PCB Design & Introduction to KiCad

ELEC-D0301 Protopaja

Aalto University
School of Electrical
Engineering

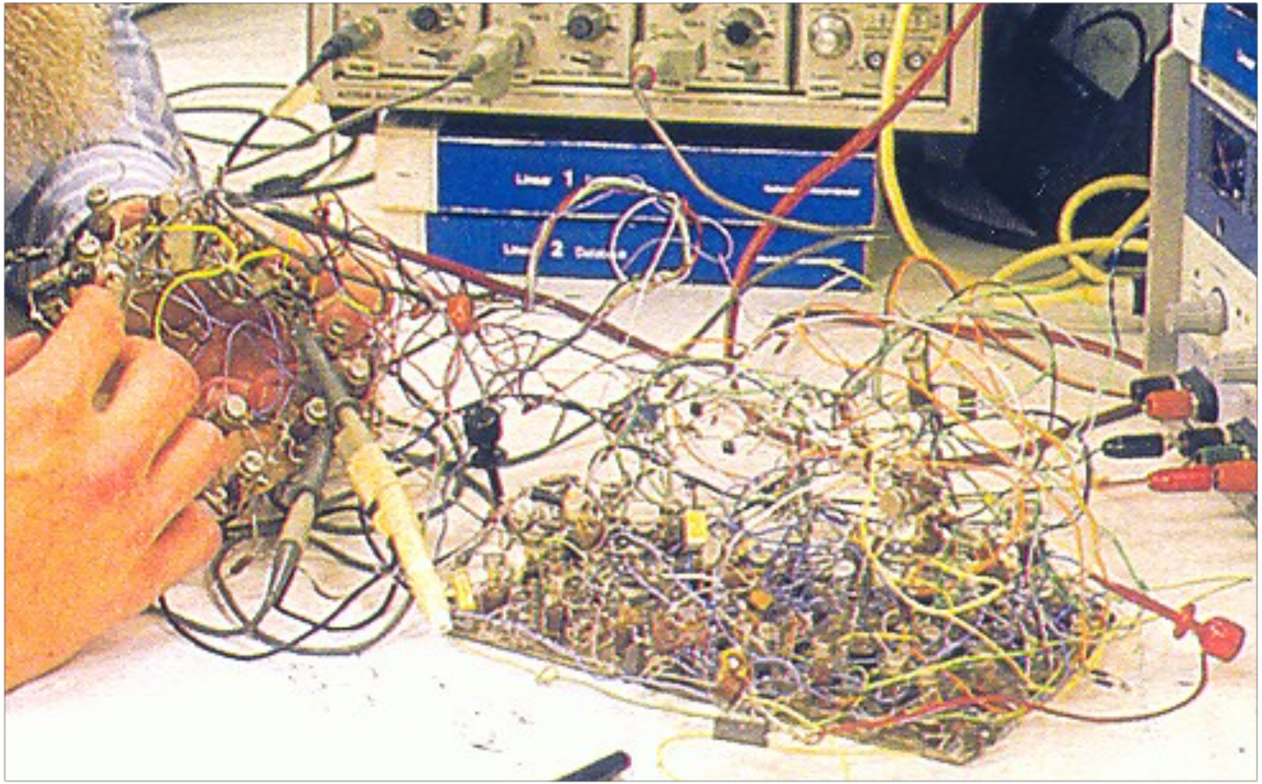
Aleksi Zubkovski Shahram Barai (Based on former lectures by Juha Biström)

