

Electronic Design Automation

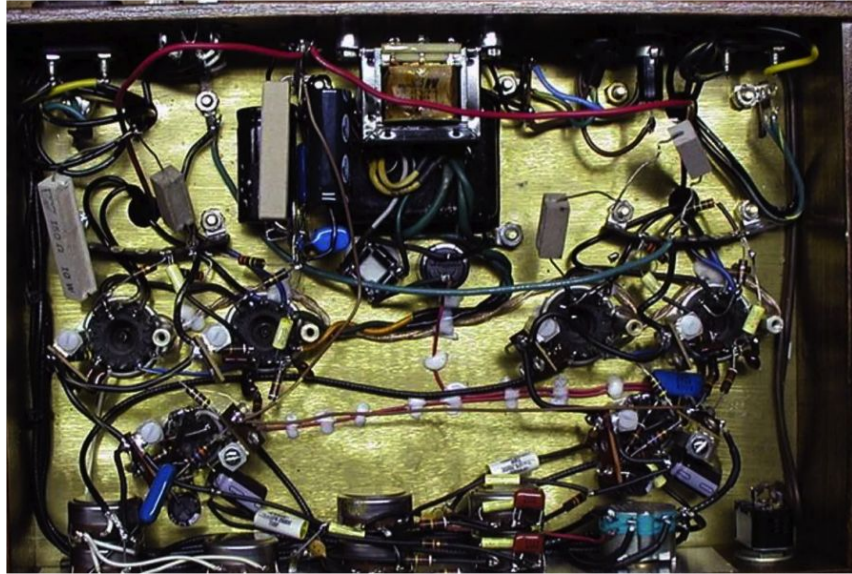
or

The art of untangling

Good/bad practice

What is a circuit board?

Circuit boards were invented to connect components **mechanically** and **electrically**



