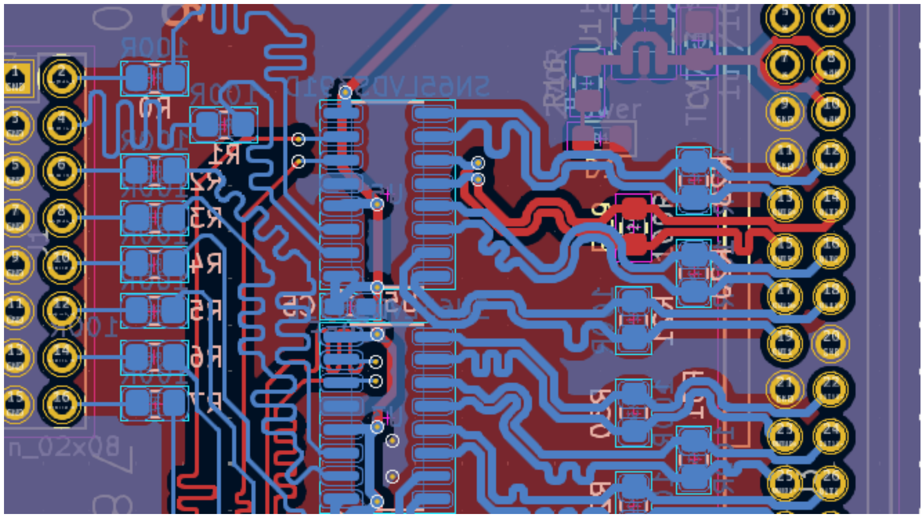

Guide to basic PCB design using KiCAD

B. Groenen
L. Lobato Azedo

April 28, 2026

Contents

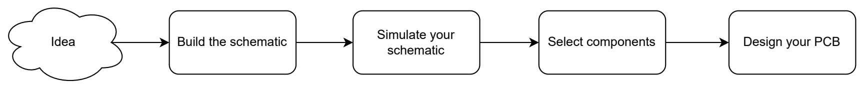

Figure 1: Flowchart of the process of creating a PCB

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.

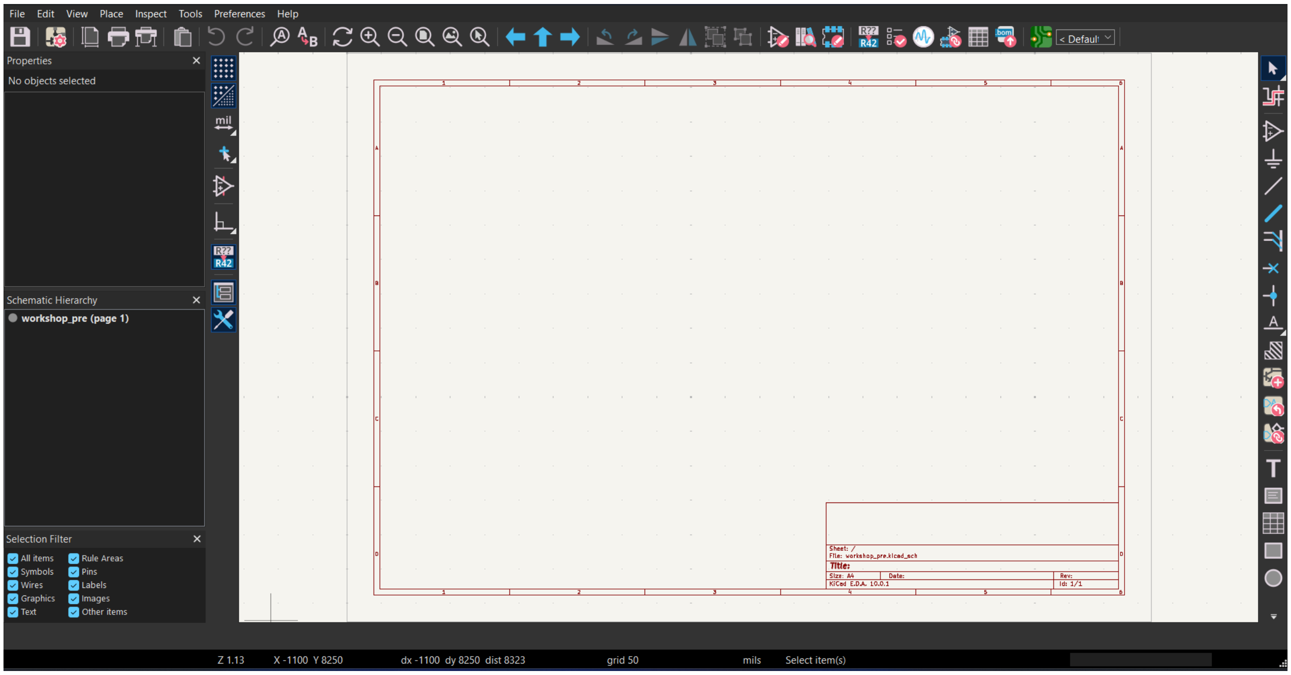

Figure 2: Schematic layout editor

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 symbol**: If clicked within the working space, a menu will pop up with all the available components. Alternatively you can use the key "A" on your keyboard. 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. Alternatively you can use the

