

# Essentials of PCB Design

04: KiCad

# welcome back

## schedule

### A-Term

**Tue, Sep. 23**

Basics of PCBs  
6-7PM; AK 233

**Thur, Sep. 25**

Designing your Project  
6-7:30PM; AK 233

**Tue, Sep. 30**

Layout + Routing  
6-7:30PM; AK 233

**Thur, Oct. 2**

Working with KiCad  
6-7:30PM; AK 233

### B-Term

**Mon, Oct. 20 - Fri, Oct. 24**

**Fri, Oct. 24**

**Tue, Nov. 4 (tentative)**

Office Hours  
**TBD**

Boards Due  
**By 11:59PM**

Assembly  
**TBD**

# course updates

## lecture 04

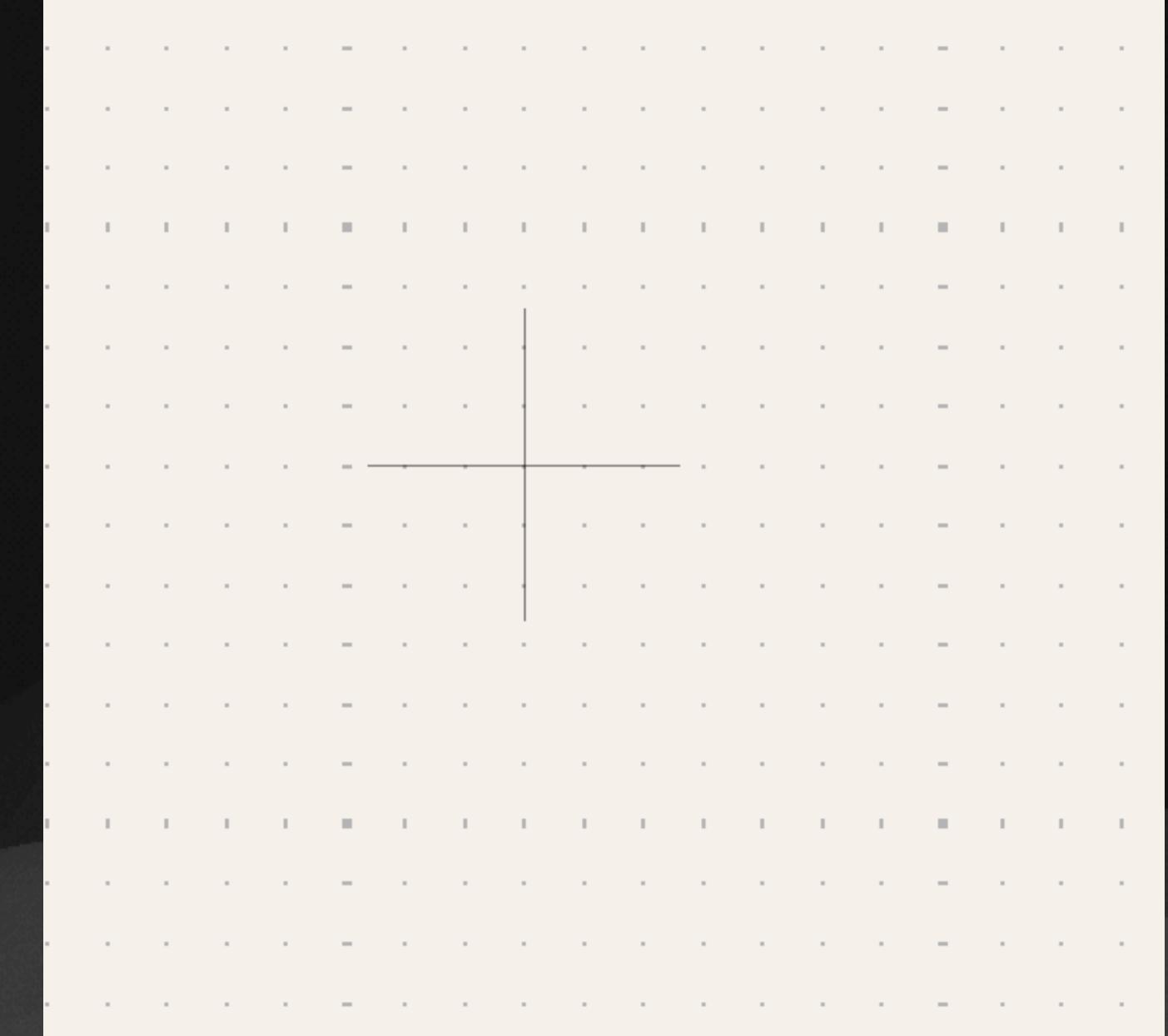
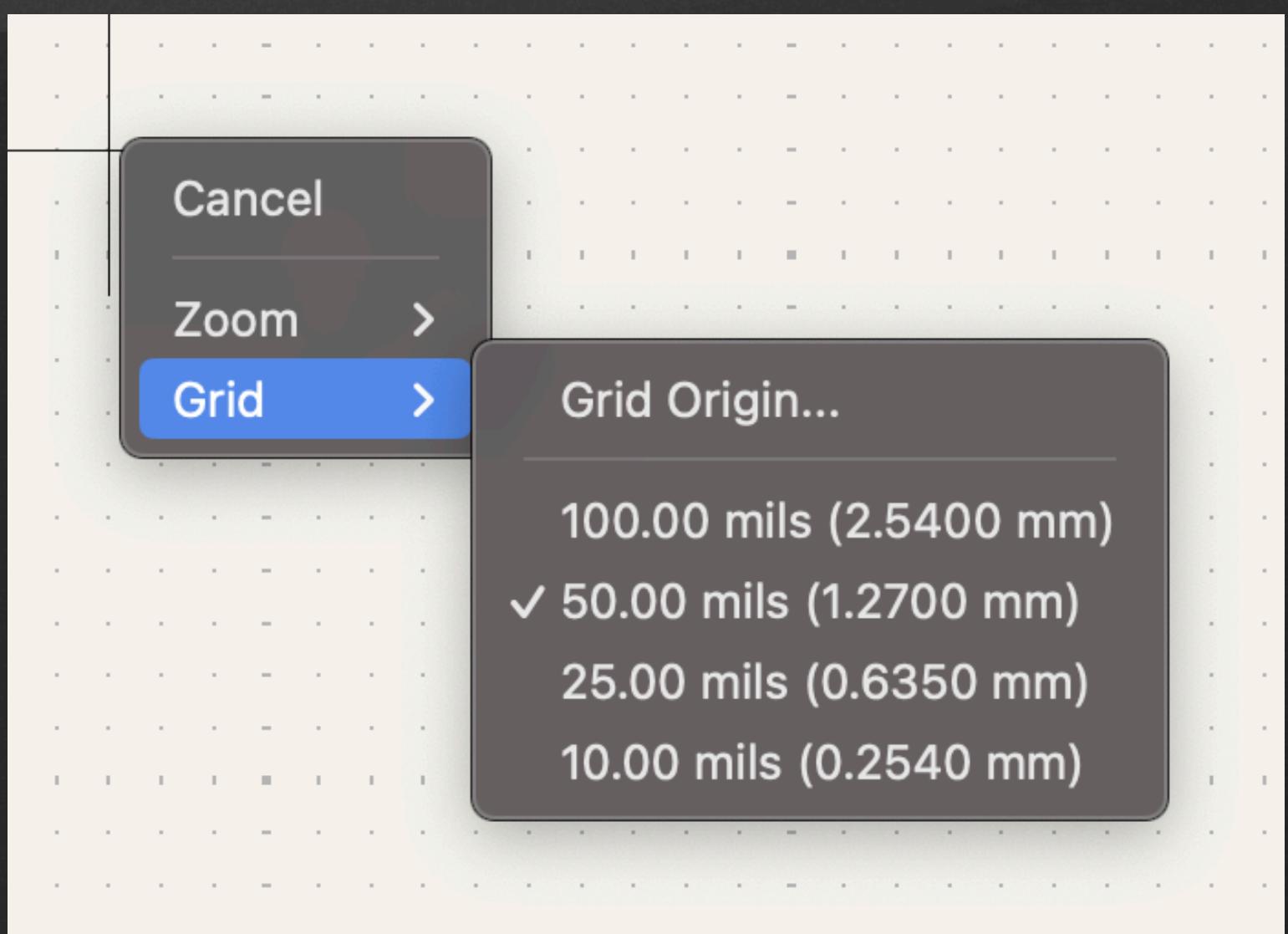
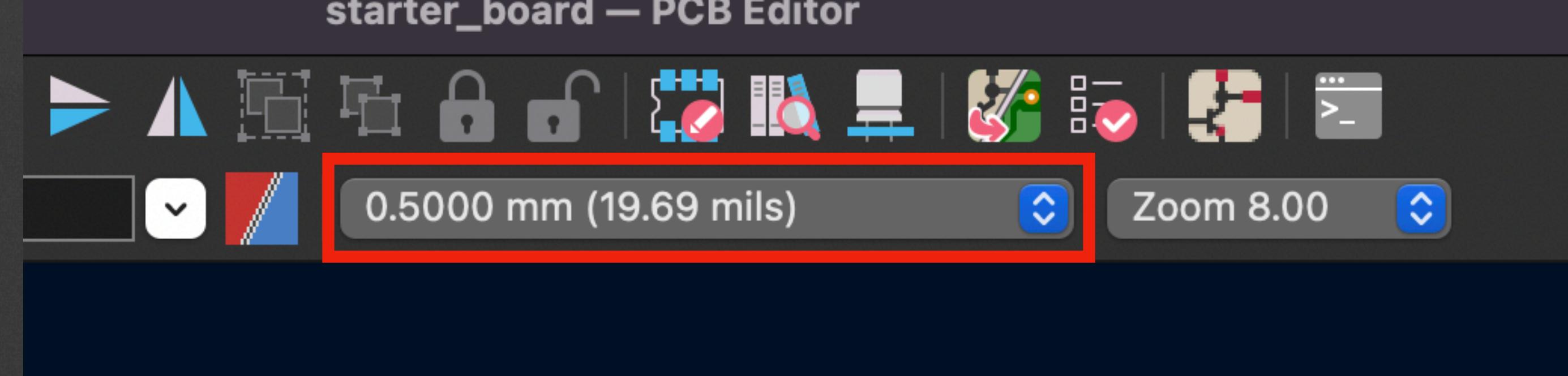
- Today you will be starting your boards
- You will use the starter files from our GitHub
- Please download/clone the entire repo:
  - [github.com/ieee-wpi/pcb](https://github.com/ieee-wpi/pcb)
- **Thank you for sticking with us!**
- We hope you learned something 😊

