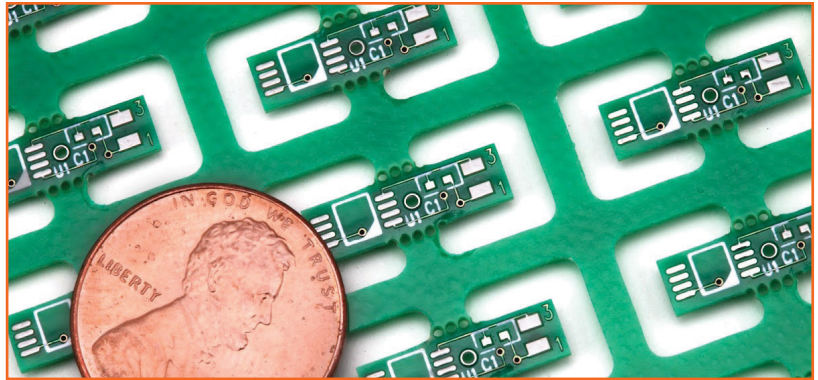


## PCB Design for Smarties



### Four Strategies for Becoming Your Own, Best PCB Designer

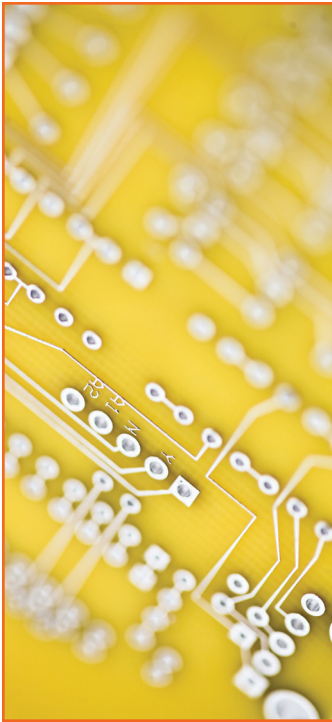
Printed Circuit Boards (PCBs) support the functionality of just about every working electronic device in use today. They are integral to every professional discipline from mechatronics to robotics and process automation. **If you want to make something that does something, PCBs are likely a component of your product design.** With boards getting smaller and demands on them increasing, off-the-shelf PCBs probably will not provide the functionality you need for your next project.

First in a new series of white papers, PCB Design for Smarties will share our expertise on the subject, helping you move forward to your next project, confident there will be smooth sailing from design through manufacturing.

Say you're an electrical engineer or a rocket scientist in a small shop, part of a lean product team, or building the next big thing in a makerspace. You're expert in a lot of areas, but not necessarily PCB design. You need a specific function out of a PCB, you need it now, and ready access to an experienced PCB designer is not at hand. This is a challenge we see every day.

Our customers are part of companies of every size, members of the maker community, or even enthusiastic, innovative hobbyists. They make everything from fuel management systems to time-lapse cameras to specialty signage, and their products all require unique, complex PCBs. Without a designer available on short notice, much less one dedicated to their projects, these professionals have learned to become their own designers.

First in a new series of white papers, PCB Design for Smarties will share our expertise on the subject, helping you move forward to your next project, confident there will be smooth sailing from design through manufacturing.



Just because the next board might be more complicated than your last, your design software doesn't have to include every imaginable feature to meet your needs.

## A Note to Our Readers

Since our title—PCB Design for Smarties—assumes the reader is, you know, a really smart person, we'll treat you like you've done your homework. Rather than make another feature vs cost comparison table or count mouse clicks in a user interface (UI), our goal here is to provide insight and analysis to help you make more sense of the myriad of available offerings.

### ***Choosing the right PCB design tool for you.***

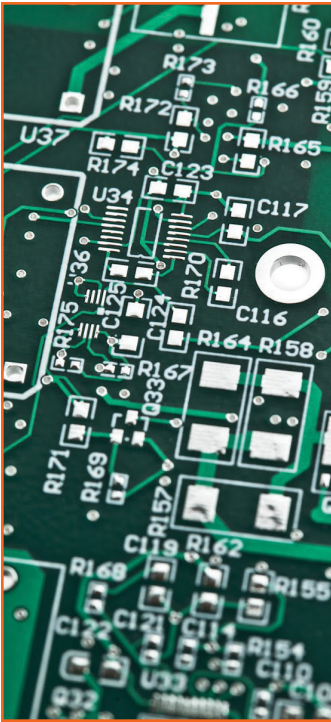
Choosing PCB design software can seem like an overwhelming task. The market offers a wide range of products with diverse feature sets and pricing models. In the interest of transparency, we recognize Sunstone's own PCB123® is part of this ecosystem. Though we are proud of our free PCB tool, this paper intends to inform the decision-making process, not promote software.

