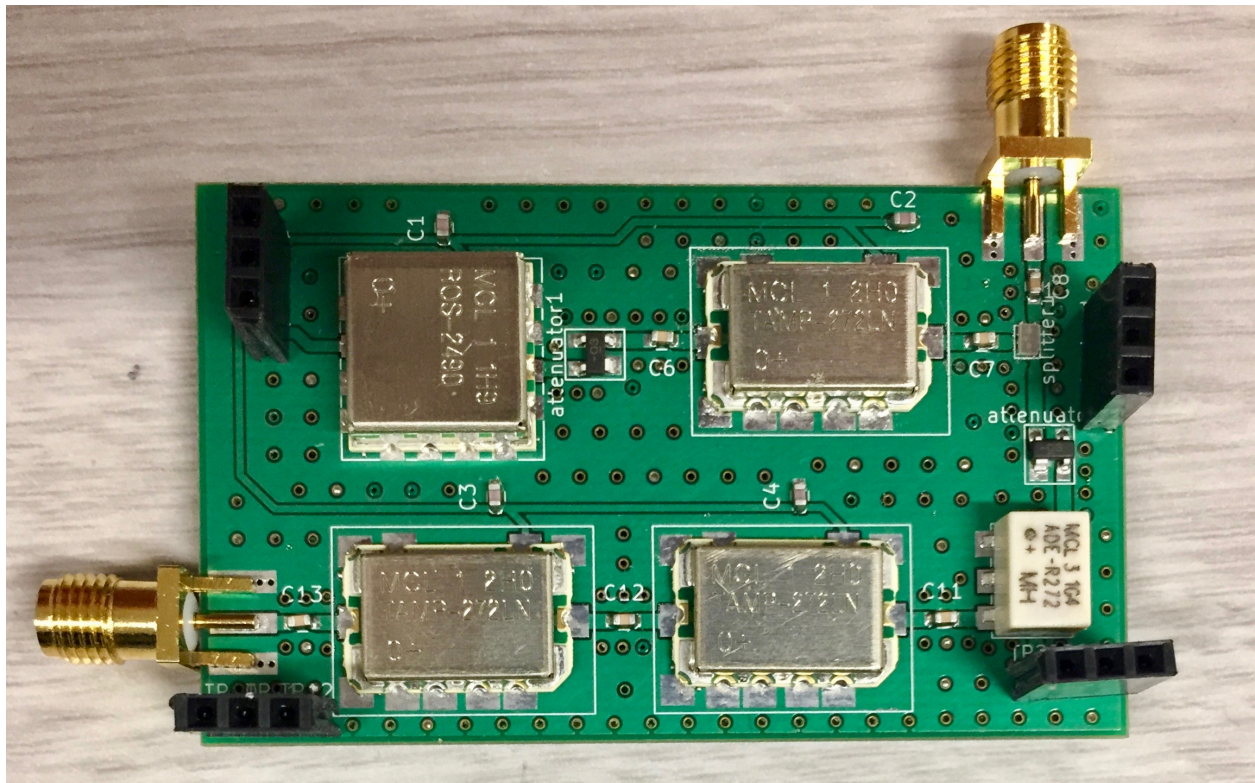


EEC 134AB

Application Note

RF PCB Design



By: Yizhi Tao



Introduction:

In quarter one, you have probably already put together an RF system using coaxial SMA components. Now you want to put the circuit on a PCB to save weight and better electrical connections. In our radar system, we opted for a stack baseband and RF PCB design because two PCB boards provide better isolation between baseband and RF signals. In addition, having separate PCBs reduces the overall footprint and helps testing and debugging. Compared to the baseband PCB, RF PCB likely has fewer components and traces to lay out, but the signal path requires extra attention to make sure little to no loss.

Schematic:

Just like any PCB design, we start by drawing out the schematic. Because most components do not have pin connections in the KiCAD library, we need to make them. We can find out the pin connections for a component on its datasheet. Take an example of the VCO, the pin connection is shown in Fig 1. Then open up Part Library Editor in KiCAD and choose “Create a new component”. Use  tool to draw the outline of this component and use  to define each pin, Fig 2. Notice that in library editor, the things that matter the most are the pins because they will directly affect how you lay out the PCB in the end, so be extra careful when defining Pin name, Pin number and Electrical type, Fig 3.

