Oleg Khokhlan
EEC 134 Senior Design
Baseband PCB
Application Notes
June 2016

I. Objective

For all students, I recommend that you watch KiCad tutorials on Youtube first, and use this paper as a secondary source. The goal of this paper is to supplement the videos. I will attempt to explain and show steps that may have been left out or are unclear in most tutorials.

II. Paper Outline

a. Introduction to KiCad
b. Schematic Symbols
c. CvPCB List
d. PCB New
e. Gerber Files

III. Introduction to KiCad

KiCad can be downloaded for free from http://kicad-pcb.org/ (if the link expired, Google KiCad). This software works on most operating systems. Upon installing the software and opening the program, you will Figure 1. This is the home screen of KiCad which acts as a menu for the tools and functions.
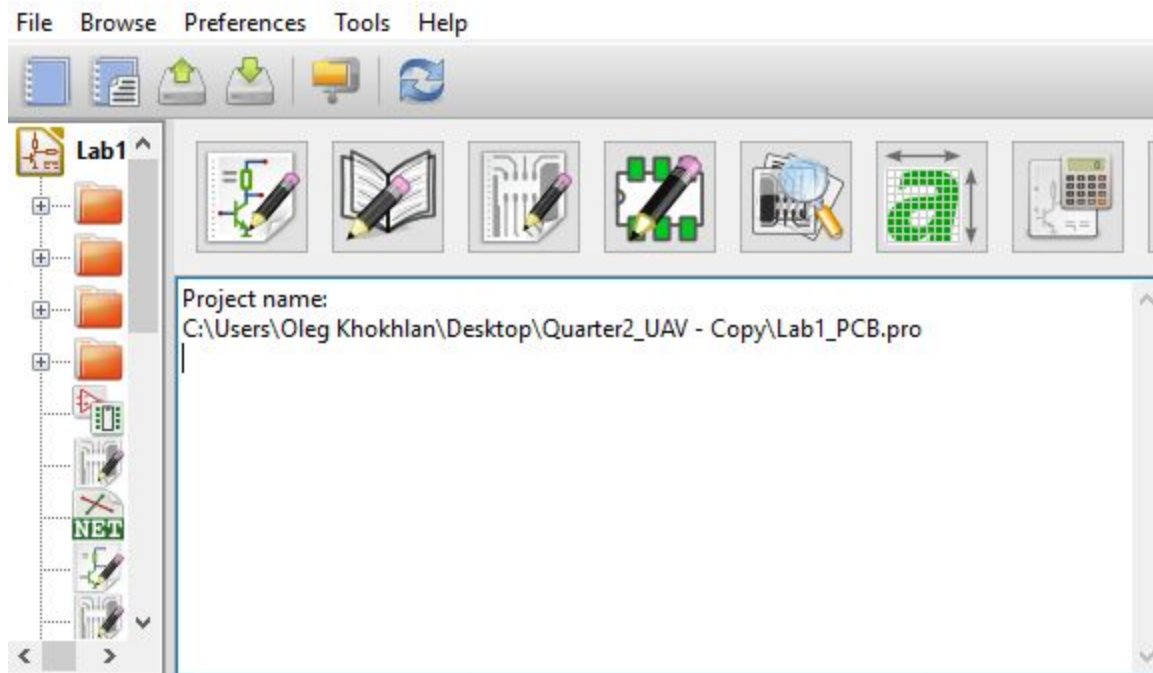

Figure 1

First step, start a new project. Click on File > New Project. Avoid naming the project with a generic name such as project1 or PCB1. Give it a relevant and a specific name to avoid confusion in the future when you have multiple projects saved. After it will ask if you would like to create a new directory, click yes.

: All files (schematic, pcb, and library) will be saved and should be saved in the project folder. The folder you created in the step above.

Figure 2

A. Introduction to KiCad layout: (boxes refer to fig. 2)

Eeschema - electronic schematic editor.

First box from the left. This is the first step of the design process. You will build a schematic. Basically, replicate the one you have. Similar approach as in 180A lab.

Schematic library editor

Second box from the left. KiCad, by default, has a large number of popular schematic symbols. However, it does not have everything, including a large number of components used in EEC 134. This editor allows you to manually design custom schematic components.

Pcbnew

Third box refers to a printed circuit board editor. After designing the schematic and assigning footprints, you will finally be able to create the layout for your PCB.

PCB footprint editor

Fourth box from the left. Same as the schematic library editor. It allows you to build custom footprints for your components which KiCad may not have.

