Will Yang 912121530

Professor Liu

EEC 134

23 March 2018

Application Note: RF PCB Design

Opening remarks:

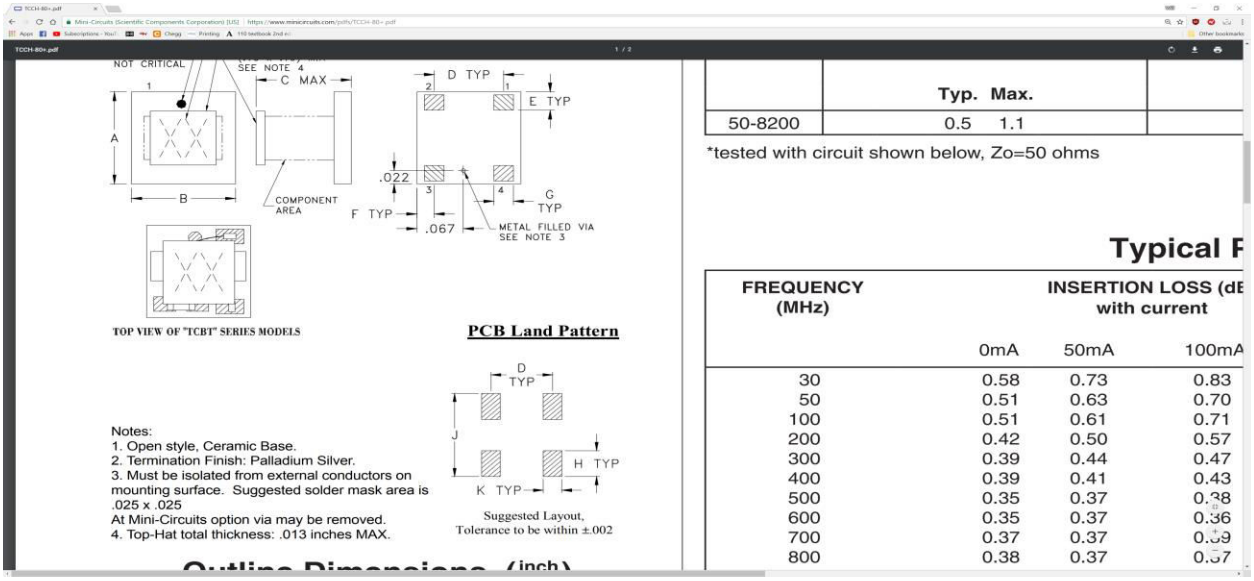
PCB design is one of the key concepts in the RF systems senior design and almost the whole second quarter is spent on making the RF and baseband PCBs since they are the most efficient and effective ways to create the radar system. This application note will focus on the details in each step of the design that the designer should pay extra attention to before sending it in for production.

Preliminary:

With the RF board, the first step you should take is to make sure that all the components cascade together in a desired fashion based on your full system design, there is no going back once you want to do the layout over and over again. When connecting all of the amplifiers and other blocks in the schematic, make sure you also go to the datasheet and check if they have peripheral circuits that are needed for each component to work. An even better idea would be to find parts that don't need a lot of peripheral circuitry to make life easier when soldering and laying out the whole PCB. When doing the peripherals, also make sure the sizes of the capacitors and inductors match what is on the datasheet or else it could cause confusion and also possibly mess with the final signal. Have your team double and triple check your work on each step mentioned here since just one little error could cause the entire PCB to fail and it is really hard to find the mistake after the fact.

