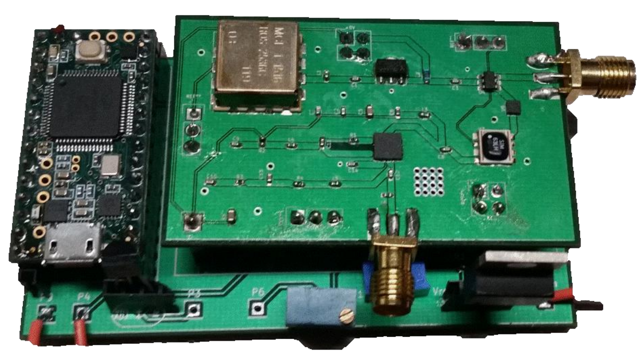
Application Note

Baseband PCB Layout and PCB Stacking

EEC 134

Team Radio Freqs

Michelle Chang



This note will be primarily about the layout of the baseband PCB and a stacking method used to connect with the RF PCB. This will not be a tutorial on how to use KiCad as that is covered extensively in the class tutorial.

Part 1: Baseband Layout

The baseband PCB consists of the Teensy, voltage regulator, low pass filter, the MCP, an op-amp, and various resistors, potentiometers, and other components. We choose these components to be on the baseband because they do not have any high frequency signals going through them. The high frequency signal gets generated and manipulated in the RF board before it gets mixed to a lower frequency and passed back the baseband board. I keep the high frequency components on a separate board so that the boards can be tested and debugged separately. The baseband board has a higher chance of working, so if the baseband board works and the RF board does not, we can redesign the RF board while keeping the baseband board intact. The baseband does not have very many parts, so the best method of reducing the overall footprint is to reduce the size of the RF board.

This layout is under the assumption that the two boards are stacked on top of each other. In order to use all the space on the board efficiently, you can arrange some parts to be underneath where the RF board will sit. This works best when you place components that do not need to be adjusted and can sit flat underneath, such as the low pass filter, the DAC, and any op-amps. These parts are soldered onto the PCB and ideally never touched again.

Another category of components are ones with a high clearance. Parts like the voltage regulator, the zener diode, and the Teensy all stick out. These parts would hit the bottom of the RF board, so they should be placed on the outer edge of the baseband board so that the RF board won't have to be raised several centimeters.

> Note: Place the Teensy with the USB connection facing an edge so that it can be easily connected to a computer or battery pack without hitting other things. (Not shown in image because we did not think to do this at the time.)

The last category of components are those that need to tested or adjusted. We use a potentiometer to adjust the gain of the received signal and this needs to be in an accessible location. Any added test points should also be accessible. Again, these are placed on the edge of the baseband board so that we can connect output pin or wires to the test points easily. The potentiometer is also easily reached with a screwdriver when it is on the edge. Leave a space for where the SMA connector will be connected on the RF board.

It is good practice to name all components and test points with silk screen. Although it is more common to name them by their reference number, I would suggest naming test points by what they are so that you don't have to memorize which test point is what.

That covers a basic arrangement of the baseband board to reduce the overall footprint and increase accessibility to certain components. While not the most important aspect of the radar or even the PCB design, neat arrangements will greatly help with debugging and assembly of the PCB.

Part 2: Stacking

Stacking PCBs is one method of organizing PCBs to keep the system compact while allowing for the two pieces to be tested separately. There are two reasons why the RF board is on top. First is that it is the smaller board of the two so it would fit within the borders of the baseband board. The second reason is that the RF board has a ground plane fill on the bottom side of the board to shield the baseband board from the high frequency signals. If the RF board was on the bottom, the high frequency signals would radiate up into the baseband board through the spaces in its ground plane.

The main goals of this stacking method we are to provide a strong connection of signals between the two PCBs and to give physical stability to the RF board. Figure 1 and 2 show the locations where we made the pin connections.

The connections were made with 2x2 and 1x3 pin header footprints. These footprints are suitable for male-male and male-female pin headers. Footprints with at least 3 pins are necessary so that the board does not wobble around, but more importantly so that the current in the node does not all have to pass through one pin. If one connection in the set fails, you have still have other pins that the current can pass through.