key "P" on your keyboard. Placing multiple of these with the same voltage results in them being connected.

- **Diagonal grey wire (thin)**: This will create a wire between two pins. You can use the key "W" on your keyboard.
- **Diagonal blue wire (thick)**: This will create a bus. You can use the key "B" on your keyboard.
- **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. You can use the key "Q" on your keyboard.
- **Wire with 'A' above it**: Placing a name above a wire gives it a name. Wires with the same name are connected within the same sheet. This allows for your schematic to be cleaner since you don't need to have wires over eachother or very long wirews. You can use the key "L" on your keyboard. There are other more complex label types, but these won't be covered.
- **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.

  For the purpouse of a simple design, it is not necessary, and many times overkill to use all these features, but it is always good to know them. And of course, there are many more, but these won't be covered directly, unless you'd like to know more. One important aspect is to always save your progress within the current file, this can be done with a CTRL+S or in File ⇒ Save.

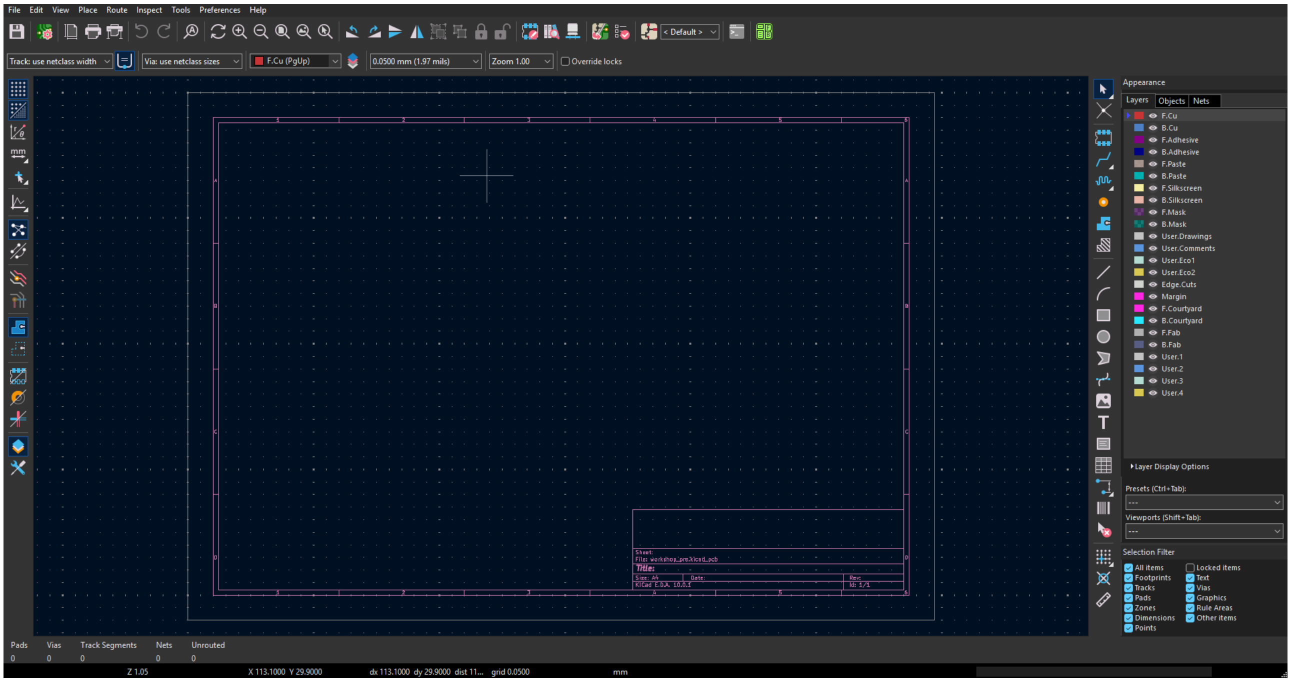

Figure 3: PCB layout editor

2.2 PCB layout editor

After creating the schematic, the PCB design itself can begin. From the project overview, or the schematic view, with the "Switch to PCB editor" button, 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 or press F8 in your keyboard. 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 (F.Cu or B.Cu respectively), 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. Alternatively you can use the key 'X' on your keyboard. 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. Alternatively you can use CTRL+Shift+X on your keyboard.
- **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 or pressing 'B' on your keyboard.
- **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

