From zero to PCB hero

A small intro to KiCad

PCB design is my passion

SEG1
R2
LED1
R1
5v
gnd
J2
J1

The plan for today

- Get KiCad installed

- Baby's first KiCad project

- Exporting for manufacture

- Organizing large projects

- Best practice, cheatsheets and all that good stuff

The important questions

- Where is it?

- What is it?

- Why should we use it?

Where is KiCad?

- https://www.kicad.org/download/

What is KiCad?

- Schematic design
- PCB design
- Bill of materials
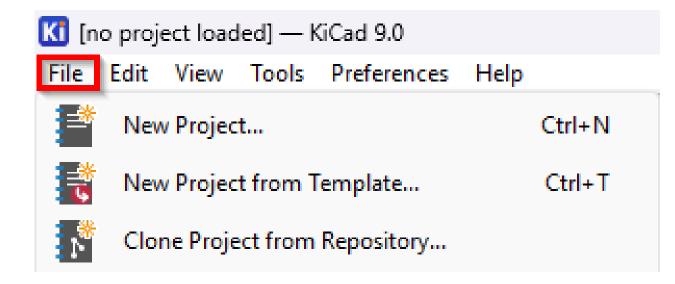- Simulation
- Much much more

Why is KiCad?

- No license needed

- Open source

- Runs on every operating system

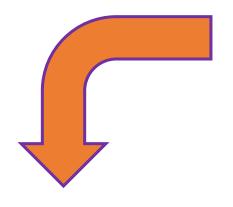- It's Free!

The design flow in broad terms

- Schematic

- Pcb layout

- (Crying because you forgot footprints)

- More Pcb layout

- Ordering / Routing yourself

- Finished physical pcb

Setting up a new KiCad project

- Empty project
- From template
- Clone from Repository

Schematic setup

Page Settings

Paper

Size:
A4 210x297mm

Orientation:
Landscape

Custom paper size:
Height: 11000 mils
Width: 17000 mils

☐ Export to other sheets

Preview

Drawing Sheet

File: [   ] 📁

Title Block

Number of sheets: 1   Sheet number: 1

Issue Date: 2025-03-13   <<<   18/03/2025 ☐ Export to other sheets
Revision: 1 ☐ Export to other sheets
Title: 2 Mic Array ☐ Export to other sheets
Company: ESD211 ☐ Export to other sheets
Comment1: ☐ Export to other sheets
Comment2: Jacob S. Wanzenried ☐ Export to other sheets
Comment3: ☐ Export to other sheets
Comment4: ☐ Export to other sheets
Comment5: ☐ Export to other sheets
Comment6: ☐ Export to other sheets
Comment7: ☐ Export to other sheets
Comment8: ☐ Export to other sheets
Comment9: ☐ Export to other sheets

OK   Cancel

Jacob S. Wanzenried

ESD211
Sheet: /
File: microphone_array_2mic.kicad_sch
Title: 2 Mic Array
Size: A4   Date: 2025-03-13   **Rev: 1**
KiCad E.D.A. 9.0.0   Id: 1/1

