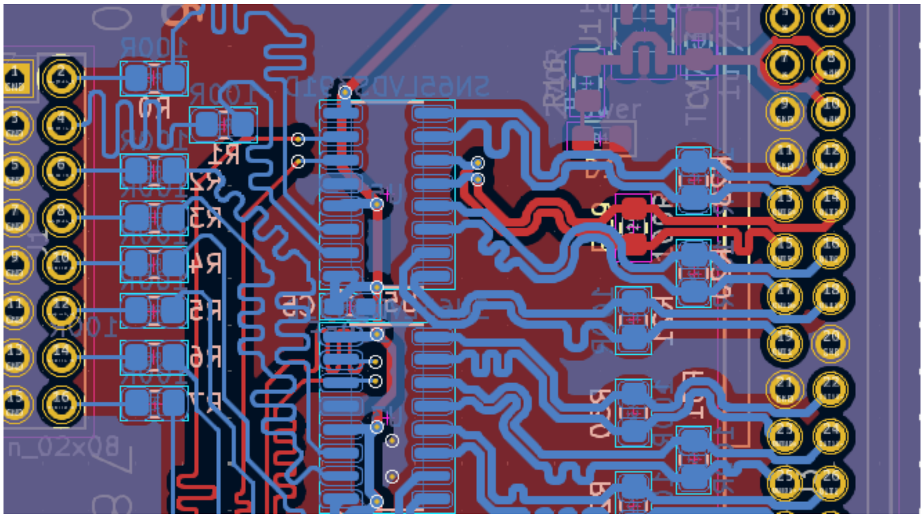

Guide to basic PCB design using KiCAD

B. Groenen

March 22, 2025

Contents

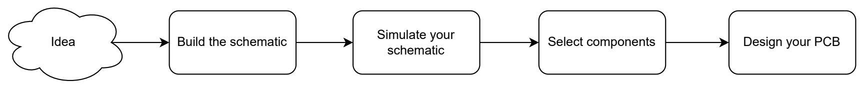

Figure 1: Flowchart of the process of creating a PCB

1 Introduction

In a world driven by electronics on PCBs, how nice would it be to be able to design your own. In this guide, the basics of PCB will be discussed with use of the free open-source software KiCAD. KiCAD is a powerful solution with support for custom libraries, SPICE simulation, a 3D viewer, and many other things. A step-by-step approach will be taken in this guide followed by do's and don'ts. Fig. 1 displays the general steps to be taken when designing a PCB.

1.1 Terminology

Some terms will be used which can be unfamiliar, they will be explained below:

- **Net**: A net is nothing more than points that should be connected. One or multiple wires can form a net.
- **Footprint**: The bottom of the physical component to be laid on top or inserted in the PCB.
- **BOM**: Bill Of Materials, the exact components ordered from suppliers to be used on your PCB.
- **Ratsnest**: Unrouted lines between components that should still be routed.