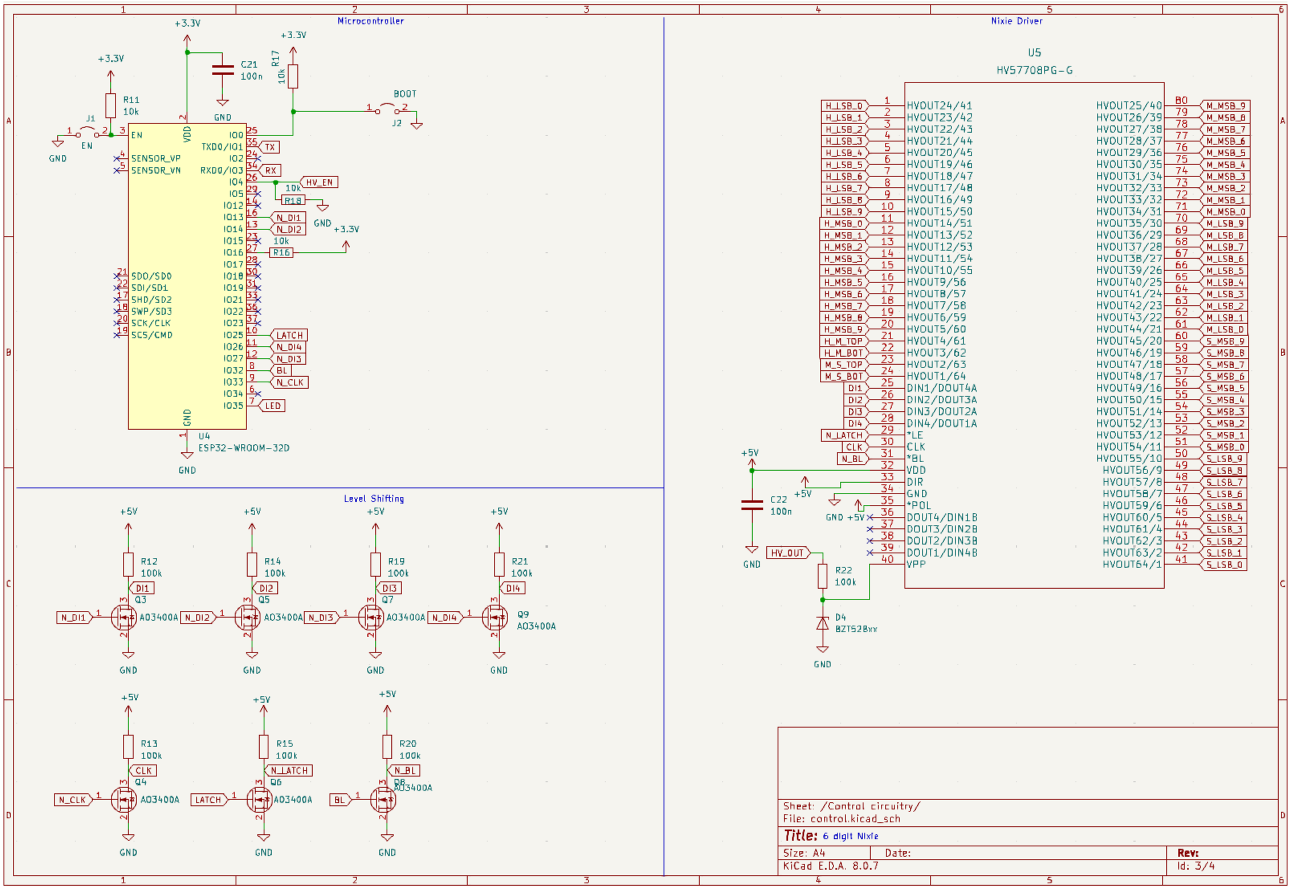

Figure 4: Sub-circuit sheet

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

The circuit sheet should be structured, by seperating it into parts and labelling them. An example can be seen in Fig. 4.

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. For high current traces, a thin line will not suffice, as it it's resistance is too high and will overheat and maybe burn, always consider calculating the necessary minimum width with a tool such as *Digikey's PCB Trace Width Calculator* (link).

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 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.