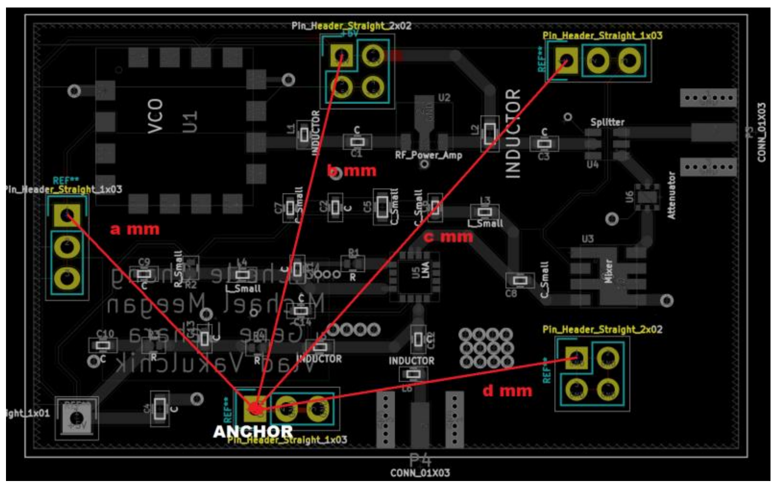


Figure 1.

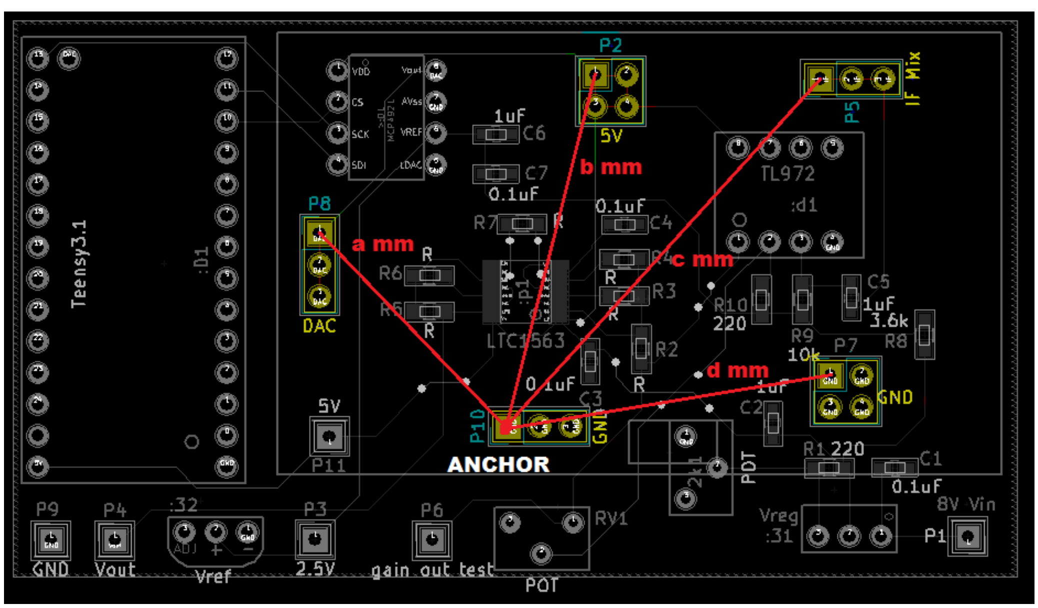


Figure 2.

The alignment of the connections are the most crucial part of this method. If the pins are even a little bit off from board to board, they may not be able to connect properly. The placement of the connections, however, do not matter greatly, just spread out enough so that the board can sit comfortably without wobbling. We use both male-male and male-female pin headers to give some clearance to the baseband components that lie underneath the RF board.

In KiCad

1. Determine how many connections you need. I needed 5V, GND, DAC output, and Mixer output.
2. Find shared areas on the two boards where you can place 2x2 or 1x3 connections.
3. Choose an anchor spot. Press the space bar to make this the new origin. This is where the position of all other connections will be referenced to.
4. Place the connections making sure that the x and y positions are the same on both boards.

Soldering

1. Solder rows of male-female pin headers to the baseband board, keeping them as straight as possible.
2. Solder rows of male-male pin headers to bottom of RF board.

You should now be able to place the RF board on top of the baseband board like in Figure 3.

Figure 3

With these two guides, you will be able to make a stacked PCB arrangement for you radar system!