PCB Design & Introduction to KiCad

ELEC-D0301 Protopaja

Aalto University
School of Electrical
Engineering

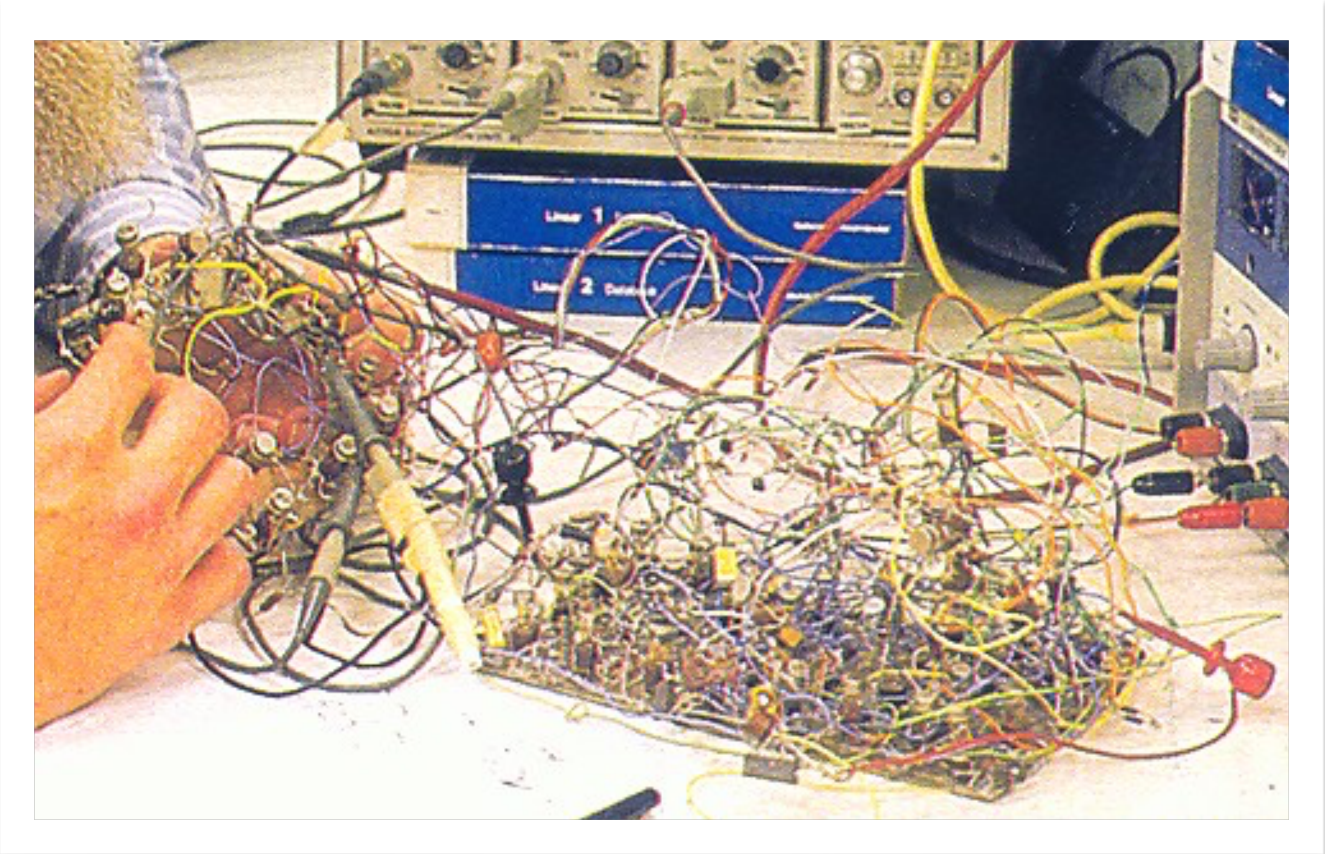
Aleksi Zubkovski (Based on former lectures by Juha Biström)