# Schematic View

# grid

## why are we in manhattan

- Exists in schematic view and board view
- The cursor will **snap** to grid coordinates
- Allows for order within your designs
- Choose a reasonable size for your purpose



# symbols

## schematic view

- **Symbols** are graphical representations of components
  - You can link a physical **footprint** with a symbol
  - Most parts are in KiCad's default symbol library
- 
- **A** -> Add symbol
  - **E** -> Properties



# wires and labels

## schematic view

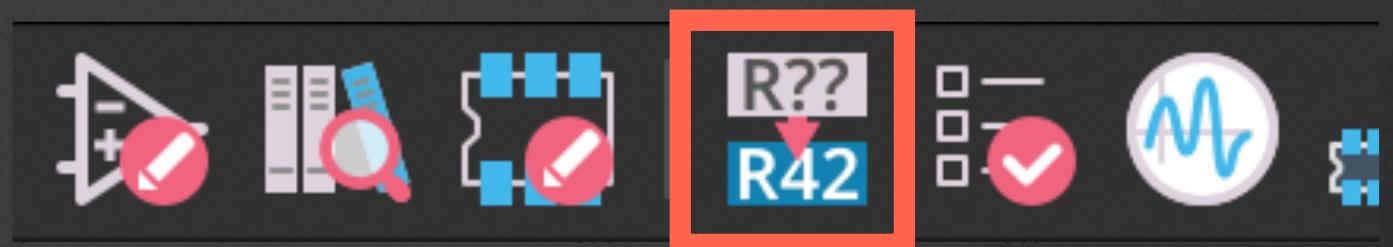
- **W** -> Add wire (connect component pins)
- **Q** -> Add no connect flag (for unused pins)
- **L** -> Add label (invisible wire)



# good practice

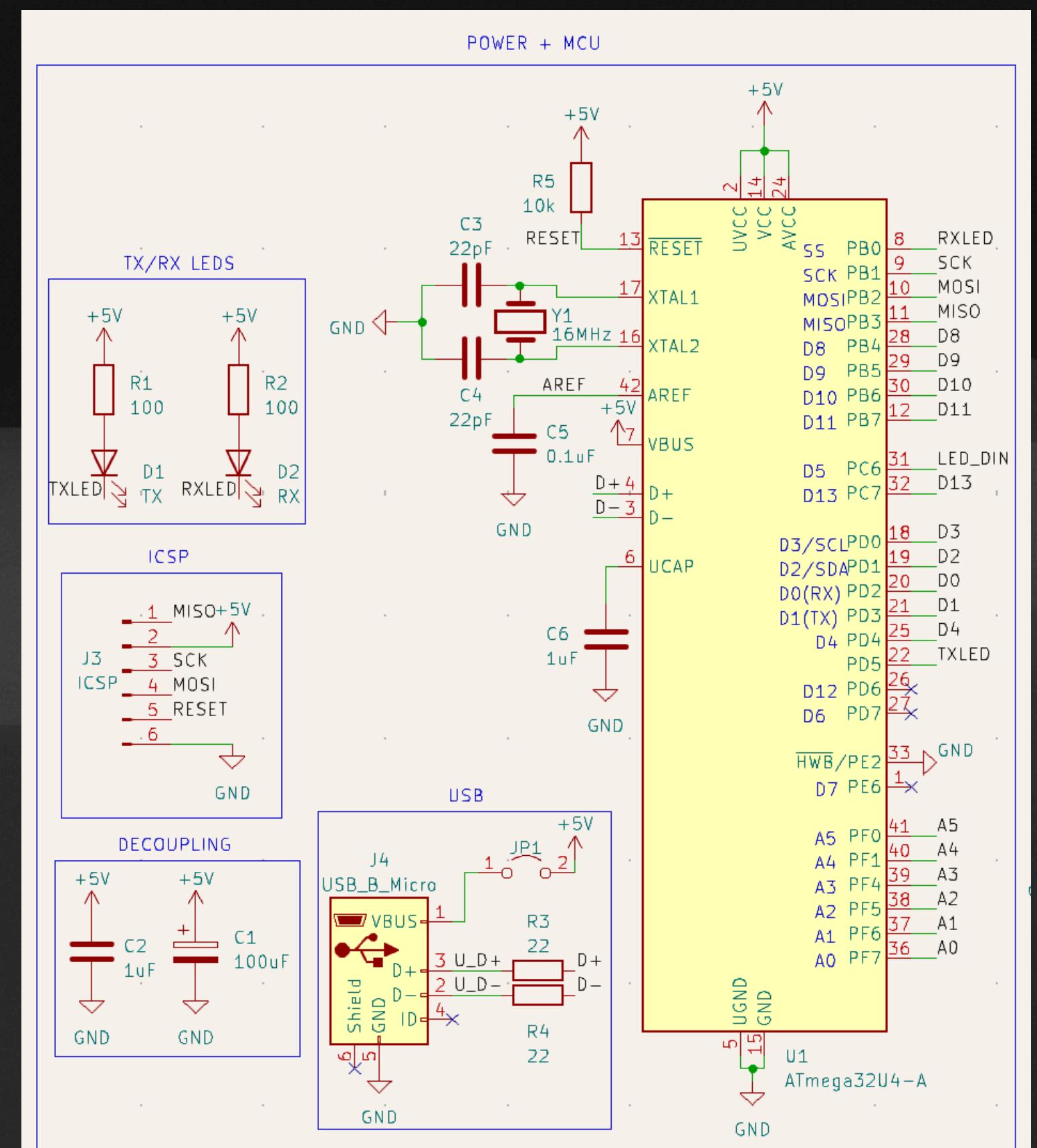
## schematic view

- Organize your schematic by function
- Add text and boxes around each section
- Typically place important sections left to right
- Can change reference designator ordering



- Update title block

Sheet: /			
File: starter_board.kicad_sch			Author: Daniel Gorbunov
<b>Title: Essentials of PCB Design Starter Board</b>			
Size: A4	Date: 2024-03-17	Rev: v1	
KiCad E.D.A. 8.0.5		Id: 1/1	



# Board View

# setup board view

- Design Rules (already setup for Track 1)

File -> Board Setup -> Design Rules -> Constraints

Category	Parameter	Value	Unit
Copper	Minimum clearance:	0.1	mm
	Minimum track width:	0.1	mm
	Minimum connection width:	0.1	mm
	Minimum annular width:	0.1	mm
	Minimum via diameter:	0.5	mm
	Copper to hole clearance:	0.254	mm
	Copper to edge clearance:	0.2	mm
Holes	Minimum through hole:	0.3	mm
	Hole to hole clearance:	0.25	mm
uVias	Minimum uVia diameter:	0.2	mm
	Minimum uVia hole:	0.1	mm
Silkscreen	Minimum item clearance:	0	mm
	Minimum text height:	0.8	mm
	Minimum text thickness:	0.08	mm

