



**Altium**

# HIGH VOLTAGE PCB DESIGN

1. **HOW TO PLAN YOUR PCB LAYOUT FOR A HIGH VOLTAGE DESIGN**
2. WHY DESIGNING A HIGH VOLTAGE PCB FOR RAPID PROTOTYPING IS MORE EFFECTIVE THAN RUSHING TO FORM FACTOR
3. HIGH VOLTAGE PCB DESIGN: CREEPAGE AND CLEARANCE DISTANCE
4. CONSIDERATIONS FOR DESIGNING A HIGH VOLTAGE PCB
5. HIGH VOLTAGE PCB DESIGN FOR ARC PREVENTION: HOW TO MANAGE PADS AND TRACES
6. HIGH VOLTAGE PCB DESIGN TIPS: MATERIALS FOR HIGH VOLTAGE PCBs

# 1. HOW TO PLAN YOUR PCB LAYOUT FOR A HIGH VOLTAGE DESIGN

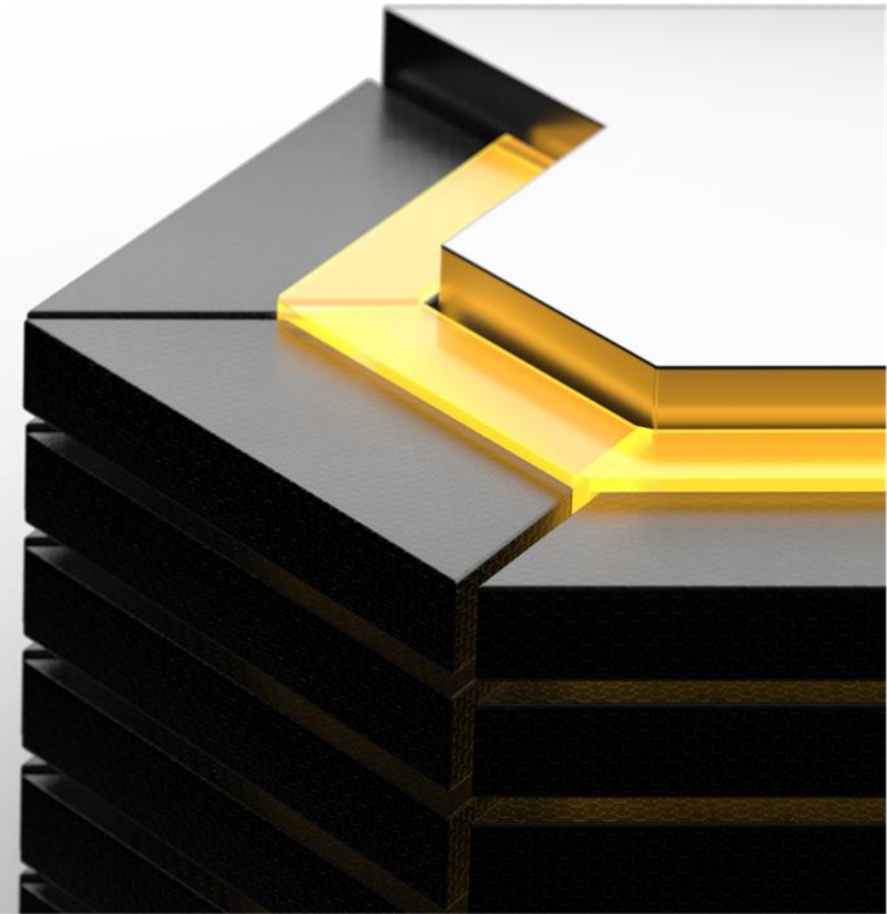


# Planning PCB Layout for High Voltage Design

Layouts for a high voltage PCB need to control and optimize field strength across the board to provide the best performance and lifetime protection in your final product. To do this, you should ...

