

# EEEC134 App Note

## PCB Baseband Design & Stacking

Connor Bard  
999780392

### Introduction:

This application note will guide you through myself and my team's process in designing and fabricating our baseband PCB, including the stacking anchors that the RF board will fit into to complete the entire system design. Hopefully, this will inform you about PCB design techniques, and a few tips & tricks along the way. Strap in.

Note: [This series of Youtube videos from Skinny R&D were extremely helpful in understanding the basics of creating a project in KICAD, and was an extremely helpful walkthrough from creating a schematic to exporting the final files for fabrication. Please watch before or while following through the app note: <https://www.youtube.com/watch?v=o6IWdPpZ4mc>.](https://www.youtube.com/watch?v=o6IWdPpZ4mc)

### Part One: Baseband Design

The first step in any design is to build the schematic in KiCAD. This was done simultaneously or slightly after our ADISim simulations, and once we sourced the right parts that gave us good performance and specifications via ADISim, we could put them and their specific biasing circuitry into the schematic. The baseband board consists of the 5V voltage regulator, 2.5V voltage reference circuit, Teensy, DAC, op-amp, and low-pass filter, along with other various passives (resistors, capacitors, and inductors). Additionally, we included a single switch double throw switch to our circuit to switch between DAC and Teensy function generators with ease, although this was not necessary and if we were to do the project again, we likely would not include it, as we only really needed the output of DAC in the end.

These components were all chosen to be on the baseband board because they all have low-frequencies associated with them. On the other hand, all of the components that have high-frequencies running through them went on the RF board. This makes it much easier to assemble and debug the radar system, as you can run some simple tests to make sure each board is running individually before putting them both together. We found that the baseband board worked perfectly, but we had some problems on the RF board, so it was very advantageous to be able to keep the working baseband and then make changes to the RF board.

The baseband board works by having the Teensy/DAC stage input a 4 kHz triangle wave that is taken up from the baseband board to the RF board, where it is input into a VCO. After going through RF processing, a mixed signal is then taken down from the RF board into the baseband board, and goes through the gain stage and low-pass filter where it is then output into the computer. The 5V and 2.5V voltage reference are there to power the circuit. Because the circuit makeup of the final baseband board is very similar to the makeup of the baseband PCB1 from quarter 1, we reused a lot of the schematic from that PCB via copy-pasting. Copy-pasting in KiCAD is somewhat less intuitive than simply using Ctrl-C/Ctrl-V, and we had to look up how to do it by creating a new window and copying the project over. From there,

however, there were only some small changes to be made. Many test points were added so we could test the signals at almost any point in the system for proper function, I added the DAC/Teensy switch, and changed the large and complicated op-amp/low-pass filter system to ICs that required much fewer passives in their biasing circuitry than the quarter 1 systems. The biasing circuitry can be found in the datasheets, and MUST be found and followed carefully, otherwise the circuit will not work properly. We ran into a serious problem when we put the wrong biasing circuitry on our RF board, and so the board was wasted and we had to order another rush order. These changes from the quarter 1 op-amp/low-pass filter to smaller ICs and biasing circuitries, however, simplified the baseband board considerably. Finally, I added blue outlines to each subsystem in the circuit and labeled them for increased organization and clarity.

