5

Layout Design

5.1 General

There are a huge number of things to remember, consider, and check before sending a PCB out for fabrication. However, the most important step before sending out your PCB is to have it reviewed by other people. Try to involve people from all across the production process: other electrical engineers, production engineers, mechanical engineers, test engineers, assemblers, etc. It's best to give all of your reviewers a day or two to look things over before the formal design review meeting. Freeze the design after you distribute your schematics and layout to be reviewed, or else you may change something that could end up slipping through without being reviewed.

I like to tell everyone to not to review together or share what they've found until the design review meeting. The more independent the reviews, the more likely you are to catch everything. For the same reason, it's dangerous to divide the design up into chunks and have each person look at a different piece. You want as much redundant coverage as possible. When I review designs, I like to print them out on a piece of paper, go to a quiet room by myself with a stack of datasheets for every part used in the design, and work through the whole thing again. It's still useful to use your CAD software to inspect layout features that may be difficult to see on a print out, but for me, looking at things with zero distraction helps me pay attention to detail.

Having engineers from different fields looking over your design will help you catch issues that you may have missed from your perspective, but will be apparent to people in other disciplines. Production engineers and assemblers in particular often have a tremendous amount of practical knowledge from experience that can dramatically improve your DFM.

Good layout is driven by good placement. Good routing should naturally fall out of your placement choices. Beginner designers will often place parts in a fairly haphazard way and spend the majority of layout time on routing traces. Instead, you should spend more time thinking about where to place parts, what rotation they should have, which side of the board they should go on, etc.

A good design is maintainable and repairable. Even though your pick and place machine can place parts with microscopic precision, think about how hard it will be to replace components by hand if something needs to be fixed. How good do your tolerances need to be? Can you fit a soldering iron down in there? How many screws do you have to remove? Do wires need to be desoldered to get at anything? If you assemble your engineering prototypes by hand, you will quickly discover any shortcomings in maintainability and repairability. Sometimes it's impractical to assemble your engineering prototypes by hand, or you'll need a technician to do it. Talk to your technician. Ask them what's frustrating to repair or rework so you can incorporate their feedback into your next revision. In general, be careful about placing components too close together or in such a way that they will interfere with tools that need to be used on the PCB.

There are also mechanical placement requirements to think about. Of course, there's the obvious examples like component height, connector overhang, and mounting and screw holes.

There are also more subtle things, like how far the leads on the bottom of through hole parts stick out. One good way to detect and avoid these problems is to use a CAD package with 3D modeling support. Most electrical CAD programs support 3D component models and even allow you to export to a mechanical CAD program (like SolidWorks or Catia) so your mechanical engineer or industrial designer can do a fit check before fabrication. Using this technique requires accurate and complete 3D models of all components, but many manufacturers provide 3D models of their components you can download for free.

One potential trip up for new designers concerns the units commonly used in PCB layout. When someone says "mils", they mean 0.001 inches (one thousandth of an inch). If you've ever worked in a machine shop, you may have heard this unit referred to as a "thou", short for thousandth. The mistake that some people make is that they think mil is short for millimeter. So in the world of electronics design, it's important to remember never to shorten the word millimeter to mil or you will really confuse people. The mechanical drawings in datasheets for the component are almost always in mils, millimeters, or inches. A good drawing should tell you what the units are, so make sure to pay attention.

If your design requires any traces have a particular impedance, tell your PCB fabricator to use impedance control on those traces. You'll first calculate the correct transmission line dimensions using a software tool like LineCalc (made by Keysight) or one of many free online impedance calculators. However, the fabrication process will have a small variance on the width, plating thickness, and spacing. Specifying impedance control will tell the manufacturer to take those variances into account and precisely hit a target impedance. Request a test coupon so you can verify that the impedance is correct.

Make sure you understand the manufacturing abilities of your PCB fabricator and design with those constraints in mind. Manufacturers are able to fabricate extremely small trace widths, complex via structures, and other exotic features. However, it will be very expensive. If you know what standard manufacturing specifications your PCB manufacturer offers, you can get your boards back at a lower cost and in less time. For example, say you design a trace that is 9 mils wide, but it would also be acceptable if it was 10 mils wide. If your manufacturer has a standard or low cost tier that requires a minimum 10 mil trace size, your design will be more expensive than necessary because of the difference of a single mil, which isn't important for your design anyway.

PCB fabricators enforce design rules so that they can be confident that they're able to produce your board to meet their quality, yield, and time standards. It's really important to set up the design rules in your CAD software to mirror those of the fabricator you'll be using. These design rules usually appear in a table on the manufacturers website. If your manufacturer has a DRC or design constraint file for the CAD package you're using, download it and use it. This will automatically import the minimum feature sizes and spacing requirements for the particular process you'll be using. Some manufacturers even have dedicated tools you can use to check your gerbers before you submit them. For example, Advanced Circuits has a website called FreeDFM.com that allows you to upload your gerbers and automatically look for and identify errors before you submit them for fabrication. While FreeDFM.com is only set up to check against Advanced Circuit's rules, those rules are fairly standard and it can still be useful for finding mistakes even if you don't ultimately fabricate with Advanced Circuits.

If you'll be designing a board with high speed or RF requirements, look at the available stackups and materials of your board house before designing the impedance controlled structures. Knowing what stackups are available can significantly impact your cost. Exotic or custom stackups are often much more expensive than the standard stackups a board house may use. They often also have much longer lead times. In many cases,