

# Essentials of PCB Design

04: KiCad

# welcome back!

## schedule

### A-Term

<b>Tue, Sep. 24</b>	Basics of PCBs <b>6-7PM; SH 301</b>
<b>Thur, Sep. 26</b>	Designing your Project <b>6-7:30PM; SH 201</b>
<b>Tue, Oct. 1</b>	Layout + Routing <b>6-7:30PM; SH 201</b>
<b>Thur, Oct. 3</b>	Working with KiCad <b>6-7:30PM; SH 201</b>
<b>Mon, Oct. 7 - Fri, Oct. 11</b>	Office Hours <b>6-8PM; IEEE Lounge</b>

### B-Term

<b>Mon, Oct. 21 - Fri, Oct. 25</b>	Office Hours <b>TBD; IEEE Lounge</b>
<b>Fri, Oct. 25</b>	Boards Due <b>By 10PM</b>
<b>Tue, Nov. 5 (Wellness Day)</b>	Assembly <b>TBD; AK 113</b>

# course updates

## lecture 04

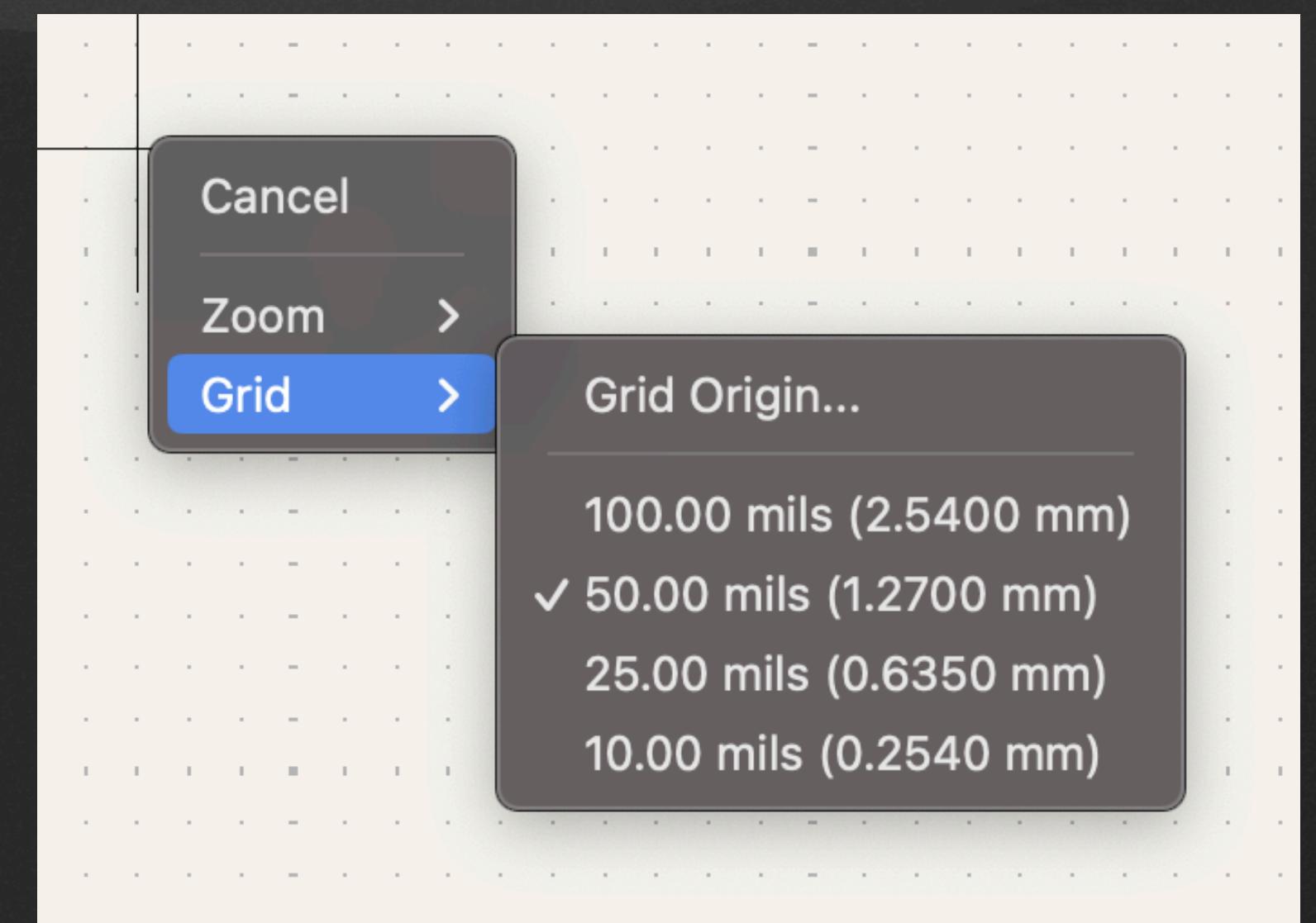
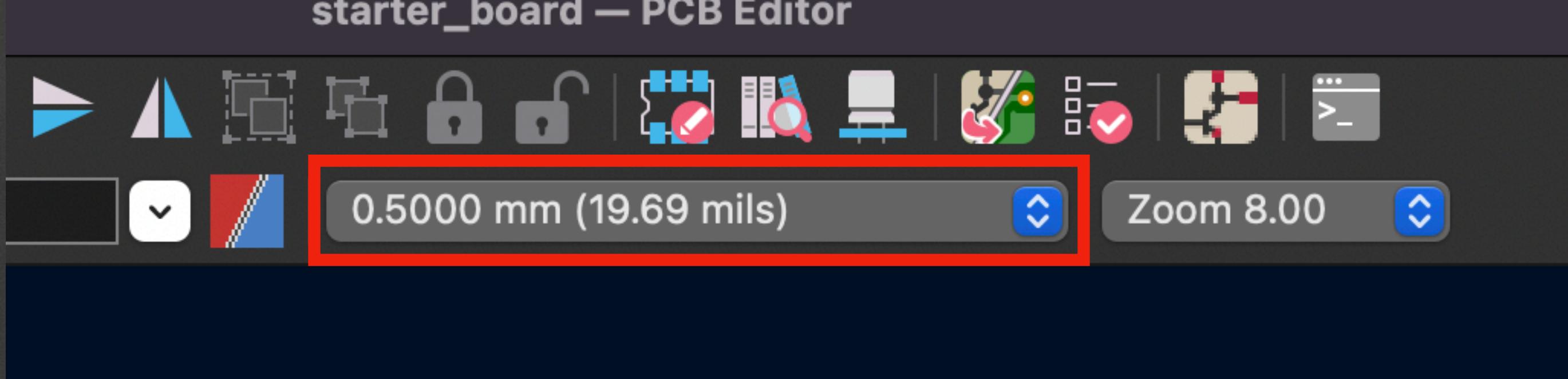
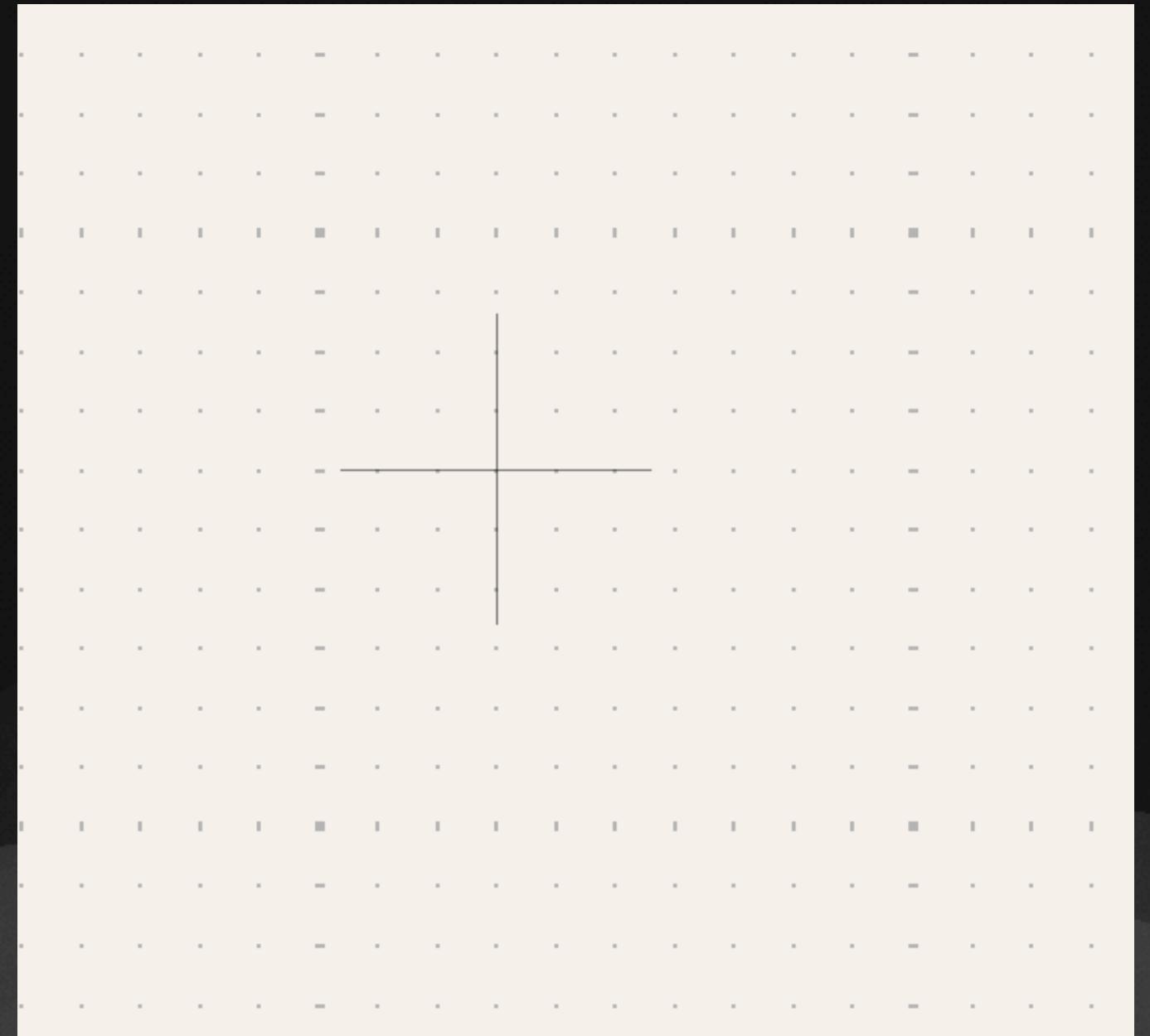
- Office hours next week: 6-8PM in IEEE Lounge (AK106)
- Pull the repo! Updated sample board w/ button
- **Thank you** for sticking with us!
- We hope you learned something 😊
- Will send out feedback for lectures