PCB setup

- Board layers
- Routing constraints
- Track and via sizes
- Net classes

- And much more…

Netclasses
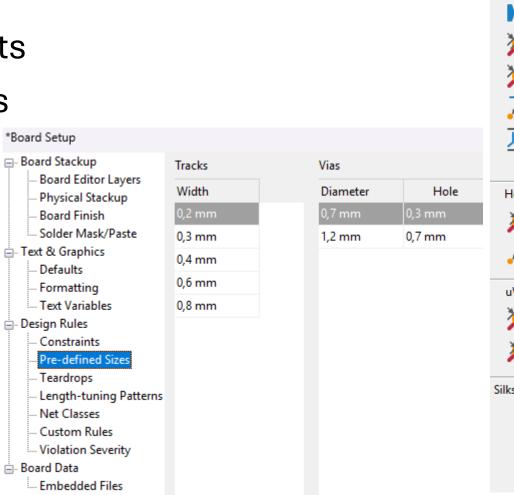
Name	Clearance	Track Width	Via Size	Via Hole	µVia Size	uVia Hole	DP Width	DP Gap	PCB Color
Default	0,2 mm	0,4 mm	1,2 mm	0,7 mm	0,3 mm	0,1 mm	0,2 mm	0,25 mm	
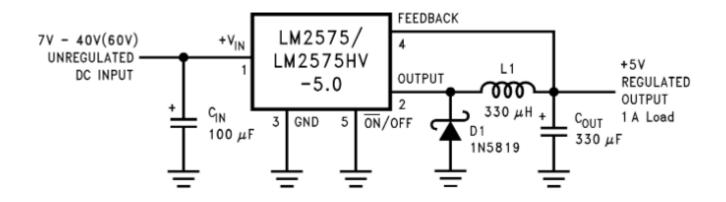
Copper

Minimum clearance:	0,1	mm
Minimum track width:	0,1	mm
Minimum connection width:	0	mm
Minimum annular width:	0,15	mm
Minimum via diameter:	0,5	mm
Copper to hole clearance:	0,25	mm
Copper to edge clearance:	0,5	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

*Board Setup

- Board Stackup
  - Board Editor Layers
  - Physical Stackup
  - Board Finish
  - Solder Mask/Paste
- Text & Graphics
  - Defaults
  - Formatting
  - Text Variables
- Design Rules
  - Constraints
  - Pre-defined Sizes
  - Teardrops
  - Length-tuning Patterns
  - Net Classes
  - Custom Rules
  - Violation Severity
- Board Data
  - Embedded Files

Tracks

Width
0,2 mm
0,3 mm
0,4 mm
0,6 mm
0,8 mm

Vias

Diameter	Hole
0,7 mm	0,3 mm
1,2 mm	0,7 mm

Simple buck converter with LM2575

A small starter project

Only 5 components ☺
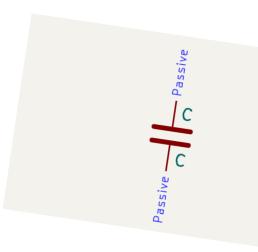
Let us go into KiCad

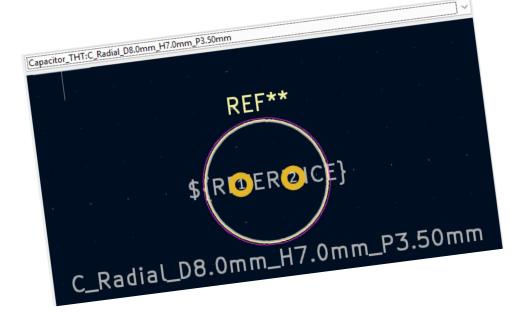But first, a 3.14 minute break

Inserting a symbol

- Press A
- Search for the symbol/part you want to insert
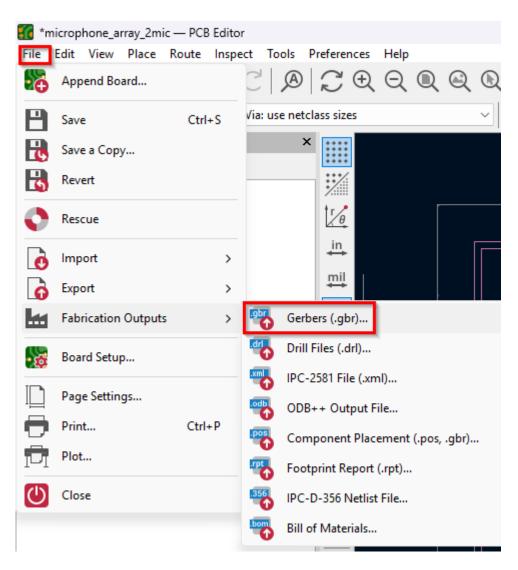- **<u>Remember to select a footprint</u>**

