EEC 193 APPLICATION NOTE

PCB Design

Senior Design Radar Project

David Rangel Alarcon

This application goes over how to design a PCB, starting with general considerations and tips on what to look out for and how to stay organized when creating the PCB.

Table of Contents

***The System*** *1*

*Know the System 1*

*Components 2*

*Off-Board Connections 2*

***Layout Considerations*** *3*

*General Layout 3*

*RF Layout 4*

***Fabrication Houses*** *6*

*Houses 6*

*Gerber Files 6*

***References*** *7*

The System

First things first, before you create the PCB you should try to understand the system, the components going on to the board, and the off-board connections.

*Know the System*

Understanding the system is a must when designing the PCB. To start you need a block diagram of the system, from this you can create the schematic.

Prototyping before building the PCB is also very necessary. Creating the board takes time and money, and you should know for certain if the system works beforehand, because chances are the system will not work as well on a PCB as it does on a breadboard.

The following is a list of some things you need to know about the system in order to successfully create the PCB:

* Signal Flow
* Signals going on/off the board
* Power Draw

*Components*

Having an excel list of components going on to the board helps organization, and later people will be asking to see this list for system details. On this list, you can include:

* Operating Voltage
* Operating Current Draw
* Max Power Input
* Gain/Loss
* Operating Frequency
* IP3/Noise Figure/1dB Compression Point
* Price
* Link to Purchase Component

Keep the data sheets of all the components in a folder to stay organized. You will likely be referring back to these when you are creating the component layouts on your CAD tool, when you are trying to connect the component pins properly, and again when you are verifying your system.

*Off-Board Connections*

Knowing the signals that you want to put on the board and the signals coming off the board is essential to planning a good layout.

For instance, if you are creating a Shield for a Microcontroller, you will need a pin map of the Microcontroller to put onto your board and plan where the components go. If you are feeding the signals to antennas you will need to plan the layout for a coax connection.

Defining these connections to the board will be the best way to start when you layout the PCB.

Layout Considerations

*General Layout*

Some general layout tips to consider when laying out a board:

* Find out the limitations of the fab house and follow them like gospel. This will tell you minimum trace size, spacing between traces, via hole size, etc. If you can leave more space between traces, that would be even better, because no fab house is perfect and there is a small chance of errors in the board.
* Try multiple configurations to fit the components the right way since you might not get the optimal layout the first time.
* Avoid auto routing because that will not give the best thought out tracing.
* Keep trace widths thick, especially for lines that have more current. Smaller lines will have a larger resistance to them, so always consider how much current will be going through the trace to see how thick it should be. Power supply lines should be the thickest.
* Avoid 90-degree turns; since currents will be disrupted by the change in direction, 90-degree turns will add resistance, which is especially bad for higher current lines. At 3 way junctions you should try to use a Y junction.
* Use different ground planes for high frequency and low frequency sections, and analog and digital sections. The ground planes need to be connected at some point to keep the same reference point, but having them mostly disconnected helps reduce the signals leaking into each other and adding noise to the system.
* Add bypass capacitors between the DC power supply and the ground plane to keeping AC signals off of the DC supply. Several should be used, and should be placed all over the board wherever components are being fed power.
* Add test points to important traces, except in RF lines since that will disrupt the microstrip line properties. For lower frequency sections though adding via holes around 1mm diameter as test points will help verify the system is working properly, and if need be you can use jumper wires to go between the PCB and a breadboard for testing. Assume every part can fail and plan a way of testing the stage and working around each stage wherever possible.

Bruce Carter’s paper on PCB design techniques for TI is also a great resource with many of the same tips and more. A link to the paper can be found in the resources at the end of this application note.

*RF Layout*

When you design PCB’s for greater than 300 MHz, and especially once you are operating in the GHz range, you should start considering RF effects in your circuit. If designed improperly you can get power reflections from impedance mismatches, coupling between lines, and essentially a lot lost power and added noise.

The reason RF lines are different from normal traces is because at the higher frequency, these short lines on the PCB may look like transmission lines, and thus need to be treated as so. Also, at higher frequencies there is more chance of radiation, and the signal will radiate from the trace to other traces and out into the air.

To draw these traces, it may be useful to use polygons on the PCB instead of using the normal trace. In EAGLE, this means giving the polygon the same name as the node you are using it for, setting the polygon to keep orphans, and setting it on high isolation priority (so it doesn’t get distorted from being around other traces).

