

**EEC 134AB**

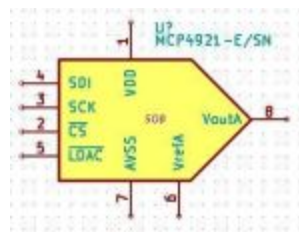
**PCB System Design**

**Falcon 9**

**Marco Venegas**

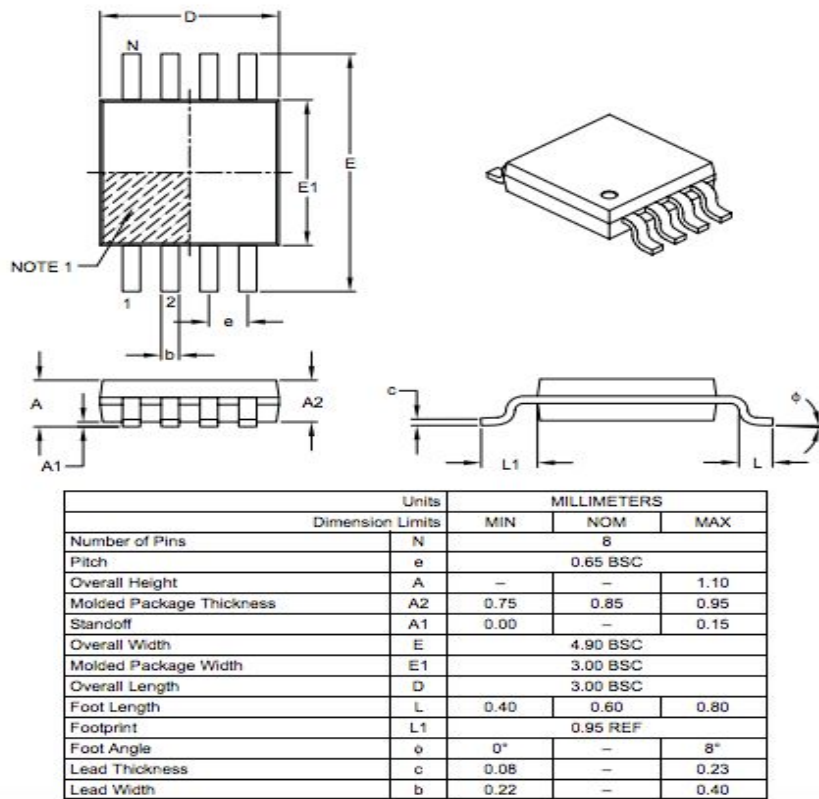
## KiCad: Schematic

For my group, my main function for the group was to be in charge of the PCB fabrication. Making PCBs is a great skill to have, but it takes time to get a feel for the software you are using. Thankfully, KiCad has a more user friendly interface when building a PCB. First thing before you build your schematic in KiCad, check if the software already has a component for all of the material you need to build your circuit. They will already have resistors, capacitors, inductors, et cetera but check for the less common components, such as the MCP4921.