9.6.2021

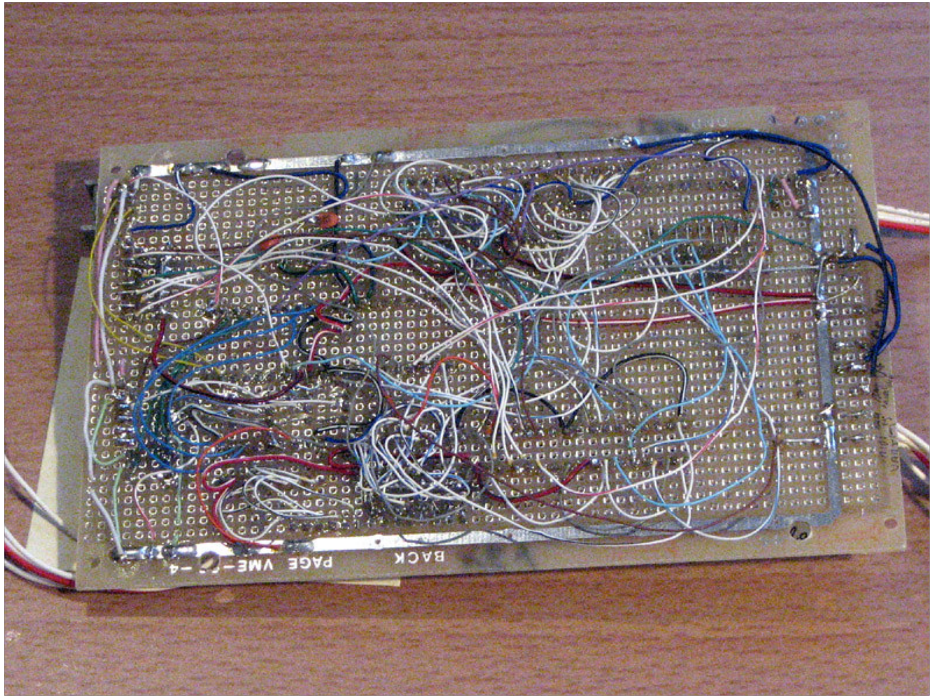
Motives on Creating a PCB

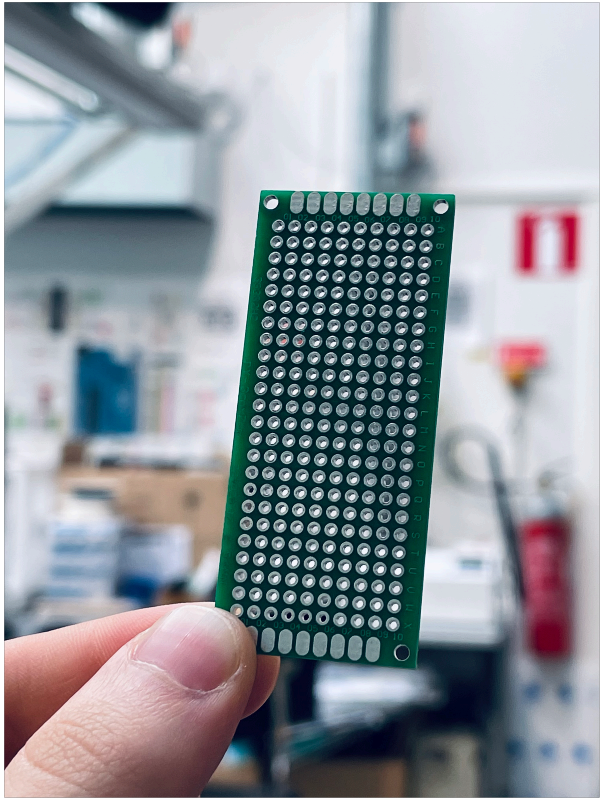
- Good & stable mounting for components
- Reliable wire connections
- PCB CAD ensures that the circuit is properly connected
- Reliable & simple PCB vs breadboard ratsnest of wires

