

# **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

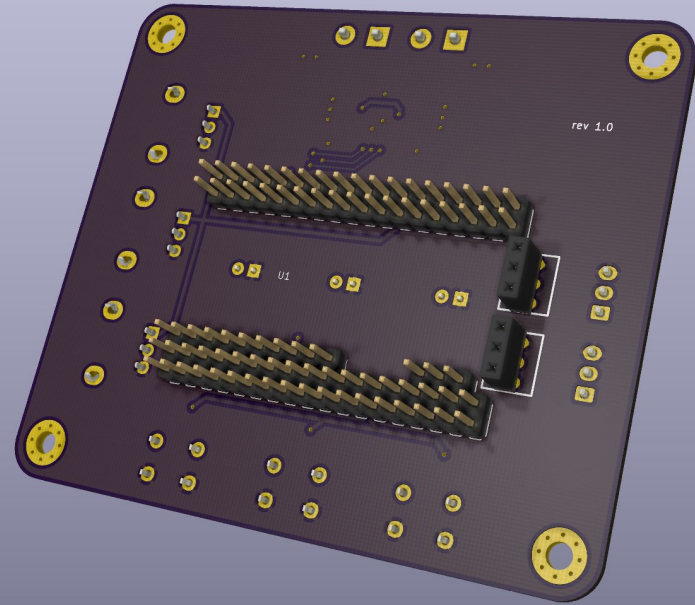
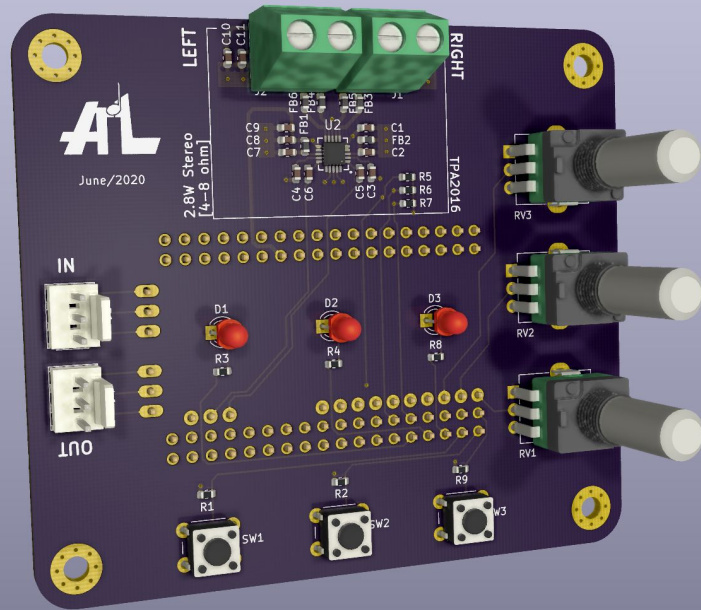
- a. Design
- b. Simulate?
- c. Choose components

## 2. Schematic

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

# 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
- g. Simulate?

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

## 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