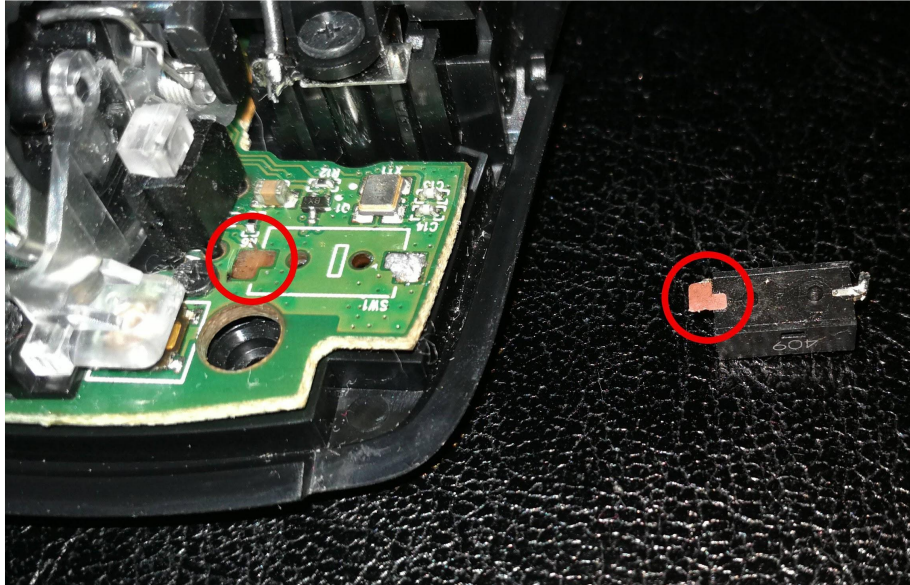
Tangle of wires



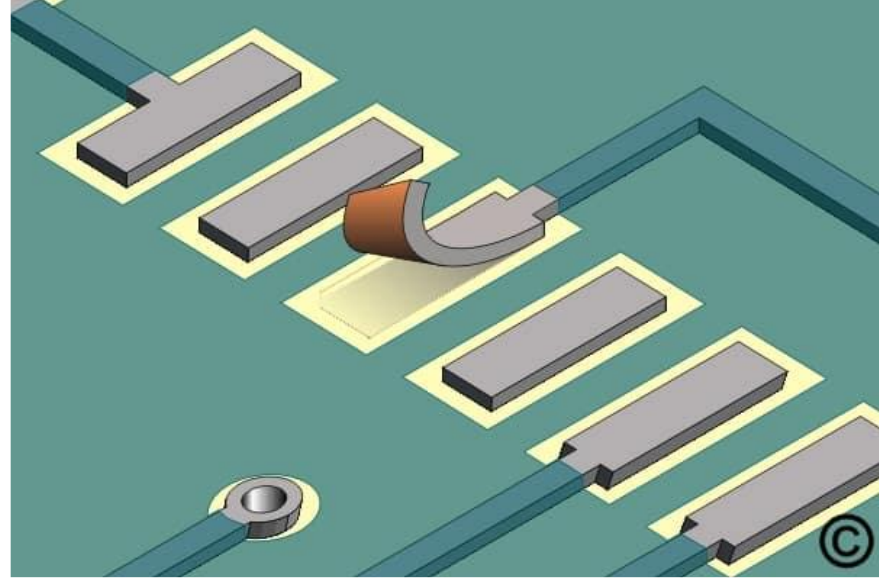
Circuit board

Circuit boards aren't magic

They can still break. Pads are especially vulnerable when strained



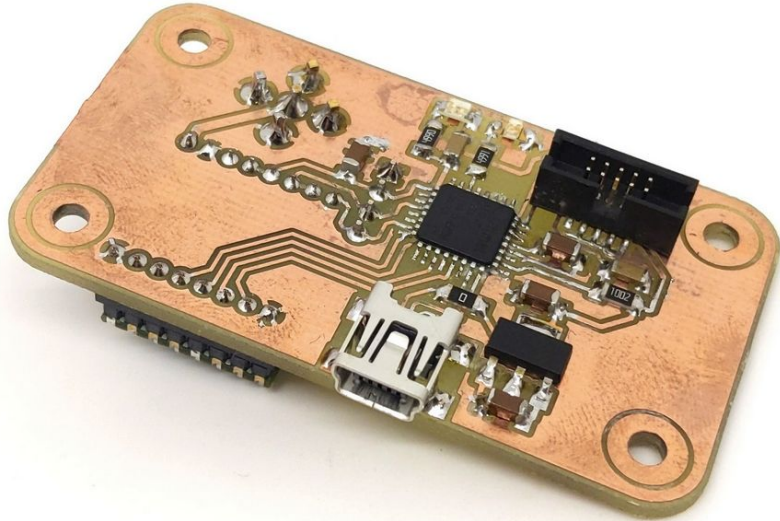
Ripped pad



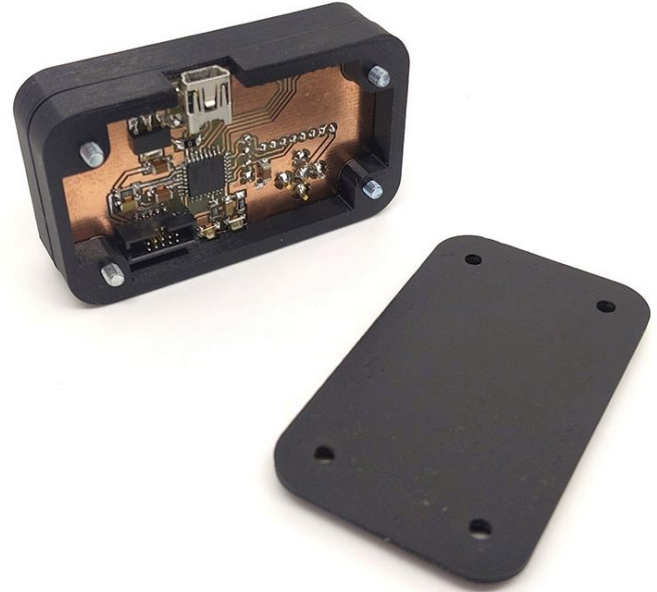
Oof

Your board needs a mechanical interface to other things

Mounting holes are the best way to secure your board into a project

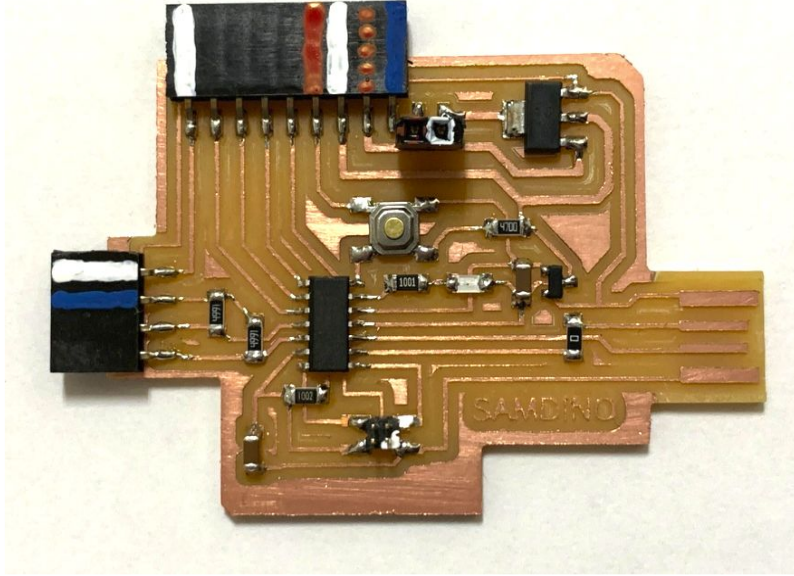


Mounting holes

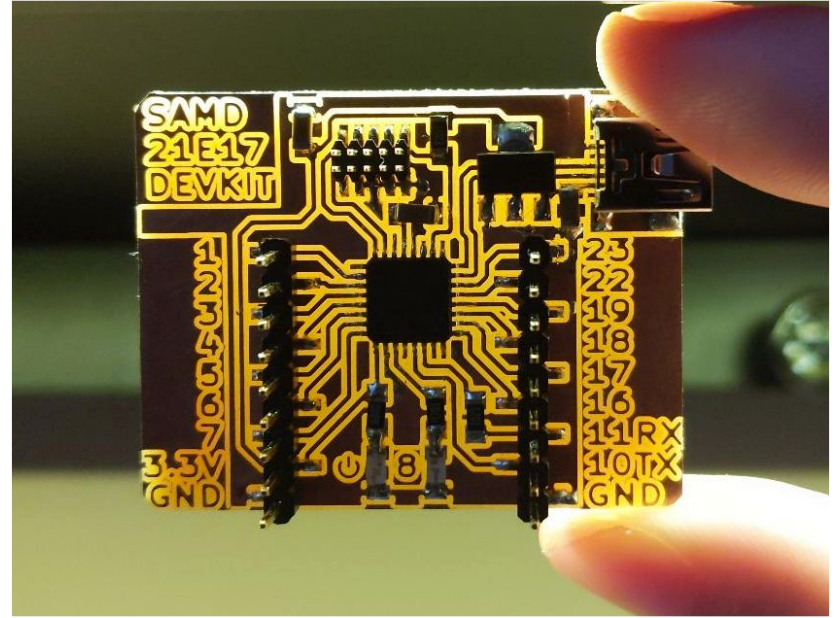