Note: Footprints are physical layouts for components. In your schematics, you will have some component such as a resistor labeled "R8" with a value of say $1k\,\Omega$ which is just a symbol. This does not provide any details on the physical representation of the actual resistor. It is just a value and a name. Footprints will consider the physical dimensions of the resistor and the type (thru-hole or SMD). Notice the difference below.
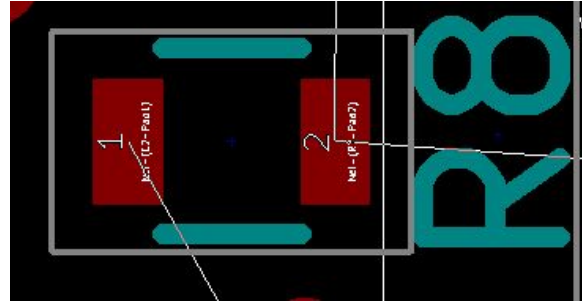
Figure 3 - Schematic R

Figure 4 - Footprint of R (SMD)

B. EEschema - Electronic Schematic Editor (1st box) - Symbols

As mentioned, the first step is to build a schematic of your design. This schematic should look exactly as the schematic you may have on paper. This schematic acts as a map for the PCB editor.

Clicking on the 1st box will open up a blank page. At the rightmost side of the screen, you will see a lot of tools. We will quickly go over them.
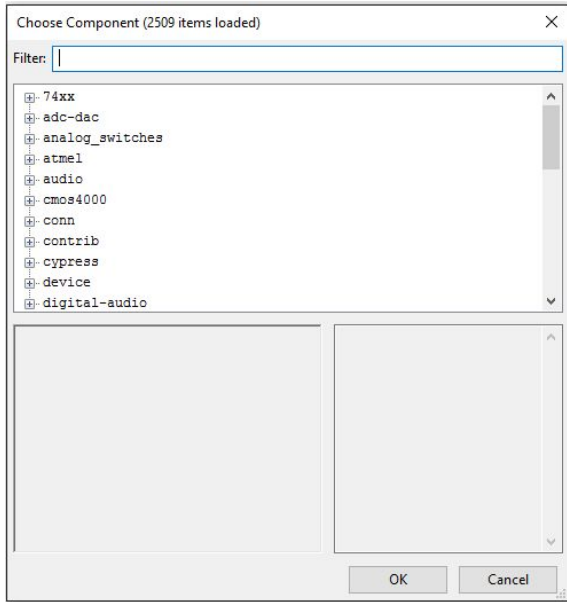
Note: After clicking on a box, click on the schematic page for dialogue boxes or wires to show.

	Click here to go back to a mouse pointer, and escape from other tools.
	"Place Component" - Click on this box, then once on the schematic page to open a component search box. See below.

This component search box lists all of the schematic symbols within Kicad. If you know the name of the symbol, you can filter search. If not, you can look through the different files.

Use deduction to narrow the search.

Ex. The file adc-dac will have ADC symbols and the file name 74xx will have symbols for ICs which are in the 74xx series.

Tips: Type R for a resistor, and C for a capacitor.

This box is the component search box equivalent for voltage sources (VCC, VDD, 5V, etc...) and ground.

The green wire (top) is for connecting pins as in any schematic.

Tips: Shortcut for the wire is "w". Double click to end the wire. Clicking once will not end it. It will maintain its shape from that point allowing you to continue without fear of the wire shifting its position.

The blue wire on the bottom is for a bus, which I have not used, and will not be covered.

This is a "no connect". It is attached to unconnected pins (pins which are floating).

This is a junction. Basically a giant circular wire connection. Junctions also form by default whenever two wires join.

Note: Two wires crossing (without a junction) is not a connection. It is the software equivalent of jumping over the other wire. Junctions, on the other hand, symbolize connections.

Be careful connecting TO a junction
I had an issue where a wire going through the junction was not actually connected. This resulted in a floating pin which I had to manually jump on the PCB. Always double check your junctions. This issue comes up when you are trying to connect a wire to an already existing junction.

The **Place net name - local label** allows you to label wires to reduce the amount of them you can see.
Ex. Label a node coming from a 5V source as "5V".
Then label all other wires which would be connected to the 5V node as "5V".

Now you know enough to build a schematic.

C. CvPCB List

After finishing your schematic, it is time to do the CvPCB list which maps footprints to your components. First, click on

(tool bar above schematic) to annotate (number) all components. Adjust the settings if you like. Next, click on

To create a netlist. You must update the netlist anytime you make an update to your schematic. To update, simply click on it again. Now, click on

To open a new window for the CvPCB page. You should get something like this:

On the left, you have a selection of different components (R ,C , etc...).