Motives on Creating a PCB



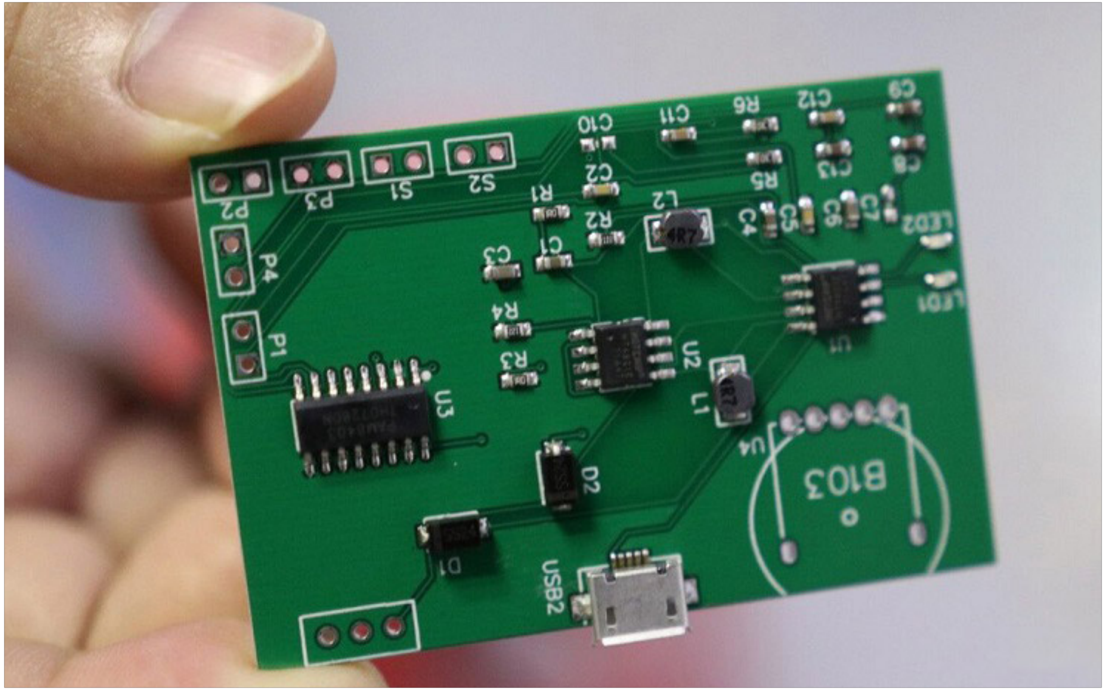
Motives on Creating a PCB



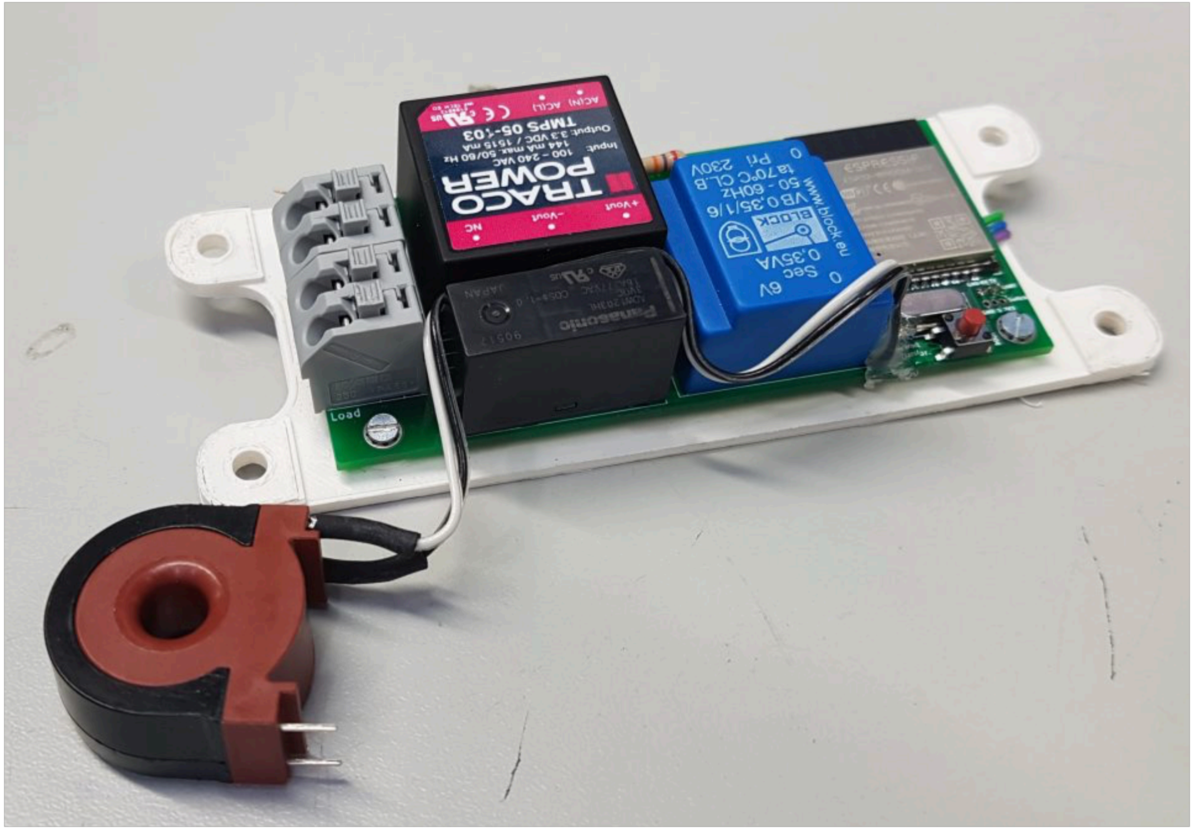


Aalto University
School of Electrical
Engineering