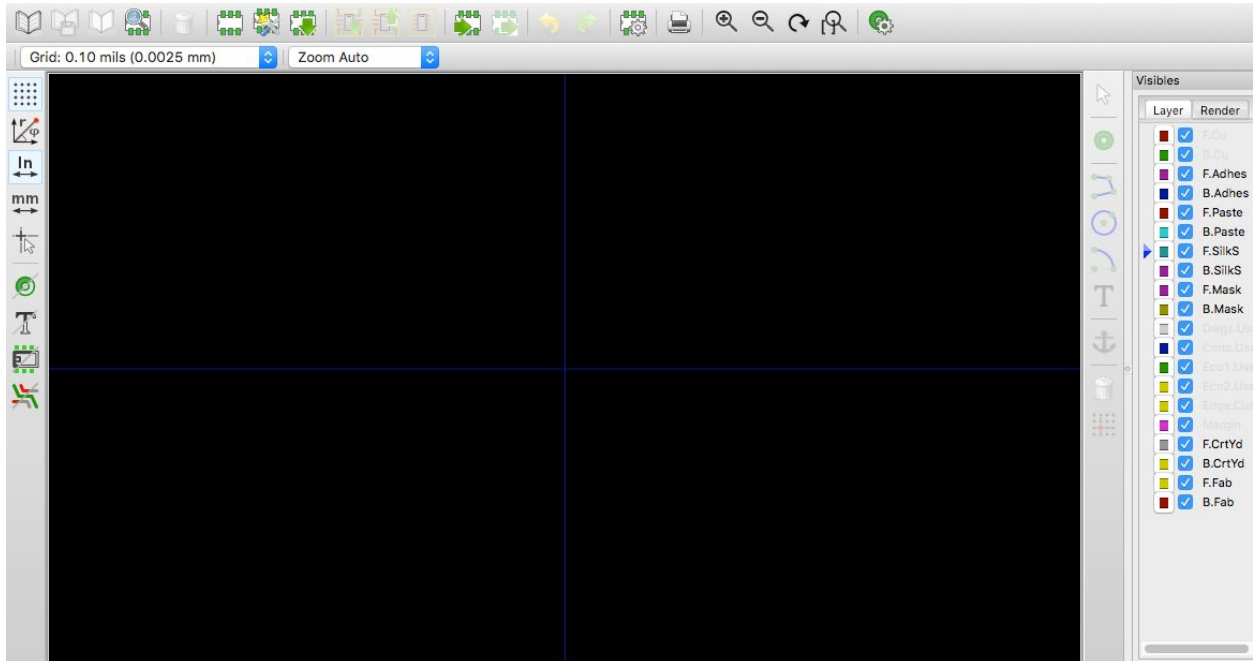
**Figure 1: Example Component**

KiCad tries to help out as much as possible and if they did not, then you will have to build the footprint of the component, first. The footprint is the actual 1 dimension of the physical components. Before starting, looking at the details of the components on its respective data sheet.



**Figure 2: Example Dimensions**

In figure 2, we see the dimensions of the MCP921. This will help you to get the best results of for your PCB. It is important to follow the dimension as much as possible because a little error that can be made so that the components can properly fit on the board. Go to footprint editor mode on KiCad.



**Figure 3: Footprint Editor**

When on this part of the PCB fabrication, it is important to remember your grid size because it will lead to much frustration when trying to connect traces in different grid sizes. 20 mils or 10 mils should be good enough. Make sure that you are on the appropriate layer, F. Silks. Double check in the right side of the interface under visibles. Next, you can change the measurement when making the footprint. Check left hand side to see you are in In or mm. Now, click file and new footprint file, this will allow you to start a new design. Next, make a new active library, its the icon that looks like a book with no lines in it. On the mid right side click on the trace that looks like a 3 sided component. When first , it is best to clear or reset the marker when making the footprint. This can be done by clicking the spacebar. Now this is when you follow the dimensions form the data sheet. It would be wise to put an indicator on the footprint to show how it should be oriented, Again, try to be precise as possible because this save you time and money. When done making the footprint, click save. Next we will append this footprint to the overall software library. This can be done by clicking preferences > Footprint Libraries Mangaer. From there a pop up screen. Click Project Specific Libraries and then click Append Library.