3D printed case

Labeling can save you time and prevent mishaps

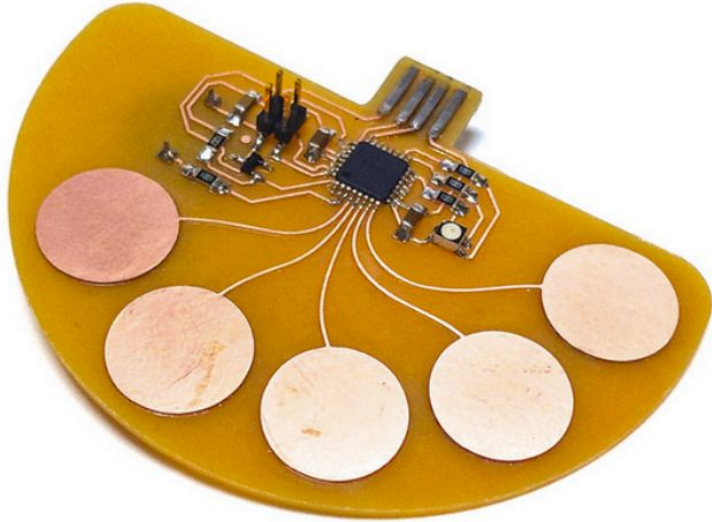


Color labels

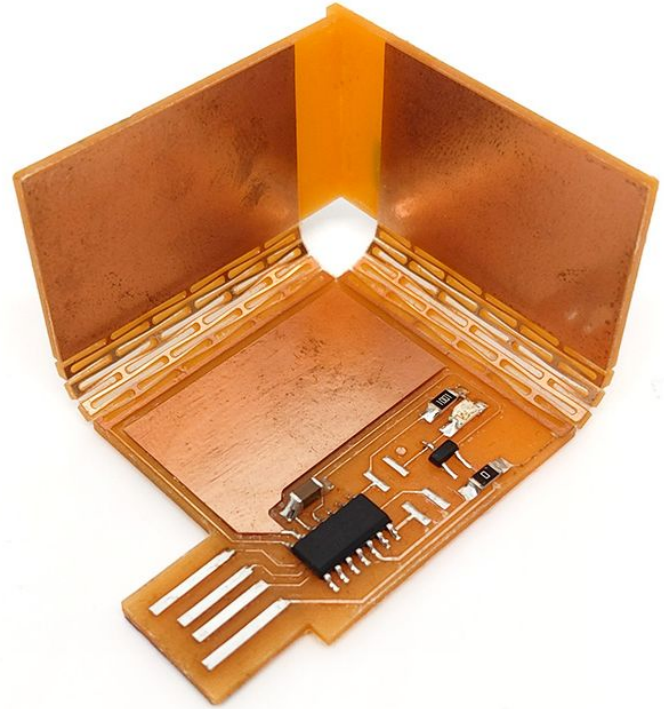


Text on board

Shape and function can be integrated directly into your board



Capacitive touch pads

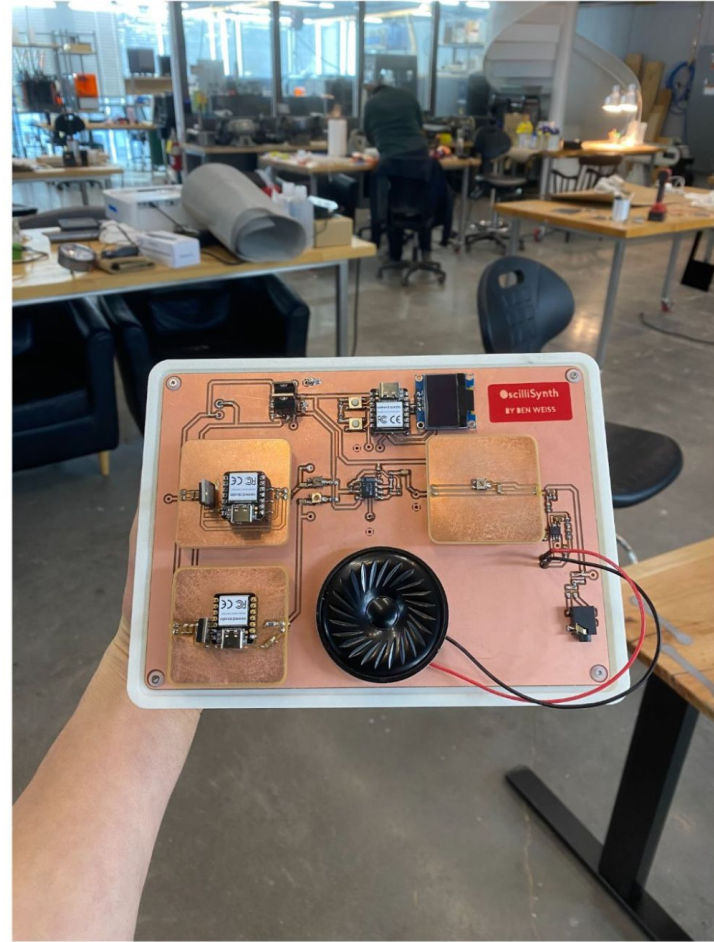


Mechanical flexures

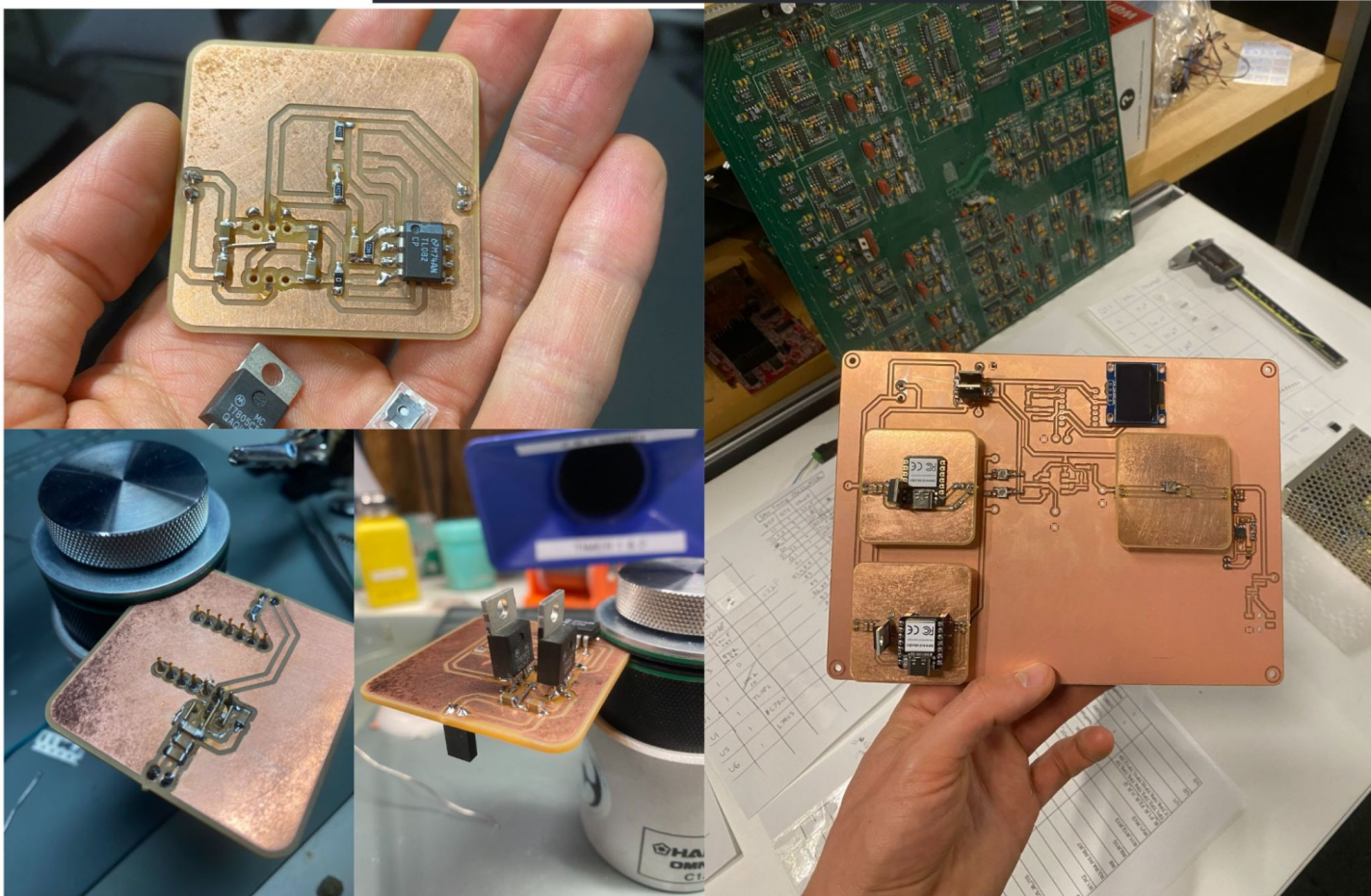
Breakout Boards

And why you should use them

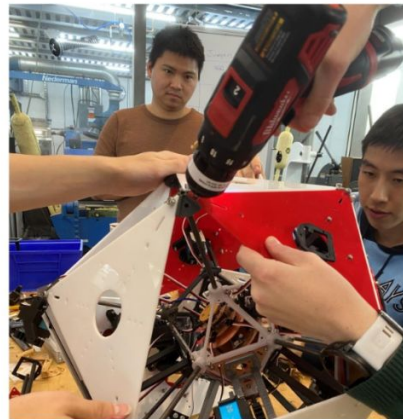
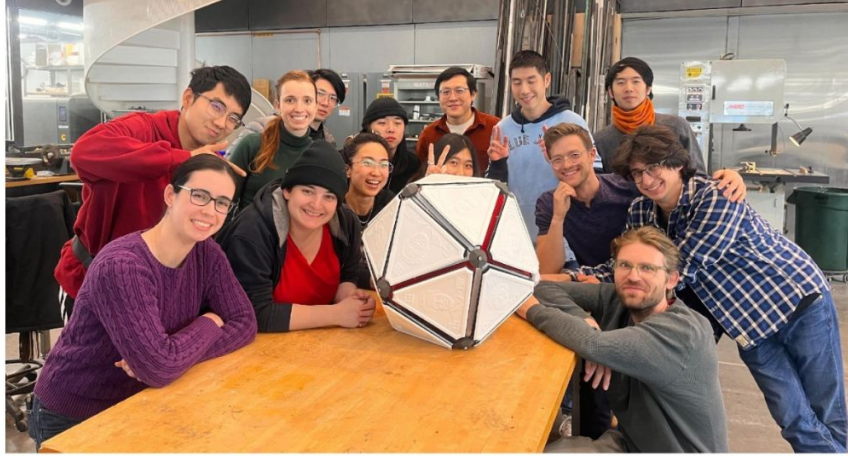
Motherboard, daughter boards



