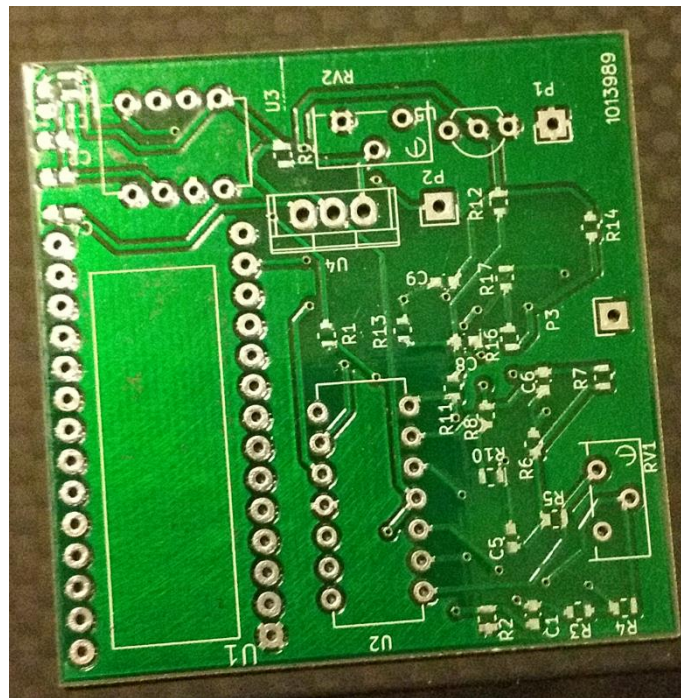


EEC134AB

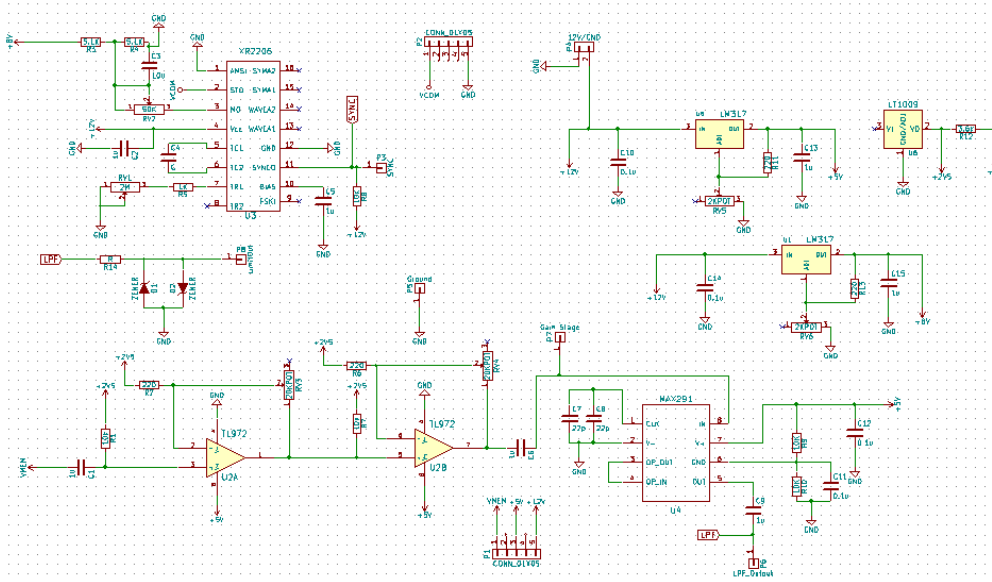
Application Note: PCB Design



Team Grildur Frostcrag  
Charles Liu

A Printed Circuit Board (PCB) is a board made up of a set of pads and copper interconnects that connect at various points to create a proper circuit. PCBs are made up of a substrate (usually fiberglass), copper wirings, soldermask, and silkscreens all stacked up in layers to achieve a compact design. The use of PCB simplifies the wiring process, reduces error caused by wiring, and decreases the overall size and weight of the device. The absence of messy wires gives us cleaner runs and user-placed test pins allow for fast and easy debugging. For our project, we used KiCAD to design the schematics and PCB layouts of our PCBs. KiCAD is not only free, but straightforward in the way that it simplifies the conversion from schematic to PCB layout design. Another option would be to use EAGLE, but the fact that its free version is not for commercial use makes it less appealing to KiCAD.

