

# Baseband PCB Suggestions – EEC134 Appnote

Yuanyuan Xiang

## I. INTRODUCTION

For the radar system we built, one of the major parts is the baseband Printed Circuit Board (PCB) which supplies Vtune for the Voltage Control Oscillator in RF PCB and amplifies and filters the output signal from Mixer. Also, baseband PCB has a voltage regulator circuit that provide operating voltage for the RF PCB. In this application note, I will discuss how to design, layout and refine a PCB of baseband circuits (no RF components). The PCB design described below was processed in ‘KiCad’ software.

## II. BASEBAND PCB

### A. Circuit design

Based on our primary proposal, the new radar system would be built upon quarter 1 design. Therefore, the functions of baseband circuit included conversion from 8 volt to 5 volt, generation of triangle wave for VCO and a low pass filter and a gain stage for output signal from mixer. In addition, the circuit also needed to convert 8 volt to 3.3 volt which is the voltage supply for the microcontroller, CC3200. With the knowledge on the functions of baseband circuit, I designed the circuit based on the circuit form lab1 manual from quarter 1. We decided to not change components for low pass filter and gain stage, since the given designed circuit on lab1 manual is refined and the required resistors were already provided by the lab. There is no need to order new SMD resistors and op-amps.

### B. Schematic

The first part of baseband circuit is the conversion of voltages. In Figure1, the detailed schematic is shown.

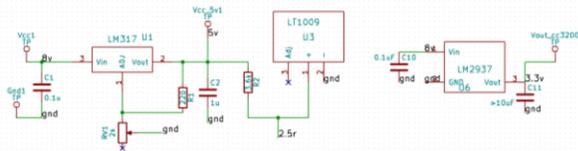


Figure1

In this part, LM317 adjustable voltage regulator is used to convert 8 volt to 5 volt. And LT1009 2.5v voltage reference is used to generate 2.5 volt as the biased voltage for low pass filter and gain stage and the reference voltage for DAC which converts digital signal from teensy to analog signal. In addition, LM2937 3.3v is used to convert 8 volt to 3.3 volt as voltage supply for CC3200. The circuit is based on the evaluated circuit shown on the data sheet.

Figure2 shows the low pass filter and gain stage design.

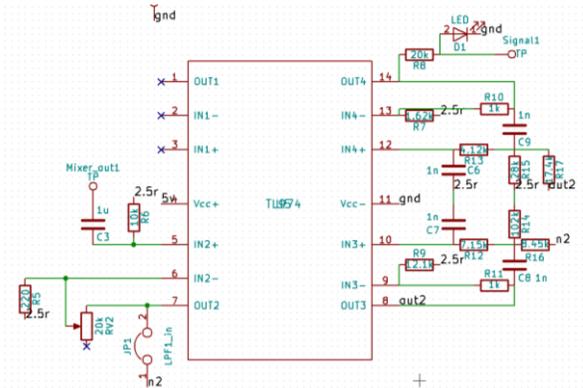


Figure2

There is no significant change on this part compared to quarter 1. The noticeable change is that the output signal is connected to an LED which prevents output voltage from exceeding 1.4 volt, the maximum input voltage for CC3200 ADC.

For Teensy and MCP4921 (DAC), the schematics are the same as quarter 1.

When design schematic of your circuit, it is important that when you built the ‘new component’ for your components in KiCad, the pin number should be label clearly; thus, when you design the footprint, the pin numbers are not mixed. In addition, for new components that you are not familiar, the best way to design a good circuit using those components is to follow or understand the evaluated circuit provided on the datasheet. Furthermore, to make the test of PCB easy and correct, it is important to set test points between gain stage and low pass filter. A jumper between Teensy voltage supply and 5 volt port is also needed to avoid burn out of Teensy.

### C. Footprint

After schematic design, we needed to create footprint for each component. The rule of thumb is that carefully review the PCB design the PCB footprint provided on the datasheet and make sure it is bottom view or top view. It is important to know the unit on the datasheet is mm or inch. In addition, assigning each pad the correct pin number is required, because later when laying out the PCB, the track net is based on the pin number on the footprint and the pin number on schematic. Make sure the two pin numbers are matched. Furthermore, using non-copper line to outline the approximate size of the components can help to define the distance between each component when doing PCB layout.