Footprints:

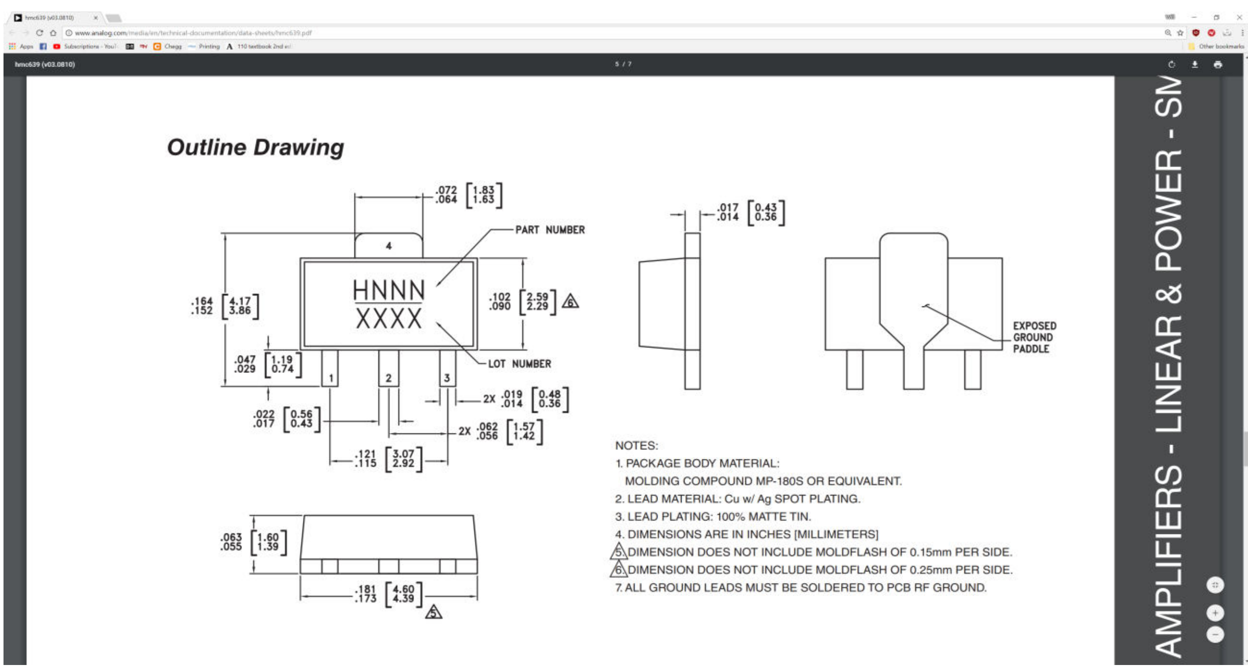
This step is interchangeable with the schematic layout and most of the time you are probably switching between the two depending on the circuit. There are a few things to keep in mind while making the footprint. The first is that all the sizes are provided on the datasheet but the axis and drawing on the software is most likely on a coordinate system and you will need to do some arithmetic to figure out where everything is. This is a step that needs to be double checked since we had some groups this quarter that simply had the wrong dimensions on their surface mounts as the cause for the PCB to not work. Also pay attention to the parts that are not regular inductors and capacitors that might be advertised as such like a RF choke that is actually written in as an inductor.



