Altium®

Multiboard vs. Multilayer PCB Design Systems

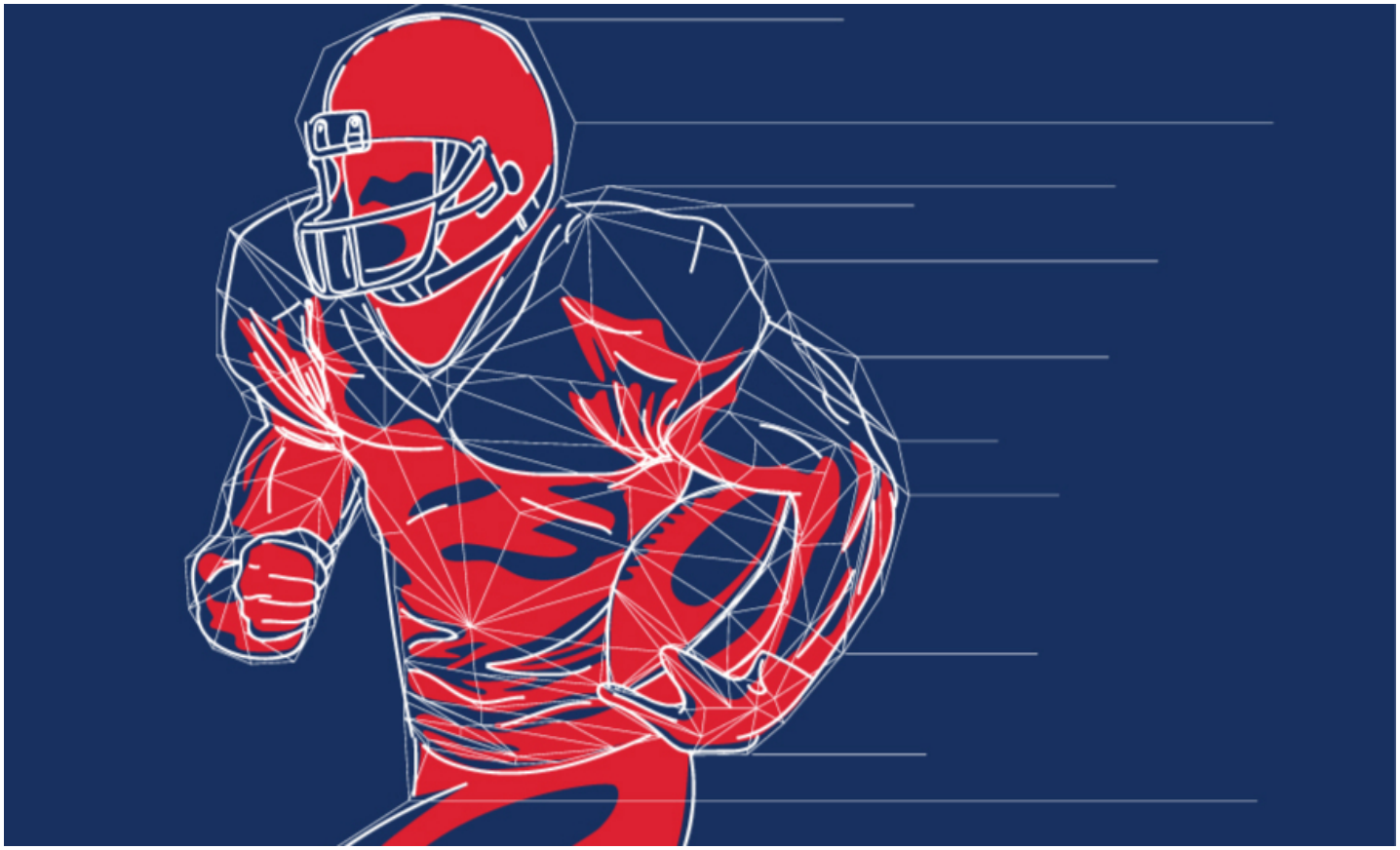
MULTIBOARD VS. MULTILAYER PCB DESIGN SYSTEMS

In the past, PCB designers were limited with system designs spanning more than one board. However, recent developments in mechanical CAD systems have given designers the ability to now design multiple boards for a system with relative ease. Thanks to 3D PCB design tool technology, designers are able to save both time and money previously spent on prototypes by confirming that their designs are system-ready

Join us as we discuss a variety of topics to help you with Multiboard vs. Multilayer PCB Design Systems including:

- Step Up Your Game: Multi-Board Versus Multilayer Designs
- The Best Tools for Multi-Board PCB Systems Design - SLP
- Finding the Perfect World for Multilayer PCB Design
- A Big Kick in PCB Interconnect Solutions: Multi-Board Best Practices
- SpaceX and Other Advanced Technologies Require Multi-Board PCB Innovations
- Annular Rings and Multilayer PCB Design: Stay Within Your Tolerances

STEP UP YOUR GAME: MULTI-BOARD VERSUS MULTILAYER DESIGNS



I don't remember much about playing football in high school. I'm not sure if that's because of wanting to forget the endless grueling practices to prepare us for getting pummeled in the game, or actually getting pummeled in the game. Either way, I tend to vaguely associate high school football with lots of hard work and pain, punctuated by the occasional moments of glory. If there is one thing that I do remember very vividly, it was the coaches encouraging us to always "step up our game".

Our coaches knew that the goal wasn't a matter of arriving at a plateau, but instead constantly advancing to the next level. To prepare us, therefore, for whatever was coming next, they pounded "step up your game" into our heads every moment that they could, and I'm actually grateful for that. It helped me through my teenage year and continues to help me today. In my career of designing circuit boards, I have had to step up my game many times. I've seen advances from thru-hole technology to surface mount, and from wide open designs to high density interconnect (HDI) designs. Now the next step has arrived: moving up from designing single multilayer boards to designing multi-board systems.

Those of us in the design world are familiar with designing a single printed circuit board, but designing multiple boards for a system is probably something new to many of us. Fortunately, there are new and enhanced PCB design CAD tools that can help with this task. Whereas designing PCBs for a multi-board system used to be a clumsy manual process, these new tools provide a design

solution that is eloquent and actually fun to use. Keep reading and you'll see what I mean.

Today's PCB designs often require designing all of the system boards together

REMEMBERING THE OLD WAYS

System level design is nothing new, but how we design the different circuit boards that make up a system is. For years, PCB design was limited to designing one board at a time as there wasn't a CAD solution that would design multiple system boards together. The mechanical design of the system would dictate the size and shapes of the different circuit boards that would fit into it, but the boards would be designed individually.

