Alexis Torres

Team Falcon 9

EEC 134 Technical Note

During the second half of the course I worked a lot with the RF PCB design. Our team originally had a baseband PCB and an RF PCB, but the baseband PCB was very complex and we were not able to get it up and running for the competition. Luckily we had the baseband circuit moved to a protoboard as a backup and then combined it with the working RF PCB. Looking back, I am really glad that we decided to design two PCBs in order to make debugging the circuits easier. If we had combined everything onto one board it would have been a nightmare to debug and it would have pushed our testing of the overall system far back. It is also a good idea to keep the two portions of the system separate because the RF board requires different techniques for the design compared to the baseband PCB.

When making an RF PCB the trace widths matter very much since we are working at higher frequencies. Impedance matching comes into the design at this stage. The characteristic impedances of the transmission lines must match the source and load impedances to minimize loss from reflections. The lines should also be as straight and short as possible, avoiding bends in the design, to keep the loss to a minimum. For this reason, we implemented a coplanar waveguide (CPWG) design in order to minimize the trace widths and achieve the 50 ohm match required for the RF components.

Some important things to note when designing the RF PCB are the necessary steps that need to be taken in order to achieve good isolation for the signals. To do this, a technique called via stitching is used throughout the board and along the signal carrying traces. This technique

uses vias that are placed along the outsides of the signal traces to connect the top and bottom layers of the boards for grounding. This basically creates a cage around the signal trace that prevents it from being heavily affected by other signals and from affecting other signals as well.

One last suggestion when designing an RF PCB, and essentially all PCBs, is to triple check your component footprints to make sure that the dimensions are correct. Have your teammates check any dimensions that you feel are correct as well just to make sure. If the footprint dimensions are off this could hold your progress back by a couple of weeks until the next PCB comes in with the corrected dimensions. This step should also include reviewing the PCB pin connections because if something is not connected correctly or grounded then the PCB needs to be redone as well. Overall there are several more things to consider when designing an RF PCB than a PCB working with lower frequencies since transmission line effects come into play with the higher frequencies.