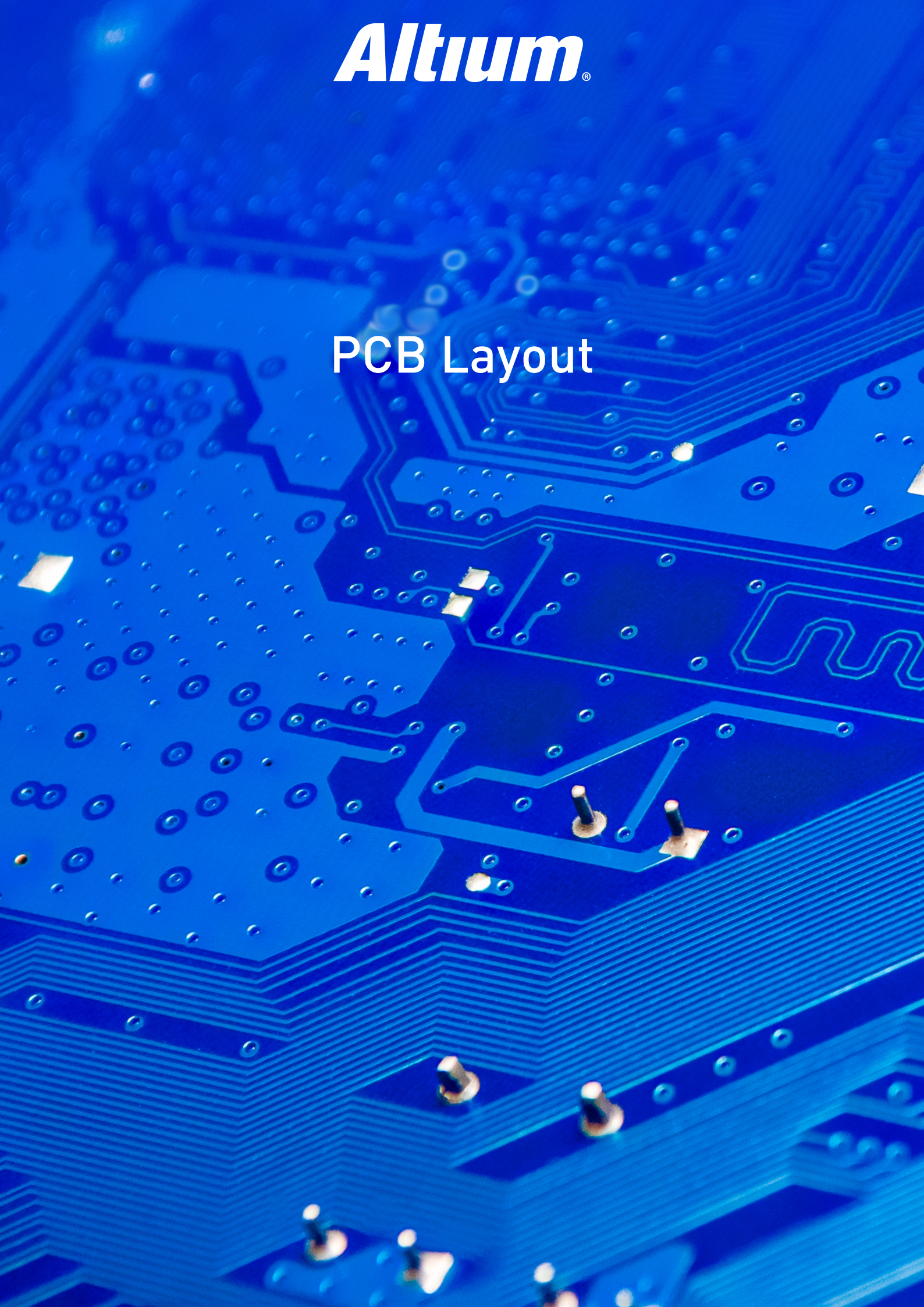
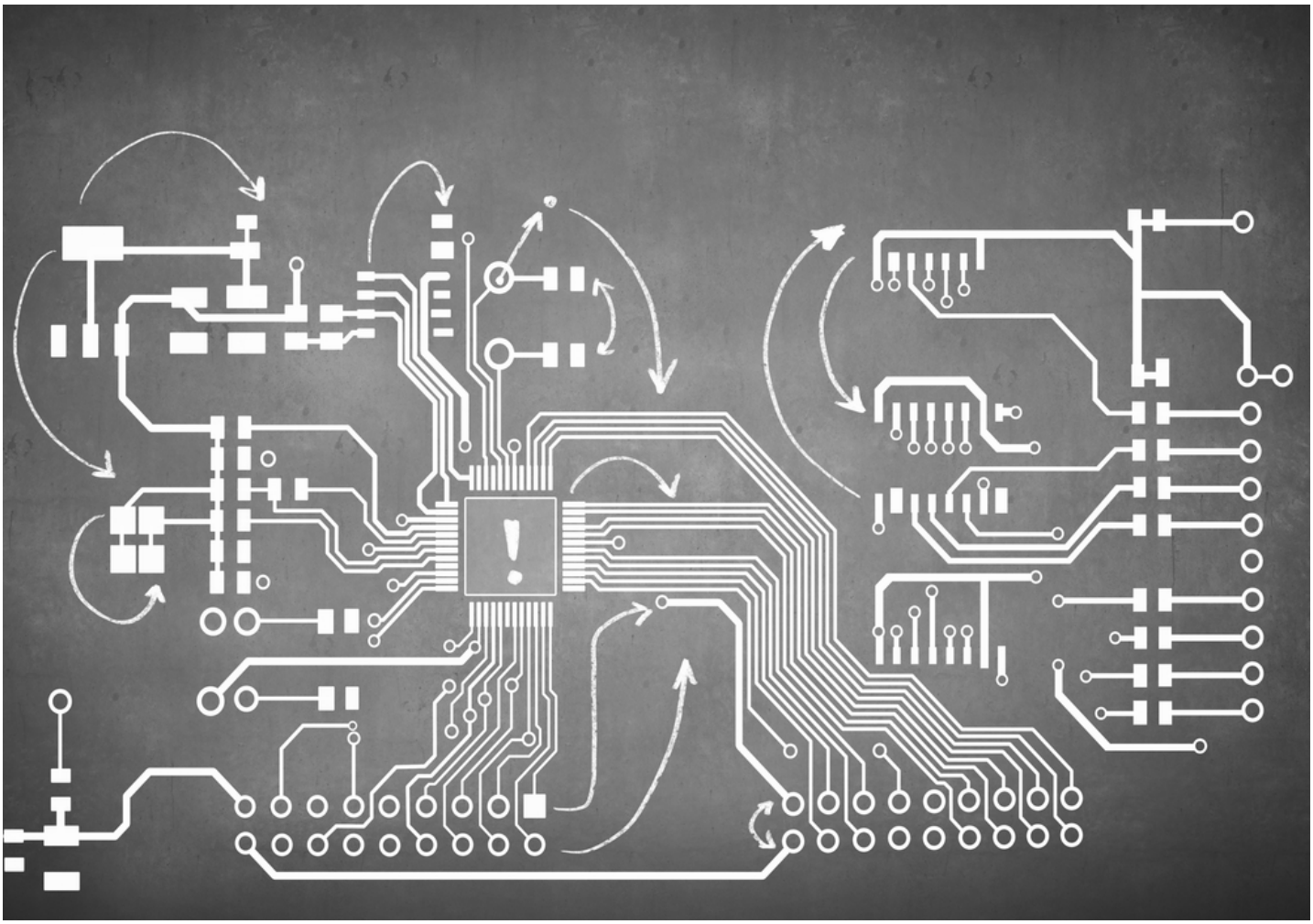
Altium®