4 Design Example

Assume a 5 V input signal has to be downconverted to 3.3 V and fed to a connector, and to see whether it is active, it should drive an LED.
Important: For your board to be manufacured by Volundr, pay attention to the this example, as some settings can make your design unfeasible. The apropriate settings will be highlighted with **bold text.**

4.1 Power supply

First, the converter from 5 V to 3.3 V chosen is the LM1117. In the datasheet [1], it states that it should be driven as seen as in Fig. 5

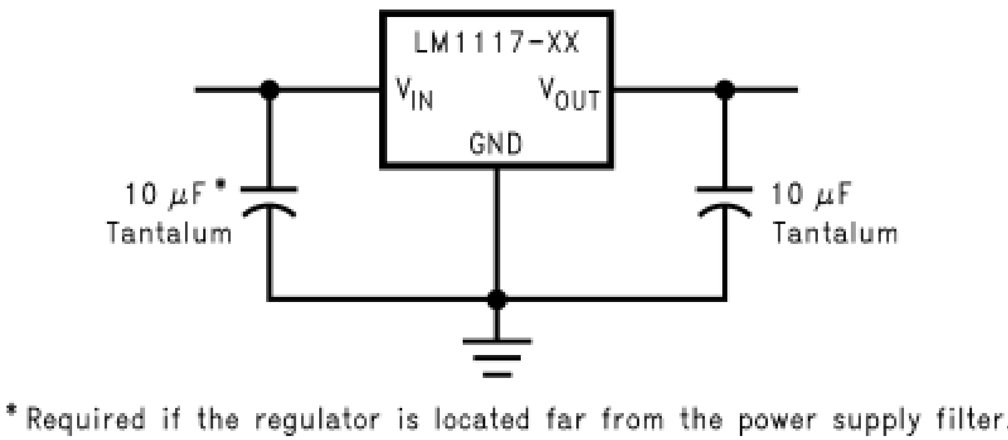

Figure 5: Power supply circuit with the LM1117

As seen in the picture, the IC should be driven by two capacitors of 10 μF. Now it is time to choose the capacitors. Any arbitrary 10 μF capacitor will do, the 12065D106MAT2A (arbitrary) from Kyocera is chosen [2]. The supplier states that the package is 1206 (Name for the SMD package).

4.2 LED

Now an arbitrary LED can be chosen. For this example, the Kingbright2 V Green LED 5mm Through Hole is chosen [3]. For a voltage of 2 V typically according to the datasheet it consumes a current of 5 mA. This means the supply voltage of 3.3 V should drop 1.3 V under a current of 5 mA. With Ohm's law, this equates to a resistor of 260 Ω. This leads to the schematic in Fig. 6.

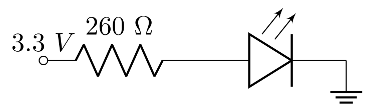

Figure 6: Schematic LED

4.3 Connectors

Once again, any arbitrary connectors can be used, in this case 2x2.54 mm male headers will be used.

4.4 Approach

4.4.1 Schematic Setup

It is best practice to, before starting anithing meaninfull, having setting up your page so other designers, and yoursef knows what is going to be there. Steps that can be regarded as this are:

1. Having your page settings as you wish, this is under File→Page Settings, an example is Fig. 7
2. Organizing your shematic by separating sub circuits on the page with lines in between, to draw a line click the key "I" or select the "Draw lines" tool on the right
3. All wires with the same name are internally connected to the same net (you do this by placing net lables, using key "L" or the tool on the right ribbon), and this also applies for same power symbols, such as GND and 3V3, use this knowladge to make your design neater

4.4.2 Schematic

Since the preparations are done, KiCAD can be opened and a new project can be created. Since this circuit is so small, no subsheets will be used and everything will be created in the main sheet. Insert all the components, give them values where necessary and organize them on the sheet. Afterwards, it should look something like Fig. 8.

4.4.3 PCB setup

This section is important for manufacturability, so read and see figrues carefully. On your PCB Editor you have a page called board setup, this page most importantly determines your constraints, pre determined sized and layers. It can be opened by accesing Files→Board Setup, here let's start with the Phisical Stackup, **for the purpouse of manufacturing, keep it default, so it should look like Fig. 9**.