Schematic editor shortcuts

- A – place component
- W – draw connection
- P – place power label
- L – place local label
- Ctrl+L – place global label
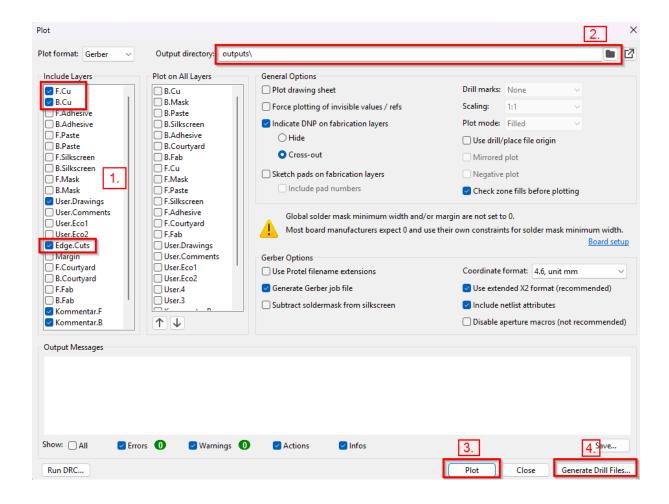
Pcb editor shortcuts

- V – Switch between top/bottom layer
- F – move selected to other layer
- X – Route single track
- U – select whole track
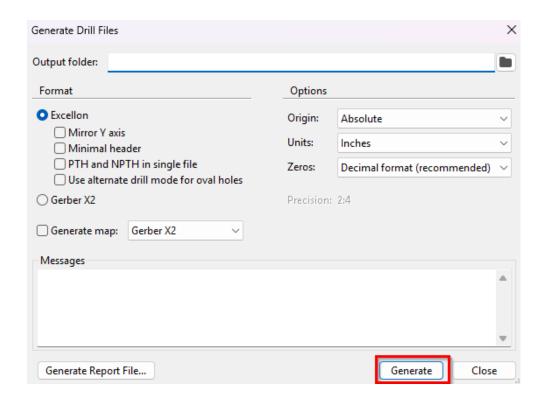- B – Recalculate all filled zones

Exporting your board for production

- In-house fabrication
- Ordering

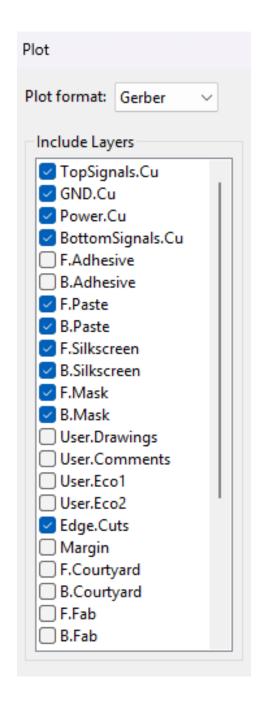# In-house

Ordering

- Same procedure, but more options can be exported
- Check exported files using JLCDFM before ordering
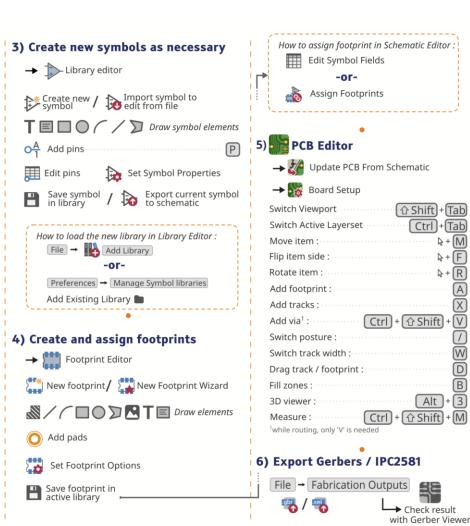- https://jlcdfm.com/

KiCad Cheatsheet
https://docs.kicad.org

1) Create a project
File → New Project → New Project

2) Schematic Editor

Add components :	A
Move item[1] :	▷ + M
Grab item[1] :	▷ + 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