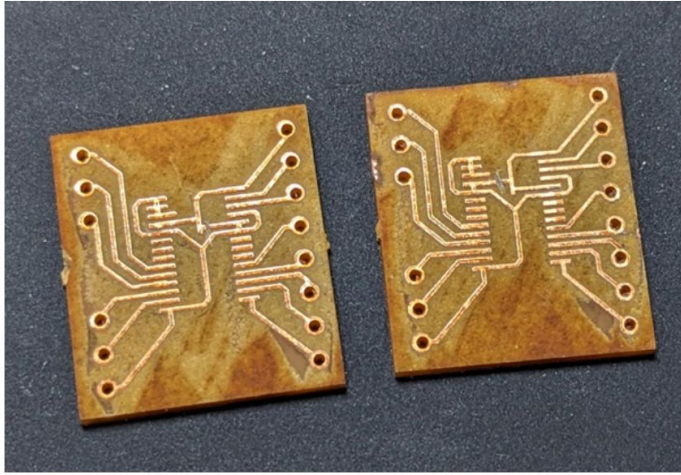
Repairability



Working on components of a project

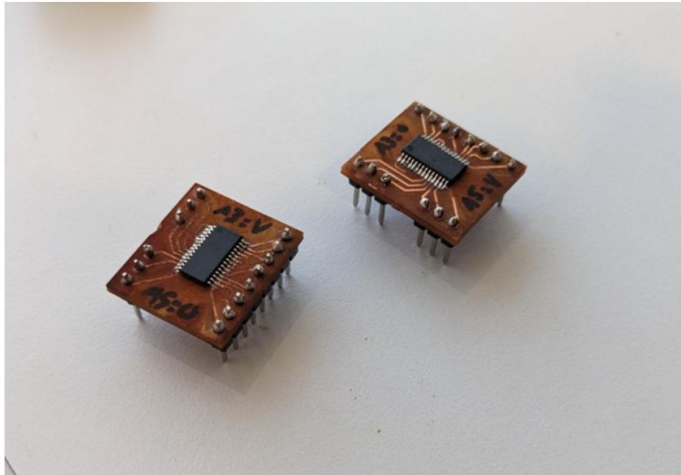


Breakout boards are neat



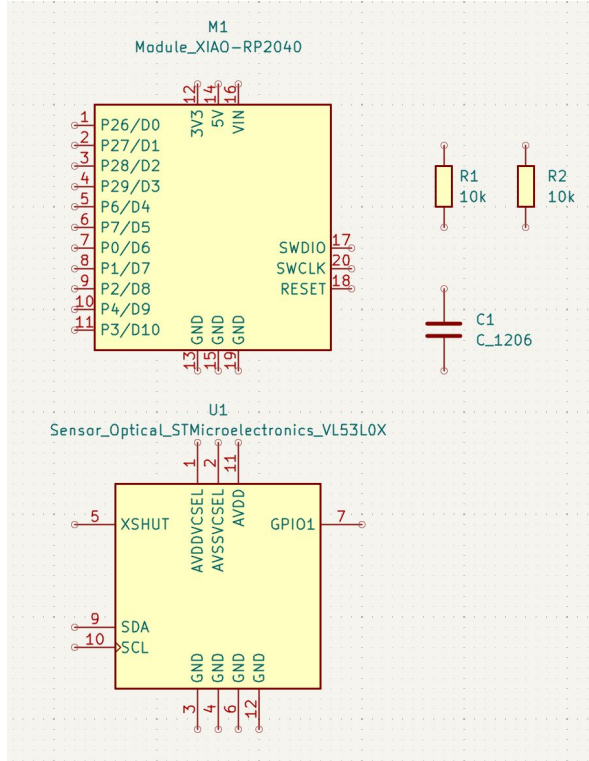
Tyler's Documentation

<https://fab.cba.mit.edu/classes/863.25/people/TylerJensenHill/Week11.html>

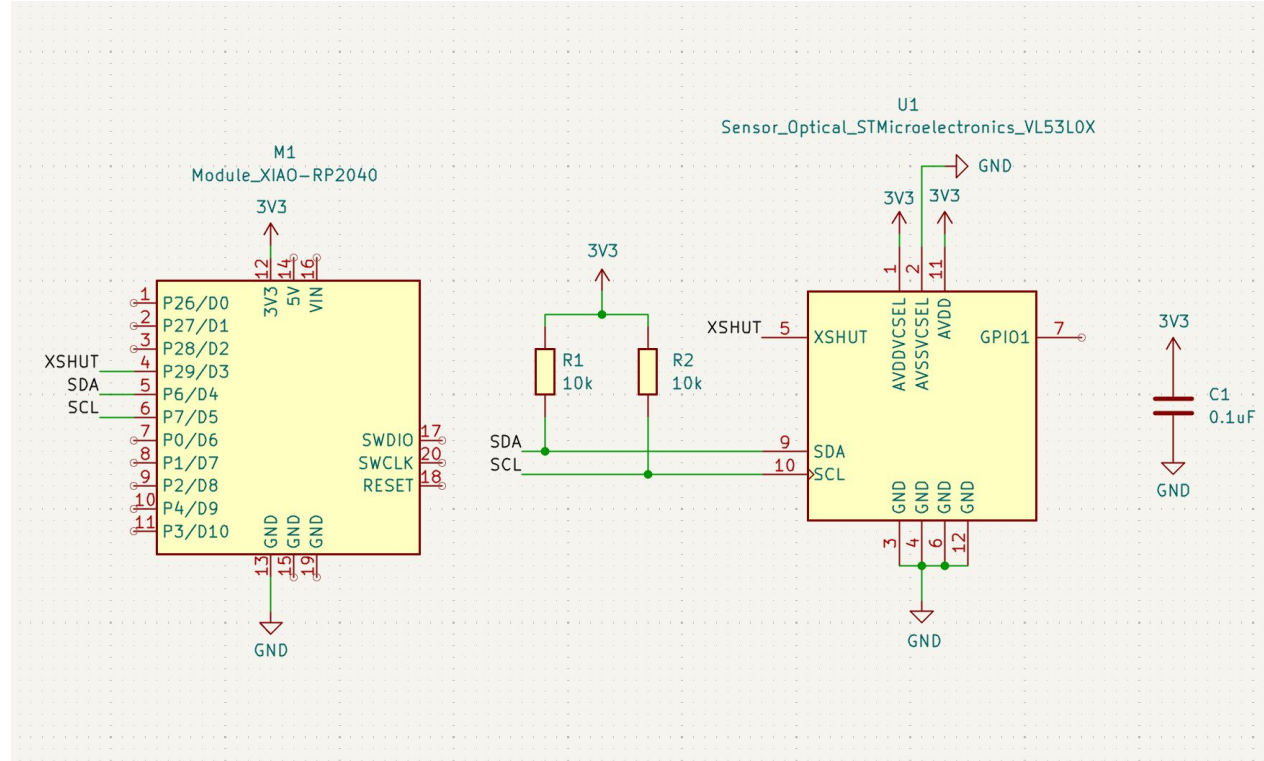