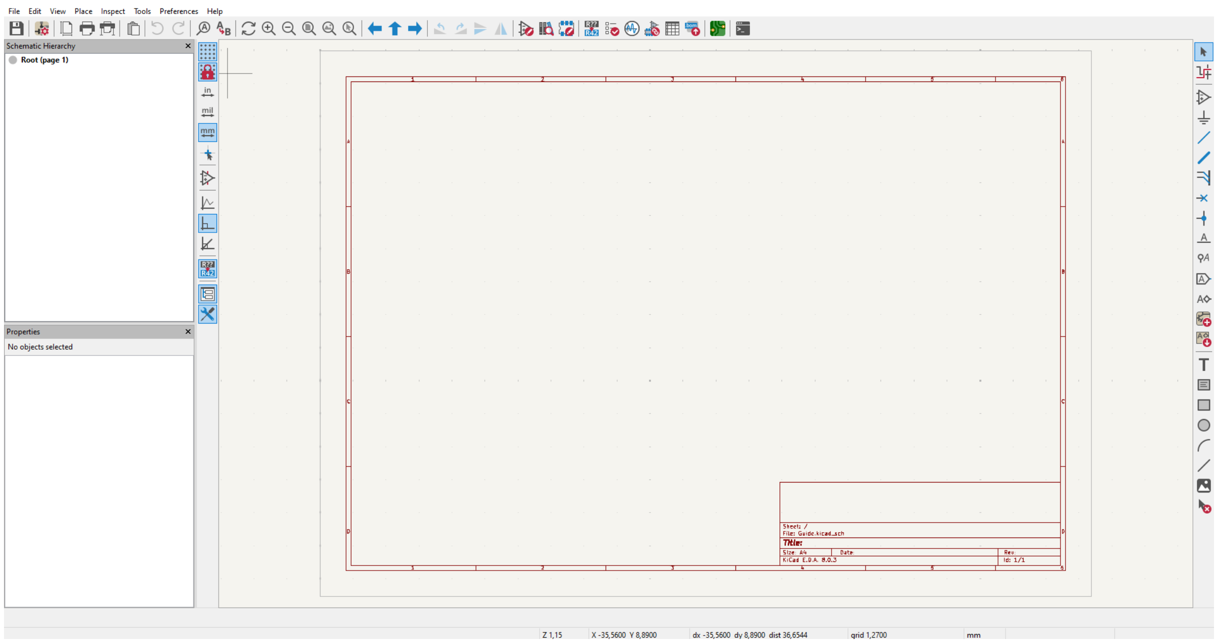

Figure 2: Schematic layout editor

2 Software

Once you open KiCAD, the project screen will appear where a new project can be created. After creation, two files will be generated. These files will contain the schematic (extension: .kicad_sch) and the PCB layout file (extension: .kicad_pcb).

2.1 Schematic layout editor

Once the schematic file is opened, the screen shown in Fig. 2 The following things and symbols depict important stuff:

- **Area within red**: This is the area in which the schematic is created.
- **Op-amp**: If clicked within the working space, a menu will pop up with all the available components. It contains a wide variety of both passive and active components. If needed components are not present, custom libraries can be created and/or loaded in.
- **Ground symbol**: This button is for placing the power nets such as ground, VDD, VSS, 3V3 and many more. Placing multiple of these with the same voltage results in them being connected.
- **Diagonal blue wire (thin)**: This will create a wire between two pins.

- **Diagonal blue wire (thick)**: This will create a bus.
- **Blue cross**: In case a pin should be left unconnected and unused, the blue cross can be placed on it. This will let the software know that it is okay to be left unconnected.
- **Wire with 'A' above it**: Placing a name above a wire gives it a name.
- **Label with 'A' in it**: In case dragging a wire makes the schematic cluttered, or the connection is towards another schematic page, a global label can be placed. Just like with the power nets, if multiple global labels share the same name, they will be connected.
- **R?? to R42**: This option will annotate components in case some components have no number or should be renumbered.
- **Op-amp and chip**: After making the schematic and checking the annotations, the real life footprint should be coupled to the component. Clicking this option will open up that menu. Since most devices (resistors, capacitors, MOSFETS and ICs) can be acquired in multiple sizes, take a good look at the datasheet and your BOM and use the built-in footprint viewer to ensure the correct component will be placed on the PCB. In case the wrong footprint is selected, it can be changed later.
- **Checklist with red arrow**: This feature checks whether the designed schematic is logical and throws errors if things are not okay, or warnings when things should be double-checked. In case an error can be ignored, the errors can be excluded by right-clicking the error message.

  To design our PCB, a netlist is needed. This draws blue lines (ratnests/rats) between footprints that should be connected together when we design a PCB. This can be generated by clicking File/Export/Export Netlist. A menu pops up with different options so you can pick which PCB design suite you want to export it for. Since we use KiCAD for this as well, keep it on KiCAD en press Export Netlist.

