EEC 134 Application Note

BASEBAND PCB DESIGN AND IMPLEMENTATION

Name: Marzhan Bekbalanova
SID: 998230364
Team EEPower
05/04/2016

Table of Contents

Components Selection

Before you actually start the PCB design, you should start looking for components and picking them based on their performance. There are many websites where you can order components, such as Mouser Electronics and Digikey. They have a variety of components, and you can place orders easily on OPS (Online Purchasing System, a system by the UC Davis College of Engineering).

In order to minimize the cost of components, we tried to order most components from Mini Circuits E-Z samples. They have over 1000 surface mount free samples.

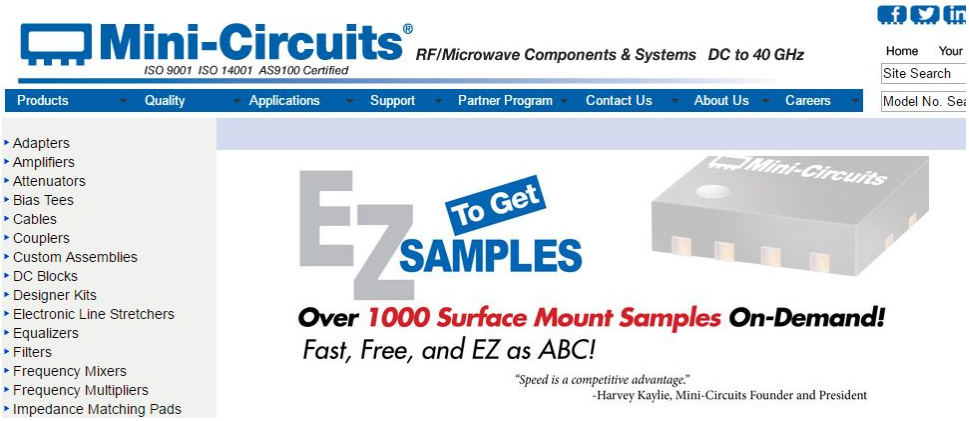

Figure 1: Mini-circuits E-Z samples page

Also if you are looking for certain characteristics, such as frequency range or package size, you can narrow down the options with the filter below: Make sure the type of the component is actually surface mount for the PCB design.

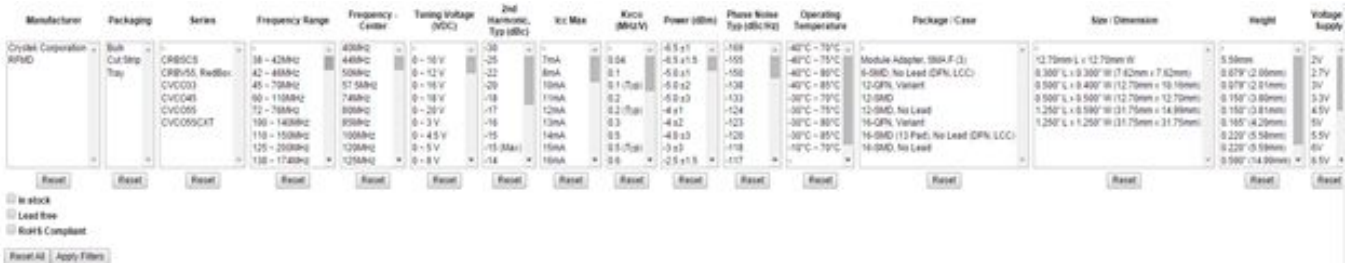

Figure 2: Filtering components

Once you click on the component to see more details, you will be able to find a link to the datasheet of the component.

Data Sheet	S-Parameters and Performance Measurements			Case Style	T&R	PCB Layout	Eval Board *	Environmental Ratings
Data Sheet	View Data Download Excel File	View Graphs	Download S-Parameters	DQ849	F104	98-PL-328	TB-547-2+ Gerber File	ENV08T1

Figure 2: Finding Datasheet

Datasheets are important during PCB layout design as you will need the dimensions of the pads when you design the footprints of the components. Although some websites do provide already pre-made footprints that you can download, it is advised that you design it yourself as we found some discrepancies and mistakes in the footprints that were on the website. But I will talk about footprints in more details in the next sections.

Schematics Design

During the PCB design process, first we have to decide whether to go with a 2 layer board or 4 layer board. Although 4 layer boards have many flexibilities and functions, our group decided to use a 2 layer board as it is less expensive to fabricate, and for our purposes, 2 layer boards were sufficient enough.

Next question that arises during the PCB design is what software to use. I used Eagle Cad for my PCB designs because it was a free software, and I already had experience using it.

Our group to decided to have 2 separate boards, a baseband board and RF board, and stack the boards on top of each other. We used male and female pin headers to keep them attached and stable.