- Disable ground nets

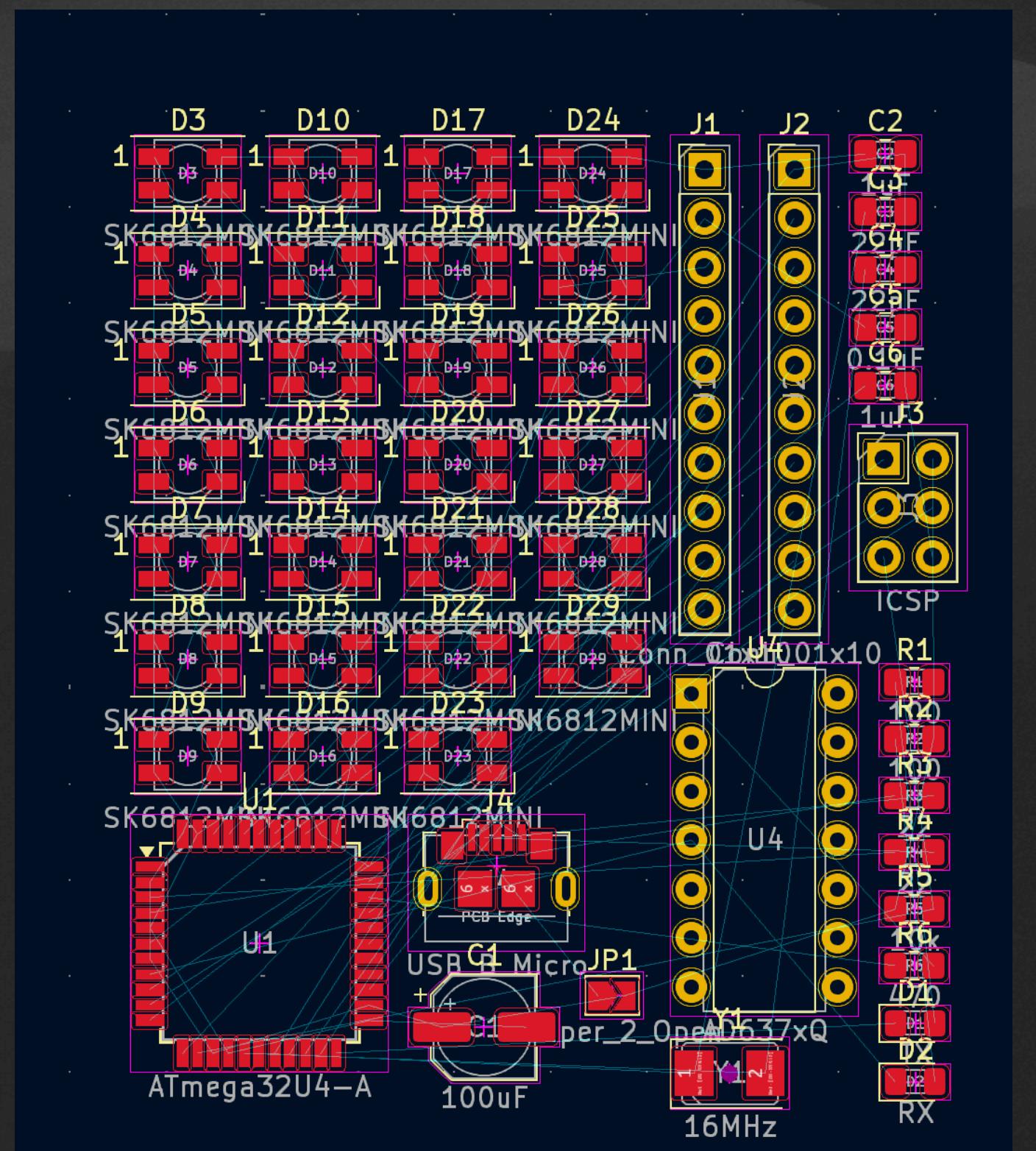
Appearance -> Nets

Your **last step** will be ground pours



# linking schematic and board board view

- Update the PCB after changing the schematic
  - Tools -> Update PCB from schematic (F8)
  - Places any unplaced components \*somewhere\*
  - It is **your** job to layout the components in a way that makes sense

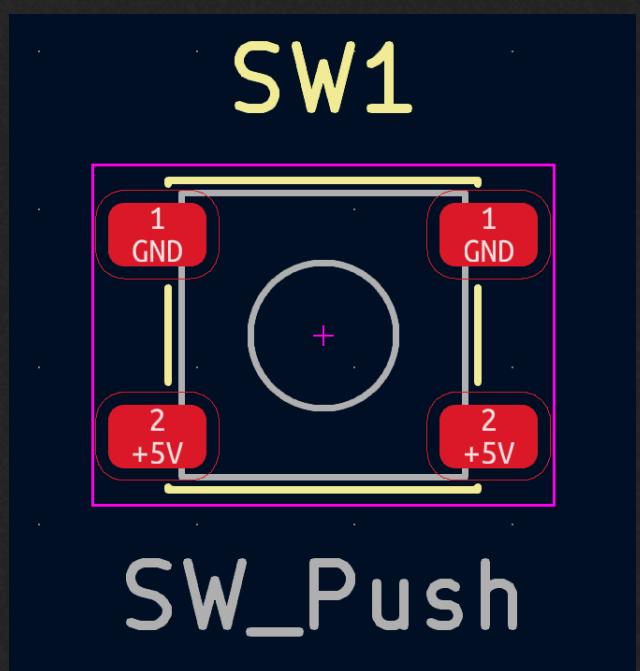
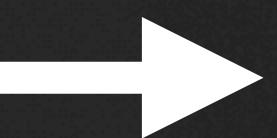
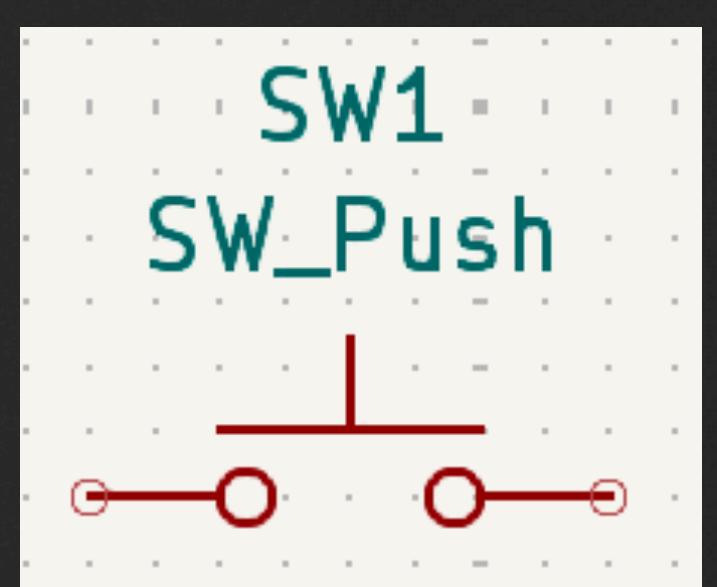
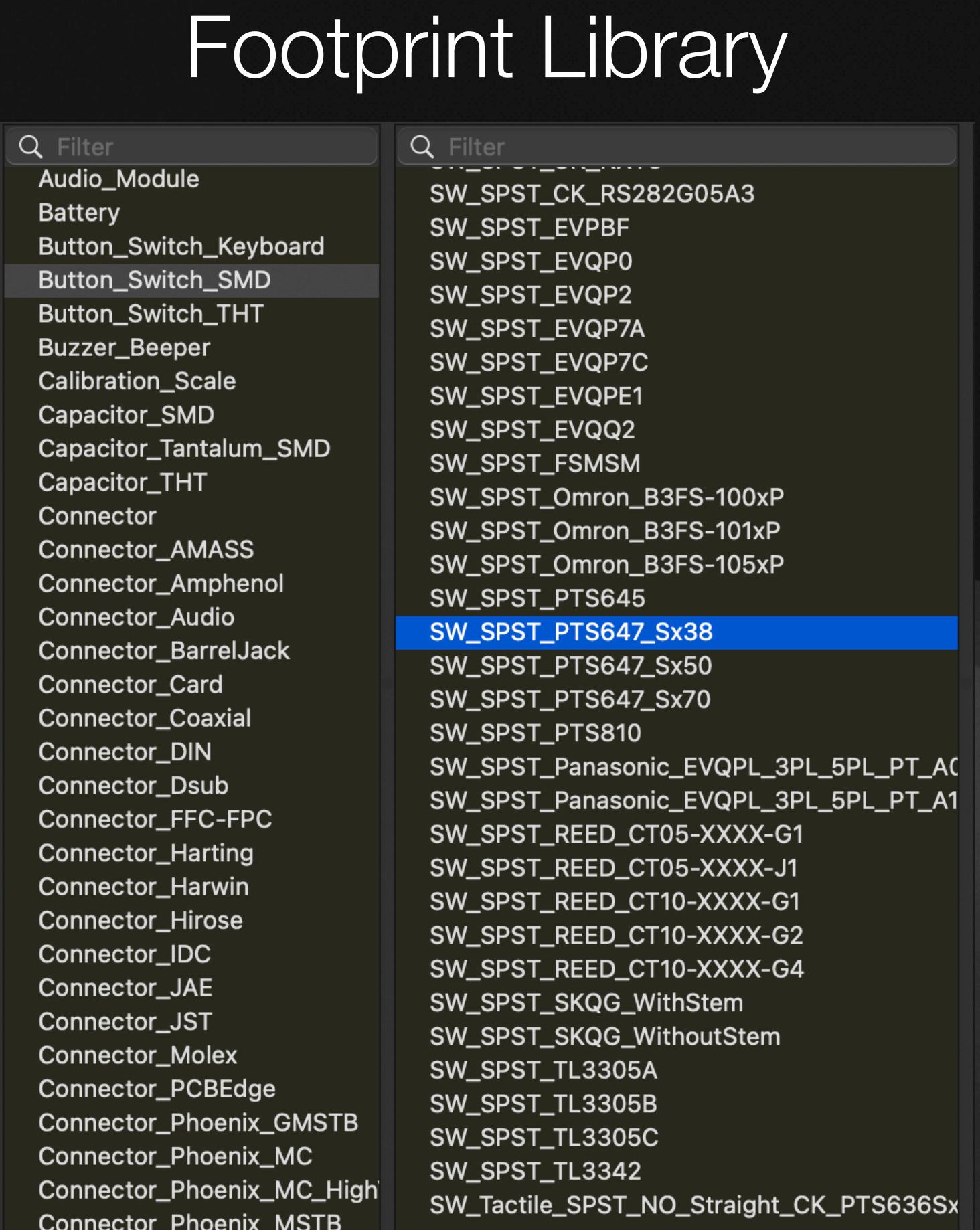


# footprints

## board view

- Footprints represent the physical layout and dimensions of component connections
- Footprints are linked with a symbol in the schematic
- Footprints include multiple layers - copper, silkscreen, solder mask, drill holes, keep out

Name	Value
Reference	SW1
Value	SW_Push
Footprint	Button_Switch_SMD:SW_SPST PTS647_Sx38
Datasheet	~
Description	Push button switch, generic, two pins