4.6.2023

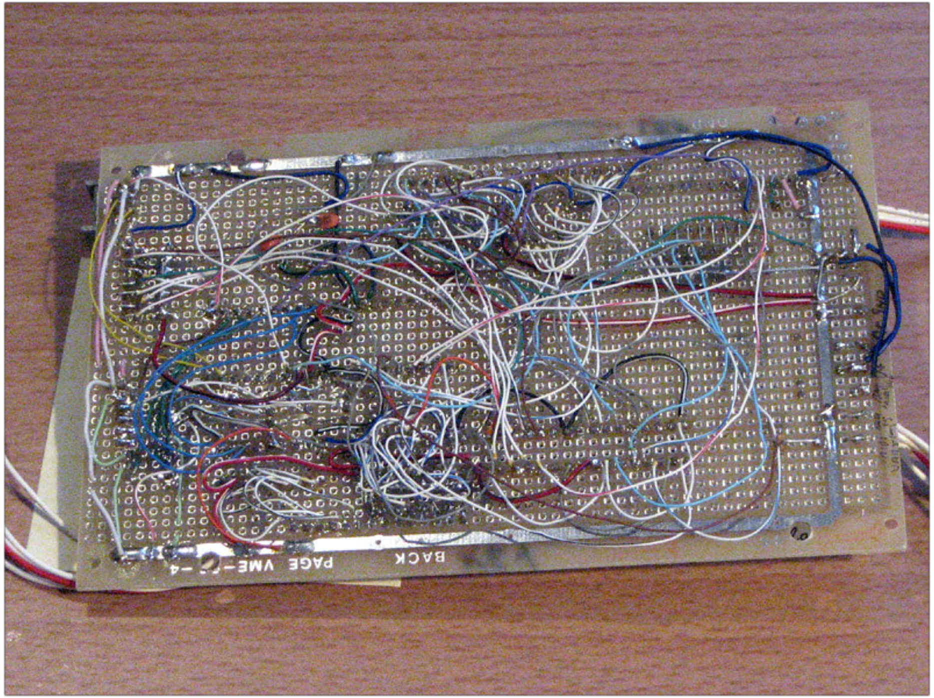
Motives on Creating a PCB

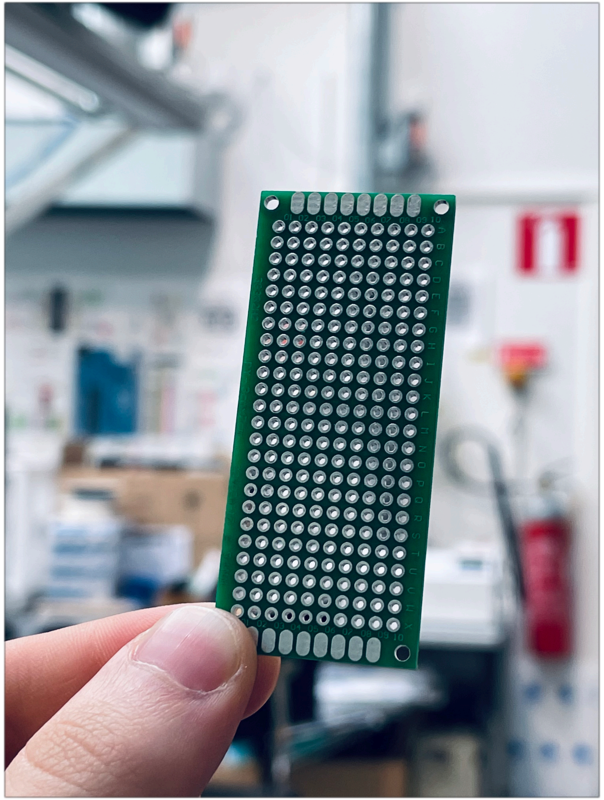
- Good & stable mounting for components
- Reliable line connections
- PCB CAD ensures that the circuit is properly connected
- Reliable & simple PCB vs breadboard ratsnest of wires