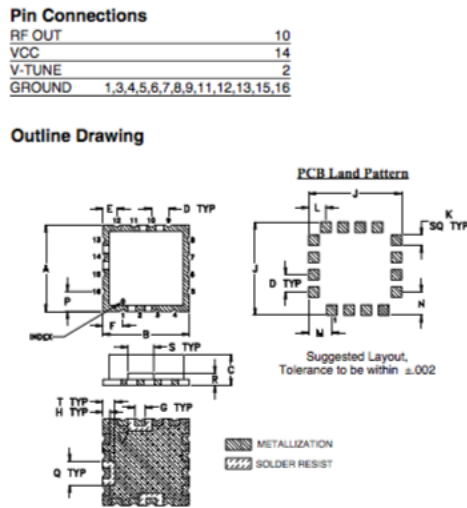


Fig 1: Pin connections of VCO on datasheet

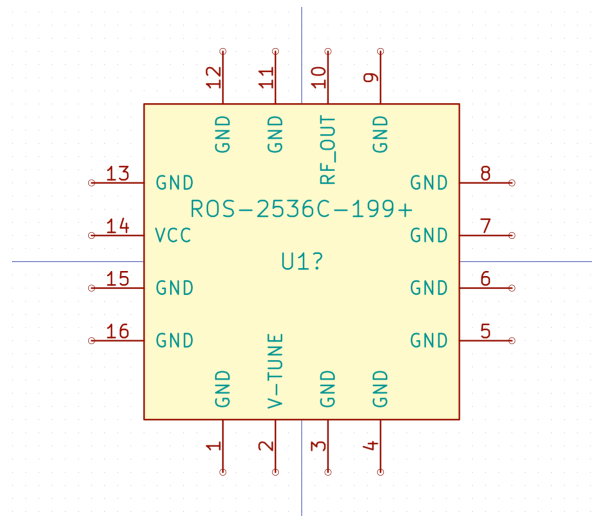


Fig 2: pin connection of VCO

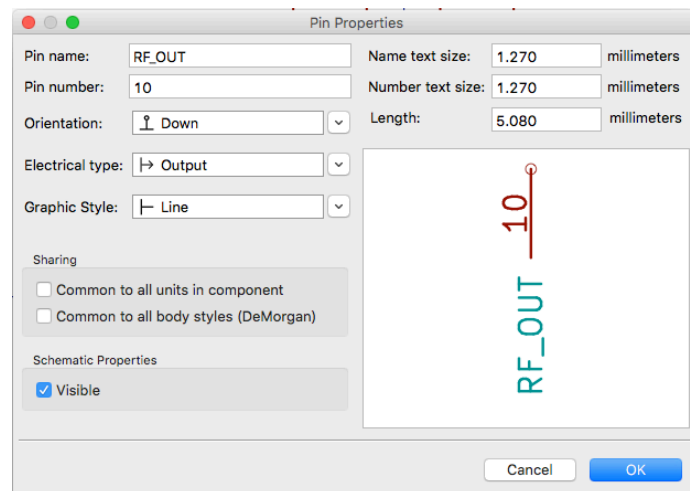


Fig 3: Pin properties

Once you have finished the pin connections, you can start drawing the schematic. Try to place the components in a fashion that makes the most sense to you. In my case, I placed transmitter and receiver in parallel, so they are easy to distinguish from each other. Also, make sure to use Label tools to reduce wire connections to gnd and vcc for example. Fig 4 shows the whole schematic for the RF PCB.

You may notice that in addition to the RF components I also placed capacitors between vcc and gnd and between some components. Capacitors between vcc and gnd are called decoupling or bypass capacitors and they are used to decouple the AC signal from power supply and protect the circuit. On the other hand, capacitors in between RF components can block the DC offset.