PCB Layout





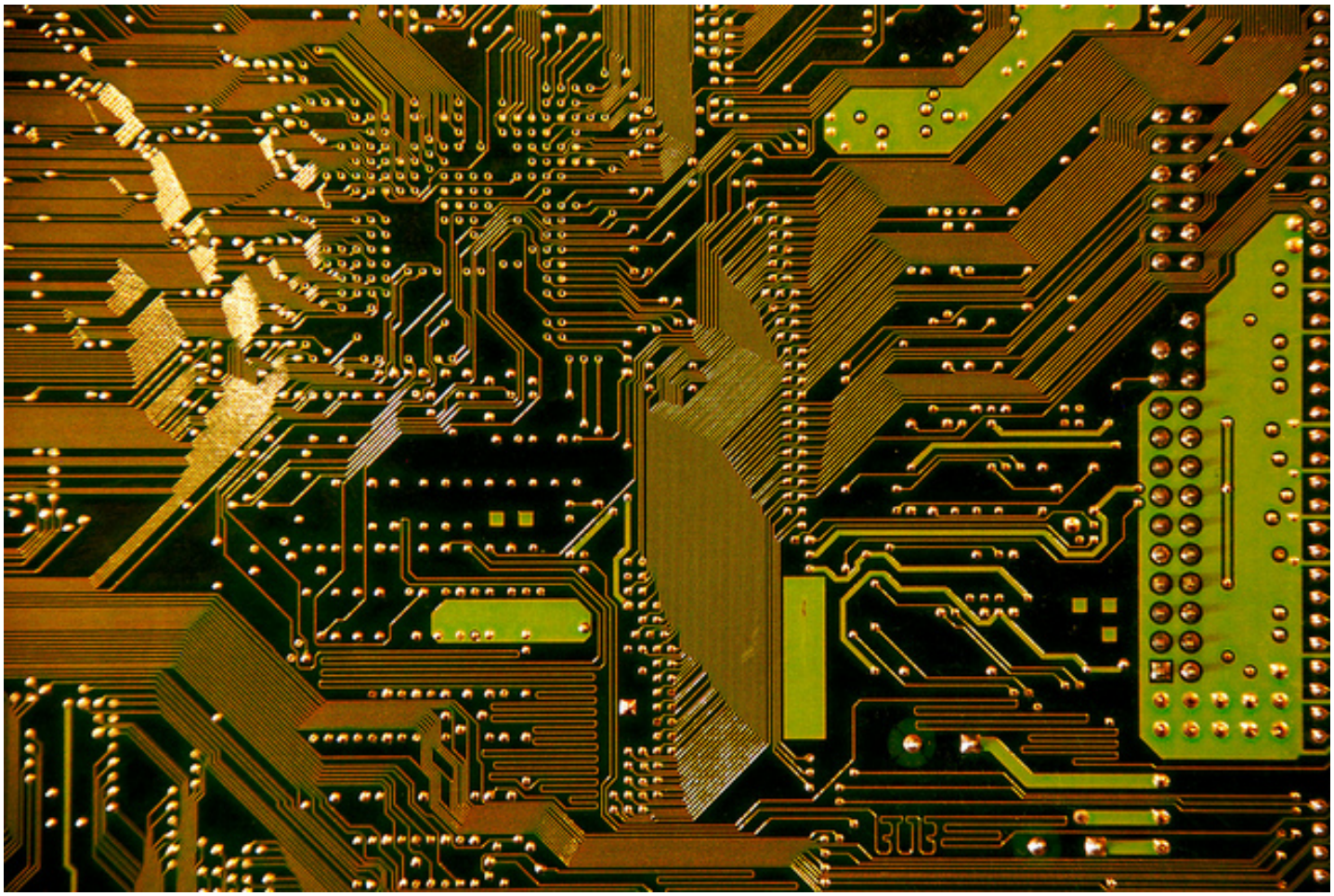
PCB LAYOUT

As electronics get smaller and more complex, so too must the boards (PCBs) that make them function. Because of this, your PCB layouts matter more than ever before. While a PCB with a poor layout may still function, a well-planned layout will go a long way to ensure efficiency, minimize risk, and even save you money in your production budget. The care you put into your PCB layout can give your product a strategic edge over competition through reducing the amount of time to market, as well as providing useful product differentiation.

Join us as we discuss a range of topics relating to PCB Layouts including:

- Syncing Your Schematics and PCB Layouts Increases Efficiency and On-Time Delivery
- How Proper PCB Layout Can Help You Avoid Adding Heat Sinks to Your Design
- The Most Important Features for PCB Layout Software Comparison
- How to Find the Best PCB Layout Help for Your Specific Design Questions
- Using Customizable Grids to Save Time on Your PCB Layout Process
- PCB Layout Constraints That Help Mitigate Component Obsolescence