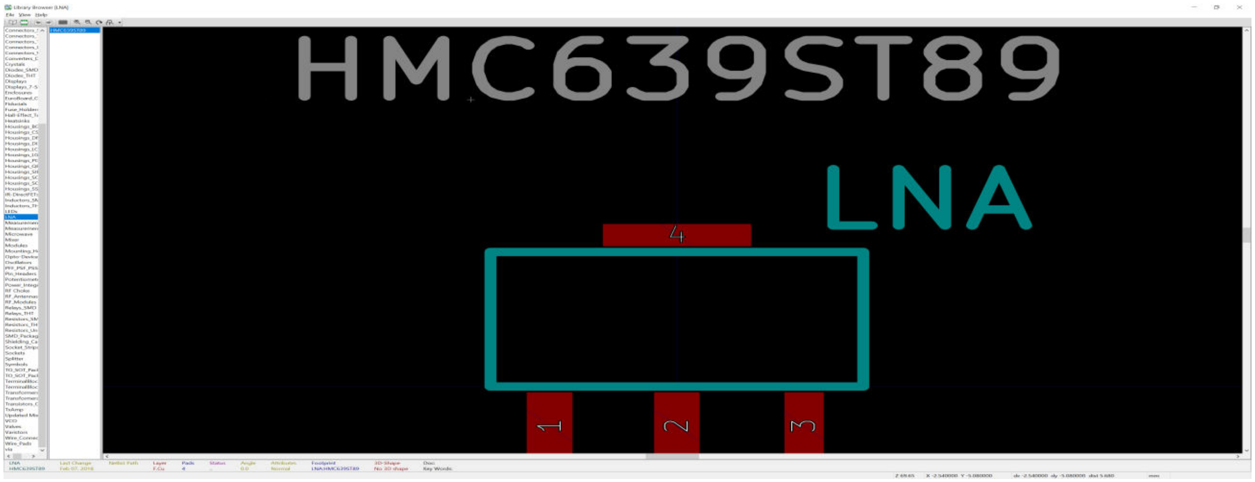
FREQUENCY (MHz)	INSERTION LOSS (dB) with current 0mA	50mA	100mA
30	0.58	0.73	0.83
50	0.51	0.63	0.70
100	0.51	0.61	0.71
200	0.42	0.50	0.57
300	0.39	0.44	0.47
400	0.39	0.41	0.43
500	0.35	0.37	0.38
600	0.35	0.37	0.36
700	0.37	0.37	0.39
800	0.38	0.37	0.37

Part of a peripheral circuit we had on our RF board that is listed as an inductor but is not as simple as that.

Next up is the settings of the pads and through holes that are being drawn. The sizes of the through holes are especially important since it is really hard to make the hole bigger once the PCB is produced. There are certain amplifiers that will have a big ground pad that extends all the way across the body of the surface mount and the problem with this is that irregular pad shapes are a hassle to make. For something like this, you could just divide the pad into two sections for the top ground pin and the bottom ground pin.



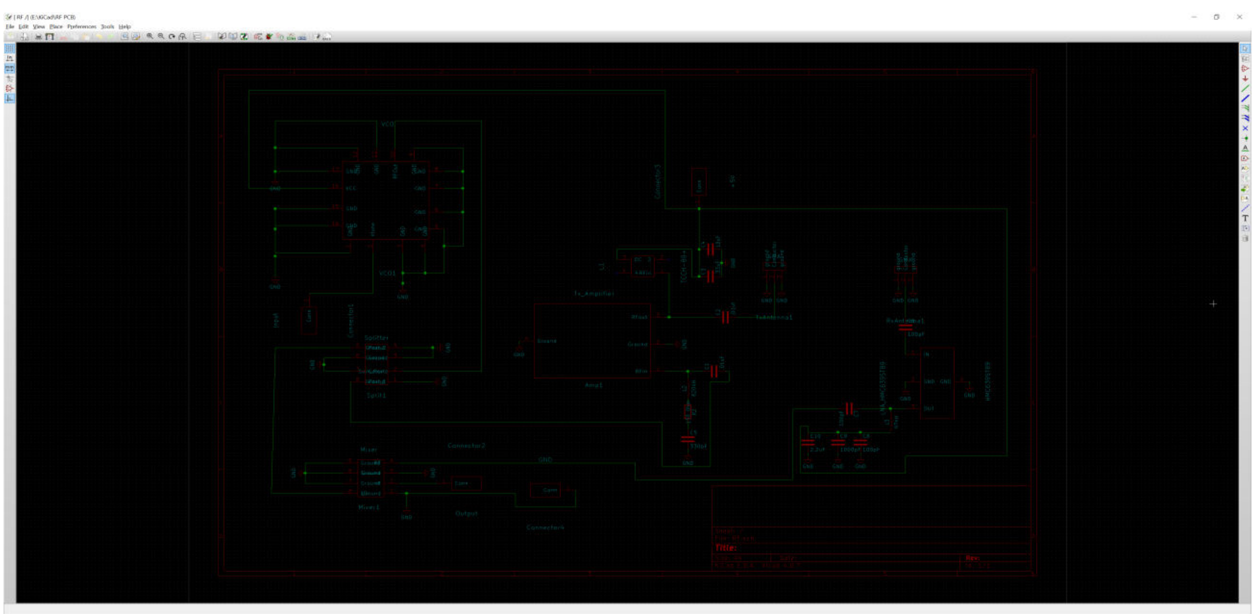
The datasheet has the full ground paddle in the back but that is not necessary unless you really want it.



