Beginner Workshop

KiCad

Presented by Thomas Pointhuber, Peter Poetzi and Bernhard Tittelbach at EH19

Schedule

- Short Introduction to KiCad EDA
- Build a simple design from Scratch
- Answer Questions

Please install KiCad 5.1.x now if not already done!

including the KiCad library

`http://kicad-pcb.org/download`

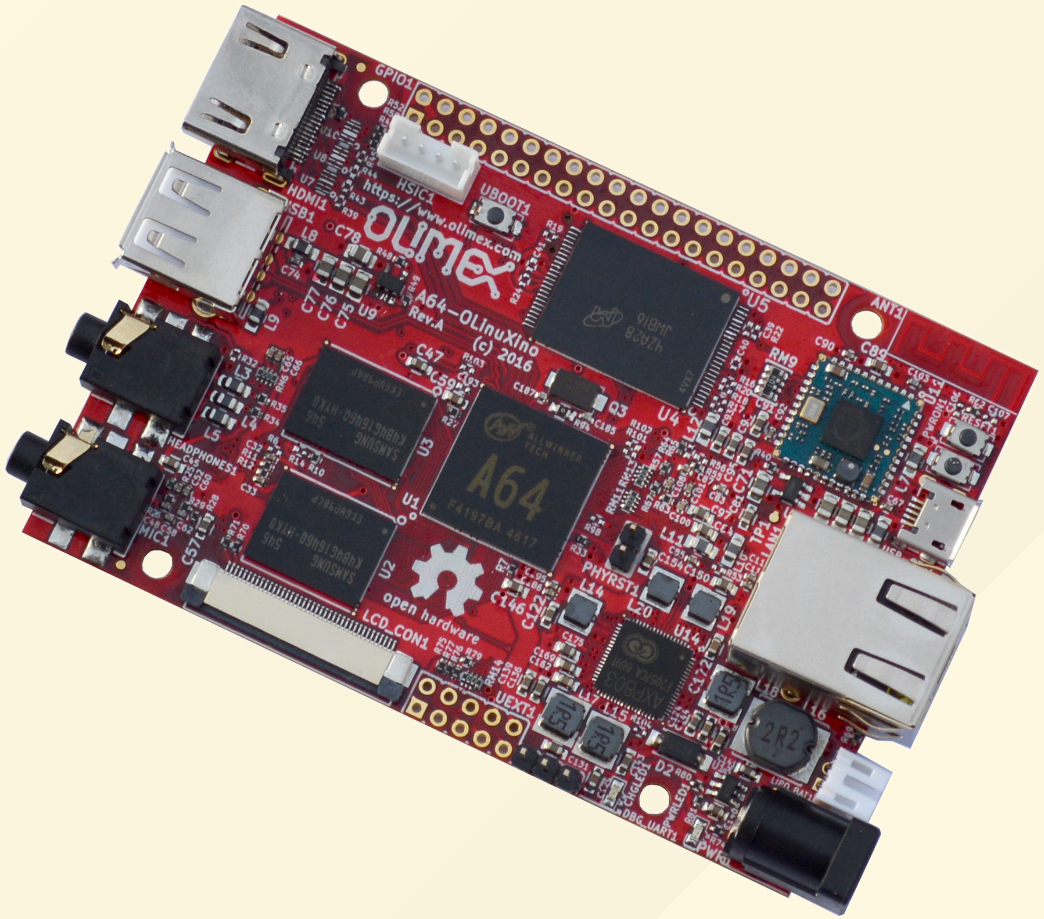
What is KiCad?

> “ A Cross Platform and
> Open Source Electronics Design Automation Suite ”

What KiCad gives us?

- Schematic Design Software
- PCB Design Software
- 3D-Viewer
- Gerber Viewer
- extensive Symbol, Footprint and 3D-Library
- exporters for BOM, Gerber, STEP, ...