[1]*grab* keeps connections, *move* doesn't

3) Create new symbols as necessary

→ Library editor

Create new symbol / Import symbol to edit from file

T ▤ ▭ ○ ⌒ / ⊿ Draw symbol elements

Add pins P

Edit pins   Set Symbol Properties

Save symbol in library / Export current symbol to schematic

How to load the new library in Library Editor :
File → Add Library
-or-
Preferences → Manage Symbol libraries
Add Existing Library 📁

4) Create and assign footprints

→ Footprint Editor

New footprint / New Footprint Wizard

▧ / ⌒ ▭ ○ ⊿ ▣ T ▭ Draw elements

◎ Add pads

Set Footprint Options

Save footprint in active library

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[1] :	Ctrl + ⇧ Shift + V
Switch posture :	/
Switch track width :	W
Drag track / footprint :	D
Fill zones :	B
3D viewer :	Alt + 3
Measure :	Ctrl + ⇧ Shift + M

[1]while routing, only 'V' is needed

6) Export Gerbers / IPC2581

File → Fabrication Outputs

gbr / .xml

→ Check result with Gerber Viewer

Anthony Gautier / KiCad Team
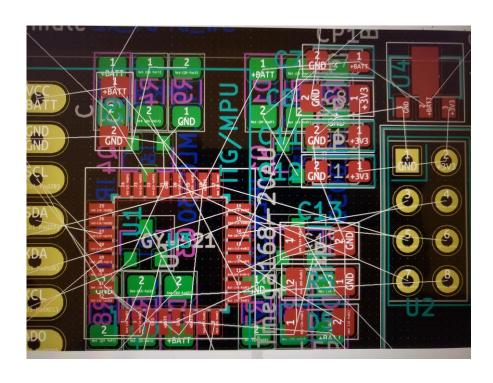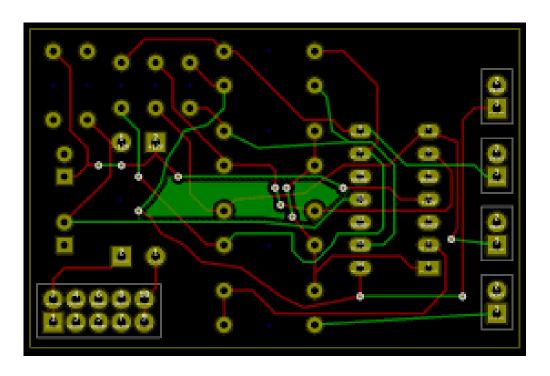
Kicad Cheatsheet

PCB design pitfalls

- Always use as wide a trace as possible
  - 0.4 mm / 0.6mm is ideal for traces
- Every component/IC/Transistor/Regulator should be decoupled using a 1-100n capacitor
  - Except some rare cases
- The goal is to get the shortest path for every trace from A to B
- Good PCB-design Requires a certain level of OCD

Proper PCBs

Our homemade PCBs
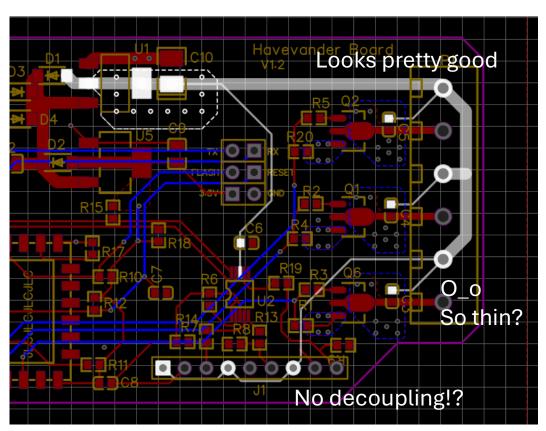
Examples – what not to do

A project doomed from the start

"You already paid for all the copper so you might as well use it" – Axel Olsson

Examples – what not to do

Looks pretty good

O_o
So thin?