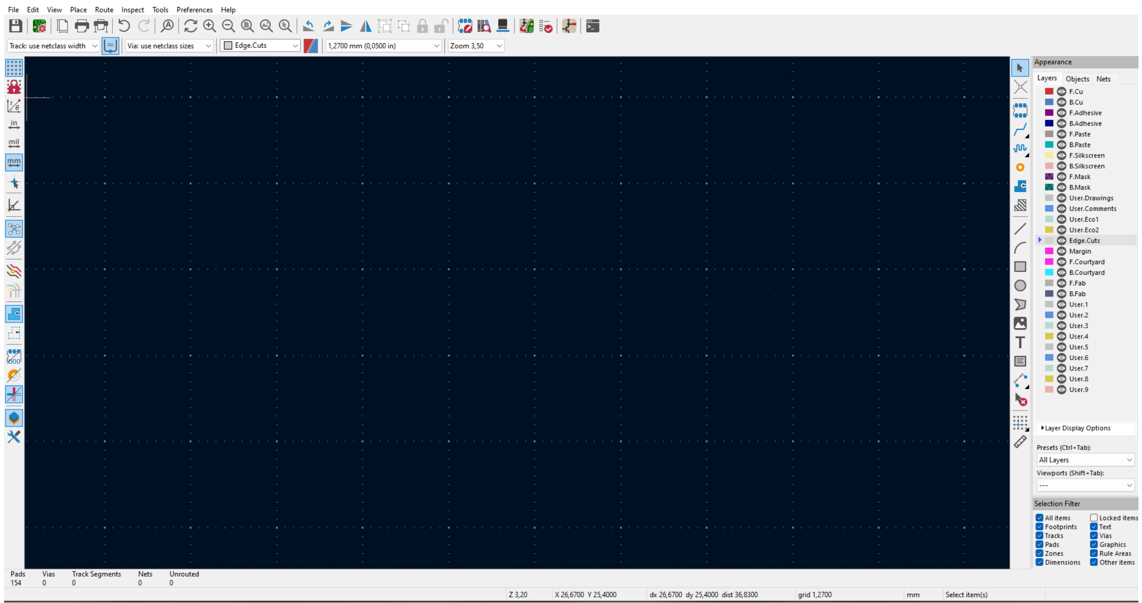

Figure 3: PCB layout editor

2.2 PCB layout editor

After creating the schematic, the PCB design itself can begin. From the project overview, the PCB editor can be opened. The overview of the tool is shown in Fig. 3 In order to load in the designed schematic, press the button in the top row resembling a conversion from schematic to PCB (fourth button from the right). Now options can be selected and 'Update PCB' can be pressed. Once again, if errors pop up, these have to be fixed beforehand.
Now all components of the schematic will pop up on screen and can be dragged towards the desired location. Make sure to set the grid size (next to the zoom bar) to something that is not too fine or course for the desired application. Components can be placed on top and bottom layers of the PCB, their location can be freely chosen.
If more than two layers are desired, this needs to be specified under File → Board Setup → Physical stackup. The amount of copper layers determine how many layers the PCB has. After this the dielectric thickness has to be verified for correctness.
The following symbols depict important functions on the right hand toolbar:

- **Thin bent blue line**: This option lets you route traces. The 'Track: use netclass width' defines the width of the trace, which determines its characteristic impedance and current carrying capability. For more information regarding current carrying capability, research IPC-2221. Pressing 'V' whilst laying traces switches the layer and connects the layers with a via.

- **Blue zig zag line**: This tool is used for length matching on existing traces.
- **Yellow ring**: This is a via placing tool, connecting layers together. The 'Vias: use netclass sizes' defines the sizes of the vias.
- **Blue shape with via**: This creates a copper pour over a specified area. The net has to be selected as well as the layer and pad connection. Use thermal reliefs when a large copper pour is used on a component that is not carrying a lot of current, and solid connections when the pad carries current. After creation of the pours, the zones still have to be filled, this can be done by right clicking this symbol.
- **Striped shape**: This creates an exclusion zone in at copper pour area, used for example under inductors to mitigate eddy currents.
- **'T'**: This places text on your PCB on the specified layer.

Important functions on top:

- **Capacitor through a board**: This is the built-in 3D-viewer, for viewing the PCB in 3D.
- **Checklist with red arrow**: This feature checks whether the designed PCB does not have any errors.

After creating the PCB, the files should be exported to the Gerber format. This is done by going to File → Fabrication Outputs → Gerbers (.gbr). Now the relevant layers should be selected, drill files should be generated followed by the generation of Gerber files. After this is done, the generated files should be compressed into a .zip file to be sent to the PCB manufacturer. At least the following layers have to be present for a 2-layer PCB:

- F.Cu - Front copper layer
- B.Cu - Back copper layer
- F.Silks - Front silkscreen layer
- B.Silks - Back silkscreen layer
- F.Mask - Front soldermask layer
- B.Mask - Back soldermask layer
- Edge.Cuts - Borders of the PCB

3 Some tips

3.1 Component Selection

Before getting started, important components such as ICs should be selected. Make sure the chosen components are solderable by the party that is going to solder the components to the PCB.