# Schematic View

# grid

## why are we in manhattan

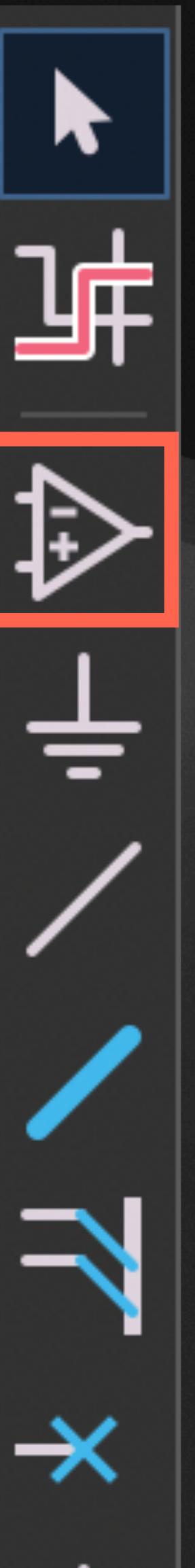
- Exists in schematic view and board view
- The cursor will **snap** to grid coordinates
- Allows for order within your designs
- Choose a reasonable size for your purpose



# symbols

## schematic view

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



# wires and labels

## schematic view

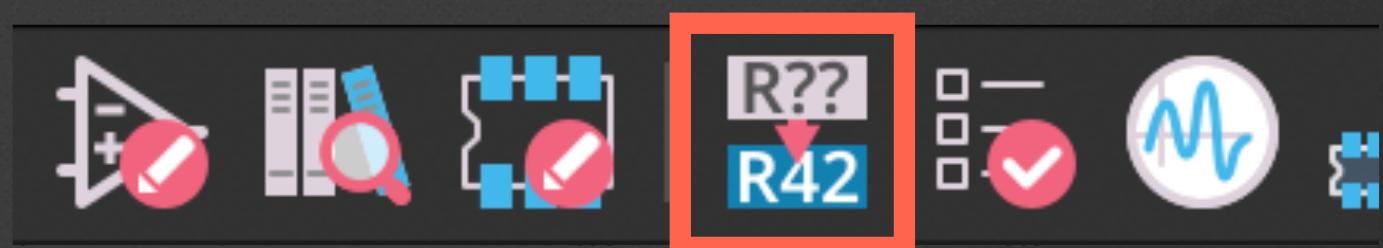
- **W** => Add wire (connect component pins)
- **Q** => Add no connect flag (for unused pins)
- **L** => Add label (invisible wire)