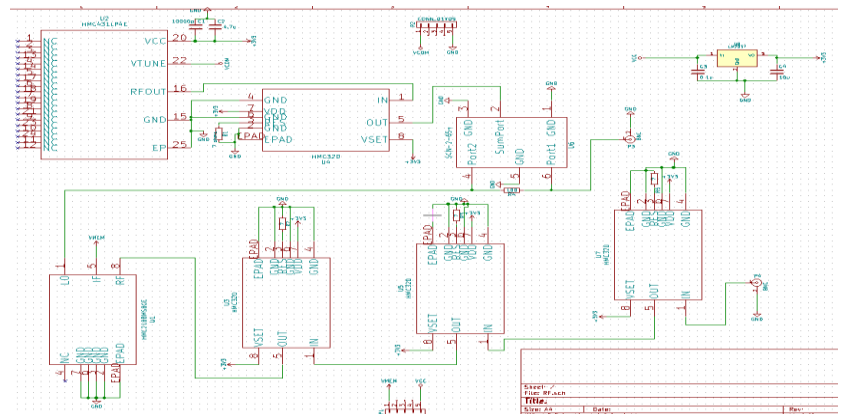
### Schematics



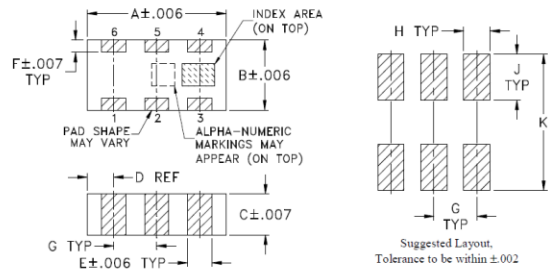
In order to start making your schematic you need to finalize your parts. Once that is done, you can start inputting the parts into the schematic. A schematic is a simplified diagram that is visually easy to comprehend compared to the mess that is the PCB layout. By making connection in the

schematic, we can convert it using KiCAD so that it will automatically make the connections apparent for us when we are at the PCB layout stage. That is why it is crucial not to make a mistake in wiring for schematic design as it will lead to guaranteed failure with the PCB. To make a schematic design, you place your component symbols onto the page and connect the available pins to each other. You have the option to create your own symbols and have various tools such as voltage sources, ground pins, pin connectors, etc. that simplify the process. Above you can see the schematic design of the baseband from our project.

Take note that each component has a name and value. This is to help differentiate the components once they are placed on the board. For our circuit U2A and U2B are the same component, but if they were two different, yet similar components then the part's label would be very important to avoid mismatching. On the right is the RF schematic for our radar.



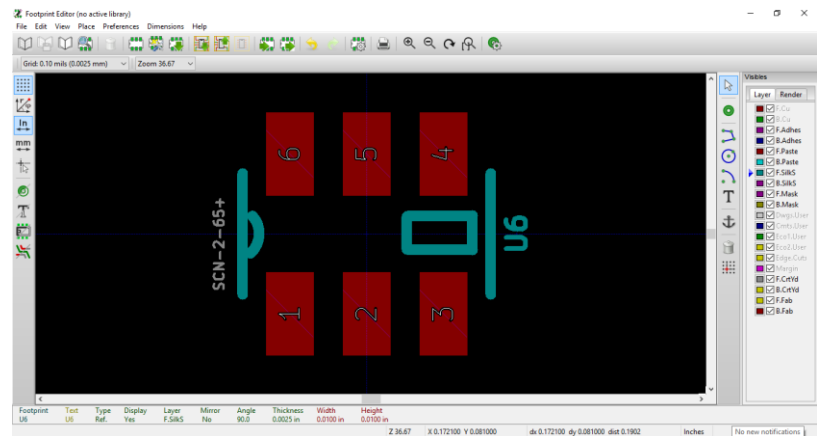
## Footnote Design



CASE#	A	B	C	D	E	F	G	H	J	K	L	M	N	P	WT. GRAM
FV1206-1	.126 (3.20)	.063 (1.60)	.035 (0.89)	.034 (0.61)	.022 (0.56)	.011 (0.28)	.039 (0.99)	.024 (0.61)	.042 (1.07)	.123 (3.12)	--	--	--	--	.020

Footnotes are designs that, when done correctly, allow components to connect with the PCB boards at only the pins specified. It is absolutely crucial that the footnote you have matches the part because a footnote that does not fit the part will make a lack of connection guaranteed. Most components come with premade footnotes or are already stored inside of KiCAD's large database, but sometimes that is not true or the companies don't supply them so we have to make them on our own. To make your own footnote, you use KiCAD and your component's datasheet to look at the dimensions of your part so that you can space out the soldermask layer correctly.

The figure shown above is a company's provided instructions on proper footnotes for the component that we purchased from them. Notice how detailed all the dimensions are and how they have some values that give us more leeway with plus/minus values, making it easier for us to create a proper footnote. On the right is our completed version of the same footnote provided shown in KiCAD's Footprint Editor. The red squares are the parts of the board that will be connected to the copper interconnects.