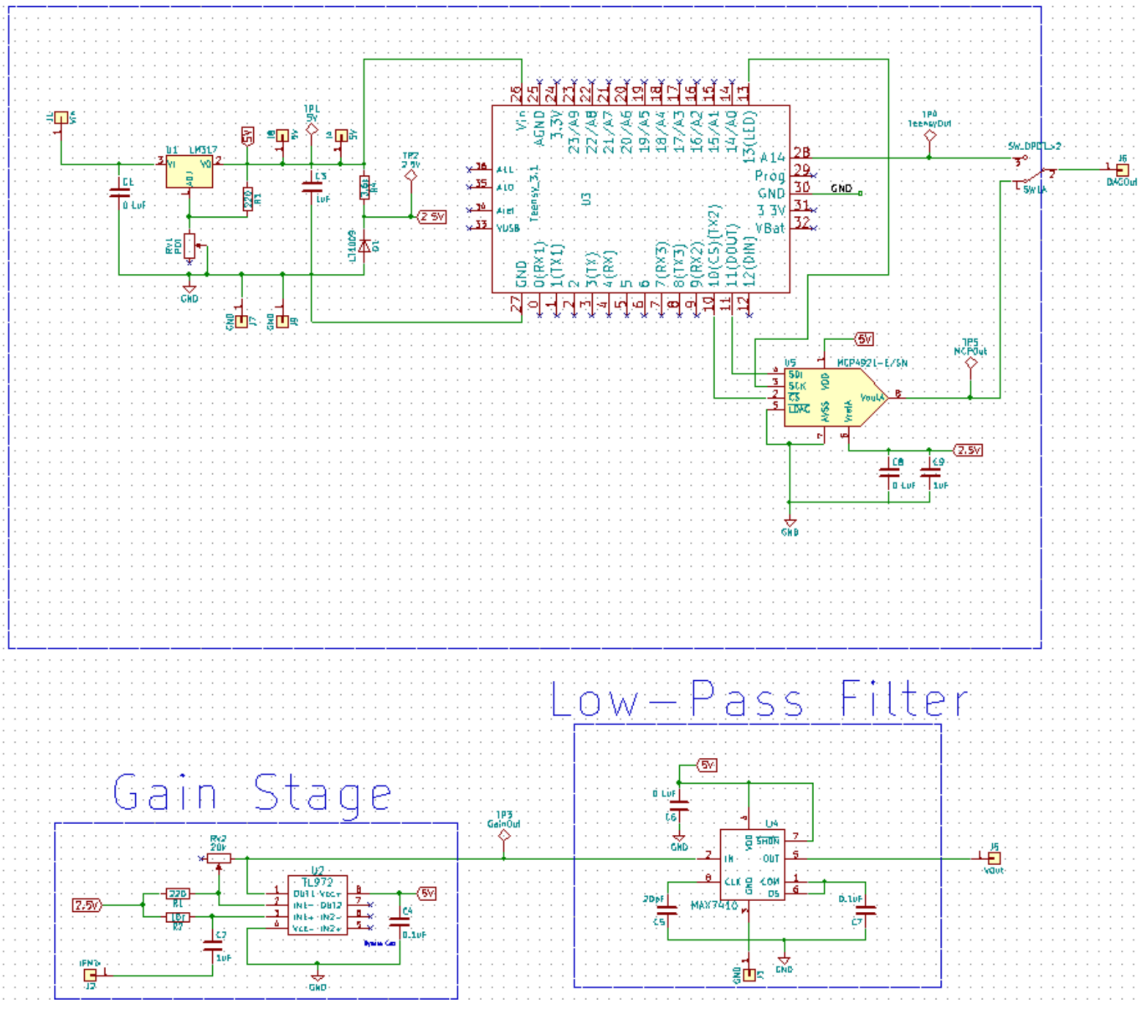


Figure 1. KiCAD schematic of baseband board.

## Part Two: Footprints, PCB Layout & Ground Planes

Once the schematic was laid out and properly connected on the schematic portion, it's time to move on to the footprint and PCB layout stage. Most of the footprints for components on your board should already be in the KiCAD component library. However, there are a handful that are not in the KiCAD library, most importantly the Teensy and probably some of your team-

specific components such as the op-amp, low-pass filter, and the majority of RF components. Additionally, the pin headers that are placed for PCB stacking may not be in the library in the number and size you want (2x2, 3x1, etc.). It is IMPERATIVE that you create the footprints with the right measurements and specifications, otherwise it is possible that your components will not fit on the board, and you will have to remake the footprint and board, wasting valuable time. The Teensy footprint was found online in various places, and you should have already found or created the Teensy footprint back in the first quarter. For the rest of the components, you need to find the datasheet for the component and find the measurements for the physical IC, which are usually found towards the bottom of the datasheet. Below in Figure 2 you can see the low-pass filter being created in the Footprint Editor. It is good practice to keep the component naming clear and organized. Two extremely valuable tools in making sure the measurements of each footprint are correct are the grid scale bar found on the upper left of the window and the x-y coordinates found at the far bottom. The grid scale can be changed in the drop down bar and allows you to change coordinates more precisely, and the x-y coordinates will tell you how far in either direction you are creating your footprint. **Additionally, you can press the space bar on a**