- **Isolate high voltage areas:** You want to group your high voltage circuitry together to minimize the impact on the rest of the board. One way to do this is by adding an insert around high voltage areas
- **Decrease voltage across the board gradually:** After isolating highest voltage regions, you should lay out the rest of the board in “zones” that let you step the voltage down gradually
- **Isolate noise sources:** You should always either use short traces or isolate your noise sources
- **Minimize interconnects:** Using fewer interconnects reduces opportunities for transient generation in your design and the propagation of high voltage
- **Use design checks:** Make sure your PCB design tool has a thorough design rules checker

[See original content here](#)



1. HOW TO PLAN YOUR PCB LAYOUT FOR A HIGH VOLTAGE DESIGN
2. WHY DESIGNING A HIGH VOLTAGE PCB FOR RAPID PROTOTYPING IS MORE EFFECTIVE THAN RUSHING TO FORM FACTOR
3. HIGH VOLTAGE PCB DESIGN: CREEPAGE AND CLEARANCE DISTANCE
4. CONSIDERATIONS FOR DESIGNING A HIGH VOLTAGE PCB
5. HIGH VOLTAGE PCB DESIGN FOR ARC PREVENTION: HOW TO MANAGE PADS AND TRACES
6. HIGH VOLTAGE PCB DESIGN TIPS: MATERIALS FOR HIGH VOLTAGE PCBs

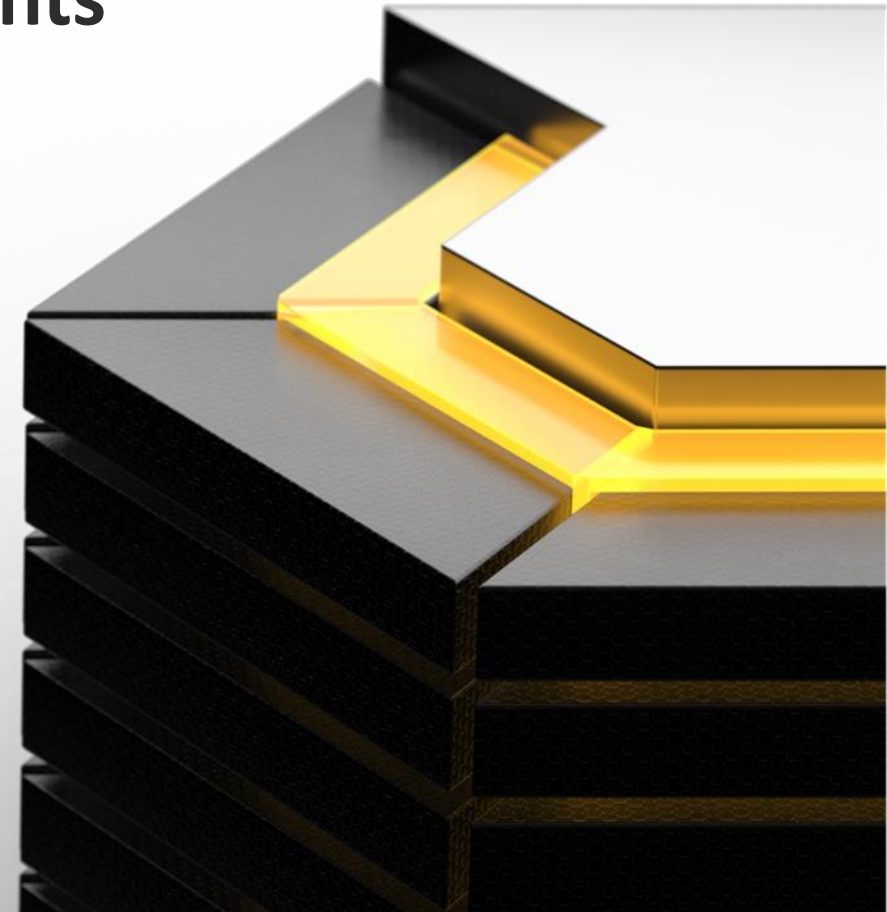
## 2. WHY DESIGNING A HIGH VOLTAGE PCB FOR RAPID PROTOTYPING IS MORE EFFECTIVE THAN RUSHING TO FORM FACTOR



## Making Critical Performance Measurements

Going to form factor too early rather than rapid prototyping can create many unanticipated issues later in the design process. A design is never actually done until its specified requirements are met, and the key to knowing a design is finished is being able to measure its performance to verify these requirements.

