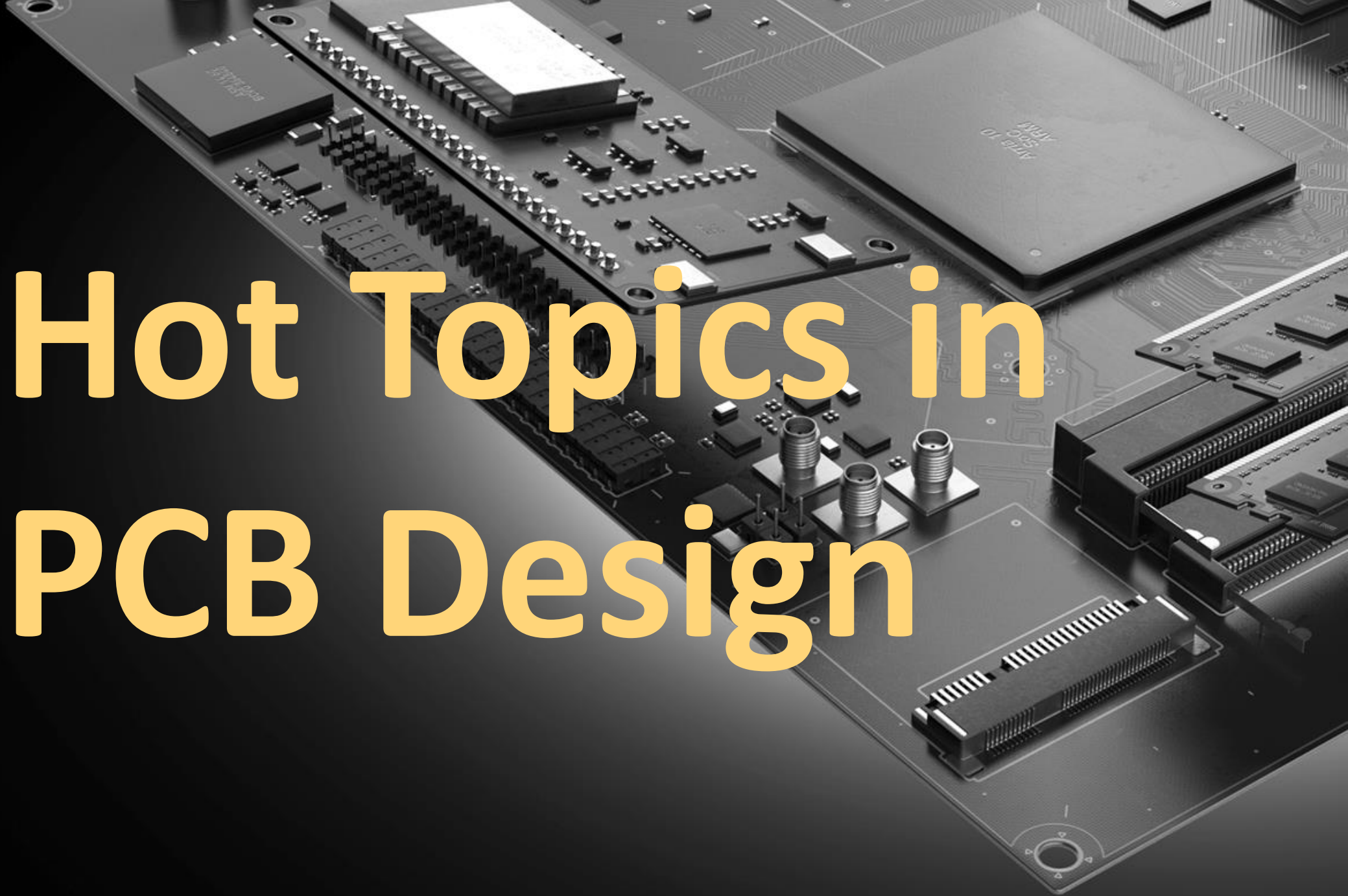


Altium

Hot Topics in PCB Design



1. HIGH CURRENT PCB DESIGN AND ITS REAL WORLD APPLICATIONS
2. HOW TO DESIGN A SIGNAL AMPLIFICATION DEVICE
3. PCB DESIGN LIMITATIONS BEYOND BORDERS. EXTENDING PAST EDGE CLEARANCE
4. YOUR PCB DESIGN LAYOUT WILL NEED REWORK
5. BEST PCB ROUTING PRACTICES AFTER AUTO ROUTING GOES THE DISTANCE

1

High Current PCB Design and Its Real World Applications

High Current PCB Design and Its Real World Applications

One of the largest constraints for PCBs was how to get PCBs handling large power distribution networks, high voltage, and high current capacities without being massive, bulking structures. Here, we're going to explore a few real-world applications:

PCB Applications in High Current - Mixing signals in PCB was too complex of an operation since many of the components on a dedicated board have their own power requirements and would be a nightmare to work with. In order to ensure that we are staying safe within this explosive domain, it is important that to maintain checks and analysis of the heat and power maps for designs.

Solar Panel Power Distribution - PCBs are usually designed for small voltage and low current signal operations, this poses a challenge for the high voltage and current that comes with functional solar panels.



High Current PCB Design and Its Real World Applications

Freeing Up Some Space - Along with the similar demand and technology that is shaping the solar panel industry, smaller and more capable designs are being pushed further into military designs. Within each craft produced by the military, most will inevitably have a radar system embedded within. One of the largest space suckers is that of the transformer—those massive hunks of iron that embody hundreds of copper windings to induct the transfer of energy

Flattened Power, High Current - One overarching issue that will arise with these new power systems is, that of thermal management. Uneven distribution of power and current within these PCB applications can wreak havoc on and in some cases even be catastrophic to design.



