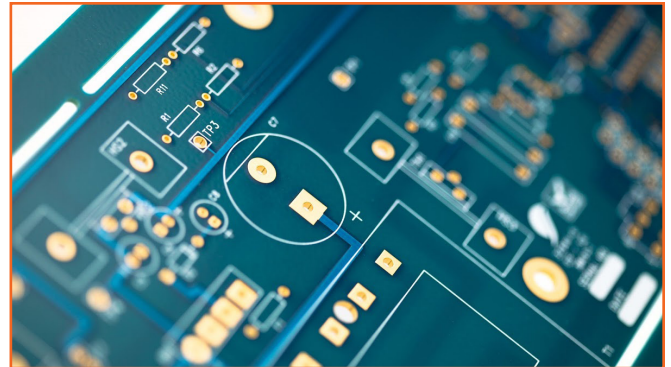


Solving Problems Before They Occur: Manufacturability for Smarties



How to avoid common manufacturing pitfalls that can cause delay, create cost overruns, and impact quality.

Despite their best efforts, even experienced makers run into trouble and need help with the transition from PCB design to manufacture. We see manufacturability issues every day—issues that can impact board performance, hinder integration with the final product, or even render the PCB nonfunctional.

While pitfalls can often be avoided by adhering to the measure twice, cut once rule, potential trouble can be difficult to detect even if you're an expert. This paper offers strategies for ensuring your design will result in a smooth manufacturing process and produce quality boards.

Designing for manufacturability is
still critically important.

The PCB world continues to be a dynamic place with broader manufacturing industry trends constantly creating new challenges for makers and generalist PCB designers. Board complexity continues to increase due to the density and performance capabilities of current and next-generation processes and materials.

Choosing the right interactive design tool matters. DFM is a methodology that considers, along with yield, any issue that could affect cost and quality before the manufacturing process begins.

Designing for manufacturability (DFM) has therefore never been more important. It's how you avoid cost overruns and reworks, as well as improve the quality of both your boards and the final product. Here, we focus on best practices in these key areas to ensure manufacturability: *tools, process, and partner*.

Tools

Choosing the right interactive design tool matters. DFM is a methodology that considers, along with yield, any issue that could affect cost and quality before the manufacturing process begins. These interactive tools integrate the design and manufacturing processes so that manufacturing requirements and capabilities can be accurately reflected in the design itself.

Interactive design tools give engineers and designers insight into how their design choices will impact the yield or manufacturability of their design while they are working in the design tool. This helps designers make design decisions that will ensure manufacturability.

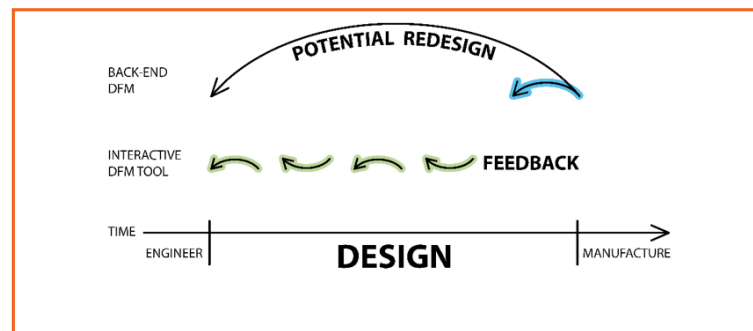
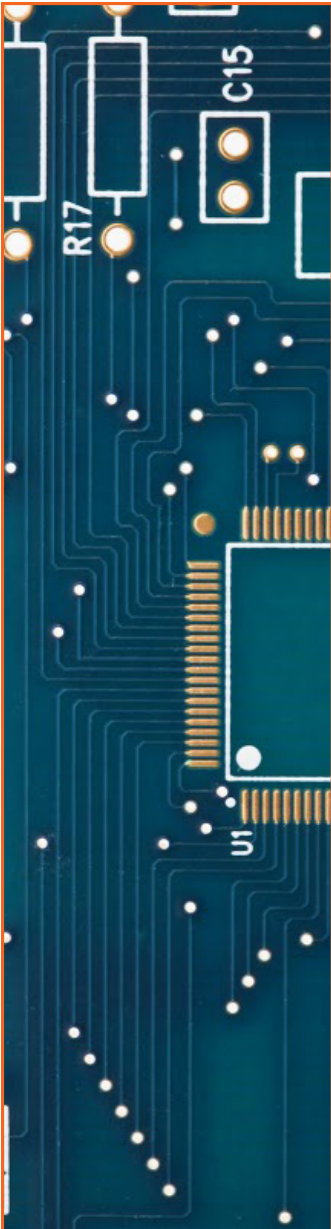


Figure 1
Interactive DFM tools provide more frequent, granular feedback throughout the design process, avoiding expensive changes or redesigns

Process

Your approach to designing PCBs matters to the ultimate quality and manufacturability of the boards. If you are juggling PCB design with other roles, it is common to overlook the simple ways in which you can optimize your design process. And if there is one constant that we have seen across different PCB designers, it is that a good design process trumps heroics every time.