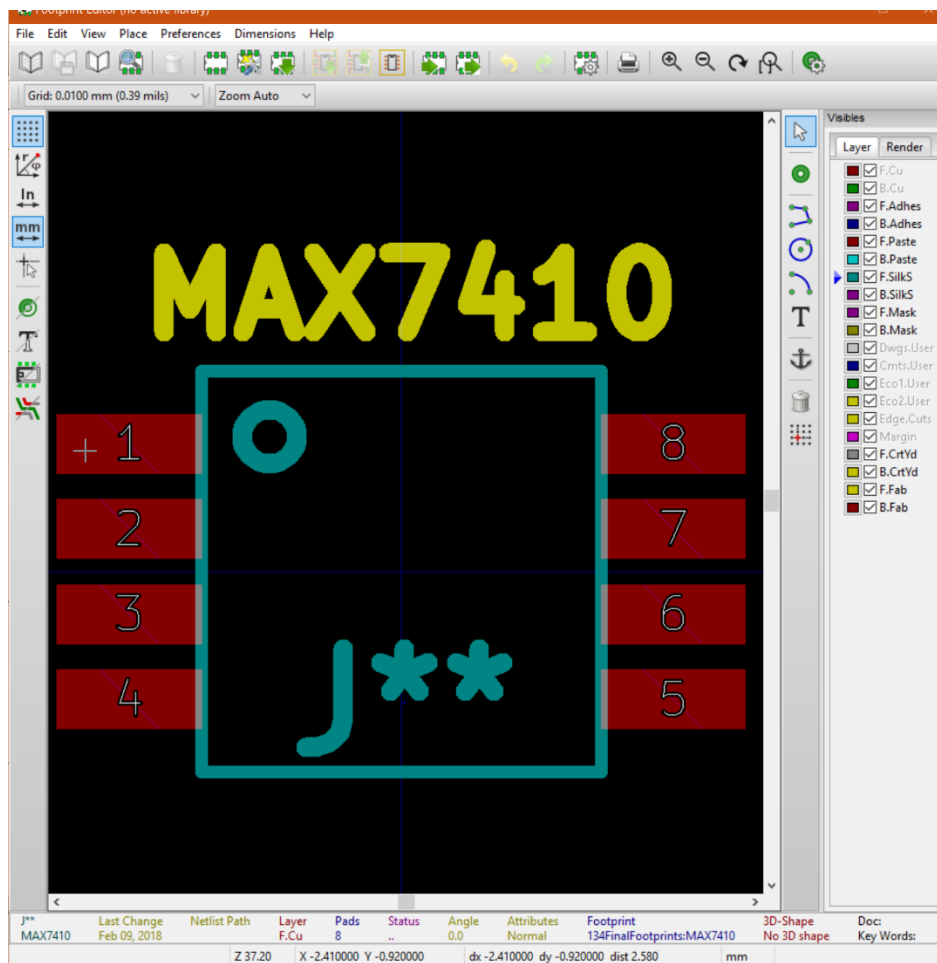


Figure 2. KiCAD footprint creation screen. The x and y coordinates can be seen at the very bottom of the photo.

point within the editor window to change the dx and dy values to zero, which can allow you to measure the distance from that specific point to any other point in the editor window. This is extremely useful in measuring distances that aren't measured with respect to the absolute center (which happens all the time), and is invaluable when making sure the distances for the PCB stacking anchors are at the same coordinates on both the baseband and RF boards (discussed later). Once all the footprints are made, it is good practice to put them all in the same library directory, which will then have to be added to the library wizard. There are many tutorials on how to create a footprint and add the library to the wizard online, on YouTube and elsewhere. Then, you can find the footprints in the footprint library, and add them to the netlist. Then, you are ready to layout your PCB!

