

Application Note: RF PCB Design
EEC 134B Winter 18'
Hannah Harter

I. INTRODUCTION

In this application note, I will discuss how to design and layout a RF Printed Circuit Board (PCB). This discussion will focus mainly on an RF board for UC Davis' EEC134 course using KiCAD software, but many of the considerations I will discuss are relevant for all RF PCBs created using any PCB design software. Also, this application note is written with the assumption that the reader has working knowledge of KiCAD software.

II. PREPARING FOR LAYOUT

A. Schematic

Before creating the schematic in KiCAD, you must first have a total system design in mind. Your design will likely be similar to the EEC134 quarter 1 system design. Split the total system design into an RF part and a baseband part. An RF signal has special considerations when putting it on a PCB board so simplicity is key. Simplicity is especially important to keep in mind when selecting RF components as many of them have complicated biasing requirements. That being said, chose the simplest design and the components with the simplest biasing circuits before creating the schematic in eschema.

Figure 1. System block diagram

B. Footprints

Although footprints for common components, such as inductors and capacitors, come with the KiCAD software, you will probably have

to create custom footprints for most of the other not as common components, such as a VCO, LNA, etc. Footprints are important to create correctly before beginning the PCB layout. If you realize a footprint has a mistake after doing all the layouts, changing it might alter a lot of the layout if components are close together, and you'd create a lot more work for yourself. Even worse, if you don't notice the footprint had a mistake until receiving the physical PCB, it will not work. You would then have to resort to some sort of hack that would probably decrease the performance of the PCB and also be a lot harder to implement than just creating the correct footprint the first time.

There are other considerations to keep in mind when creating footprints that will help the whole process go more smoothly. The first is making the pad numbers on the footprints correspond to the port numbers in the schematic. Also, if different pads are all connected to the same net, give them the same pad number and they will automatically connect. By doing that, you'll avoid having to add traces between those pads. I would also recommend using the edit pad function in the footprint editor to make sure the pads are the correct sizes and the correct distances apart instead of altering the grid sizes and moving pads around that way. Although altering the grid size and moving pads around does work, just inputting the exact spot you want the pad is a lot easier and that way you can be positive it's in the correct spot.

III. LAYOUT

A. Traces

The traces are one of the most important parts of the board design. Turns in the traces significantly alter the signal and therefore, should be avoided. Align all your components so that they can be easily connected with minimal turns in the traces. You only need to be concerned about traces for RF signals and not for baseband signals. Some turning of the RF traces is probably necessary, but try to minimize them and definitely arrange the components so that you have no 90 degree turns on your board.

RF traces also need to be impedance matched. If you don't know how to do that, the PCB tutorial Professor Leo wrote covers it. For my board, the traces need to be 37 mils for impedance matching. Since this trace is rather large compared to some of the smaller pads, like the 0603 capacitors, you'll have to taper the trace. Since tapering a trace takes up a certain length on the board, sometimes it is better to just use a smaller trace width between two components if they are really close together. For example, since biasing circuitry should be put as close to the main component as possible, it is better to use a smaller trace width between the main component and the biasing component than to move the biasing component farther away solely for the purpose of leaving space to taper the trace.

Ultimately, while it is important to try and limit the size of the board, following the above specifications is more important. It is better to have a slightly larger, well-performing board than a slightly smaller low quality board.

B. Connecting to Baseband

For the purpose of the EEC134 project, you'll have to connect the RF board to the baseband board. Our group decided to stack our PCB's. Stacking the PCB's requires test points between the two boards for each signal needed. For this project, we needed VCC, GND, DAC output, and Mixer

output. Since the RF board is a lot more "touchy" than the baseband board. Deciding where the test points work best on the RF board and then just making that work for the baseband board is best approach for connecting the two boards.

IV. ADDITIONAL CONSIDERATIONS

A problem I ran into was that I forgot to refill the ground plane after moving a couple of components around. This caused almost half of the board to be grounded and we ended up having to do a rush order of a new PCB. So, be extra careful that refilling the ground planes is the last thing you do before creating the drill files.

I already mentioned that a slightly larger PCB doesn't make much of a difference in the weight for the overall system. But I just wanted to reiterate that because having to resort to using the quarter 1 blocks will increase the weight significantly.

One last tip is to actively double check your board. Instead of just sending the design to another group member and asking them if everything looks good, sit down with them and go through the entire design and make sure everything is good. A couple hours spend doing this can save you many many more hours later and make the difference between having a working system at the competition or not.