# good practice

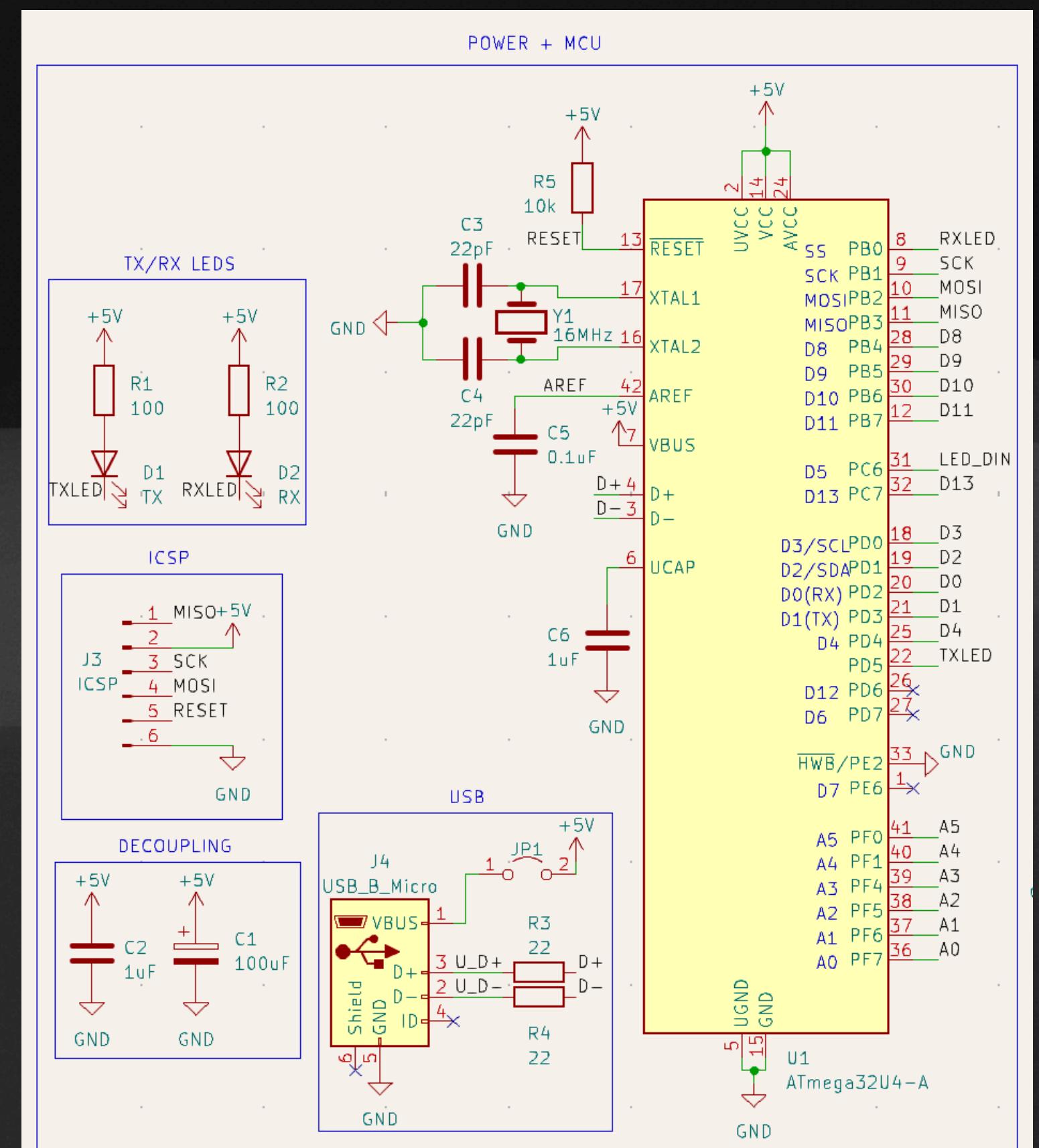
## schematic view

- 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

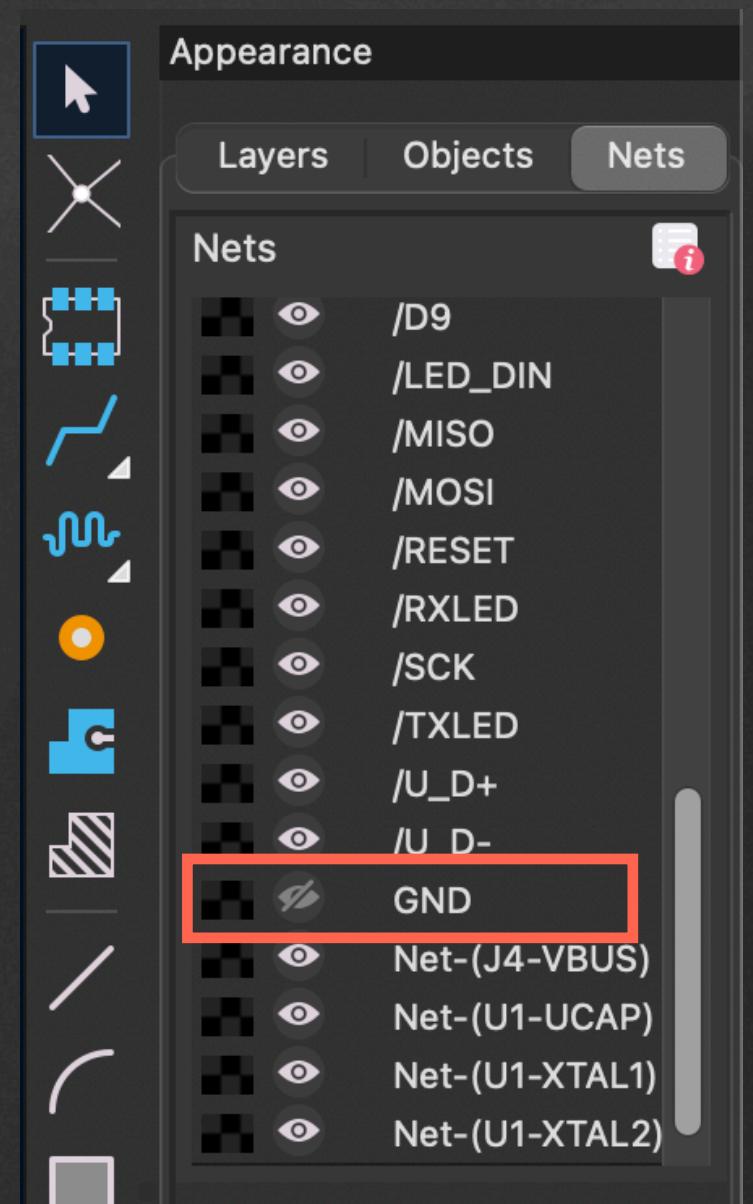
- Design Rules (already setup for Track 1)