Motives on Creating a PCB



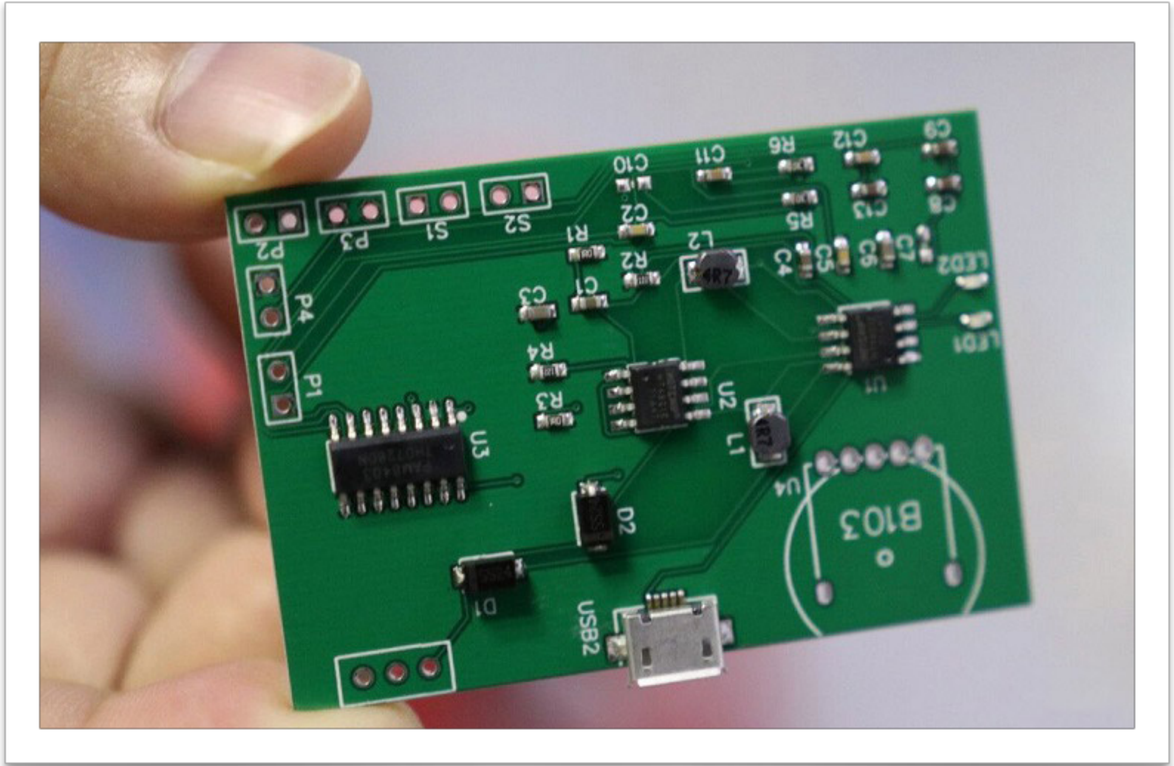
Motives on Creating a PCB





Aalto University
School of Electrical
Engineering

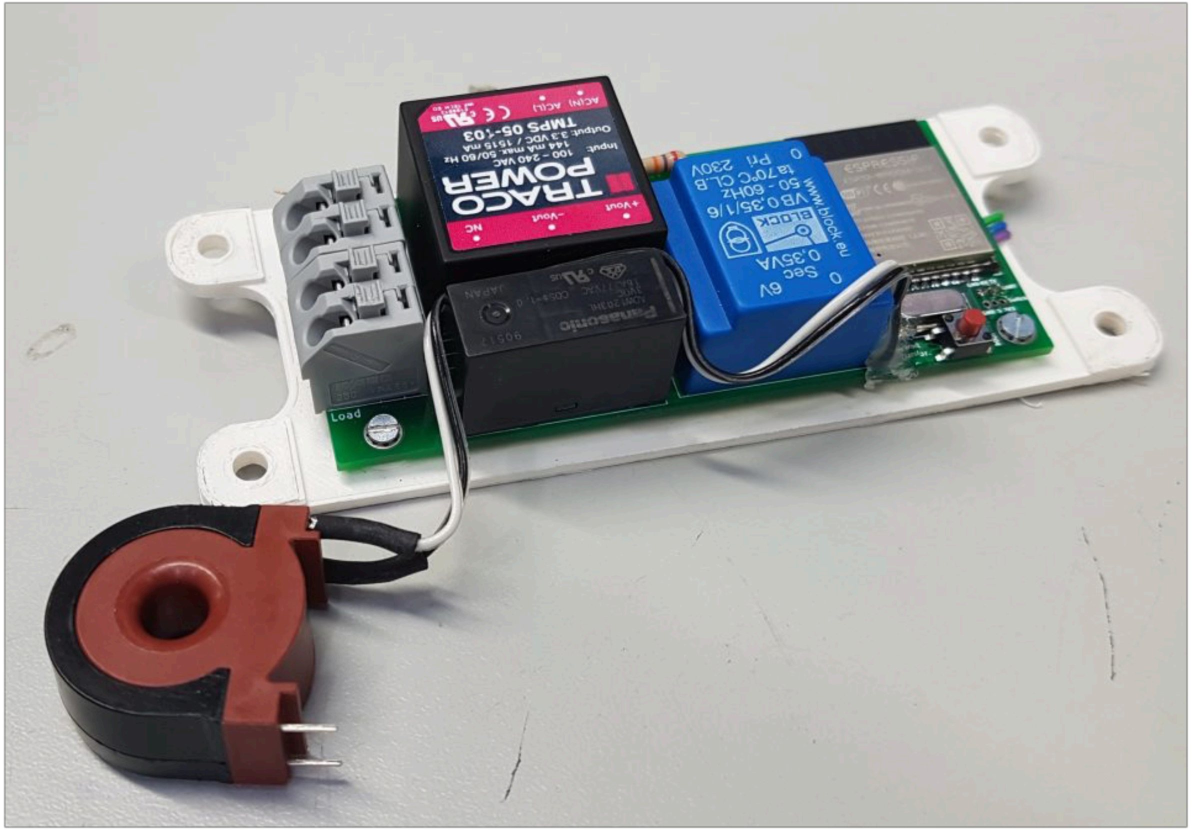
Motives on Creating a PCB



Motives on Creating a PCB

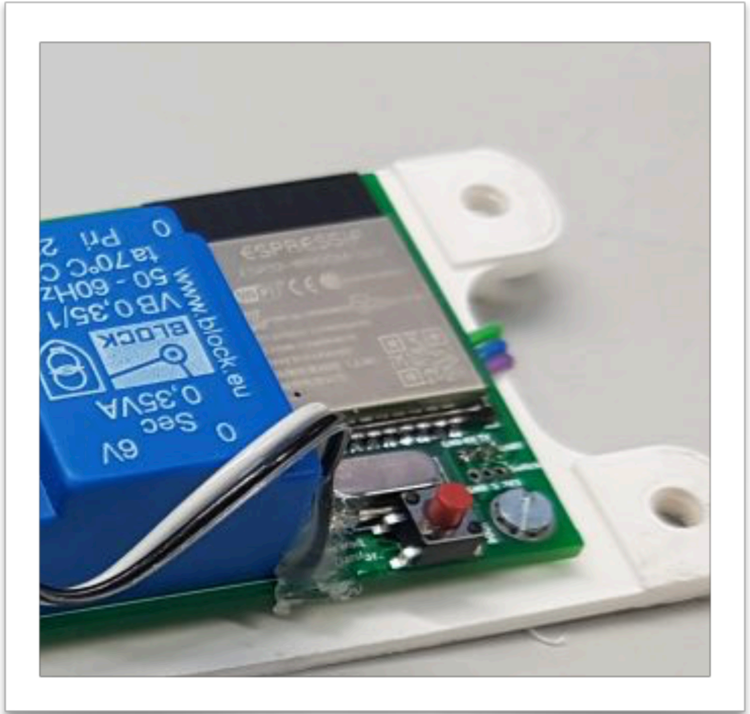
- Good & stable mounting for components
- Reliable wire connections
