STILWELL BAKER

5 Basic Steps to a Successful PCB Layout

Often times PCB layout becomes the sponge that absorbs all the schedule slips in the R&D process. But a great design, with a poorly executed layout, can be lethal to your product development schedule and budget. We know that we *should* take the time to review a design prior to layout and resolve any issues early. The problem is finding time to do it. So no matter how compressed your schedule, pay attention to these 5 basic steps for a successful PCB layout.

"Why is there never time to do it right the first time, yet always time to do it over?"
– an Engineer's common lament

Step 1: Know your library

Incorrect part libraries will certainly get your layout off to a bad start. Be particularly mindful of the following:

Verify part symbols & footprints. Part symbols in the schematic and part footprints in layout need to be verified and checked somewhere in your library process. Without this check, an incorrect part will find its way on to your designs before it is actually assembled and tested. These types of part errors can be as simple as a not having the correct polarity to having the schematic symbol referencing the wrong footprint. Some errors will be easy to see when the part is being assembled, while others won't appear until after some hair pulling and a few hours on the lab bench.

Use industry-standard parts. Try to stick with industry standard part package types when possible. This will help the designer, fabricator, and the assembler. Figuring out how to create a footprint for something special, and then trying to get it fabricated and assembled, may not be the best idea for designs on the fast track to market or near the production phase of your R&D.

Involve your fabricator/assembler. If you need to get creative and use a new part type or technology, it's advantageous to include the fabricator and assembler along with your layout resource early in the design process.

Step 2: Mechanical constraints do matter

Trace density, layer count, and physical size all matter, but the parts still need to fit on the board. Make sure to consider these mechanical constraints.

Connectors, connectors. Connector locations and orientations can quickly limit placement and routing. This is a big source of mistakes! Don't forget about the connections at the other end of the cable—this also needs to be verified.

Finalize the mechanical specification. Finalize the mechanical specification before moving into placement of a design. Changes to mechanical specifications during placement and routing can significantly increase the amount of time in layout.

Consider mechanical interfaces. Be sure to budget extra space for mechanical interfaces that plug into or interact with the board.

Don't forget the z axis. You've checked your part spacing and connector locations in the x and y direction, now it is time to make sure that you have accounted for component heights and clearances in the z axis. Check for external mechanical interferences, such as the curvature of the case that the board is mounting in, or the motorized arm that moves along the adjacent interface. Most times an IDF or DXF transfer to check your design in the larger mechanical world is advisable.

info@stilwellbaker.com 503-885-1231

Step 3: Be careful with layer count

Additional layers can speed routing, but if your end product will require fewer layers, use caution. A working 6-layer prototype may not easily translate into a 4-layer design. You may be in a situation where you need to quickly get firmware engineers a development PCA. Extra connectors and additional layers make it easier and faster to deliver the PCA. But, if the same electronic design needs fewer layers for production, you may not learn what is needed to get to production.

Look carefully at connector and HDI escape routes. Have you leveraged layers that will not be there in the reduced layer count design? If you have, is there space on a different layer? Will you have to modify planes to accommodate these traces?

Give routing priority to critical nets. On high speed designs, the layer reductions can have dramatic effects on return paths. Giving routing priority to critical nets, in any layer count, may save added length, disrupted grounds, and plane splits that compromise the design for production.

Step 4: Know your PCB fabricator

Every PCB fabricator has a preferred design guideline. Make sure you have a copy and verify the following:

Verify the fabricator's technology expertise. Be sure your fabricator is capable of easily and cost-effectively producing the type (technology) of design that you are creating. Are there materials that are new to them? Are the drill sizes so small that only one PCB can be drilled at a time? Is the PCB size slightly larger than the standard working panel size?

Work with the fabricator to build a stackup. This is especially important on many of the high speed designs. It's easy to download a free impedance calculator and punch in what you think are the correct numbers. However, you should at least call your fabricator and verify what impedance they will guarantee with the materials they're going to use. It's often easier to have them work out a stackup for you.

Know their design capabilities. Get the fabricator's range of capabilities and design to their standards where possible. Find out what "advanced" means and how much it will cost in schedule and budget.

Consider R&D vs. volume fabricators. If you are creating a design for R&D at a prototype fabricator, and plan to send the completed R&D design to a different volume fabricator, be sure that the R&D design has been created to accommodate both fabricators' guidelines.

Step 5: Know your PCB assembler

Why design a board that's hard to assemble?

Get the assembler's DfX guidelines. A working prototype that's difficult to build will need modifications for volume production. What impact do those modifications have on electrical performance, compliance, safety, or EMI? Get the DfX design guidelines from your assembler of choice, and integrate them into your design during the layout process.

Involve the assembler early in the layout of a design. Assembly can provide valuable input on panelization, component location and rotation, and the soldering process (i.e. reflow, wave, selective wave, etc). Involving the assembler early in your design can help save you time and money during this process.

Using a different assembler for production? If you are creating a design that will be sent to a different assembler for production level assembly, make sure that you know what their DfX capabilities are, and you are accounting for that in your prototype design.

info@stilwellbaker.com 503-885-1231