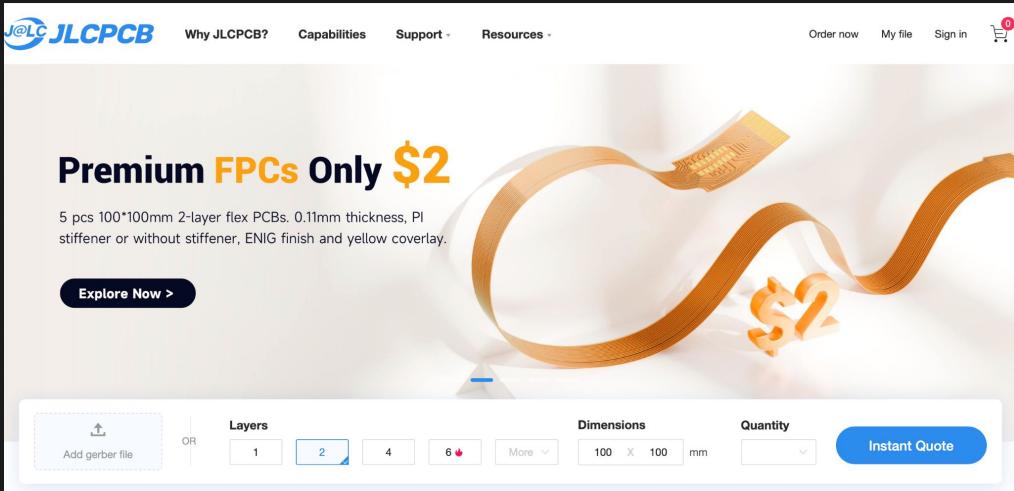
- In the PCB Editor click the Fabrication Toolkit icon in the top toolbar and click generate
- Once complete the folder with the production files will open automatically



Ordering your first PCB

 Intro_to_KiCad.zip

- Navigate to
<https://jlpcb.com>
- Drag & Drop the zip file in the production folder onto the Add gerber file box in the website



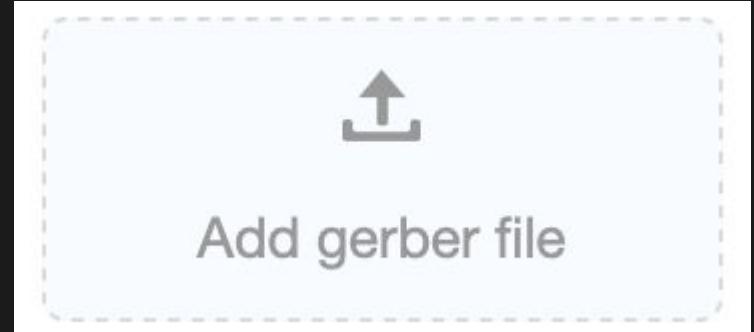
J@LC JLCPCB Why JLCPCB? Capabilities Support - Resources - Order now My file Sign in 0

Premium FPCs Only \$2

5 pcs 100*100mm 2-layer flex PCBs. 0.11mm thickness, PI stiffener or without stiffener, ENIG finish and yellow overlay.

Explore Now >

Add gerber file OR Layers 1 2 4 6 More Dimensions 100 x 100 mm Quantity Instant Quote



Ordering your first PCB

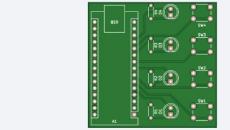
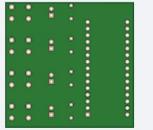
- Congratulations! You can now order your first PCB

 **Intro_to_KiCad.zip**

USD
[Order now](#)
[My file](#)
[Sign in](#)
[Cart](#)

JLCPCB

Standard PCB/PCBA Advanced PCB/PCBA SMT-Stencil 3D Printing/CNC

[Back to Upload File](#) Detected 2 layer board of 48.26x49.53mm(1.9x1.95 inches). [Gerber Viewer](#)

Base Material: FR-4 Flex Aluminum Copper Core Rogers PTFE Teflon

Layers: 2 4 6 8 10 12 14 16 Select

Dimensions:

PCB Qty:

Product Type: Industrial/Consumer electronics Aerospace Medical

PCB Specifications

Different Design: 1 2 3 4 5

Delivery Format: Single PCB Panel by Customer Panel by JLCPCB

PCB Thickness: 0.4 0.6 0.8 1.0 1.2 1.6 2.0

PCB Color: Green Purple Red Yellow Blue White Black

Silkscreen: White

Surface Finish: HASL(with lead) LeadFree HASL ENIG

[High-spec Options](#)

Charge Details

Special Offer: \$2.00

Via Covering: \$0.00

Surface Finish: \$0.00

Build Time:

PCB: 2 days \$0.00
 24 hours \$7.30
 24 hours **PCBA Only** \$0.00

Calculated Price: \$4.00- **\$2.00**

Additional charges may apply for [special cases](#)

[SAVE TO CART](#)

Shipping Estimate: \$1.52

Global Standard Direct Line 8-13 business days

Weight: 0.16kg

Coupons: [View your coupons](#)



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 - Fine	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!