- PCB CAD ensures that the circuit is properly connected
- Reliable & simple PCB vs breadboard ratsnest of wires
- Possible to manufacture quickly industrially
- Clean & representable solution for a final product / final PROTO

Motives on Creating a PCB



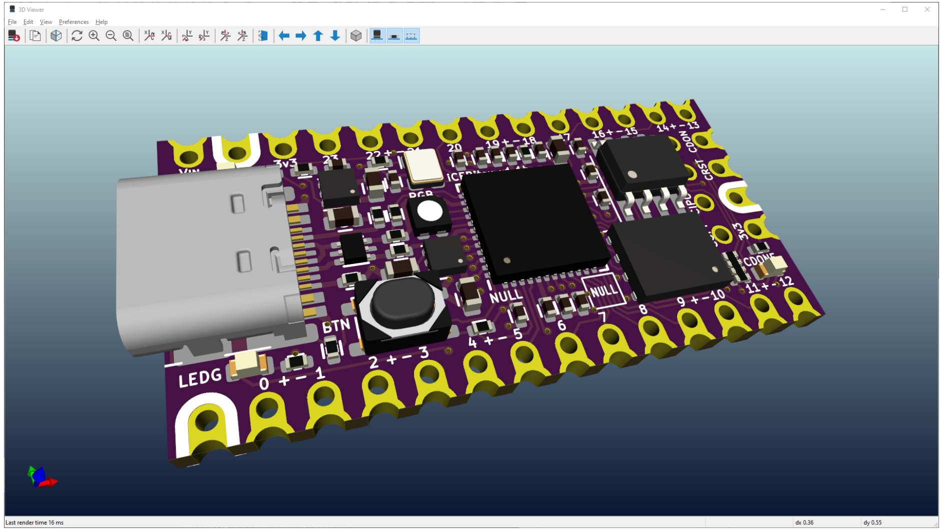
Aalto University
School of Electrical
Engineering