Projects made with KiCad

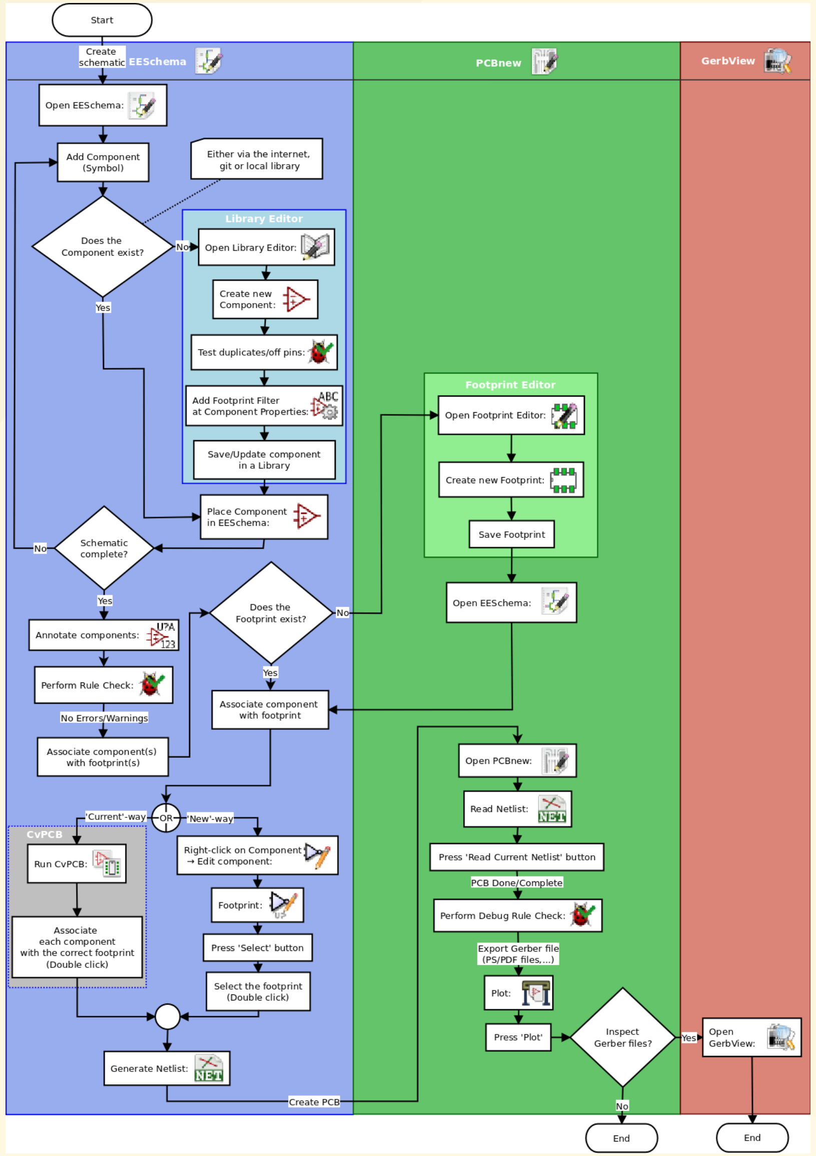


- ✅ (simple) hobby stuff
- ✅ RF circuits up to 6GHz
- ✅ Single-Board Computer (DDR3, WiFi, Ethernet, HDMI,...)
- *... your future application?*

- http://kicad-pcb.org/made-with-kicad

Basic Workflow

1. Draw **Schematic**
2. run Electrical Rule Check (**ERC**)
3. Export **Netlist** to Board Editor
4. Draw **Board**
5. run Design Rule Check (**DRC**)
6. **Export** BOM, Gerber,... 🎉



