

Scan to Sign-In!

Insert Check-In QR

Week 6

PCB Design Intro

Month Date Year

Agenda

1 Design

2 Schematic

3 PCB

IEEE@UCI

Who are we?

We are a student branch in the global IEEE, a professional organization with a mission to advance technology for humanity.

Where?

Our lab is in ICS 225

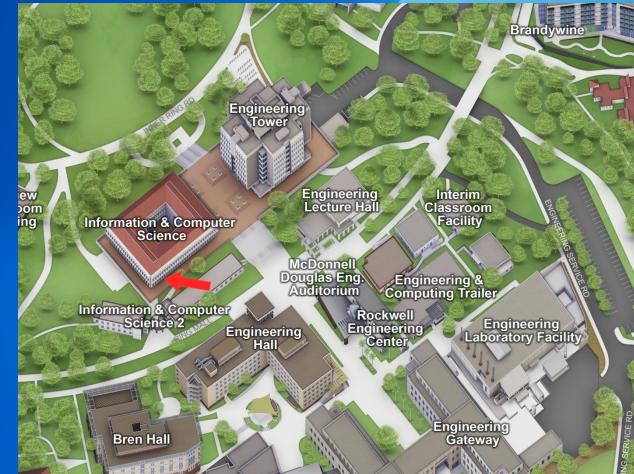
What do we offer?

- Workshops
- Micromouse
- Open Project Space
- Socials
- Fam System



The IEEE Room

- Check `# room-status` to see if the room is open
- 3D printers
- Soldering Stations
- Textbook Library
- ICS 225
 - Giant Post-it “IEEE” on the window



Click [here](#) for directions

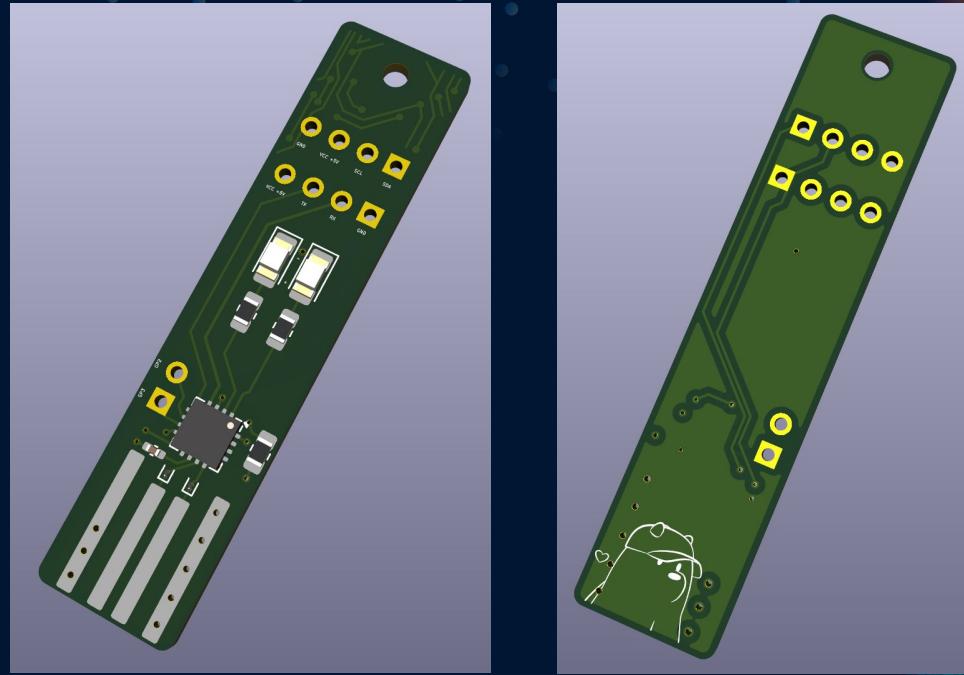
Design Overview

USB Keychain PCB

We are going to be making a USB to I2C convertor board in a USB stick form factor.

PCB topics covered in slides

- SMD
- THT
- Vias
- Polygon Pour
- Mounting Holes
- Layer Stackup
- Differential Pairs
- Impedance Matched Traces



Design Inspiration



Sage Wu Light Up Keychain



IEEE-Eta Kappa Nu, Zeta Omega
University of California, Irvine

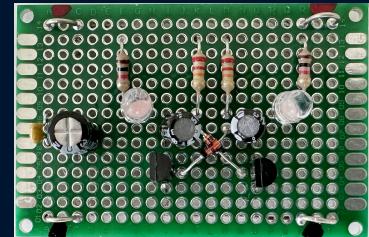
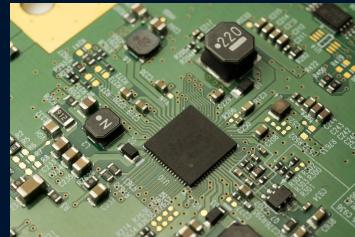


General PCB Design Process

1. Component Research
2. Create Schematic
3. PCB Layout
4. Gerber Files
5. Order and Test PCB



Why Design a PCB?



- Allows for more complex designs over breadboarding or on perfboard
- More reliable and stable connections between parts
- Much more compact

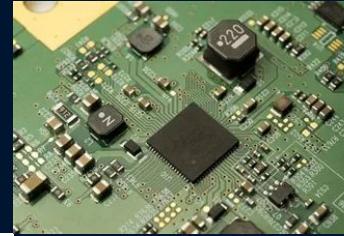


Terminology

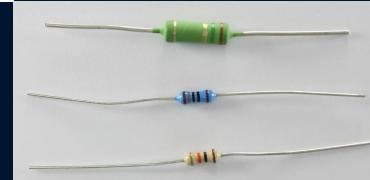
- **PCB:** Printed circuit board
- **SMD:** Surface Mount Device
- **THT:** Through Hole Technology



Ex. SMD Resistors



Ex. PCB



Ex. THT Resistors



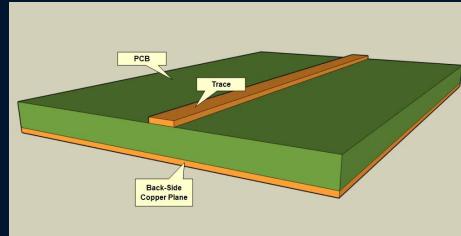
Terminology Continued

- **Symbol**
- **Footprint**
- **Trace/Track**
- **Via**
- **Net**

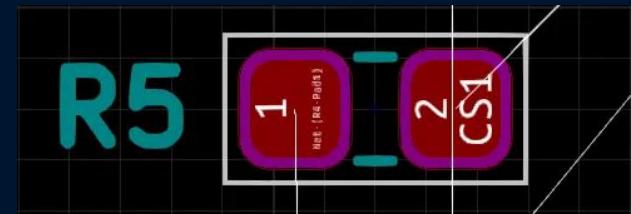
Resistor Symbol



PCB Trace



SMD Resistor Footprint

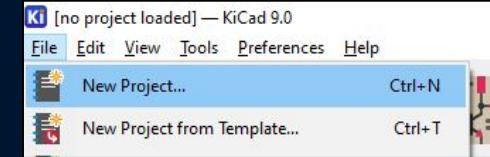


PCB Via



Setup KiCad and Create New Project

1. Install KiCad: <https://www.kicad.org/download/>
2. Go to Files -> New Project -> name your project and save
3. Click on Schematic Editor



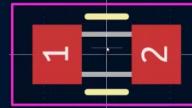
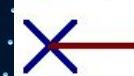
Creating Schematic