No decoupling!?
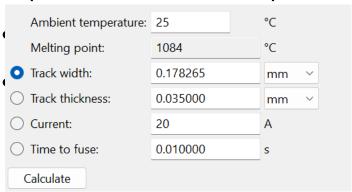
A garden pump controller
Designed by yours truely
(It never worked that well)

Other things

- Avoid Highspeed
    - If $l_{trace} > \frac{1}{3}\lambda_{signal}$ you have a problem
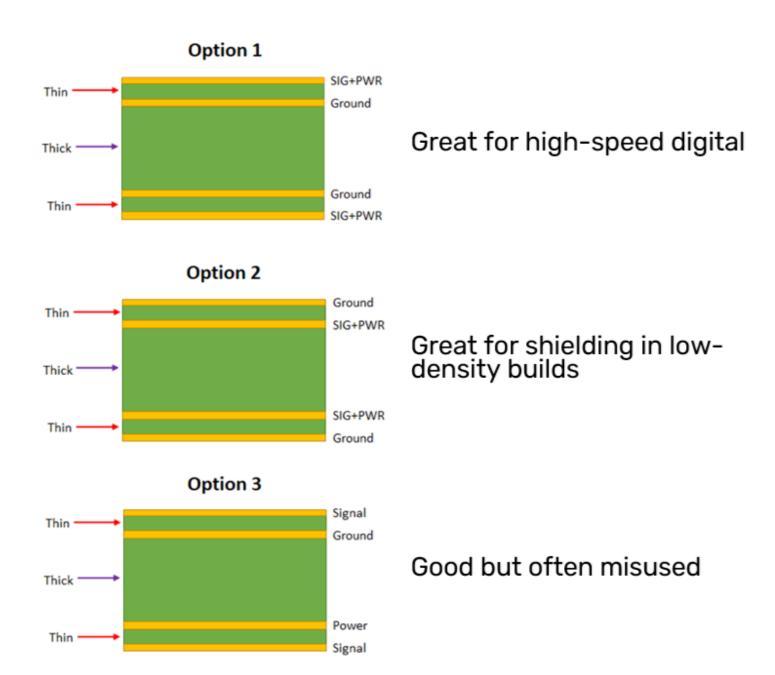
- High currents?
    - Your traces are resistors (and fuses as well)

Ambient temperature:	25	°C
Melting point:	1084	°C
⦿ Track width:	0.178265	mm ⌄
○ Track thickness:	0.035000	mm ⌄
○ Current:	20	A
○ Time to fuse:	0.010000	s