### D. Layout

At first, I outlined a fixed size (1.5inch by 2.5inch) for my baseband PCB to make sure RF and baseband PCBs have the same size. The I load netlist to the layout window. Based on netlist to place the components. Using track via can help avoid

cross of tracks. And use fill zone to fill ground is a good way to save tracks from grounding. Before, upload the PCB design, it is necessary to refill zone.

### E. Stacking

For good connection between two PCBs, we choose to use stack two PCBs together. The stacking PCB is shown in Figure3.

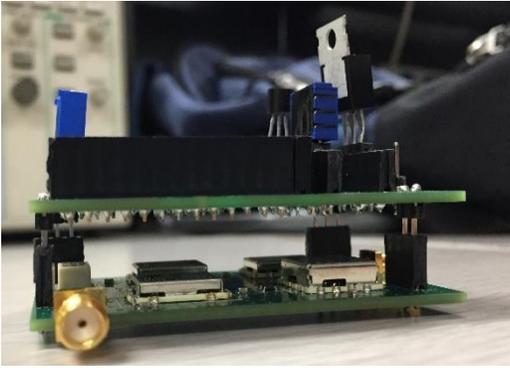


Figure3

First, I created three pins: VCO input, ground and 5 volt on the top left corner. And make sure the positions of pins are the same as the positions of pins on the RF band. Using larger grid makes the position matching easier. Then, to make sure the RF PCB can support baseband PCB, I created several ground pins at other three corners. The final PCB layout is shown in Figure4.

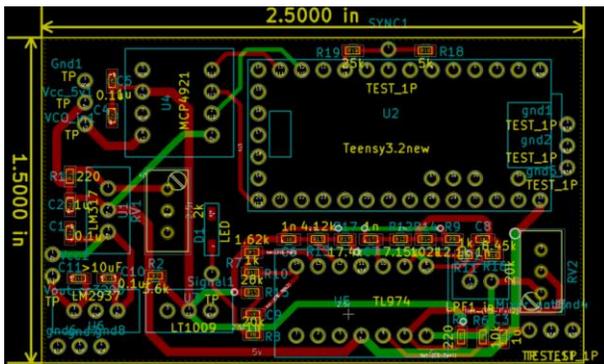


Figure4

### F. Assembling and Test

The first step to assembly the PCB is to collect all needed components and mark each component correctly. Then, based on the PCB design file, I soldered each component on its place. Personally, placing solder paste to all the SMD copper sections and then place the components is the most efficient way to assembly a PCB.

The first soldered PCB is shown in Figure5. We used normal female pin headers for Teensy, op-amp, DAC and potentiometers. And then we test this baseband PCB. It works fine. Later, when we stacked the baseband PCB on the RF PCB and tested the whole system, we had troubles on our baseband PCB. Since the potentiometers have very thin pin, the contacts between potentiometers and pin headers are loose. Also, there are loose contacts between op-amp pin headers. Therefore, we decided to re-soldered baseband PCB.

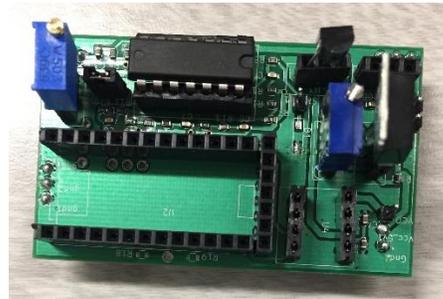


Figure5

We used “14 Position Pin Standard Circular Connector” for op-amp and “8 Position Pin Standard Circular Connector” for DAC to avoid loose contacts. Also, we directly soldered potentiometers on PCB. Figure6 shows the re-soldered PCB.



Figure6

This re-soldered PCB worked much better than the old one.

## III. CONCLUSION

PCB design is the most important part to realize our radar system and reduce its weight. To get a good baseband PCB, we should first know how the whole system works and the desired functions of your circuits. Then, based on the knowledge about you radar system, we can design an improved circuit schematic and then create a refined PCB layout with carefully made footprints. Before creating footprints or designing circuits, it is necessary to read through datasheet to know well about all the components.