Application Note:

Baseband PCB

EEC 134

Team MegaHurtz

Charlie Saechou

This note will be used for the baseband PCB of the radar system. This note will explain how to create the schematic and PCB layout of the baseband component. Though the course provides a PCB tutorial, this app note will serve as a step by step guide to help create an efficient baseband PCB.

Baseband Schematic

The baseband PCB consists of the gain stage, the low pass filter, the teensy, the voltage regulator and the DAC. The gain stage includes two amplifiers to ensure the signal has enough power at the output. When determining which components to be on the baseband PCB, an important thing to consider which components can process high frequency signals. This components will be included in the RF PCB, which will eventually be connected to the baseband PCB. Since both the baseband and RF components are a lot to test, I decided to split up both parts into two separate PCBs. This allows the group to test each board individually and debug the PCB a lot easier.

To begin the baseband schematic, organize the schematic in a way that separates all the components. Though this isn’t required, this allows the user to easily look back and find out the different connections. An example below is an easy way to keep the entire schematic organized for future use when soldering the PCB.

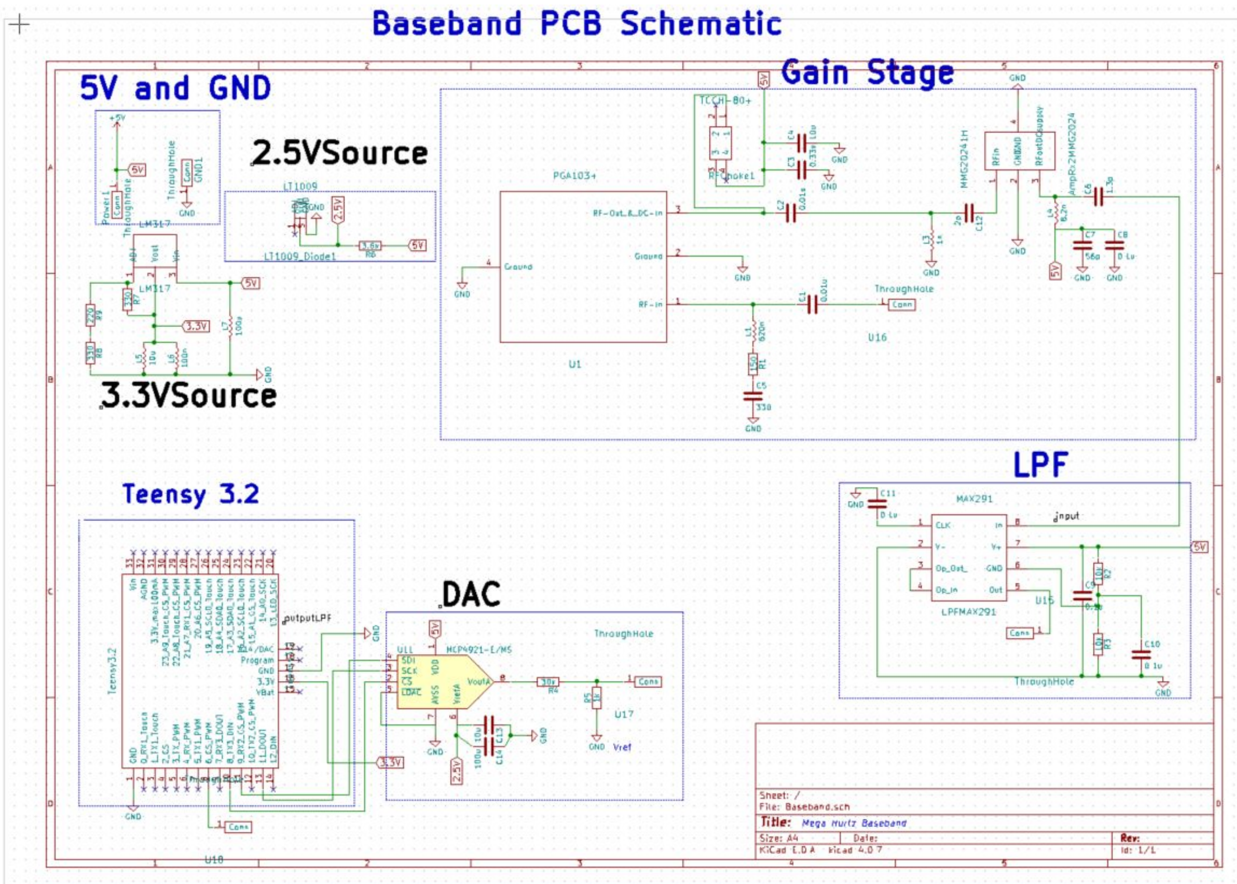
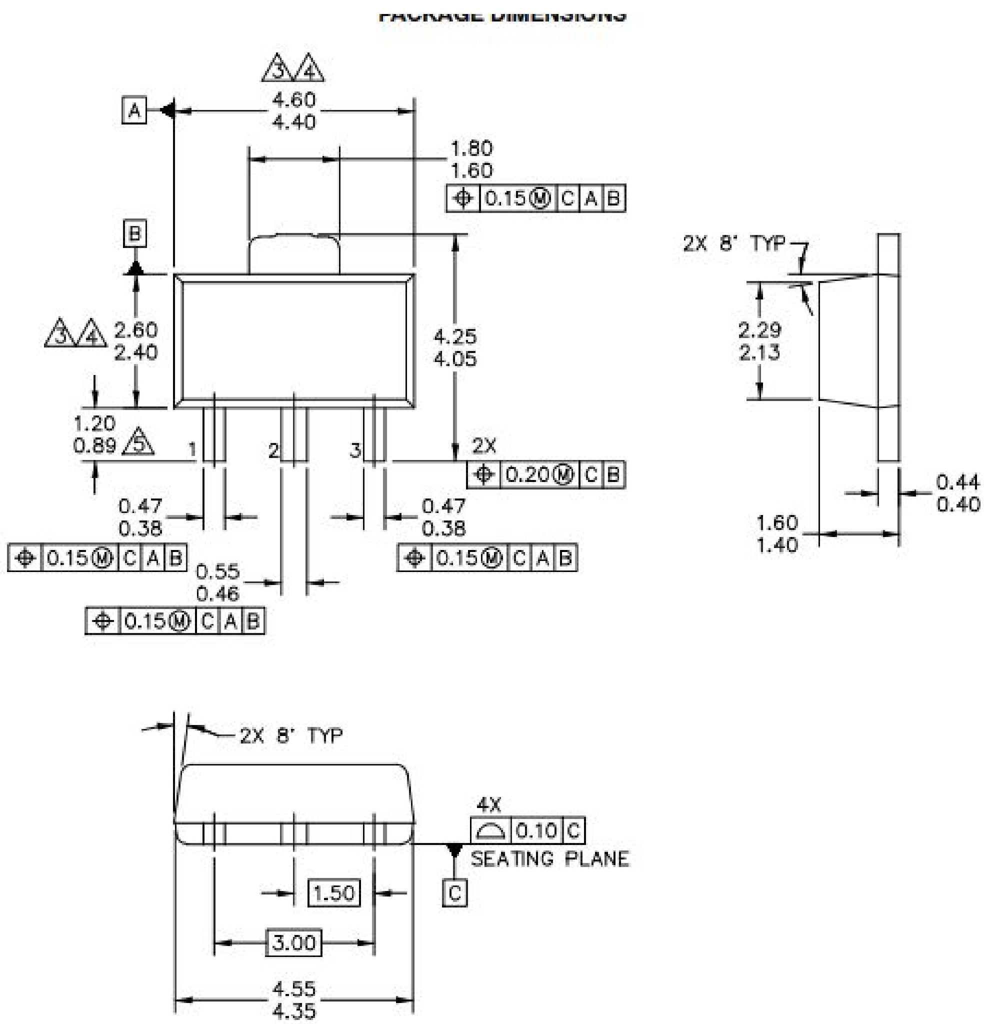