Motives on Creating a PCB

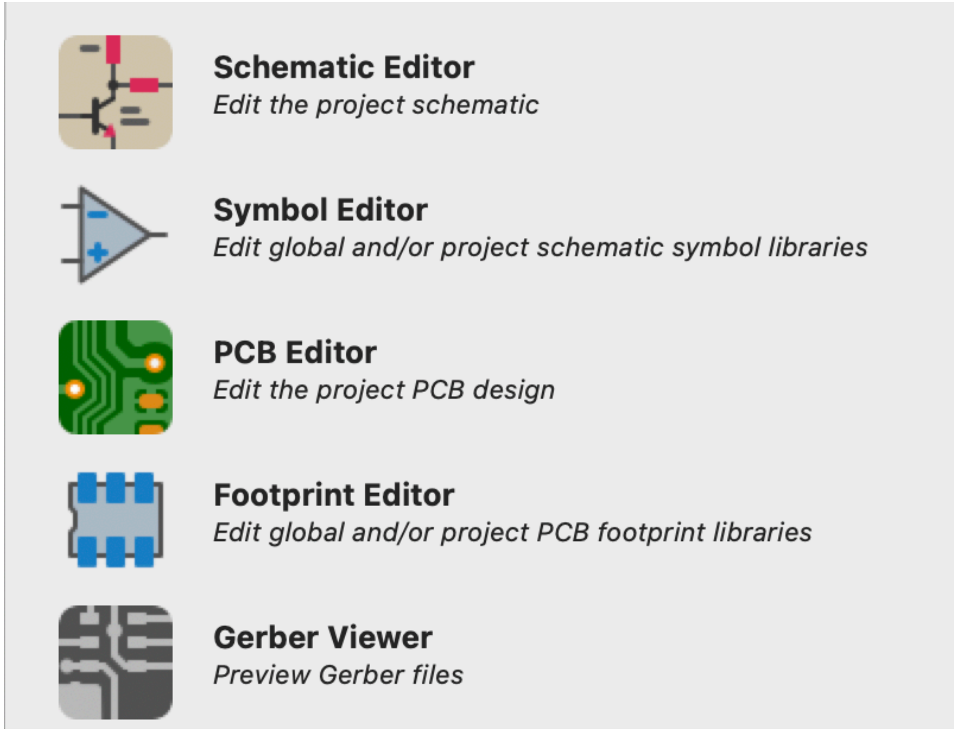


KiCad

- OPEN SOURCE <u>PCB CAD</u>
  - Main supporter: Cern
- Commonly used, efficient, versatile tool for designing circuits & PCB’s
- Free!



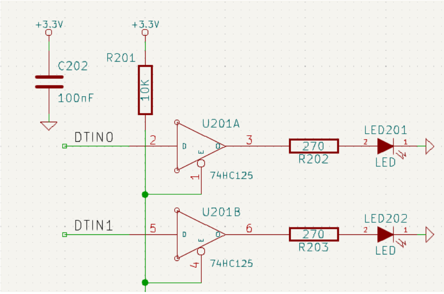
KiCad Subprograms



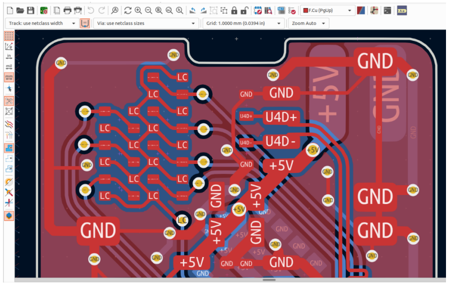
And more…

KiCad Workflow (from Afar)