In the middle are all of your components from the schematic.

On the right are the different varieties (different sizes and types of the same component) of the component you selected on the right.

For example, first click on one of your components (middle column). Say C1. Now find the Capacitor selection of the left column. Click on that. On the right column you should see different sizes for capacitors. Select the one you need. You can view footprints by clicking on

Remember to save. Now, it is time to go back and update the netlist. Update the netlist by going back to the schematic page and clicking on the box (below) again.

Creating Footprints

If you cannot find the necessary component, you must create your own. You will find that you may have to do so for almost 50% of your design.

There are many tutorials on this process, so I will not cover it.

D. PCB New

To open, select the 3rd box on the main KiCad window. You should get a blank window. Click on

Select "Read Current Netlist". Afterwards you get a pile of components stacked onto each other. At this point, you have to manually separate the individual components manually. As you are separating them, spread them out first. Choose the most important components, and place them where you need them. Then begin placing smaller components around them.

To connect the components, use

To lay a trace for the PCB. Keep the trace width in mind. That can be accessed in Design Rules > Design Rules. Then change the Track Width.

The trace/track can be either on top or the bottom of the board. Red means F. Cu which is front copper. Green is B.Cu which is back copper. They may connect through a via which is selected by clicking on "v" on the keyboard while laying out the trace.

Tips: Avoid 90 degree angles for fabrication related reason.

To do edges, select the Edge.Cuts at the right of the screen. Then click on

To draw the line of the screen.

Instead of using trace for GND, fill the PCB with copper which is grounded. Use

To draw the trace (should be just inside the PCB outline). Do it for front and back. Then pour copper in. In settings, make sure that the copper selected is GND.

Once you place all components, think about the PCB size. You can measure the distance between edges by taking note of the x y coordinates at the bottom of the screen.
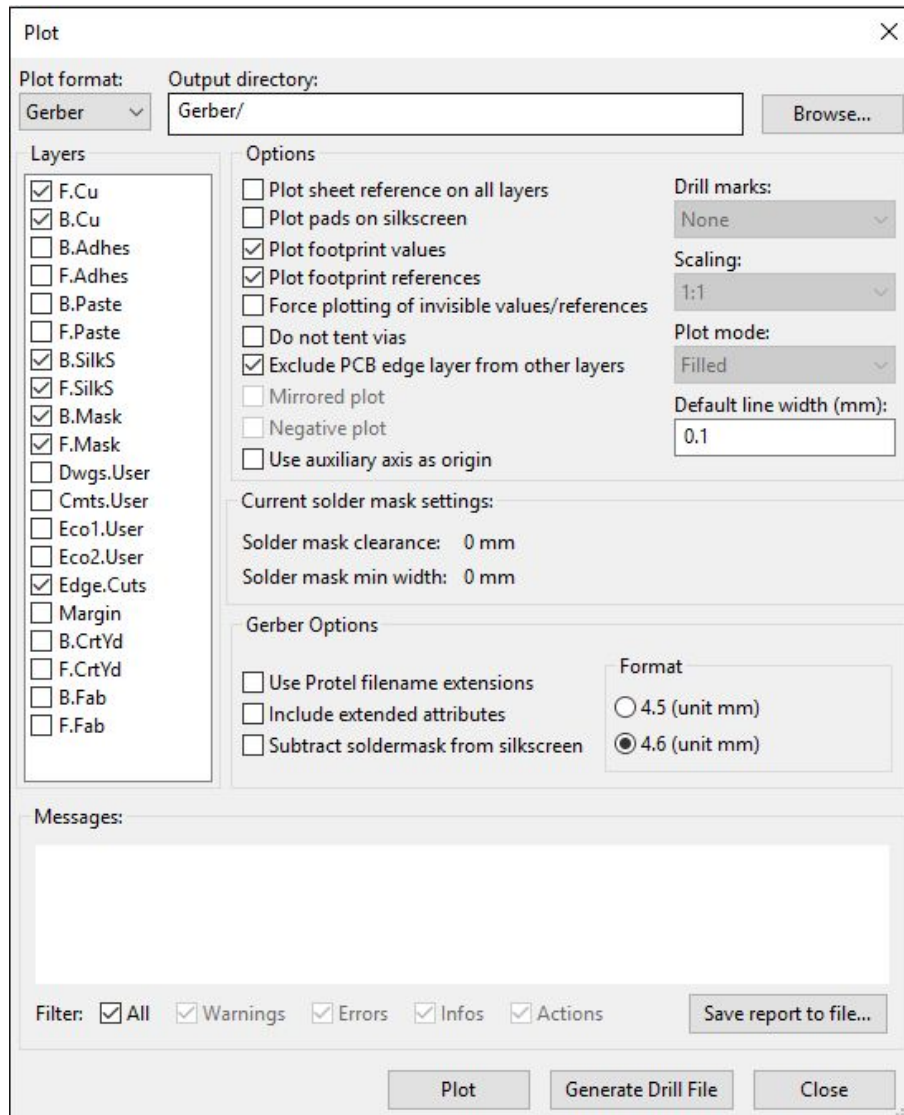
E. Gerber Files

