Guide to basic PCB design using KiCad

B. Groenen

March 2021

Contents

1   Introduction

In this guide, the free open-source software "KiCad" is used. KiCad is a powerful solution which offers a 3D viewer and support for layouts with 32 copper layers. Furthermore, this guide will show step by step how to start designing basic two-layer PCB's Figure 1 follows which displays the general steps you should take designing a PCB.
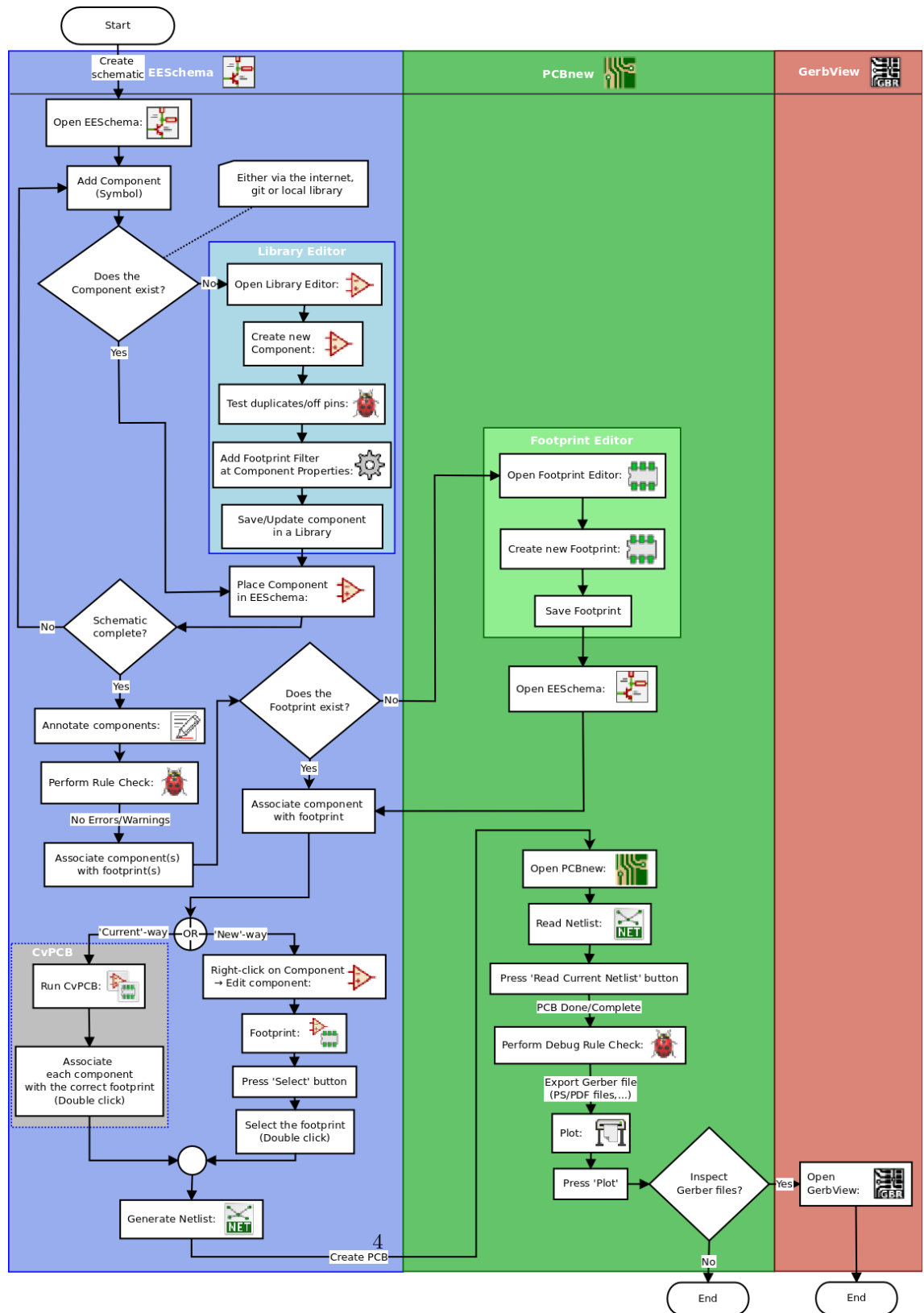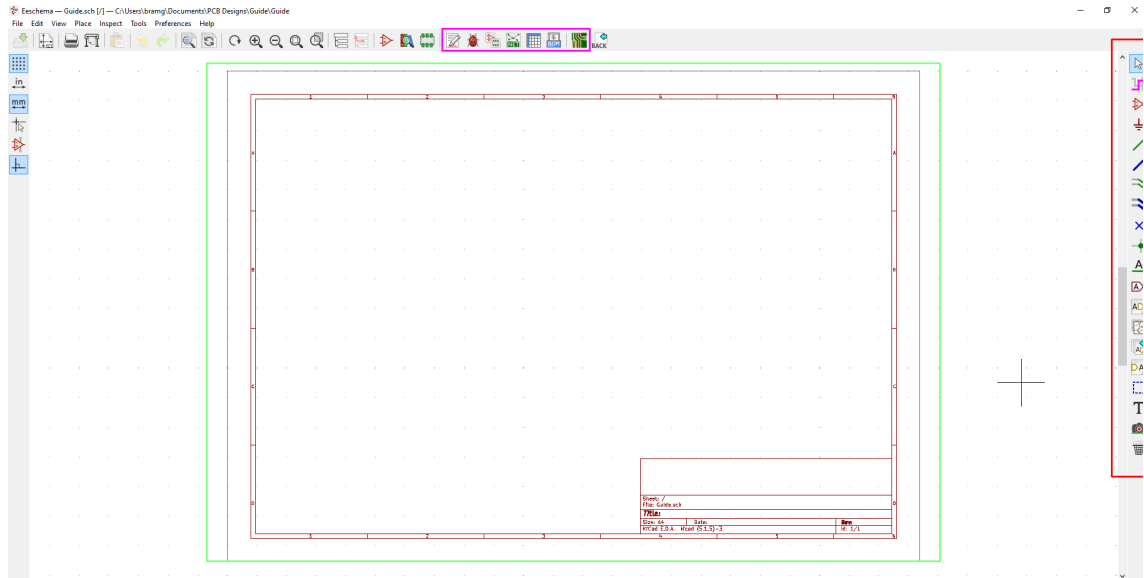
Figure 1: Flowchart with steps