IEEE-Eta Kappa Nu, Zeta Omega
University of California, Irvine

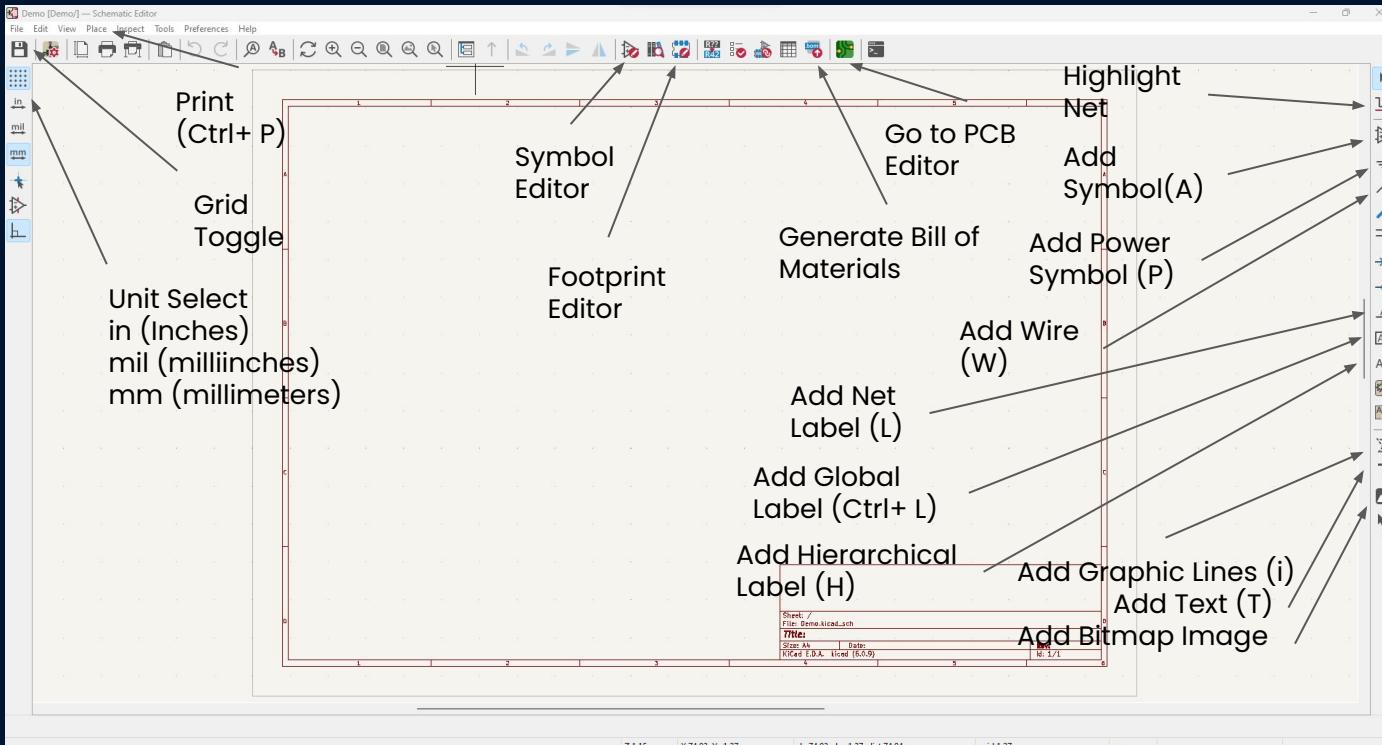


Schematic Design Process

1. Import components
2. Create and place all symbols 
3. Assign footprints to symbols 
4. Connect all symbol pins to one another as desired
5. Mark all pins that are not connected with NC flag (No Connect) 
6. Label pins and Nets as needed 



The Schematic Editor



Important Shortcuts

M - Move an Object

R - Rotate an Object

X - Mirror an Object horizontally

Y - Mirror an object vertically

W - Add Wire

B - Add Bus

Z - Add Bus Entry

E - Edit Properties

V - Edit Value

D - Open Datasheet (If there is a datasheet linked to part)

G - Drag (Interactive Move)

A - Add Symbol

P - Add Power Symbol

Ctrl+D - Duplicate

L - Add Label

H - Add Hierarchical Label

Ctrl+L - Add Global Label

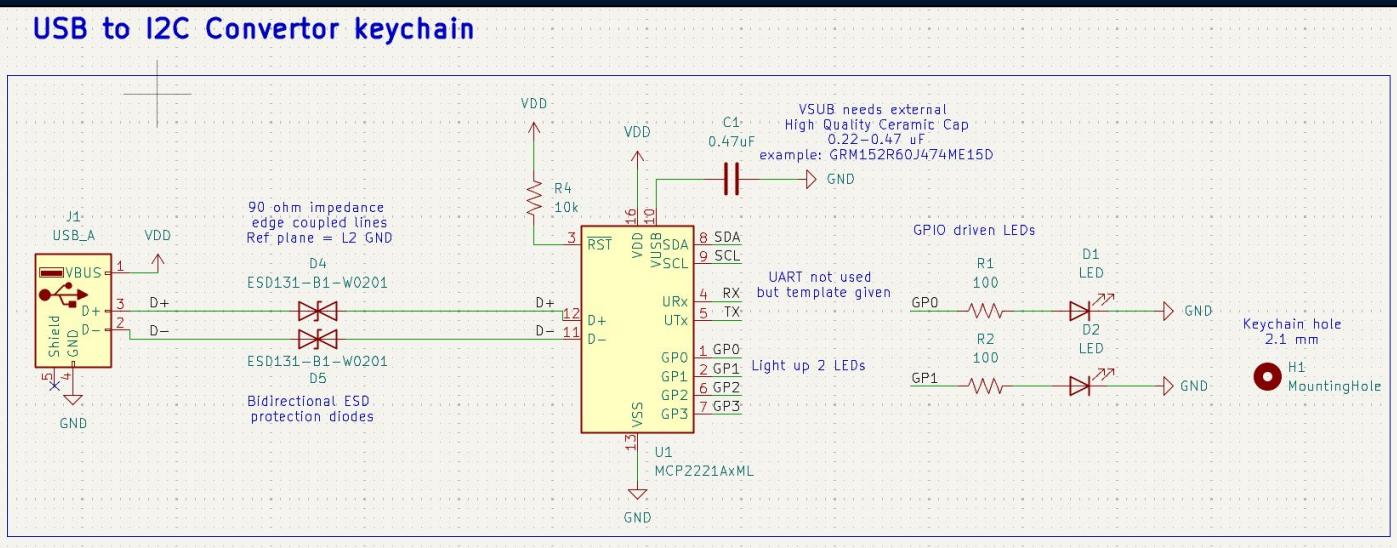


Schematic Layout

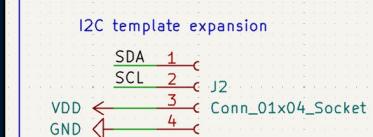
Here is what a completed schematic could look like.

There is creative freedom in the selection of
expansion headers and exact LED form factor (THT, SMD, etc).

USB to I2C Convertor keychain



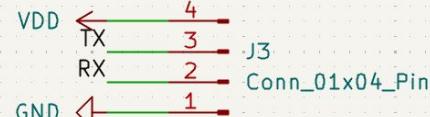
Expansion Ports (2.54mm pitch used)



2 GPIOs connector template



UART Pin Holes Template



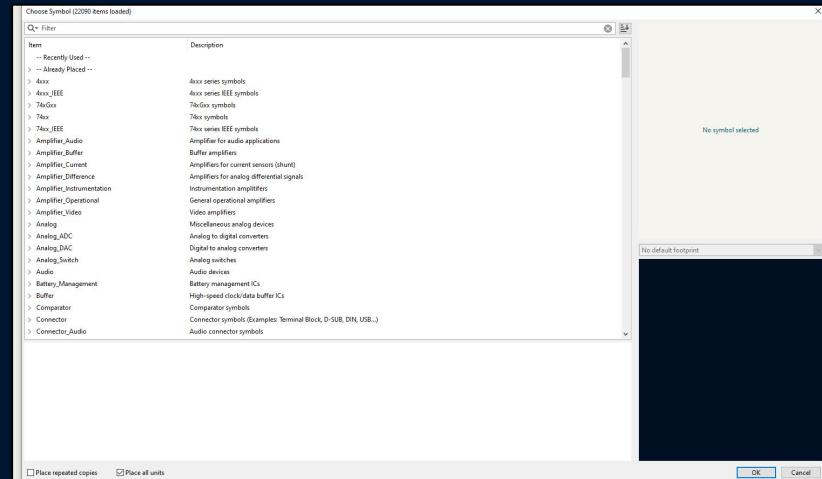
Adding Symbols

First add all symbols needed
 Parts list on next slide



1. Press **A** and search in the box
2. Once you find the correct item press **OK**
3. Then place the symbol down somewhere in sheet
4. To reuse any symbol, select and duplicate with **Ctrl+d**

Note: Edit a component to desired properties before duplicating component



List of Components (symbol placement)

1.	USB A	USB A
2.	USB Converter Chip	MCP2221AxML - Datasheet
3.	Resistors	2 x 220 Ohm, 1 x 10k ohm
4.	LEDs	SMD 1206
5.	Capacitors	0.47 uF
6.	4 Pin header	Conn_01x04_Pin
7.	2 Pin header	Conn_01x02_Pin
8.	4 Pin socket	Conn_01x04_Socket
9.	ESD Protection Diodes	ESD131-B1-W0201
10.	Mounting Hole	MountingHole