Once the footprints are all made, the library is added, the footprints are added to the netlist, and the netlist is then imported into the PCB layout screen, you are ready to work. The layout of our two boards stacked on top of each other means that the vertical height of each component must be taken into account as well as the length and width. Tall components such as the Teensy, DAC, 5V and gain stage potentiometer, diode, and 5V regulator must be placed outside the RF board footprint on the board so the RF board can fit. These components can be seen placed outside the RF footprint below in Figure 3. Additionally, it is good practice to place the Teensy in such a way that the Micro-USB port faces away from the board, so it can be easily connected to a computer or battery without getting in the way of other components on the board.

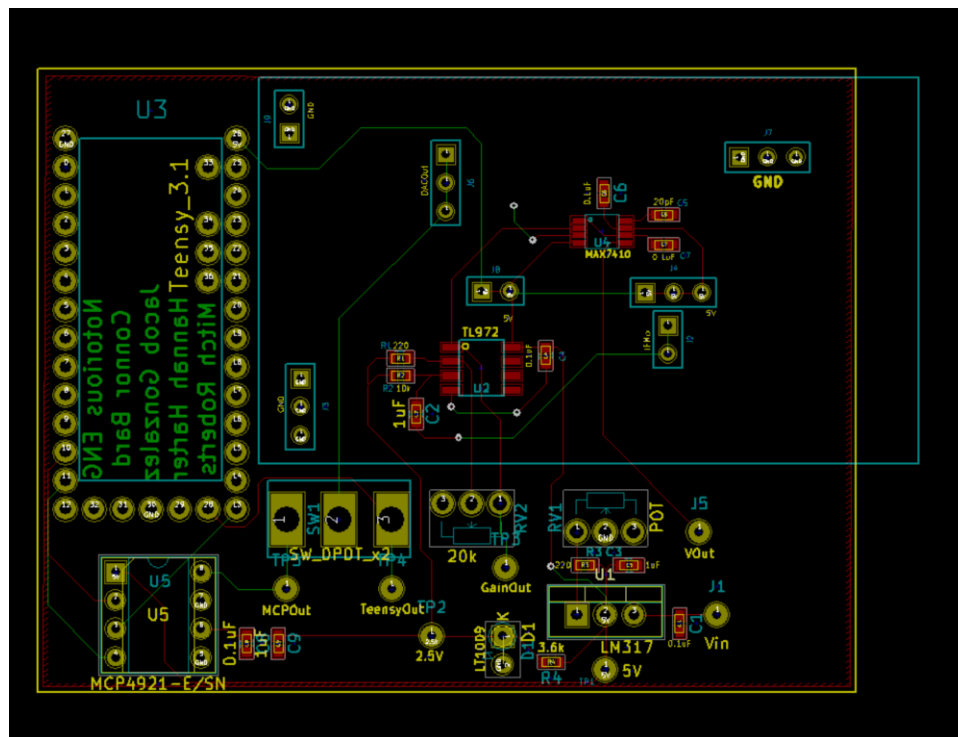


Figure 3. Photo of the completed baseband board in KiCAD. The RF board footprint is outlined in the light blue rectangle that hangs over the board.

After the components were placed in the right configurations, a ground plane needed to be added to the top of the board. This was easier than creating one singular ground point and having tracks go from every component to that singular point. Thus, a ground plane was vastly simpler. However, due to the placement of some other signal tracks, there were occasionally some ground plane islands that were cut off by other signal tracks. To remedy this, vias (holes through the board that allows tracks to move from the top side of the board to the bottom and vice versa) were created that stitched the ground planes together. This can be seen below in Figure 4.