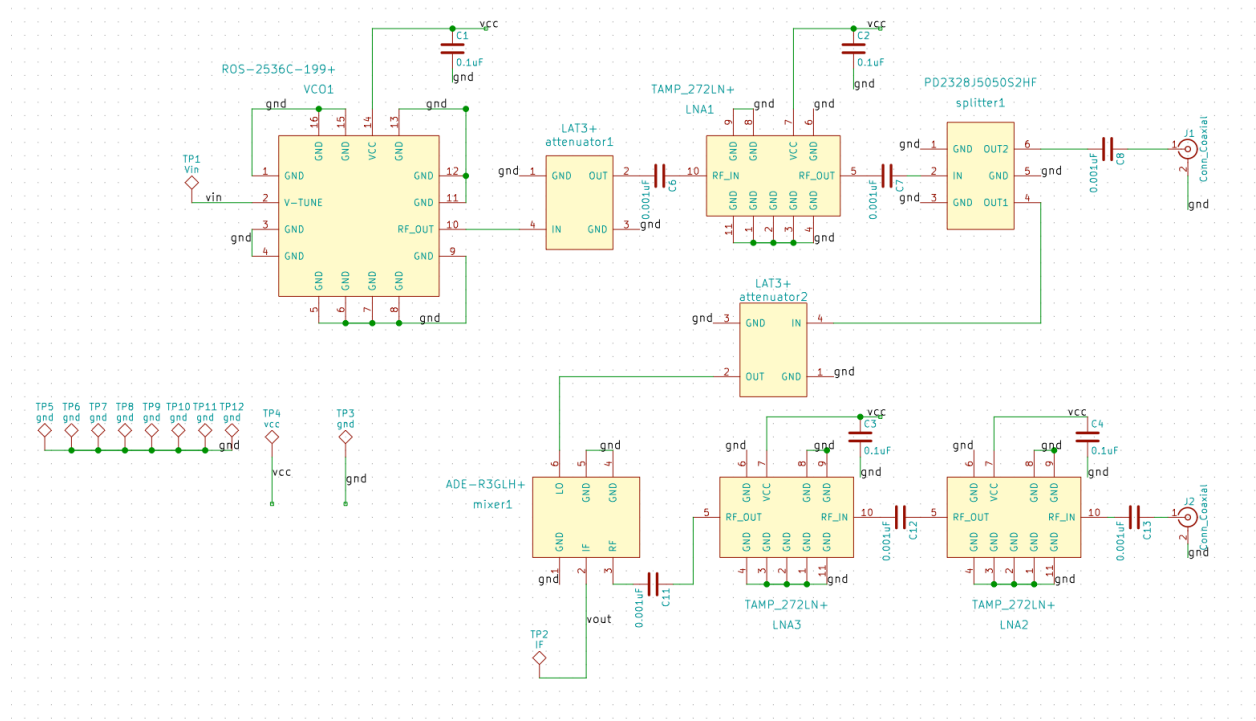


Fig 4: RF PCB schematic


After you are satisfied with the schematic, make sure to run a design rule check by clicking the ladybug  and it will tell you if you have missing connections. Notice that in Fig all pins are connected to gnd while in Fig only pin 5 is connected.



Fig 5: gnd connection

PCB Footprints:

The next task is creating custom-designed footprints. Again, because KiCAD likely does not have the correct footprints for the component and you should not trust footprints on the internet, so it is safest to create your own footprints. Using the VCO as an example, study the outline drawing and outline dimensions on the datasheet, Fig 6. Use Pad tool to create SMD or through-hole pads according to the datasheet, Fig 8. Use “create pad array” function to create equally spaced pad connections. Take advantage of different grid sizes to help you pinpoint a part’s location. For example, with a grid size of 10 mils, the component can only move at a step of 10 mil, Fig 7.

Creating the correct footprint may feel tedious but it is probably one of the most important parts of PCB design. So, take your time to make sure that your footprint matches as closely as possible to the datasheet. Double check and triple check if necessary.