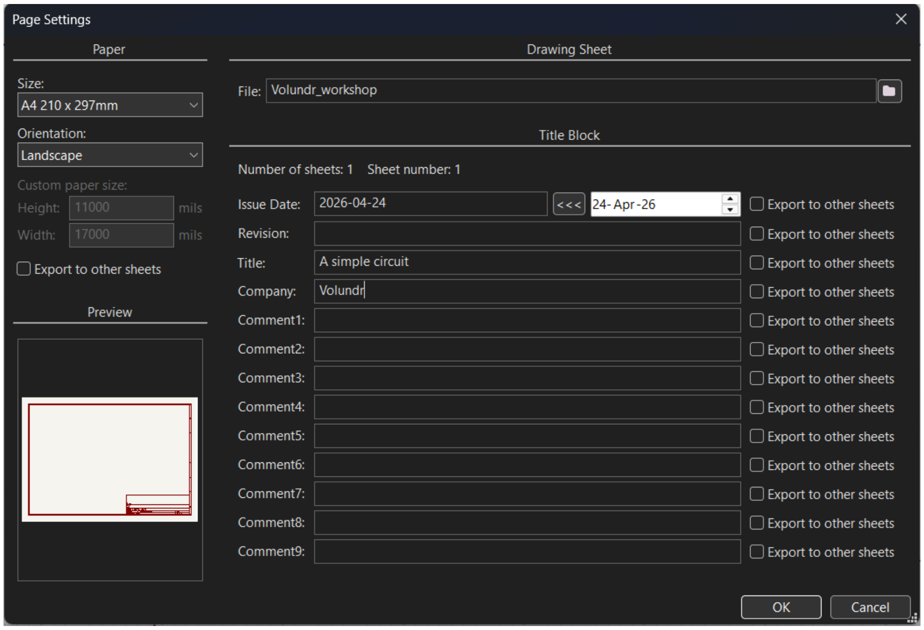

Figure 7: How your page settings might look after setting up

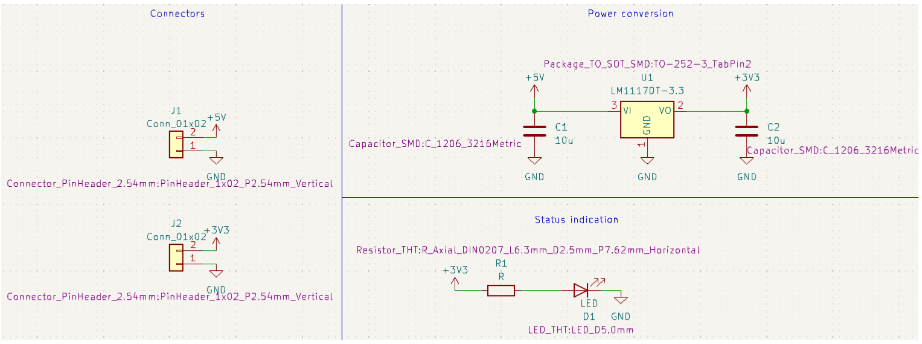

Figure 8: Schematic including footprints (not necessary in actual schematic)

Constraints are another area of importance, as they prevent you from making mistakes, **for the purpouse of manufacturing, keep it like shown in Fig. 10, note that we will not manufacture your board if it has microvias, as they are relitively expensive to make**.

The last page you should look are Pre-defined Sizes, there you can set sizes for the tracks and vias you will make. **Make sure these are not smalled than the sizes we set the constraints to**.

One section you can also use it netclases, but this type of feature is more suited to a more complex design, so it won't be covered.