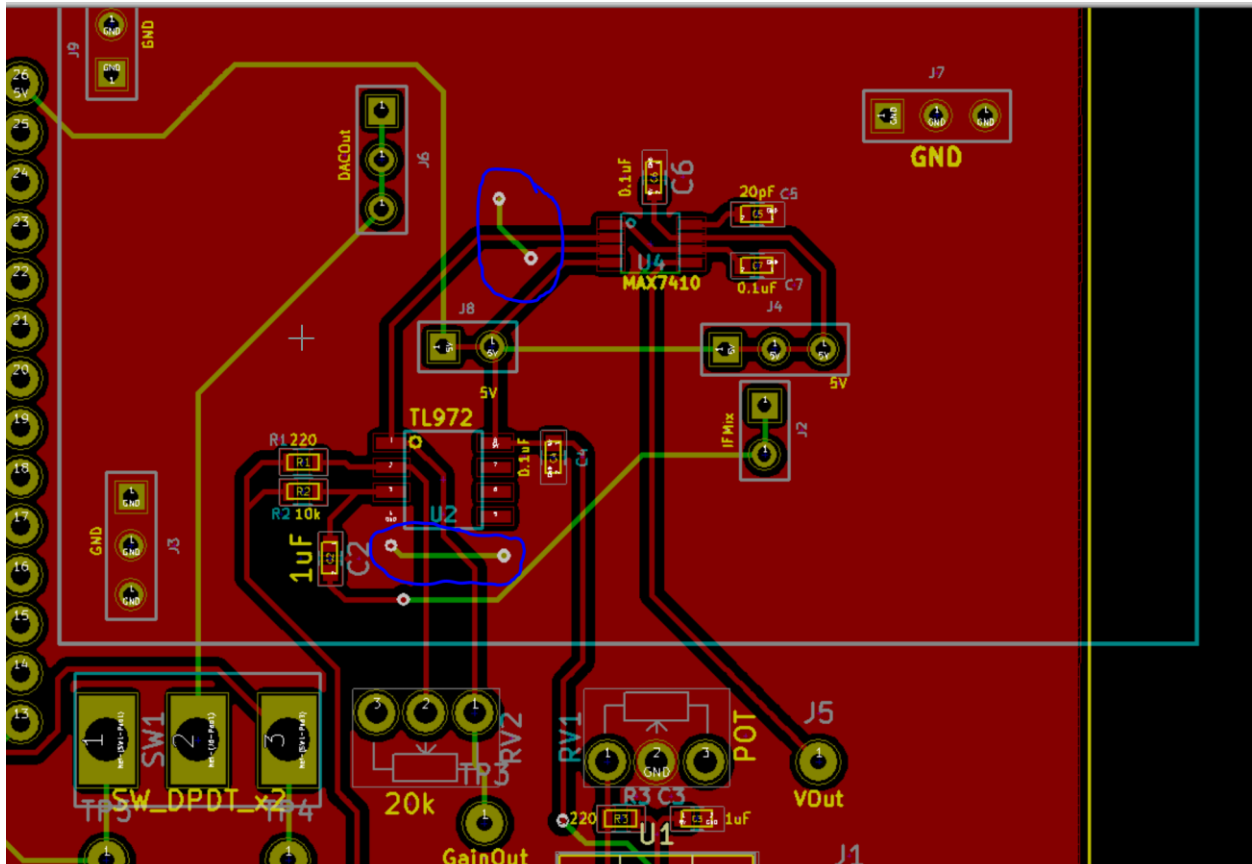


Figure 4. Final KiCAD PCB design with ground planes filled. The two vias connecting isolated ground plane islands are circled in blue.