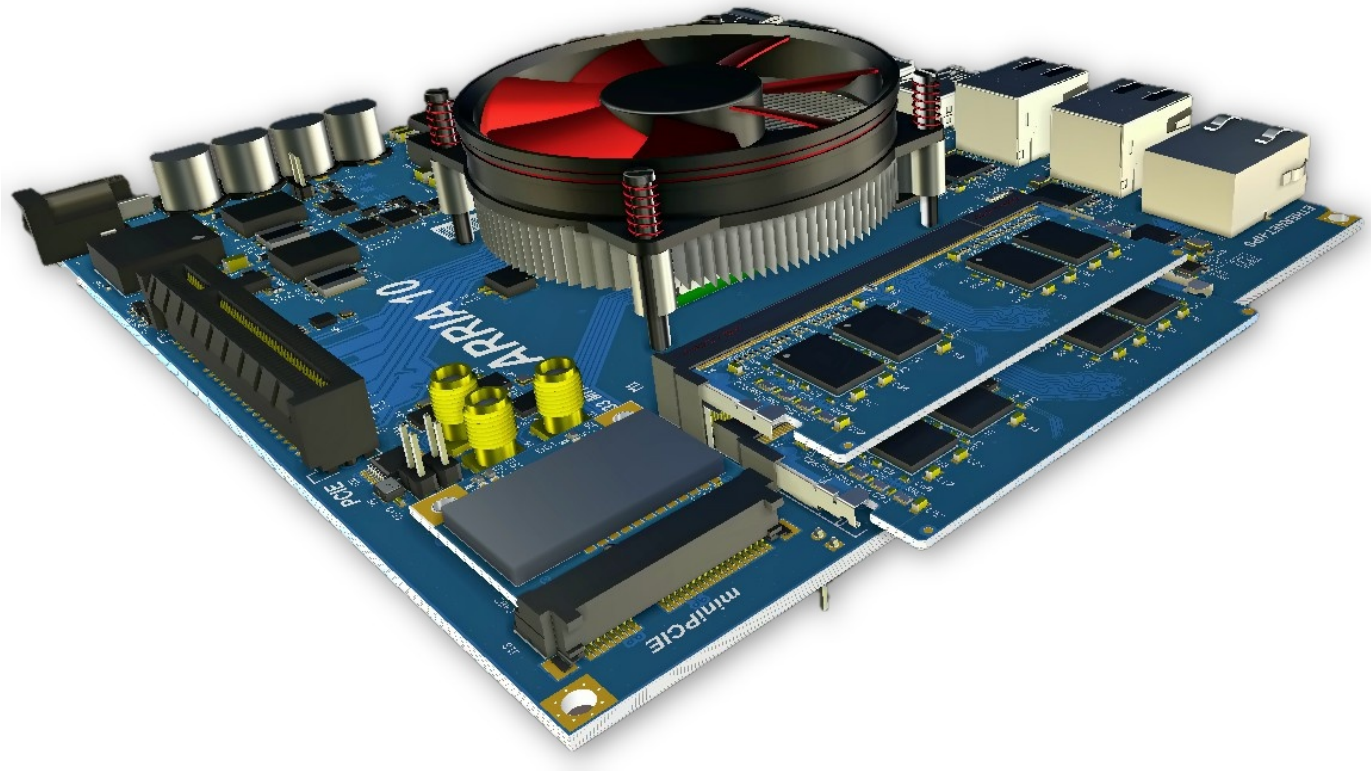
After individual PCBs were designed, they would be built as prototypes. Once these prototypes were up and running, they would then be put together as an entire system. This would allow system level electrical and mechanical checking to be run. Any problems discovered would have to be sent back to the design engineers for another spin of the boards. I have worked on designs in systems

like this that required major changes to board outline shapes and component placement in order to make the board work with the rest of the system. It was a tedious process, to say the least.

A STEP IN THE RIGHT DIRECTION

As 3D mechanical CAD systems grew in their abilities, they became an important part of the design process. They could create the board outline shape with the appropriate keepout and mechanical clearance zones, and send that data to the PCB design CAD system. Once the printed circuit board was designed, they could then read the PCB design placement and library files in order to accurately depict the populated circuit board within the system.

Mechanical CAD systems were a major step forward towards total system design. Design teams could now check for physical conflicts between the board and the rest of the system before building prototype boards. However, there still wasn't a way to verify electrical connections, and all changes and corrections still had to be sent back to the individual PCB designs.



Multi-board design within a 3D PCB CAD system

BEYOND MULTILAYER DESIGNS: MULTI-BOARD DESIGNS AT THE PCB DESIGN LEVEL

Today, system level design is a whole lot easier due to the ability to create multi-boards at the PCB design level. Where PCB design systems in the past could only create individual designs, you can now link together multiple board designs in one multi-board project. This makes it possible for individual board designs to be treated as child objects within the main system assembly.

Interconnections between the boards are identified for electrical checking by assigning multi-board properties to connecting nets on the schematic. In this way, the PCB design system knows how the system boards are connected just as if they are physically connected together in the actual system. This facilitates electrical checking to verify that intra-board connectors have the correct pin assignments.

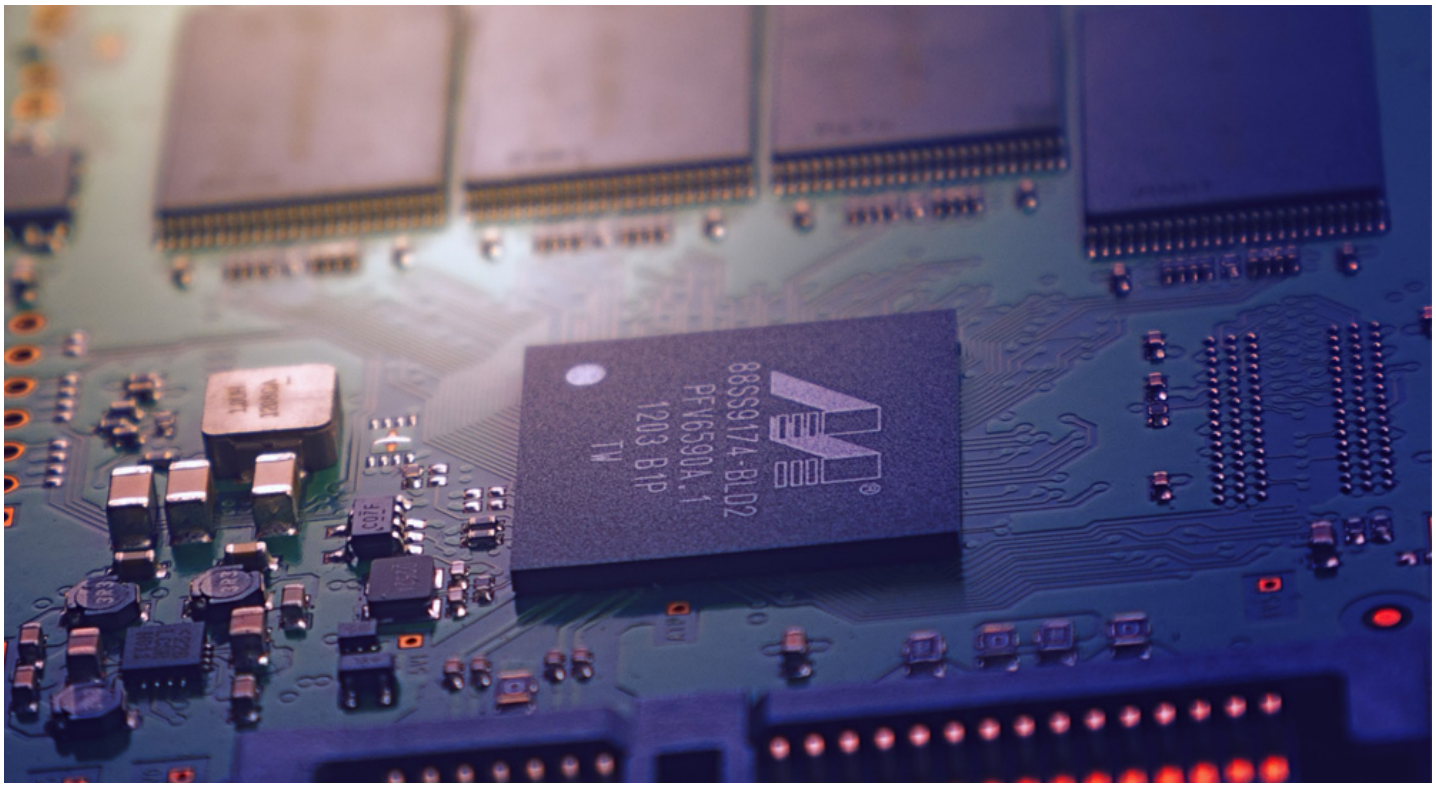
In the 3D environment of the PCB layout tools, the different system boards can be brought together within their enclosure. This allows for physical conflict clearance checking as well as checking for the correct location of interconnecting components such as plugs and connectors. Any changes that are required can be made at the 3D system level in order to verify that all connectors are aligned correctly and any physical conflicts have been resolved.

Multi-board design in today's PCB design CAD tools is not just a step up in your game; it is a complete game changer. You will save time and money previously spent on system prototypes by confirming beforehand that your circuit boards are system ready.

Altium Designer 18 is a top-of-the-line PCB design software that is made for multi-board design. It has a 64-bit architecture and other performance enhancements that will enable you to create your different system PCB designs all within a single multi-board design project.