The EDA process

Step 1: the schematic

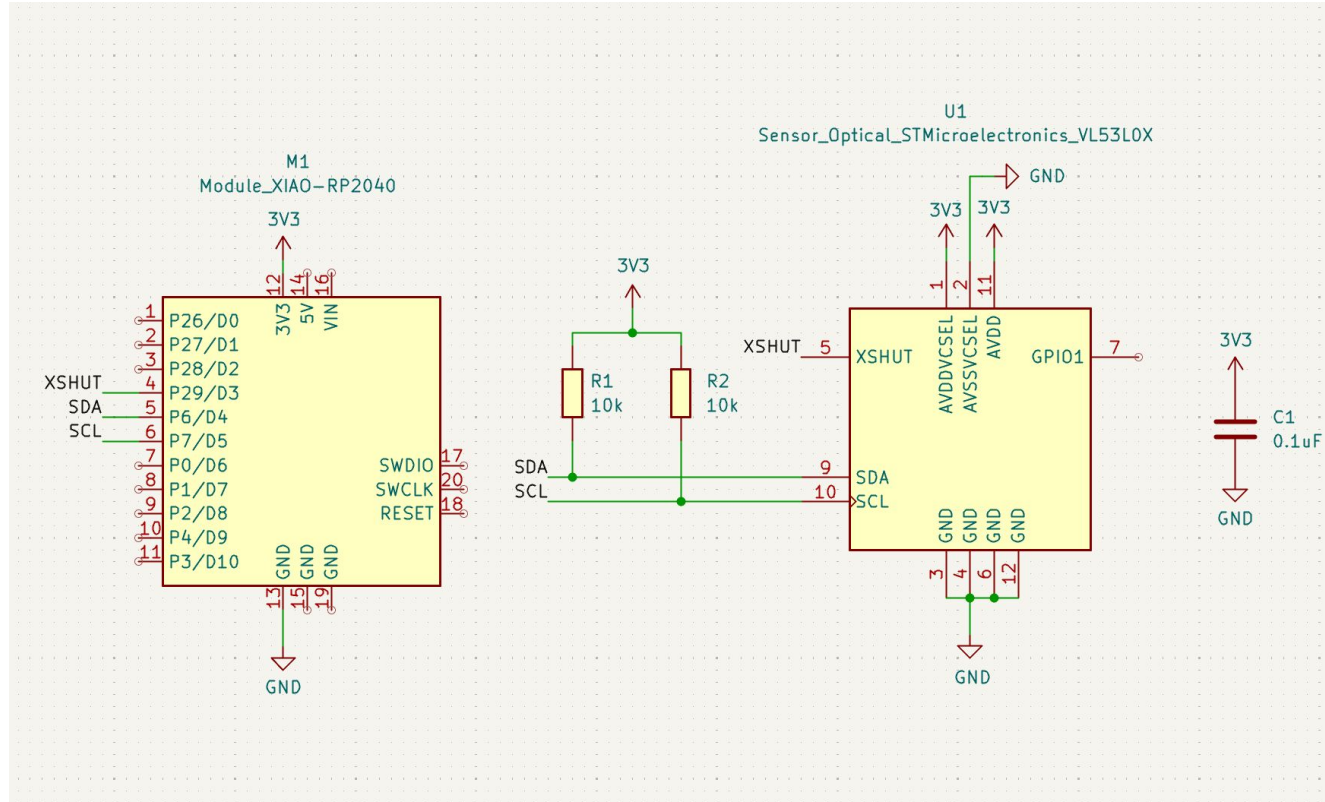


Components



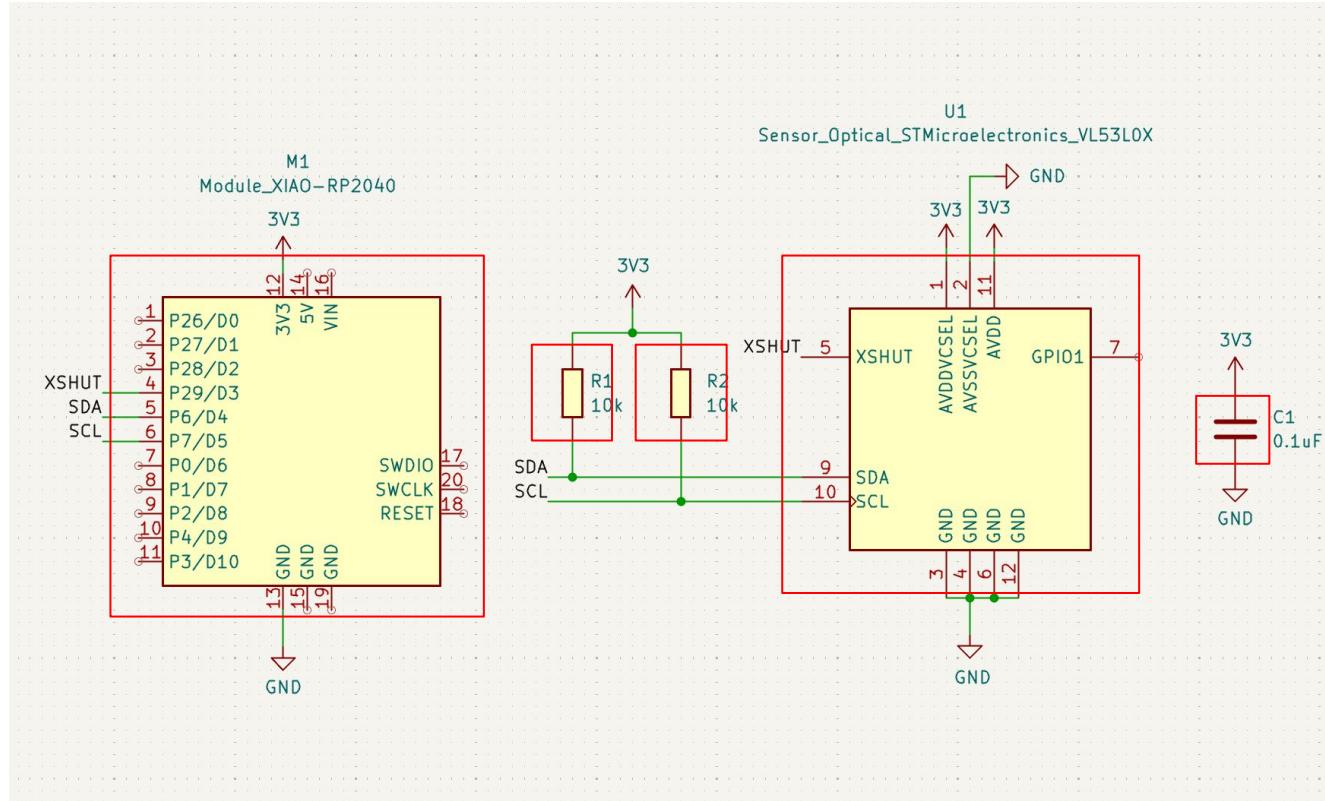
Completed schematic

How to read a schematic



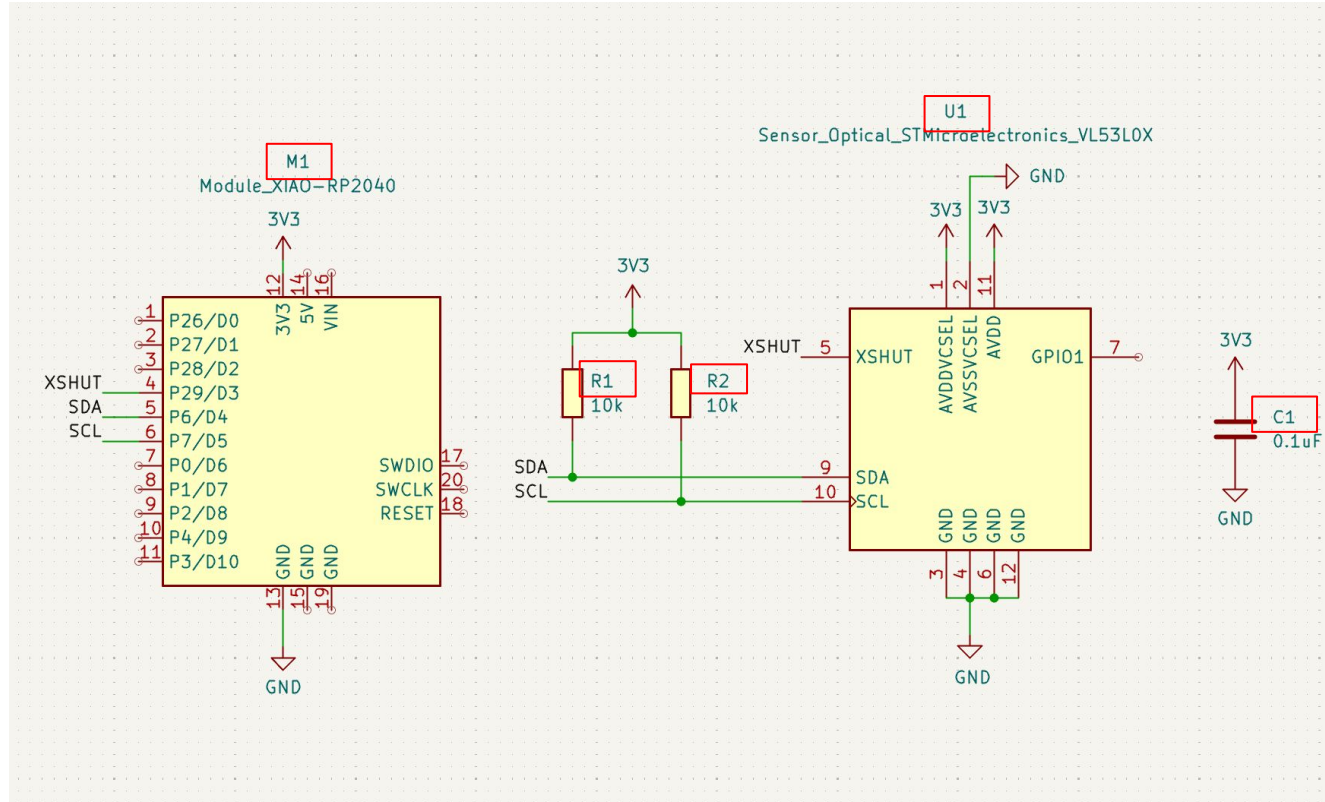
Symbols: abstract representation of components

Their shape is irrelevant. Only its pins matter, each with its own name and number