SYNCING YOUR SCHEMATICS AND PCB LAYOUTS INCREASES EFFICIENCY AND ON-TIME DELIVERY

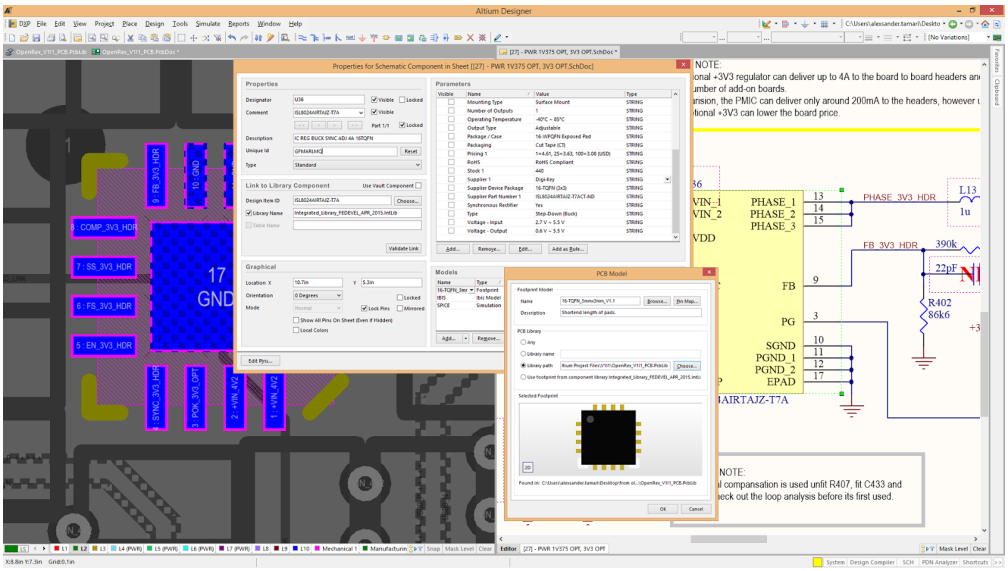


A while back, I was working at a company that designs and installs systems for automating smart irrigation and environmental monitoring. We had gotten into the habit of making last minute “hacks” (pin swapping, gate swapping, etc.) to our boards to accommodate a specific need from a job (or to compensate for a design flaw or oversight on a particular model after getting it back from the manufacturer). While this improvisatory method worked in the short term, and the company was always able to meet the requirements expected from a project, this “fly by the seat of your pants” approach was simply not scalable.

Part of the problem was that we were spending too much time tweaking the PCB layout (we’d receive boards back from the manufacturer, only then realizing that we had neglected some detail) and not enough time taking a big-picture approach to the design through detailed schematics. A Unified Data Model approach might have helped us at the time if we had known about it.

KEEP YOUR SCHEMATICS AND PCB LAYOUTS IN SYNC WITH A UNIFIED DATA MODEL APPROACH

A Unified Data Model approach allows users to seamlessly integrate multiple aspects of the design process, including schematic symbols, PCB footprint, supplier procurement info, SPICE models, and more.



These are some of the benefits that keeping schematics and PCB layouts in sync through a Unified Data Model approach offers:

- **Simultaneous Design:** Creating a schematic is crucial for effective planning. However, for many designers, it is also useful to get deeper into the details of the circuit design through the PCB layout. By integrating schematics and PCB layouts through linking files and automated updates, your design process can be free to simultaneously attack both a “big picture” and a detailed, “real world” approach.

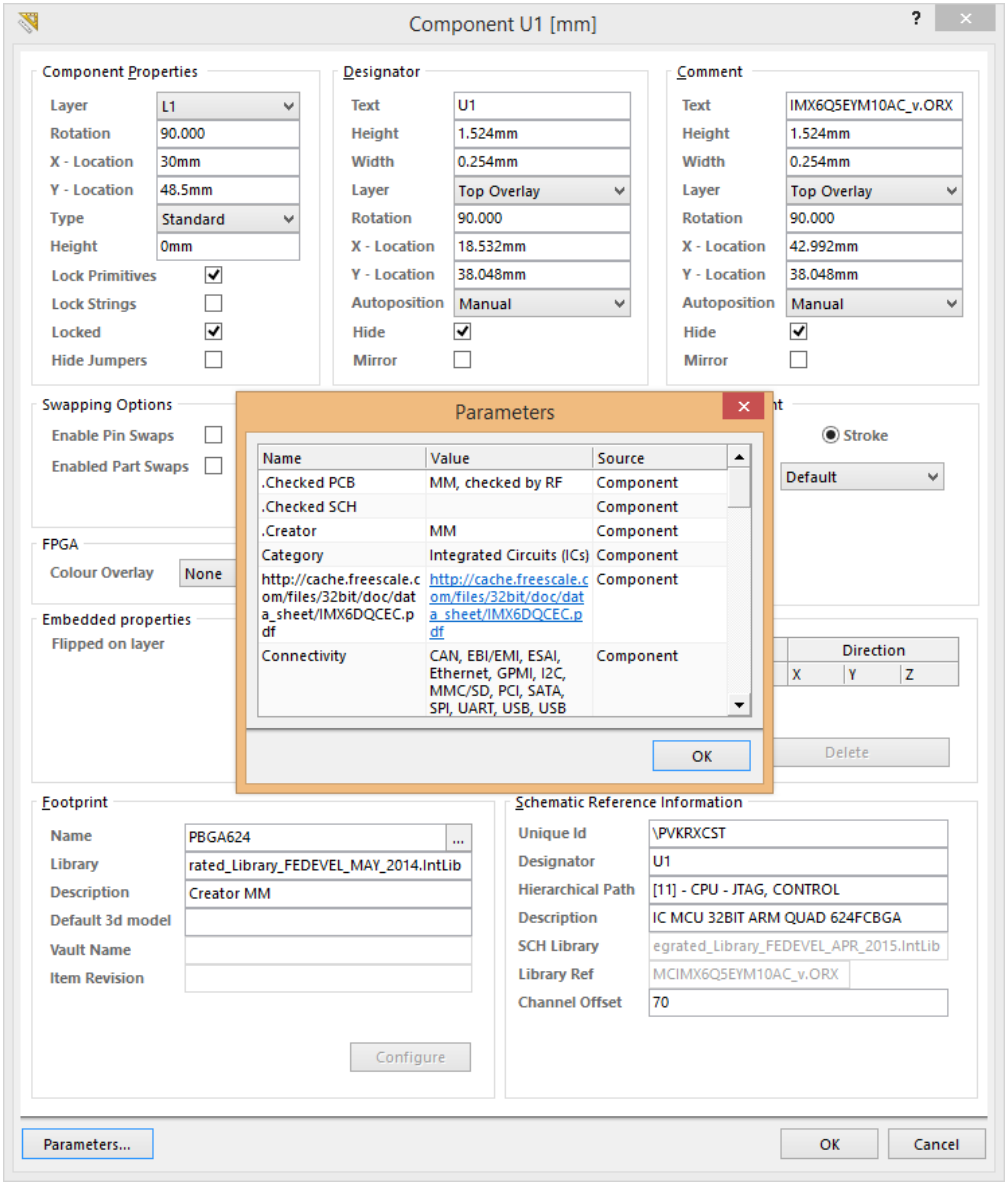
- **Time Savings:** Taking the necessary measures to effectively plan out schematics can save you precious time in the long run. In the past, when my team improvised our design straight to the board, we’d end up wasting our time. In many instances, we’d receive our print back from the shop only to realize we missed a “minor” detail that would end up costing us hours of revision. Syncing PCB layouts and schematics can help ensure that your design assets are consistent and validated before being sent to the manufacturer.