Figure 1. Baseband Schematic

When creating the schematic, be sure to look at the datasheet of each component you will be using. Included in the datasheet is a recommended circuit that the user should follow. This recommended circuit will include different resistors, capacitors, and inductors. For every component you will be using, be sure the actual component already exists in the KiCad Library. The resistors, capacitors and inductors all should be included in the library. However, the op-amps will be a different dimension.

Footprint Editor:

When designing the footprint for a new component, look at the datasheet for the desired component. Included will be a pinout that shows the dimension of each pin. This page will be used to determine the spacing and the size of the actual component. I will use the MMG20241 amplifier as an example. Looking at the datasheet below, you can see the size of each pin and the distance between each. These values will be used in “Footprint Editor” to create the MMG20241 component. When creating the component, be sure to label each pin correctly, as this will be used to connect to other components in the schematic.



© FREESCALE SEMICONDUCTOR, INC. ALL RIGHTS RESERVED.	MECHANICAL OUTLINE	PRINT VERSION NOT TO SCALE
TITLE: SOT–89A, 3 LEAD, 4.5 X 2.5 PKG, 1.5 MM PITCH	DOCUMENT NO: 98ASA00241D	REV: 0
	CASE NUMBER: 2142–01	15 JUL 2010
	STANDARD: NON–JEDEC	

Figure 2. MMG20241H Dimensions

- TIP: I recommend drawing it out on a piece of paper and mathematically determining the spacing between each. This ensures that you correctly created the right component.

Be sure to start with simpler ones provided in the PCB tutorial before moving onto more complicated footprints.

Assigning footprints to components

When creating the footprint, save the component in the current directory by exporting the file and creating a folder. Once a folder is created, click preferences on the top of the footprint editor. A screen will show up labeled as, “Add Footprint Libraries Wizard”.