In order to properly design the RF PCB, some special considerations should be taken into account:

* Try to keep all the RF components and lines 50 Ohms at your center frequency. In order to determine the width of a 50 Ohm line is on your board, you can use a microstrip line calculator online, or the ADS software from the 132 series, which other students or the professor might have access to.
* Changing trace widths on the board will change the line impedance, and can cause power reflection. In order to avoid this, you can make a simple impedance transformer by tapering the line out with polygons.
* Keep traces with different signals far away from each other, to keep signals isolated. Especially do not run two lines with different signals parallel to each other and close to each other, there will definitely be coupling in that case. Keeping traces perpendicular to one another helps in keeping them isolated.
* Do not disrupt the ground plane under the RF traces, this will change the impedance of the line and cause reflections from the mismatch. This also means you need to avoid running RF traces underneath each other, all the RF should stay on the same plane.
* Using RF components, the data sheet usually has a suggested layout including pads and a flooded ground, meaning there is a large ground polygon around the component. Consider using these and a flooded ground to increase the size of the ground.
* Use via holes generously to break up your ground plane, this is necessary to avoid RF signals from resonating on your ground plane and radiating out or generating noise. It may do so anyways, but vias will help reduce the effect.
* Try to keep lines between components very short, as this will minimize the RF effects and make it easier to design the PCB. If the pads are close and different sizes, consider just using a tapered line to connect those pads.

Iulian Rosu has a geat site linked in the resources on RF Basics, Antennas, PCB software, etc. He’s compiled great resources for RF microstrip designs, something you can really come to appreciate after the 132 series and this senior design project. There is a link in the resources section to one of Iulian’s resources written by Rick Hartley on RF/Microwave PCB design. In there you can find simple microstrip designs for power splitters, filters, direction couplers, capacitors and inductors. A lot may be unable to be used in this project, but on page 15 is some useful layout techniques.

Fabrication Houses

Although the fab house is the last step for your PCB, you should think of where your fabricating before hand so you know the minimum trace widths and minimum spacing between traces when you design your PCB. In general, 6mil (1/1000ths of an inch) is the minimum trace width and distance, while the minimum hole size is around 15mil.

*Houses*

Two possible fab houses are Osh Park and Bay Area Circuits. Osh park has 2 layer boards for 5 dollars per square inch and 4 layer boards for 10 dollars per square inch. 2 layer boards have a turn around time of 2 weeks, while 4 layers are 2-4 weeks. Bay area circuits has multiple different options that are generally more expensive, but they have 30-dollar student special that may be a better option than Osh Park if your board is bigger than 6 square inches. Bay Area also offers slightly tighter specs for the 30-dollar student special. They also have other more costly options, such as different dielectrics, board coloring, and being able to submit more than one design.

*Gerber Files*

To decide what kind of board you need for your circuit, consider the differences between 2 layer and 4 layer boards. 4 layer boards have a smaller space between layers, and usually have a better dielectric, making them better for RF circuits. 2 layer boards are still usable for up to 3GHz, but you can expect more loss and noise at the higher frequencies. Still, if you can fit a two layer board, chances are they will have a faster turn around time and be cheaper, meaning you can fit in more iterations of the project in the short time available.

Once you have the fab house decided and are ready to send in the PCB, you will need to convert your CAD files to gerber files. Most fab houses will take the RS-274X standard gerber format. This file format contains a file for the top and bottom copper layer, the top and bottom silk screen, the via drill locations, and other mask layers that you will likely want to include.

In Eagle, generating these files is as simple as opening the CAM processor and running a CAM job, which you can download from Osh Park or Spark Fun. The link for the Osh park CAM job is posted in the references, this creates gerber files that both Osh park and Bay Area Circuits will accept. Once the gerber files are generated, you can put them into a folder and zip the folder, this zipped folder will be what you give to the fab house, and you can test them online with free gerber file viewers.

References

Blum, Jeremy – Eagle Tutorial

<https://www.youtube.com/watch?v=1AXwjZoyNno>

(A good way to get started with Eagle is following this man’s tutorial)

Carter, Bruce. Texas Instruments – Circuit Board Layout Techniques <http://www.ti.com/lit/ml/sloa089/sloa089.pdf>

(More tips and good literature on PCB design considerations)

Hartley, Rick – RF /Microwave PC Board Design and Layout <http://www.qsl.net/va3iul/Files/RF-Microwave_PCB_Design_and_Layout.pdf>

(RF PCB design, including some basic circuits and tips on page 15-20)

kd7nn. Instructables – How to make a custom library part in Eagle CAD tool <http://www.instructables.com/id/How-to-make-a-custom-library-part-in-Eagle-CAD-too/>

(Useful tutorial on creating components and libraries in EAGLE)

Osh Park – CAM job for Eagle CAD

<http://support.oshpark.com/support/articles/106997-cam-job-for-eagle>

(Eagle Cam job, will generate gerber files used for Osh park and Bay Area Circuits)

Rosu, Iulian – RF Technical Articles

<http://www.qsl.net/va3iul/>

(A good source for more RF knowledge, including microstrip designs, antenna’s, noise figure, etc.)