- If you are *not* using a real-time, interactive DFM tool in your CAD software, you will want to consider how to best integrate your back-end DFM tool into your design workflow.



- Close communication between you and your partner manufacturing is extremely helpful, even critical. Collaborating with your manufacturing partner regularly throughout the design process will help you avoid surprises.
- To avoid costly redesigns, use a simulation tool to balance the electrical performance of your design against other physical characteristics like size and yield.
- If you are using a rapid prototyping service for your PCB manufacturing, be aware of any process requirements that may be specific to your manufacturer.

Partner

When you are selecting a manufacturing partner, we recommend that you look for the following capabilities:

- **Manufacturer-Sourced DFM Rule Sets**—Design for manufacturing may as well be called design for *the* manufacturer, meaning *the specific* manufacturer you will be working with.
- **Front-End Expertise**—When shopping for a PCB manufacturer, be very curious about the internal process they use to move a design from CAD file to completion.
- **Quick Access to Support Staff at the Manufacturer**—Choosing a manufacturing partner with readily available support staff can help you solve manufacturability puzzles during the design process rather than after submission.

The real key to optimizing the manufacturability of your PCB designs is to choose tools, processes, and a manufacturing partner that meet your needs.

Methods for Ensuring Smooth PCB Production

Even after the most meticulous design process, issues can still arise during PCB production. In the engineering & tooling phases we often find design elements that will potentially impact the production process or cause the board to not function as intended.

PCB layout software: trust but verify.

Try to avoid being too reliant on layout software—especially if you haven't been formally trained or haven't used the tool very much. It takes time to get the feel for layout software, so it's easy to get a false impression of hole size, parts fit, or the space between the traces.

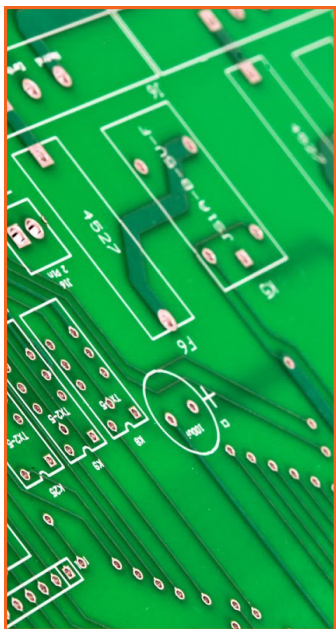
Try to avoid being too reliant on layout software—especially if you haven't been formally trained or haven't used the tool very much

You can work together with your manufacturing partner to ensure you really have enough real estate on the board for your components or optimize routing. This type of fine-tuning can reduce costs, improve performance & increase yields.

Keep an eye out for inaccurate output.

PCB CAD packages and Gerber files can on occasion output incorrect information. Here, too, your fabrication partner will be of immense help. Here are some instances where output files may cause you problems:

- PCB CAD programs let the designer create a logo for the board, but logos often come out garbled or with pieces missing. Logos often have to be split into separate data layers in order to appear correctly. This opens the door for misinterpretation and slows the manufacturing process.
- Uncommon shapes can output differently than intended in Gerber files. Rotations can change and create spacing problems.



It may seem more efficient to bundle four or five designs together on one build, but the practice often causes more problems than it cures.

- PCB CAD packages are notorious for creating issues with soldermasks. Polarity for soldermask output is critical to ensure the correct areas are open or covered. You should make sure the mask leaves the pads open, so they'll take solder while leaving a protective barrier between the thin lines of copper. It's also tricky when there are tight clusters of surface-mounted components and you can't trust the program to get it right.
- Check your tolerances. Sometimes ultratight tolerances are specified even if they are not necessary. For example, if hole position tolerance is too tight, most manufacturers cannot reliably hold to it. This is not an issue if there is only one hole with an ultratight tolerance, but the law of averages will catch up to you if there are multiple holes with too tight tolerances. In this situation, it is best to overbuild to be sure you produce enough boards that meet the tolerance.

In each of these cases, collaborating with your manufacturing partner can avoid unexpected problems.

More is not always better.

It may seem more efficient to bundle four or five designs together on one build, but the practice often causes more problems than it cures. Unique designs often use different CAD packages, and if your bundle contains more than one type of PCB, they can't easily be manufactured in one batch. Each design must be treated differently during manufacturing, with varying process constraints and handling practices.

It's good practice to produce boards together when their characteristics match but bundling dissimilar boards together can delay the manufacturing process or produce sub-optimal yields.

Learning from real-world experience

Very small details can dramatically impact manufacturability. Every day, Sunstone seizes opportunities to help our customers with smooth manufacturing of quality, cost-effective PCBs. We believe constant communication and a commitment to double- and triple-checking every component is the umbrella best practice for ensuring smooth production.