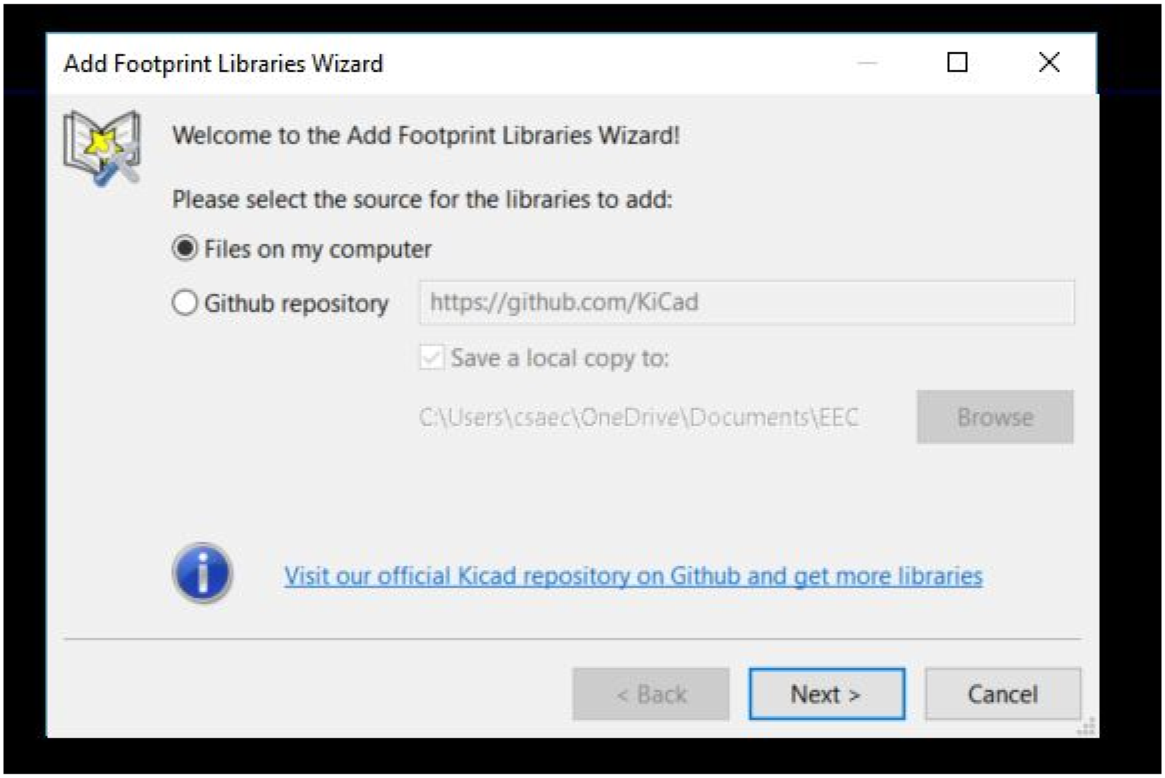


Figure 3. Footprint Library Wizard

Click next, then select the folder you created. **Be sure to have different folders for different components.** Now set the active library. This ensures that you will be able to connect this footprint to the component. After opening up “CvPCB”, click the component you wish to associate. Then, on the right column, find the correct footprint. Once found, double click that footprint so that the component is included in the middle column. Repeat this for all the components in the list as shown below.

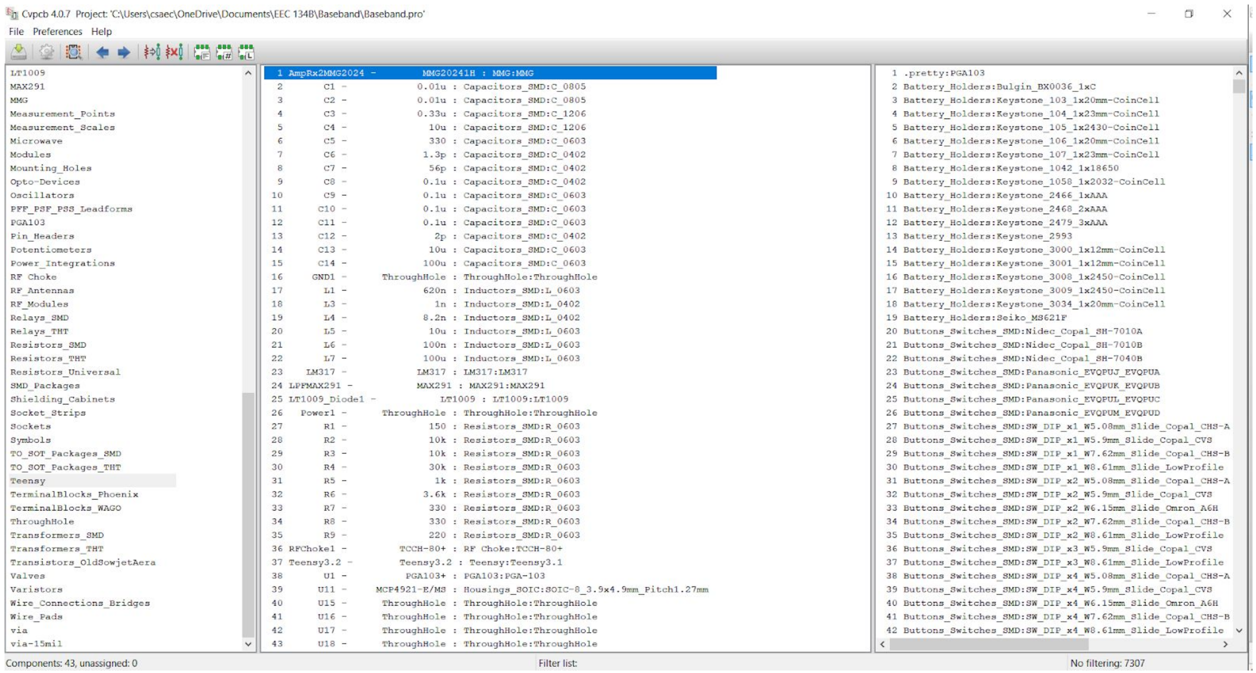


Figure 4. Associating footprints

We can now move onto the PCB design.