Calculate
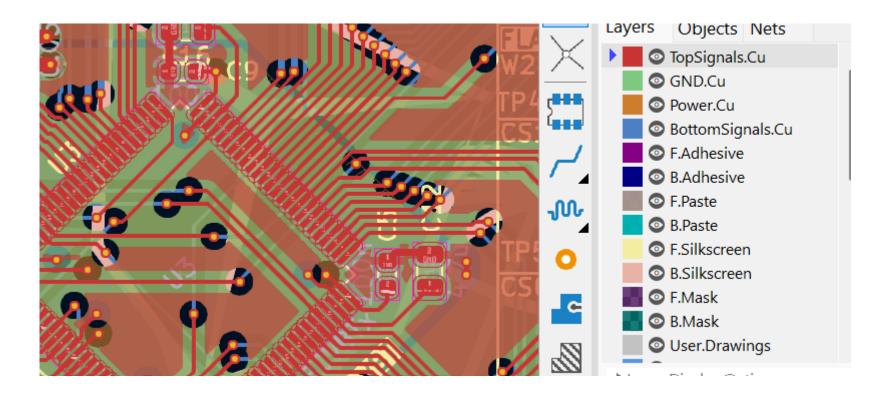
BREAK TIME

Power and Ground stackup

- Life is short, use 4 layer for "real" pcb's

Option 1

Thin → SIG+PWR
Ground
Thick →
Ground
Thin → SIG+PWR

Great for high-speed digital

Option 2

Thin → Ground
SIG+PWR
Thick →
SIG+PWR
Thin → Ground

Great for shielding in low-density builds

Option 3

Thin → Signal
Ground
Thick →
Power
Thin → Signal

Good but often misused
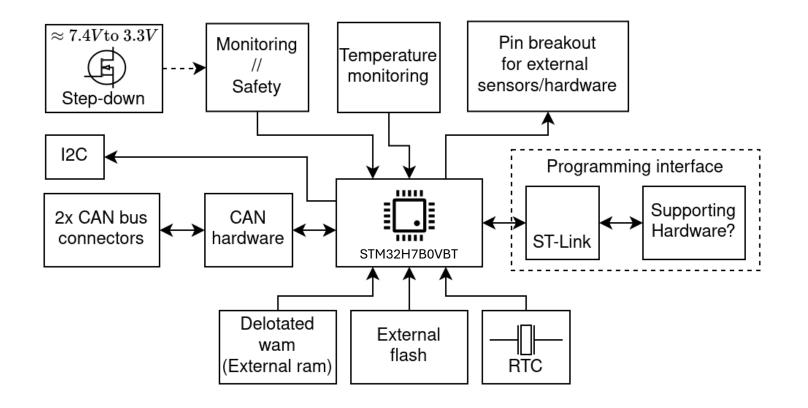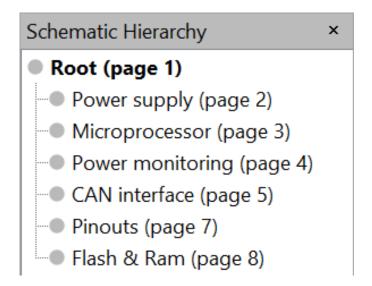
Organizing larger projects

- Building on Ben's lovely PCB course
    - Hierchal sheets
- Pin management
- Vector/Group Busses

Hierarchical design

- Splitting the board into functionalities
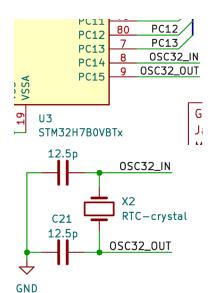
Hierarchical design – interfaces

- The pin connectors
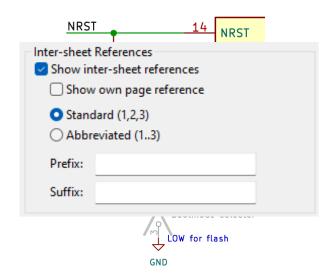
Local labels

Use for local connections local sheet only

Global labels
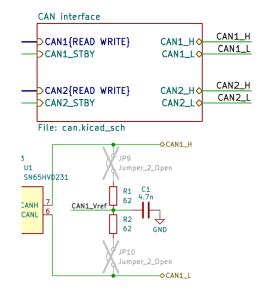
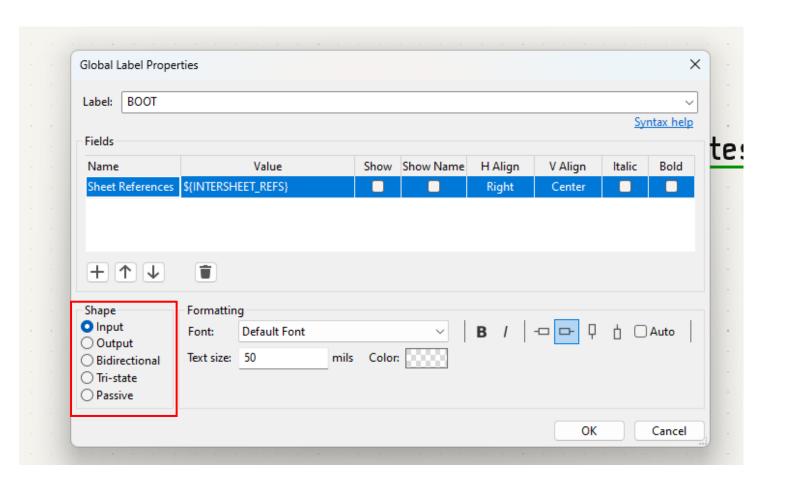Global variable across all sheets – Shares properties with "power" labels