Figure 2: Schematic layout editor

2 Schematic layout editor

When you open KiCad, you are greeted with a screen that shows your last edited project. Here you can create a new project and open the "schematic layout editor". This is where you create your electronic/electrical circuit that you want to build a PCB for. The following colors depict what options are of importance:

- **Green**: This is the area from which you work and on which you design your circuit.

- **Red**: Over here the tools for designing your circuit are placed. If you click the Op-amp symbol and then click on where you want to place a component, you can choose from a wide variety of components to place. To connect these components, click the single diagonal green wire and click on the terminals you want to connect. In case a junction is wanted, either click the green dot and place it, or click with the green wire from the start location to the wire. All terminals have to have a connection, or a "No connection". This "No connection" can be placed by clicking the blue cross and placing it on the terminal that should not left unconnected.

  In case you have a lot of connections on the same line, it might be easier to use global labels, which is the red shape with the letter 'A' in it. With this, names for nets can be defined, nets are connections to the same node. For example, in a circuit in which 5 V is needed on a lot of connections,

make a global label with 5 V and connect that label to every point that needs 5 V.

- **Pink**: In the pink area, the tools are located for preparing the PCB for designing. To start off, you need to annotate the components and give each individual component a name. Luckily, this doesn't have to be done by hand, but this is done automatically with the annotate function. If you click the "white paper with the red marker" icon, the annotating menu pops up. In here you can just click annotate and then it's done.