Informal BOM

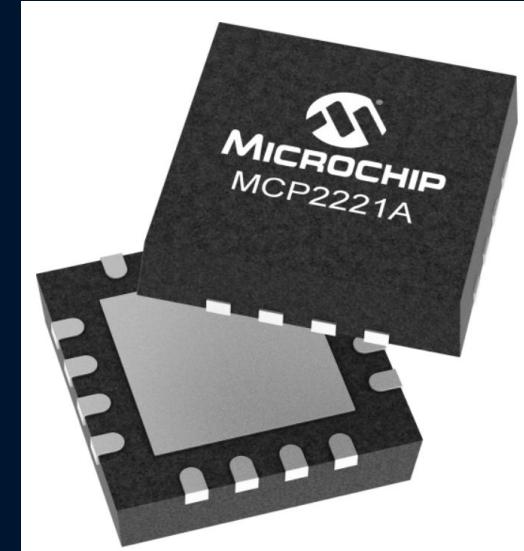
	A	B	C	D	E	F	G	H	I
1	Reference	Qty	Value	Description: Footprint				Datasheet	
2	C1	1	0.47uF	Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder				~	
3	D1,D2	2	LED	LED_SMD:LED_1206_3216Metric_Pad1.42x1.75mm_HandSolder				~	
4	D4,D5	2	ESD131-B1-W0201	Diode_SMD:Infineon_SG-WLL-2-3_0.58x0.28_P0.36mm				https://www.infineon.com	
5	H1	1	MountingHole	Excluded MountingHole:MountingHole_2.1mm				~	
6	J1	1	USB_A	USB A Conn PCB Leads:USB_A_Conn_Leads				~	
7	J2	1	Conn_01x04_Socket	Connector_PinSocket_2.54mm:PinSocket_1x04_P2.54mm_Vertical				~	
8	J3	1	Conn_01x04_Pin	Connector_PinHeader_2.54mm:PinHeader_1x04_P2.54mm_Vertical				~	
9	J4	1	Conn_01x02_Pin	Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical				~	
10	R1,R2	2	100	Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder				~	
11	R4	1	10k	Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder				~	
12	U1	1	MCP2221AxML	Package_DFN_QFN:QFN-16-1EP_4x4mm_P0.65mm_EP2.5x2.5mm				http://ww1.microchip.com	
13									



USB to I₂C Converter Chip

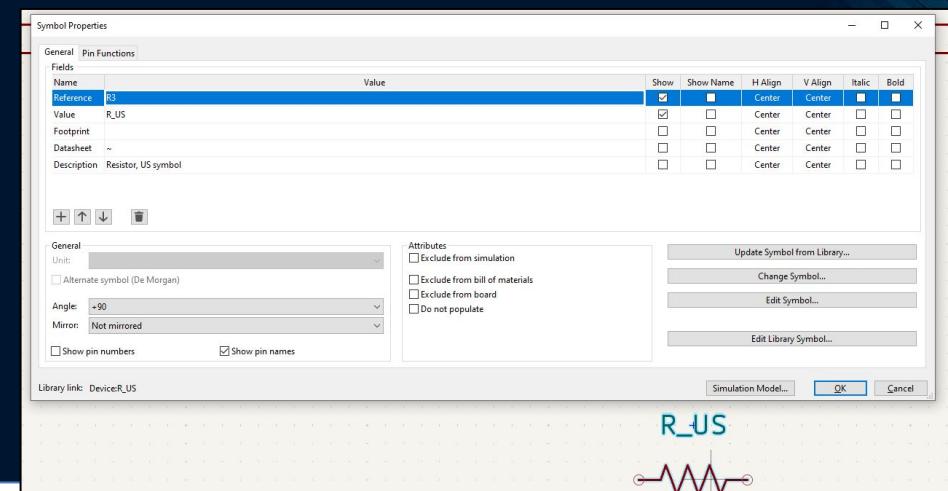
MCP2221AxML Features

- Implements USB Protocol Composite Device
- Human Interface Device (HID) for Both I₂C Communication and Control
- The Device Runs as an I₂C Master.
The Data to Write/Read On the I₂C Bus is Conveyed By the USB Interface
- Four General Purpose Input/Output Pins



Set Component Values

1. Hover cursor over a symbol and press E (or double click component) to edit the properties of that symbol/component, a menu like the one below will show:
2. Input desired value of resistance, capacitance, inductance (e.g. 200)
(How does one determine this value?)
 - The value scheme is as follows:
{base value} {scale} {unit}
 - Scale (Power of 10):
p: -12, n: -9, u: -6, m: -3, k: 3, M: 6, G: 9
 - Units of common discretes:
Capacitors: F, Inductors: H,
Resistors: None



Manually Importing Symbol and Footprint

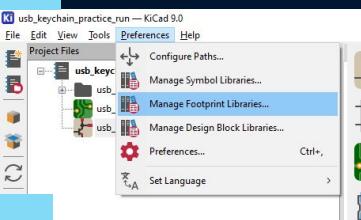
- In general, sometimes there's no available footprint and/or symbol in library, so we will have to search up, download, and import them to kicad.
- Footprint library search resources:
 - SnapMagic: <https://www.snapeda.com/home/>
 - UltraLibrarian: <https://www.ultralibrarian.com/>
- General steps:
 1. Download symbol and footprint, select for KiCad
 2. Import library to kicad
 3. Add the symbol and footprint to your project
- We will import the USB A PCB Connector manual (but use existing symbol)



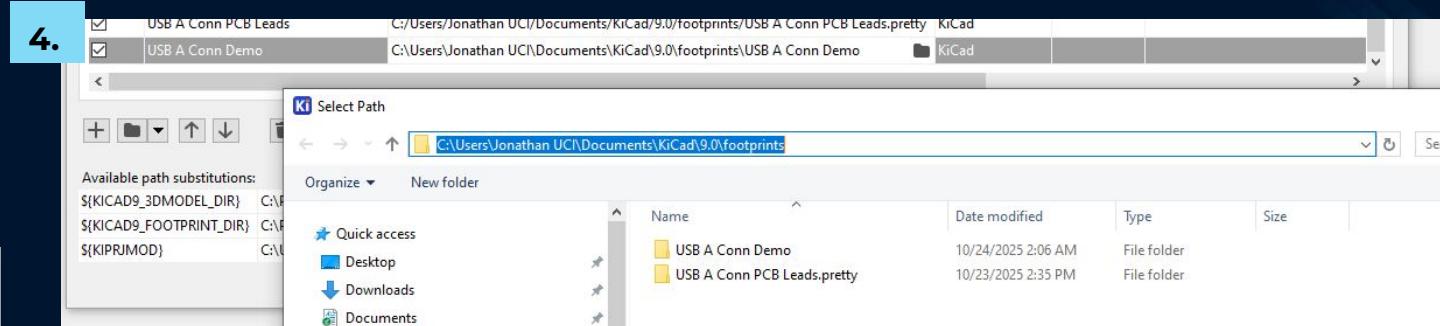
Manually Importing USB A Pads Footprint

- Go to Home Screen, then Preferences → Manage Footprint Libraries → hit plus sign at bottom left “Add empty row to table” → name it **USB A Conn**
- Add the library path at “C:\Users\{username}\Documents\KiCad\9.0\footprints\USB_A_Conn\”
- Move downloaded footprint file into this \USB_A_Conn\ folder

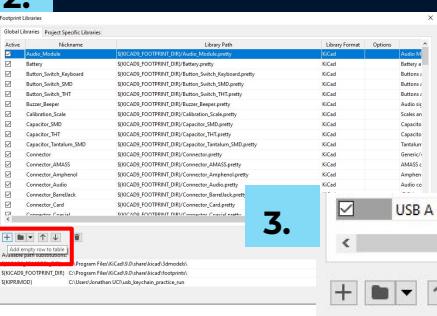
1.



4.



2.