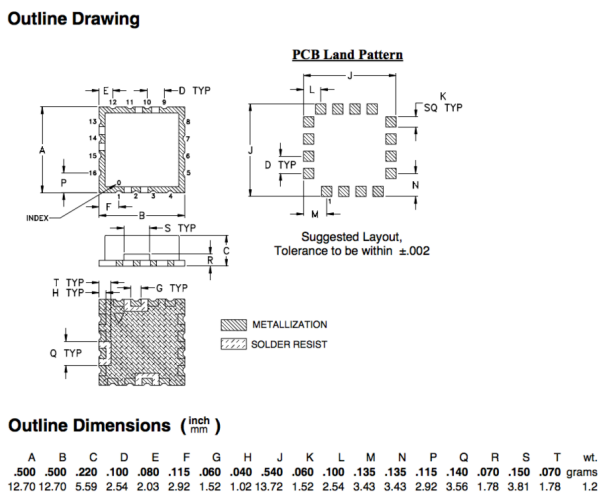


Fig 6: Outline drawing and dimension of VCO

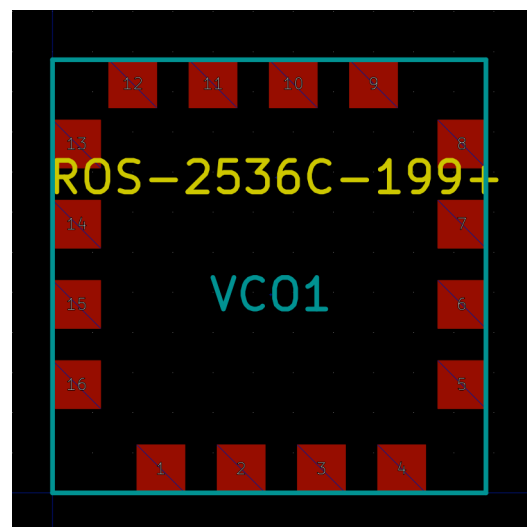


Fig 7: footprint of VCO

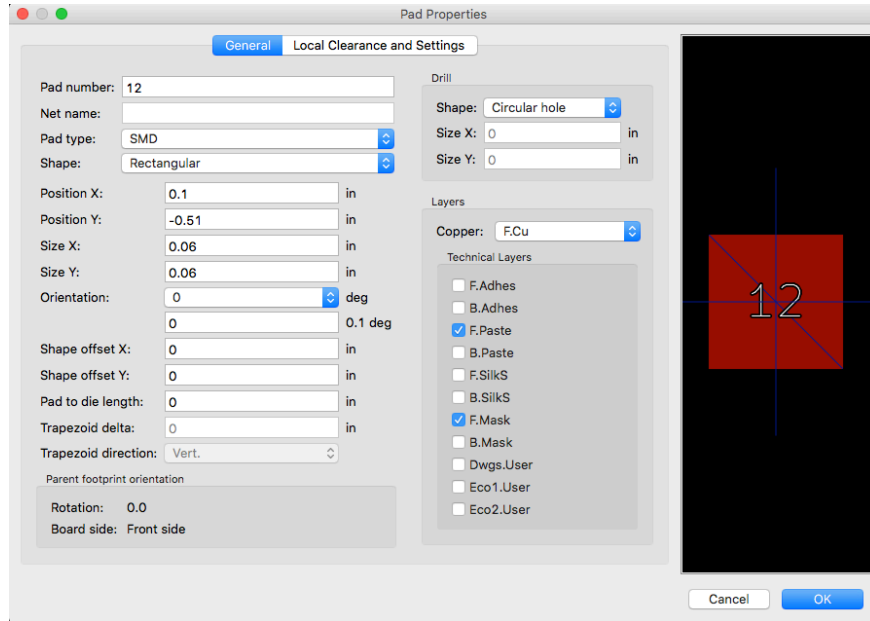



Fig 8: Pad properties

Once you have finished all the footprints, it is time to associate components with footprints. Go back to the Schematic editor and run Cvpcb . First select a component in the middle window, then go through the right window to find the associated footprint, Fig 9. After all components have been assigned, click Save footprint association.




```

1 attenuator1 -          LAT3+ : RF_PCB:LAT-3+
2 attenuator2 -          LAT3+ : RF_PCB:LAT-3+
3 C1 -                   0.1uF : Capacitors_SMD:C_0603
4 C2 -                   0.1uF : Capacitors_SMD:C_0603
5 C3 -                   0.1uF : Capacitors_SMD:C_0603
6 C4 -                   0.1uF : Capacitors_SMD:C_0603
7 C6 -                   0.001uF : Capacitors_SMD:C_0603
8 C7 -                   0.001uF : Capacitors_SMD:C_0603
9 C8 -                   0.001uF : Capacitors_SMD:C_0603
10 C11 -                  0.001uF : Capacitors_SMD:C_0603
11 C12 -                  0.001uF : Capacitors_SMD:C_0603
12 C13 -                  0.001uF : Capacitors_SMD:C_0603
13 J1 -                   Conn_Coaxial : RF_PCB:SMA
14 J2 -                   Conn_Coaxial : RF_PCB:SMA
15 LNA1 -                 TAMP_272LN+ : RF_PCB:TAMP-272LN+
16 LNA2 -                 TAMP_272LN+ : RF_PCB:TAMP-272LN+
17 LNA3 -                 TAMP_272LN+ : RF_PCB:TAMP-272LN+
18 mixer1 -              ADE-R3GLH+ : RF_PCB:ADE-R3GLH+
19 splitter1 -           PD2328J5050S2HF : RF_PCB:Anaren-splitter

```

Fig 9: footprint association

PCB Layout:

In Schematic editor, click  to generate netlist. Then open Pcbnew  and click  to read netlist, Fig 10.

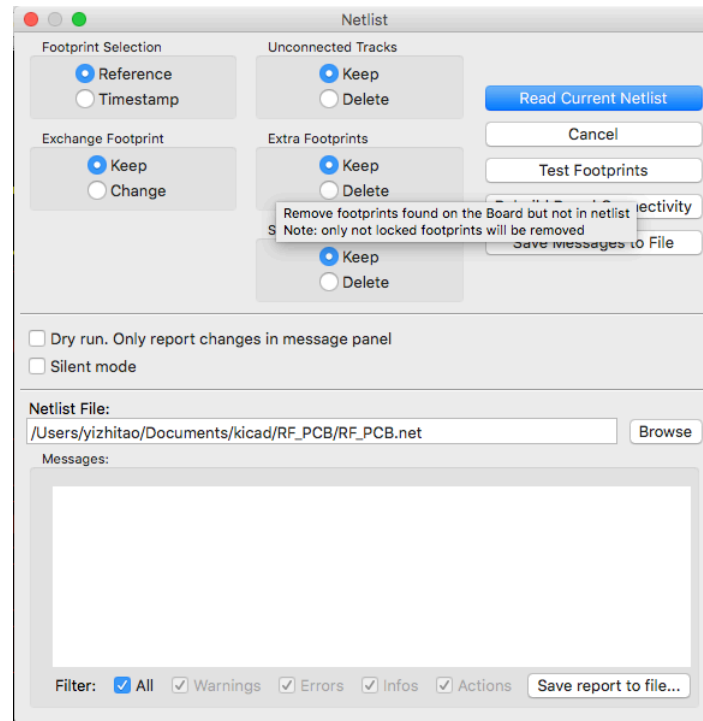


Fig 10: generate netlist

After that, you should see all the components stacked together in one place. Use right click -> Move or key board shortcut M to separate them and place them in the desired location. Note that you can change the layout any time in the future, so it is not necessary to decide the location of each component in the beginning.

On RF PCB, there are two types of traces you should distinguish between, low frequency (or DC) signal paths and high frequency signal paths. In my design, the only low frequency and DC signals are the V-tune signal from baseband PCB and VCC power lines. Other traces are all high frequency RF (2.4 GHz) signals in transmitter and receiver. For low frequency signals, you do not need to worry too much about the track width as long as they are not too thin. However, for high frequency RF signal traces, you do have to pay attention to the track width and length.