Pins 2 and 4 are both ground and making them separate would save a lot of time (unless there is some really simple way I'm missing)

Schematic:

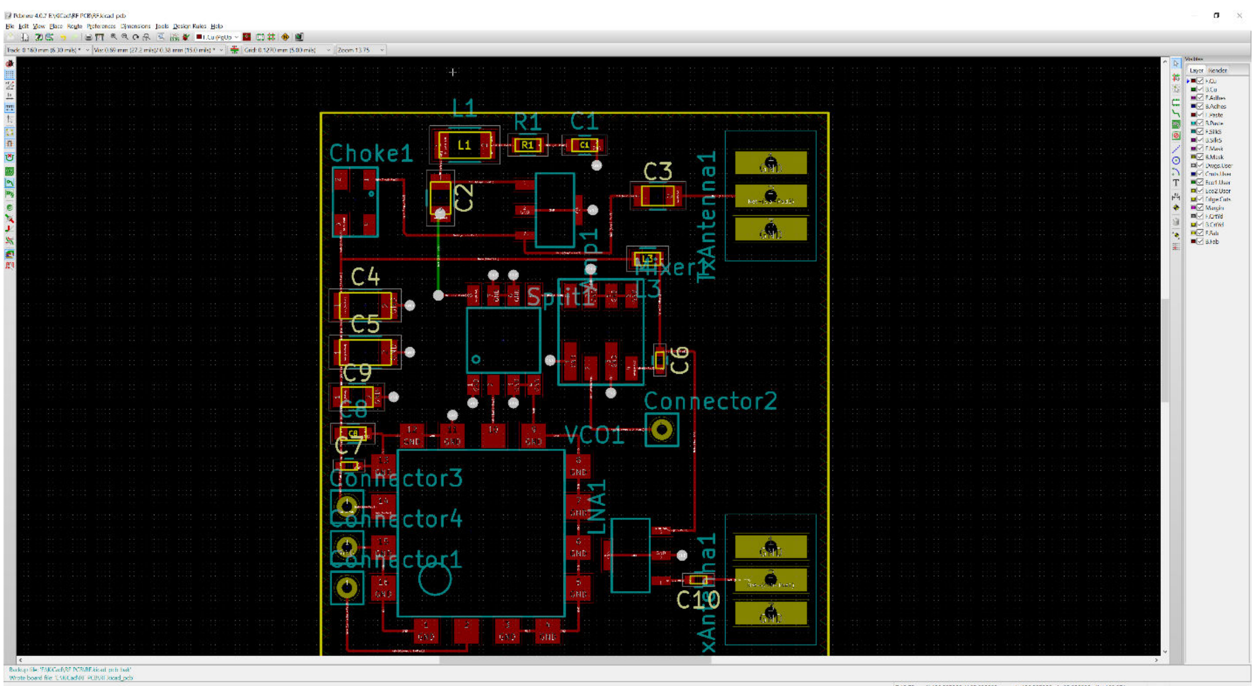
This is perhaps the easiest part of the PCB design process since there isn't much to pay attention to. Just make sure the connections are correct so the PCB layout will have the correct nets. For this specific PCB, you can make each block in the RF circuit their own little section to keep it tidy although that isn't absolutely necessary. Make sure the components that need a footprint is created as a part first and then be sure the footprints you created for them are assigned to them or else they won't show up and there won't be any error messages! Also don't forget to add in external connectors (whether that is a footprint or available in the library) like the SMA connector and your voltage input/ground.



