KiCAD Cheat Sheet

Dainish Jabeen

May 15, 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
-

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

7. Link footprint to symbol

6 SPICE Simulation

Simulation program with integrated circuit emphasis.

Example statement: .[type] [step time] [fixed time] i.e. .tran 1u 1m

- SPICE uses component sim models (.lib files) to describe the behavior of components.
- Simple components (resistor and capacitor) models are auto assigned, others need to be manually assigned.
- **KiCAD and sim model may use different pins**
- Placing SPICE statements as text, allows them to be saved
- Set compatibility to PSPICE and LTSPICE
- Use labels to read probe sim values easier
- Netlist shows connections in sim
- #branch: shows the current
- Show cursor on signal list to see val over time

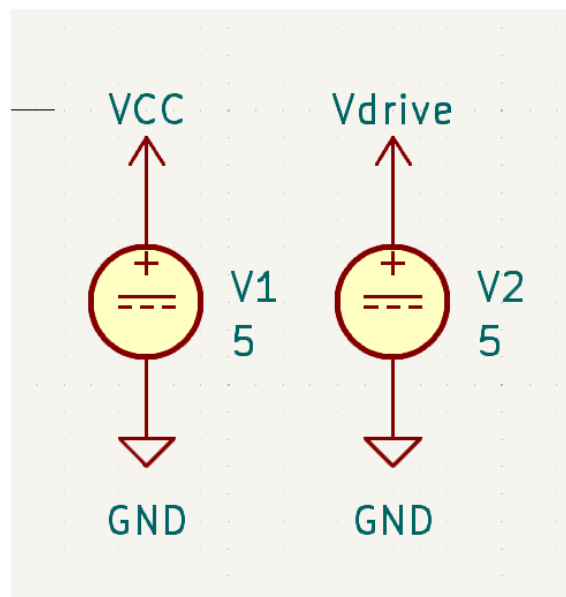Use global power ref points with SPICE power components 6.1, to simulate voltages. VSIN is for variable voltage waveform.

Figure 6.1: Power comp ref