- **Money Savings:** In addition to saving time, taking a Unified Data Model approach can also save money in the long run. I’m sure my former boss would have loved to hear that, had my company applied this approach sooner to our design process.

Doing so would have been significantly more cost-effective and helped salvage all the time and labor devoted to correcting mistakes that could have easily been avoided.

- Multiple Team Members: From my experience managing teams of engineers and designers, I know how difficult it can be to coordinate (and efficiently plan out) the deliverables from each of my team members. You might have one designer that's particularly gifted at planning out a product through schematics, and another who loves working directly with the PCB layout. Effectively coordinating the assets from these team members can be stressful, especially when you're working under a manufacturing deadline. I can't tell you how many times I've been in the frustrating predicament of having to pour through multiple emails from different designers, or to scan an assortment of files and spreadsheets, just to get everyone on the same page. Through a Unified Data Model approach (including a centralized repository of all your design assets, where schematics and PCB layout are immediately synced) you can help facilitate efficient communication of design changes between multiple team members.

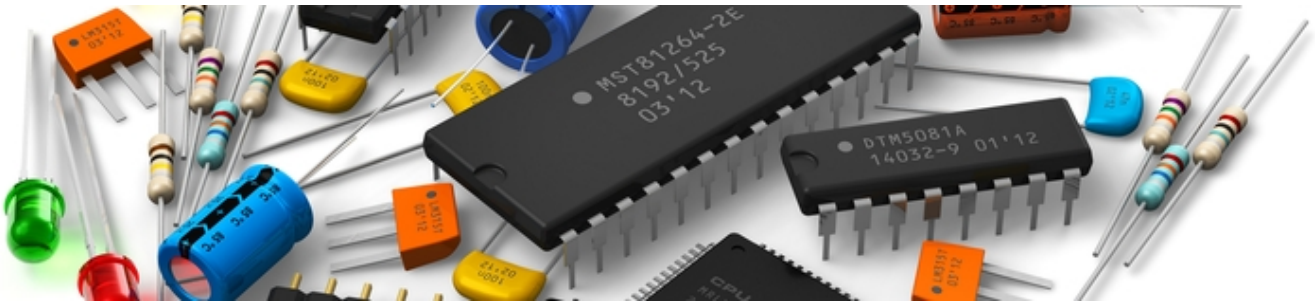
- Spend More Time on the Design Process: Let's face it—as engineers and designers, we should be spending the majority of our time on creating and implementing innovative, well-built products. Unfortunately, too much of our day-to-day is often spent on unnecessary (and circular) communication processes, searching for the right file, and trying to get everyone on the same page. Syncing up your schematics and PCB layout through a Unified Data Model approach can help you cut out time spent on the things you don't really want to do, so that you can focus on creating excellent designs.



I think most designers would agree that a Unified Data Model approach is the way to go. So why isn't this practice more commonplace? Perhaps the reason it hasn't been fully adopted yet is due to many designers not being aware of the emerging and powerful tools that can help streamline the design process.

Altium Designer allows designers to execute a Unified Data Model approach by syncing up their design assets. By integrating your schematics and PCB layouts to automatically reflect all updates made in the system, you can streamline your entire design process to be more efficient in bringing your idea to market.

HOW PROPER PCB LAYOUT CAN HELP YOU AVOID ADDING HEAT SINKS TO YOUR DESIGN



Everyone knows that excessive heat can kill components or drive them out of spec, but sometimes it isn't obvious when heat will be a problem. Heat mitigation issues may appear over time, like a solder joint that fractures due to thermal expansion and contraction over numerous heat cycles. These issues can go undetected in testing, which is why it is important to layout your board correctly in order to avoid these scenarios. Read on to find out more!

Having to recall products, like faulty computer monitors, is a financial burden that can be avoided.

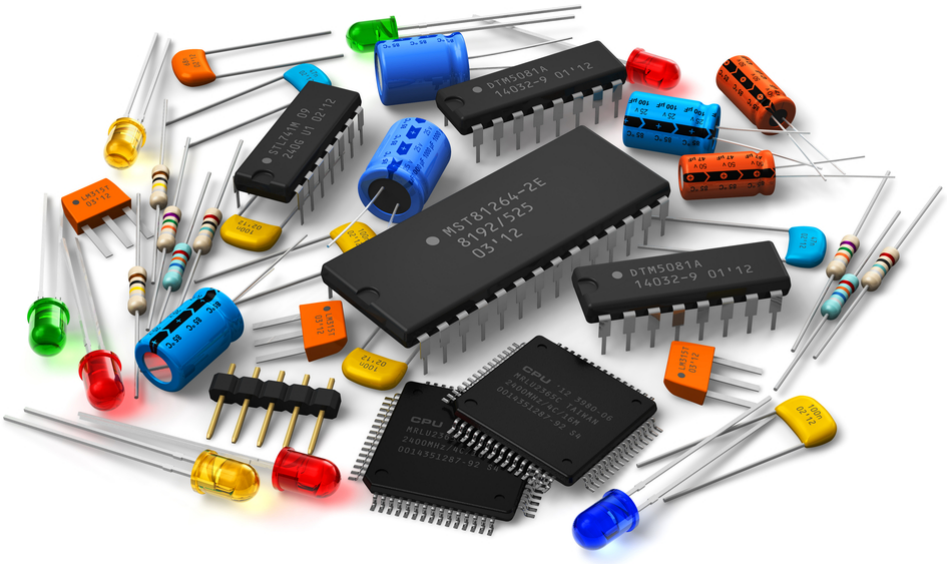
Many years ago I bought myself a new computer monitor capable of displaying the highest resolutions and color depths that computer technology had to offer. I could barely contain my excitement when I plugged it in and began experimenting with various

apps to test its capabilities. Everything was working beautifully so I left my office to brag about my new display. Upon returning, I was shocked to see that the display was now a flickering mess. I cycled the power to see if it would recover, but it did not. My new monitor was apparently broken.