Schematic Editor
Edit the project schematic



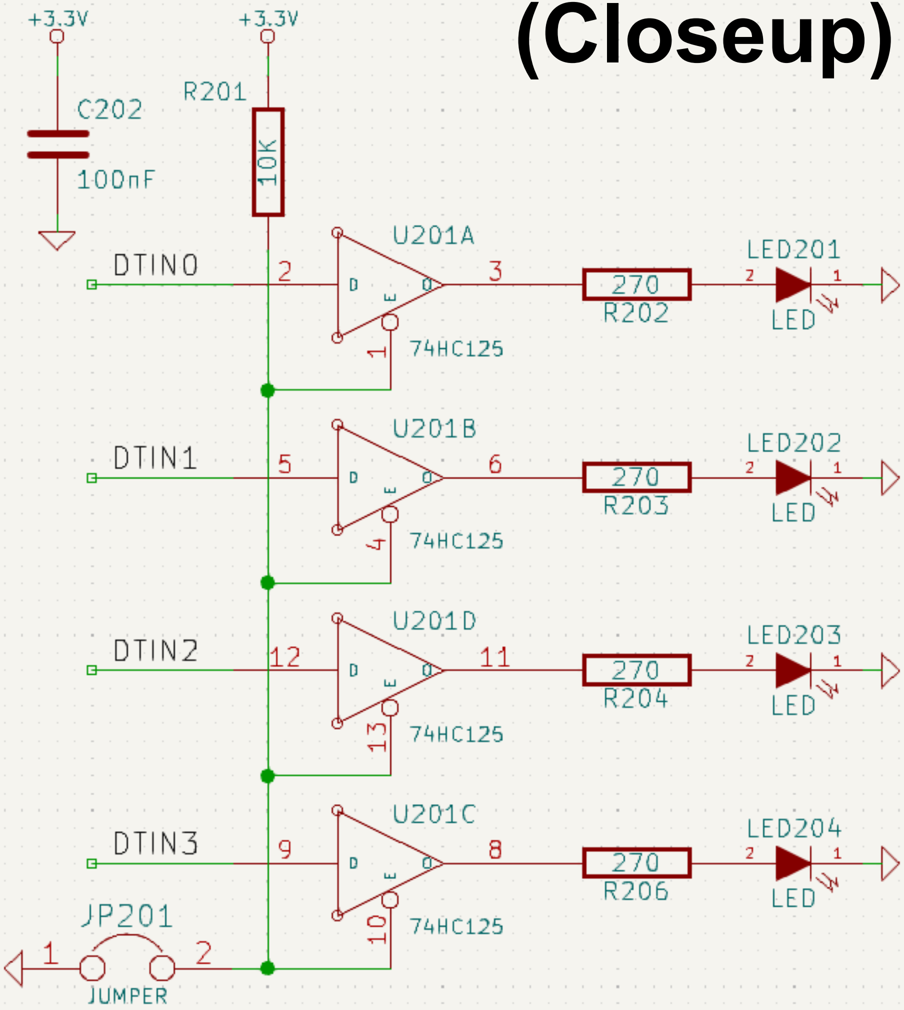
PCB Editor
Edit the project PCB design



- **Export outcome (Gerbers)**

KiCad Workflow (Closeup)

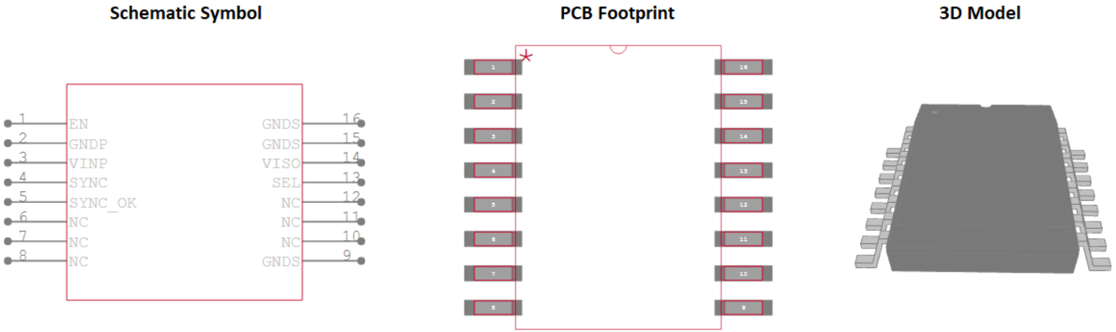
- **Create schematic**
  - (Create own symbols if needed)
- **Annotate symbols (components)**
- **Check for electrical rules**
- **Assign footprints to symbols**