In order to incorporate components on the PCB, we first need to design footprints. The footprints are the designs of the actual pads that components later will be placed onto. On Eagle, there is library that already had footprints of some common components, like resistors and capacitors. However, in case they are not in the library, you might have to design your own footprints based on the given dimensions from datasheet. Also make sure to pay attention to the pin assignment and carefully follow the order. It is a crucial step. (Look at Figure 3)

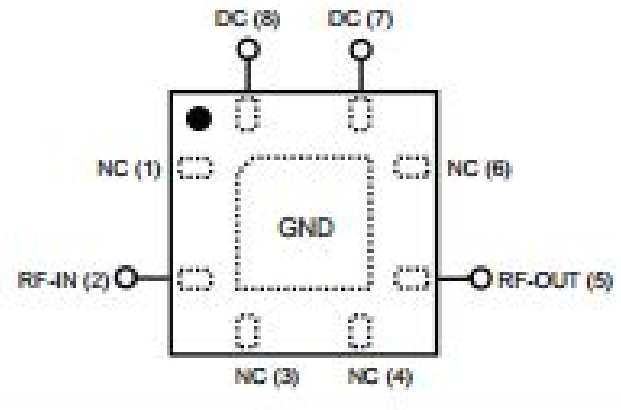

Figure 3: Example of a datasheet pin assignment

Once we have footprints ready (as in Figure 4), we can start designing the PCB schematics.

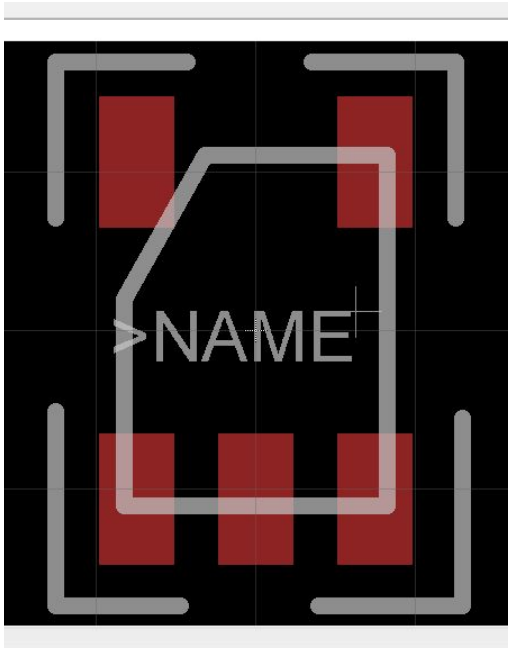

Figure 4: Footprint Example

When we do schematics on Eagle, using pin header is very handy as it is not only helpful for debugging purposes, but also minimizes the long wires between pins. I used label pins for some long wires such as 5V and 3V that need to be connected to many other pins. Instead of having long wires, it looks cleaner and less likely to have wiring mistakes.

