

AIL Kicad Tutorial

19/06/2020

Adán L. Benito

Why Kicad?

- Open source
- Free
- Cross-platform
- Community
 - User-contributed action plugins
- Clean interface (IMO)
 - Shortcuts
 - Layered
- Good for hobbyist and professional
- Maintained by people at CERN

If it is good enough for people that design hadron colliders it is good enough for me

Other PCB design software

- Eagle cad
 - Autodesk / subscription based
 - Integration (Fusion 360)
 - Educational license, mildly costly for professional use
- Altium Designer
 - Industry standard
 - Massively complex
 - Price: £7530, down to £4795 on offer
- Diptrace
 - Less intimidating
 - Adapted pricing: from free to £900 (limitations in number of pins/layers)
 - Worst interface of the lot IMO (but check yourself). Some people love it

Designing a PCB

1. Concept

- a. Design
- b. Simulate?
- c. Choose components

Designing a PCB

1. Concept

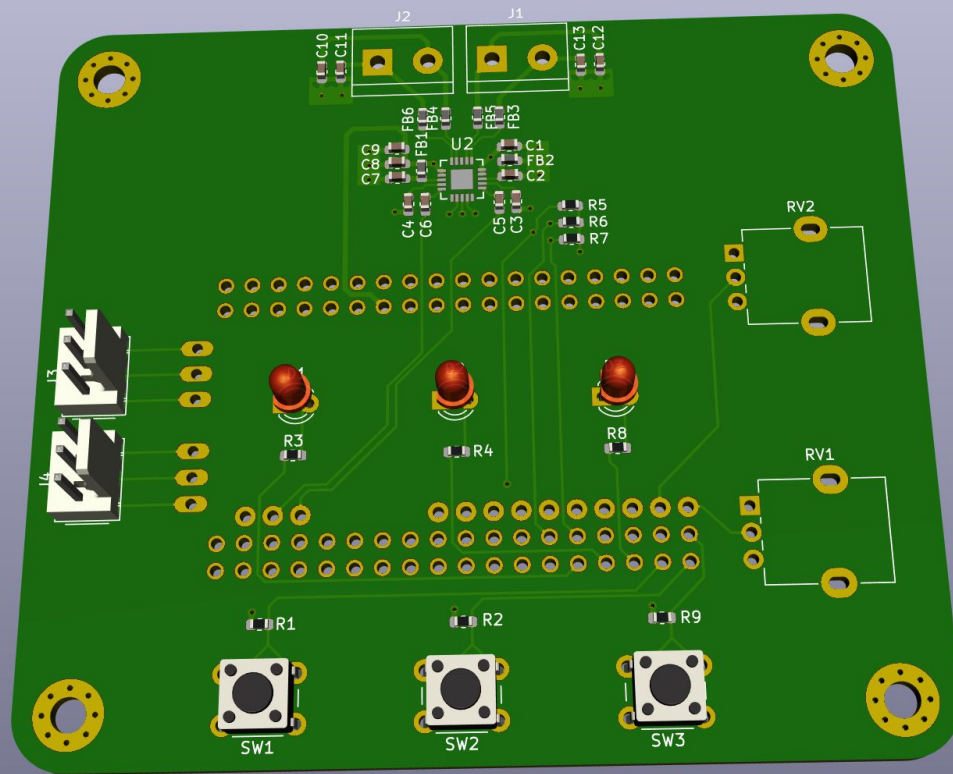
- a. Design
- b. Simulate?
- c. Choose components

2. Schematic

- a. Symbols
- b. Wiring: nets
- c. Labels
- d. Annotate
- e. Assign Footprints
- f. Generate Netlist

Eeschema shortcuts

- `a` - Add component.
- `c` - Copy component.
- `r` - Rotate component.
- `w` - Drag wire.
- `g` - Drag component without breaking wire.
- `e` - Edit component value.
- `Esc` - Return to pointer mode.
- `ctrl(cmd)+z` - Undo
- `ctrl(cmd)+s` - Save
- `ctrl(cmd)+c` - Copy block to clipboard.
- `ctrl(cmd)+v` - Paste block from clipboard.



Designing a PCB

1. Concept

- a. Design
- b. Simulate?
- c. Choose components

2. Schematic

- a. Symbols
- b. Wiring: nets
- c. Labels
- d. Annotate
- e. Assign Footprints
- f. Generate Netlist

3. Layout

- a. Layers
- b. Tracks
- c. Grid
- d. Netlist & footprints
- e. Outline (maybe later?)
- f. Routing
- g. Filled zones & copper pours
- h. DRC

Layers

- **F.Cu** and **B.Cu** : copper layers
- **F.Silk** and **B.Silk**: artwork on the silkscreen layers.
- **F.Mask** and **B.Mask**: area free of soldermask
- **F.Paste** and **B.Paste**: area that will be covered with solder paste
- **Edge.cuts**: final board shape.
- **F.Adhes** and **B.Adhes**: adhesive (=glue) areas.
- **F.CrtYd** and **B.CrtYd**: define a courtyard area.
- **F.Fab** and **B.Fab**: documentation.
- **Dwgs.User** and **Cmts.User**:user drawings and comments.
- **Eco1** and **Eco2**: no specific defined purpose.
- **Margin**: margin relative to the edge cut.

Pcbnew shortcuts

- `m` - Move item.
- `e` - Edit item.
- `r` - Rotate item.
- `f` - Flip item (switch to different layer).
- `x` - Route new track.
- `v` - Add through via.
- `b` - Update ground polygon pours.
- `Delete` - Remove a trace or component.
- `n` - Next grid size.
- `shift+n` - Previous grid size.
- `w` - Next track size.
- `shift+w` - Previous track size.

Designing a PCB

1. Concept

- a. Design
- b. Simulate?
- c. Choose components

2. Schematic

- a. Symbols
- b. Wiring: nets
- c. Labels
- d. Annotate
- e. Assign Footprints
- f. Generate Netlist

3. Layout

- a. Layers
- b. Tracks
- c. Grid
- d. Netlist & footprints
- e. Outline (maybe later?)
- f. Routing
- g. Filled zones & copper pours
- h. DRC

4. Fabrication

- a. Specification
- b. Gerbers & drill files