File => Board Setup => Design Rules => Constraints

- Disable ground nets

Appearance => Nets

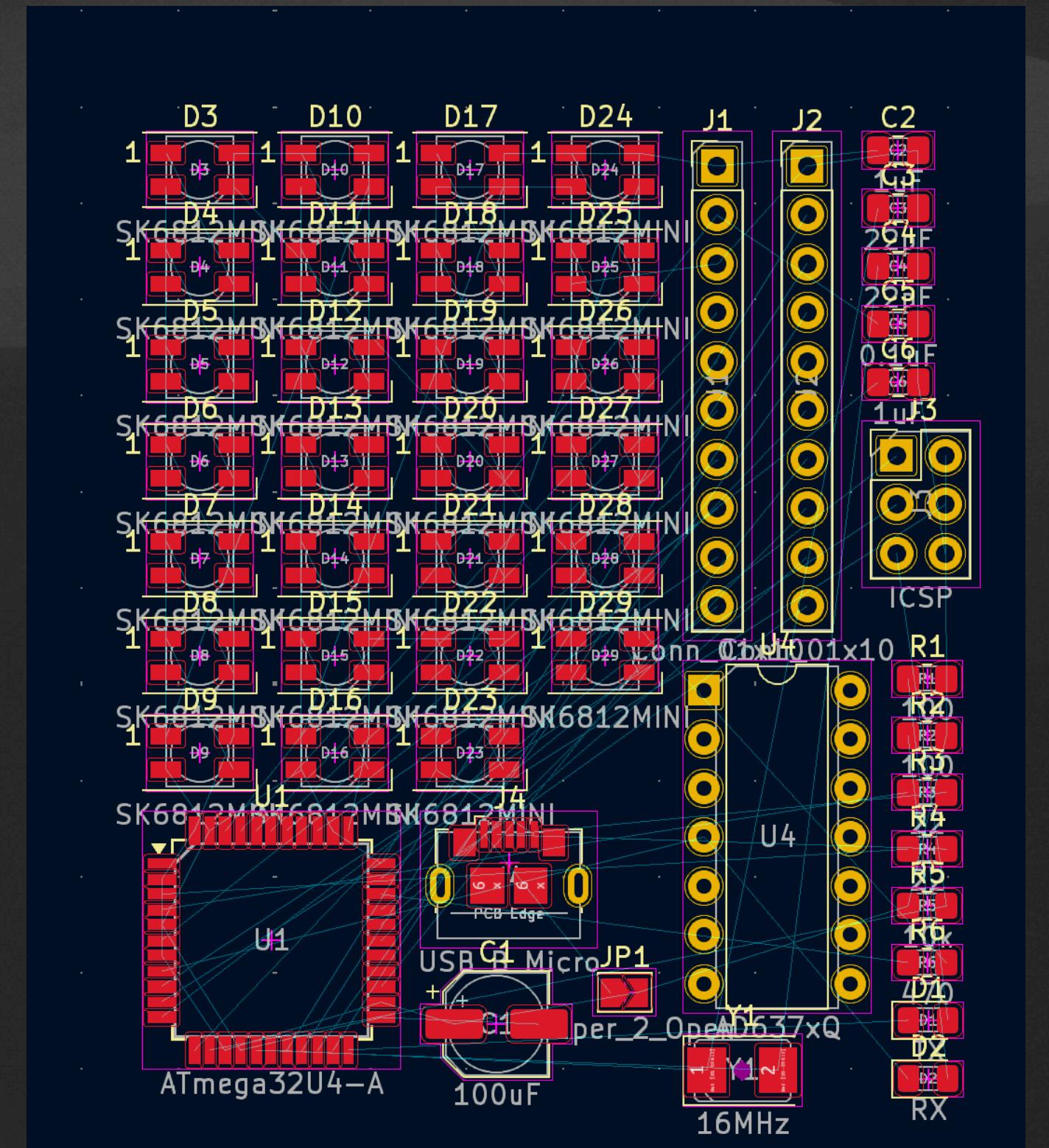
Your last step will be ground pours



Copper	
	Minimum clearance: 0.1 mm
	Minimum track width: 0.1 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

# 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



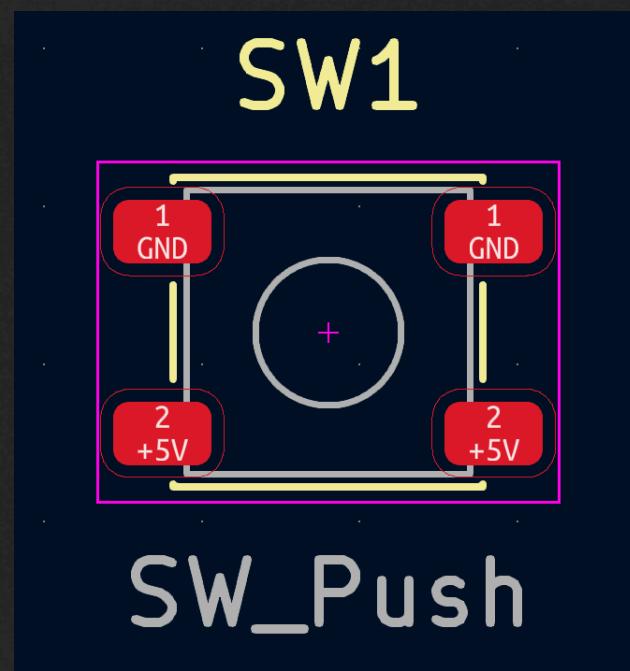
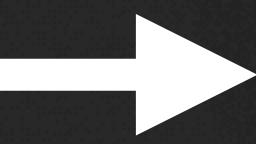
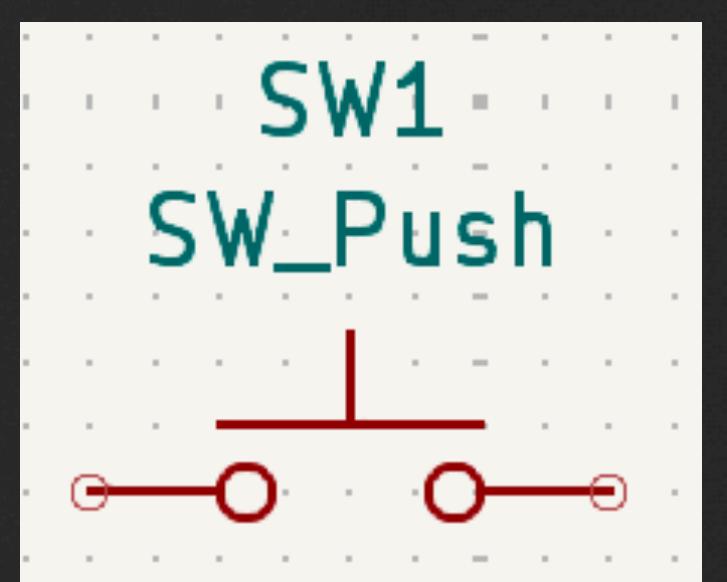
# footprints