PCB Design

On the schematic window, generate a netlist. Once the netlist is complete, click “Run PCB new” to begin the design. Click the “read netlist” button located on the top bar. When read in, be sure no errors are present. Once the netlist is read in, the design should show all the components in one location as shown below:

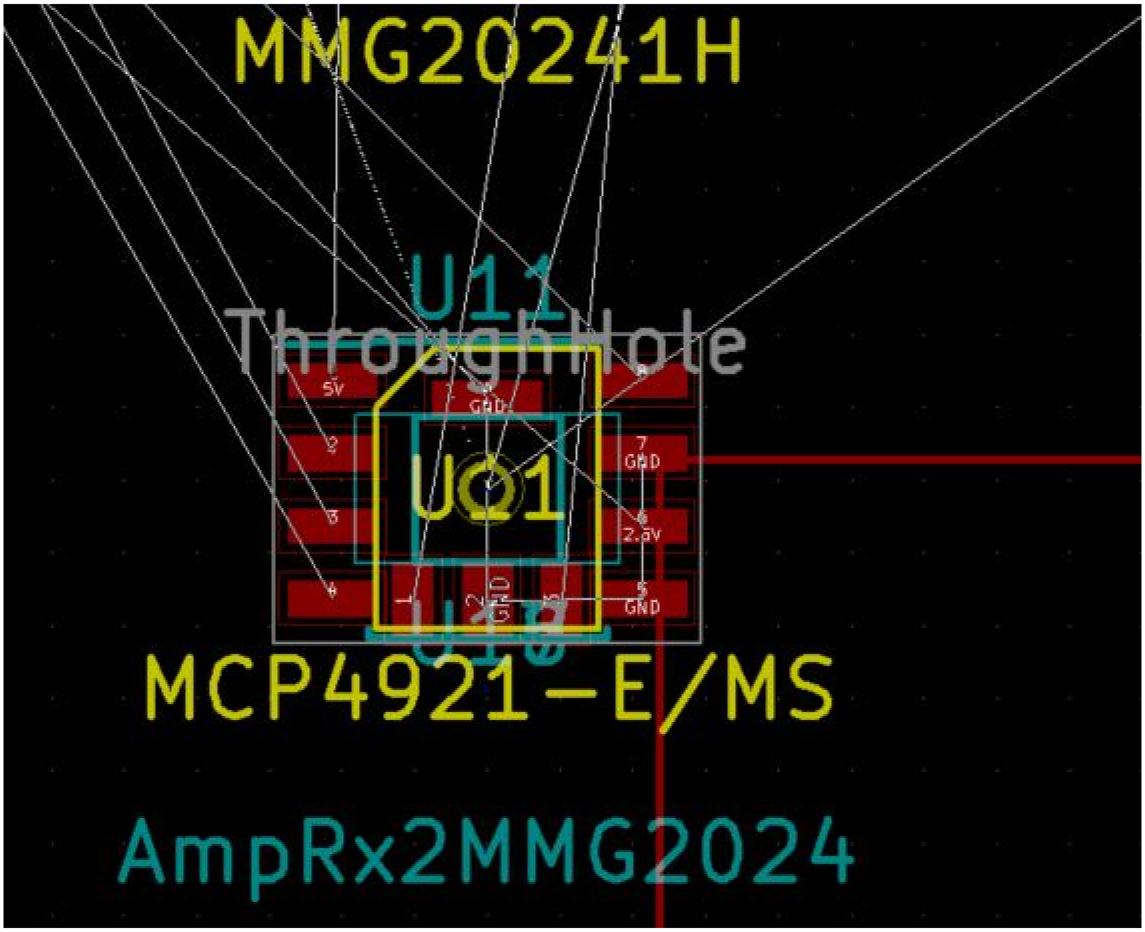


Figure 5. Rats netlist