### Part Three: Anchors and PCB Stacking

In order to keep the system compact, and also to make debugging easier (as explained above in Part One), we decided to stack the RF PCB on top of the baseband board, instead of having all the components on one board. We chose to have the RF board on top instead of the bottom because it was smaller, and because the ground planes on the RF board needed to be on both sides to shield the baseband board from high frequency signals. If the baseband board was on the top instead of the bottom, the high frequency signals from the RF board could radiate up from the top of the RF board into the baseband board and cause problems. When stacking the PCBs we want to ensure that there are enough anchors so the RF board doesn't wobble on top

of the baseband board and cause connection problems. Thus, you should have at least 3 or 4 to prevent wobbling, although we ended up having 6 to ensure all of the signals that we needed from the baseband board reached the RF board in the right places. Additionally, these connection points should have more than just one pin header attached to each one, to have connection redundancy and prevent wobbling (2x2 or 3x1 pin headers are recommended). Figures 5a and 5b below show the anchor points of a previous team's baseband and RF boards. The anchors were placed with the same spacebar method discussed earlier in Part Two. The cursor was placed at the center of a designated pin header, and then spacebar was pressed to zero out the dx and dy reference points at the bottom of the screen. From there, the distances to each other pin header can be calculated. With these measurements, the pin headers can be placed on both boards with the same distances away to ensure that the connections meet exactly. If the measurements and connections aren't exactly accurate, the boards might not fit into each other, and you will have serious connection problems (and will likely have to remake one of the boards).

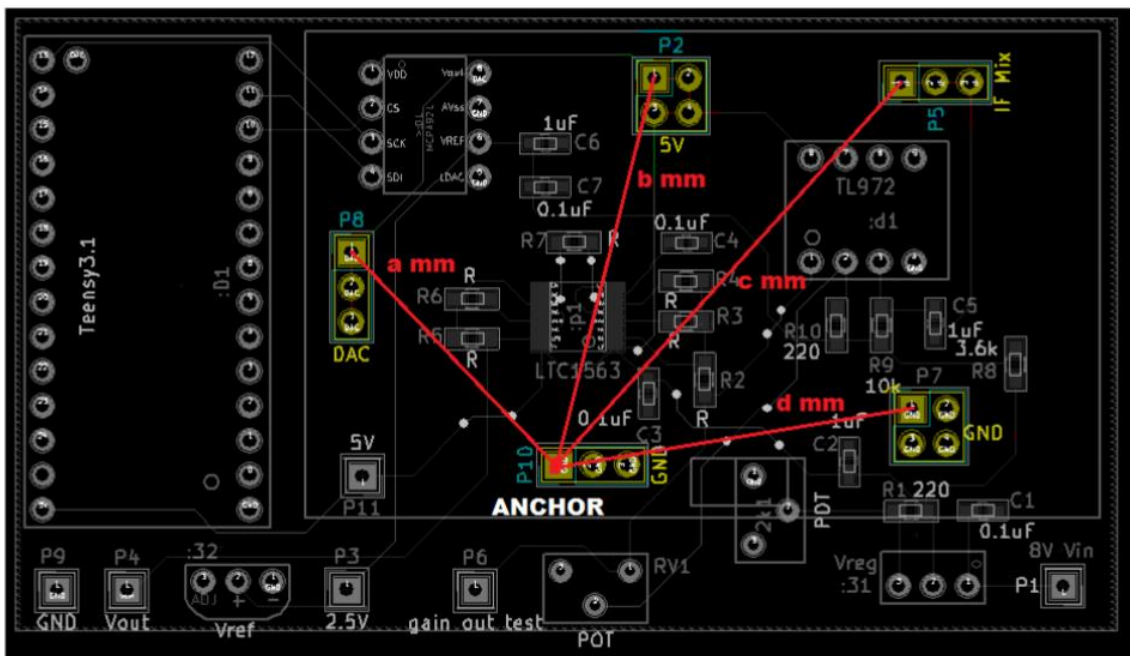


Figure 5a. PCB footprint showing stacking pins relative to a center anchor point on the baseband board. (Courtesy of Team Radio Freqs, Michelle Chang)