# footprints

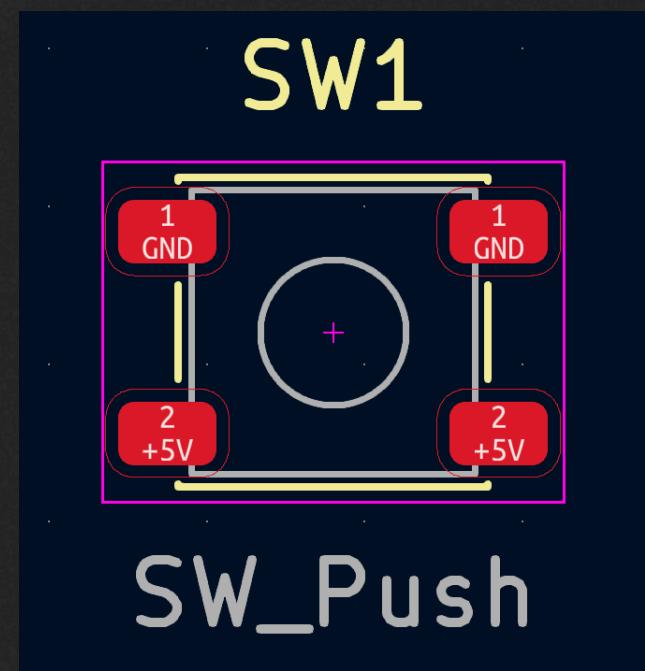
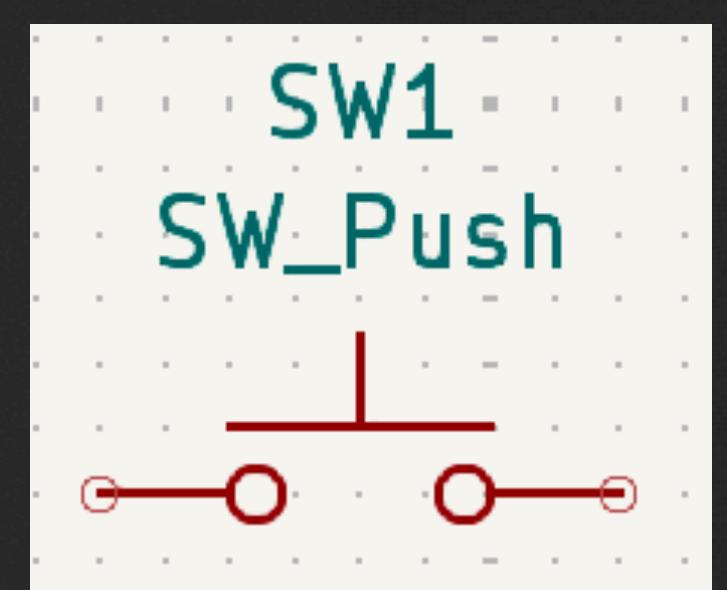
## board view

- Many parts are in KiCad's default footprint library
- For parts that are not in the default library, you can download their footprints online

## Footprint Library

Filter
Audio_Module
Battery
Button_Switch_Keyboard
<b>Button_Switch_SMD</b>
Button_Switch_THT
Buzzer_Beeper
Calibration_Scale
Capacitor_SMD
Capacitor_Tantalum_SMD
Capacitor_THT
Connector
Connector_AMASS
Connector_Amphenol
Connector_Audio
Connector_BarrelJack
Connector_Card
Connector_Coaxial
Connector_DIN
Connector_Dsub
Connector_FFC-FPC
Connector_Harting
Connector_Harwin
Connector_Hirose
Connector_IDC
Connector_JAE
Connector_JST
Connector_Molex
Connector_PCBEdge
Connector_Phoenix_GMSTB
Connector_Phoenix_MC
Connector_Phoenix_MC_High
Connector_Phoenix_MSTB
<b>SW_SPST PTS647 Sx38</b>
SW_SPST PTS647 Sx50
SW_SPST PTS647 Sx70
SW_SPST PTS810
SW_SPST_Panasonic_EVQPL_3PL_5PL_PT_A0
SW_SPST_Panasonic_EVQPL_3PL_5PL_PT_A1
SW_SPST_REED_CT05-XXXX-G1
SW_SPST_REED_CT05-XXXX-J1
SW_SPST_REED_CT10-XXXX-G1
SW_SPST_REED_CT10-XXXX-G2
SW_SPST_REED_CT10-XXXX-G4
SW_SPST_SKQG_WithStem
SW_SPST_SKQG_WithoutStem
SW_SPST_TL3305A
SW_SPST_TL3305B
SW_SPST_TL3305C
SW_SPST_TL3342
SW_Tactile_SPST_NO_Straight_CK PTS636Sx

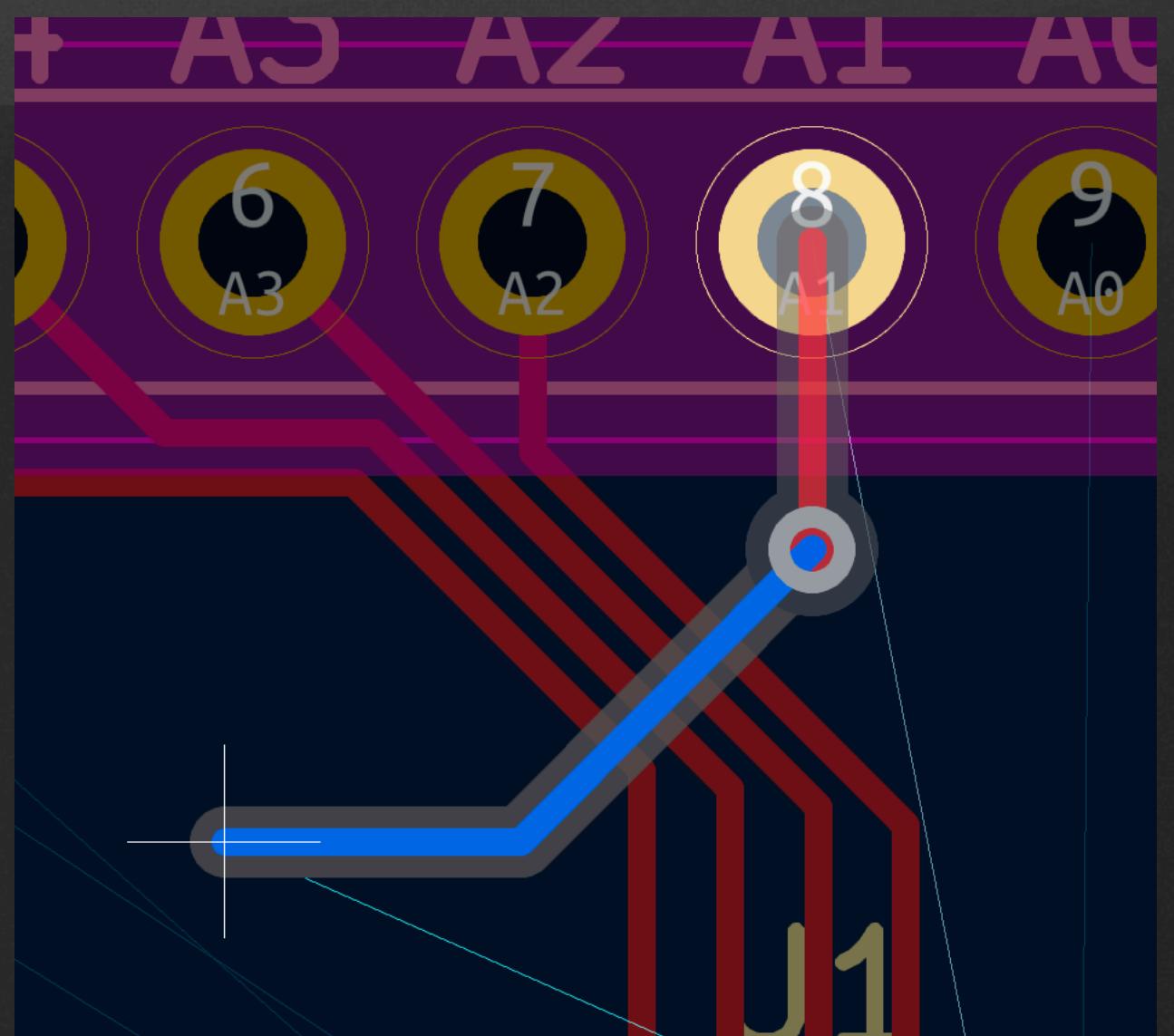
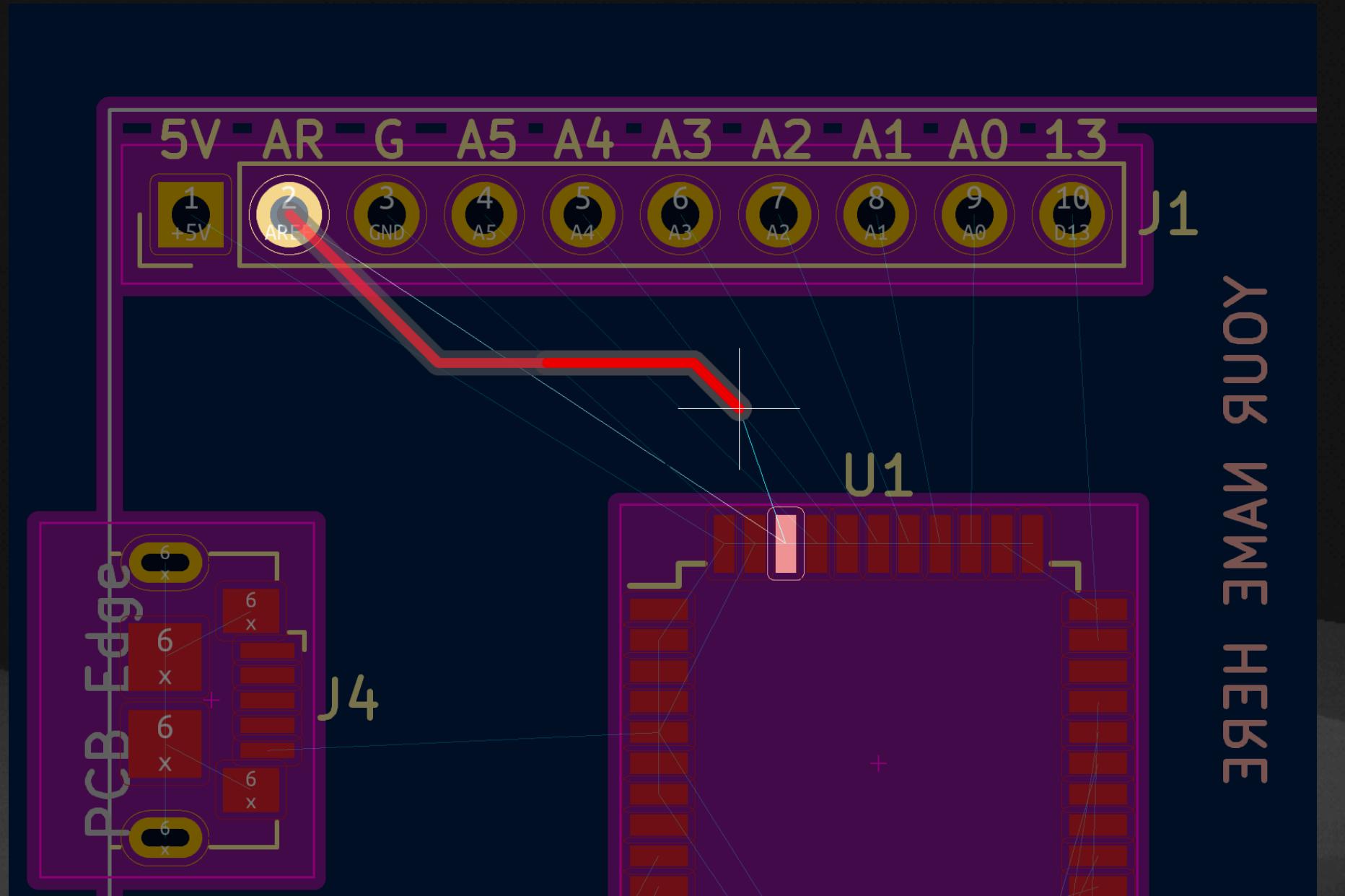
Name	Value
Reference	SW1
Value	SW_Push
Footprint	Button_Switch_SMD:SW_SPST PTS647 Sx38
Datasheet	~
Description	Push button switch, generic, two pins