  Next, to see if the circuit isn't faulty, you need to do a DRC, short for Design Rule Check. This checks whether there are any faults in the circuit like unconnected terminals. Once the menu is open, press run to see af there are any errors. Any errors occurring should be displayed in the dialog. In case any errors are present, try to fix them, annotate the circuit again, and then check if the errors still persist.

  Since a lot of components can be acquired in multiple packages/types, footprints have to be assigned so the program knows which component package/type you will be using exactly. To do this, press the Op-amp symbol with an IC in front of it. Here you can select a footprint for every component, also for connectors and battery-holders. In case they are wrong or don't fit, you can change these later. Make sure to give each and every component a footprint. It can be that the required footprint for your project is not listed. If that is the case, try to download a library and load it into KiCad. How to load libraries will be discussed later.

  To design our PCB, a netlist is needed. This draws white lines (rat-nests) between components/footprints that should be connected together when we design a PCB. This can be generated by clicking the icon that features an odd cross and says NET in the lower half. Just press "Generate Netlist" when the menu opens and that generates one.

  Now we can run Pcbnew, the actual PCB designer of the software. It can be found by clicking the small PCB icon. This opens a black screen which will be talked about in the next chapter.
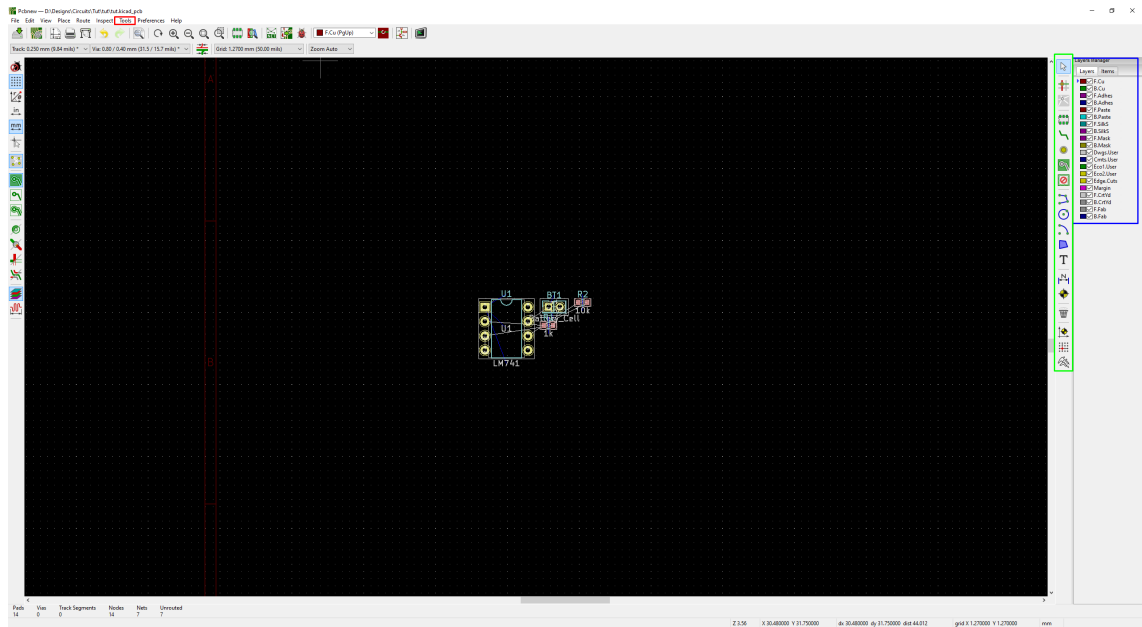
Figure 3: PCB layout editor

3 Pcbnew

Now we are in the actual PCB design part of the software. It has a wide variety of tools that can be used. In this guide, we will be using only a few of these tools.