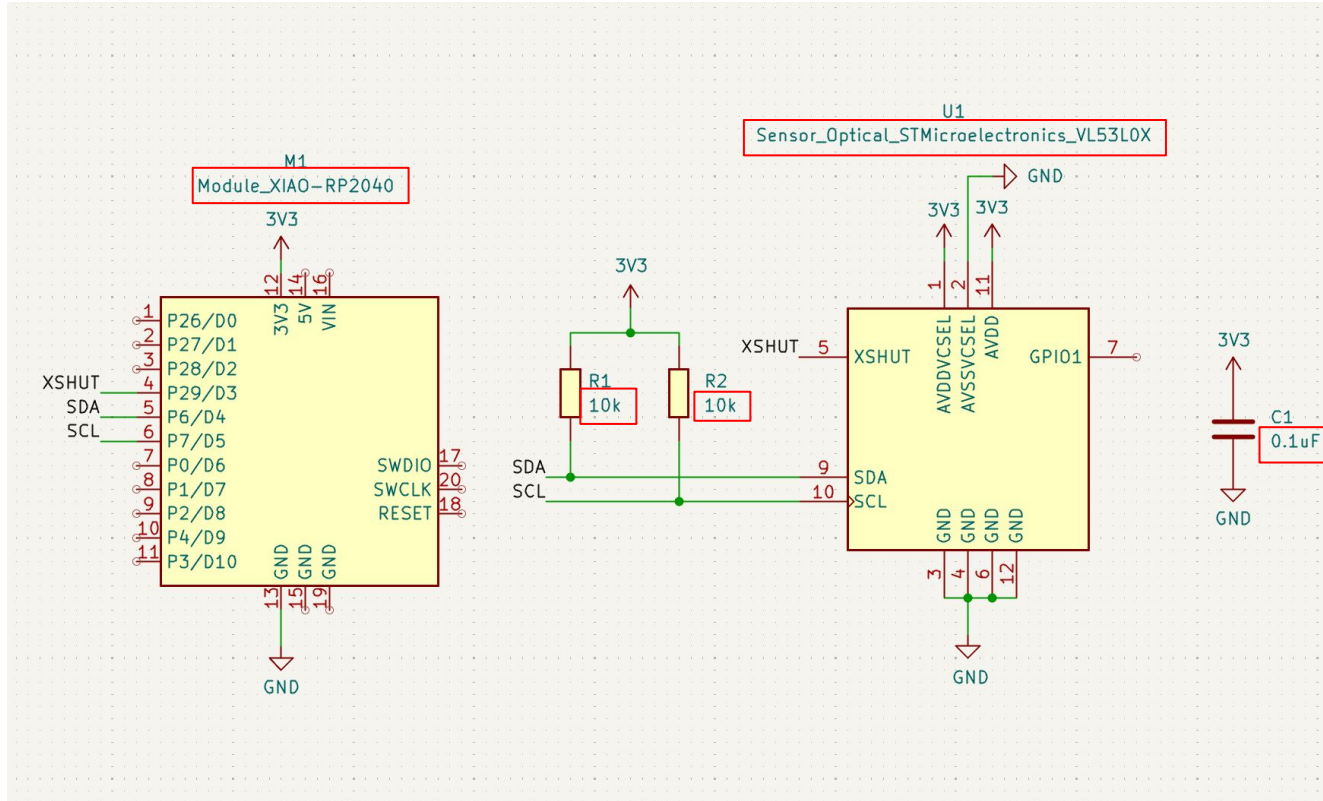
Symbol references

One letter followed by a number. Unique identifier for each symbol

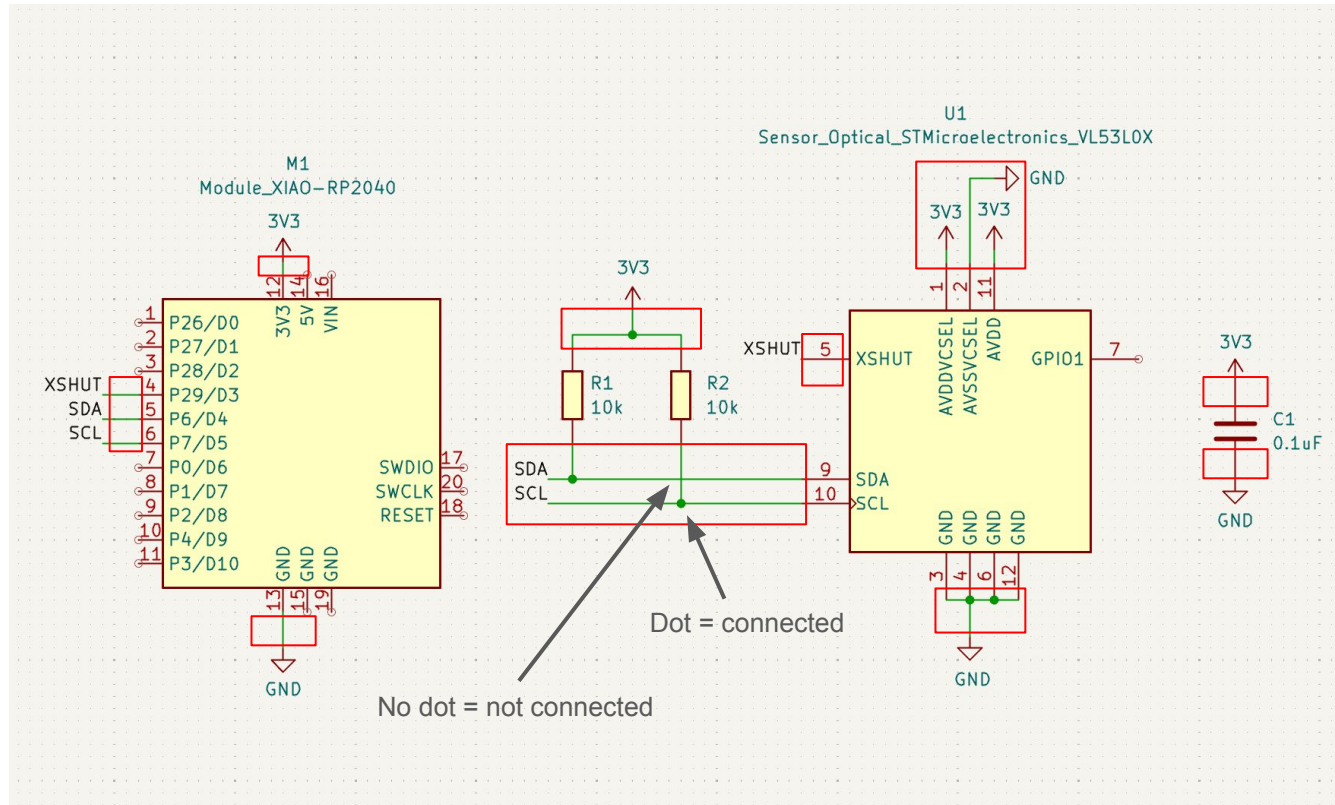


Symbol values

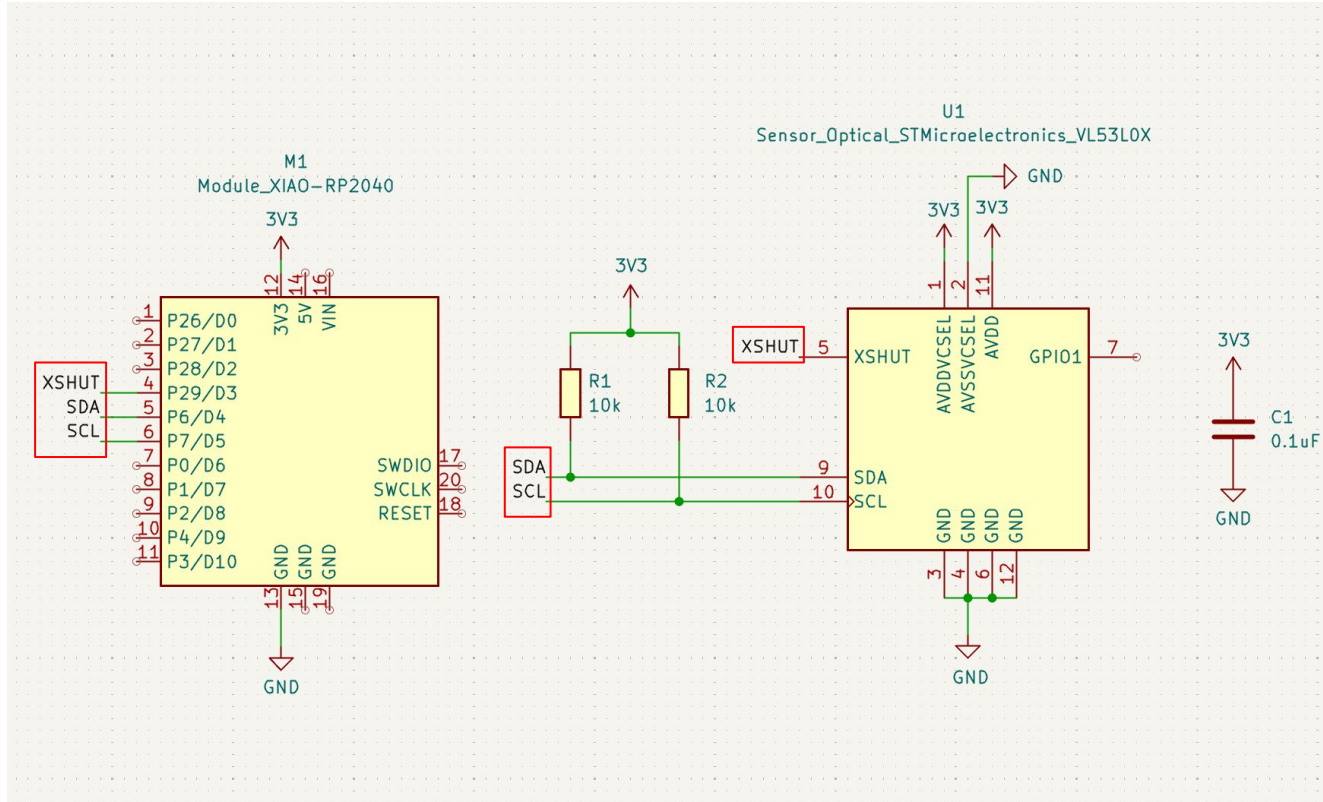
Additional information about a component, arbitrary