## 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

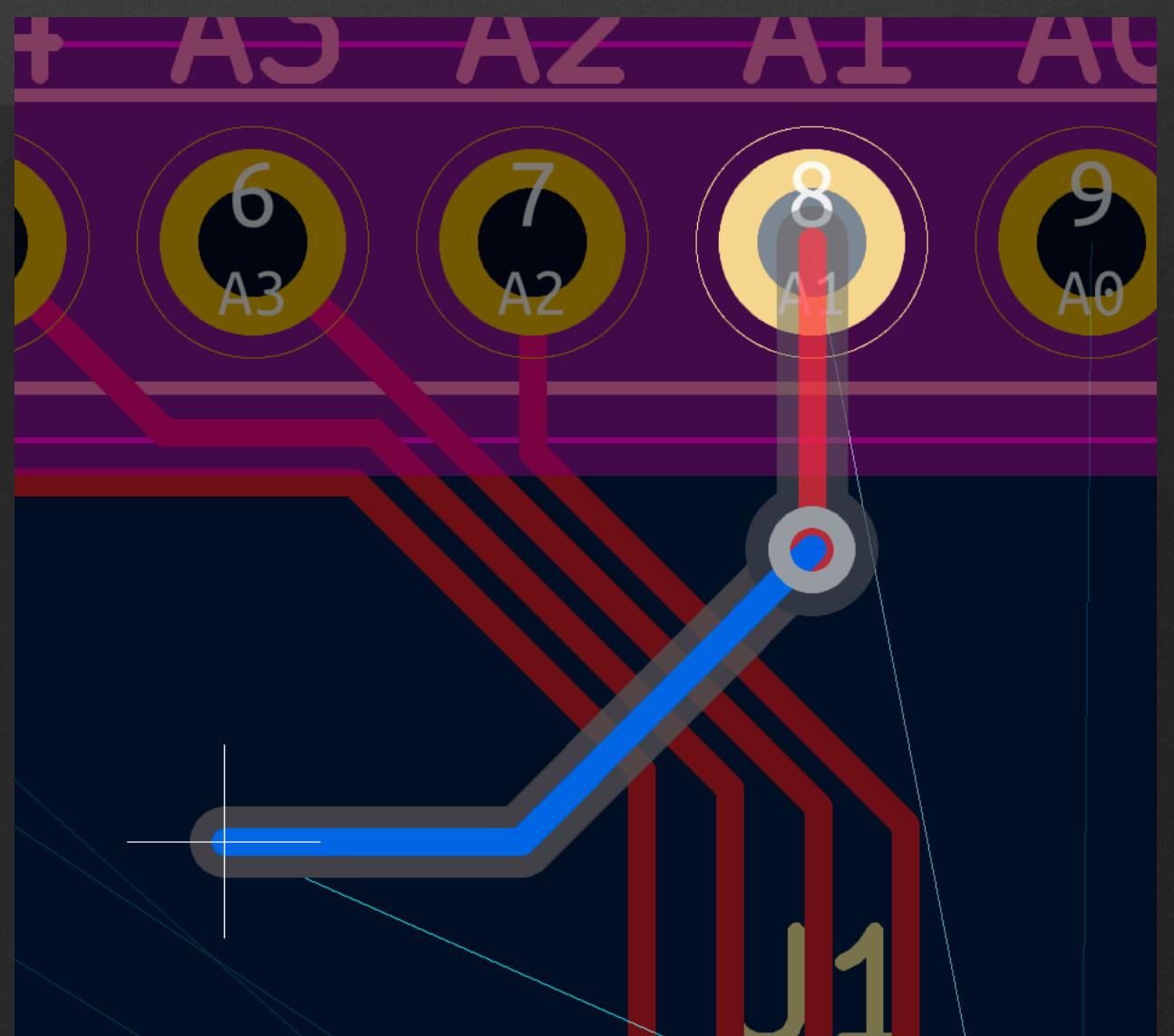
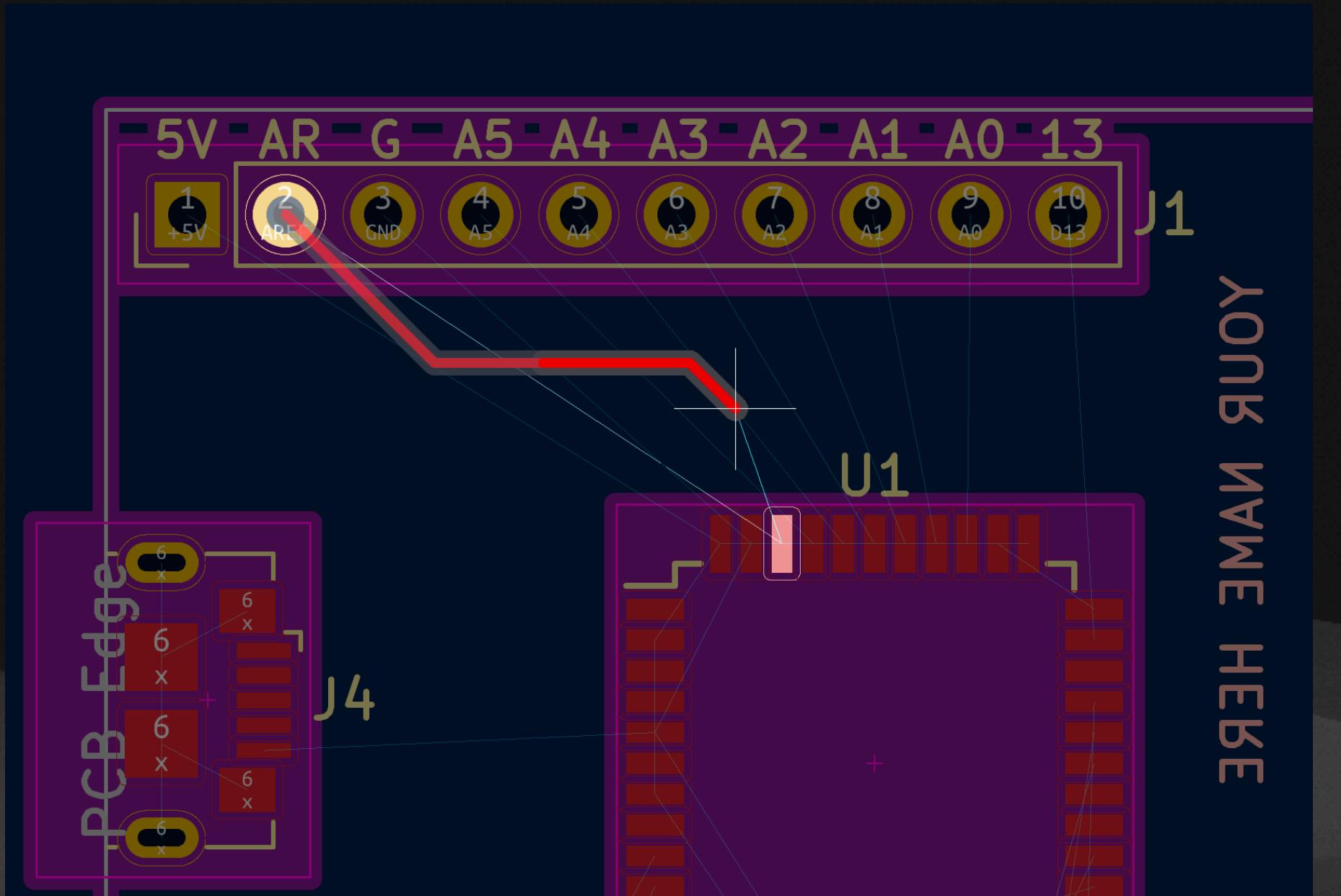
Name	Value
Reference	SW1
Value	SW_Push
Footprint	Button_Switch_SMD:SW_SPST PTS647_Sx38
Datasheet	~
Description	Push button switch, generic, two pins