Our RF PCB schematic, we didn't have it as tidy as it should have been but since it is small it's not a huge deal.

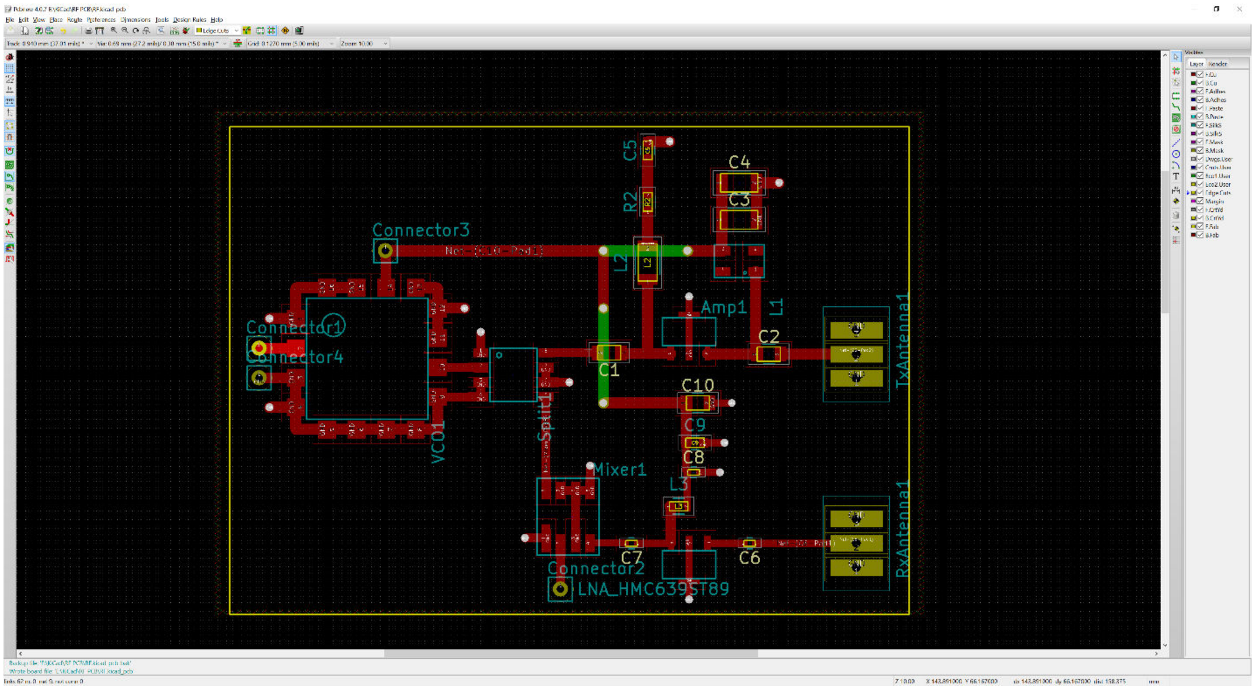
Layout:

This is the part that will take up the majority of the time on the PCB design. The amount of nets you will have to manually connect is very time consuming and there are a couple of special precautions you have to take to make sure the RF signals are passed correctly and clearly as possible. We had to lay out our board three times because we didn't know there are certain rules that needed to be applied for this specific PCB since it is transmitting high frequency signals that are more demanding than normal signals. On the first run, we were trying to make the PCB as compact as possible so there were a lot of bent/small traces along with a via that actually carries the RF signal. The one thing we were able to do well on this is the ground pins. Simply drag a small trace out from the ground pin and via down to the ground trace to save some connections and space on the board, there is no need to connect them all together.