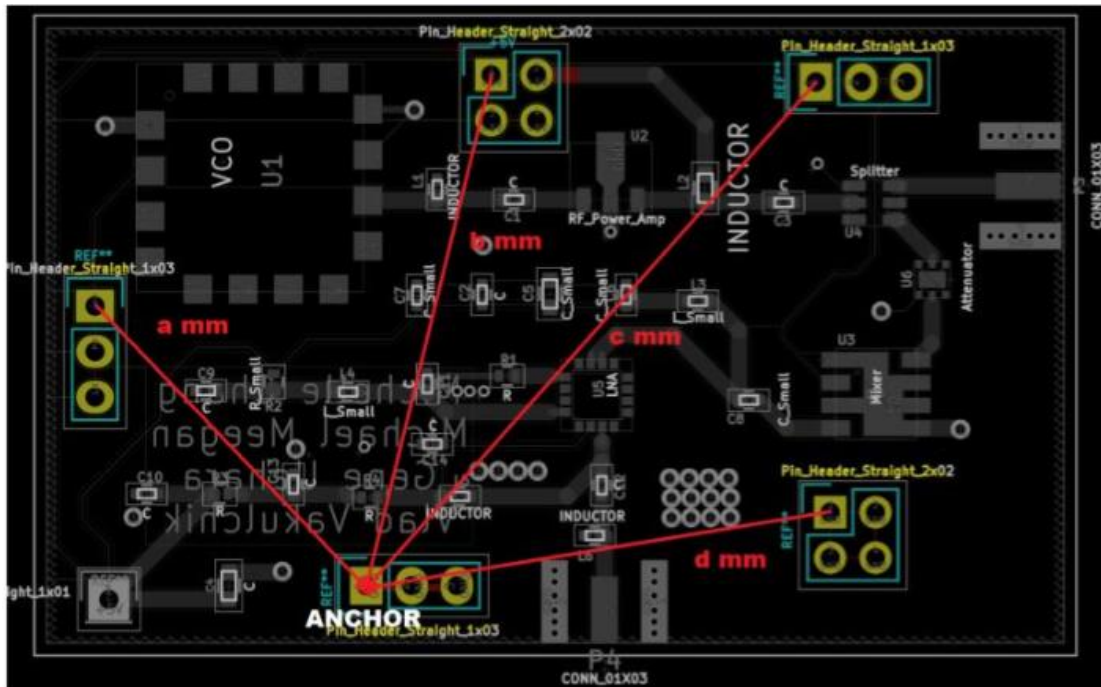


Figure 5b. PCB footprint showing stacking pins relative to a center anchor point on an RF board.  
 (Courtesy of Team Radio Freqs, Michelle Chang)

Once the boards are manufactured, you can then solder male-female pin headers onto the top of the baseband board, and male-male pin headers onto the bottom of the RF board to place the RF board on top of the small baseband board. Figure 6 below shows an example of male-male pin headers fitting into male-female pin headers. Once the pin headers are on you should be all set to go!

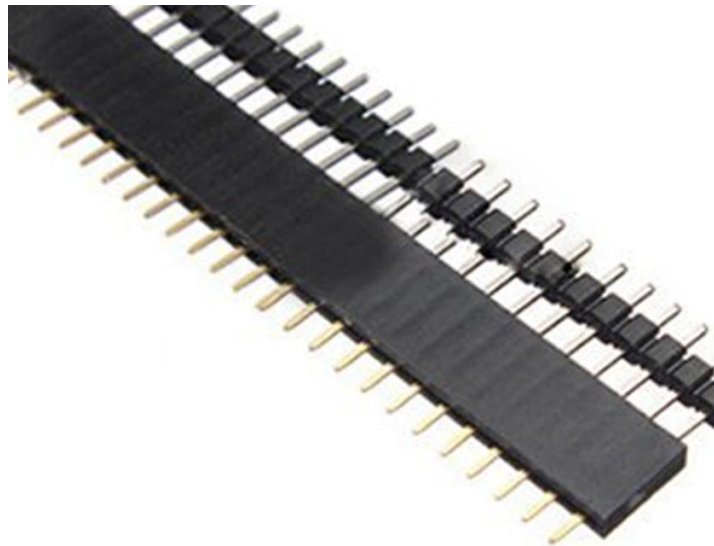


Figure 6. Male-male pin headers inserted into male-female pin headers.