Track width for RF signal is important because we need to have a $50\ \Omega$ transmission line to minimize signal reflection. To determine the RF track width, use the built-in PCB calculator in KiCAD, Fig 11. I chose to use Coplanar wave guide with ground plane because it gives relatively thin traces compared to having Microstrip lines. Note that you need to fill the top plate with ground for this method to work. First enter the substrate parameters given by the TA. On the right-hand side in Physical parameter, you can select W and S values and calculate Z0. Conversely, you can specify Z0 and one of W and S parameters to calculate the other.

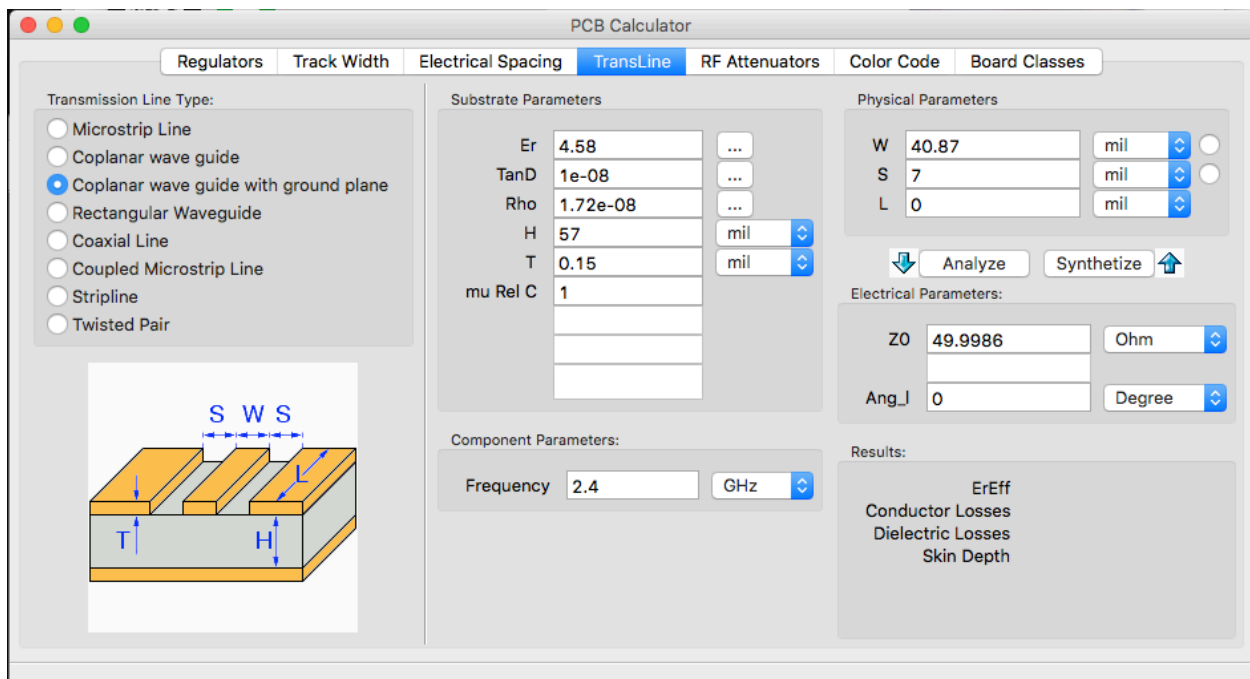


Fig 11: PCB calculator

At first, I tried to use 6 mil (minimum copper to copper width of Bay Area Circuit) as the separation. However, the DFM reported less than 6 mil (5.45mil) copper to copper width and never passed the DFM check, Fig 12. This is due to the fact that some copper fills do not have smooth round edges and the actually clearance is less than 6 mil. In the end, I switched to 7 mil clearances and passed the DFM check.

