



Altium

PCB Design Systems

Multi-board vs. Multilayer

1. MULTI-BOARD VERSUS MULTILAYER DESIGNS
2. THE BEST TOOLS FOR MULTI-BOARD PCB SYSTEMS DESIGN
3. FINDING THE PERFECT WORLD FOR MULTILAYER PCB DESIGN
4. PCB INTERCONNECT SOLUTIONS MULTI-BOARD BEST PRACTICES
5. ADVANCED TECHNOLOGIES REQUIRE MULTI-BOARD PCB INNOVATIONS
6. ANNULAR RINGS AND MULTILAYER PCB DESIGN

1

Multi-Board Versus Multilayer Designs

Multi-Board Versus Multilayer Designs

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.

Remembering the Old Ways - 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. 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.

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. Mechanical CAD systems were a major step forward towards total system design.

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. Interconnections between the boards are identified for electrical checking by assigning multi-board properties to connecting nets on the schematic. In the 3D environment of the PCB layout tools, the different system boards can be brought together within their enclosure



[See original content here](#)

2

The Best Tools for Multi-Board PCB Systems Design

The Best Tools for Multi-Board PCB

There are 3 kinds of features that help with PCB design , but are particularly useful when it comes to multi-board PCBs- board to board connectivity, MCAD integration, and modular organization

MCAD Integration - Using MCAD to model PCBs can stop from making expensive mistakes. It can also put mind at ease when sending off a board for prototyping or production.

3D Clearance Checking - The stakes are high when it comes to clearance checking, especially when working with multiple PCBs. Each one has to fit together. All of that then has to fit into a custom enclosure.

Modeling at Your Fingertips - There are tools available now that can generate 3D models of individual components and create models of board with everything on it. The best ones will also allow to import a 3D model of enclosure and do clearance checking with that as well.

Inter-board Electrical Rule Checking - Checking electrical rules isn't usually a big hassle because the software will do it 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.



The Best Tools for Multi-Board PCB Systems Design

Error Checking During Development - A good multi-board PCB systems design program will not only help trace connections through the system, but automatically notify when things aren't quite right.

EMI Reduction - Software should also allow to highlight the nets to see where they all go. That kind of feature is especially helpful when dealing with EMI.

Redesign Made Easy - Highlighting nets isn't just useful for EMI reduction, it can also saving the design. If there is a software that can show where every trace is going, what was impossible now becomes simple.

Modular Organization -This kind of tool will allow to significantly reduce time wastage in design process and help focus on new and exciting work.

Standard Designs - Great multi-board software allow to make entire system modular, so that there is no need to reinvent the wheel each time designing a new product.

Save Time During Design - PCB design software that allows to modularize designs will help save time that can be used to focus on making new, cutting edge devices. True modular design and integration isn't really possible without the first two features mentioned. It's really the combination of all three of these tools that can revolutionize design process.



[See original content here](#)

3

Finding the Perfect World for Multilayer PCB Design

Finding the Perfect World for Multilayer PCB Design

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 the decision to design a multilayer PCB or not.

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. For communications applications, where various signal types are being routed across multiple layers performance issues may arise due to crosstalk or unmatched impedances.

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%.

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.



Finding the Perfect World for Multilayer PCB Design

Absolute, Multilayer Power

Functionality- Multilayer PCBs allow for the incorporation of more complex circuitry. Therefore, more functioning can be built into a smaller package.

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.

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.



[See original content here](#)

Finding the Perfect World for Multilayer PCB Design

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, choice should be based on sound decisions that soberly weigh the pros and cons to ensure a good result.



[See original content here](#)

4

PCB Interconnect Solutions Multi-Board Best Practices

Interconnect Solutions Multi-Board Best Practices

Whether it is consideration of space within a product or to keep signal isolated from one another, the boards need to talk to each other.

First PCB Interconnect Solutions: Crosstalk, impedance discontinuities, and electromagnetic emissions are real issues that needs to be mitigated, here are some quick tips to incorporate into considerations.

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



Interconnect Solutions Multi-Board Best Practices

Whether it is consideration of space within a product or to keep signal isolated from one another, the boards need to talk to each other.

Designing for Manufacturing and Serviceability

Maintaining strong communication with manufacturer will allow to keep design working far into the future. Space requirements needed to manufacture product will need to be kept in mind.

In addition, if the product is maintained regularly or if it breaks but is designed for repair, then physical space required to carry out such procedures should also be understood and designed for such.



PCB Interconnect Solutions Multi-Board Best Practices

Ensuring Your PCB Doesn't Collapse - Testing procedures of PCB design will, of course, need to be implemented for quality assurance in the flow. Testing procedures involving the use of probing can be a physical challenge when are vertically stacked. Keeping in mind the space requirements of not just the product's chassis, but testing apparatus too will be a crucial step.

A Butter Knife Won't Cut a Tree, Use the Right Software - Utilize the design software which can make process smooth and intuitive. For multi-board systems specifically, software that has features like multi-board PCB signal analysis can be extremely beneficial.

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 design look cool.



[See original content here](#)

5

Advanced Technologies Require Multi-Board PCB Innovations

Advanced Technologies Require Multi-Board PCB Innovations

Now multiple boards within a project can be designed to give system-level design

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.

The Challenges of Designing Multiple Boards for a System – 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 there could be a miss on some important design details that could be cleaned up. Another area of design that was a challenge was in designing the circuitry.

How Design Tools with Multi-Board PCB Innovations Can Help - Fortunately, there are PCB design tools today that allow to design from a system level perspective. By starting at the project level, designate the individual PCB designs below that for all of system boards. On the layout side, either work on designs individually, or work at the project level bringing them all together as they would appear in the final system.



[See original content here](#)

6

Annular Rings and Multilayer PCB

Annular Rings and Multilayer PCB

Annular rings come into play when considering the physical location of the drill holes through each via trace. Factors that determine the proper annular ring size for 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. Working with multilayered PCBs, however, poses a critical issue in connecting these layers 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.

Manufacturing Tolerances for Your Multilayer PCB Design - 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 annular ring can safely de-risk the whole manufacturing process ensuring minimum values are always reached.



Annular Rings and Multilayer PCB Design- Stay Within Tolerances

Calculating Annular Ring Width - An easy way to verify that width will be acceptable for design is by calculating the maximum width seeing post-production run. 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.



[See original content here](#)



ALTIUM
DESIGNER18



Altium Designer 18

New to Altium Designer?
Don't be!

Try the latest release and get time saving workflows and enhanced team
collaboration options today at-

www.Altium.com/free-trials

Altium

Thanks for your attention!