# routing board view

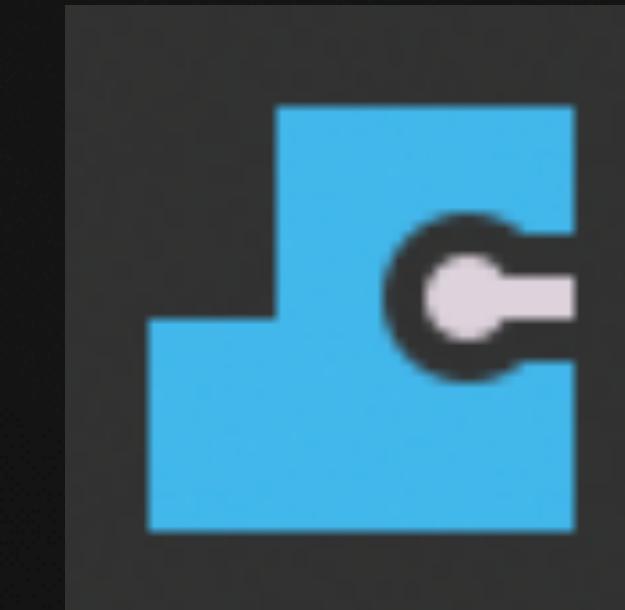
- Follow nets to see what needs to be connected
- Use vias sparsely to bypass intersections

- **X** -> Route a single trace
- **V** -> Place a via (while routing)
- **Cmd+Shift+V** -> Place a via (anytime)
- **U** -> Select neighboring trace segments

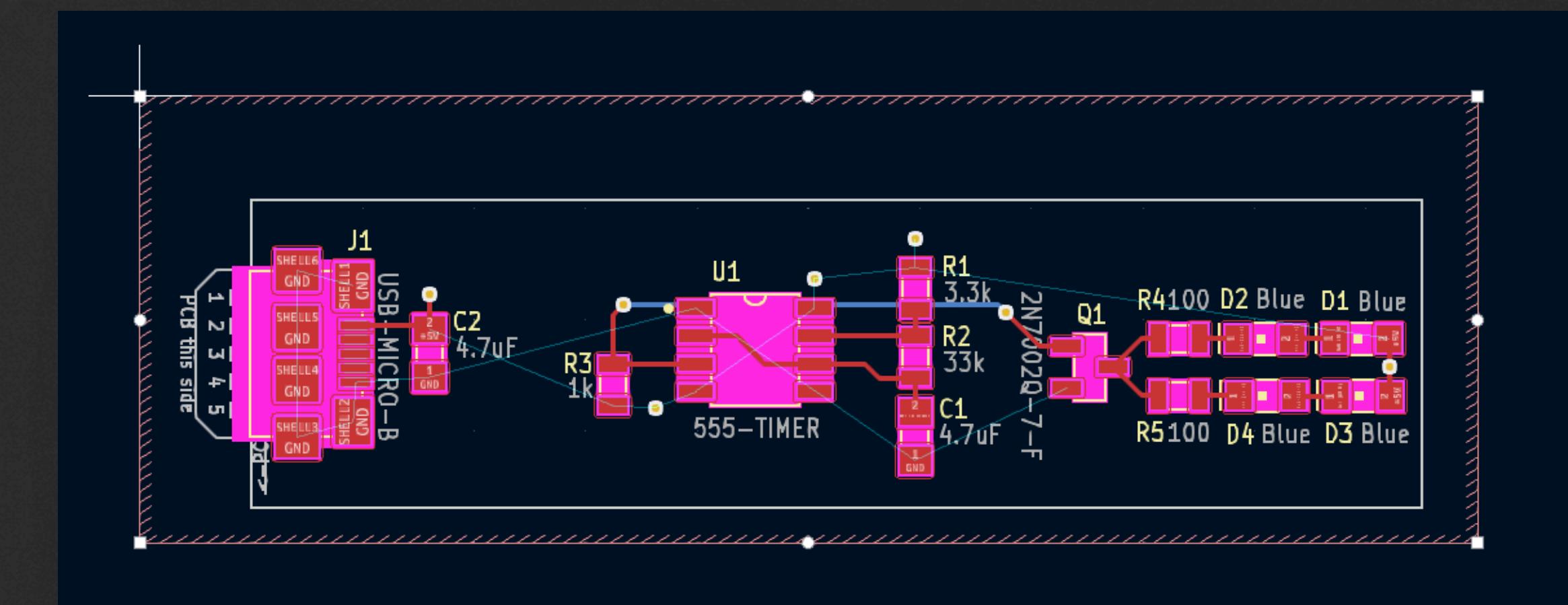
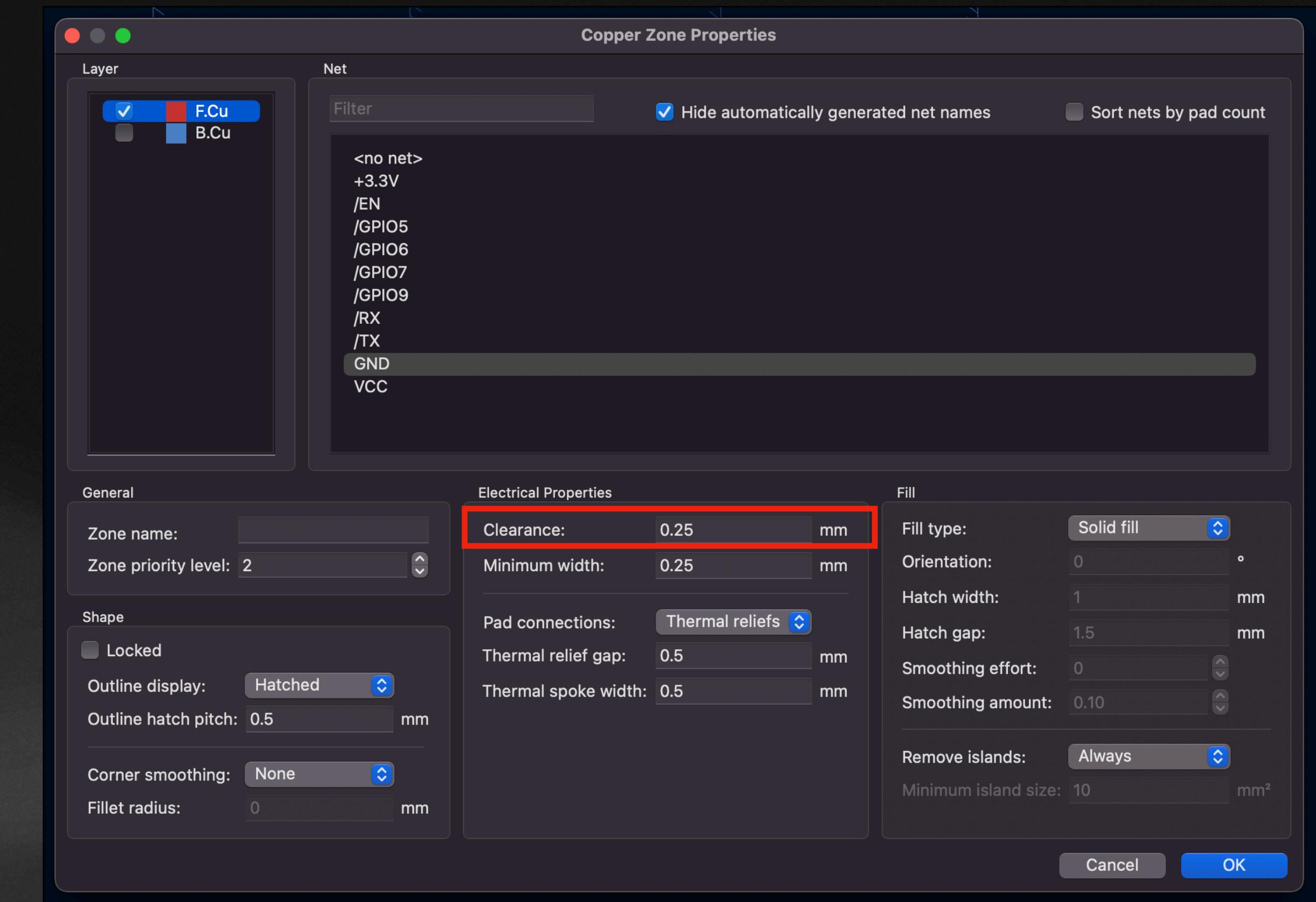