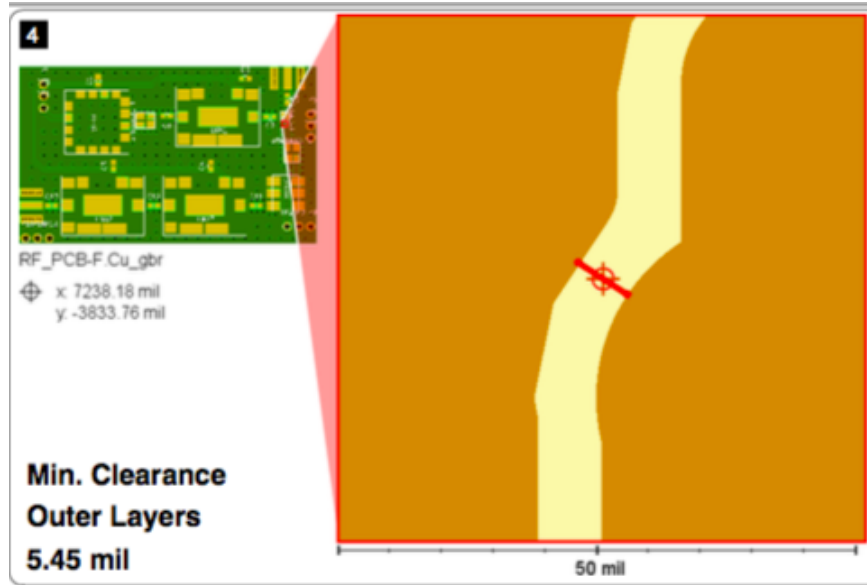


Fig 12: DFM report min. clearance

In addition to selecting the correct track width, designing short and straight RF traces is another important thing to consider. Long and bending RF traces can create loss in the signal. Therefore, I centered the design of my RF PCB and placement of components around achieving the shortest and never bending RF traces. Below is my first attempt at the RF PCB, Fig 13. Although I kept all traces straight, there was a lot of wasted space on the top and in the middle. Components could also be placed closer together.

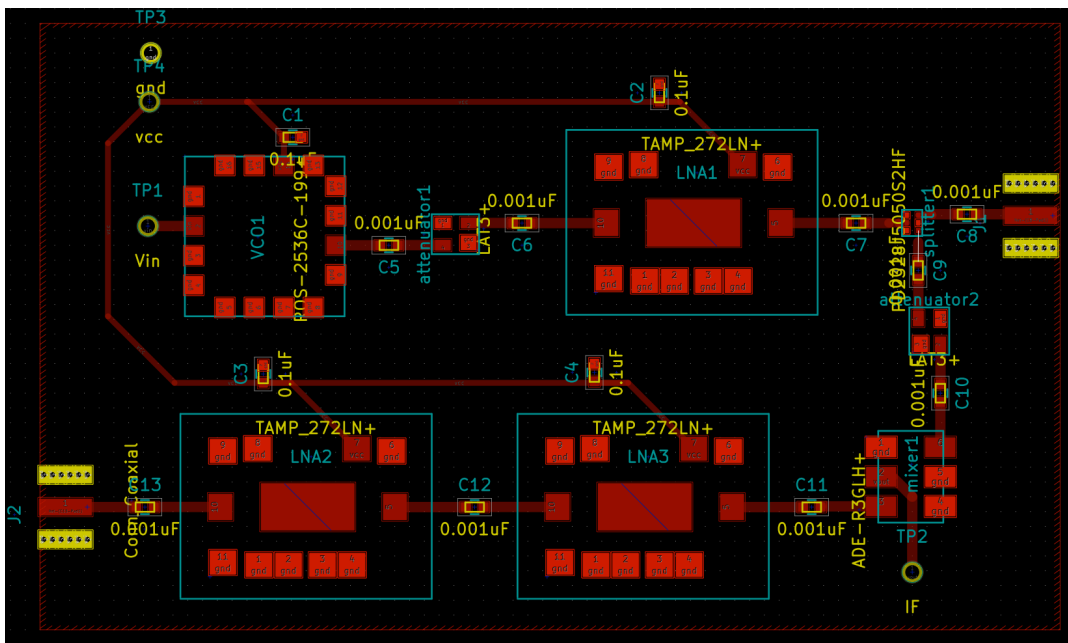


Fig 13: preliminary RF PCB

After a few iterations and cooperating with the baseband RF designer, I finally landed on a satisfying design, Fig 14.