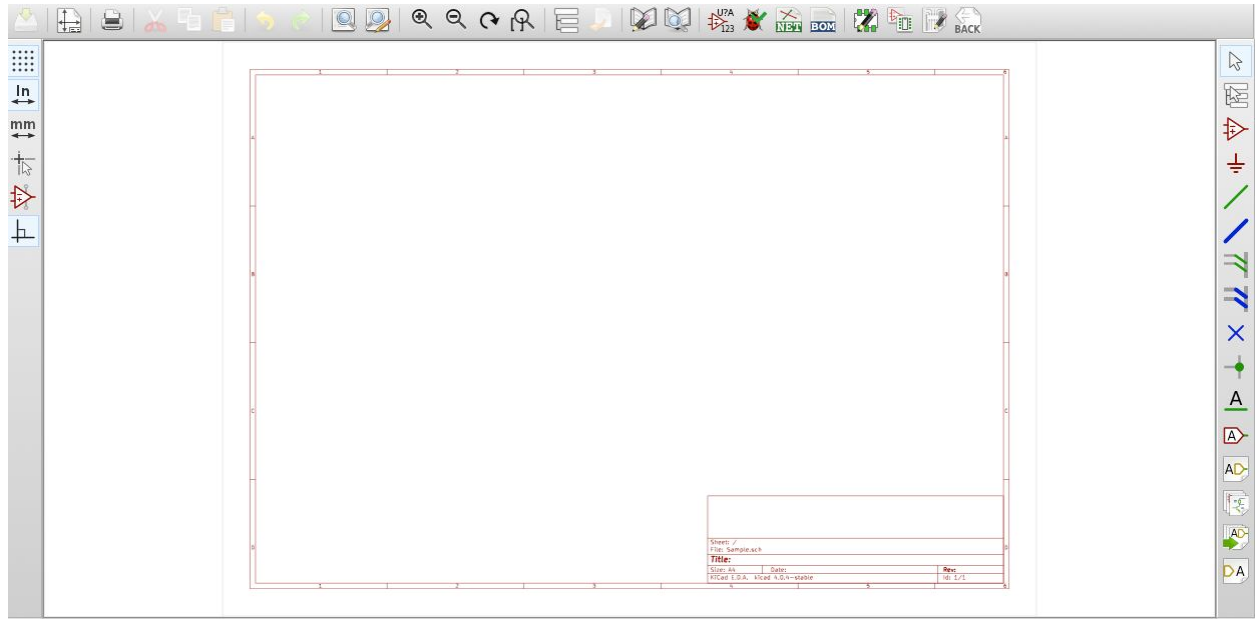


Figure 4: Electronic Schematic Editor

When you are done making all of components schematic, it time to make the actual schematic. Electronic Schematic Editor. It is an icon of a circuit schematic with with a pencil. Here, remember the grid size again, this is not the same as the other grid size. It can be but they do not have to be the same. Before making your actual schematic, you will have to make a physical components for the components your make. This can be made by using the library editor, it is the icon that has a book and a pencil on it. This is a similar process as the footprint editor by the dimension do not have to be the same. It is just a block to be placed in the circuit, however, make sure you follow the pins as seen on the data sheet because this can save time when actually connecting on the components. Save the components and it can be implemented on the schematic. When making the schematic, it best to design the circuit that is similar to one that will be provided. Also, do not make continuous connections because it will get messy. It is best to make a node and name the node. Next, you will have to associate each component to a components.

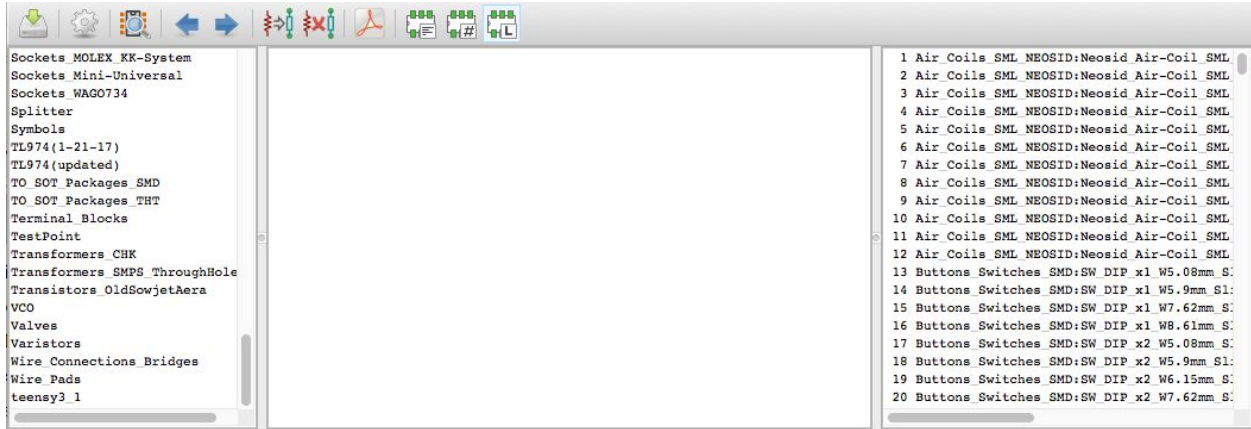


Figure 5: CvPcb

Run the CvPcb, which brings you to another pop-up screen. It will have 3 sections. The left section is the libraries available, the middle is the components that are used in the schematic and the right section is the components to within each library. When done making the whole schematic, it is time to save the schematic.

When you are back on your Electronic Schematic Editor, you will need to drill the fill. This is used to make the actual PCB layout.

