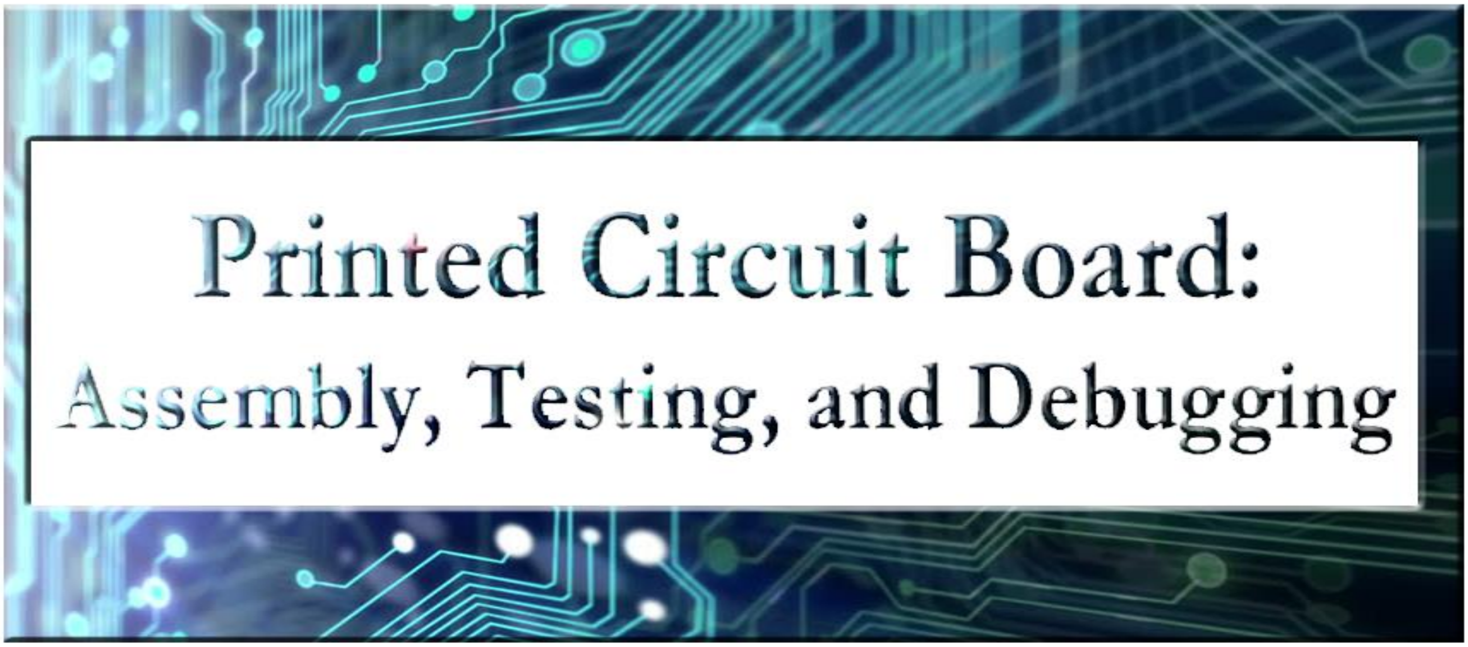


Stefi Katada
sgkatada@ucdavis.edu
TEAM HERTZ
EEC 134: RF Senior Design Project

Table of Contents

Introduction

This paper is a supplement to Team Hertz's Final Report for EEC 134: RF Senior Design Project. This paper will explain Printed Circuit Board (PCB) (1) assembly techniques, (2) tests performed, and (3) debugging done for Team Hertz's radar system. To ensure that the system is working at its outmost performance, a pcb well-soldered or assembled is a must. There are certain techniques that can be used for easier and faster pcb assembly. The most challenging part of the whole project is debugging the PCB. Hence, a documentation of the ways our team has debugged our radar system may be of help to others.

Get To Know the System

Our system is a Frequency Modulated, Continuous Wave (FMCW) Radar. This system will measure the distance of an object detected by the radar within its cross-section. Figure 1 shows the block diagram of the system design. The detailed explanation of the system design and its expected performance is written in Team Hertz's Final Report.