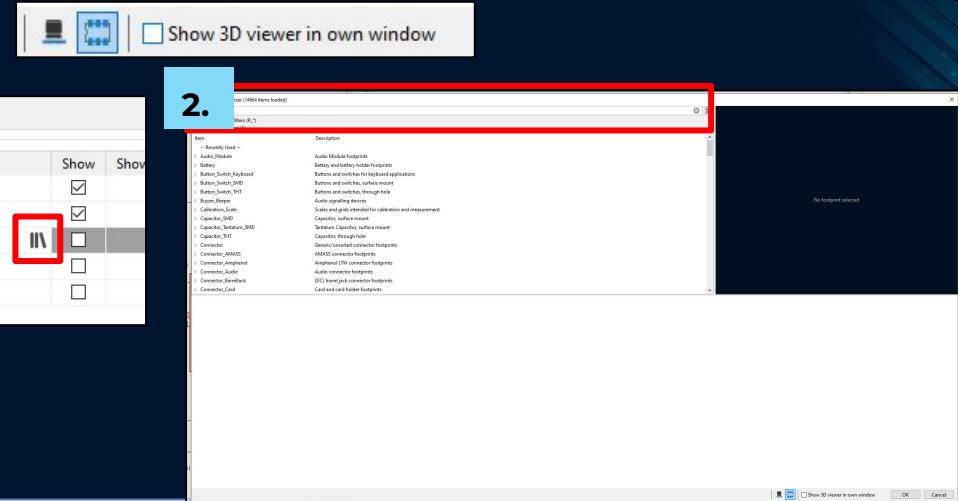
3.

5.



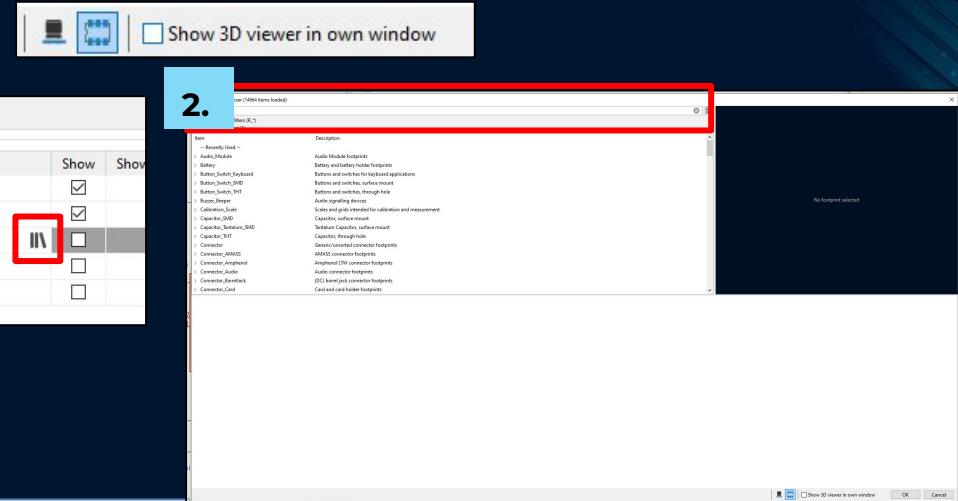
Set Component Footprints (properties menu)

1. Hover over a symbol and press E (or double click component) to edit the properties of that symbol/component, (same menu as last slide will show)
2. Click on books icon in “Footprint” field
3. Search for desired part via browsing or part number (demo)
4. Reference side by side 3D model viewer



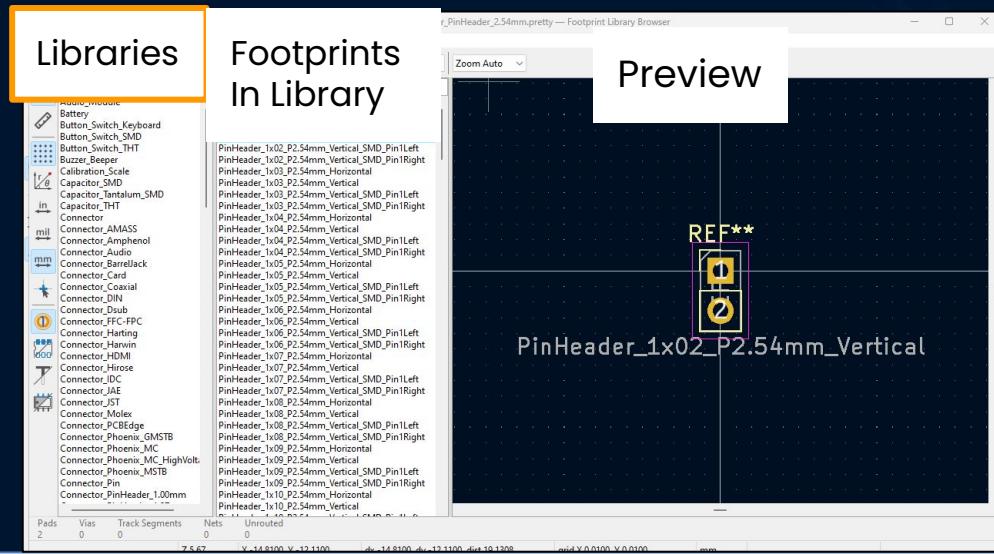
1.

General Pin Functions	
Fields	
Name	Value
Reference	R3
Value	R_US
Footprint	
Datasheet	~
Description	Resistor, US symbol



Set Component Footprints (Continued)

- Resistors, capacitors, inductors, and other components come in different shapes and sizes
 - Can be SMD or THT
 - Pin headers have different pitches
 - 2.54mm pitch is common
- Each **library** covers a type of component



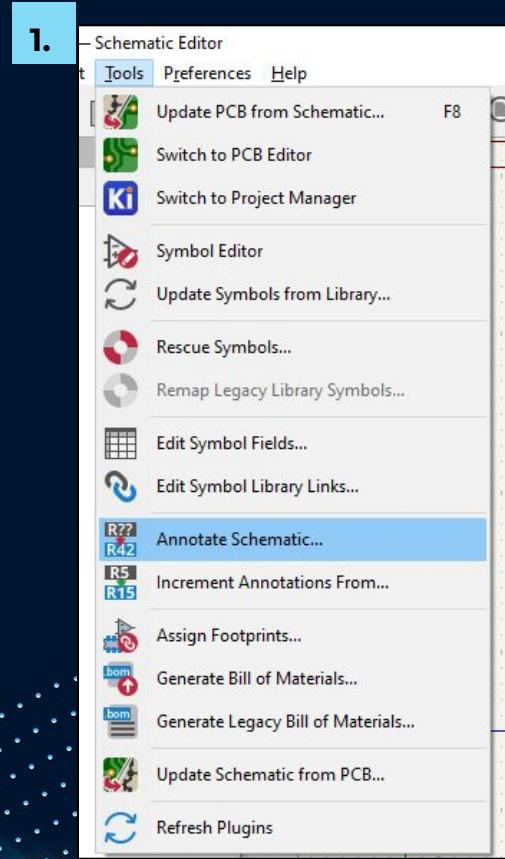
Set Component Footprints (Continued)

- Resistors:
R_0805_2012Metric_Pad1.20x1.40mm_HandSolder
- Capacitor:
C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
- LED (LED_D3.0mm):
LED_1206_3216Metric_Pad1.42x1.75mm_HandSolder
- ESD Protection Diodes:
Infineon_SG-WLL-2-3_0.58x0.28_P0.36mm
- Mounting Hole:
MountingHole_2.1mm
- USB to I2C Chip (MCP2221AxML):
QFN-16-1EP_4x4mm_P0.65mm_EP2.5x2.5mm
- Pin Headers (2.54mm pitch):
 - 4 pin vertical
PinHeader_1x04_P2.54mm_Vertical
 - 2 pin vertical
PinHeader_1x02_P2.54mm_Vertical
- Pin Socket (2.54 mm pitch):
 - 4 pin vertical
PinSocket_1x04_P2.54mm_Vertical
- **USB-A PCB Connector Pads:**
 - Footprint file also in [HKN Discord Server](#) #resources channel
USB_A_Conn_Leads (Download this [footprint file](#))



Annotating Symbols' Reference Designators

- There will be multiple of one type of component
 - Example: 2 identical resistors and 2 identical LEDs
 - How to know which wire goes to which components?
- Reference Designators provide unique labels to each component on a schematic
- Tools → Annotate Schematic → Click “Annotate”