Move the components so they're all organized. A simple way of organizing the components is to follow the schematic created previously. Move all the resistors, capacitors, and inductors so that they're in the same layout shown in the datasheet. When placing the teensy, place the teensy in the corner of the board so that the USB port will be sticking out. A complete design of the schematic is shown below.

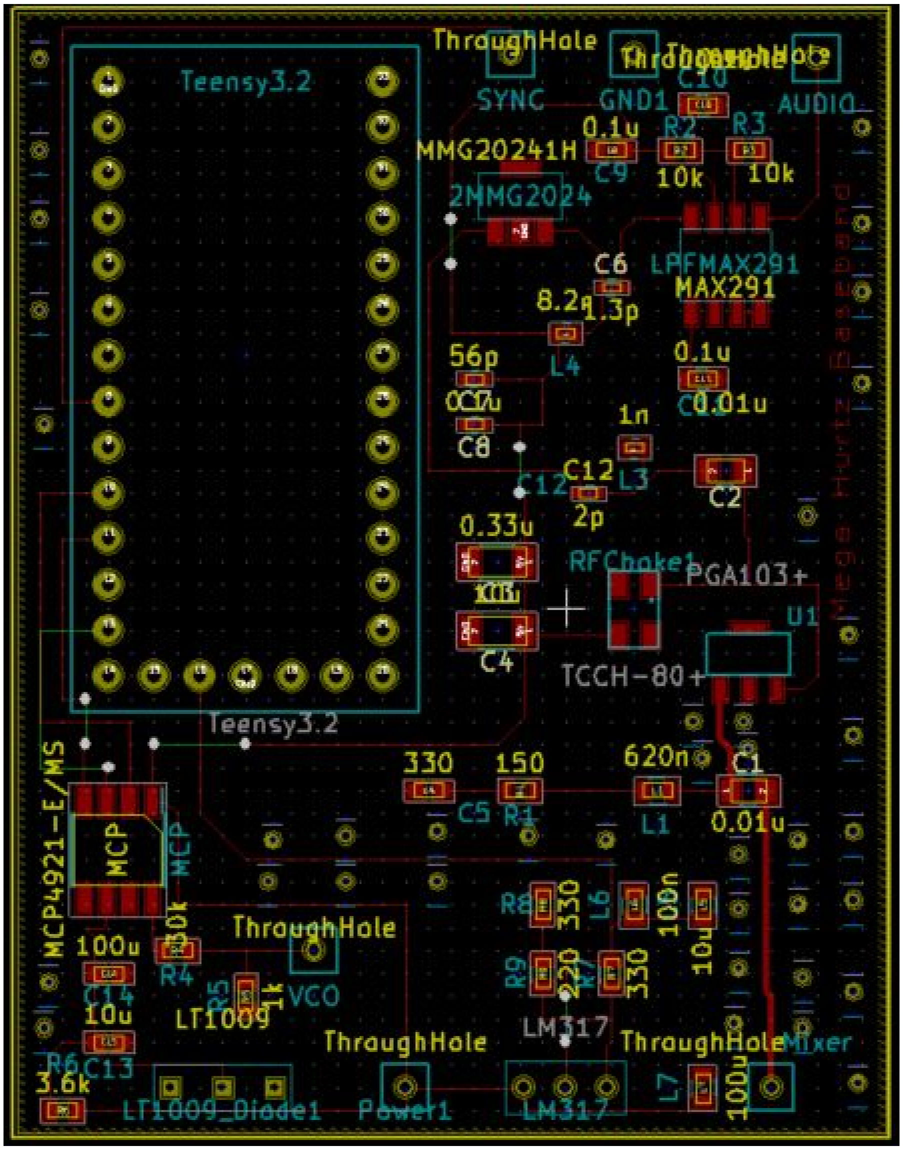


Figure 6. Complete Design of Baseband Board.