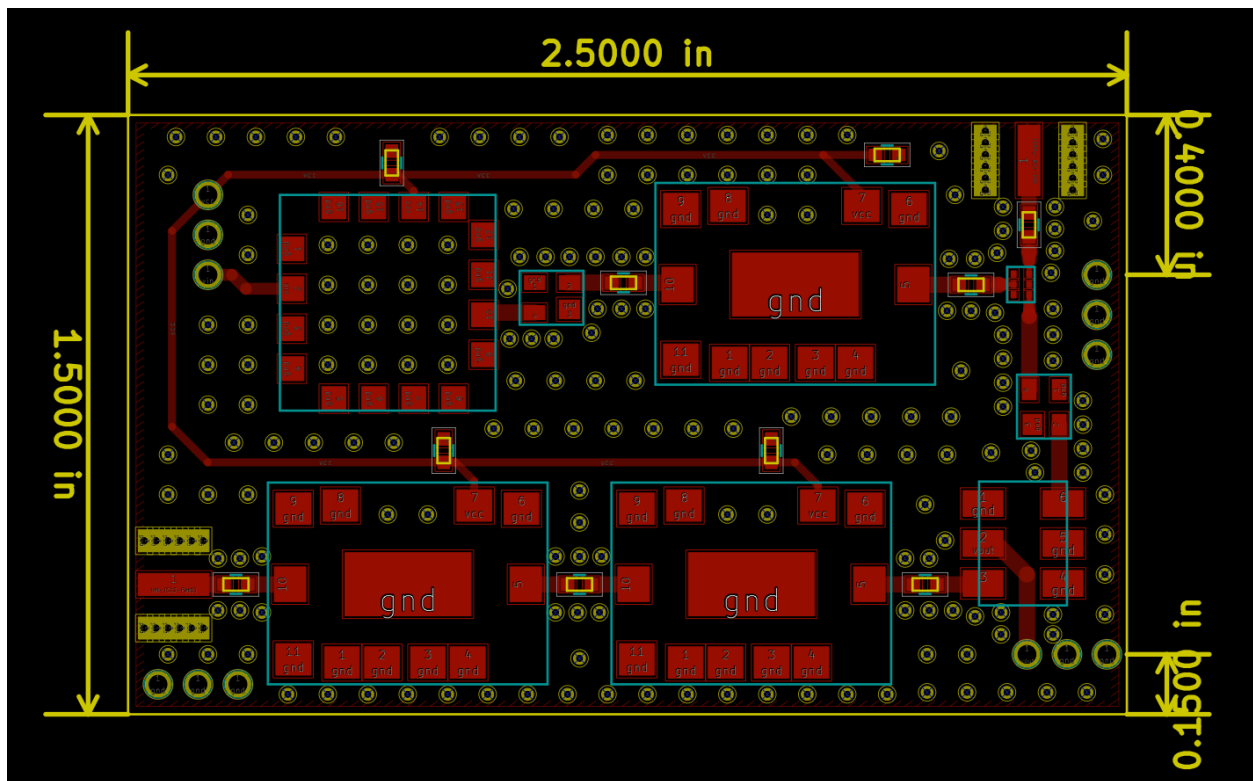


Fig 14: RF PCB final layout

When placing the components, it is also helpful to use different grid sizes to line up pads of two adjacent components. Otherwise, the traces might have a very small bend at one end. When placing the via fences, use array tool to save time. Although there is no exact rule to how far via fences should be placed from the signal trace, I recommend leaving some space between via fences and track. I also created 4 sets of through-holes on each corner to connect to baseband PCB via pin headers. Vcc and V-tune signals are placed on the top left and IF signal is placed on lower right, with other connections connected to gnd. Having extra ground connections between RF and baseband PCB provides better electrical and mechanical stability so it is highly recommended.

Notice that in the previous layout, there is a blue diagonal line across the center pad, Fig 15. At first, I did not pay attention to this line. However, when my team mate and I went through the data sheet, we found that the center pad should be shorted to gnd in suggested PCB layout, Fig 16. This part is very easy to miss because the center pad is not a pin specified in

the datasheet and therefore not in pin connection or footprint. To solve this issue, I edited the pad's pin name to /gnd to short this pad to ground. We were glad that we caught this bug before sending it to fabrication.