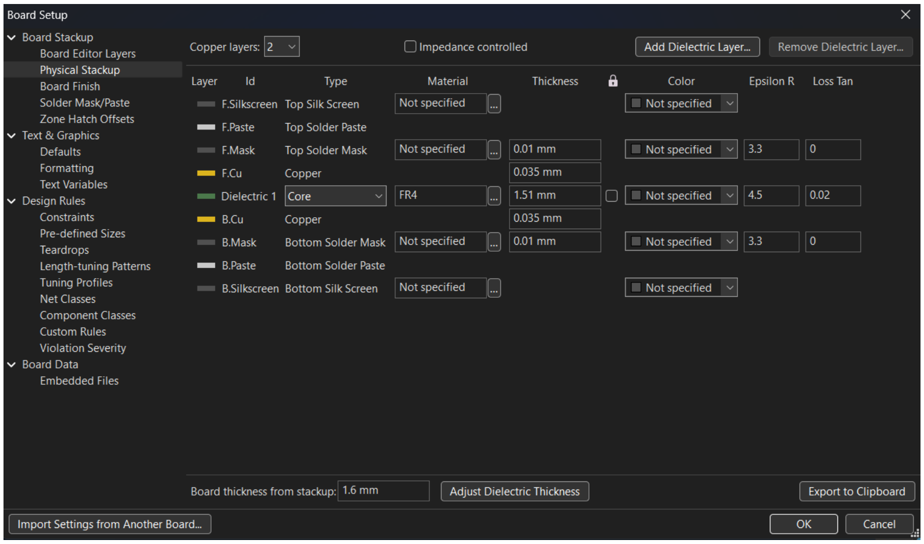

Figure 9: Stackup of your Board

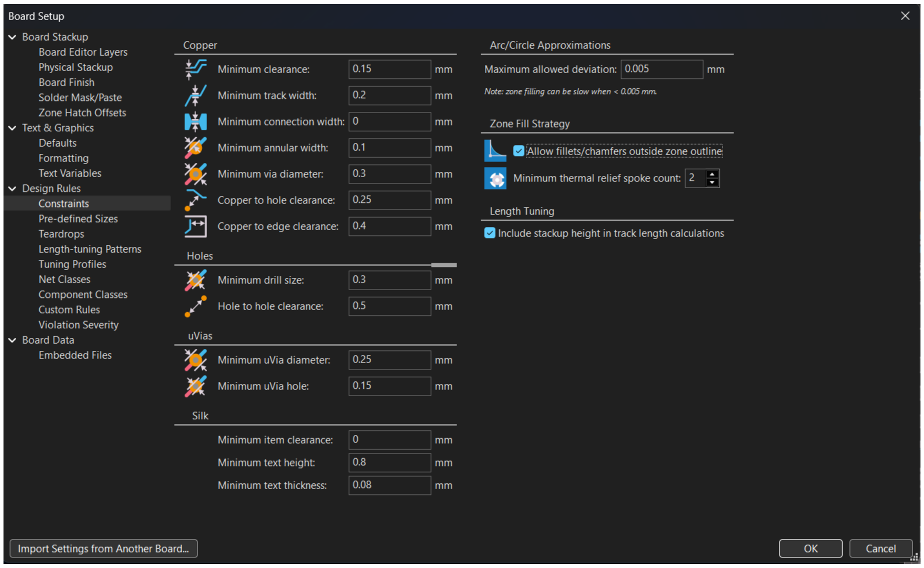

Figure 10: Board constraints

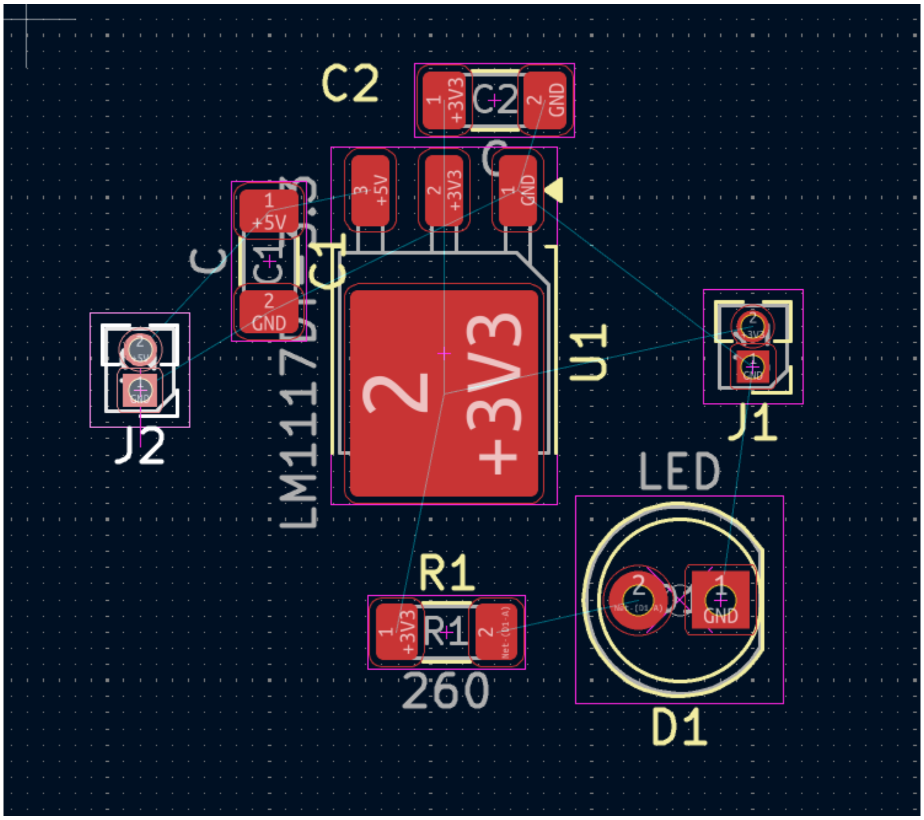

Figure 11: Placement of components

4.4.4 PCB

Now the PCB can be created and the components can be placed where desired. How to do this is stated in Section 2.2. An example of component placement where compactness is kept into consideration is shown in Fig. 11. There will be blue Ratsnest lines visible, this is the line between component pins that should be connected.
Since the connectors have no text saying what should be connected. It should be added in this stage. Next to that, the board outline can be drawn on the edge cuts layer. The result is visible in Fig. 12 The LM1117 can handle a current draw of 800 mA, which means according to the IPC-2221, a trace width of at least 0.434 mm is required to handle 800 mA while maximally heating 10 $^{\circ}C$. Also make sure to place a ground pour in at least one layer. In the end, it could look something like shown in Fig. 13 and Fig. 14. The last step is to run a DRC with the checklist with the a red arrow symbol on the top ribbon, discrepancies and mistakes will be pointed out there, make sure you are either aware and accept of the current warnings and errors (some things might look wrong for