Motives on Creating a PCB

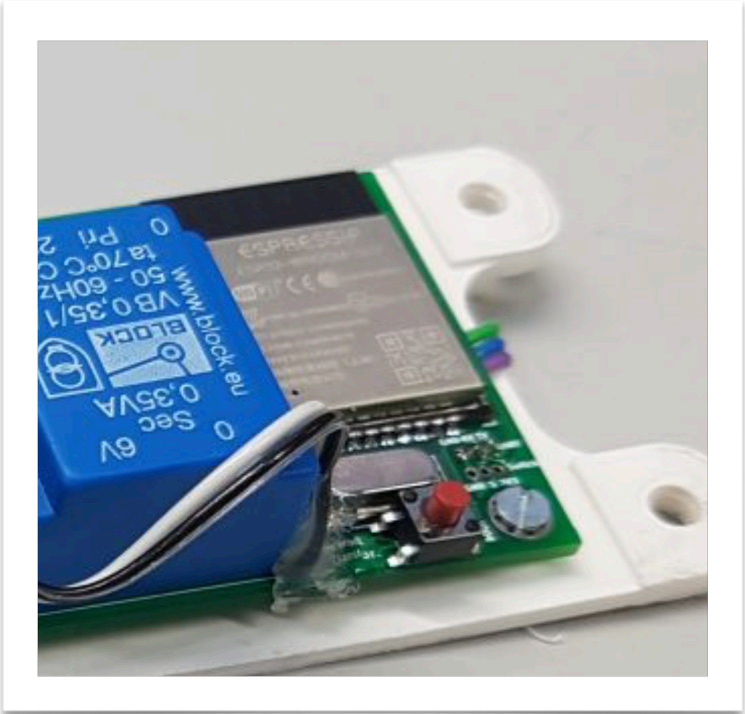


Motives on Creating a PCB



Aalto University
School of Electrical
Engineering

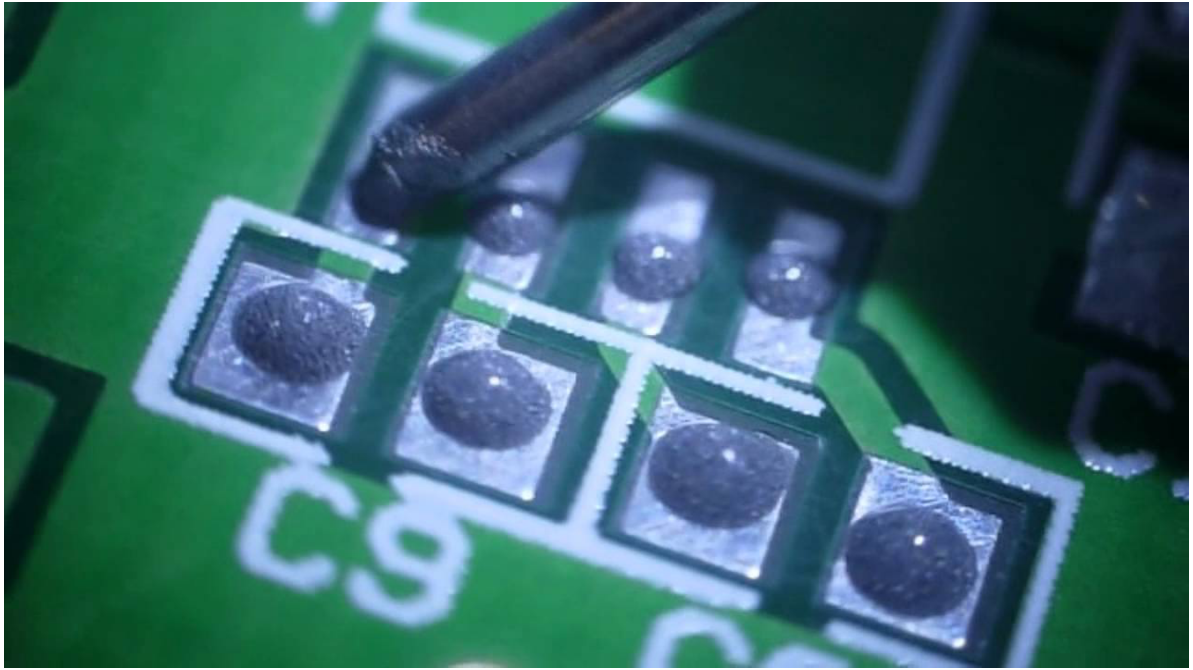