3.2 Schematic

When laying out the schematic, it is good practice to work with sheets as seen in Fig. 4. This creates a neat overview from which subcircuits can be managed. New sheets can be created on the root sheet by pressing Place → Add Sheet and then clicking somewhere within the root sheet. The sub-circuit sheet should also be structured, by seperating the sheet into parts and labelling them. An example can be seen in Fig. 5.

3.3 PCB

The component placement is a very important step in creating a PCB, it should take more time than routing the traces to the components to ensure a good PCB. Components cannot be placed in each others courtyards. Courtyards are the boundaries of the components.

Make sure that datasheets of the used ICs are read to see whether there are any routing constraints. An example of this could be a power supply IC, which will always say not to route the feedback line under switching nodes. Next to this, respect the PCB manufacturers constraints. If these are not respected, the PCB might not be produced correctly.

After placing the components, it is time to route the traces or lay copper pours. It is a good practice to make at least one ground pour covering the whole board. This provides easy and uniform access to signals going to ground. If there are more layers to spare, cover these with ground pours or even power planes (3.3 V,5 V). In case of high-speed PCB (PCBs with a lot of lines with signals larger than a few MHz), it is recommended to create a four layer PCB and use the inner two layers for signalling and the outer layers for the power planes. Make sure to only cross high-speed traces with other high-speed traces when absolutely necessary. Also try to not route high-speed traces close together, as this can incurr *crosstalk*. These signals are often differential or require a specific impedance, my personal recommendation is to calculate this with use of the *Saturn PCB design tool* (link). Furthermore, when switching layers when impedance matching, route two ground vias on the side of the switching point. It should be noted that switching layers can bring a different impedance. It is layer specific.

Make sure to couple ground pours together using enough vias. This ensures that noise spikes induced by components on board are minimized. Edge stitching can also be done. This tactic involves placing a lot of vias in a repeating pattern