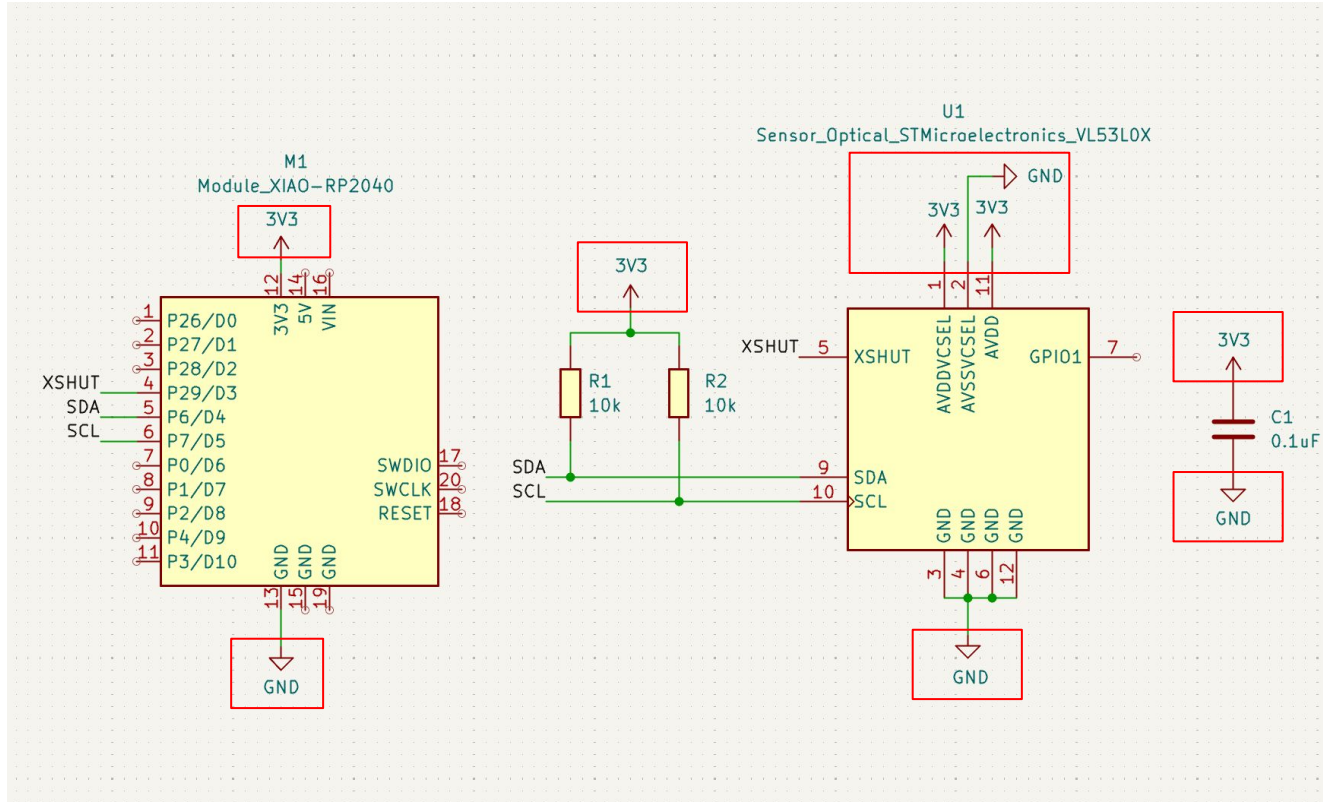
Wires: connections between pins



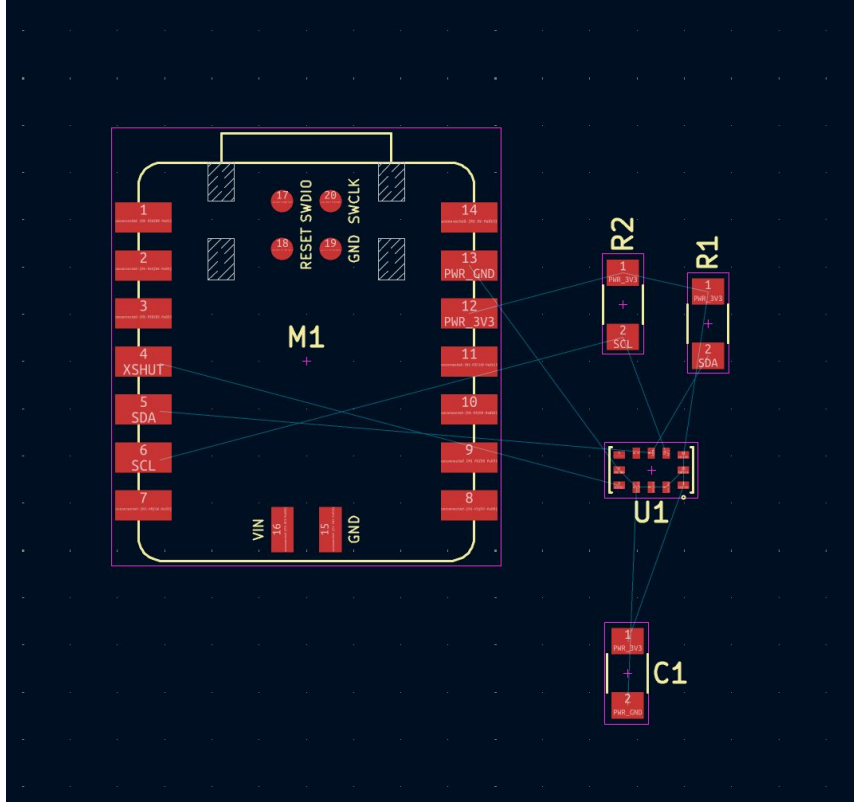
Labels: another way to connect pins



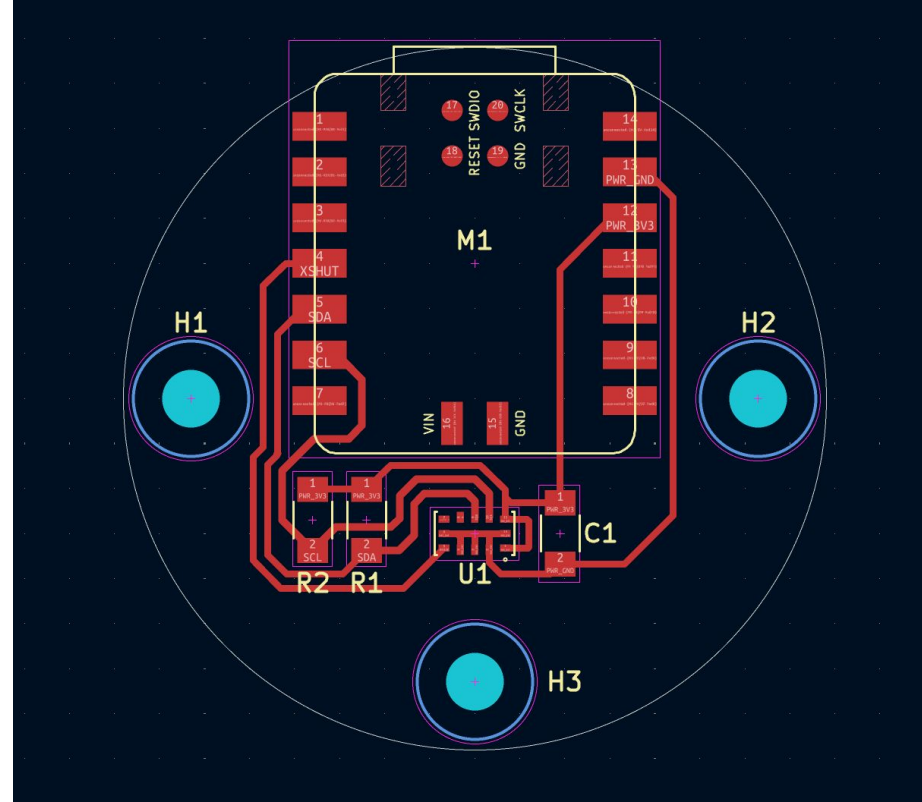
Power symbols: same as labels, just fancier



Step 2: board design



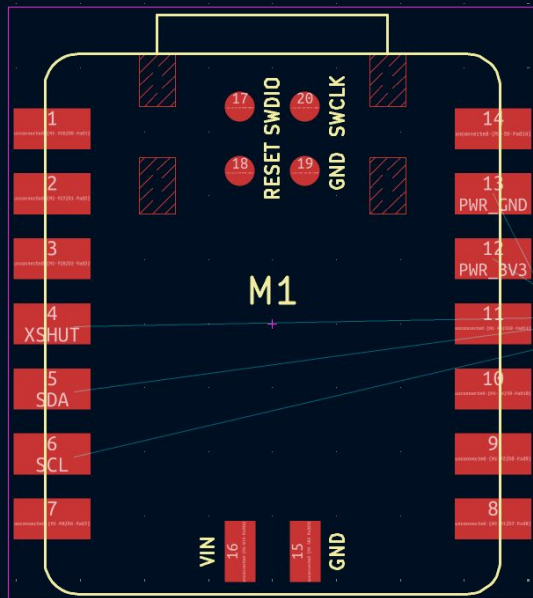
Components



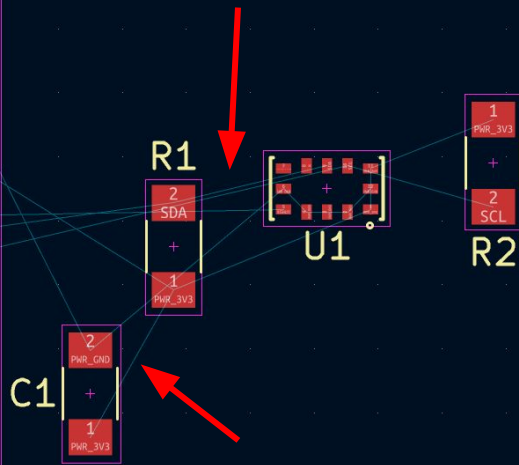
Completed board

Your job: choosing component **placement** and adding **traces**

Rastnests show you which pin wants to be connected to which other pads.

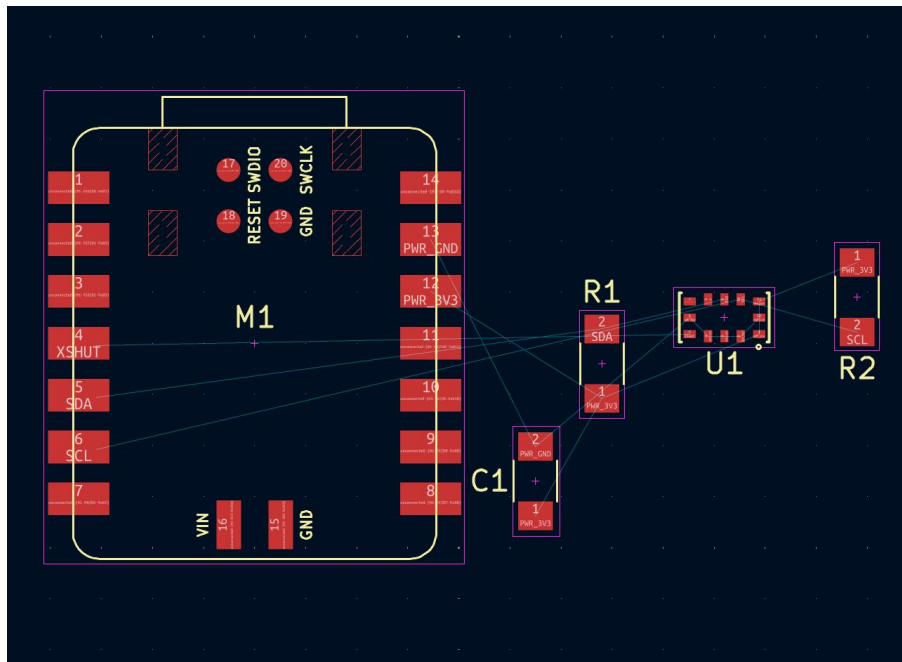


Tangled ratsnest: probably worth relocating components

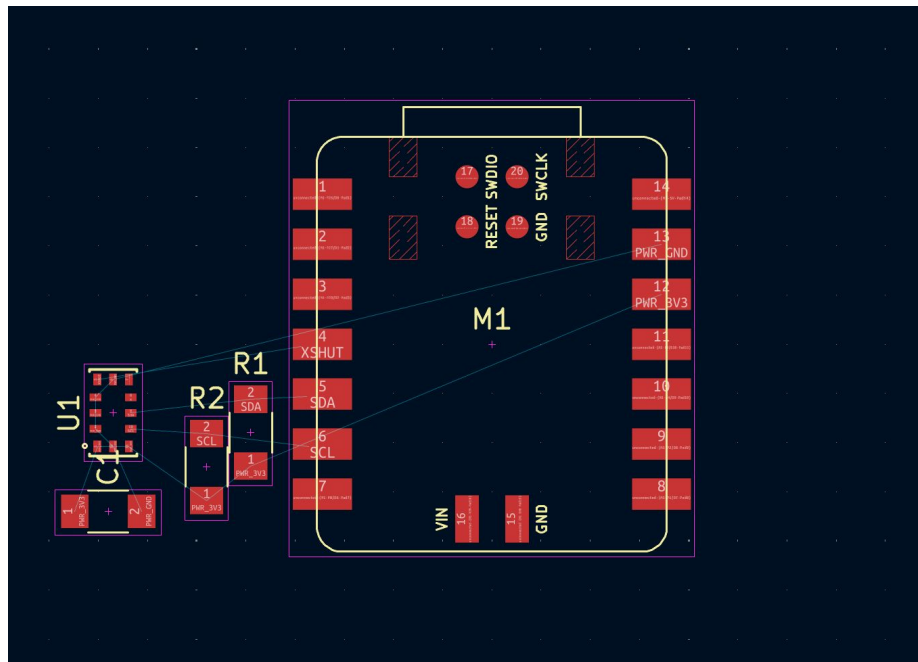


A connection should be made here

Try different placements until the ratsnest simplifies



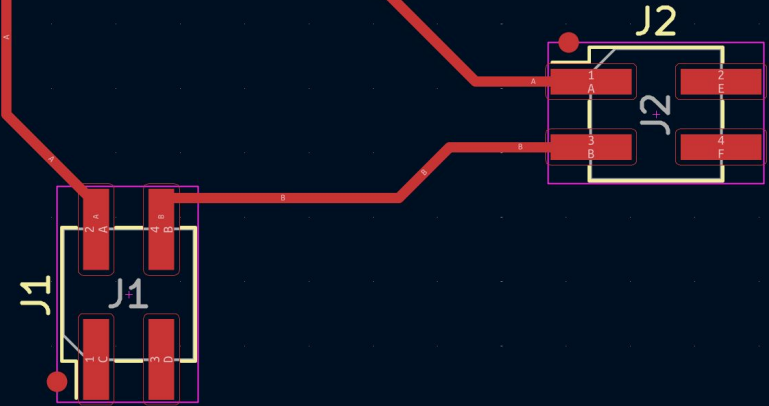
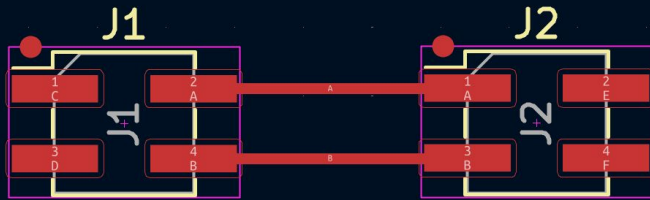
Not ideal