Would you like to find out more about how Altium can help you to move into a multi-board PCB design system? Talk to an expert at Altium.

THE BEST TOOLS FOR MULTI-BOARD PCB SYSTEMS DESIGN



They say that time flies when you're having fun. Maybe that's why I feel like time slows to a crawl when I'm constructing a multi-board PCB device. Designing even one board can be an agonizing task, what with trying to avoid EMI, ground everything properly, mitigate static discharge, and optimize your routing. Creating a system of PCBs that all fit together physically and electrically sometimes tests the limits of tedium. Lucky for us there are tools that are just on the horizon that will make creating multi-board schemes more of a pleasantry than a punishment. There are three kinds of features that help with PCB design in general, but are particularly useful when it comes to multi-board PCBs: board to board connectivity, MCAD integration, and modular organization. Interboard rule checking can be a nightmare, and can even lead to scrapped boards. This tool makes tracing connections throughout the various boards simple and can help you when last minute changes come up. Interactive models can ease the hassle of clearance checking and making sure that everything fits together and into the enclosure. Lastly, modular organization assist you when you're creating new circuits that could use boards or connectors that you designed in the past.

MCAD INTEGRATION

Having a 3D modeling tool incorporated into your program may not seem like a big deal, but it can make a huge difference in your PCB design process. Using MCAD to model your PCBs can stop you from making expensive mistakes. It can also put your mind at ease when you're sending off a board for prototyping or production. When that same kind of tool is at your fingertips, instead of those of a mechanical engineer somewhere else, it will totally revolutionize the way you PCB circuit design boards. Let's look at how it does this:

3D Clearance Checking

Have you ever designed a board and found out the expensive prototype, or maybe even a production run, didn't fit inside the enclosure? That happened to me once, after I'd added an electrolytic capacitor to a design at the last second before sending it off to be prototyped. It turned out my clearance calculations had been a tiny bit off. So I had to send the designs off the mechanical guys to model and check it, then back again for mock-up. Imagine making that kind of mistake on a production run - good luck explaining that to your boss. The stakes are high when it comes to clearance checking, especially when you're working with multiple PCBs. Each one has to fit together, like a very expensive 3D puzzle. All of that then has to fit into a custom enclosure, which may or may not have been designed by people at your company. I consider myself a good designer, but even I make mistakes that are difficult to catch without a 3D model of my system.

Modeling at Your Fingertips

By now most of us use computer models of our boards, but we're often not the ones making them. That's generally the domain of the mechanical engineers. What if, though, you could be the one making models and doing clearance checking yourself, without having to send off changes over and over to someone else? I'm not talking about you learning a standalone MCAD program and literally doing the work yourself. I'm talking about your circuit board software doing the work for you. That's what a great, well integrated, MCAD tool will do. There are tools available now that can generate 3D models of individual components and create models of your board with everything on it. The best ones will also allow you to import a 3D model of your enclosure and do clearance checking with that as well. This kind of feature allows you to cut out the middleman when it comes to MCAD. Need to see if an electrolytic capacitor fits? A few button clicks and you'll see.

INTERBOARD ELECTRICAL RULE CHECKING

Checking electrical rules isn't usually a big hassle because your software will do it for you at a board level. However, It can be nearly impossible to keep track of connections when they cross from board to board. There are very few programs that can check an entire

system as a whole to ensure that everything fits together electrically and mechanically. This kind of tool will help you keep all your ducks in a row during design by allowing you to do rule checking between various boards. It can also help you a great deal during re-design, when an enclosure change or other external factor forces you to make significant connection alterations on your PCBs.

Error Checking During Development

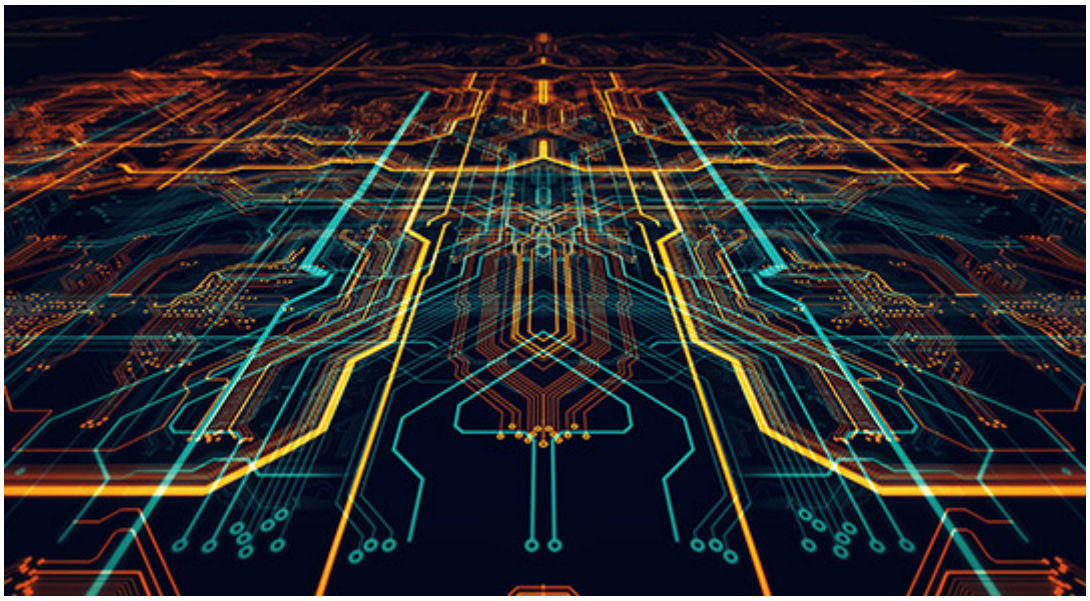
There's nothing more tedious than tracing a net through your system, board by board, to ensure that it makes all the right connections. It makes me feel like a dog chasing its own tail. I go along until I find an error, fix it, then start tracing everything back from the beginning. A good multi-board PCB systems design program will not only help you trace connections through your system, but automatically notify you when things aren't quite right. You need to know at the beginning if one of your routes has a dead end, not just when you're reviewing everything near the end of design.

EMI Reduction

Your software should also allow you to highlight your nets so you can see where they all go. That kind of feature is especially helpful when dealing with EMI. FCC certification can be a hassle, and you don't want to have to go through radiated or conducted emissions testing twice. There are so many interference concerns that come with routing signals across boards. You need to be careful with high speed traces, differential pairs, and the analog and digital portions of your PCBs. Proper grounding is also essential as you're probably not going to be able to use a star ground, and will have to watch out for current return paths through your entire system. Keeping track of all of these nuances is nearly impossible without a tool that can show you exactly where all your connections are routed.

Redesign Made Easy

Highlighting nets isn't just useful for EMI reduction, it can also end up saving your design. I've been on projects where one PCB had to be re-worked or scrapped entirely due to late game system level changes. Sometimes you can't control when a mechanical engineer will need to tweak the enclosure, and that adjustment may mean you need to move several pins or maybe a bus connection. Doing so without a tool that can highlight your nets can be an exercise in futility. However, if you have software that can show you where every trace is going, what was impossible now becomes simple.



MODULAR ORGANIZATION

At first glance this feature may not excite you as much as MCAD integration or system level electrical rule checking. You'll see, however, that it is quite useful and can help you reuse old designs. No more wasting time re-designing the same old PCB that you've made 1,000 times. This kind of tool will allow you to significantly reduce time wastage in your design process and help you focus on new and exciting work.

Standard Designs

King Solomon once said that there's nothing new under the sun, and that's starting to apply to PCBs. If you're designing for areas like the Internet of Things (IoT) you already know many of the different components you're going to need. Of course you'll need a module for WiFi, Bluetooth, or soon maybe even 5G to allow the device to communicate. If you're making a wearable and you want it to be convenient for the user you'll hopefully utilize a micro USB connector instead of devising your own proprietary kind. There's no reason you should have to build the PCBs that these modules sit on and connect to over and over. Great multi-board software will allow you to make you entire system modular, so that you don't have to reinvent the wheel each time you design a new product.