3.1 Loading in your circuit

Loading in your circuit is very easy. To do this, open the "Tools" bar seen in the red area in figure 3 and select load netlist. Then select the netlist that you just generated in the schematic layout editor and press "Update PCB"

3.2 Laying out your circuit

Now that your circuit is loaded in, you can drag it around within the red sheet that you see in front of you. This is the area you work in. You can select individual components to drag them on a place you would like it to be. Take enough time to lay out your circuit efficiently to so it fits on a smaller surface.

3.3 Making an outline

When you are happy with the layout of your circuit, you can make an outline with the tools provided in the green area. In that area, select "Add graphic

lines" (if you want rounded corners select "Add graphic arc") and select the area you want your PCB to be.

3.4 Laying traces and copper pours

3.4.1 Traces

In order to connect the components together, traces or copper pours should be laid down. Traces are the "wires" connecting two or more components/pins together. Traces can be laid using the "Route track" tool seen in the green area in figure 3. To lay a track, click the point you want to route from and where you want to route to. Sometimes, one side simply isn't enough, and two or more layers have to be used. In this guide, we will focus on 2-layer PCB's only, which means that the toplayer and bottom layer can be used. To switch layers, press "v" whilst not laying a trace.

3.4.2 Traces between layers

Sometimes, traces have to get to a component on the other side of the board. To do this, tiny holes in the PCB that connect to both sides are used. These tiny holes that connect both sides are called "vias". To lay a via, route one part of the trace on one side, press "v" and continue the trace on the other side to the goal.

3.4.3 No-fill zone

Say that you want to have an area where nothing can be routed or poured, then you add a no-fill zone. No tracks can be routed in there, nor can vias or copper pours be laid there. To do this, click the "Add keepout areas" tool in the green highlighted area in figure 3 and click where you want the area to start. You can select in the menu which layer you want the zone to be and what to keep out. If you want copper pours to keep out that area, lay these before you lay the copper pour.

3.4.4 Copper pours

A copper pour is an area on the PCB filled with copper. Copper pours can be used for: heatsinks, ground/voltage planes, EMI reduction and more things. To lay a copper pour, first click where you want the pour to start. Then a menu will pop up and you select the layer and net you want the pour to connect to. If you want the pour to be "floating", select the "¡no net¿" option. Then, you can select an area where you want your copper pour.

3.4.5 Adding text to your PCB

You can add text or images on the silkscreen layer or the copper layers on the board. Click the "Add text on copper layer or graphic text" tool, click where

you want the text to be. Then a menu will pop up where you can configure how you want the text and on which layer. If you want text in the copper layers, select either the layers "F.Cu" or "B.Cu". If you want the text on the silkscreen layers use either the "F.Silks" or the "B.Silks" layer.

3.4.6   Soldermaskless areas

Sometimes, you need some bare copper on your board or need the soldermask to be absent. To do this, select either the "F.Mask" or the "B.Mask" layer, select the "Add graphic lines" or "Add graphic arc" tool and select the area you don't want a soldermask to be present.

3.5   Exporting your PCB for production

If you want to export your PCB for production, Gerber files have to be created to be sent to the PCB fabricator. First, see if your PCB has any DRC errors. To do this, press the red ladybug tool and press run DRC. If no errors are here, cou can continue. If not solve them before continuing. Next, press the "File" menu seen in figure 3 and then press "Plot". Here, you select which layers have to be fabricated. You need to at least have the following layers to fabricate a 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

Once you have these layers or more, you can select an output folder and then press "Generate Drill files" and then press "Generate Drill files". This lets the machine in the factory know where holes should be. Next press "Close" and press "Plot". Now all the files are generated in the output folder you selected. Place all these generated files in a .zip file and now it is ready to be sent to the PCB fabricator!