NAME HERE

# pours board view

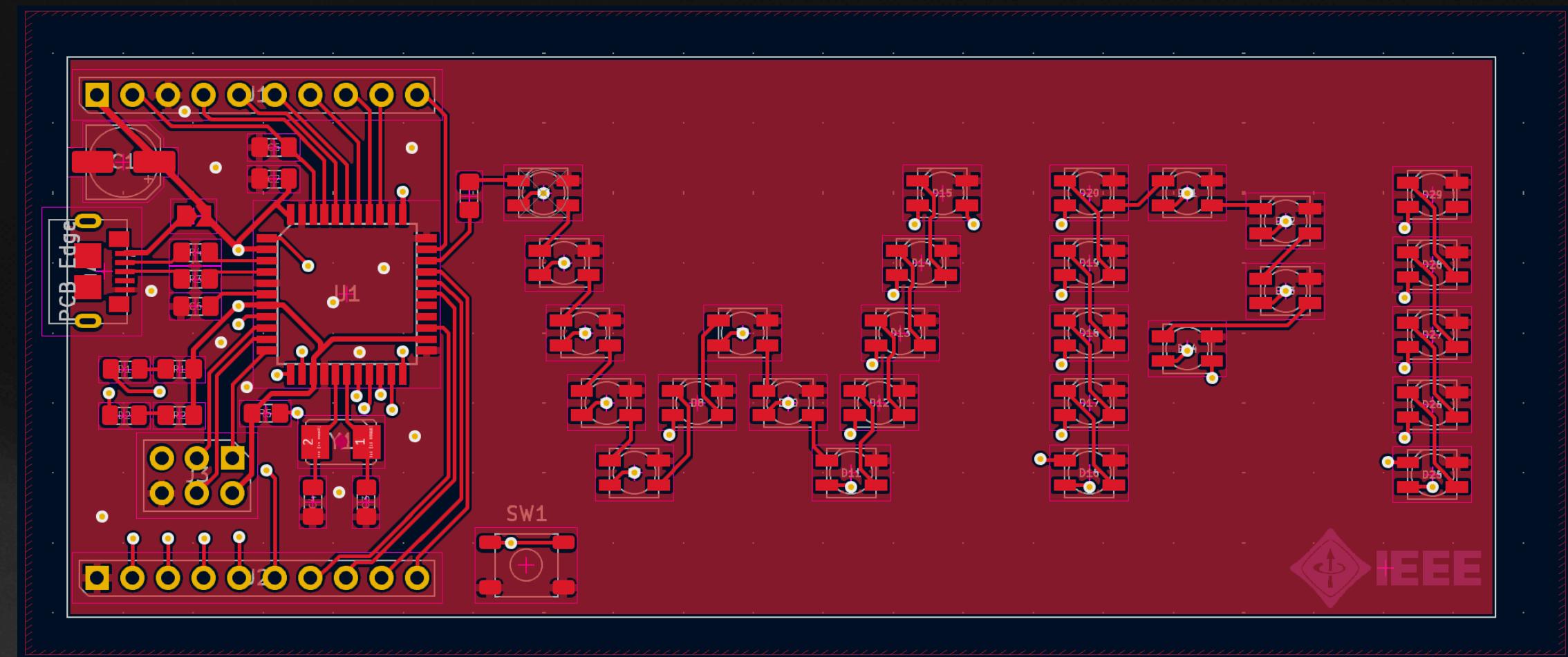


- You will need to use multiple ground pours on your board
- This will help simplify a lot of routing
- Click “Add filled zone”
- Select the correct layer and net
- Choose the desired properties
- Draw a zone around the desired area
- **B** -> refill all pours

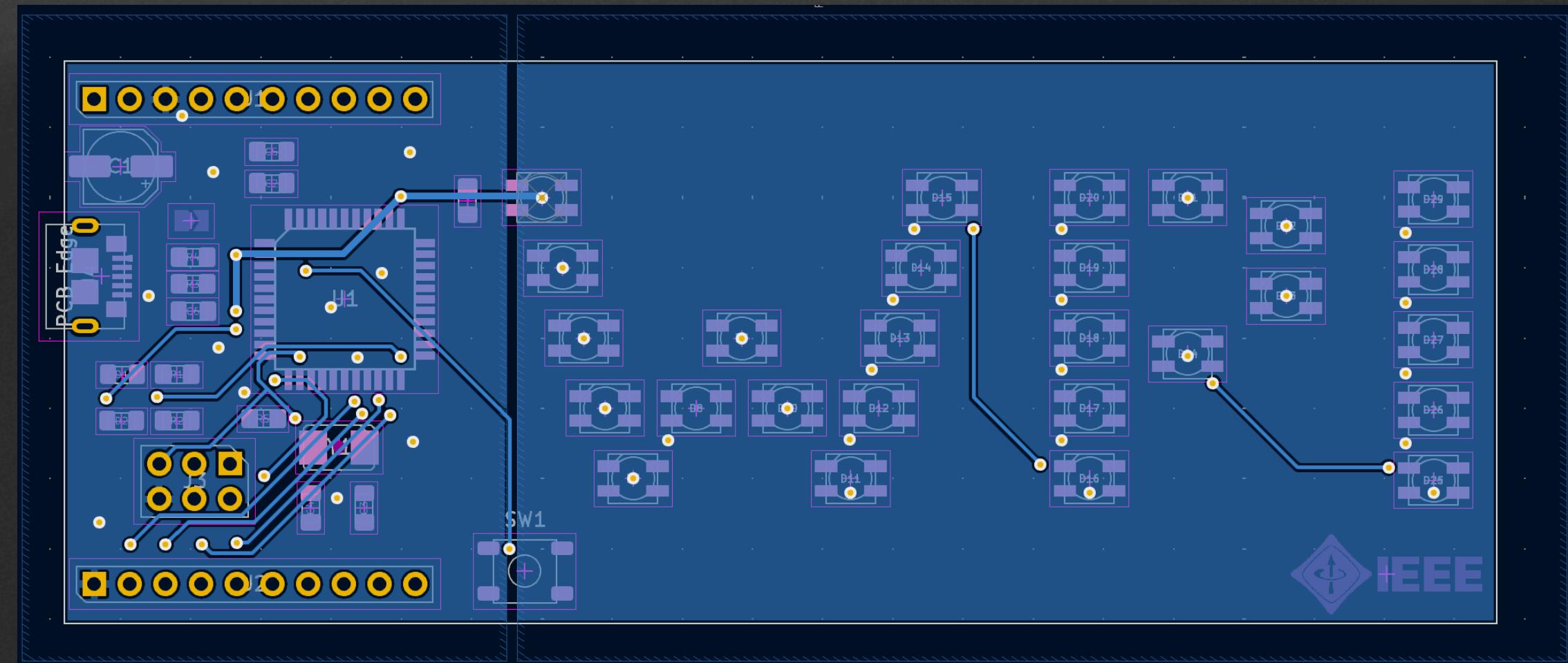


# pours board view

- Disable all ground nets
- You should not be routing any ground traces until the rest of your routing is complete
- Vcc pour can also be useful on your board (ultimately up to you)



Front Layer (GND)



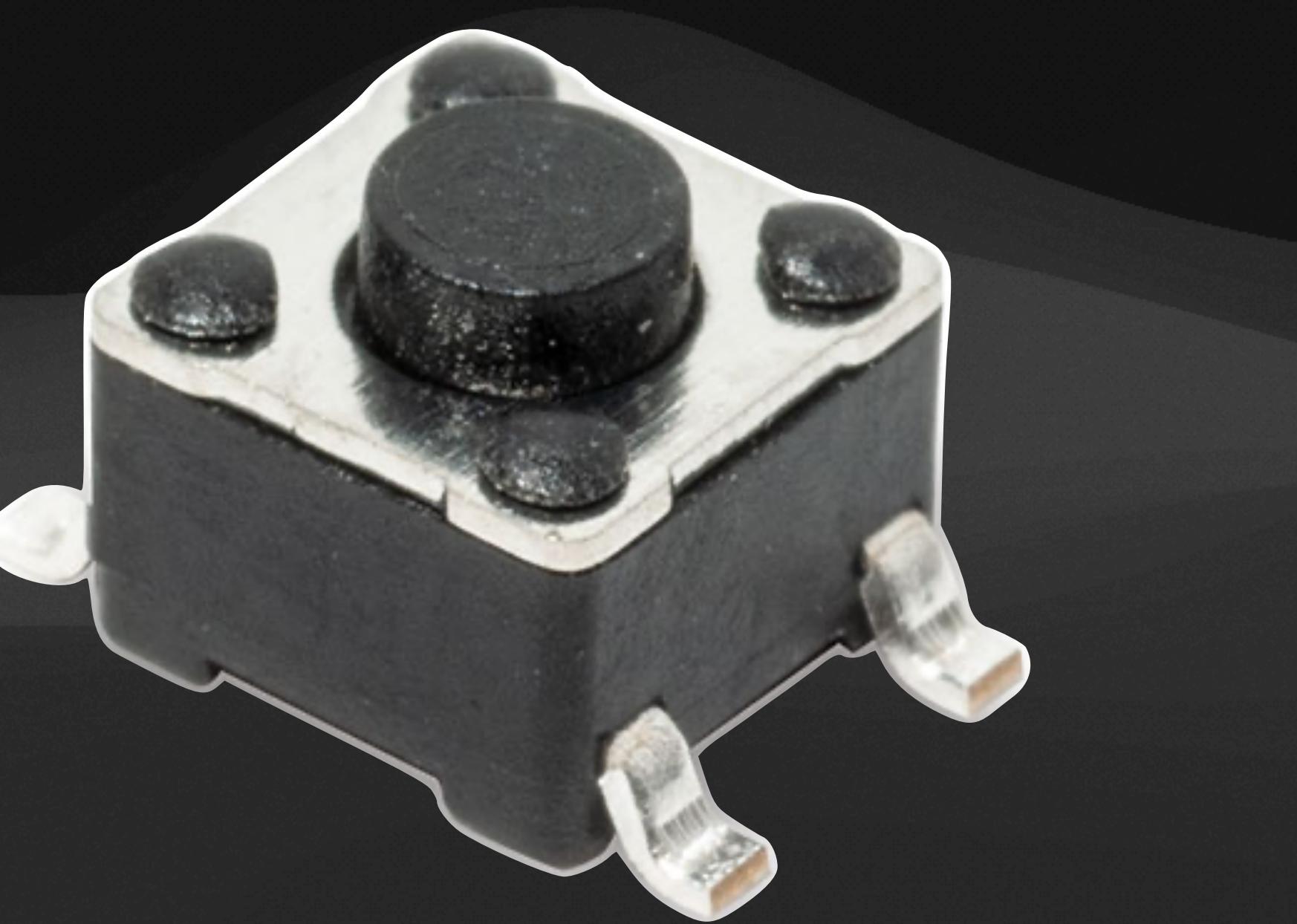
Back Layer (GND left, VCC right)