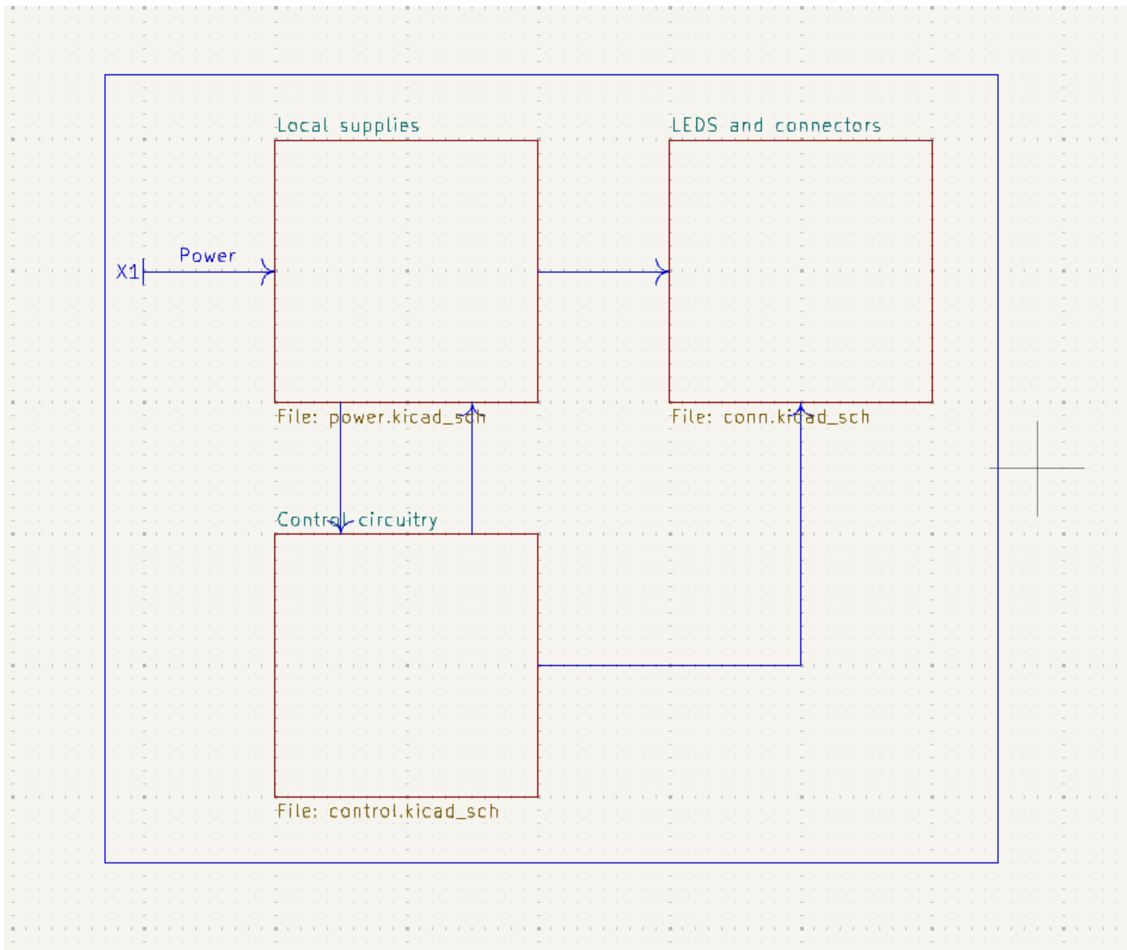

Figure 4: Root sheet

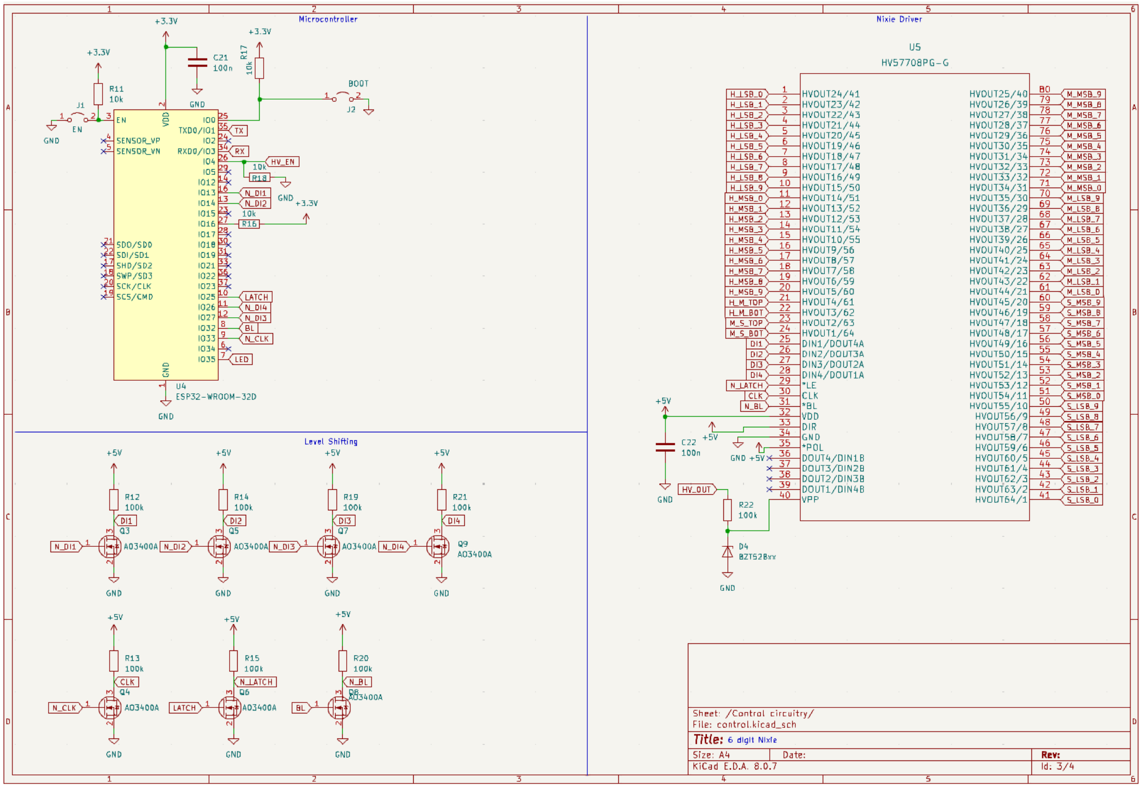

Figure 5: Sub-circuit sheet

along the edge of the PCB.
Do not forget to make a PCB outline, which highlights the edges of your PCB. Components should fall into this area (with an exception for overhanging connectors). It does not have to be square and can even feature arcs or circles.

It is strongly recommended to add text to the PCB such as the title, revision and connector names. This way, if more revisions are needed they can easily be discerned. This text can be added in the copper layer, soldermask layer or silkscreen layer.