Motives on Creating a PCB



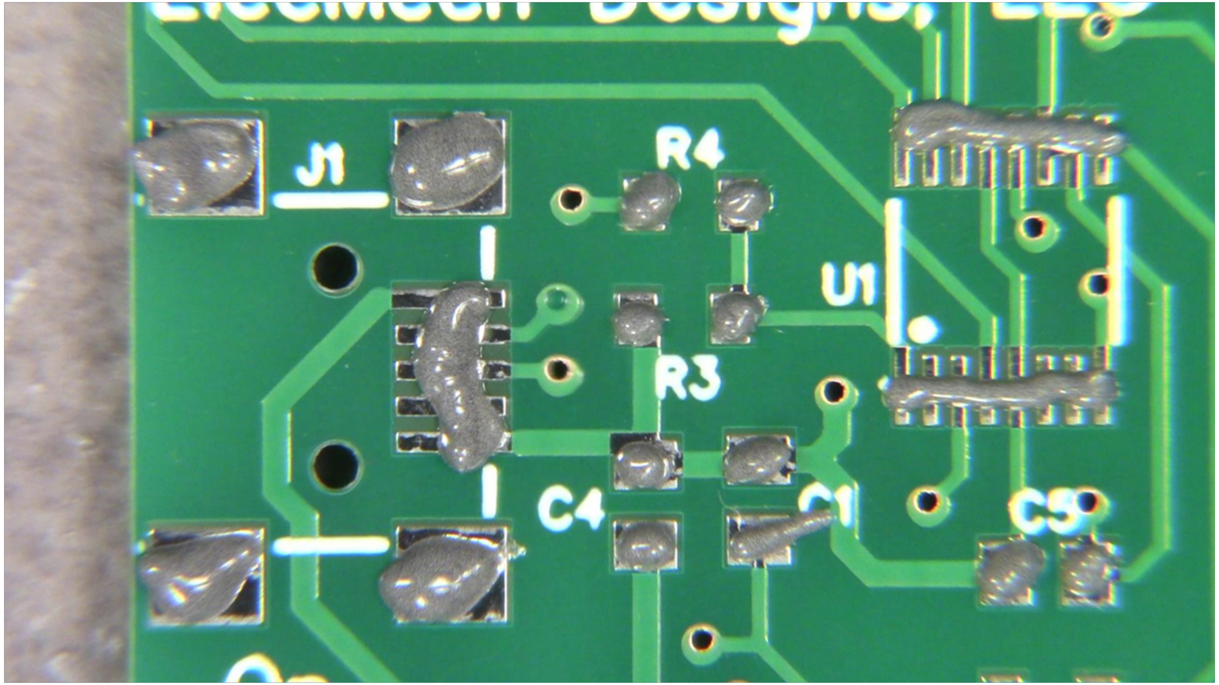
Motives on Creating a PCB

- 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
- Clean & representable solution for a final product