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

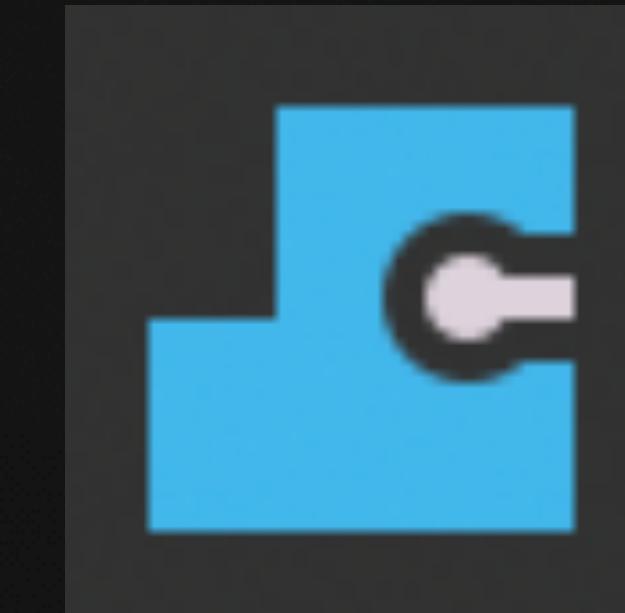


# routing board view

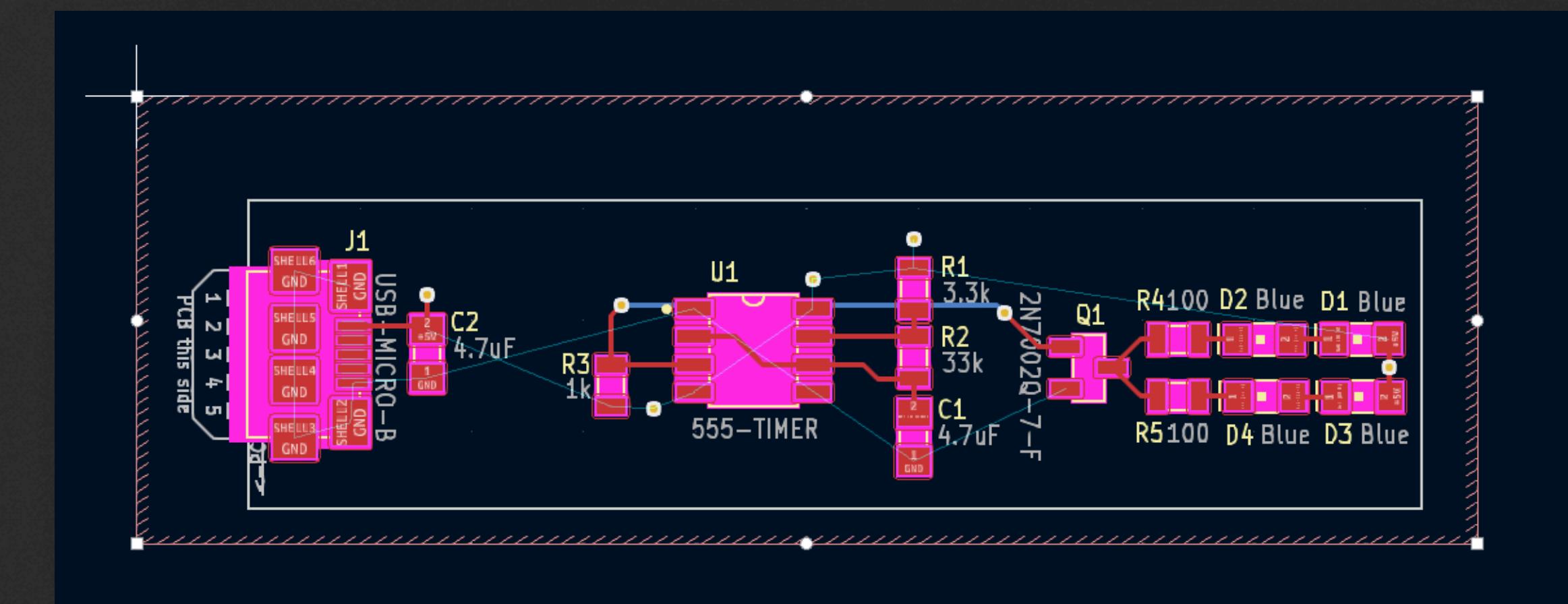
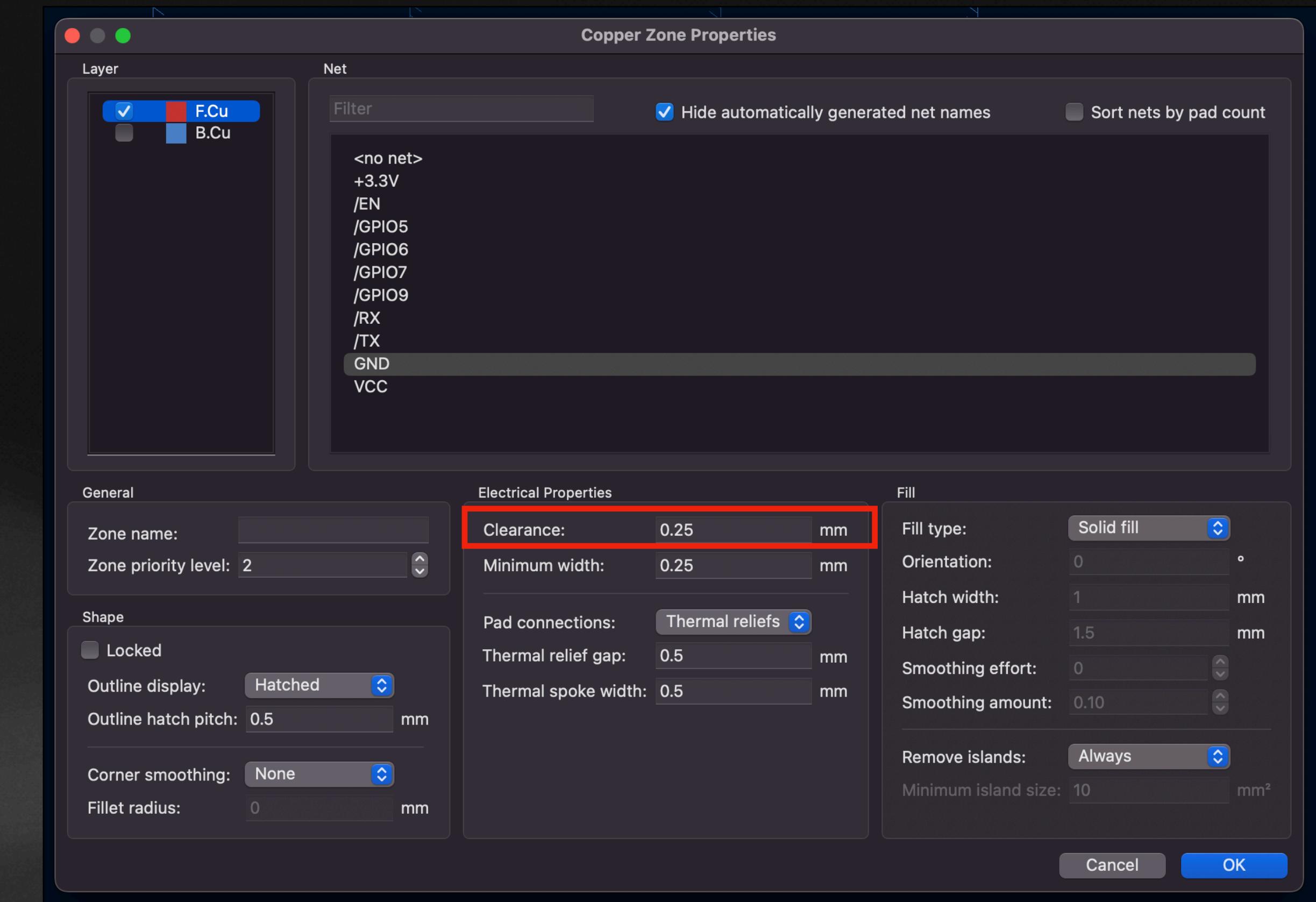
- Follow nets to see what needs to be connected
  - Use vias to simplify design
- 
- **X** => Route a single trace
  - **V** => Place a via (while routing)
  - **Cmd+Shift+V** => Place a via (anytime)
  - **U** => Select neighboring trace segments



# pours board view



- You will need to use multiple ground pours and Vcc pours
- 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**