These files contain vital information concerning PCB manufacturing. To plot these files, select
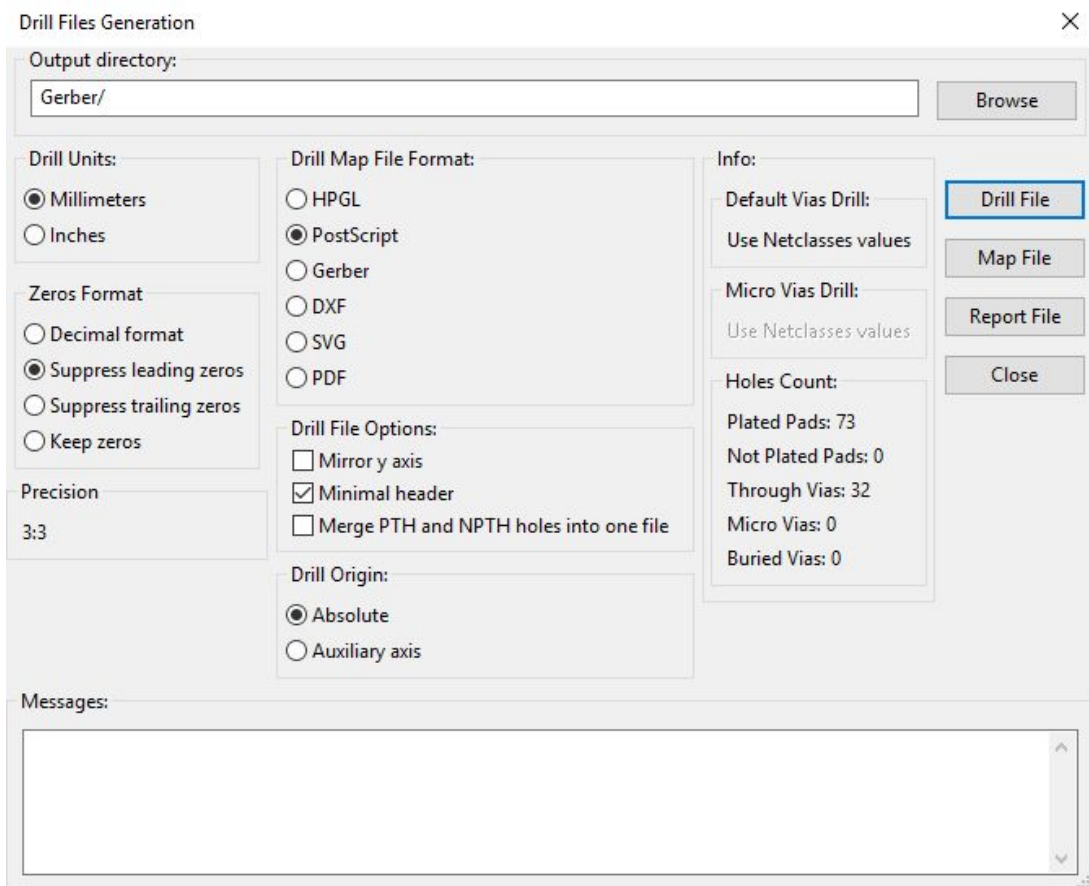
Before anything else, make sure to set the directory to a new folder named Gerber inside of your workspace.

At a minimum, make sure the boxes on the left are selected as following:

You must also click on Generate Drill File.

Click on Drill File to complete this step, then finally click on Plot to finish the Gerber files.

Now you have a complete set of Gerber files.

Conclusion

By now you should a good understanding of KiCad, and should be able to complete the task at hand. Some more tips on getting a successful design: have others review your design. They will be able to spot mistakes that you always skip. Always try Googling an issue if it is not covered in this tutorial or some other video. If there are mistakes in this tutorial, or you would like to contribute, feel free to take this document and add to it.