# Practice!

# adding a button

## practice

- You'll be adding a button to the starter board!
- You'll have software control over what this button does!
- We need to figure out:
  - Schematic
  - Layout
  - Routing



The **PTS 647 SK38 SMTR2 LFS** 😊

# button

## PTS647

**PTS 647 SK38 SMTR2 LFS**



*Image shown is a representation only. Exact specifications should be obtained from the product data sheet.*

<b>DigiKey Part Number</b>	PTS647SK38SMTR2LFSTR-ND - Tape & Reel (TR) PTS647SK38SMTR2LFSC-ND - Cut Tape (CT) PTS647SK38SMTR2LFSDKR-ND - Digi-Reel®
<b>Manufacturer</b>	C&K
<b>Manufacturer Product Number</b>	PTS 647 SK38 SMTR2 LFS
<b>Description</b>	SWITCH TACTILE SPST-NO 0.05A 12V
<b>Manufacturer Standard Lead Time</b>	9 Weeks
<b>Customer Reference</b>	[Redacted]
<b>Detailed Description</b>	Tactile Switch SPST-NO Top Actuated Surface Mount
<b>Datasheet</b>	 <a href="#">Datasheet</a>
<b>EDA/CAD Models</b>	<a href="#">PTS 647 SK38 SMTR2 LFS Models</a>

**PTS647 Series  
4.5 mm Tact Switch**

**Features/Benefits**

- Compact size 4.5 x 4.5 mm
- SMT terminals
- Different actuator heights
- Choice of actuation forces
- Tape & reel

**Typical Applications**

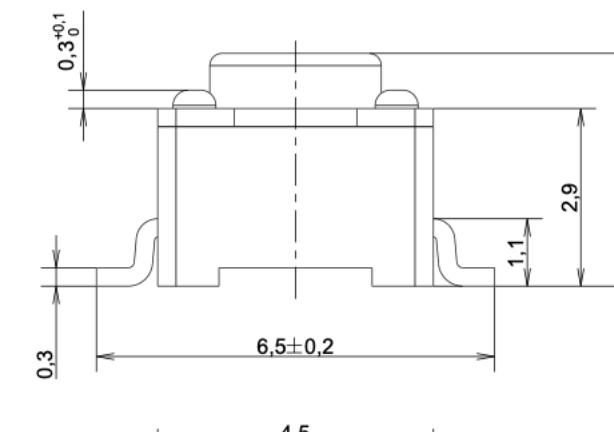
- Consumer
- Industrial control panels
- Computer products



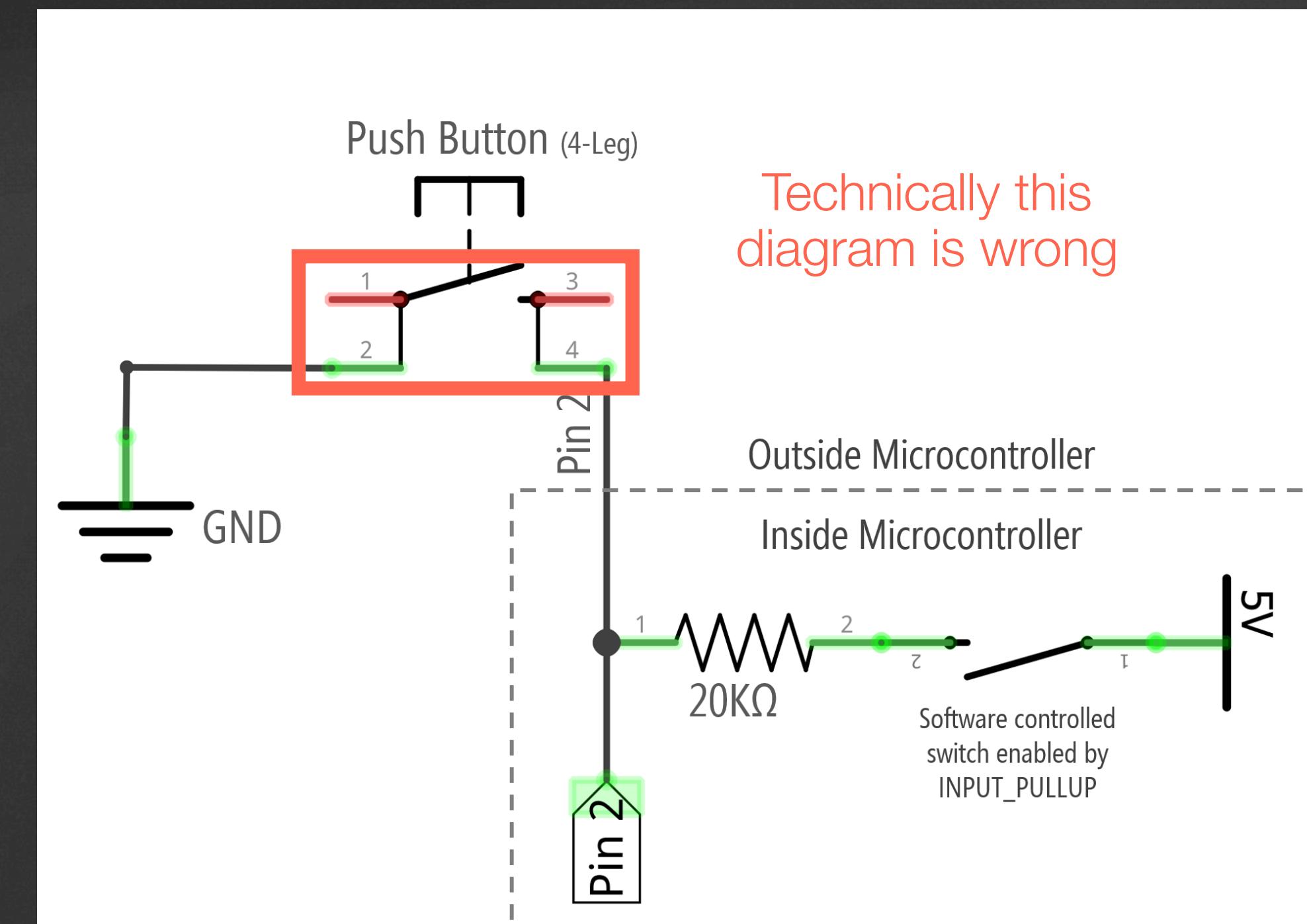
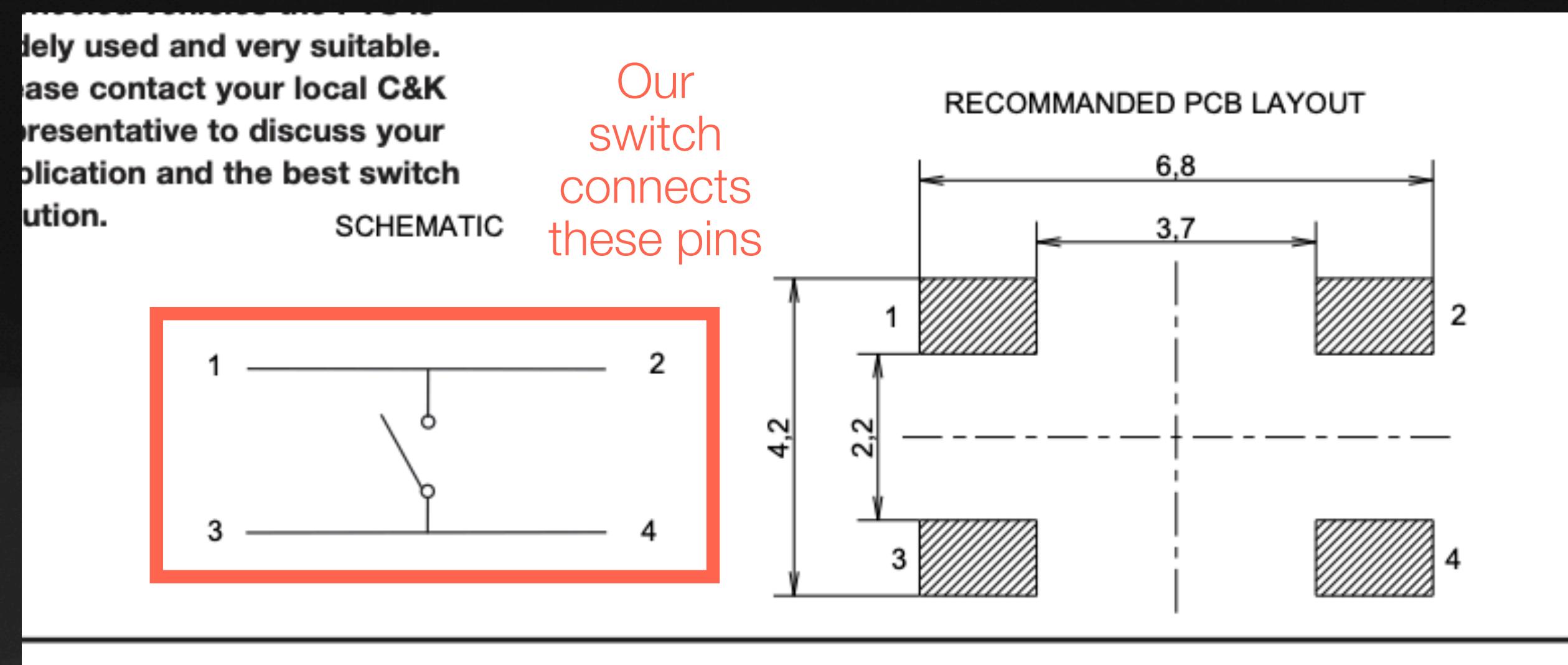
**B**