# 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**

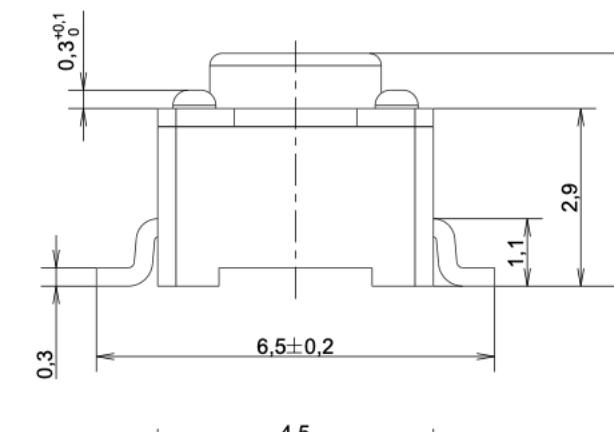
- Consumer
- Industrial control panels
- Computer products



**B**

**Tactile Switches**

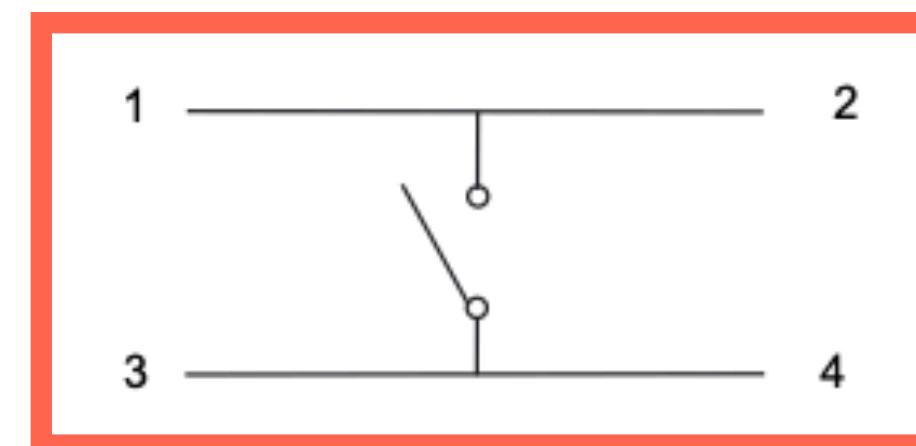
Part Number	Operating Force (gf)	Life	Height (H)
PTS647 SN38 SMTR2 LFS	100 +/- 50	100,000	3.8 mm
PTS647 SM38 SMTR2 LFS	180 +/- 50	100,000	3.8 mm
PTS647 SK38 SMTR2 LFS	250 +/- 50	100,000	3.8 mm
PTS647 SN50 SMTR2 LFS	100 +/- 50	100,000	5.0 mm
PTS647 SM50 SMTR2 LFS	180 +/- 50	100,000	5.0 mm
PTS647 SK50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