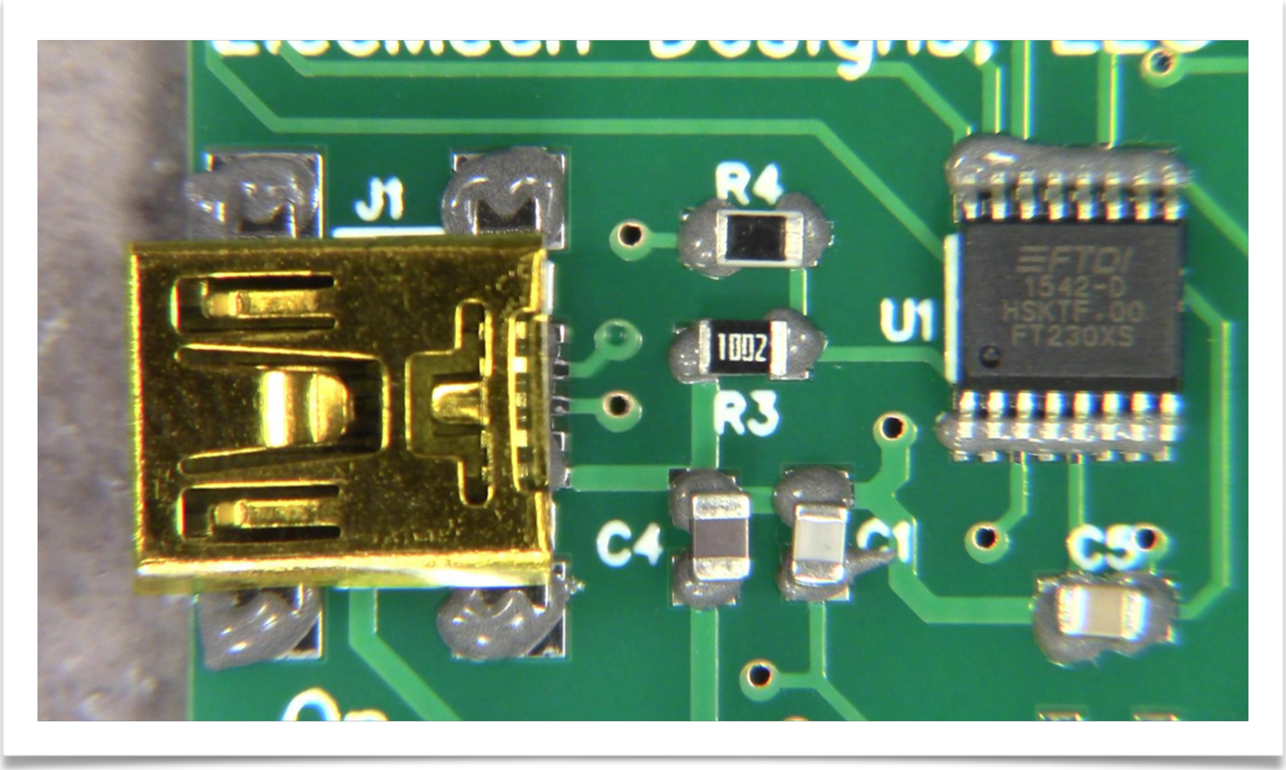
Components mounting on a ready PCB



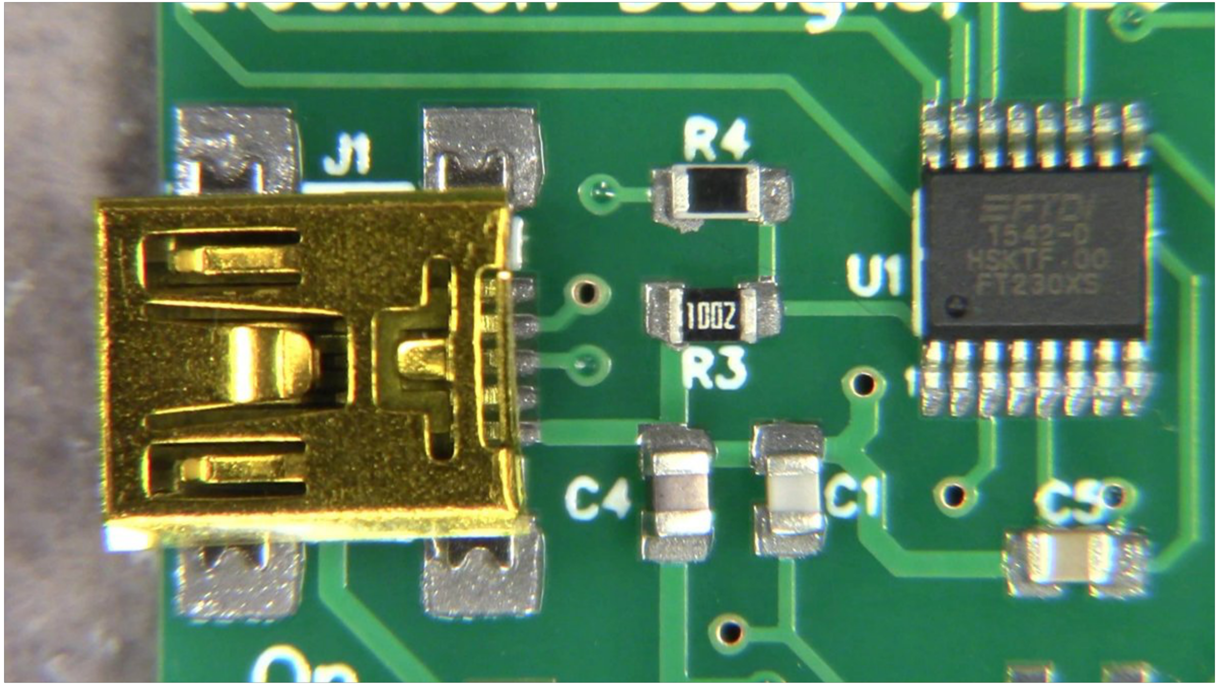
Components mounting on a ready PCB



Components mounting on a ready PCB



Components mounting on a ready PCB



Aalto University
School of Electrical
Engineering

Pictures by C. R. Hampton https://www.allaboutcircuits.com/projects/tools-techniques-hot-air-soldering-surface-mount-components/

KiCad

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

KiCad Subprograms

- **Eeschema – circuit design**
- **Symbol editor – create and edit own symbols**
- **Pcbnew – Layout design based on circuit**
- **Footprint editor – create own footprints**
- **Gerber viewer – view manufacturing files**
- **PCB calculator – relevant calculators to help PCB design**
  - E.g. minimum trace width for specified current

KiCad Workflow

- **Create schematic**
    - (circuit Create own symbols if needed)
- **Annotate symbols (components)**
- **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 previous steps**
    - Previous steps can be visited if needed
- **Generation of manufacturing files**

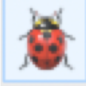
KiCad Tips

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

- **Add component (a)**

- **Automatic component annotation**

- **Electrical rules checker**
    - e.g. two outputs shall not be connected together, supply voltage has to be present



    - Problems might arise from errors in symbols (Especially self made)
    - Mystic power supply problems are usually due to missing power flags in external supply connection
- **A lot of instructions on the Internet**
    - http://docs.kicad-pcb.org/5.1.2/en/getting_started_in_kicad/getting_started_in_kicad.html

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 but at the edges
  - 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

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

Aalto University
School of Electrical
Engineering

KiCad Review