Save Time During Design

There's a lot of work to be done in PCB design, what with the IoT blowing up and connected cars revving their engines. We don't have time to waste in re-hashing the same old designs. PCB design software that allows you to modularize your designs will help you save time that you can use to focus on making new, cutting edge devices. Of course, true modular design and integration isn't really possible without the first two features mentioned. If you import an old design and have to send it off for multiple rounds of modeling and clearance checking you might as well build it from the ground up. Multi-board PCB electrical rule checking will let you import a sub-system and instantly check its connections. It's really the combination of all three of these tools that can revolutionize your design process.

Currently there's only one program out there that's looking to add all of these features together on top of what is already great PCB

design software. Altium Designer has a host of great features that will enable you to create exciting new PCBs. Its tools include those listed here, as Altium is expanding their scope to include multi-board PCB systems design. Individual PCB design has already been re-vamped by Altium, with elements like a PDN analysis tool and increasingly important rigid-flex support.

Have more questions about multi-board design? Call an expert at Altium.

FINDING THE PERFECT WORLD FOR MULTILAYER PCB DESIGN

I have always been a huge Clint Eastwood fan. Dirty Harry always seemed to be a little too quick on the draw for an urban twentieth-century environment, but The Man with No Name—he was perfect for the ruggedness and lawlessness of his time. Obviously, I always rooted for him to beat up the bully, outdraw the outlaw, or get the girl, but the most intriguing thing was that he was usually quick to make a decision. Of course, the results varied. Sometimes they were good, sometimes bad, and sometimes pretty ugly.

The process of creating a PCB can have similar outcomes depending on the decision to use or not to use a multilayer PCB. A number of PCB designers incorporate multilayer boards when it is not necessary. Doing so complicates the design, increases fabrication cost and makes field repair or modification virtually impossible.

In most of the Man with No Name movies, the plot proceeds from some dastardly deed(s) or intolerable burden being inflicted upon the victims by the villains and ends with my hero righting the wrong in the end. Following the same outline, let's look at what can go wrong in the PCB design and development process first and then see what factors we need to incorporate into our decision to design a multilayer PCB or not.

YOUR MULTILAYER PCB HEARTBREAK RIDGE

From a PCB design perspective, it can be quite tempting to use multiple layers when we design PCBs. After all, the notion that "smaller is better" seems to be pervasive throughout electronic design these days. However, there are significant reasons to avoid the smallness trap unless doing so is a primary design consideration. For example:

- Design Complexity: When designing multilayer PCBs, it is imperative to have all through holes and vias line up properly. Mistakes can impact the current flow and create mounting issues. Additionally, using odd numbers of layers or different thicknesses for internal layers may cause bowing or twisting of the board making it unmountable or being relegated to test-only status. Bad. For communications applications, where various signal types are being routed across multiple layers performance issues may arise due to crosstalk or unmatched impedances. Bad.

- Increased Fabrication Cost: Manufacturing multilayer PCBs is significantly more expensive than other boards. More material is required, more time is required, and technicians must be highly skilled. Simply going from two to four layers can increase the fabrication cost by 100%. Bad.

- Difficult or Impossible Bench Repair: As with any manufacturing or fabrication process, small errors do occur. Especially for boards with odd numbers of layers or variable-sized layers. With single (or 2-layer) PCBs, these can usually be easily repaired, and the board may still be usable. If the problem lies with an internal layer, this is practically impossible and the board(s) are useless. Bad.

Another issue that is sometimes overlooked is the increased heat due to the increased layers, which may not show up until the product has been in the field for a while. If severe enough for product failure, this may result in the three R's: recall, redesign and remanufacture. Ugly.

As we can see, opting for a multilayer PCB can result in severe problems, if not done properly. However, there is a bright side. If we are deliberate in our selection and the design process, we can ride off into the sun just like Clint always does.

Take charge of your PCB design process to find your own sunset.

ABSOLUTE, MULTILAYER POWER

Now, if you are willing to spend the extra time on design and fabrication, pay the higher cost, and follow the best multilayer PCB design guidelines, then you should proceed with the multilayer PCB option. Especially, if size is the primary concern. Let's assume that it is not. What reasons are to justify using multilayer PCBs?

- Functionality: Multilayer PCBs allow for the incorporation of more complex circuitry. Therefore, more functioning can be built into a smaller package. The marketing advantages of this are quite evident. Good.

- Durability: Due to care and additional considerations that must be incorporated into multilayer PCB design and development, they are better constructed components and are highly reliable. Good.

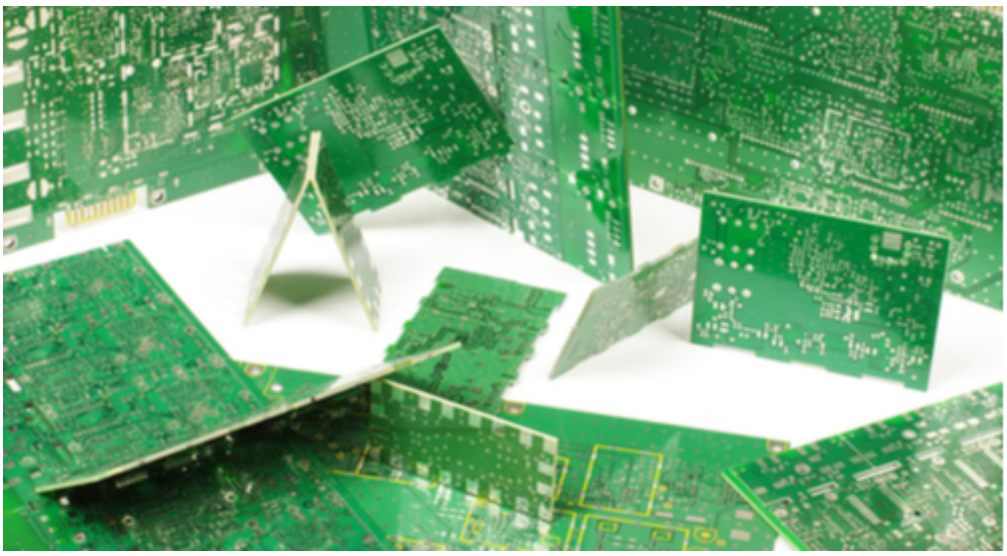
- Mountability: The smaller the PCB, the easier it is to mount. It requires less mounting holes and may be placed in more locations within the larger system. In fact, it may allow for reduction of the size of the overall product, which can increase portability, storage options and marketability. Good.

TRUE DESIGN

Multilayer boards carry a higher cost, longer fabrication time and require more expertise to design and manufacture. Therefore, deliberate consideration should be undertaken prior to determine if multilayer is the way to go. When deciding on whether to use multilayer PCBs, do not simply follow the "smallness" crowd. The results may be bad or quite ugly. Instead, base your choice on sound decisions that soberly weigh the advantages and disadvantages to ensure a good result.

NO: SOME PCB DESIGNS ARE SIMPLY BETTER SUITED FOR SINGLE-LAYER LAYOUT. A GOOD EXAMPLE IS A COMPUTER MOTHERBOARD. THESE BOARDS TYPICALLY HAVE MANY PORTS OR ASSORTED SIZES, NOT TO MENTION DETACHABLE PARTS THAT NEED TO BE ACCESSIBLE. THIS DOES NOT RENDER THE USE OF MULTILAYER DESIGN IMPOSSIBLE; HOWEVER, IT IS IMPRACTICAL AS THE BENEFITS OF REDUCED SIZE DO NOT OUTWEIGH THE DISADVANTAGES OF COMPLEXITY AND INCREASED COST.

Yes: The most common reason to opt for a multilevel PCB design is to reduce the footprint of the board. In situations, where the design flexibility is not bounded by the number of inputs and outputs, fanout requirements for single ICs (i.e. MPUs, FPGAs, etc.) or similar trace length limitations then multilayer may be the way to go. This may be especially true when the product allows for the use of flexible circuits, which remove much of the precision difficulty from the fabrication process while giving more freedom to the designer.