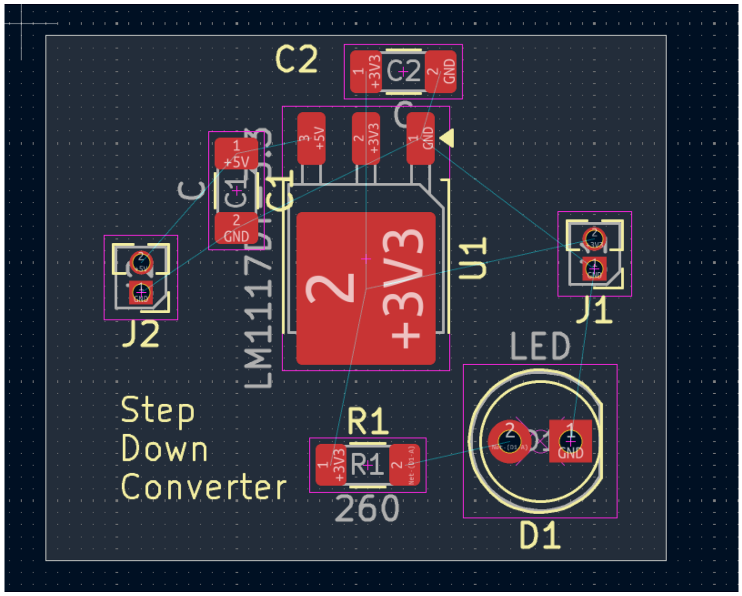

Figure 12: Text and edge placement

the software but are a certain design choice) or, ideally, that there are none.

4.4.5 Manufacting files

For you to have your board made this step is very important, so follow it strictly. Once your design is ready to proceed from an idea to a reality this step is what gives manufacures the necessary details to do so.

On the top left of your page click on File→ Fabrication Outputs → Gerbers, you should see a page like in Fig. 15. Leave all as it is presented to you, exept that you should make a directory to put your gerbers in, it is fine if that directory is inside your project directory. Click on **Generate Drill Files**, leave default settings and click **generate**, then close that small window. After this click **plot** on the page as it is in Fig. 15. Your directory should have somethin like what is on 16. Put all these on a .zip file and with this your design is complete.

4.4.6 Sending your results

If you want this design to be made and delivered we can help with that, we only need your zip file sent to volundr@thor.edu and you should mention in the email if you are a thor member or not (fee is the same). You have a week counting from the day the workshop takes place to send it and the delivery tends to not be very fast, but you'll know once it arrives.