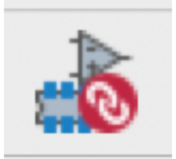


Aalto University
School of Electrical
Engineering

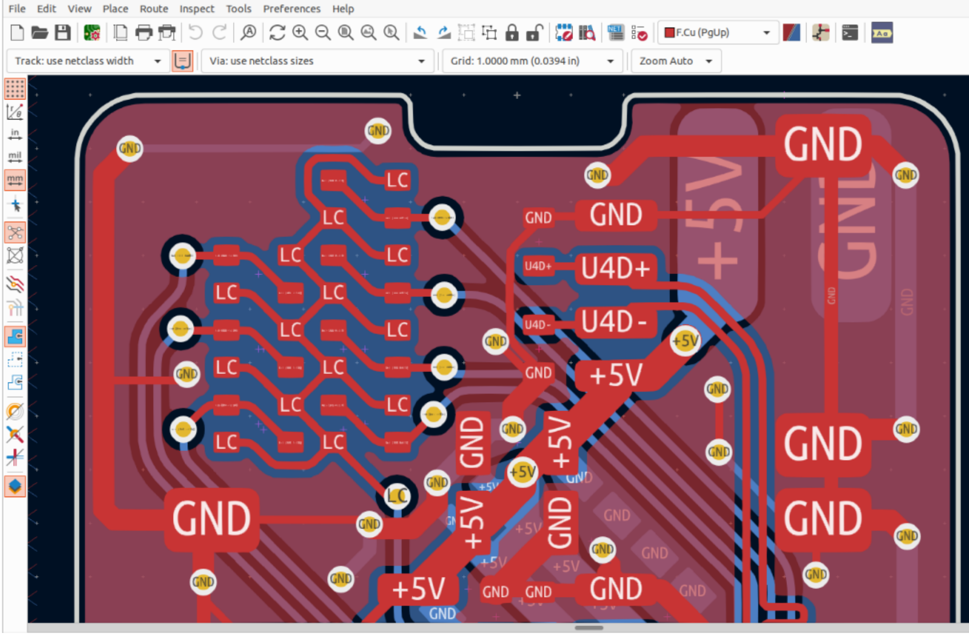
KiCad Workflow

- **Create schematic**
  - (Create own symbols if needed)
- **Annotate symbols (components)**
- **Check for electrical rules**
- **Assign footprints to symbols**
  - Choosing component casing types & sizes
  - Create own footprints if needed
  - Can be changed later in case of running out of space on pcb





KiCad Workflow



- **Create schematic**
  - (Create own symbols if needed)
- **Annotate symbols (components)**
- **Check for electrical rules**
- **Assign footprints to symbols**
  - Choosing component casing types & sizes
  - Create own footprints if needed
  - Can be changed later in case of running out of space on pcb
- **Design layout based on schematic with PCB editor**
  - Previous steps can be visited if needed
- **Generate manufacturing files**

KiCad Tips

- **Memorize at least the most common key shortcuts — Speeds up workflow a lot**
    - **Grab (g) ↔ Move (m)**
    - **Rotate (r)**
    - **Wire (w)**
    - **Add component (a)**

- **Automatic component annotation - do not annotate yourself**

- **Electrical rules checker — POSSIBLE ERRORS SOLUTIONS:**
    - e.g. two outputs shall not be connected together
    - Problems might arise from errors in symbols (Especially self made)
    - Mystic power supply errors are usually due to missing power flags in external supply connection

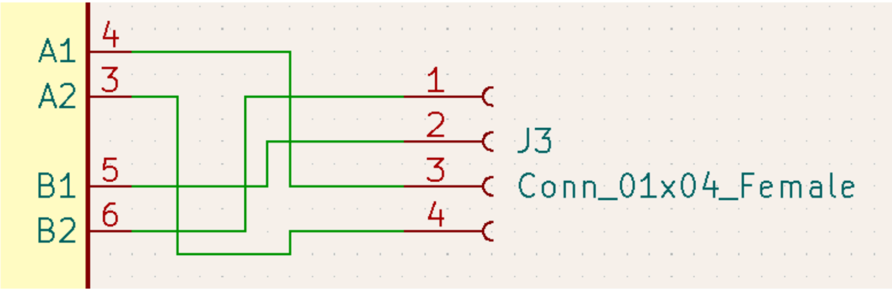
- **A lot of instructions on the Internet**
    - https://docs.kicad.org/6.0/en/getting_started_in_kicad/getting_started_in_kicad.html
    - VILHO-LAB Circuit design exercises: https://wiki.aalto.fi/display/ELECPROTO/Circuit+Design+Exercises

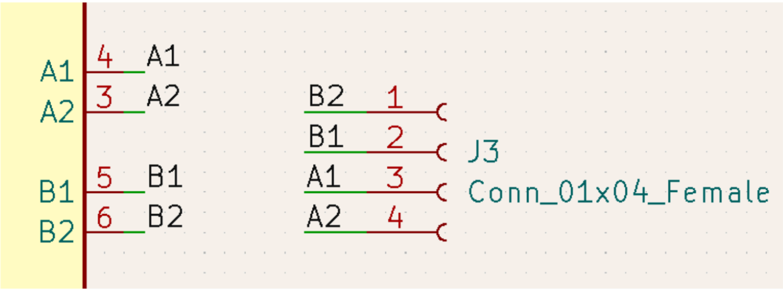
KiCad Schematic Tips

- Spread components in groups, by functional blocks
- Avoid SPAGHETTIS

Using local label

To avoid tangling with wires (making 'spaghetti'), it is recommended to use local label A .