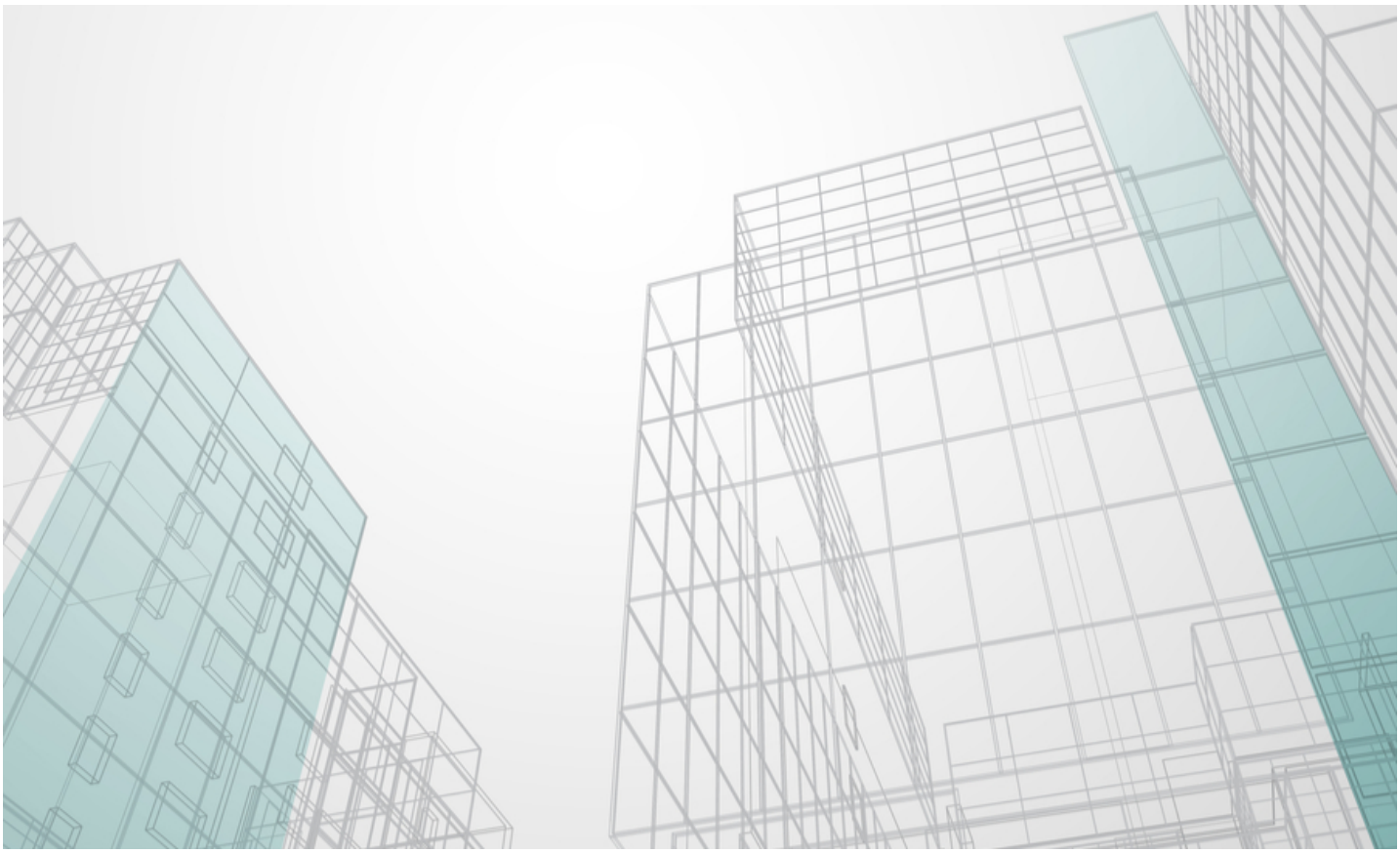


Don't settle for using the same PCB for every design, make sure to choose what works best for you.

For single, double, or multilayer PCBs, Altium Designer's smooth and efficient software will enable you to make the most effective and accurate design with your PCB.

If you are unsure and want to discuss which PCB would be best for your design project, the experts at Altium are just a phone call away.

A BIG KICK IN PCB INTERCONNECT SOLUTIONS: MULTI-BOARD BEST PRACTICES



I've always enjoyed thinking about circuits similar to tiny cities. Copper routing as the streets, components as building structures, and the following electrons as the inhabitants running around. But imagine what PCB would be in the city from Inception where the flat ground starts turning on its head and extending overtop, or if Salvador Dali became a city-planner.

In today's world of densely packed components, advanced manufacturing capabilities, and wonky product shapes, you may find yourself at a loss for horizontal real estate and looking into vertical space. More specifically, you may come across a situation in which you need to separate your PCB into two, three or even twelve boards. The beauty of your PCB-city comes in when you can start producing a multi-board PCB design.

Whether it be your consideration of space within a product or to keep signal isolated from one another, you'll likely need your boards to talk to each other. Enter PCB interconnection: the practice of running physical, non-printed cables between multiple printed boards in order to achieve a fully connected circuit. With the city laid out (and a three-hour break for Inception), we can know more about the situations requiring our multi-board PCB cities to bend and stack.

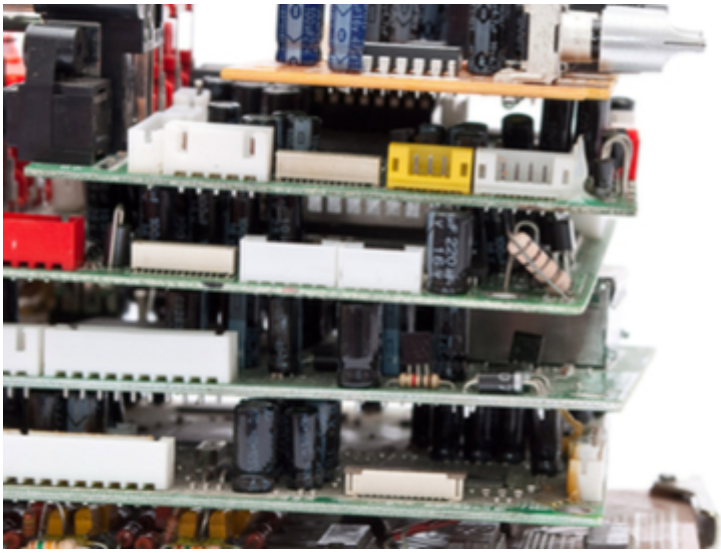
FIRST PCB INTERCONNECT SOLUTIONS: HOW'S IT LOOK?

Some of the more common stacking techniques involve a vertical stack, one on top of another. This is personally my favorite method to see implemented as the final product has potential to look like a skyscraper with the floors as PCBs. What PCB city is complete without PCB skyscrapers?

When stacking any boards, similar to designing a single PCB, you'll need to take care of the orientation of signal lines that are running through. Crosstalk, impedance discontinuities, and electromagnetic emissions are real issues that will bite you if you do not pay heed. To mitigate these issues, here are some quick tips to incorporate into your considerations:

1. All signal layers should be adjacent and closely-coupled to an uninterrupted reference plane.
2. Reduce AC impedance at the top end and dramatically reduce electromagnetic radiation with closely-coupled planes.
3. Further reduce radiation by routing high-speed signals between the planes.

In addition to these tips, you'll also need to appreciate the space that comes with the physical, non-printed interconnect cables or pins that will connect your boards together.



