

# KiCAD Cheat Sheet

---

Dainish Jabeen

May 14, 2023

## 1 KiCAD

KiCAD is designed to:

- Create circuit schematics
- Create PCBs

To ensure functioning PCBs are created, KiCAD offers the following:

- PCB Calculator (Determine electrical component properties)
- Gerber viewer (inspect manufacturing files)
- 3D viewer
- Integrated SPICE simulation

KiCAD has a manual workflow of first building the schematic then completing the PCB layout.

4 Base programs:

- Schematic editor
- PCB editor
- Symbol editor
- Footprint editor

**First step is to create a project, which has connected files for schematic and PC.**

## 2 Tips

- R: rotate object
- G: move object with wires attached
- M: move object without wires
- U: select multi traces connecting components

## 3 Create a schematic

1. In page setting give a title and rev number.
2. Use under lined A for any additional labelling
3. Choose component values (E: properties)
  - Comp val: Manufacturers part number

4. Choose Footprint for each item (E)
5. Run ERC
  - Power symbols (power flag) are required to power output pins (VCC, GND)
6. Create BOM

## 4 Create a PCB

The appearance panel on the right is for controlling the layers.

1. Setup page (page settings)
2. Setup board
  - Physical stackup (number of layers)
  - Constraints: Should match the fab house
  - Can have different constraints for different fab house
3. Update from schematic
4. Draw board outline (layer Edge cuts)
5. Place comps
  - F: Flip to other side of board
6. Route tracks
  - F.Cu : Front
  - B.Cu : Back
  - V: Insert via
7. Copper zones
  - Draw zone (should be slightly bigger than board)
  - Fill all zones (has to be done manually after changes, use B)
8. Design rule check
9. 3D viewer
10. Fab output (Plot + Drill files)

## 5 Create custom symbol

1. Create new global symbol library (Symbol editor -> new library)
2. Create symbol in lib
  - Add pins
  - Switch grid size to make changes easier
3. Set symbol properties (double click on page)
  - Set keywords for easy find
4. Create footprint via footprint editor (of components physical object)
  - Pin 1 is normally placed at 0,0
5. Add pads
  - Add margin for diameter and pads
6. Add outline
  - Fab layer
  - Silkscreen layer

- Courtyard
- 

$$\text{silkscreen margin} = fab_{width}/2 + silk_{width}/2$$

7. Link footprint to symbol