PCB Layout



IEEE-Eta Kappa Nu, Zeta Omega
University of California, Irvine

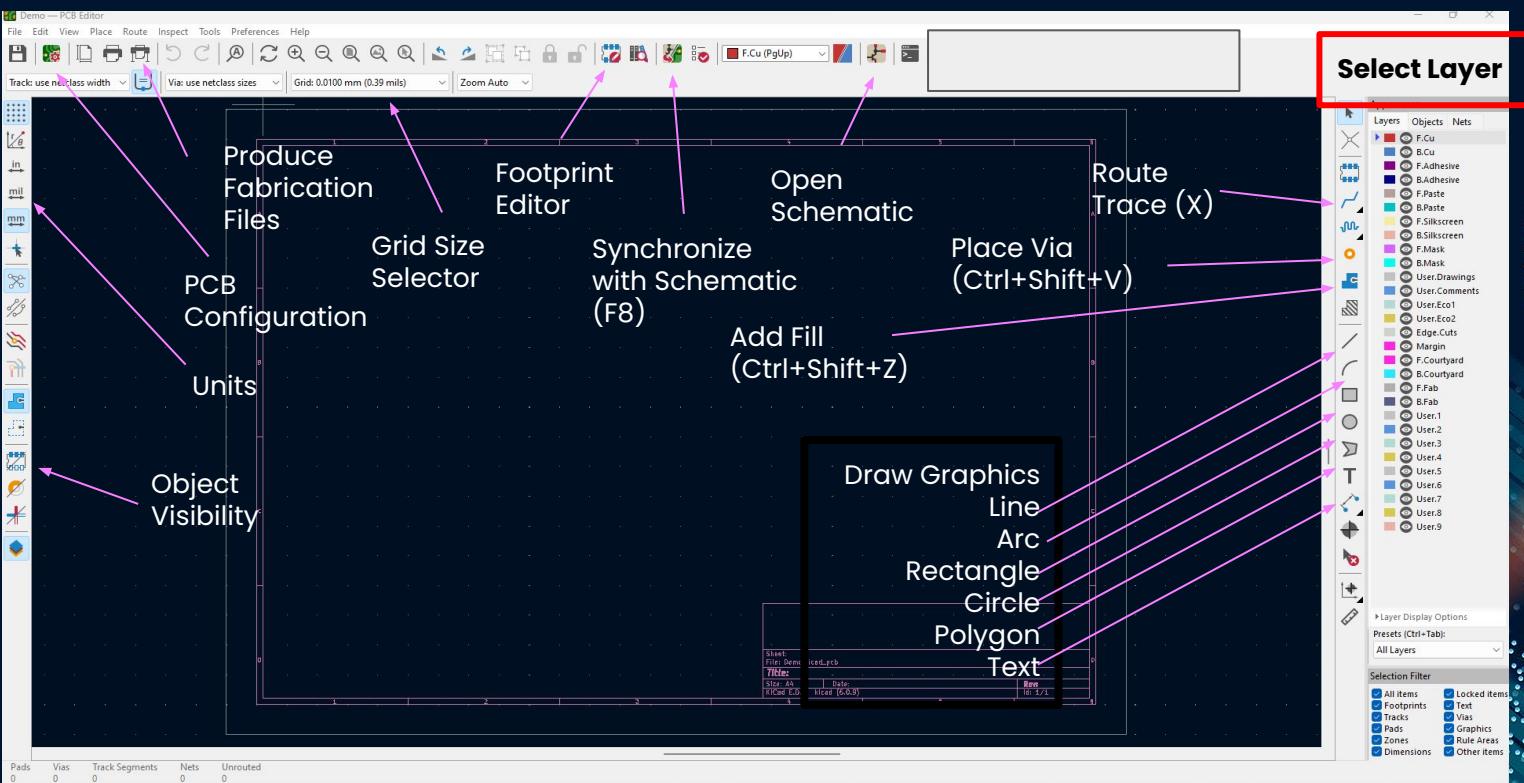


PCB Design Process

1. Define board outline
2. Place components where they should be
3. Decide layer stackup
4. Route traces and use vias as needed
5. Add any necessary fills (e.g. solid copper ground pour)



The PCB Editor



IEEE-Eta Kappa Nu, Zeta Omega
 University of California, Irvine



Important PCB Editor Shortcuts

- **R** - Rotate an Object
- **X** - Create a Trace
- **V** - Switch Layers (Between Top and Bottom Copper), Adds Via if routing a trace
- **M** - Move an Object on its own
- **E** - Edit Properties
- **F** - Flip (Moves Component from the Top to Bottom or Vice Versa)
- **D** - Drag (Interactive move)



PCB Layers (2 layer board)

- **F/B.Cu** - Top/Bottom Copper layer, where the traces are
- **F/B.Courtyard** - Defines the area around parts which there can not be other parts on the Top/Bottom
- **F/B.Silkscreen** - Graphics that are on top of the solder masks, usually white, usually for text
- **Edge.Cuts** - Board Outline
- **F/B.Paste** - Solder Paste On the Top/Bottom (The exposed metal there will be solder not copper)



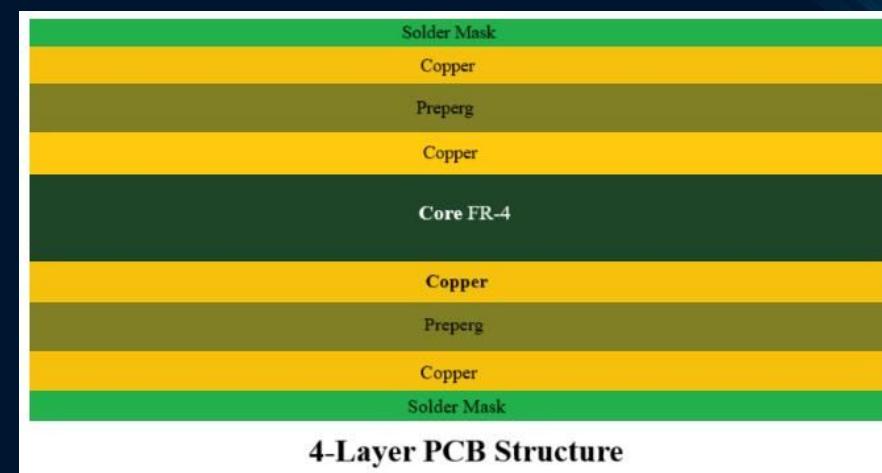
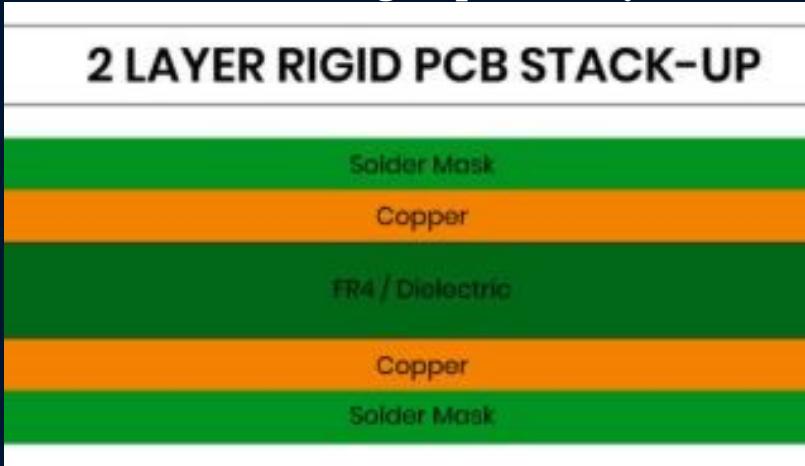
PCB Layers Continued

- **F/B.Mask** – Solder Mask on the Top/Bottom (Where we want to have an opening for the copper, e.g. for pads)
- **F/B.Fab** – Shows the component name and value for components on the Top/Bottom (For Manufacturing)
- **F/B.Adhesive** – Placement Adhesive mapping for the Top/Bottom (Not Important for us)
- **User.Drawings/Comments** – Layers to add dimensions or comments for documentation



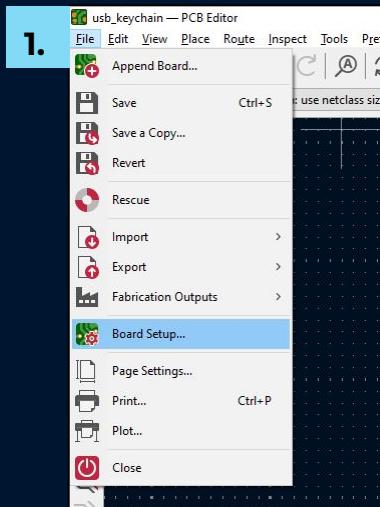
Deciding Layer Stackup

- 2 layer boards are simple and cost effective (used for today's project)
- 4 layer boards help with complex designs (e.g. RF, High Speed, Mixed Signal)
 - USB is often a high speed signal where impedance matching matters
 - PCBs can go up to 14 layers, and even more in some cases!