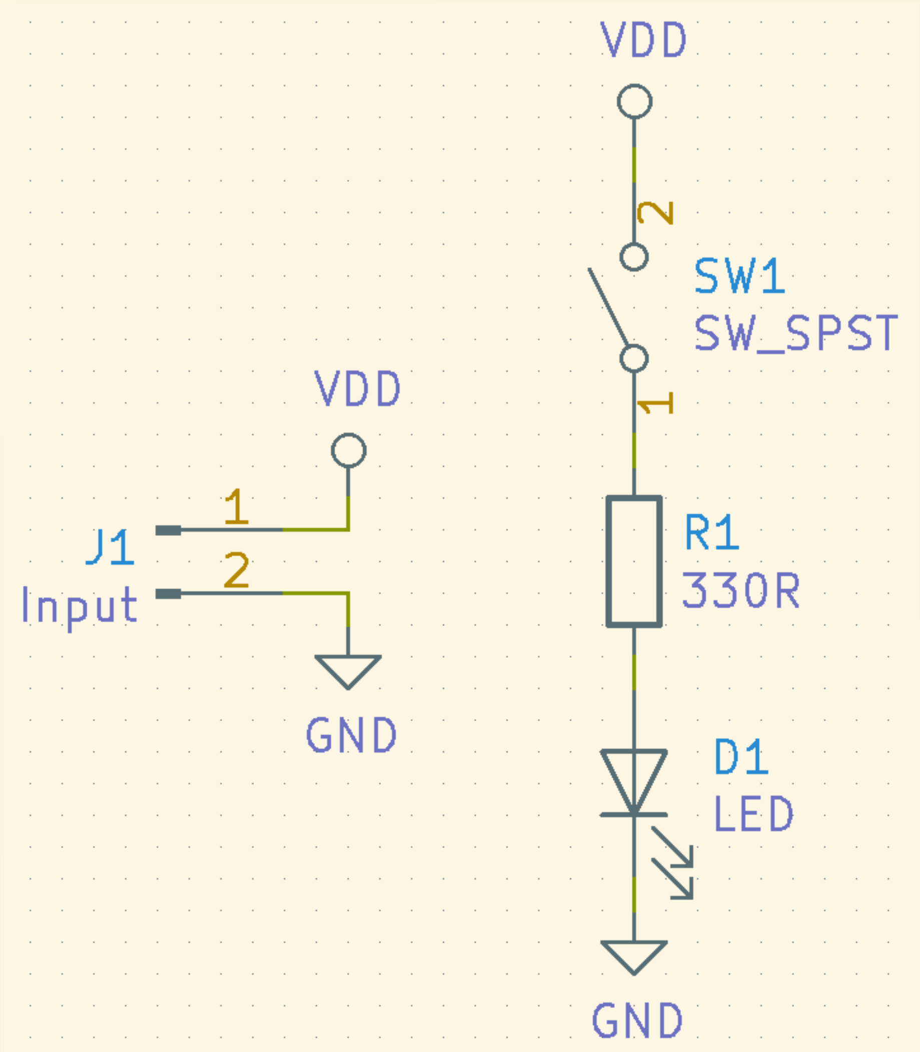
- http://docs.kicad-pcb.org/stable/en/getting_started_in_kicad.html