4   Loading in libraries

4.1   Footprints

4.1.1   Preparing the files

To import a footprintlibrary into KiCad, files have to be prepared first. Make a folder in the directory where you want your library to be stored. Tip: Make
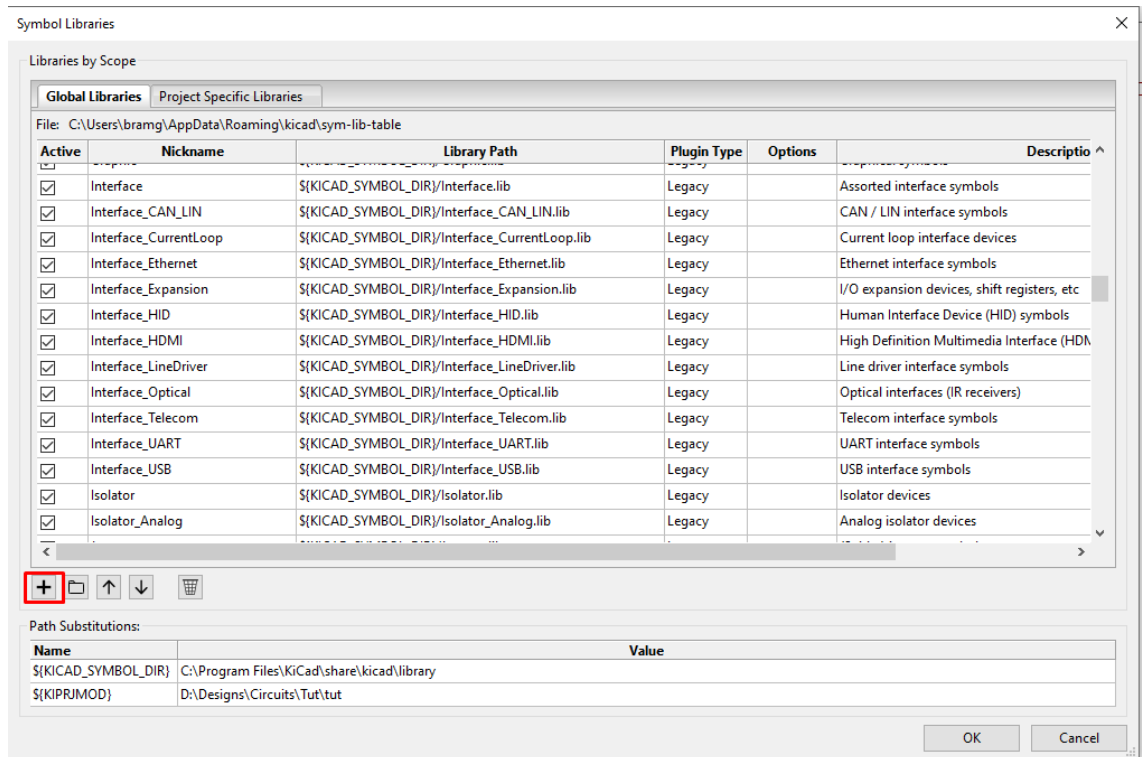
Figure 4: Footprint management area

this a permanent location easy to find. Then, name your newly created folder INSERTLIBRARYNAME.pretty. In here, you drop the files for your footprints, footprintfiles you drop in here have the ".kicad_mod" file extension.

4.1.2   Importing the created library

To import a footprintlibary into KiCad, you need to go to the "Assign PCB footprints to schematic symbols" tool seen in the pink area in figure 2, and then select "Manage footprint libraries" in the "Preferences" menu. Then press the + symbol highlighted in the red area in figure 4. Here, you can give your library a name and path to the library. This path will look something like: C:/Users/USER/PATH/INSERTLIBRARYNAME.pretty. Now the library is loaded in and is ready for use.

4.2   Symbols

4.2.1   Preparing the files

To import a symbollibrary into KiCad, the contents of the library download have to be dragged into a folder where you want the library to be. Tip: Make this a permanent location easy to find. Next, press "Manage symbol libraries" in the "Preferences" menu. Here, although file extensions are different of the ones used in figure 4, press the + button seen in figure 4 and name your library and put the path of the library in. The path will look something like: C:/Users/USER/PATH/INSERTLIBRARYNAME.lib. The library is now imported and ready for use.