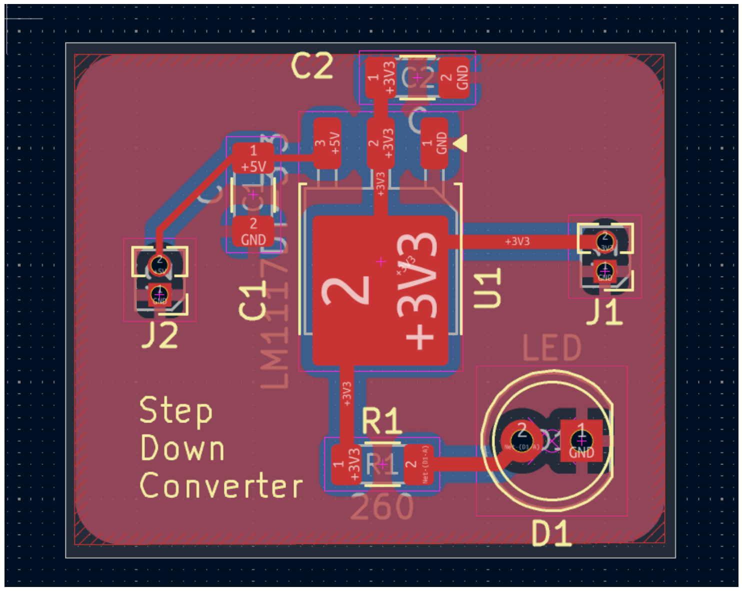

Figure 13: Final PCB

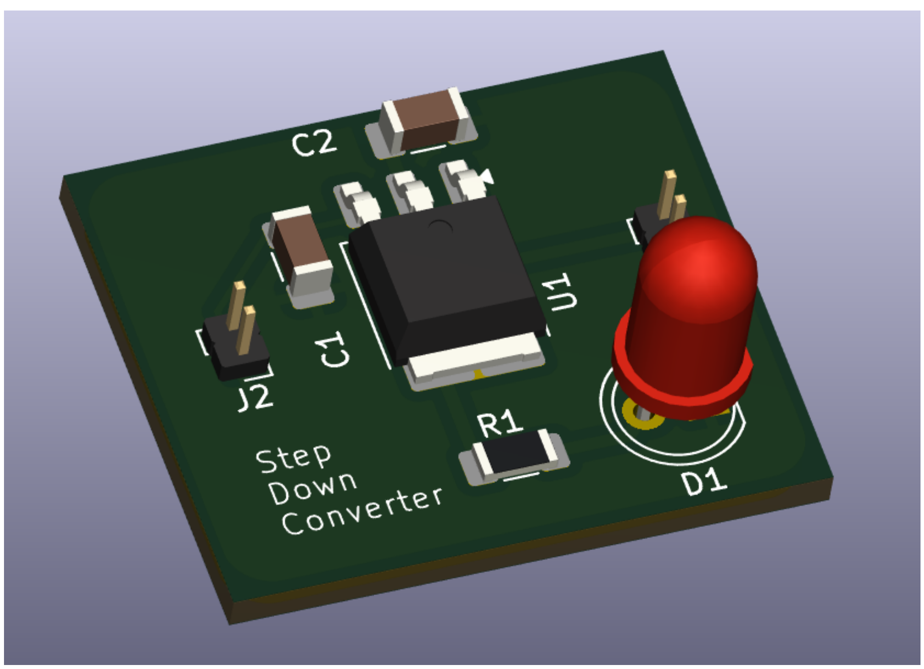

Figure 14: 3D-view

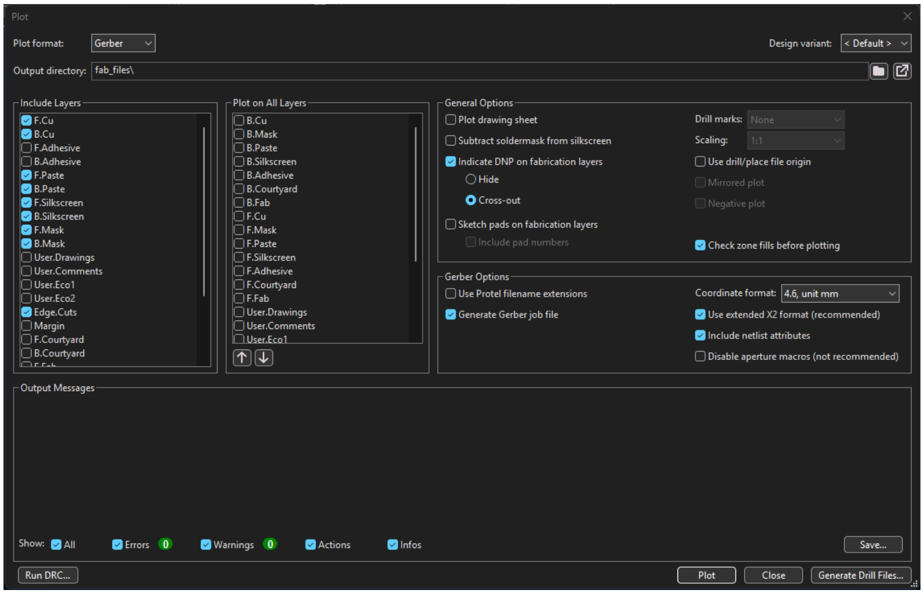

Figure 15: Page to set your manufacturing files

Figure 16: Manfucating files in directory

References

[1] T. Instruments, “Lm1117 800-ma, low-dropout linear regulator.” [Online]. Available: https://www.ti.com/lit/ds/symlink/lm1117.pdf

[2] Kyocera, “X5r dielectric, kgm series.” [Online]. Available: https://nl.mouser.com/datasheet/2/40/cx5r_KGM-3223198.pdf

[3] Kingbright, “T-1 3/4 (5mm) solid state lamp.” [Online]. Available: https://docs.rs-online.com/4066/0900766b8151e408.pdf