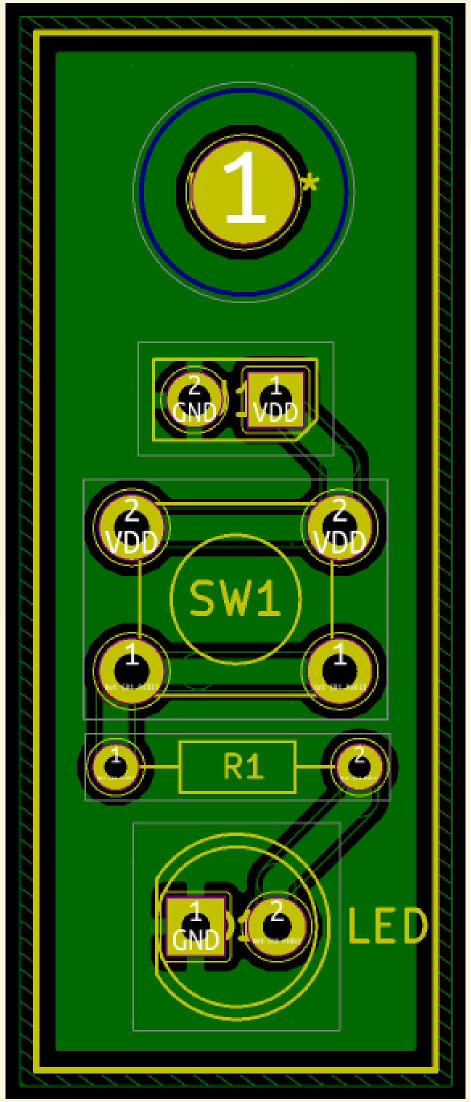
Library Management in KiCad

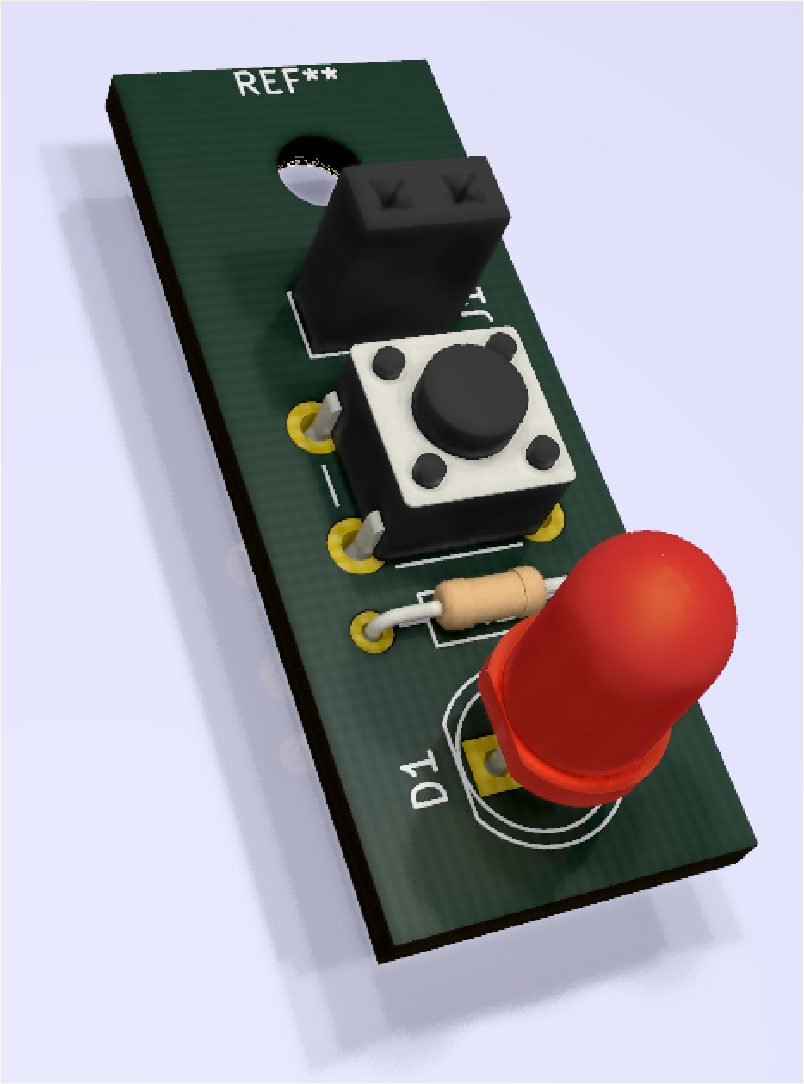
- Symbols and Footprints are located in distinct libraries
  - https://github.com/KiCad/kicad-symbols
  - https://github.com/KiCad/kicad-footprints
- Those types of libraries can be **Global** or **Project Specific**
- KiCad has it's own standard called KiCad Library Convention (KLC)