Better

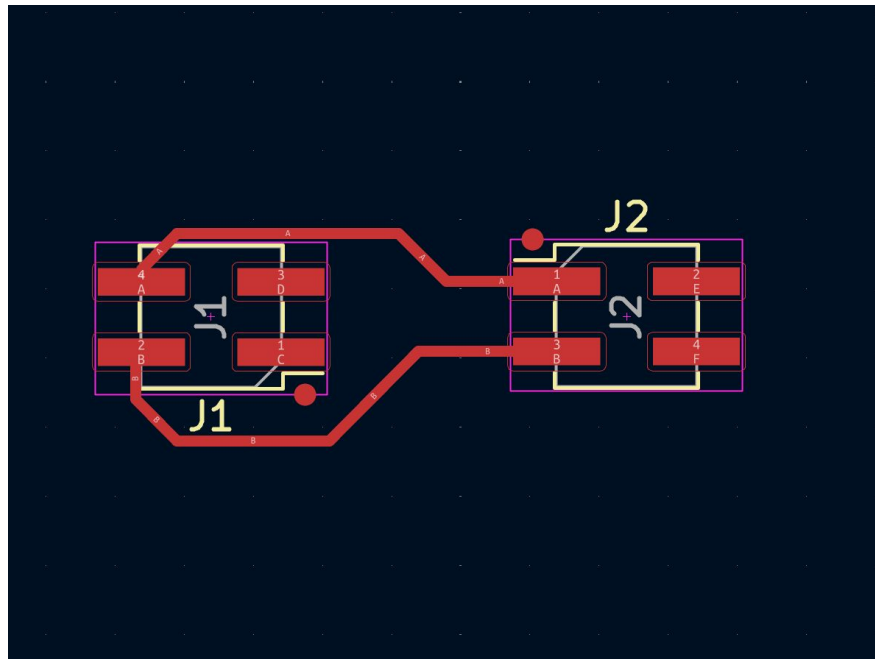
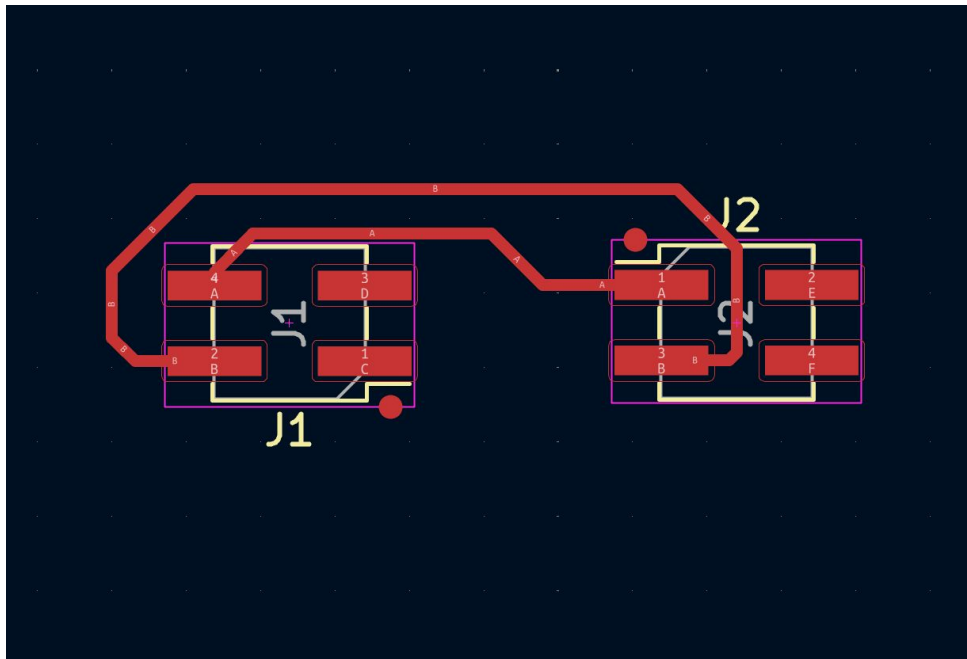
When adding traces, think about **topology**, not **shape**

These two situations are **topologically equivalent**



When adding traces, think about **topology**, not **shape**

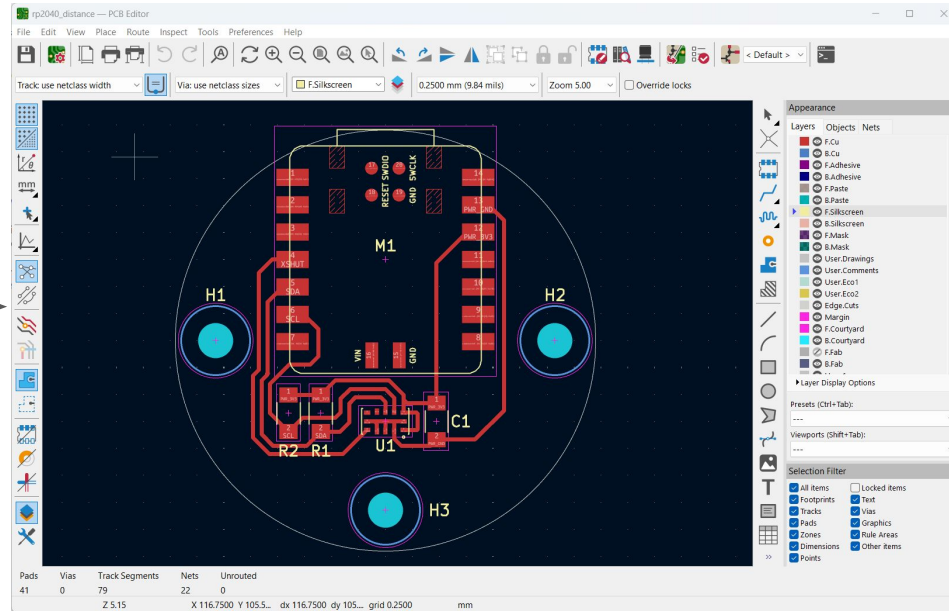
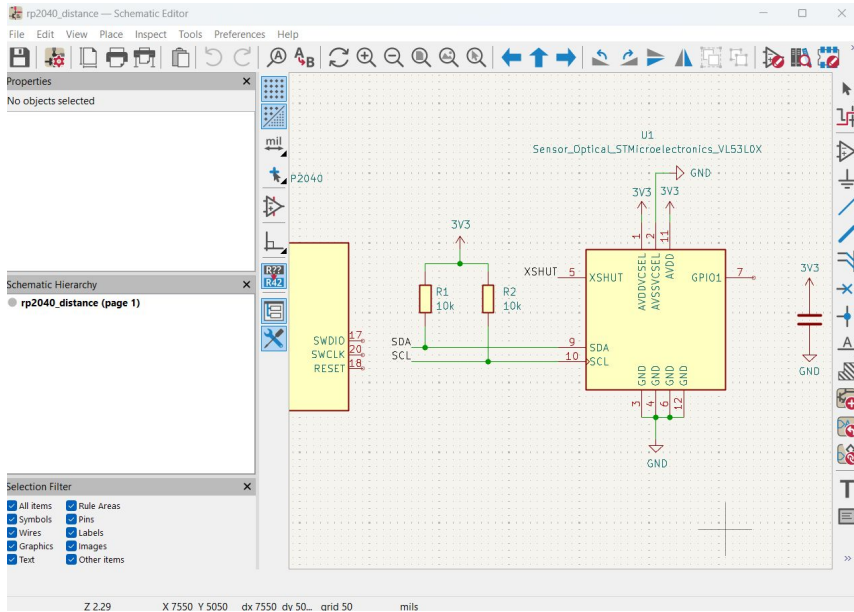
These two situations are **not topologically equivalent**: pins C and D are now encompassed



Good design involves going back and forth

You will switch often between **schematic** and **board** design. This is normal and encouraged.

The two are kept in **sync** by your EDA software, and changes are reflected accordingly



KiCAD demo