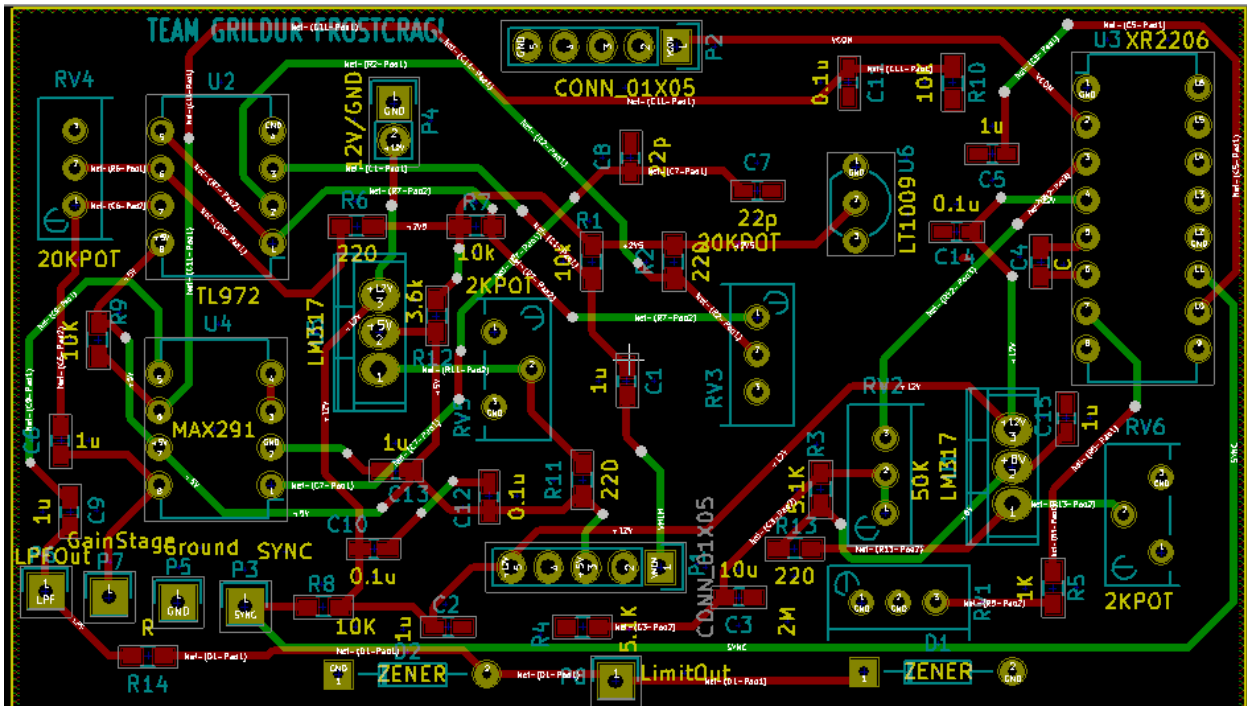
## PCB Layout Design

The next step is where we generate the actual layout of our PCB. Using KiCAD, we can specify how we want the area of our PCB by making an outline of the borders. Generally you want to create one that is rectangular for portability and cleanliness. KiCAD automatically converts your premade schematics and supplies all the components onto your PCB layout for easy designing. The key to PCB layout design is to organize your components in a way that will minimize distance from paths to avoid messy connections and decrease overall size of the board.

Once you have decided on where you want your components you need to connect them by placing traces in between them. Since KiCAD generated all the components from your translated schematic, when you trace the components together there will be lines that show where to connect each part. This is really convenient because without this tool you would need to manually check with the schematic to make sure of where you should be connecting and that would take too much time. You can place a trace on either the front or back copper layer and connect them together by using a via. This simplifies the process when there are too many connections already on one side giving us more space to connect. It is ideal to only use one layer, but it is not realistic for circuits that have many components especially when trying to decrease board size. **Make sure not to place the vias directly on the pins of the components, this will cause improper connections.** Another thing to point out is to be cautious of trace length and width. The longer the length the greater the dissipation of power so we aim for shorter length to give

more power to the circuit. Same goes for width where we want thicker traces, however this is harder to do when trying to minimize board area since larger trace width means less space to connect with. Another important note is that trace connections should avoid making sharp ninety degree turns. These result in issues due to manufacturers not being able to etch it properly.