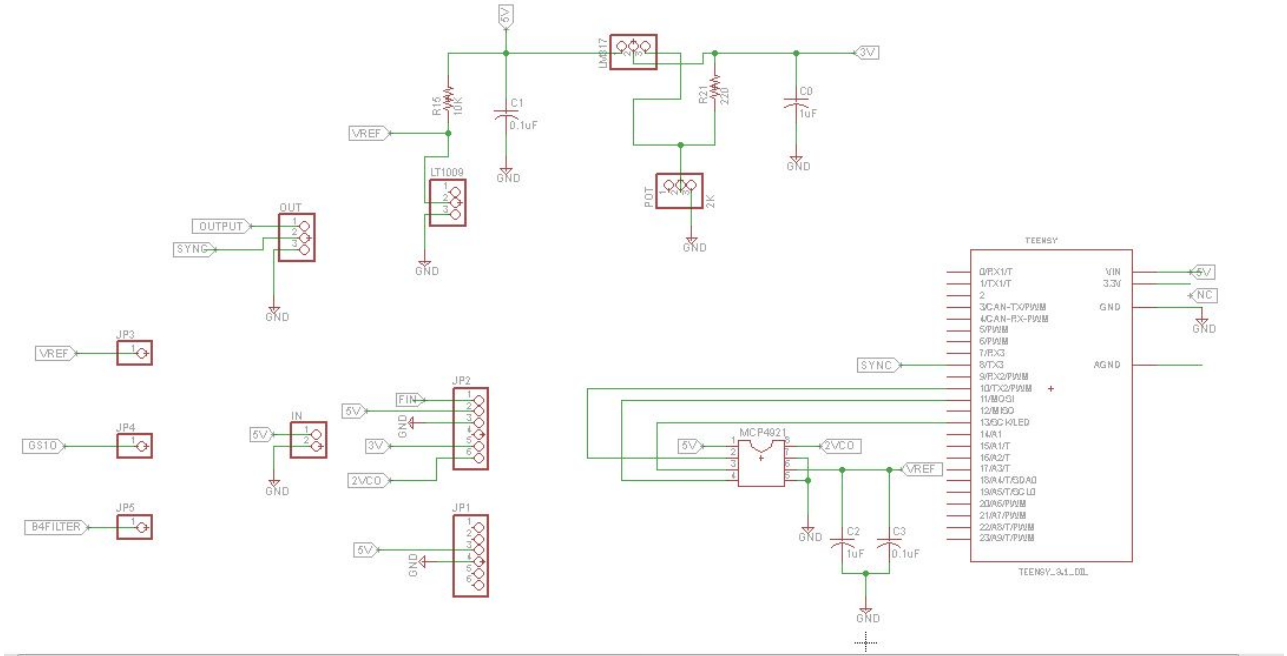

Figure 5: Baseband PCB Schematics 1 on Eagle

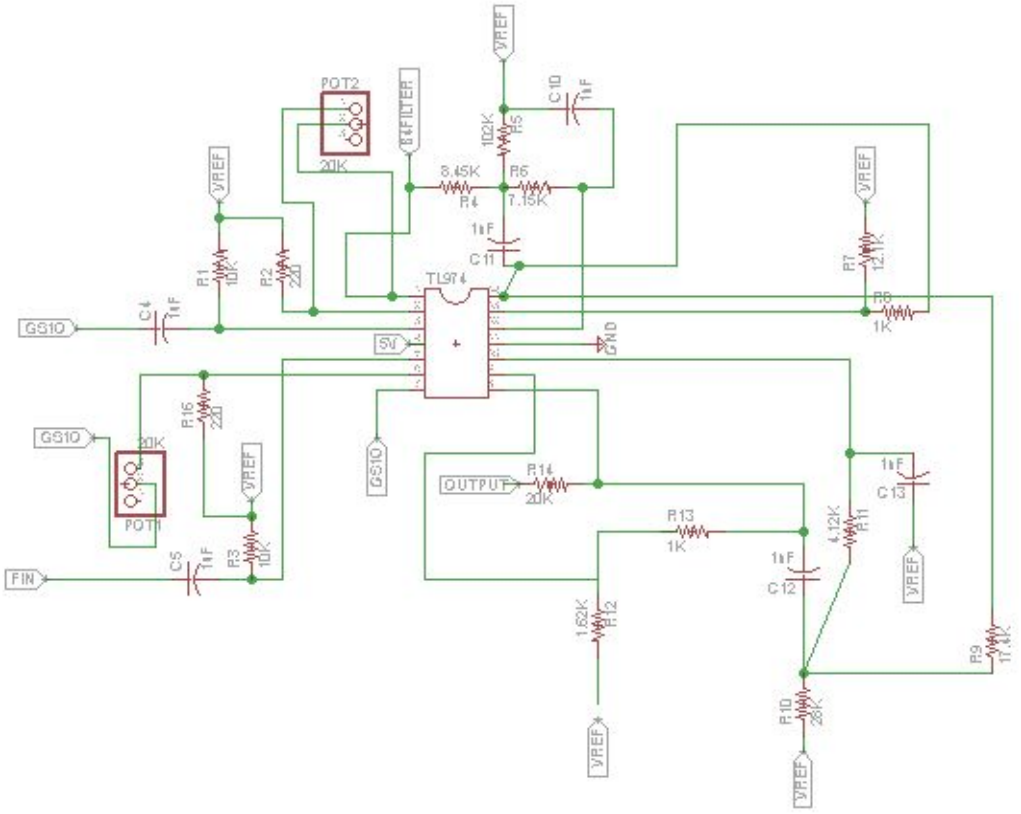

Figure 6: Baseband PCB schematics 2 on Eagle

I divided the baseband PCB schematics into 2 figures here to see the connections more clearly. But they are in the same schematics file.

PCB Layout Design

When designing a PCB there are a few rules that are recommended to follow

- For 3V and 5V lines, use thicker traces as there will be more current flow
- Try to make angles of traces more than 90 degrees to avoid build ups during the manufacture
- Use many ground vias around the board to make sure the ground planes are connected properly
- For the baseband board, it is recommended to have test pins for later debugging purposes

Later after designing the layout, I printed out my design for the peer review to see if I missed anything.