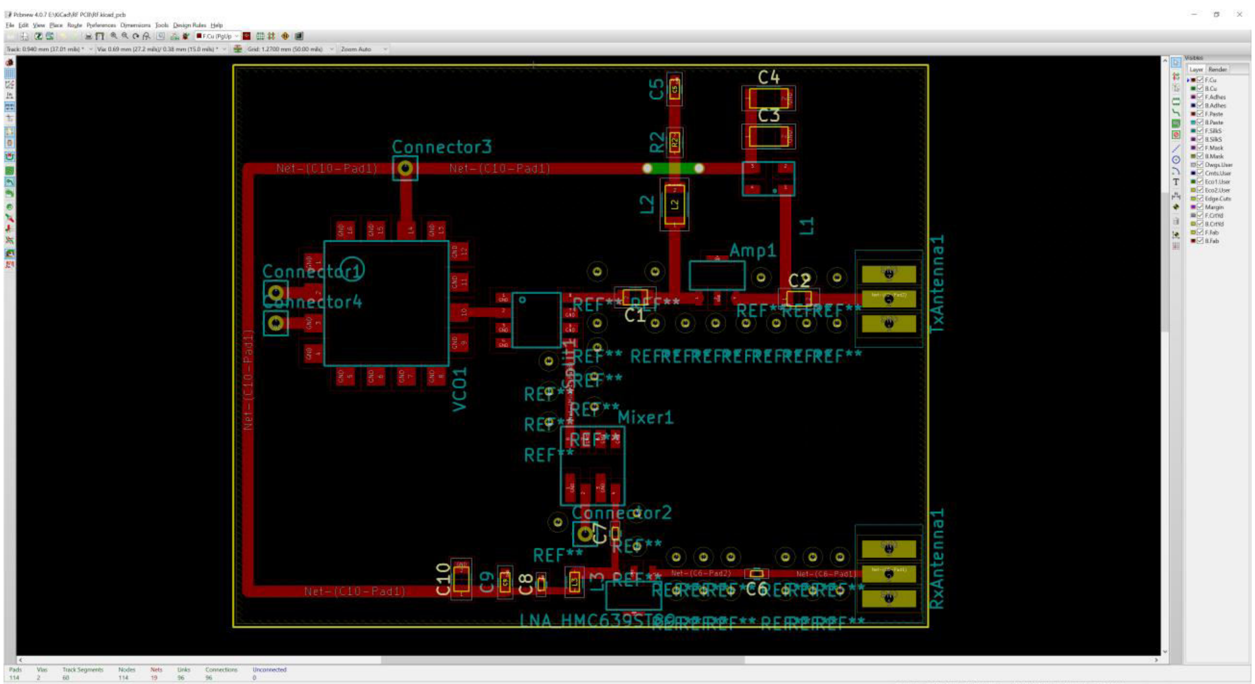


First iteration board, RF traces are bent/too small and via carries signal which is not good.

On the second iteration, we fixed many of the problems described above but there were still some small details that needed to be fixed. We changed the trace width on the RF carrying traces so they are large enough to create a clear signal. The traces were also straightened out on the signal path (the peripherals and voltage carrying traces don't matter). After going to the professor we found out the problem was that the middle via went through a component, which should never happen, and that the peripheral circuits were not next to their correct blocks to keep the board tidy. Just for good measure, there should also be via stiches around the RF carrying traces to minimize noise (they are just little through holes around the traces).



Fixed some problems but still needed to fix the via and peripheral placement along with via stitching.



Final iteration keeping in mind that you need via stitching, straight RF traces, large RF traces, no vias on RF traces, and keep the peripherals around their main component.

After the placement, all that's left is to fill the layers with the right material and to generate the files for production which shouldn't be too much trouble.

Concluding remarks:

This document serves as a guide to common mistakes in the RF PCB process but not a full guide on creating the RF PCB. Make sure you check on YouTube and previous application notes for the full tutorial on KiCad PCB design. Get familiar with the advanced soldering machine from quarter 1 by practicing a bit more even if the quarter 1 designs don't work, you will save time the second quarter, which is much more rushed in terms of pacing. Don't worry about making the PCB pretty and making everything compact since that is not the goal this quarter anyway (there is also no size limit on production for your own design). Having a functional PCB is much more important than having a pretty one that doesn't work!