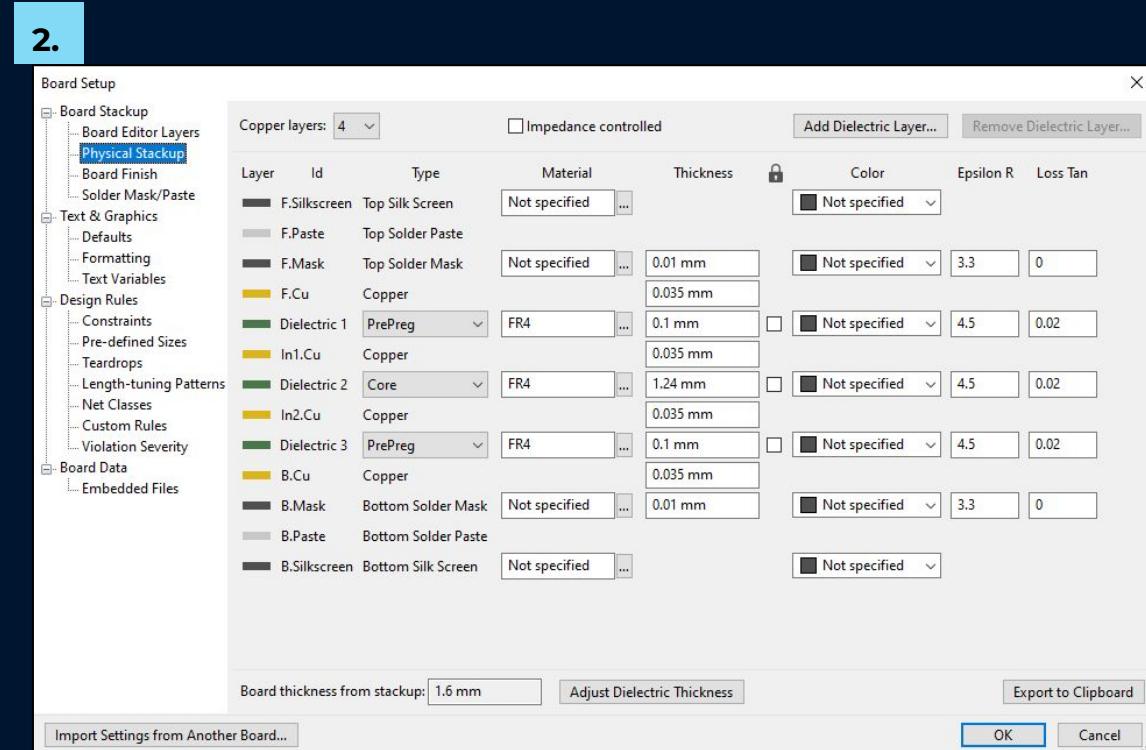


Configuring PCB Layer Stackup

- Go to: File → Board Setup → Physical Stackup
- We are using default 2 layer board for this design



2.



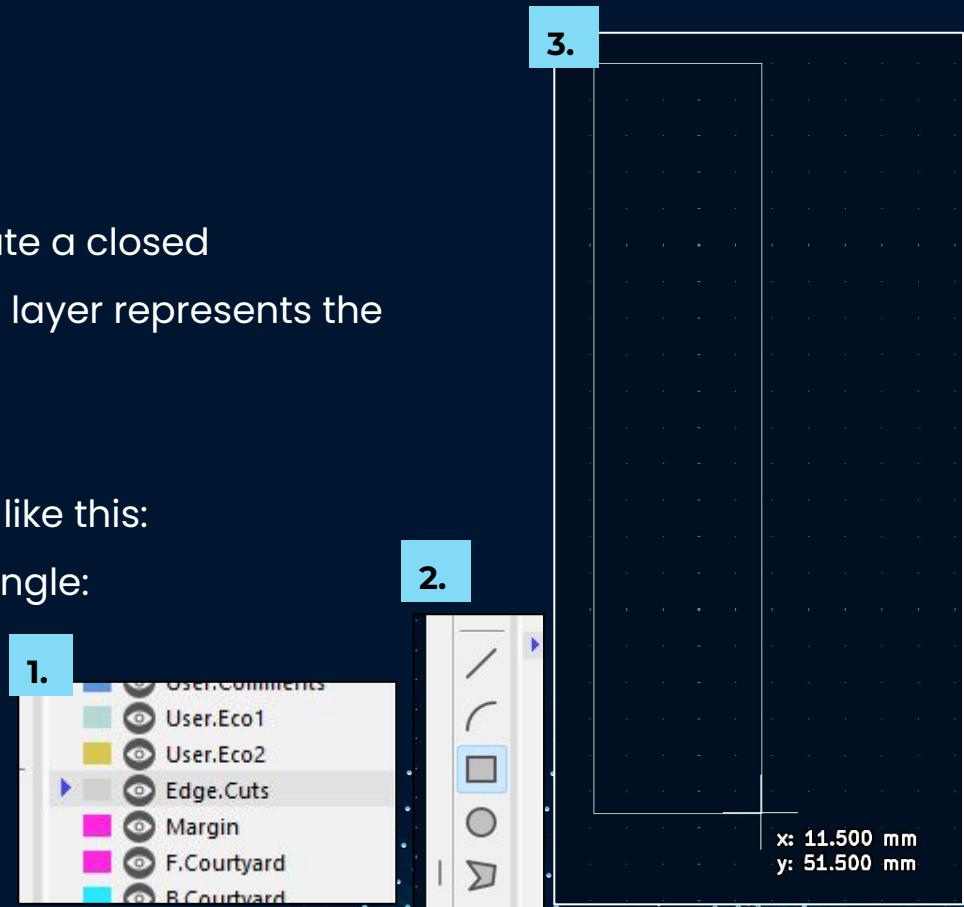
Layer	Id	Type	Material	Thickness	Color	Epsilon R	Loss Tan
	F.Silkscreen	Top Silk Screen	Not specified		Not specified	3.3	0
	F.Paste	Top Solder Paste	Not specified	0.01 mm	Not specified	4.5	0.02
	F.Mask	Top Solder Mask	Not specified	0.035 mm	Not specified	4.5	0.02
	F.Cu	Copper	FR4	0.1 mm	Not specified	4.5	0.02
	Dielectric 1	PrePreg	FR4	0.035 mm	Not specified	4.5	0.02
	In1.Cu	Copper	FR4	1.24 mm	Not specified	4.5	0.02
	Dielectric 2	Core	FR4	0.035 mm	Not specified	4.5	0.02
	In2.Cu	Copper	FR4	0.1 mm	Not specified	4.5	0.02
	Dielectric 3	PrePreg	FR4	0.035 mm	Not specified	4.5	0.02
	B.Cu	Copper	FR4	0.01 mm	Not specified	3.3	0
	B.Mask	Bottom Solder Mask	Not specified	0.035 mm	Not specified	4.5	0.02
	B.Paste	Bottom Solder Paste	Not specified	0.035 mm	Not specified	4.5	0.02
	B.Silkscreen	Bottom Silk Screen	Not specified	0.035 mm	Not specified	4.5	0.02

Board thickness from stackup: 1.6 mm Adjust Dielectric Thickness Import Settings from Another Board... Export to Clipboard OK Cancel



Creating Board Outline

- To define the board outline we need to create a closed graphical shape in the **Edge.Cuts** layer, this layer represents the outline/cut-out of the board.
- Any closed shape of any size works
- Select Edge.Cuts by clicking on it, it will look like this:
- Dimensions of this board's design is a rectangle:
 $W \times L \times H = 11.5\text{mm} \times 51\text{mm} \times 1.6\text{mm}$
- Select rectangle tool → click canvas
→drag dimensions→click to confirm shape