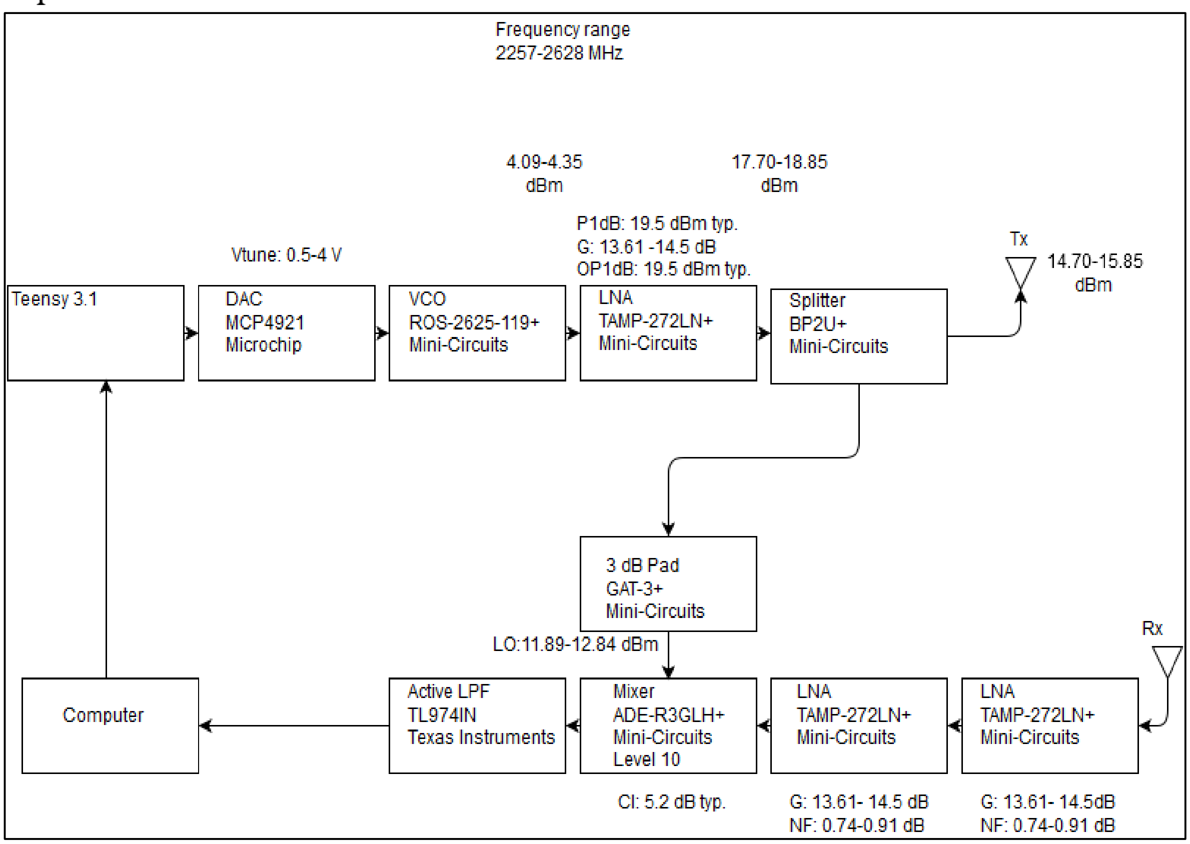


Figure 1. FMCW Radar System Block Diagram

PCB Layout

There are three PCB designs built for this system throughout the whole project. The first PCB is for the baseband portion of the system and the second PCB is for the RF part. The third PCB is for the whole system (i.e., combined baseband and RF).