This collaborative strategy can help makers and product engineers innovate, reduce costs, and get to market faster. The following real-world use cases reveal additional methods for improving manufacturability and the quality of the end result.



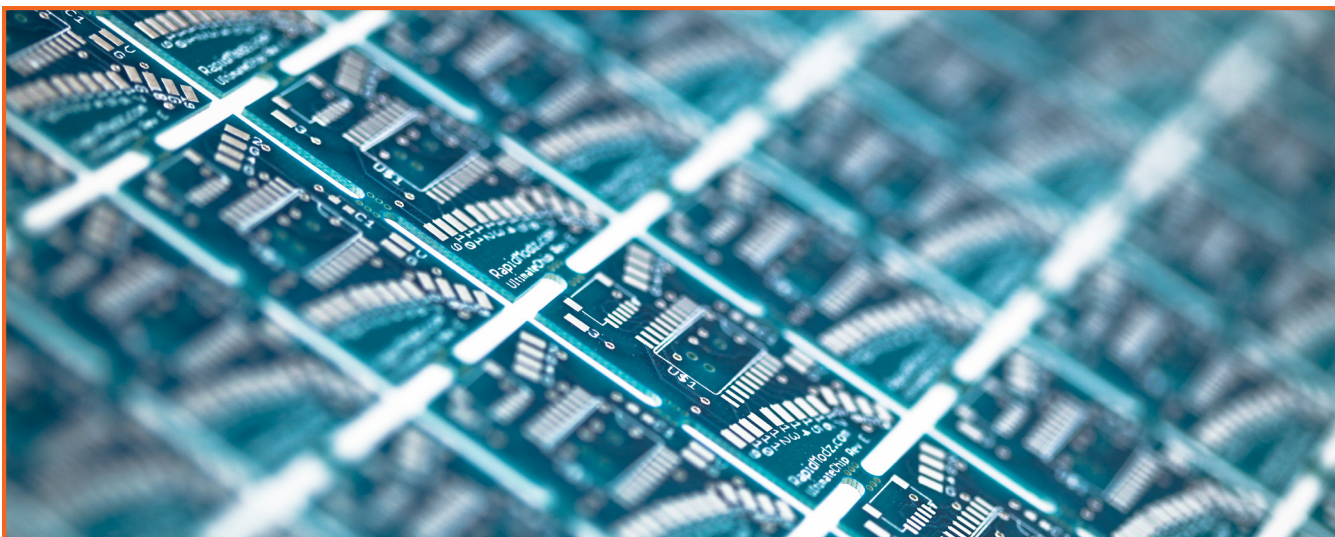
Small Production Runs, Big Improvements



When Amplion Clinical Communications sought to revolutionize patient-care communication, collaboration with experts was critical to success during a highly iterative product development process.

We not only provided the methods and expertise necessary to iterate the system, our team also helped streamline manufacturing—saving time and money in the process.

“Sunstone made us realize we needed to panelize in order to reduce waste and speed up the process,” said Jonathan Musselwhite, Product Manager at Amplion.



Learning to become a PCB designer on the fly.



Aerospace engineer Miki Szmuk specializes in the engineering of tomorrow's unmanned aerial vehicles (UAVs). While at the Controls Lab for Distributed and Uncertain Systems (C-DUS) of the Department of Aerospace Engineering at the University of Texas (UT), in Austin, he recognized the value of becoming his own PCB designer.

He needed the ability to design a board himself quickly and cost-effectively. Using our design tool, PCB123[®], Szmuk was able to teach himself the art of PCB design with our help.

"It was safe to say I didn't have a lot of design experience," said Szmuk. PCB123 had an intuitive user interface and a logical design process. ***"They had people take a look at my designs to make sure they were feasible and that really reduced our stress levels."***

The collaboration with our engineering team helped Szmuk avoid production issues while developing his skills as a designer.



Design input improves manufacturability.



Krytar, Inc. provides the highest standard of radio frequency [RF] test and measure services available to the broadband industry. When dealing with these high-frequency and high-performance devices, if the physical circuit deviates from design by even a tiny fraction of millimeter, it can have a very negative impact.

“Sunstone isn’t afraid to provide input on the design if they think they have a better idea about how to build a circuit that will do what it’s supposed to do,” said Doug Hagan, president of Krytar.

When your partner participates in a collaborative design and manufacturing process, issues with manufacturability are addressed before they can become problems.

Conclusion

We clearly believe open communication between designer and fabricator, as well as attention to detail, are key to a smooth production process and a quality end result. Both represent preventative measures that serve to fix problems before they arise on the production line or after the PCB is in use.

For more tips on PCB design and prototyping, check out our white papers [here](#). And our customer success [stories](#) that offer real-world case examples of solving problems before they occur.