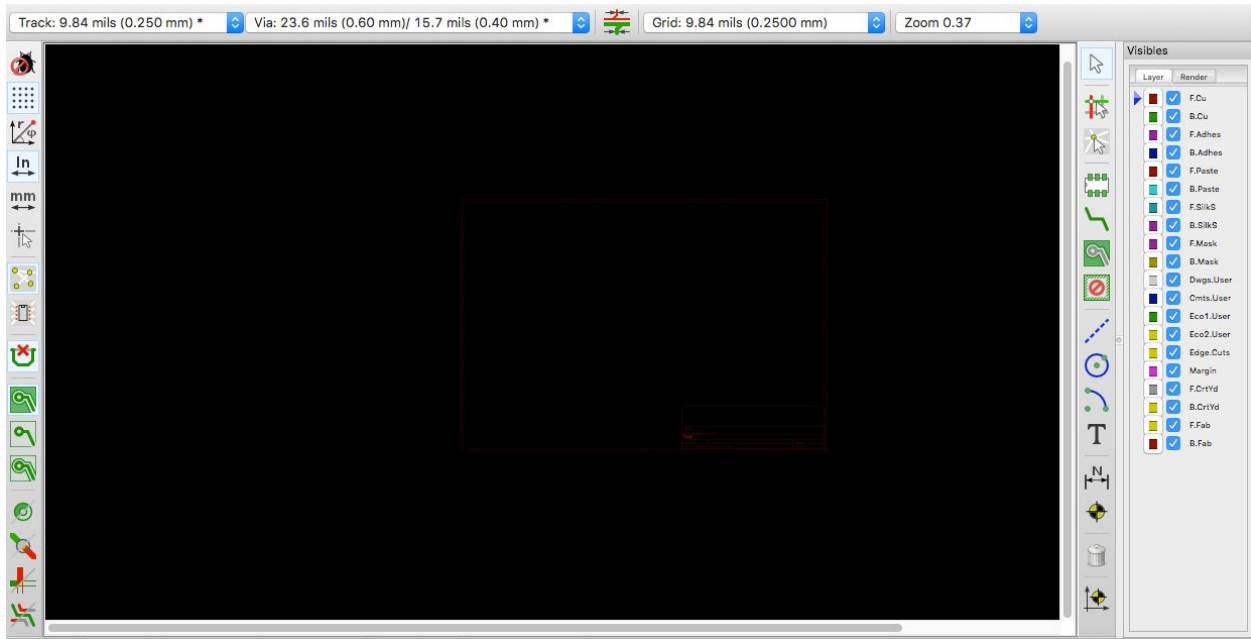
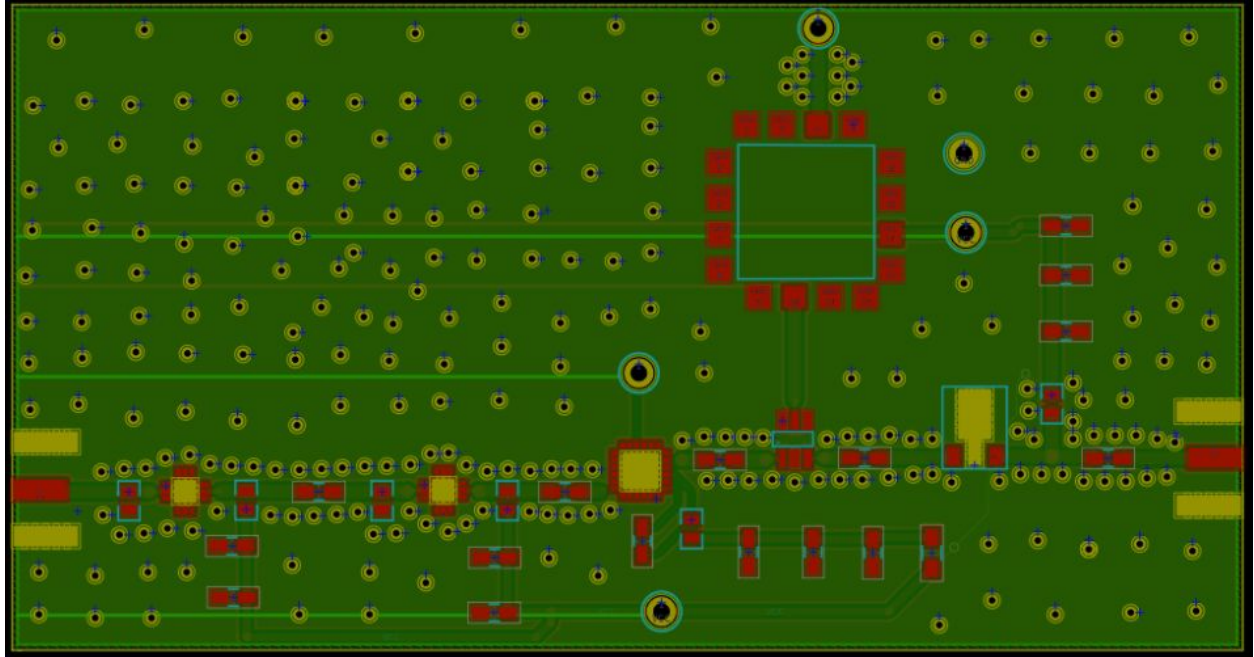