Hierachal label

Makes a pin visible on the hierachy block
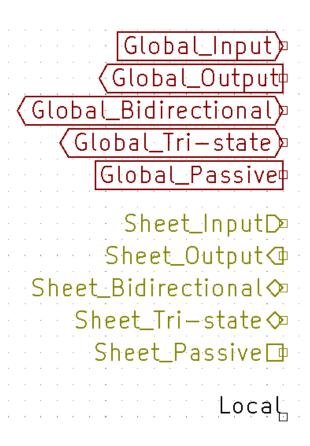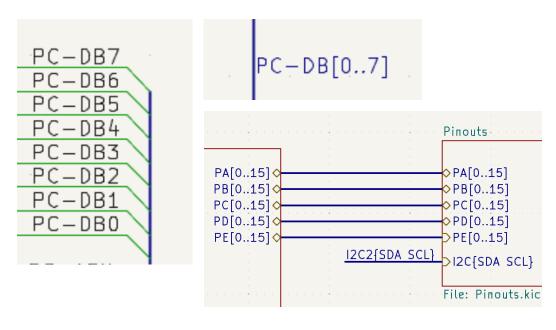
This one is different well come to him later

Hierarchical design – interfaces cont.

- (like humans) Almost all pins have a type
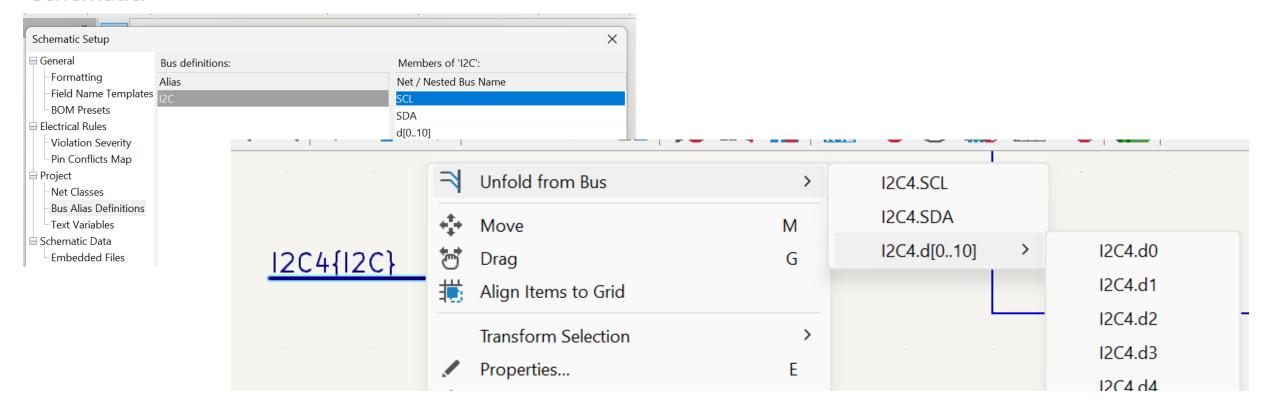
Vector / Group bus

Vector bus

Group bus

Vector / Group bus

Bus aliases

Bus aliases are shortcuts that allow you to work with large group buses more efficiently. They allow you to define a group bus and give it a short name that can then be used instead of the full group name across the schematic.

The Design rule checker <3

- Our constraints
    - In house fabrication
    - JLC standard
- DFM checker from kicad

Life saving plugins

- Interactive HTML-BOM
- JLC export

- https://octopart.com/

Exercise: How to make shield

- You dont have to know everything, fortunately.
  - We've done some work for you.
- https://github.com/aausat/a6hwcommon/tree/main/CommonDEVBoard/shield_template_rev1
  - kortlink.dk/2s42v