Curved Board Edges

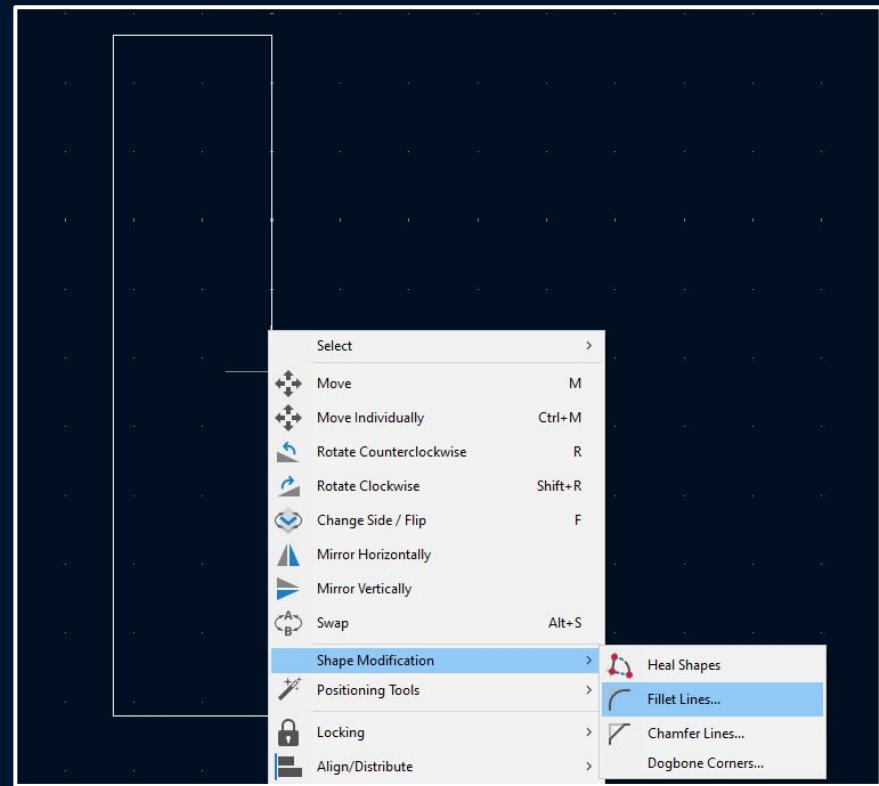
Instructions:

Draw Rectangle (board shape)

→ Select & Highlight Shape

→ Right Click → Shape Modification

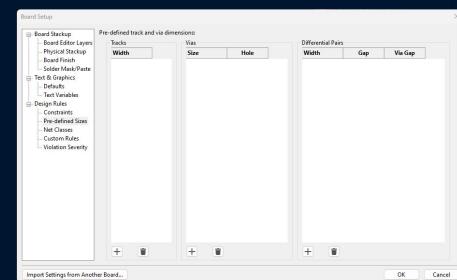
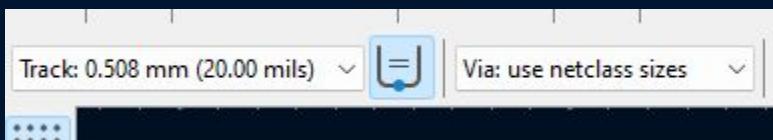
→ Fillet Lines → 1mm



Setting Up Traces and Vias

We want to set the size of vias and traces. The reason we do this is for resistance. The wider the trace the less resistive it is, same with vias. We want to make traces that carry any serious amount of current have as wide of a trace as possible. For data signals, the trace size doesn't matter very much as minimal current is carried.

We want click on the via size selection list or the track size collection list. From there we will click "edit predefined sizes"



We want to press the plus on the via and add a via size of "30 mils"

We want to make the hole "20 mils" We want to add 2 Track sizes, "30 mils" and "40 mils"

After press Ok

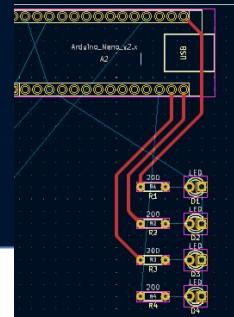


Laying Traces and Vias

Now we have the track sizes and via sizes set we can start drawing traces. To place a track we need to select a track size from the drop down menu mentioned in the slide before. We will use 40 mils for all power lines and 30 mils for all signal lines. We will be demonstrating laying a track for VCC so we will be using 40 mils and the 30 mils/20mils hole Via size. Unselect the icon between the track size and via size dropdowns

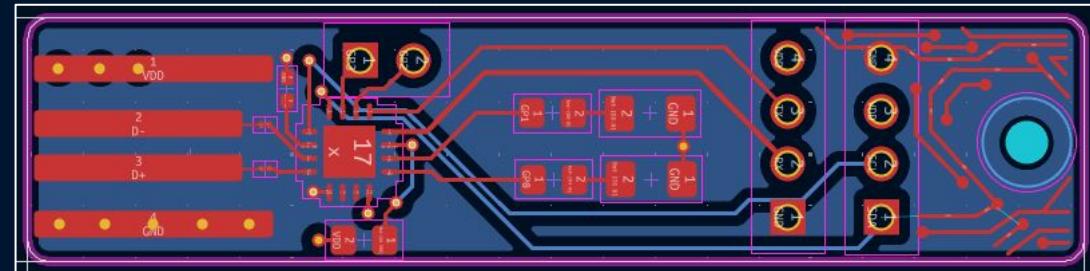


We will press **X** while hovering over a pin that we want to draw a track from. If we need to make a turn or pivot we will click in that pivot point and route from there. Do not wire ground, we will do that later. If you can't route wires press **V** while routing and you can use the other layer via a via. Click on the F.Cu layer in the selector and start routing



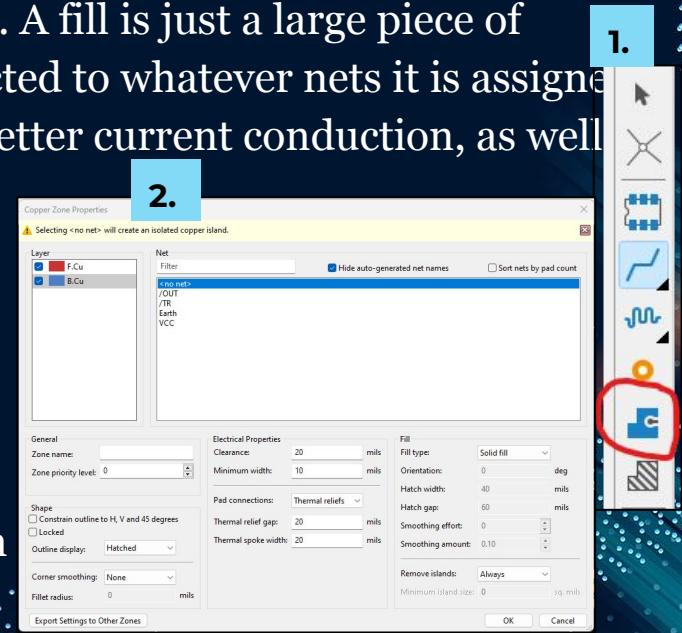
Routing Traces

- Hover a pad and press **X** to start a trace
- We want to try to route all of the traces on one layer (top copper layer)
- We also want to try to minimize the path from one pad to the next
- Here is an example of routing done for this particular circuit
- The only unrouted net is ground
- Cross check the datasheets and schematic
- Run DRC (Design Rule Check)



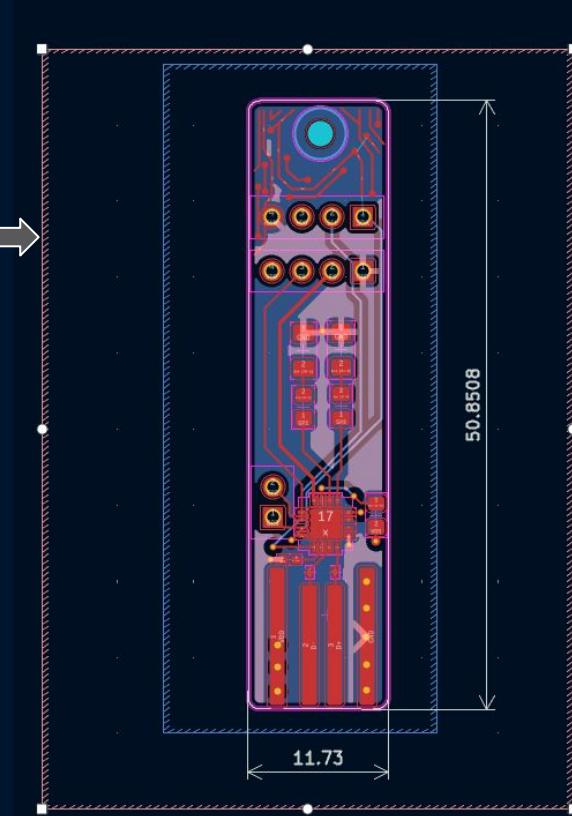
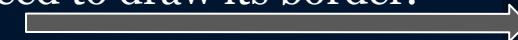
Adding Fills