**Tactile Switches**

Part Number	Operating Force (gf)	Life	Height (H)
PTS647 SN38 SMTR2 LFS	100 +/- 50	100,000	3.8 mm
PTS647 SM38 SMTR2 LFS	180 +/- 50	100,000	3.8 mm
PTS647 SK38 SMTR2 LFS	250 +/- 50	100,000	3.8 mm
PTS647 SN50 SMTR2 LFS	100 +/- 50	100,000	5.0 mm
PTS647 SM50 SMTR2 LFS	180 +/- 50	100,000	5.0 mm
PTS647 SK50 SMTR2 LFS	250 +/- 50	100,000	5.0 mm
PTS647 SN70 SMTR2 LFS	100 +/- 50	100,000	7.0 mm
PTS647 SM70 SMTR2 LFS	180 +/- 50	100,000	7.0 mm
PTS647 SK70 SMTR2 LFS	250 +/- 50	100,000	7.0 mm

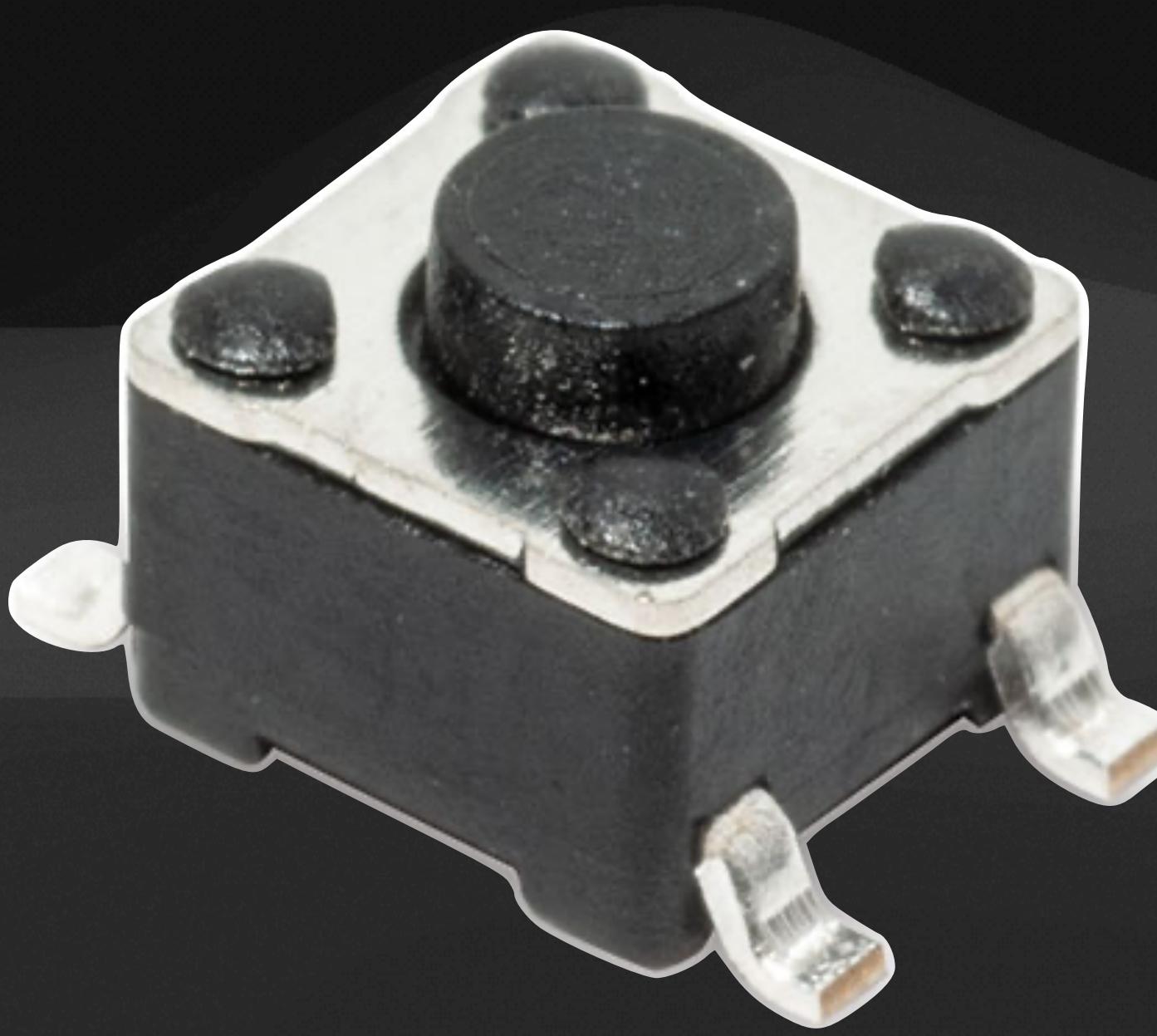


# button schematic



# button schematic

- Connect to microcontroller pin 1
- Pin 1 is interrupt enabled
- This means we won't need to poll the button's state in software
- Footprint built into KiCad's default library



The **PTS 647 SK38 SMTR2 LFS** 😊

# kicad: cheatsheet

## useful keybinds

- **Left click** -> select + move objects
- **Right click** -> move view
- **E** -> edit properties

# KiCad

## Cheatsheet

<https://docs.kicad.org>

### 1) Create a project

File → New Project → New Project

### 2) Schematic Editor

Add components : A  
Move item<sup>1</sup> : ⌘ + M  
Grab item<sup>1</sup> : ⌘ + G  
Expand selection: ⌘ + Shift + ⌘  
Deselect items: ⌘ + ⌘ + ⌘ + ⌘  
Delete item : ⌘ + Del  
Edit Symbol : ⌘ + ⌘ + E  
Rotate item : ⌘ + R  
Mirror item : ⌘ + X / Y  
Add wires : W  
Edit properties : E  
Edit value : V  
Add power symbols : P  
Add no-connect : Q  
Add text : T  
Add labels : L  
List of shortcuts : ⌘ + ⌘ + Shift + F1

<sup>1</sup>grab keeps connections, move doesn't

How to assign footprint in Schematic Editor:  
Edit Symbol Fields  
-or-  
Assign Footprints

### 5) PCB Editor

Update PCB From Schematic

Board Setup

Switch Viewport ⌘ + Tab

Switch Active Layerset ⌘ + Tab

Move item : ⌘ + M

Flip item side : ⌘ + F

Rotate item : ⌘ + R

Add footprint : A

Add tracks : X

Add via<sup>1</sup> : ⌘ + ⌘ + Shift + V

Switch posture : /

Switch track width : W

Drag track / footprint : D

Fill zones : B

3D viewer : Alt + 3

Measure : ⌘ + ⌘ + Shift + M

<sup>1</sup>while routing, only 'V' is needed

### 6) Export Gerbers / IPC2581

File → Fabrication Outputs

.gbr ↑ / .xml ↑



Check result with Gerber Viewer

# practice adding a button

- PTS 647 SK38 SMTR2 LFS (4.5x4.5mm) connected to microcontroller pin 1
- Add SW\_PUSH symbol to schematic (A)
- Edit the symbol (E) and attach the SW\_SPST PTS647\_Sx38 footprint
- Practice basic layout + routing with the button
- **Afterwards, start your design!**

## KiCad Cheatsheet <https://docs.kicad.org>

### 1) Create a project

File → New Project → New Project

### 2) Schematic Editor

Add components :	A
Move item <sup>1</sup> :	↑ + M
Grab item <sup>1</sup> :	↑ + G
Expand selection:	↑ Shift + □
Deselect items:	Ctrl + ↑ Shift + □
Delete item :	↑ + Del
Edit Symbol :	↑ + Ctrl + E
Rotate item :	↑ + R
Mirror item :	↑ + X / Y
Add wires :	W
Edit properties :	E
Edit value :	V
Add power symbols :	P
Add no-connect :	Q
Add text :	T
Add labels :	L
List of shortcuts :	Ctrl + ↑ Shift + F1

<sup>1</sup>grab keeps connections, move doesn't

How to assign footprint in Schematic Editor :  
Edit Symbol Fields  
-or-  
Assign Footprints

### 5) PCB Editor

Update PCB From Schematic

Board Setup

Switch Viewport ..... ↑ Shift + Tab

Switch Active Layerset ..... Ctrl + Tab

Move item : ..... ↑ + M

Flip item side : ..... ↑ + F

Rotate item : ..... ↑ + R

Add footprint : ..... A

Add tracks : ..... X

Add via<sup>1</sup> : ..... Ctrl + ↑ Shift + V

Switch posture : ..... /

Switch track width : ..... W

Drag track / footprint : ..... D

Fill zones : ..... B

3D viewer : ..... Alt + 3

Measure : ..... Ctrl + ↑ Shift + M

<sup>1</sup>while routing, only 'V' is needed

### 6) Export Gerbers / IPC2581

File → Fabrication Outputs

.gbr ↑ / .xml ↑



Check result  
with Gerber Viewer