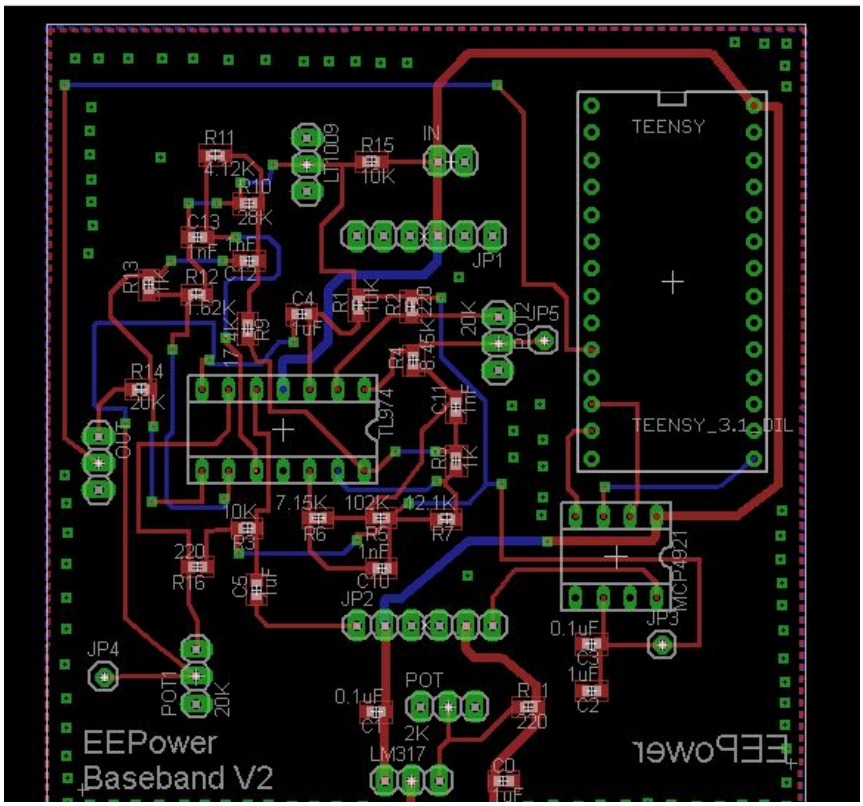

Figure 7: PCB layout for the baseband PCB

PCB Fabrication

Once the layout is complete, we can proceed to the fabrication. Our senior design group sent the PCB design to the Bay Area Circuits and Oshpark.

For the Bay Area Circuits, you have to upload Gerber files to the website. Based on the gerber files, the website can detect any possible errors and overlaps and will not let you submit the PCB design.You will receive instant DFM files that shows the locations of the errors. You will have to fix those errors. For instance, if you did not follow the copper to copper minimum width requirements, you will receive a warning as illustrated in the figures 8 and 9. (See Figure 8 and Figure 9)

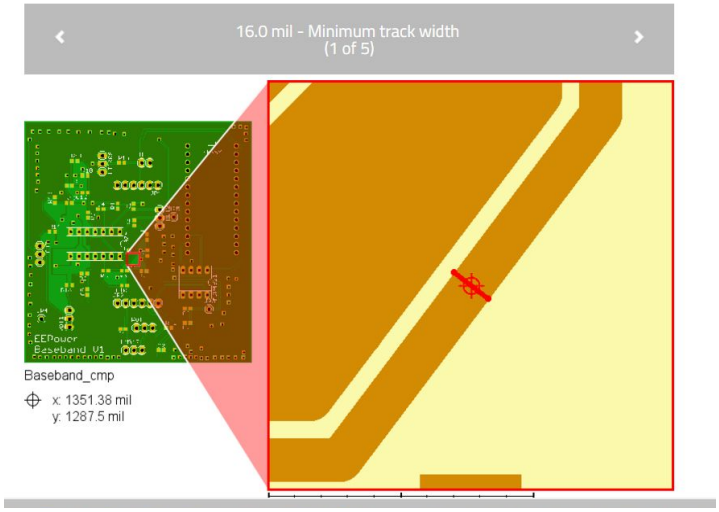

Figure 8: Instant DFM file

DFM Checks:

	Min. Distance
copper to board edge:	-
trace to plated hole:	13.0 mil
trace to non-plated hole:	-
copper to copper:	5.9 mil
copper ring:	7.0 mil
track width:	16.0 mil

Figure 9: DFM scheck results

PCB Assembly

Once we receive the PCB boards, they are ready for soldering. For surface mount components, it is highly advised to use a hot plate to melt solder on the pads. Due to the small sizes of the components (0603), this method is the most efficient and practical.

Here are some tips for using a hot plate:

- It is good to have a magnifying glass to spot even a tiny short circuit
- Try using a temperature no higher than 200 degrees as higher temperatures might damage some components. You can look at each component's datasheet to verify what temperature is safe to use.
- For the baseband board try soldering components in stages. For instance, first do the voltage regulators, then gain stage and then finally the filter. This way it is easier to debug.