Careful not to topple your PCB stack, that's a lot of hard work

DESIGNING FOR MANUFACTURING AND SERVICEABILITY

Just with any design, you'll want to make sure you plan out your board for its full livelihood: will your design be lasting years or decades? How often will your board require updating and maintenance? Maintaining strong communication with your manufacturer will allow you to keep your design working far into the future. When in search of manufacturing facilities, ensure you do your due

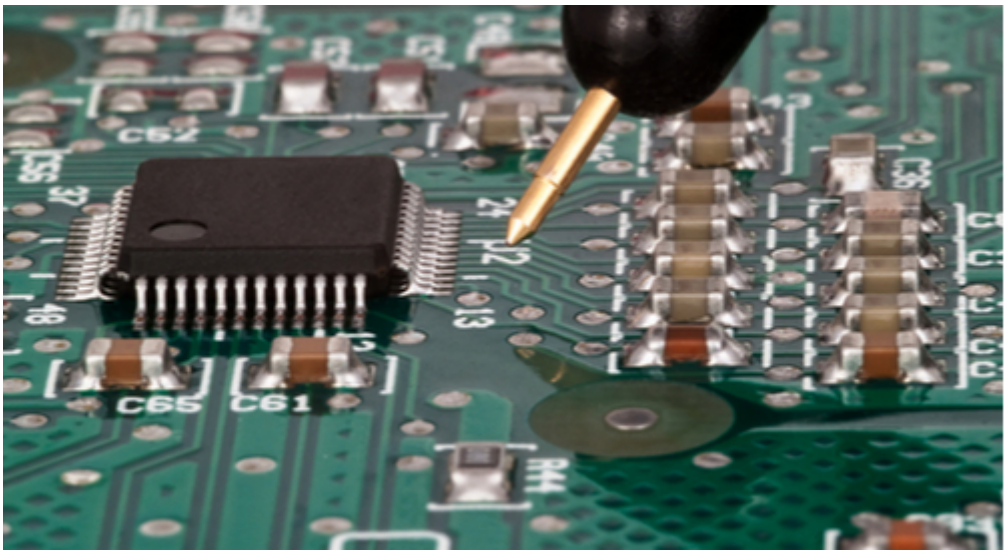
diligence and attain their limitations and requirements for manufacturing.

Space requirements needed to manufacture your product will need to be kept in mind. In addition, if your product is maintained regularly or if your product breaks but is designed for repair, then physical space required to carry out such procedures should also be understood and designed for such. Again, don't forget about the interconnect cables that may or may not run between the boards.

ENSURING YOUR PCB DOESN'T COLLAPSE

Often an afterthought, testing procedures of your PCB design will, of course, need to be implemented at some point if you are to keep any amount of quality assurance in your flow. From producing prototypes to test in various conditions to determine any potential overlooked design flaws to using your software tools to simulate and run consistent, reliable testing on rule checking as well as product analysis, your PCB tests need to be implemented.

Testing procedures involving the use of probing can be a physical challenge when you are vertically stacked. Keeping in mind the space requirements of not just your product's chassis, but your testing apparatus too will be a crucial step when it comes time to test. It's like trying to make a sandwich out of your PCB city with PCB skyscrapers—you have to prepare it properly.



A testing apparatus of PCBs require space that may not be present in your current multi-board PCB design

A BUTTER KNIFE WON'T CUT A TREE, USE THE RIGHT SOFTWARE

While your PCB multi-board can be done, painstakingly, with tools as primitive as a pencil and paper; you're going to want to utilize the design software which can make your process smooth and intuitive. After all, software is something that you'll be working in the

majority of the time in your designs—why shouldn't you be using software that you both like and find incredibly helpful?

For multi-board systems specifically, I find using software that has features like multi-board PCB signal analysis can be extremely beneficial. Your design as a whole will be acting as a single PCB, so being able to properly analyze the separately communicating parts as a single PCB will be critical to maintaining your product's integrity. Furthermore, 3D visualization capabilities also assist when determining the physical stacking of your boards.

MULTI-BOARD PCB DESIGNS AREN'T YOUR ENEMY

Without over-thinking the process, designing Multi-board PCBs with interconnects are fantastic solutions for overcoming space requirements, isolating signals, or simply making your design look as cool as you imagine. By keeping in mind the purposes and utilities of a multi-board system, as well as understanding the full effect of your PCB design software on your capability for smooth layout, you'll be able to build your PCB city in no time.

With the software tools listed above all wrapped into one unified design environment, Altium Designer will empower you to smooth out your design and keep your city looking like Manhattan instead of Inception. After all, the goal for your PCB software is to make your design process as intuitive as it can be, with as little time spent outside of the software and consulting other tools and services as possible.

If you would like to discuss how Altium can assist in your interconnected multi-board design, talk to an Altium expert today.

SPACEX AND OTHER ADVANCED TECHNOLOGIES REQUIRE MULTI BOARD PCB INNOVATIONS

When I was in middle school I used to love to build and fly model rockets. We had a rocket club at school and except for the number of rockets that I littered the trees with, it was a lot of fun. One thing was certain though, although the rockets we launched went up, no one knew exactly where they might come down. Suspended by their parachutes (assuming that those chutes actually deployed as intended), they might land anywhere.

Recently I watched as the two side boosters of the Falcon Heavy from SpaceX executed perfect simultaneous landings under their own power. These boosters were not dangling under parachutes with the hope of hitting near to their landing zones, they used their remaining fuel to soft-land exactly where they were programmed to. The lesson that I learned as a middle schooler, that rockets go up but who knows where they'll come down, had been redefined.

For years I've been designing printed circuit boards, and I've always designed them one board at a time. It was the only way to do it, as multi-board system level design was beyond what our CAD systems could do. Today though, that fact has also been redefined.

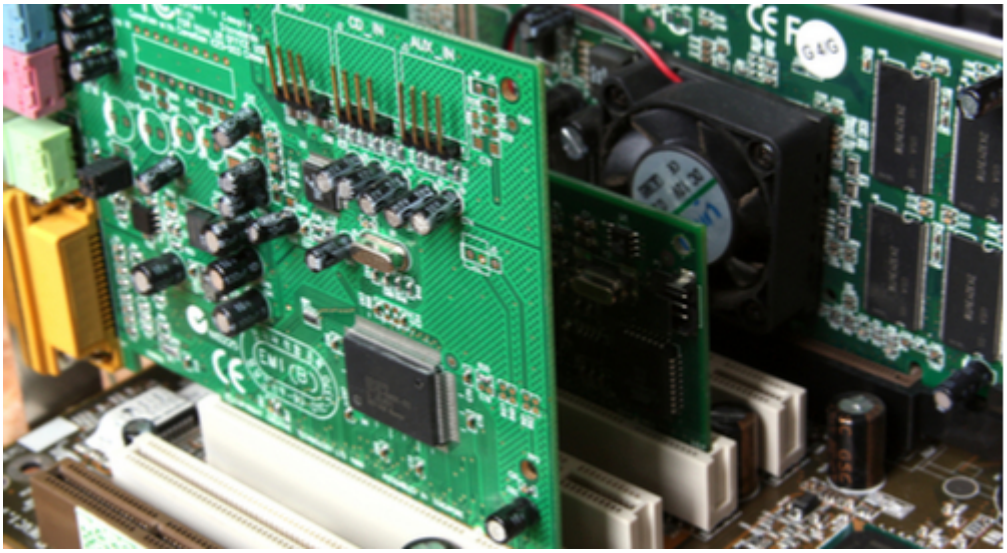
As technology evolves, our design methodology needs to change along with it so that we are ready for it. We now have the ability to design multiple boards within a project to give us system-level design. This will give you significant advantages over single board

design. System-level design like this is going to be essential as we design the latest advancements in technology exemplified by the Falcon Heavy and other aerospace applications.

MORE THAN MEETS THE BOARD: MULTI-BOARD PCB INNOVATIONS

The standard for PCB design has always been to design one printed circuit board at a time. These designs were often part of multi-board systems, but due to the limitations of our PCB CAD tools we only had visibility into the design that we were currently working on. Although we knew that there were other designs involved, we couldn't see them and had to rely on input from other sources to match our work up with the rest of the system.

The mechanical engineers would give us board outlines with keepout zones so that we could avoid potential placement conflicts. This would tell us where not to place components over a certain height, or in some cases not to place any components at all. In the same way, the electrical engineer would match up net names from different designs and the schematics would indicate which connectors would pair up between designs.