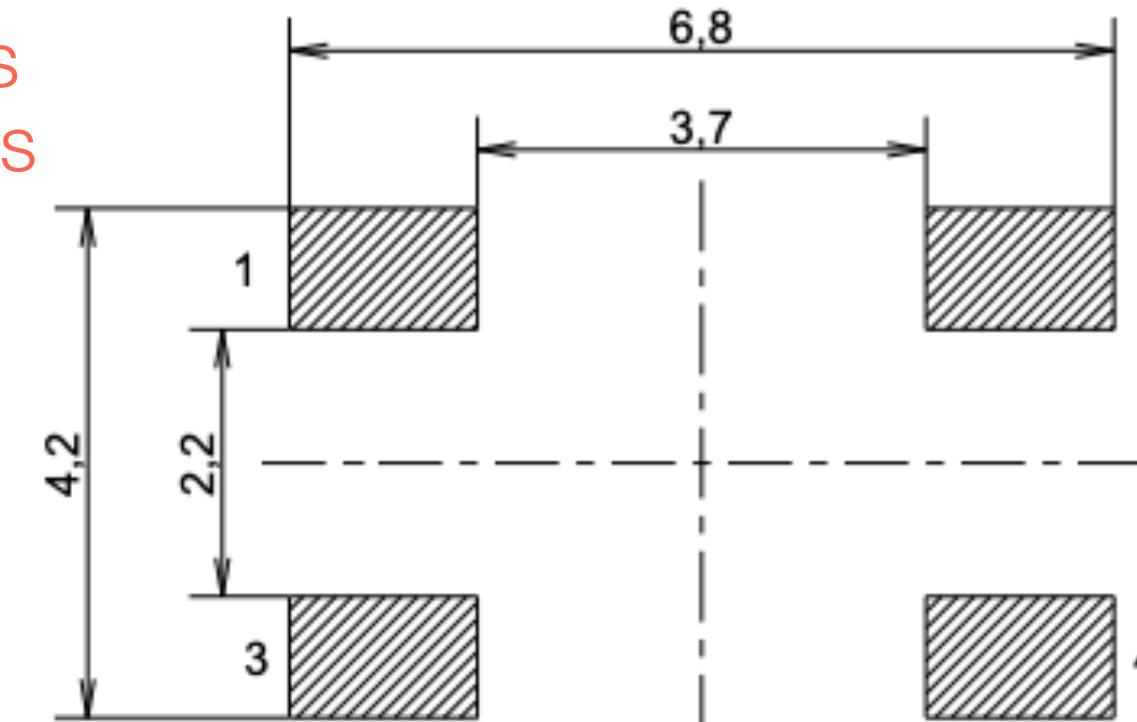
widely used and very suitable.  
Please contact your local C&K  
representative to discuss your  
application and the best switch  
solution.

SCHEMATIC

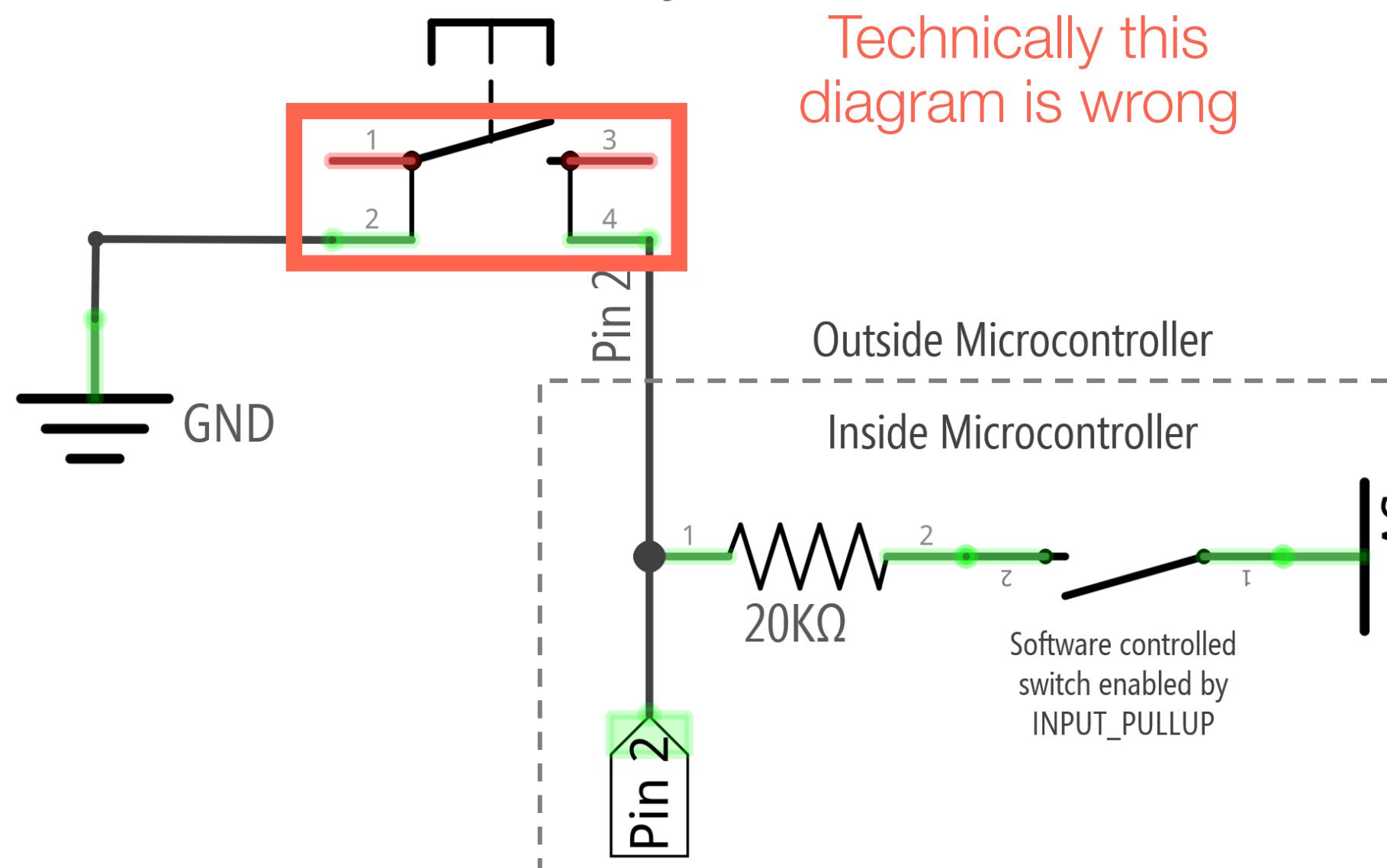


Our  
switch  
connects  
these pins

RECOMMENDED PCB LAYOUT



Push Button (4-Leg)



Technically this  
diagram is wrong

# 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