A few days later I cracked the case and poked around to see if I could find the problem. It turns out one of the video circuits was mounted directly over the power supply. As a result, it heated up the video circuit to the point it was no longer was functioning in spec, which resulted in a distorted display. The fix was simple, install a heat shield to direct the thermal energy away from the video circuit. That being said, the real problem was in the design of the product. The video circuit should not have been in such close proximity to a heat source.

HEAT IS A PCB'S WORST ENEMY

Everyone knows that excessive heat can kill components or drive them out of spec, but sometimes it isn't obvious when heat will be a problem. At times, heat management can be particularly difficult to troubleshoot and might only be an issue in uncommon situations. For example, when extreme temperatures occur. More so, heat mitigation issues may appear over time, like a solder joint that fractures due to thermal expansion and contraction over numerous heat cycles. These heat mitigation issues can go undetected in testing, which is why it is important to lay out your board correctly in order to avoid these scenarios.



Consider the location of external heat sources before placing components on your PCB.

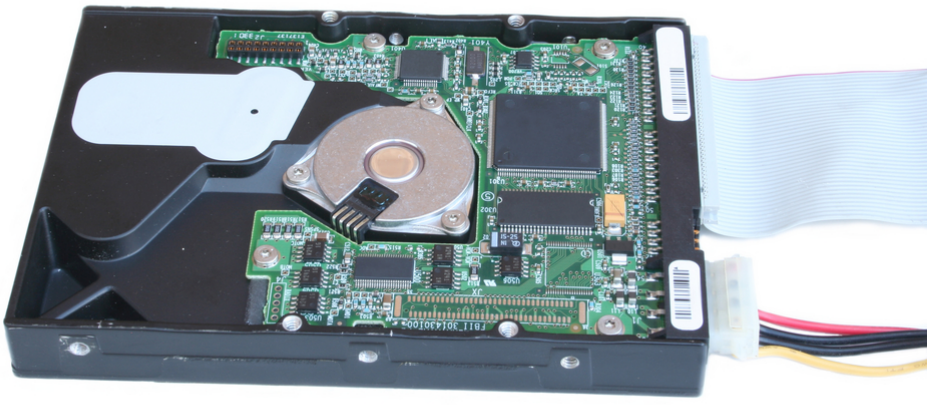
WHAT'S A PCB DESIGNER TO DO?

It is common in PCB design to mitigate heat with SMT heat sinks, fans, thermal vias, and other techniques. These can be very

effective but add cost and complexity to a design. The least expensive and most reliable solution is to design a PCB such that undesired heat transfer from external sources is not an issue. Design around the problem, don't throw components at it!

When you plan your design, remember your PCB is part of a larger system. You have to consider both the placement of heat producing components relative to one another on the PCB and to the other parts of the product as a whole. Since the location of PCBs varies depending on the device it is designed for, external heat sources can be located in three dimensions around your board. With this in mind you should consider:

- Isolating heat sources: Keep heat-generating subsystems, like the power supply, isolated in your layout so that they don't impact other components.
- Other PCBs: There is a good chance nearby PCB's will have heat-producing or heat-sensitive components on them.
- Physical parts: Mechanical parts, wires, and mounting frames may absorb and transfer heat from your PCB to other parts of the product. This includes parts that the user may come in contact with.
- The enclosure: As designers, we are under constant pressure to make our products smaller and more compact. This means that the enclosures around PCBs are getting tighter. We have to give careful thought to how that will affect heat transfer from our PCB. Tighter spaces offer less opportunity for convective cooling and a higher chance of conductive heat transfer to other areas of sensitivity.



Consider how tight your PCB enclosure will be, this may affect how your board layout.

THINK IN A 3D SPACE WHEN DESIGNING YOUR PCB

You are not designing your PCB in a vacuum. From the outset, you should think of your design in three dimensions. Consider how your components will interact with each other and how they affect the space around the PCB.

To do this, I recommend reviewing the product design specs to see how your PCB is positioned in the product, and which elements are close to it. Make a note of anything that may be sensitive to heat before you begin your layout. This information will affect the first broad strokes you take at your design and can help you avoid reworking things later, which is always more difficult. Next, you should consider how the PCB is positioned within the enclosure. If it is a tight fit, or there are constricted areas, avoid putting heat-producing components where they will be unable to shed their heat.

If your computer monitor is still functional (unlike mine), you should consider using the design tools included in your professional PCB design software to help you out. Many, such as Altium Designer, let you define clearances around your components and verify that the clearances are honored in the design. By defining clearances for your heat-producing and heat-sensitive components you can let the software keep you safe. Finally, you can run your design through a thermal simulator to highlight any thermal constriction points and adjust the layout to mitigate them.

KEEP YOUR PCBS COOL AND COLLECTED

Heat is a constant challenge in PCB design and will pose a bigger challenge as products are becoming more compact. Careful forethought and planning can help avoid heat-related issues that are difficult to reproduce, troubleshoot, and fix. No one likes to watch their new computer screen have a meltdown. Even worse, no company wants to recall a product. Avoid this by contacting the experts at Altium for more information on good design practices and the PCB tools available for heat management.

THE MOST IMPORTANT FEATURES FOR PCB LAYOUT SOFTWARE COMPARISON

Have you ever gone to a used car dealership and tried to get a good deal on a car? It's nearly impossible for anyone but a mechanic. When I went I ended up with a choice between two cars that seemed nearly identical to me, except for their price. I picked the cheaper one and took it to a local shop, and found out that I'd chosen a jalopy. Sometimes choosing PCB design software can feel the same way. You use a free program or spend some money on an intermediate one, just to find out it only goes half the distance you need it to. Before deciding on an electronic design automation (EDA) tool, you need to find out it if supports the advanced features necessary to design your board. It's also important to be sure that all of these elements are available in a unified environment that can be customized to fit your specific needs.

FEATURES TO LOOK FOR

The reason I chose the car I did was simply the price. It looked the same as the other vehicle but cost a couple thousand less. It turns out that I got what I paid for, and that rings true for ECAD software as well. You probably want to use less expensive, or even free, software to design your PCB. The problem with the knock-off brand is it doesn't come with all the bells and whistles, but as PCB design becomes more complex these "bells and whistles" are turning into necessities. Here are a few of the things to look for when choosing your design program.

- Board Size - This one is pretty obvious, but I'll mention it anyway. Lots of free tools are severely limited onboard space. Make sure your software supports enough space for your circuit.