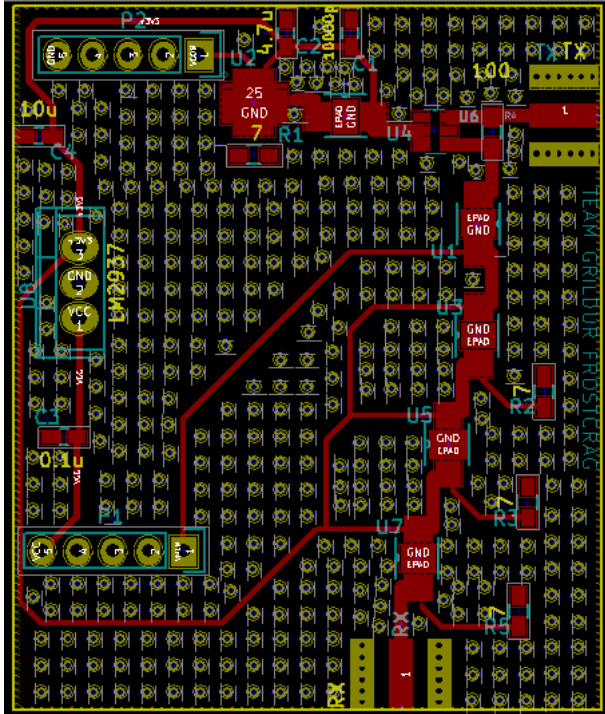
Below you can see one of our finished Baseband PCB layouts. Notice that there are two colors for traces. The red means that it is on the top copper layer and the green means it is on the bottom. We made it so that all of our components will all be on the top layer so in order to connect resistors and capacitors you must place the traces on the top layer. However for parts such as the potentiometers (RV1-6) which have in hole connections, you can use either the top or bottom since they are connected in both layers.



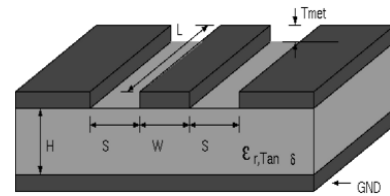
Another thing to mention is to notice the green square shaped components lined up in the left hand corner. Those are test pins that we placed to simplify input and output connections while making debugging less chaotic. We mentioned in an earlier section that PCB design makes debugging easier and it is the capabilities that test pins give us that allow for clean and simple debugging of our circuit.

### RF PCB Layout

Because of the high frequencies used for RF circuits we must take note of the traces. The transmission lines must not be perpendicular to each other and should remain as straight as possible because turns can cause a 180 degree phase dissonance causing the signal to weaken or cancel. Notice that there are two paths so they may look like they are going ninety degrees but they are separate paths.



In the figure on the left is one of our finished RF PCB layouts. As you can see there are numerous amount of vias scattered around the PCB. The vias are necessary around the transmission line in order to prevent RF signals from resonating onto the ground plane and stops the generation of noise. Using a web app called Wcalc we were able to find the parameters needed to make accurate transmission line width and spacing. As shown in the picture below, by inputting our fixed 6.15 mil spacing parameter we were able to get our transmission line width which is 37.5 mil.



Metal width (W)	<input type="text" value="37.6913"/>	<input type="text" value="mil"/>	<input type="button" value="calc"/>
Metal spacing (S)	<input type="text" value="6.15"/>	<input type="text" value="mil"/>	<input type="button" value="calc"/>
Trace length (L)	<input type="text" value="142.802"/>	<input type="text" value="mil"/>	
Metal thickness (Tmet)	<input type="text" value="0.8"/>	<input type="text" value="mil"/>	
<a href="#">Metal resistivity (RHO)</a>	<input type="text" value="5e-08"/>	<input type="text" value="Ohm"/>	<input type="text" value="- m"/>
Metal surface roughness (RGH)	<input type="text" value="0.001"/>	<input type="text" value="mil"/>	<input type="text" value="-rms"/>
Substrate thickness (H)	<input type="text" value="62"/>	<input type="text" value="mil"/>	<input type="button" value="calc"/>
<a href="#">Substrate relative dielectric constant (Er)</a>	<input type="text" value="4.8"/>		<input type="button" value="calc"/>
<a href="#">Substrate loss tangent (tand)</a>	<input type="text" value="0.01"/>		
Frequency	<input type="text" value="5.8"/>	<input type="text" value="GHz"/>	
<input type="button" value="Analyze"/> <input type="button" value="Reset"/>	<input checked="" type="checkbox"/>	Include bottom side ground	
Characteristic Impedance	<input type="text" value="50"/>	<input type="text" value="[ohms]"/>	
Electrical Length	<input type="text" value="43.4184"/>	<input type="text" value="[degrees]"/>	