- AKA ground pour
- We want to finally route ground, we want to do it via a fill. A fill is just a large piece of copper that fills all of the gaps in the design and is connected to whatever nets it is assigned to. These fills are done to stabilize the ground, allow for better current conduction, as well as allow for more heat to be dissipated.
- To add a fill we want to select this icon or use Ctrl+Shift+Z, we need to be on the F/B.Cu layers.
- Click a point outside of the border, you will get this menu
- We want to select the F and B layers
- We want to give it the Earth net as the assigned net
- The rest of the parameters are for a more advanced lesson
- Press OK and we will move on to drawing the fill



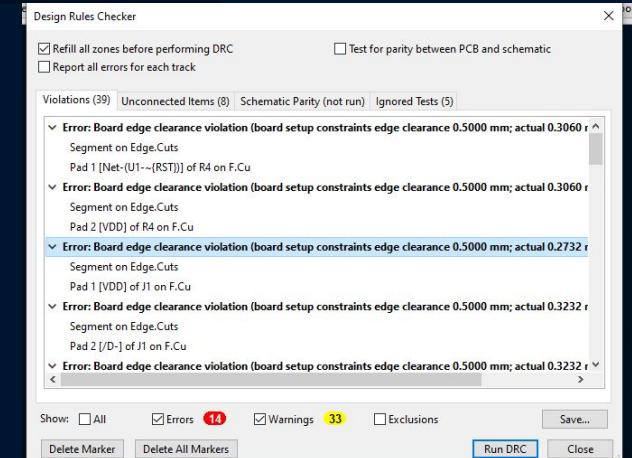
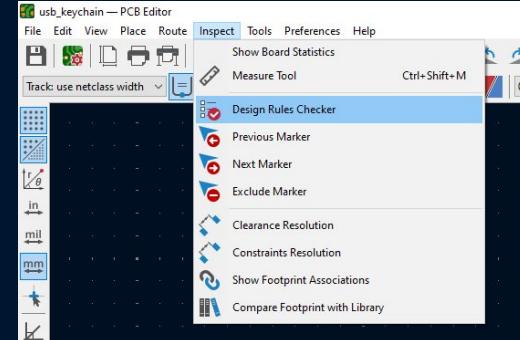
Finishing Fills

- Once we have a fill started we need to draw its border.
- Draw a closed rectangular shape outside of the perimeter enclosing whole board
- Press **B** to repour fill



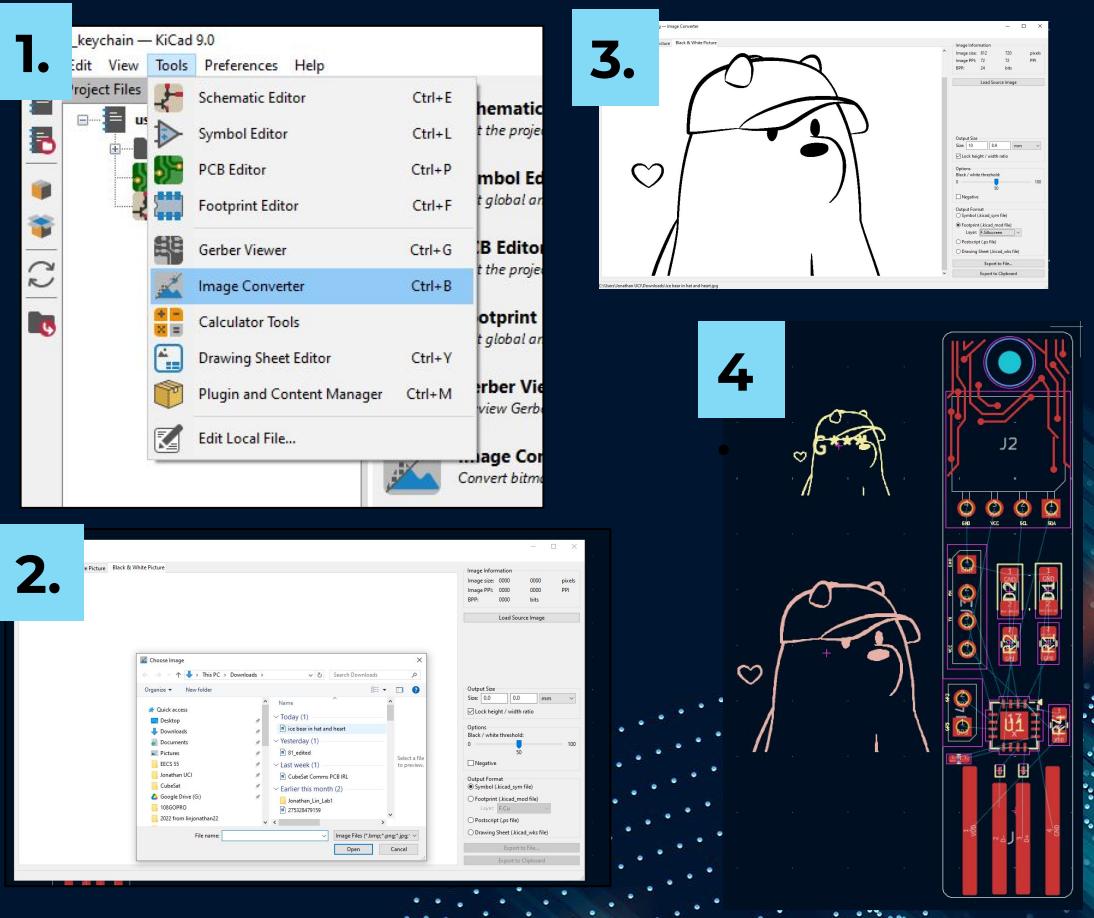
Finishing the PCB

- Ensure no more ratsnest
- Resize and organize silkscreen markers
- Run DRC



PCB Silkscreen Art

- Convert image to PCB art in KiCad
 Tools → Image Converter → Load Source Image
 Down Size image, output to front silkscreen layer, export to clipboard, paste to PCB



Any Questions?



IEEE-Eta Kappa Nu, Zeta Omega
University of California, Irvine



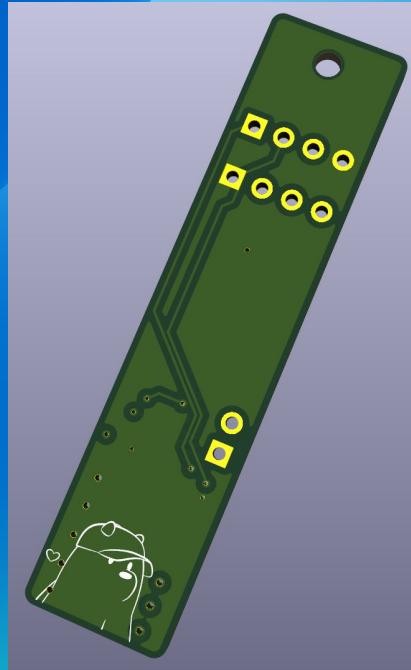
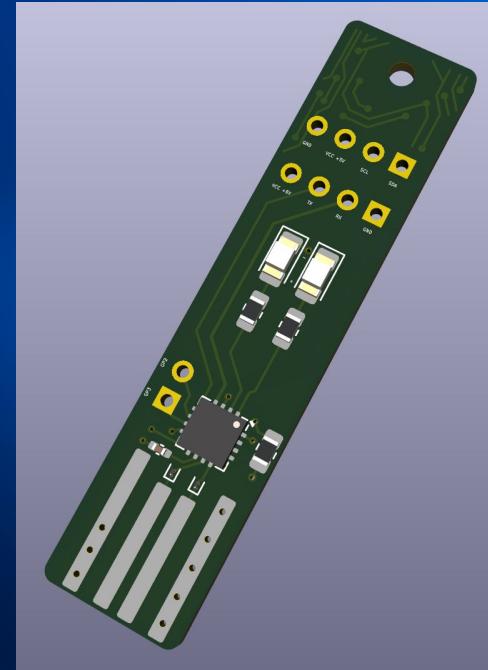
Design Overview

USB Keychain PCB

We are going to be making a USB to I2C convertor board in a USB stick form factor.

PCB topics covered in slides

- SMD
- THT
- Vias
- *Polygon Pour*
- *Mounting Holes*
- *Layer Stackup*
- *Differential Pairs*
- *Impedance Matched Traces*



Thanks!

Scan to Keep Up with Us!:



Forgot to Check In Today?:

Insert Check-In QR