- Advanced Via Design - If you're designing things like high density interconnect (HDI) boards or high-speed boards this is very important. You'll need to use things like blind and buried vias, via-in-pads (VIPs), microvias, and back drilling. Support for these is not always included in the lower-end software, so make sure you can use them if you need them.

- Number of Layers - This one is a bit self-explanatory as well. Most free and nearly free tools limit the number of layers you can use. If your board is going to use lots of layers, you probably want to go for professional PCB design software.

Integrated software will ensure you don't end up missing a piece.

- 3D Modeling - You may not think much about 3D modeling, just like how I don't bother much about the inner workings of my car. However, 3D modeling can be very important for you, especially if you're designing a PCB for embedded systems. You'll need to make sure your board fits inside its enclosure. Great software will be able to import a model of your enclosure, generate a model of your circuit, and see if they fit together.

- Power Delivery Network Analysis (PDNA) - PDNA used to be just for the power experts, but more and more you'll find it's important for you to do at least a little yourself. No one wants to learn how to use the signal analysis programs that make up the basis for PDNA. It's much easier to find a program that can do this for you.

- Differential Pair Routing - I often decide to try and navigate my car to my destination without a GPS. On a related note, I also

frequently get lost. If you're designing a board with differential pairs, good design software can be like a good GPS. You'll need to route your differential traces to reduce length and preserve signal integrity. Cheap programs may let you route things yourself, but they won't give you the help you need to find the best path possible.

This isn't an exhaustive list, but it should give you an idea of the types of things you should look for in a good EDA tool.

UNIFIED ENVIRONMENT

Some people have a different car for every situation. A big truck for moving things around, an SUV for taking the family to the woods, and a sedan with good gas mileage for long trips. Unfortunately, I'm not rich enough for that and I have a small driveway. I prefer one car that does it all. This is what you should be looking for in PCB design software, an all-in-one solution. This will greatly reduce your headache and ensure that everything meshes together at the end of the day.

The funny thing about all the different free tools out there is that lots of them do different things well. Some have advanced functionality, but have such a steep learning curve that makes them hard to take advantage of. Others are easy to use but lack all the features you need. In addition, most of them lack peripheral tools like simulators and PDNA. It's already a pain to learn the design software, so you don't want to have to read up on 7 different analysis tools as well. It's much easier to buy solid software that has everything included. I would identify what features you want, and then find software that has them all integrated.

The other side of the consolidated approach is that you can be sure everything will actually work. Lots of software can "plug in" different third-party tools. Sometimes that works, and sometimes you end up spending more time trying to fix the interlink than you do actually using the plugin. Additionally, there's the possibility that the external tool doesn't do what it's supposed to and leads you to make inaccurate design changes. If one company has integrated everything into their program, it's sure to work the first time. When your tools are all made by one developer that you trust, you can be confident in the results they provide.

Sometimes no one makes what you need. Customization will let you make it yourself. Editorial Credit: LandFox / *Shutterstock.com*

CUSTOMIZABILITY

If there's one thing car junkies love, it's customization. If I'd been more into it maybe I could have souped up that clunker I bought. PCB designers love customizing as well. Even if you buy the absolute best design software out there, chances are that it won't have extremely specific functionality that you need. At that point, the best tool you can get is one you can change.

Design software like Altium Designer comes with a software development kit (SDK) that lets you create your own tools. That way if you think of something revolutionary and don't want to wait for a developer to implement it, you can do it yourself. That being said, Altium Designer should have nearly every tool you need. It includes all of the features mentioned above, and much more. Of course, they're all available in a unified environment. No more clicking through different windows, trying to find where that waveform simulator went.

Have more questions about PCB design? Call an expert at Altium.

HOW TO FIND THE BEST PCB LAYOUT HELP FOR YOUR SPECIFIC DESIGN QUESTIONS

Have you ever heard stories about the gamblers in the Wild West pulling an ace out of their sleeve in order to cash in with the winning hand? Even though it's not the most moral example I've ever used, the principle here is still a good one. Always make sure that you have something in your pocket that you can draw from when you need it. And this is just as important for us as designers as it is in everyday life.

At some point, everyone needs help with either a design, a CAD tool function, or a manufacturing query. It's easy to get swept away by problems, but that doesn't have to be the case. There are helpful resources both online and offline that designers can keep in their pocket to draw from. Here are some of the ways successful designers accomplish this.

COMMUNICATE WITH YOUR COLLEAGUES

This one might seem obvious, but we sometimes forget how much valuable experience our colleagues have. It's also important to remember that there is no shame in asking for help. This is especially true if you have a network of professional associates with a wealth of design experience. I learned a long time ago to put my pride behind me since it did nothing but get in the way when I was trying to resolve a problem. Since then I have steadily added to my network of professional associates who I go to when I am stuck on a problem. I am also not afraid to go to my friends, family, and other non-professional associates for help as well. Sometimes the

solution to a problem can be staring us in the face, only we are too close to see it. My spouse, who doesn't know a thing about laying out a PCB design, has at times helped me through a design related problem simply by listening to me patiently and then pointing out errors in my thinking.

To make a long story short, if you need help, ask for it.

Give your colleagues a call when you're questioning your circuit layout.

THERE IS A LOT OF INFORMATION OUT THERE ALREADY WAITING FOR YOU

If you're reading this article then you already know that the world wide web is an excellent source for finding answers to your design questions. The trick is doing this effectively so that you get the answers that you're searching for. Here are some tips:

Use specific search terms: Regardless of whether you're dipping your toe into PCB design or if you've been doing this for decades, there's a good chance that someone else out there has had the same problem. Ideally, they'll also have shared it online. By specifying your design issue you'll find information that is targeted to that particular problem and will expedite your ability to solve it.

Think critically about sources: The abundance of information online can be both a blessing and a curse. There are so many types of resources to pick from it's hard to know which will be the most effective.

1. Web forums. Web forums are often a helpful resource for tackling specific design problems, but they are also a great forum for debate. While it is good to look at a problem from every angle, you will encounter biased information. This can be the result of each forum member having their own unique design experience and from working with different PCB design tools. Bottom line is, you can get what you need from forums but you might have to do some digging.

2. Blog articles and FAQs. At the risk of getting too meta, I won't dig into this one much, but blog articles and FAQs hosted on professional sites are a good resource for getting started. They're designed to get you on the right path to finding a solution to your design query and ideally will direct you to more detailed information. Most importantly, they are reviewed prior to being published to ensure credibility.