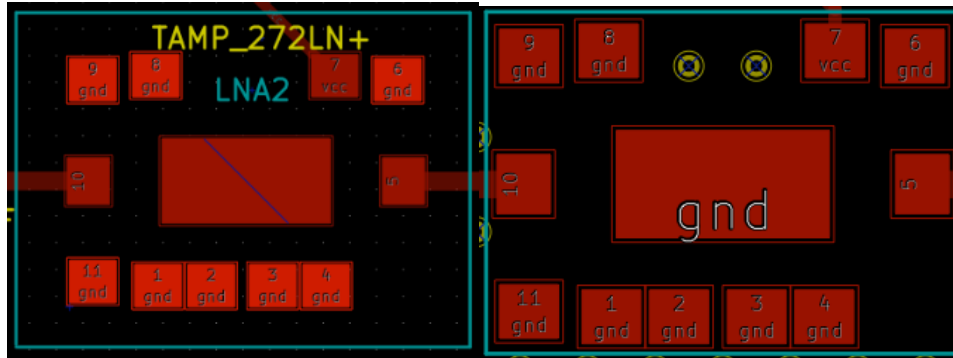


Fig 15: unconnected gnd pad vs. connected gnd pad

***Demo Board MCL P/N: TB-468+
Suggested PCB Layout (PL-293)***

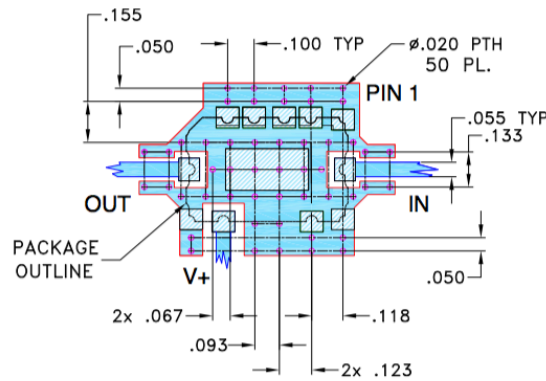
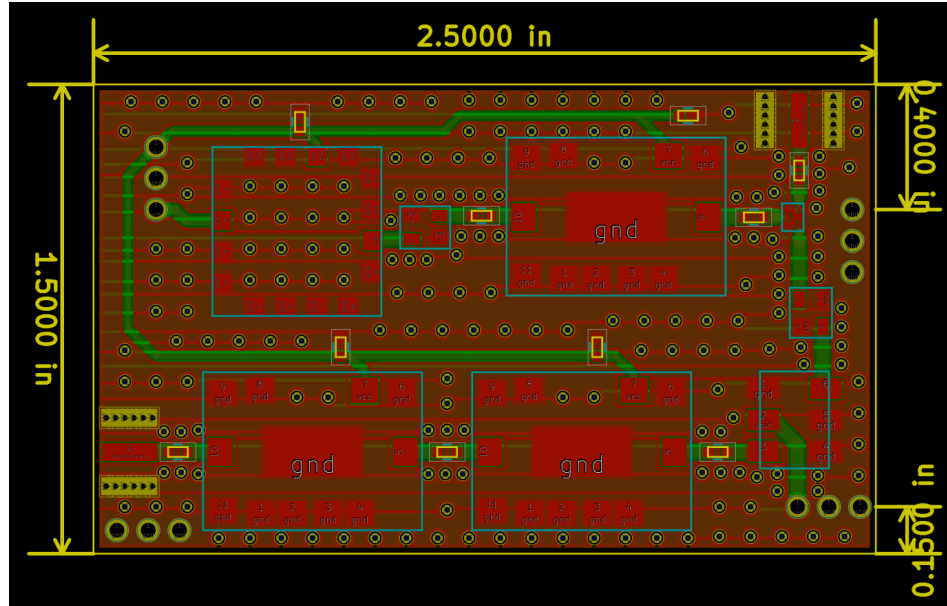


Fig. 16 suggested PCB layout

The figure blow shows the PCB with ground fill on both front and back, Fig 17. Because most of the time we work on PCB with filled zone turned off, it is critical that we turn the filled zone back on and refill all zones every time some edits have been made. KiCAD will not catch outdated fill zone for you.

After you are satisfied with the PCB design, run DRC to make sure there are no unconnected nodes. Then generate Gerber and drill files and submit them to DFM. If the design fails the DFM report, you need to fix the issues otherwise the PCB may not function properly.



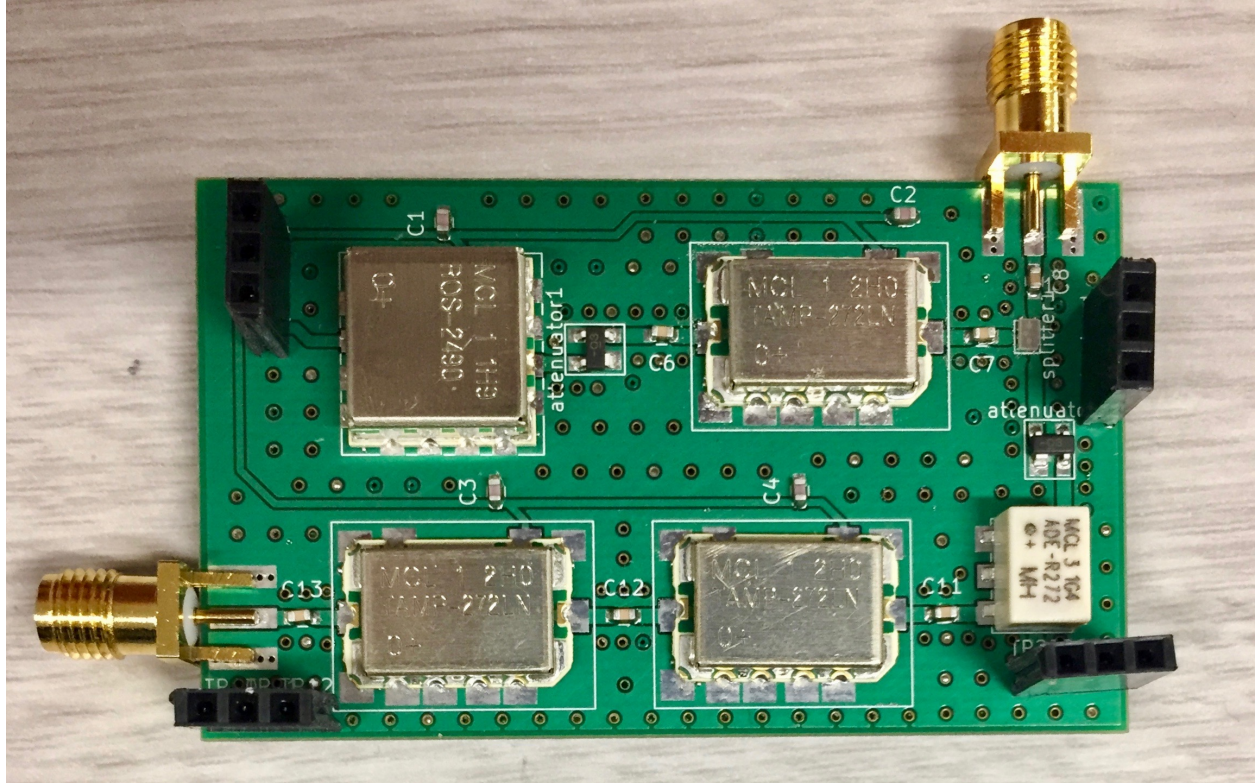


Fig 18: RF PCB with all components soldered on