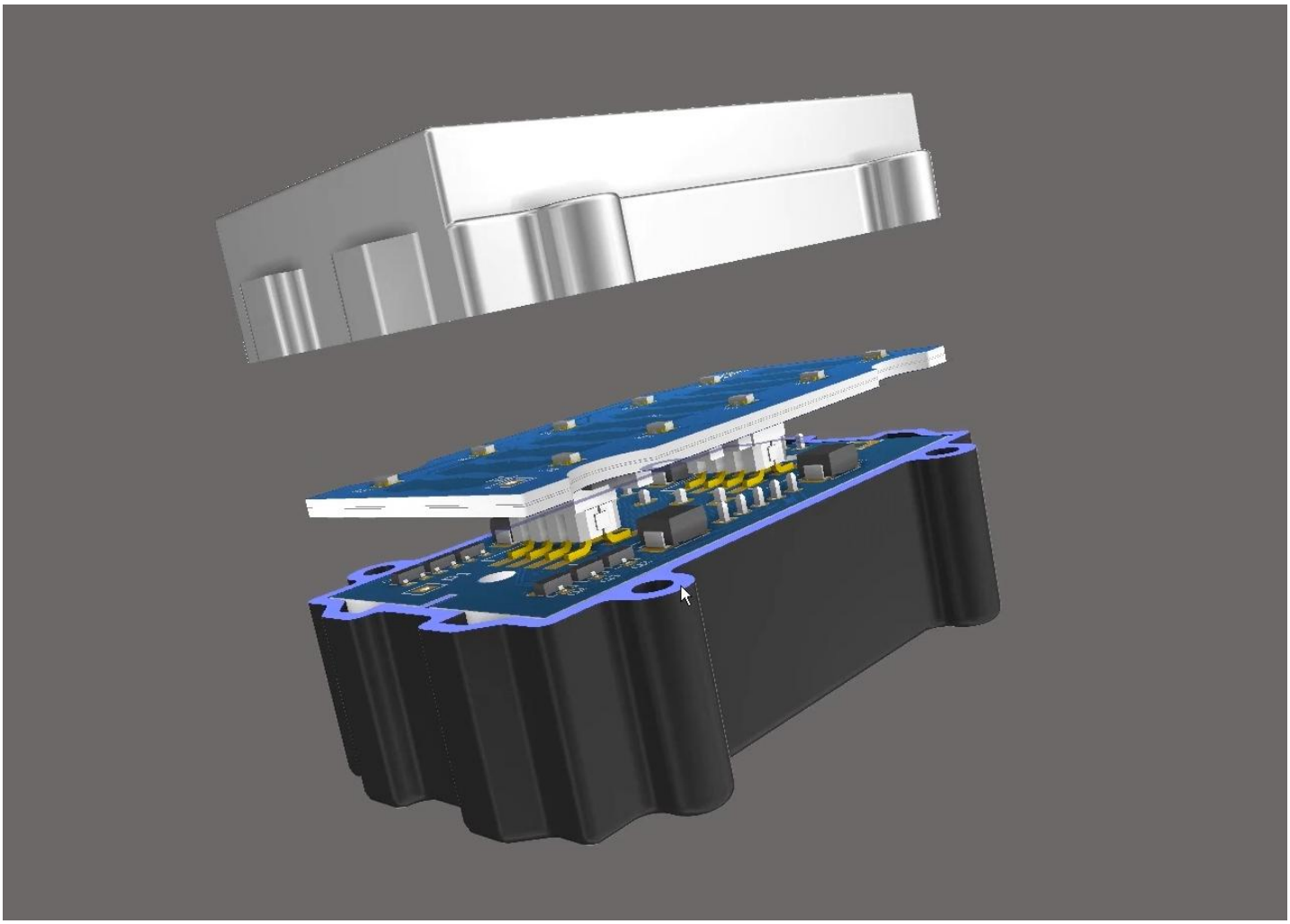
PCB design for multiple system boards can be a challenge when done individually

THE CHALLENGES OF DESIGNING MULTIPLE BOARDS FOR A SYSTEM

The input from mechanical and electrical engineers went a long way towards helping us to design a single PCB so that it worked together with the rest of the system boards. The drawback, of course, was that those directions were still a little vague. It was sort of like driving down the road and seeing a "caution" sign. What are you supposed to be cautious about? We needed more details in order to do a complete job in design.

Take placing connectors for instance. It is easy to place a connector at a specified location, but without the ability to look at it in relation to what it is connecting too you may miss some important design details that could be cleaned up. Is the connector in the best location? Will the connector harness reach the way it is supposed to? With the harness plugged into the connector will that cause any unforeseen difficulties with the surrounding placement?

Another area of design that was a challenge was in designing the circuitry. It is a regular function of PCB design to create good inter-board connectivity with placement and routing. It is more difficult though to design for intra-board connectivity without the ability to work with multiple designs simultaneously. What may seem like a great placement and routing scheme on an individual board may change a little when looked at from a multi-board system perspective.



The ability to design multi-boards in a 3D environment is a big help for system-level design

HOW DESIGN TOOLS WITH MULTI-BOARD PCB INNOVATIONS CAN HELP

Fortunately, there are PCB design tools today that allow you to design from a system level perspective. By starting at the project level,

you can designate the individual PCB designs below that for all of your system boards. This gives you the ability to specify net names that will cross between the designs so that you can work with consistent net name connectivity between designs.

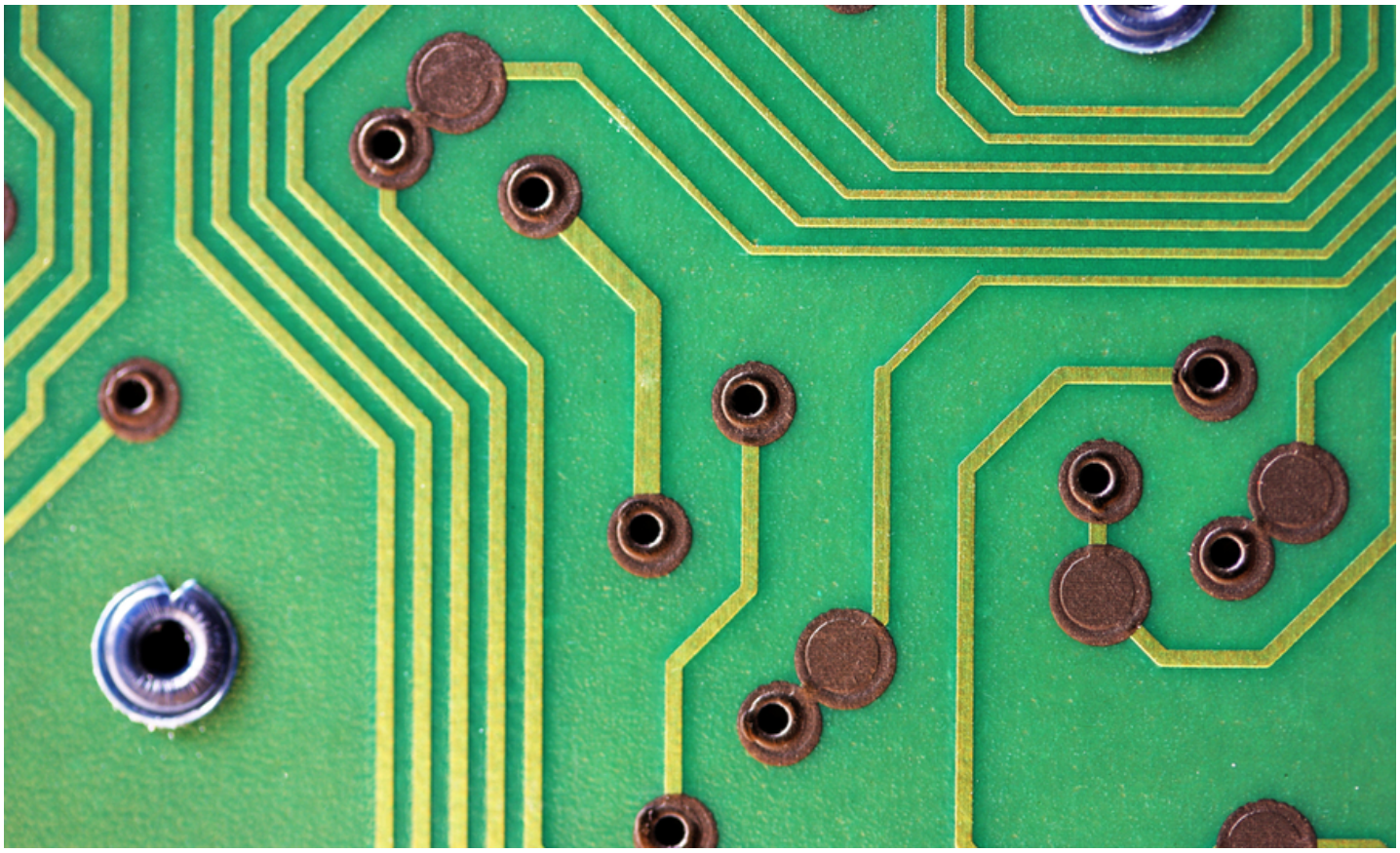
On the layout side, you can either work on designs individually, or you can work at the project level bringing them all together as they would appear in the final system. This gives you the ability to adjust your component placement according to the needs of the actual system instead of relying only on "keepout" zones.