now we can start with the Workshop

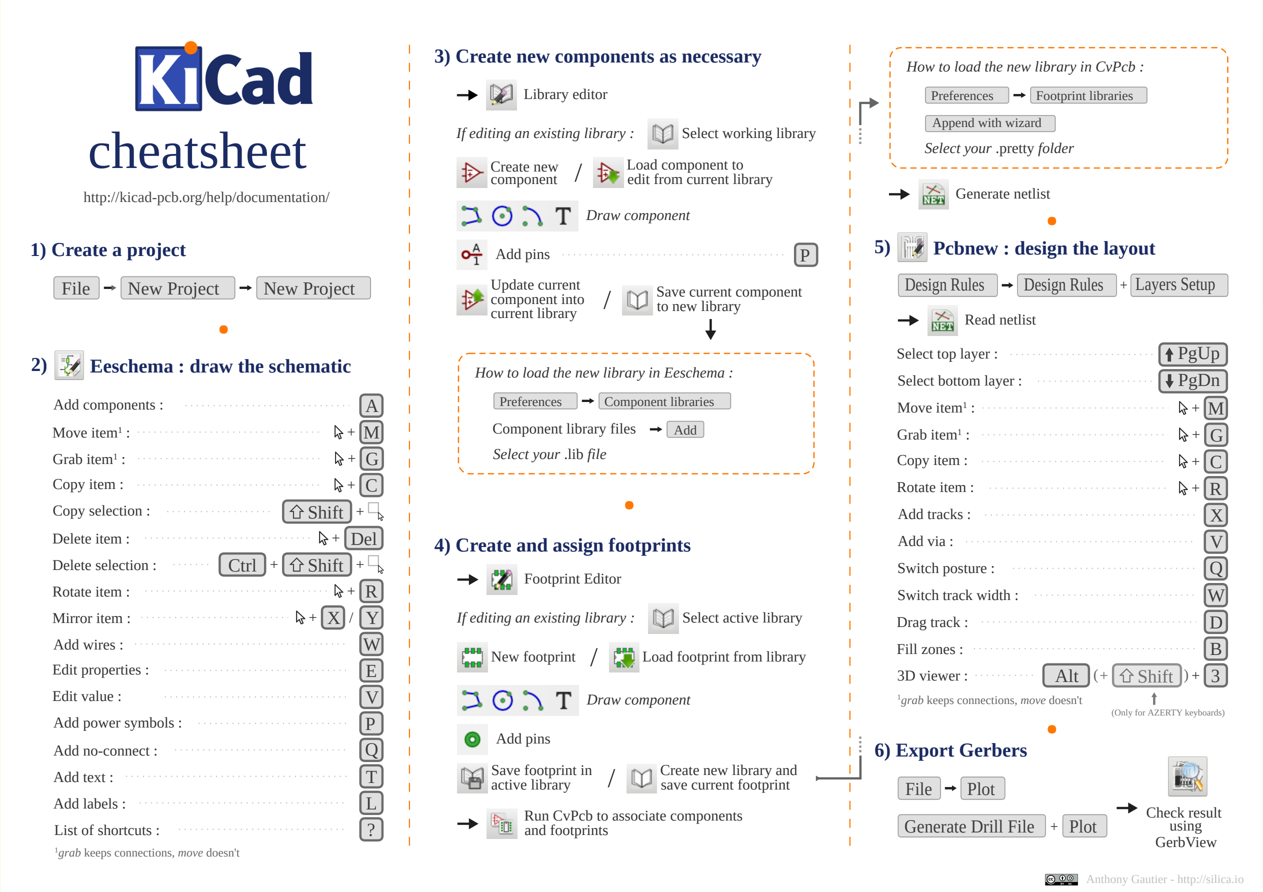
What will we design today?







Keyboard Shortcuts



- https://forum.kicad.info/t/cheatsheet-for-kicad/5247