When it comes to PCB design, conventional wisdom for the most part assumes bigger is better. Professionals choose established computer-aided design (CAD) tools such as OrCAD or Altium, and free or "simple" electronic design automation (EDA) tools are supposedly for students and amateurs. It is therefore tempting to pick software with a familiar name or larger feature set, especially if you come from a corporate engineering background or aren't sure of future PCB design complexity.

To optimize your design capabilities, you'll want to choose a software provider that will best support your process for transition from design to manufacture. Look for design tools that will:

- Automate most, or all, of the prototype quote and ordering process
- Confirm the design integrity of your prototype and provide status reports
- Perform design rule checks (DRCs) as you design
- Offer unlimited customer support both via email and phone

Try not to bite off more functionality than you'll ever need. Just because the next board might be more complicated than your last, your design software doesn't have to include every imaginable feature to meet your needs.



We recommend a different approach, one that acknowledges the inherently subjective nature of the evaluation, but still focuses on the basics. When evaluating design tools, you should first look for the following attributes:

- Be easy to learn
- Have an intuitive user interface (UI)
- Possess features key to your design needs
- Include access to an expansive parts library
- Be in widespread use
- Generates Gerber files usable by most manufacturers
- Integrates smoothly with your manufacturing process

We believe these are the foundational elements of good design software.

### Save Time with Pre-Defined Parts

Designing your own parts and footprints takes time, lots of time. Even CAD tools with the larger parts libraries can use a boost. Look for integrations that provide you access to the largest, constantly updated parts libraries available. Doing so can reduce time spent on design by as much as 35%.

How you work and what you build will help you determine what additional functionality or support you'll require from your software provider or providers. Whether you're iterating an existing line of products, a freelance engineer working on anything from a weather drone to guitar effects pedals, or a member of the maker community competing at a hackathon, you want confidence in your ability to quickly generate manufacturable PCB designs.

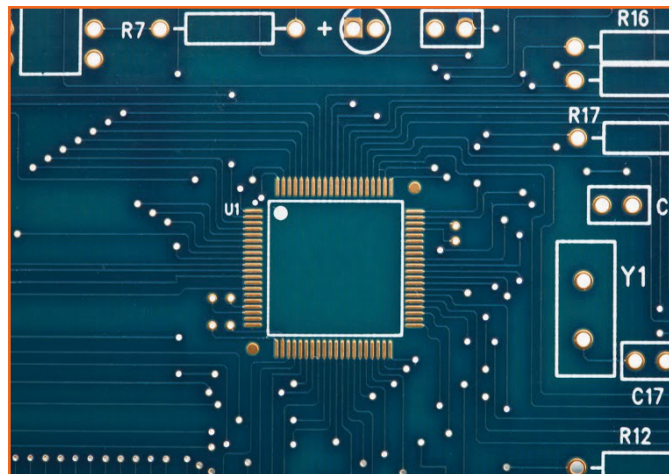
Support is the critical human element of design software, the one that will help you learn faster and improve your designs. A dedicated support team will collaborate with you to help get the most from the design tool, making it easier for you to make the best designs possible today and into the future.

## PCB Design is a Continuous Learning Process

When it comes to board design, each new discovery creates a new path for learning. The remaining sections of this paper will address some common challenges for PCB designers working to expand and hone their skills.

### **90% of PCB design is artwork.**

To the uninitiated, PCB design may seem like a rigid, by-the-book process. Determine a need, decide on a concept, and get to work on a schematic. There's also nothing artsy-sounding about a bill of materials or documentation. With so much emphasis placed on speed-to-market and automation, it's easy to commoditize this crucial skill.



There is no universal right way to place your components.

We urge you to always be studying the craft of PCB design rather than just memorizing and replicating basic skills. We recognize that PCB design is a world filled with limits, tolerances, and parameters—in other words, rules. But, like in *The Matrix*, some of those rules can be bent and others broken.

Placement of components is where art and science converge.

Placement of components is where art and science converge. Though there is no single right way to perform this task, the orientation, placement, and organization of your components will influence the manufacturability of design.

- Orient similar components to facilitate efficient and error-free soldering.
- Do not place components on the solder side of a board that would sit behind through-hole components.
- Organize through-hole and surface mount components to minimize assembly steps.

Make no mistake, an auto-router can be useful, but there can be drawbacks depending on how you use it.