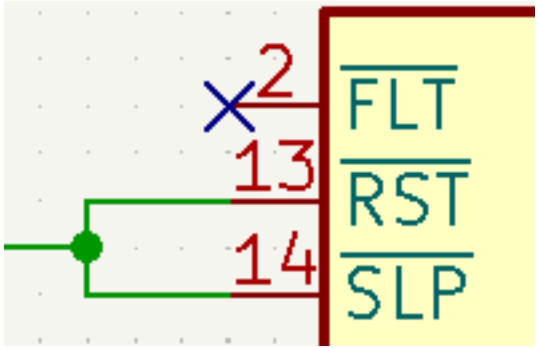




Aalto University
School of Electrical
Engineering

KiCad Schematic Tips

- **Spread components in groups, by functional blocks**
- **Avoid SPAGHETTIS**
- **Remember to set unused pins**
  - You will not pass E-rules checks



KiCad Layout Tips

- Start by filling project information & design rules
  - E.g. minimum trace widths & spacings, sizes
  - PCB supplier specific considerations?
- Remember mounting holes, Edges
- Spread components in groups, by functional blocks
- First place critical parts & their traces, then other stuff
  - Connectors are convenient to be placed at the edges
  - Led-Indicators, connectors, check points and debug ports should be located on same side of board
  - Switches, potentiometers, displays and other mounted to housing can be wired

KiCad Tips

KiCad cheatsheet

http://kicad-pcb.org/help/documentation/

1) Create a project

File → New Project → New Project

2) Eeschema : draw the schematic

Add components :	A
Move item[1] :	+ M
Grab item[1] :	+ G
Copy item :	+ C
Copy selection :	Shift +
Delete item :	+ Del
Delete selection :	Ctrl + Shift +
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 :	?

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

3) Create new components as necessary

→ Library editor

If editing an existing library : Select working library

Create new component / Load component to edit from current library

Draw component

Add pins .... P

Update current component into current library / Save current component to new library

How to load the new library in Eeschema :

Preferences → Component libraries

Component library files → Add

Select your `.lib` *file*

4) Create and assign footprints

→ Footprint Editor

If editing an existing library : Select active library

New footprint / Load footprint from library

Draw component

Add pins

Save footprint in active library / Create new library and save current footprint

→ Run CvPcb to associate components and footprints

How to load the new library in CvPcb :

Preferences → Footprint libraries

Append with wizard

Select your `.pretty` *folder*

→ Generate netlist

5) Pcbnew : design the layout

Design Rules → Design Rules + Layers Setup

→ Read netlist

Select top layer :	PgUp
Select bottom layer :	PgDn
Move item[1] :	+ M
Grab item[1] :	+ G
Copy item :	+ C
Rotate item :	+ R
Add tracks :	X
Add via :	V
Switch posture :	Q
Switch track width :	W
Drag track :	D
Fill zones :	B
3D viewer :	Alt (+ Shift) + 3

[1] *grab* keeps connections, *move* doesn't (Only for AZERTY keyboards)

6) Export Gerbers

File → Plot

Generate Drill File + Plot → Check result using GerbView

Anthony Gautier - http://silica.io

PCB Trace Width Calculator

Printed Circuit Board Width Tool

This Javascript web calculator calculates the trace width for printed circuit board conductors for a given current using formulas from IPC-2221 (formerly IPC-D-275).

Inputs:

Current	10	Amps
Thickness	2	mm

Optional Inputs:

Temperature Rise	10	Deg C
Ambient Temperature	25	Deg C
Trace Length	1	mm

Results for Internal Layers:

Required Trace Width	0.328	mm
Resistance	0.0000270	Ohms
Voltage Drop	0.000270	Volts
Power Loss	0.00270	Watts

Results for External Layers in Air:

Required Trace Width	0.126	mm
Resistance	0.0000701	Ohms
Voltage Drop	0.000701	Volts
Power Loss	0.00701	Watts

Notes:

The trace width is calculated as follows:

First, the Area is calculated:

Area[mils^2] = (Current[Amps]/(k*(Temp_Rise[deg. C])^b))^(1/c)

Then, the Width is calculated:

Width[mils] = Area[mils^2]/(Thickness[oz]*1.378[mils/oz])

For IPC-2221 internal layers: k = 0.024, b = 0.44, c = 0.725

KiCad Exercise

Aalto University
School of Electrical
Engineering