[See original content here](#)

2

How to Design a Signal Amplification Device

How to Design a Signal Amplification Device

Building a signal amplification device requires a thorough consideration of THE entire design, follow these tips to optimize design .

First Signal Amplification Steps - Get a sensor that is capable of measuring within the required limits and is sensitive enough to detect the slightest of changes, as required by the design specification. After identifying the right sensor for design, check the output signals.

Ensure Noise Immunity - Compared to the operating voltage of the ADC and processing controller, an unamplified raw signal from a sensor can have a relatively low amplitude. The analog track carrying the raw sensor signal should be well-routed and kept as short as possible. High frequency and power tracks must be kept away from the analog signal and a separate ground plane for the analog circuitry should be established.



How to Design a Signal Amplification Device

Follow Amplification Circuitry Design Best Practices - Before the signal is processed by the ADC, it has to be amplified by a certain magnitude. An op-amp connected with a couple of resistors can boost the signal as desired. Ensure that the range of the amplified signal stays within the output swing of the operational amplifier.

Prepare for Analog-to-Digital Conversion - In data processing, analog signals need to be converted to digital and this is where an ADC comes in. There is a choice of using a built-in ADC within the microcontroller or a dedicated ADC chip. Ensure the ADC has adequate bandwidth for sampling the analog signal.

Understand Signal Processing Principles - Depending on the purpose of signal amplification device, the amplified signal may be represented by visual display, retrieved by software, or simply converted to an analog output. In devices like environmental sensing controllers for aquaculture, the amplified signal is constantly measured to a threshold value with an ensuing action triggered if the threshold is crossed.



[See original content here](#)

**3**

PCB Design Limitations Beyond Borders, Extending Past Edge Clearance

PCB Design Limitations Beyond Borders

Sometimes components need to be extended past acceptable edge clearance tolerances and into the realm of the cliffs. The following guidelines will ensure components don't go falling off.

Keep Ample Copper Connections – A rule of thumb for component placement at any position on the board is to ensure that there is adequate copper enough for a sturdy electro-mechanical connection. While some things may change when venture past edge clearance tolerances, the rule for copper is no different.

Designing Outside the Box – Design beyond the chassis. When the product is opened up by the intended user, they might be using it for purposes never thought possible, but the user will still expect the product to hold to their standards. By looking at the space surrounding potentially hazardous components, be proactive about preventing damage even from unintended product uses.



[See original content here](#)

PCB Design Limitations Beyond Borders

Sometimes components needs to be extended past acceptable edge clearance tolerances and into the realm of the cliffs. The following guidelines will ensure components don't go falling off.

Manufacturing Limitations and How to Prevent Them - Moving components closer together or closer to the edge clearance of design will make it that much more difficult for manufacturing teams to incorporate.

Depending on the capabilities of manufacturer's machines, one may find at a loss when sending out initial RFQ's. One way to combat this is by performing an audit and manufacturing facility review before committing to a manufacturer for design.

This can be as simply as a phone call and building a relationship with your point of contact at the manufacturer or looking for a specific manufacturer who will work with you to customize your assembly.



[See original content here](#)

4

PCB Design Layout, When The Design Needs Rework

PCB Design Layout Needs Rework

We all know that those changes are going to come, let's make sure to prepare for them.

Design Changes are Inevitable for PCB Design Layout - Design changes may be required to accommodate new specifications, or someone may have made a mistake. Time and creativity is invested in order to make that design as perfect as it can be. When the design gets changed, it can be very difficult to deconstruct what was done.

Use PCB Design Layout Tools That are Up to the Task of Rework - Another way that can be prepared for changes is by using PCB design software that will help instead of being a hindrance. A schematic capture application will allow to make changes quickly without having to do a lot of manual edits to reconnect circuitry.



[See original content here](#)

PCB Design Layout Needs Rework

We all know that those changes are going to come, let's make sure to prepare for them.

How Can You be Prepared for Design Changes?

- **Check up front for anticipated design changes** - When starting a PCB design layout, work with the entire design team and find out what stage is the design in. This way at least you can put off working on those areas until the changes come through.
- **Leave some space** - Whenever possible, leave some space in the layout so that the layout won't be impacted as much by future changes.
- **Accept the fact that rework happens and it isn't a reflection on you** - Rework is an important part of the design process and it isn't ever going to stop.



[See original content here](#)

5

Best PCB Routing Practices after Auto Routing goes the Distance

Best PCB Routing Practices after Auto Routing goes the Distance

Best PCB Routing Practices- Review Your Auto Routing Results - Don't set up for disappointment by anticipating more than what is reasonably expected from the router. Look for how the router dealt with different areas of routing.

Catching Errors- Antennas, Jogs, and Traps - Spend some time correcting the auto routing results. Expect to find certain areas of routing that were mishandled by the router and now need to be cleaned up.

One such area is traces that are left as antennas. Two other similar problems that auto routers can create in routing are jogs and acid traps.

Auto router clean-up routines should detect and remove these problems, but it pays to keep an eye out for them nevertheless.



[See original content here](#)

Best PCB Routing Practices after Auto Routing goes the Distance

Cleanup PCB Routing Techniques, or is There a Better Way?

Probably time will be spent in manual routing cleanup to improve the routing that came from the auto router. There is another option though, and that is to use an auto-interactive router. These routers give s control over the direction of the routing by following the path that is specified for the traces to follow.



[See original content here](#)

Altium

PCB DESIGN



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!