3. White papers. White papers are definitely the most thorough of the resources in this list. Depending on your design query they could be overkill or they could be a perfect fit. They do take more concentration to dig into, so ideally you'll have come across them from a blog article or a reference that indicates that they are worth focusing on and will answer your question.

A final best practice is to stay up-to-date with subscriptions to design periodicals whether they are online or hard copy. You might already have the answer to your question tucked away in your inbox.

If you have access to CAD support, use it.

IF YOU HAVE ACCESS TO CAD SUPPORT – USE IT

Lastly, be sure to acquaint yourself with the support services offered by your CAD vendor. Whether web-based, E-mail, or phone support, the company that provides your CAD tools wants you to be successful. Depending on your design needs, software for PCB design sometimes has live support. What this means is, you can opt into being connected with their wider user community. For example, Altium Live gives you access to software updates, design templates, forums, videos, high-quality customer support and more.

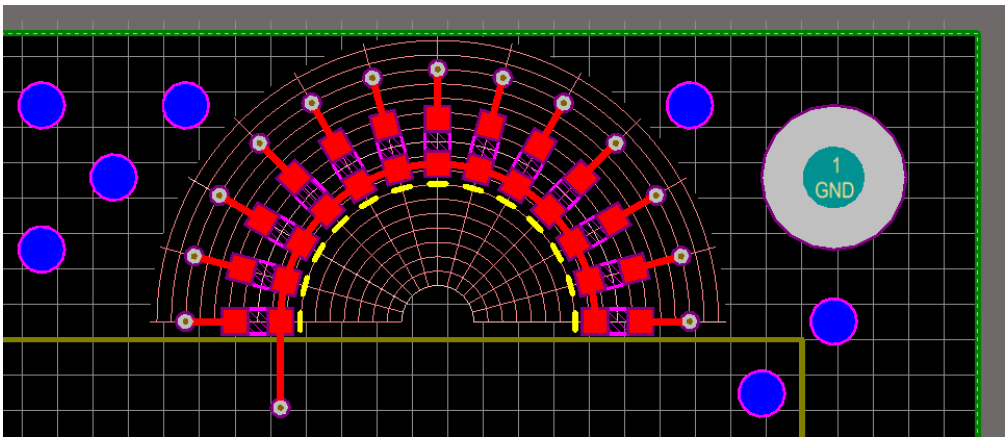
Never underestimate the value of customer support. I've seen the most perplexing of problems that potentially could have ruined a design get solved in a matter of minutes simply by involving the customer support department.

There are many ways that you can get help with your design and with your CAD tools, and we've talked about a few of them here. You just need to be prepared beforehand so that you aren't taken by surprise. Then the next time a problem throws you up against a wall, take a deep breath and reach into your pocket for that hidden ace. You will be really glad that you did.

Would you like to find out more about the different support options that Altium offers its users? Talk to an expert at Altium.

Reading this but don't use Altium yet? Be sure to check out your own free trial to find out for yourself why Altium is the first choice of PCB designers.

USING CUSTOMIZABLE GRIDS TO SAVE TIME ON YOUR PCB LAYOUT PROCESS



Components and Arcs placed precise on a 15 degree angular step and 0.5 mm radial step.

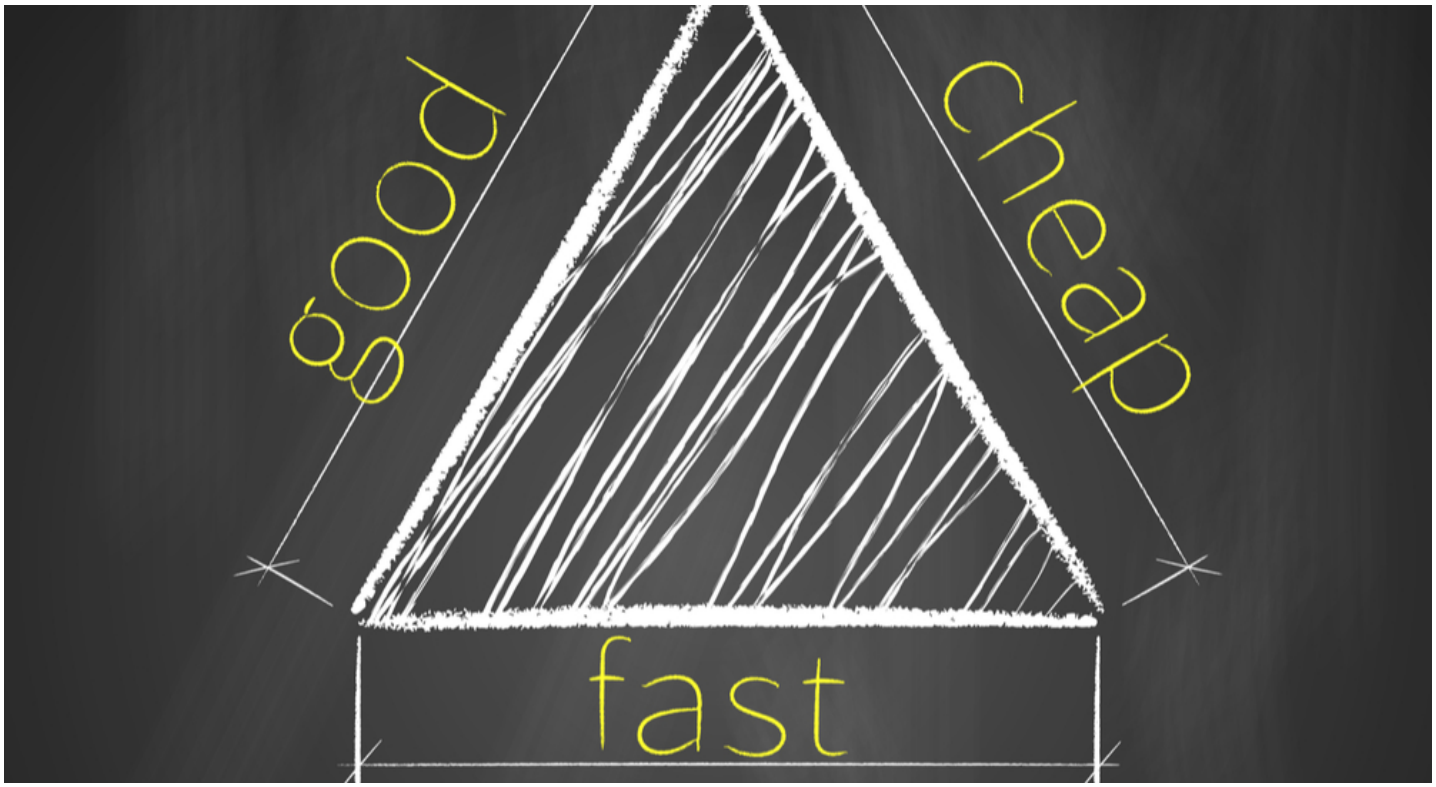
Relying on one grid size can require frequent step size changes, as well as a unit of measure change, and possibly even board origin location changes, in order to achieve all required grids. Read to understand how customizable grids can help to resolve this complicated and manual process.