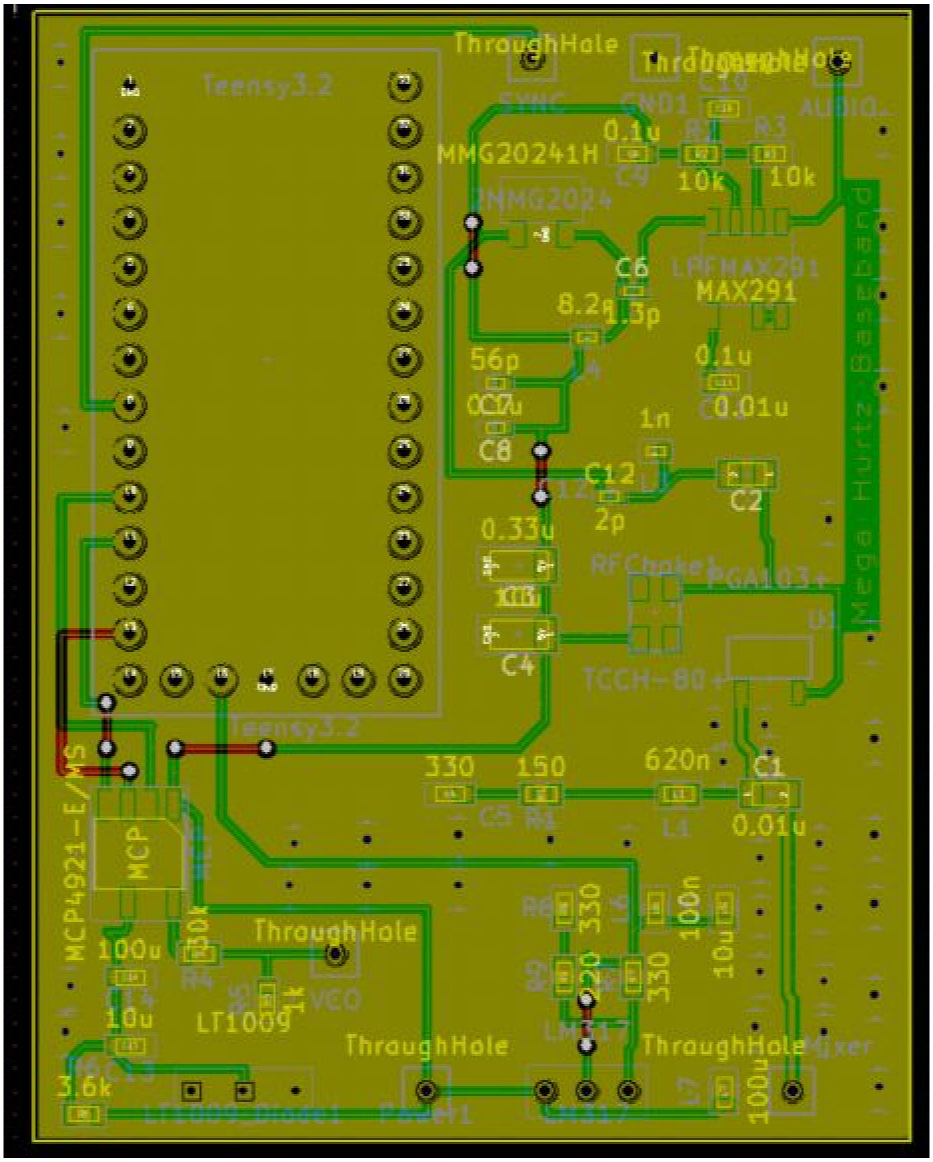


Figure 7. Complete Design of Baseband Board.