Figure 6: PcbNew

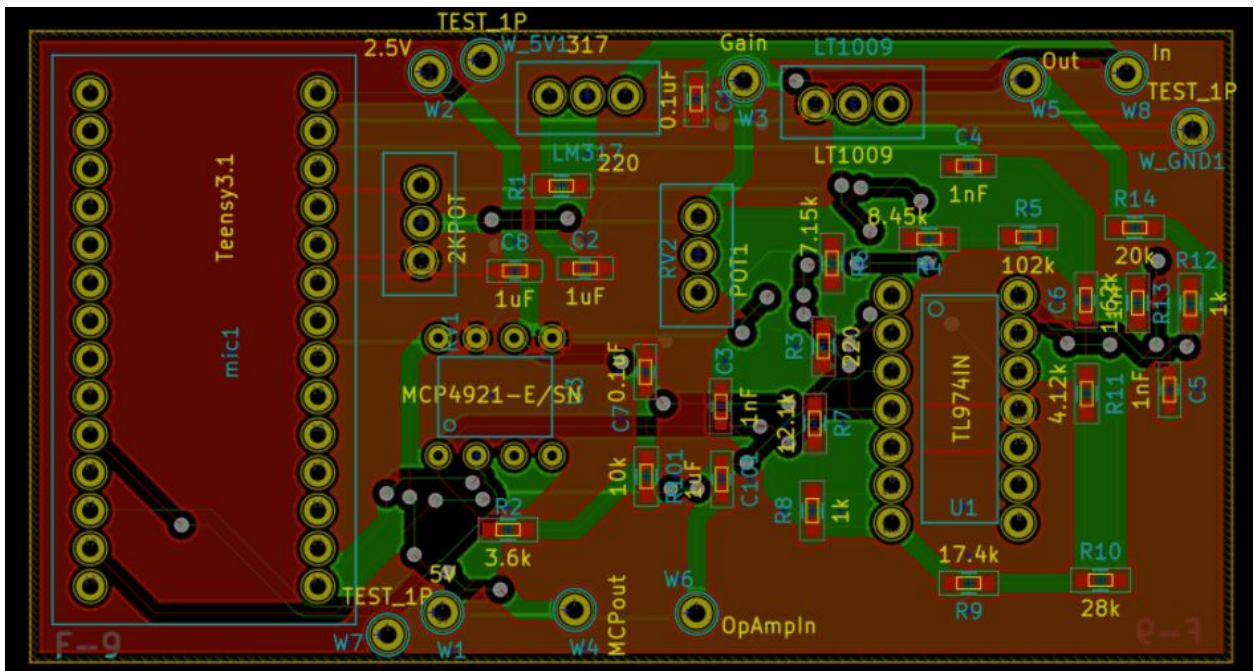
Run PcbNew and this run a new screen. Click “Net List” and it will bring up your ‘rat's nest.’ This is will bring up physical layout that will be printed out. Separate all your figure by their components. It is best to layout your schematic around the major components of the circuit, like the op amps, adc, dac, et cetera. This will make the flow of your circuit much smoother and less messy. Make sure you are on F.Cu, this is under visibles. Now you can trace the out your components. Make sure you do not auto route because it will connect all the components in a horrible fashion. When done making the schematic, make an outer barrier of your design. This will then to make your fields, which should be ground. This should be done for your front and back copper. Save your files and then create your gerber files. To check for errors, zip up all your gerber files and let Bay Area Circuits to analyze them. It will show you all the errors within the PCB.

Final Thoughts: Always triple check your circuit and diagram and have peers looks at the schematic for errors. It takes to fabricate PCBs and an error can greatly put a team in a major set back.

Falcon 9: Final Scheamtics:



RF Board



Baseband Board