It's possible to rely exclusively on your default grid for all aspects of component placement, trace routing, and mechanical or special object placement. However, reliance on one grid size requires frequent step size changes, as well as a unit of measure change, and possibly even board origin location changes, in order to achieve all required grids. This is where customizable grids come in to resolve this complicated and manual process.

USING GRIDS TO SPEED UP YOUR DESIGN PROCESS

Placing components of a specific grid can help speed up the component placement and alignment process. It can also help achieve optimal component density. As most component footprints are defined in millimeters, one or more dedicated, user-defined metric grids can be defined to accommodate component placement. Such grids can be specified to be active only when in component placement mode, and to remain disabled and invisible while routing traces. Want to learn how to save time on your PCB layout process? Download our free whitepaper about Defining and Using Multiple Grids in Altium Designer.

PCB LAYOUT CONSTRAINTS THAT HELP MANAGE COMPONENT OBSOLESCENCE



Just about everyone in the world of engineering has heard some variation of the “Iron Triangle” theory. It’s a popular way to talk about what is or isn’t possible among those given the task of getting things done. Basically, it is a philosophy that says you can get something done fast, cheap, or good; pick two. For example, when designing a PCB for an aircraft, cheap goes out the window, fast takes a back seat and quality is paramount. Alternately, when designing a PCB for next season’s flashy throwaway toy, quality is the lowest concern, fast is a must, and it better be cheap. Once a circuit design is basically finalized, finding the best way to use the allotted space for a layout requires making many decisions about tradeoffs. These will have a large effect on the quality of the product throughout its lifetime. What follows are some suggestions regarding these important considerations.

PLAN ON EXTRA TIME FOR ROUTING TIGHT SPACES

Since there is no iron shoe-horn for cramming dozens of inexpensive features into a tiny space, tradeoffs quickly become necessary. In many products, the outline of the board is predetermined by mechanical properties of the larger assembly, and the location of the larger components are restricted by their heights. These layouts will usually take much longer because of a lack of component placement flexibility and the need to comply with numerous design requirements. Redesigning a circuit using more expensive specialized components in order to reduce the required footprint is sometimes a necessary tradeoff when all else fails.

Devices that use high voltages have the constraint of needing to maintain a substantial isolation voltage between signals or mechanical assembly features. Spending additional time on placement and routing should be expected with these designs as well. Tradeoff decisions, if they are considered, will also address quality and expense usually within the circuit design itself.

Small spaces require careful planning.

TIME IS MONEY VS. MONEY IS MONEY

Board designs that end up mounted inside of large pieces of equipment are not usually constrained by the amount of space available. In these situations, there are a lot of tradeoffs that can drive the size of the PCB's outer dimensions. Many designers will try to reduce the cost of the bare board by prioritizing a smaller board size when doing part placement. This is understandable, since it's logical to not want to pay for things that you don't need. That said, it can be a counterproductive use of a designer's time if they fixate too much on space saving parts placement when it isn't a necessity. The temptation to cram everything together can lead to problems and lost opportunities down the road, and the repercussions are not obvious at the time.

Pay attention to cost vs. benefit with respect to time spent adjusting your layout.

PART PLACEMENT FORM AND FUNCTION

When working on circuit boards that will be manufactured in low quantity but have a high probability of requiring intensive troubleshooting, it is good practice to prioritize arranging components in a geometric fashion that reflects the functionality of the circuit than trying to save space. These tend to be boards used in experimental machinery or custom manufacturing equipment. This practice helps with troubleshooting since the function of each part is more obvious. Keeping similar blocks of circuitry arranged in a duplicated fashion will probably use more board real estate, but you won't waste time figuring out which field effect transistor does what from one section to the next. Here, the small expense of more board space can be a trade-off for a big headache.

In general, when you're grouping the initial blocks of components, if possible, it's a good idea to try to leave multiple paths to route each signal. That may mean backing off a pull-up resistor a tad or moving the decoupling cap to the bottom layer, but it will make your life easier in the long run. Of course, there are exceptions, such as oscillator circuits and high speed data components, whose distance from part to part has to be minimized because of noise considerations. However, backing off a bit where you can will help you avoid getting boxed in during the routing process. That may sound like simple common sense, but it's easy to overlook in the early stages of part placement.

PLANNING FOR THE NEXT REVISION

It is a good practice to ensure that your PCB's layout is flexible for redesign, especially if your design is meant intended to have a long lifecycle. Sooner or later, every PCB in a continuing product line will require a redesign as a result of part obsolescence. Having to move unrelated components so that you can replace an obsolete part adds risk to a design change. Even for new designs it is possible to choose a part that is obsolete to begin with when relying on a standard parts library. Checking parts using Online Search is a good way to ensure that this doesn't happen. Altium's BOM management tools have advanced parts selection and the BOM tool integrates seamlessly with Altium's software. Using it is a great way to increase your productivity and improve quality for all of your

board designs.

As discussed above, specialized parts can be necessary in a design, but they have a higher likelihood of becoming obsolete. Being cognizant of this when doing your layout can help your placement orientations accommodate the need to change the route of a signal in the future, even when parts are close together. Historical lifecycle data helps you to determine in advance which areas of a circuit might be better suited for a specialized part and which areas of the circuit should stick with standard parts. The tradeoff of taking a small amount of time upfront to use these convenient tools can save a lot of time, money and extra work down the road.

Have a question about part obsolescence? Contact an expert at Altium.

ADDITIONAL RESOURCES

Thank you for reading our guide on PCB Layout. To read more Altium resources, visit the Altium resource center here or join the discussion at the bottom of each original blog post:

- Syncing Your Schematics and PCB Layouts Increases Efficiency and On-Time Delivery
- How Proper PCB Layout Can Help You Avoid Adding Heat Sinks to Your Design
- The Most Important Features for PCB Layout Software Comparison
- How to Find the Best PCB Layout Help for Your Specific Design Questions
- Using Customizable Grids to Save Time on Your PCB Layout Process
- PCB Layout Constraints That Help Manage Component Obsolescence