Figure 2: Baseband PCB

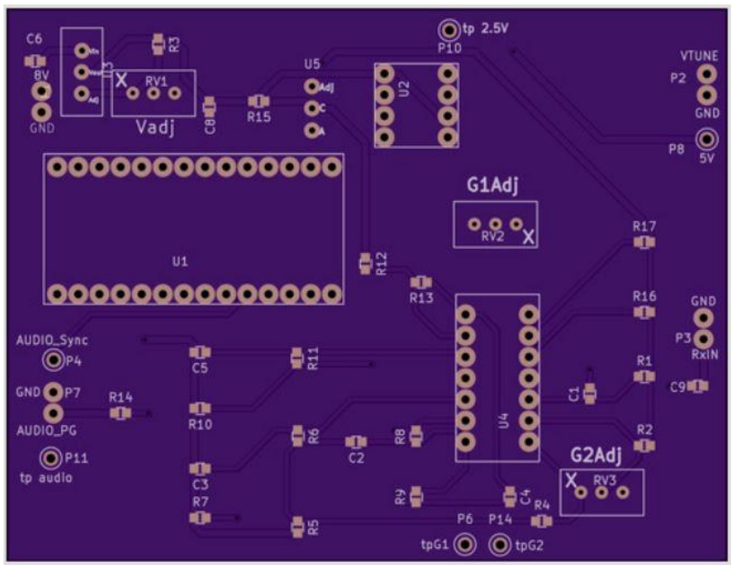


Figure 3: RF PCB

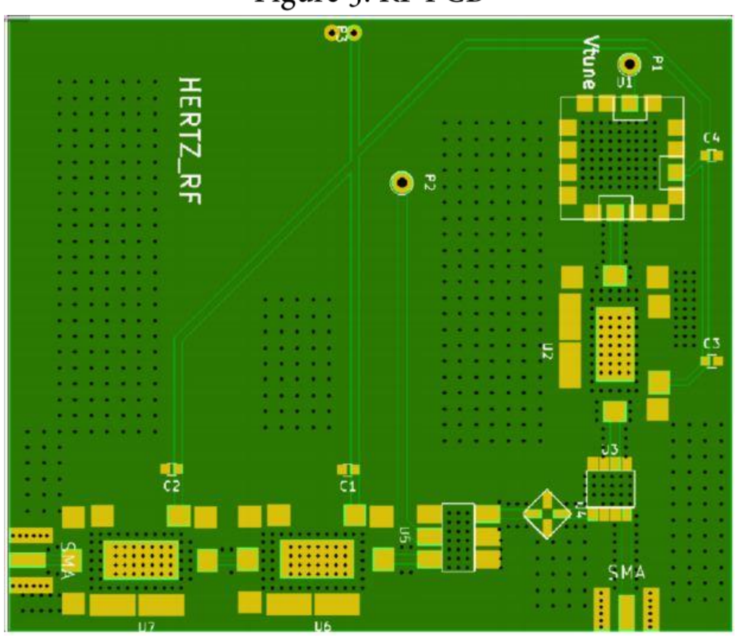
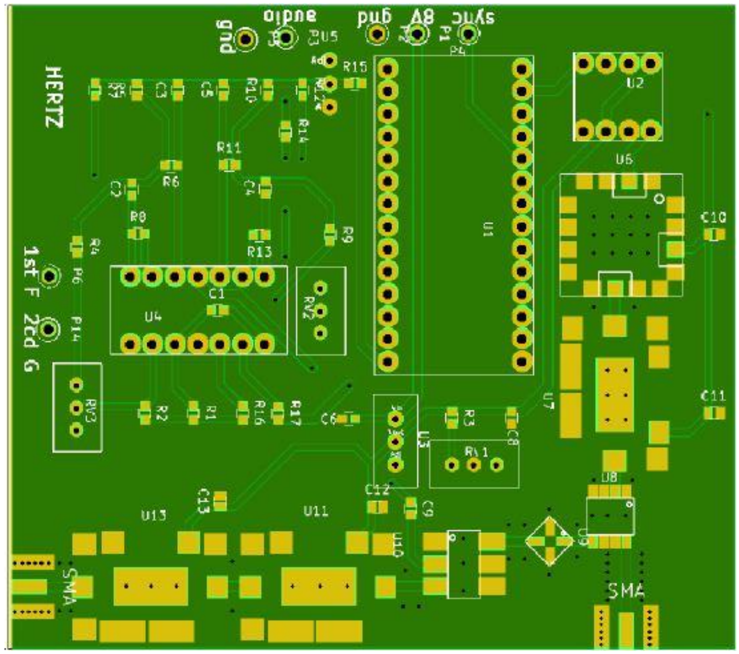


Figure 4: Combined RF and Baseband (Radar System)



Assembly

Notice that in our PCBs there are through hole plating rather having everything in pads for surface mount components. Time consumption becomes the main element in deciding which type of component should be soldered in the PCB first. We find it better with regards to time consumption to solder the surface mount components before soldering the through hole components since the heat distribution from the hotplate is quicker that way. If we solder the through hole components first, the back of the PCB might not be at the same level when placed on top of the hotplate. The heat distribution then becomes slower and uneven. It becomes problematic when the heat reaches a component that you do not wish to move first.

Soldering Surface Mount Components

Note that in soldering surface mount components, the size of the pads in the PCB and the size of the component must corresponding correctly. A bigger or smaller size component than the required size of the pads will result in a bad connection.

Materials needed for soldering surface mount components are as follows:

- Solder Paste
- Hotplate
- PCB Fine Pointed Tweezers
- Flux
- Thin Fine Pointed Wooden Skewers

All of the above materials are provided in the Kemper lab rooms; thus, no purchase is necessary.

The pad sizes for the resistors and capacitors in our system are very small in comparison to the tip of the syringe of the solder paste dispenser. Do not use the solder paste dispenser syringe in placing solder on the pads. Grab a tiny dot of solder paste using the pointed tip of a thin fine pointed wooden skewer and place the solder paste in the pads. Use a fine pointed tweezer to place the component as precisely as possible. Repeat this process for all the surface mount components then place the PCB in the hotplate. Turn the knob of the hotplate to warm or low heat setting. Avoid using medium or high heat setting for soldering the surface mount components of our system to prevent burning the components and PCB damage.

There is a way to make the above process easier and that is by using flux. Place a generous amount of flux across the pads before placing the solder paste. The flux ensures that the solder paste attaches only to the metal pads evenly. There is a disadvantage in using this process. The flux will become a hindrance for debugging the PCB later on.