You'll also want to be sure that power, ground, and signal traces have clear paths of travel. Avoid components that will give you heating issues when the board is in use. At a high level, this sounds relatively easy, but to consistently create highly functional and manufacturable designs, you'll need to become a true craftsman. That's an ongoing process, one that requires:

- Practice, lots of it
- An understanding of how your CAD tool functions
- Manufacturing partners that will help you improve your designs
- Fellow collaborators with whom you can exchange ideas and experiences

When you focus on design as an *opportunity for creativity*, rather than a task standing between you and production, you put yourself in position to continuously improve at your craft.

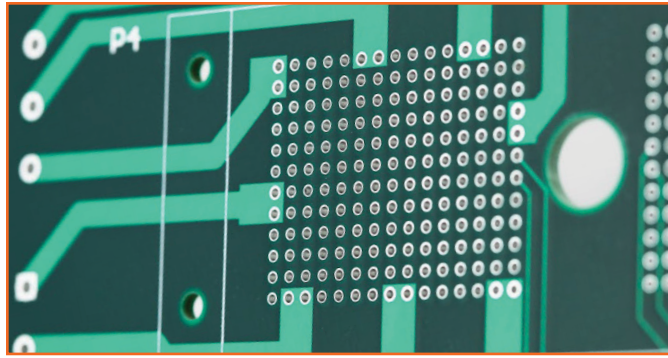
***Only you can determine the best route.***

Almost every piece of PCB design software has an auto-router feature. If you're in a hurry, have a heavy workload, or just don't feel confident about your design skills, this feature can be alluring. Make no mistake, an auto-router can be useful, but there can be drawbacks depending on how you use it.

*First, the obvious benefit: The auto-router feature can save you time.*

The auto-router, however, is not a substitute for performing the bulk of this task yourself. Assuming the feature will solve all your routing challenges can get you into trouble. Experienced designers all have horror stories about auto-routing run amok, resulting in hours of manual cleanup.

Leave the auto-router in charge and the results can be asymmetrical, not easily manufactured, and lacking in performance. It's like a pilot who doesn't know the destination of the flight. Things are going smoothly in the cockpit, no turbulence or crying babies, so why aren't we in Denver yet?



To fully realize the value of an auto-router, we suggest using it as a compliment to your routing efforts, not a replacement for you. Hand route the critical traces yourself, then auto route the remaining ones. In this capacity, the auto-router can improve the precision of your established routes, perform clean up, discover bottlenecks, and teach you how to create more efficient routes yourself on your next board design.

***To multi-layer, or not to multi-layer—You may not have a choice.***

That next design is increasingly apt to be for a multi-layer board. PCBs are getting smaller, making single and double-sided boards less useful in many ways. Only a multi-layered PCB can offer the desired functionality if it occupies a small space in your device. Even if space is not as much of an issue and multiple single layer boards could be used, higher performance, multi-layer PCBs are easier to mount and usually require only one main connection point.

PCBs are getting smaller, making single and double-sided boards less useful in many ways.

If you have limited or no experience with multi-layer board design, the prospect of an attempt can be daunting. Multi-layer boards don't have to be a hassle. Think of them as one or more single-layer boards stacked together. If you're prepared and follow a few basic guidelines, designing a multi-layer board can be a snap. Here are some guidelines that will help get you started:

- Become familiar with multi-layer design tips and guidelines provided with your CAD tool.
- Familiarize yourself with components, tolerances, and requirements unique to multi-layer PCBs.
- Make sure your design tool library is set up for multi-layer boards.
- Know your manufacturer's requirements for multi-layer designs.
- Don't hesitate to ask for help.

Becoming your own, best PCB designer is a learning process, a continuous journey. We recognize there's an element of trial and error involved.

Well-timed guidance from your manufacturing partner or design software provider can be the difference between a smooth transition to manufacturing and a do over. With a robust tutorial, it's certainly possible to self-teach multi-layer board design, but the risk of errors and omissions increases.

It makes sense to rely on expert help when it's available. You may discover that an assist with layout is invaluable if you're under the gun. Layout services are a great way to fill knowledge gaps, improve board design, and ensure manufacturability.

### **Conclusion**

Becoming your own, best PCB designer is a learning process, a continuous journey. We recognize there's an element of trial and error involved. Resources like this one can help get you started on the right track. To stay on the path, we encourage you to stay curious and keep learning.

Take advantage of all available design resources. Whether it's a video tutorial, software manual, or a shared experience from one of your peers, the next phase of discovery is a mouse click away. Join our community forum to share ideas, stay up to date, and read what industry experts are saying.