Multi-Board PCB design systems like this can make the difference between designing in the blind for a system and actually designing at the system level. With the ability to design within a native 3D environment in the Multi-Board PCB design system, you can see where placement conflicts are and make real-time corrections in order to resolve the problem.

For both the ability to design at the system level and to work in those designs within a 3D environment, PCB design software such as Altium Designer 18 can be pivotal to your design success. You will be able to see how all of the system boards in your project will work together allowing you to make design changes as needed to resolve layout conflicts.

Would you like to find out more about how Altium can help you with your next system-level design? Talk to an expert at Altium.

ANNULAR RINGS AND MULTILAYER PCB DESIGN: STAY WITHIN YOUR TOLERANCES



One of my high school teachers had a pretty successful business for when he wasn't teaching: he harvested maple syrup. I learned that a bit more goes into the process than simply taking the maple syrup from the tree after tapping a hole into it. You've got to let it rest and let some of it evaporate so you can distill the maple syrup into the tasty, flavorful product that will then go onto all of your pancakes.

While there's a whole process attached to making maple syrup, it still starts with that first hole being tapped in the maple tree. Similarly, there's a whole process attached to making multilayered PCB designs work; however, they would be nowhere without vias. Vias are essentially a vertically drilled shaft that bridges the gap in between any number of layers.

But, even vias have considerations that need to be followed or your design may encounter failures. Annular rings come into play when considering the physical location of the drill holes through each via trace. There are a few factors that come into play when determining the proper annular ring size for your application.

ANNULAR RING CONNECTION STRENGTH: VIAS AND MULTILAYER PCBS

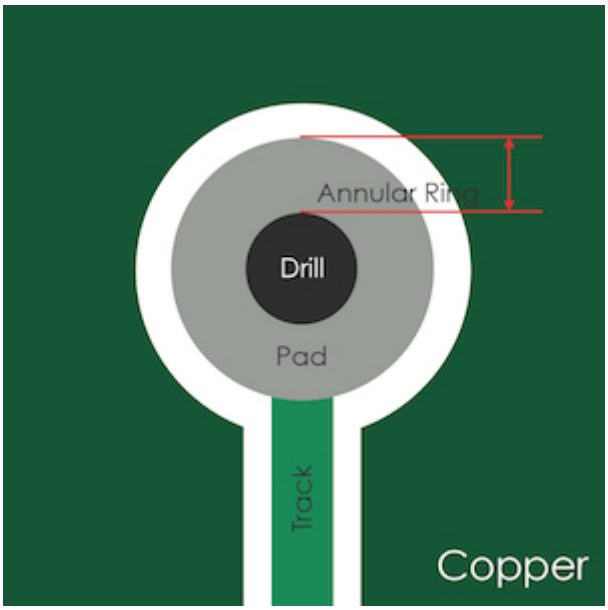
Multilayered PCBs are beneficial to some due to the greater capacity of complexity they can handle. Computers, phones, and medical equipment are some examples of applications in which are benefited via multilayered designs. Working with multilayered PCBs, however, poses a critical issue in connecting these layers to one another. Without connecting each layer to its corresponding points, you'll just end up with multiple single layered PCBs glue to one another.

Enter vias; our ingenious solution to vertically connecting each layer. Vias require an understanding of annular rings to work properly, though. These rings are defined as the minimum distance between the drilled hole and the edge of the via trace. The greater the annular ring, the greater the copper connection around the drill hole will be.

The use of your annular ring will often determine what size you should be shooting for. Are you soldering a component to one or both sides of the board? You'll likely need a larger area for soldering's sake. Are you simply using this via as a testing point and won't solder? A smaller annular ring size will get you by.

Whatever the application you are using with your annular ring, you can easily determine the size by referencing our trusty old friend, the IPC-7251. This document recommends an annular ring width of 250 µm for maximum material condition (MMC). MMC simply means you'll have the most robust solder joint. On the other hand, 150 µm is the recommended width of an annular ring to achieve the least material condition (LMC). LMC simply meaning that you'll walk away with the least robust solder joint connection.

Obviously, these are just recommendations and can (and should!) be altered depending on your particular application.



How an annular ring is measured.

MANUFACTURING TOLERANCES FOR YOUR MULTILAYER PCB DESIGN

When the marriage of any multiple manufacturing processes happen within a production environment, there will often be slight errors that overlap due to tolerancing imperfections here and there. Specifically, when the process of etching the copper traces on your PCB and the process of drilling the vias through said traces come together, your drill holes will often not align completely on the center of these traces and will leave you slightly off-kilter. However, don't fear; tolerances are here!

Since you already should be designing for manufacturing tolerance errors, designing for annular ring tolerance errors are no different. First, by identifying the manufacturer specific tolerances they'll be able to accommodate for errors, and by determining the least amount of width your annular ring can safely be you can the de-risk the whole manufacturing process ensuring your minimum values are always reached.

In short, knowing that there will exist some manufacturing errors along the way but designing for said errors will keep you above your minimum values specifically speaking to annular rings of drilled via holes.

Drilling tolerances can play a huge part in determining your annular ring size.

CALCULATING ANNULAR RING WIDTH

An easy way to verify that your width will be acceptable for your design is by calculating the maximum width you'll be seeing post-production run. The following equation can be used:

((Diameter of the trace pad) - (Diameter of the drilled hole)) / 2 = (Max annular ring width)

The greater the manufacturing errors in tolerance are, the less the annular ring width will be. The greater the diameter of the drilled hole, the less the annular ring width will be.

Knowing that the width of your annular ring should enable a strong enough connection for both electrical and mechanical connections, and being cognizant of manufacturing tolerancing will keep your annular ring width to acceptable distances and out of situations where the drilled hole doesn't even touch the trace pad (heaven help us).

As vias are being designed into microvia territory, you'll need PCB design software which can handle adequately designating annular ring widths and manufacturing tolerances for vias. Thankfully, Altium Designer can approach the sensitive layout complications with ease with their complete list of design rule checking as well as with intuitive board layout software.

So go forth and design a strong and error-free annular ring that will keep your components on for the longevity of your design! If you are curious about what Altium software can do for your annular ring issues, talk to an Altium expert today.

ADDITIONAL RESOURCES

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

- Step Up Your Game: Multi-Board Versus Multilayer Designs
- The Best Tools for Multi-Board PCB Systems Design - SLP
- Finding the Perfect World for Multilayer PCB Design
- A Big Kick in PCB Interconnect Solutions: Multi-Board Best Practices
- SpaceX and Other Advanced Technologies Require Multi-Board PCB Innovations
- Annular Rings and Multilayer PCB Design: Stay Within Your Tolerances