Soldering Through Hole Components

Materials needed for soldering the through hole components in our system are as follows:

- Soldering Iron (Chisel Tip)
- Solder Wire (less than .032 diameter)
- PCB connectors

The several through hole plating in our PCB is for pin connectors. Most of the PCB connectors used are straight dip type connectors and female round pin header connectors.

There are several soldering irons available at the Kemper lab rooms. A soldering iron with a chisel tip works best in soldering the PCB connectors. The chisel tipped soldering iron delivers heat evenly compared to a conical tipped iron. It is not advisable to use a soldering iron where you cannot adjust the temperature. Depending on the size of the through hole plating, the heat distribution varies. The size of the through hole plating is directly proportional to the heat distributed by the soldering iron. The solder wire that has a diameter smaller than .032 is sufficient. Web site links to videos regarding PCB assembly are listed in the course lab 1 manual from Quarter 1 (Fall Quarter 2015).

Testing

Baseband PCB

To test the baseband PCB, we will need an Oscilloscope, Wave Function Generator, Power Supply, and Multimeter. Before inputting a voltage to the PCB, check that the connections between each components are not shorted. There are multimeter devices and machines available in the Kemper lab rooms in order to measure resistances and voltages. However, those multimeters cannot measure capacitance. Our team used Amanda Williams's personal multimeter which can measure capacitance. To know whether a component is shorted or not, we use the multimeter to measure the resistance of a resistor or the capacitance of a capacitor. The resistances or capacitances do not necessarily have to agree with the original value of the resistances or capacitances of the components especially when those components are in a parallel connection. However if the multimeter reads a zero, then the connection across that certain component is shorted. It is a must to make sure that the multimeter is reading via the pads and not the surface mount components. If multimeter measures zero for a particular component, then resolder that component only. Once the connections are all examined, connect the PCB to a power supply.

Initial testing of the baseband PCB does not require the teensy to be connected. The wave signal is generated using a Wave Function Generator available in the Kemper lab rooms. We connect the baseband PCB to a power supply that is outputting 8 V. If the power supply machine displays zero voltage, turn it off immediately to prevent damage on the PCB then debug the PCB. If there is no short, then proceed with the test. There are test points in our PCB in order to evaluate the output of the first gain stage, the second gain stage, and the overall baseband output. The test point pin labeled RxIn in our baseband PCB will be used to connect a wave from a function generator. Connect the first gain stage test point pin to the first channel of the oscilloscope, while the second gain stage test point pin connects to the second channel of the oscilloscope. If voltage peak-peak and frequency does not match the desired values then debug the PCB.

RF PCB

Testing the RF PCB alone is extremely difficult and may not be possible. To test the RF PCB, we will need to connect it to the baseband PCB. The test for the RF PCB can only be done after the baseband PCB is debugged and performing approximately as it should be. Once the baseband is up and running, we connect the RF PCB. Testing whether the RF PCB is working requires the whole system to be connected. Debug the RF PCB if the system does not work.

Debugging

Baseband PCB

Modify the settings in the multimeter to measure DC voltage. First, check that the baseband PCB is getting the right supply voltage (8 V). Second, check that the center pin of the voltage regulator measures 5.0 V using the multimeter. Adjust the potentiometer to make the output of the voltage regulator to 5.0 V if the output initially was not the right value. Third, check whether the 2.5 v test point pin outputs a 2.5 v. If all the voltage test points are validated to have the correct voltage values yet the oscilloscope displays the wrong output, adjust the potentiometer of the gain stages. The gain stages' potentiometers should be around 890 ohms or less. On the other hand if the 2.5 V test point pin outputs a different voltage, there must be a shorted connected across a resistor or a capacitor somewhere in the PCB. Usually when we get a value of less than 2.5 V on the 2.5 V test point pin that means there are components that are not connected properly in the gain stages or the low pass filter stage. We use the multimeter to check the DC voltage value across each component of the PCB. We have yet to find a way to debug this quicker than checking the voltage across each component. For now when we find a component that has an inaccurate voltage reading across its pads, we resolder that particular component. We repeat this process until every component is connected properly.

RF PCB

A spectrum analyzer and power supply will be needed. We have to guarantee that the RF components are connected at the right pad. When soldering the RF surface mount components, one must not carelessly place the components without checking its datasheet. Otherwise, the system will not work. The 2.5 V should flow through the RF components without a problem. Check the 2.5 volt line of the RF PCB using a multimeter. Resolder the component if the multimeter gives the wrong voltage. After confirming the 2.5 volt line, check the output of the VCO using a spectrum analyzer. Note that the measurements should be done via the pads rather than the component itself. This assures us that we are measuring the PCB trace connection. Inspect whether each RF component receives the wave output of the VCO. If the other RF components are not receiving the wave output, resolder the VCO. Repeat the initial process for debugging the RF PCB until the problem is solved.