- If there is little room for probing or simulating active circuits, the design cannot be rigorously tested and must rely on functional testing to validate performance metrics
- If testing results in changes in the design, having plenty of space to directly solder components to the PCB makes the process much easier than making a complete respin of the design every time a change is required

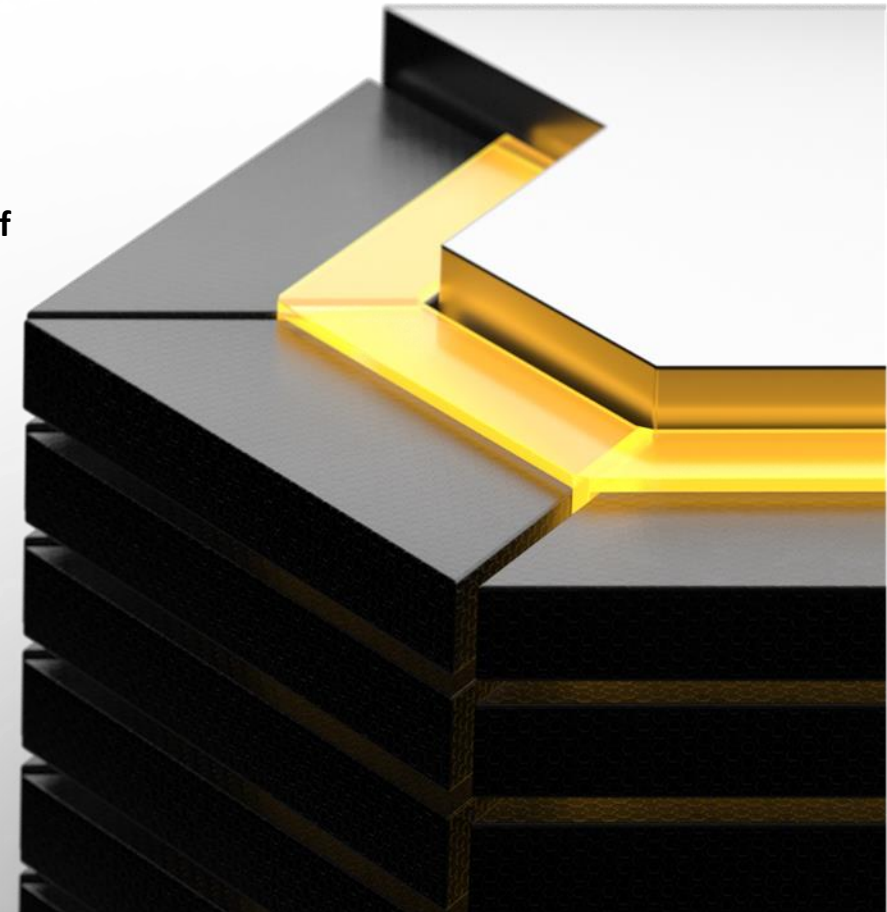


## Making Changes to Layout and Design

Often, you will have to deal with field failures or other issues reported from testing. You may also get requests for a new feature that wasn't previously scoped. However, any true product design will go through several cycles for a multitude of reasons. These changes will be costly if the layout is already form factored. However, you will eventually have to go form factor.

- Even with state of the art PCB design software and inventory management systems, creating form factor designs take a lot of work
- In addition to just planning a layout, it is also second sources for components and a real inventory management system as production begins to ramp up.
- While rushing to form factor can slow a design down, true designs must go to form factor eventually

[See original content here](#)





1. HOW TO PLAN YOUR PCB LAYOUT FOR A HIGH VOLTAGE DESIGN
2. WHY DESIGNING A HIGH VOLTAGE PCB FOR RAPID PROTOTYPING IS MORE EFFECTIVE THAN RUSHING TO FORM FACTOR
3. **HIGH VOLTAGE PCB DESIGN: CREEPAGE AND CLEARANCE DISTANCE**
4. CONSIDERATIONS FOR DESIGNING A HIGH VOLTAGE PCB
5. HIGH VOLTAGE PCB DESIGN FOR ARC PREVENTION: HOW TO MANAGE PADS AND TRACES
6. HIGH VOLTAGE PCB DESIGN TIPS: MATERIALS FOR HIGH VOLTAGE PCBs

# 3. HIGH VOLTAGE PCB DESIGN: CREEPAGE AND CLEARANCE DISTANCE

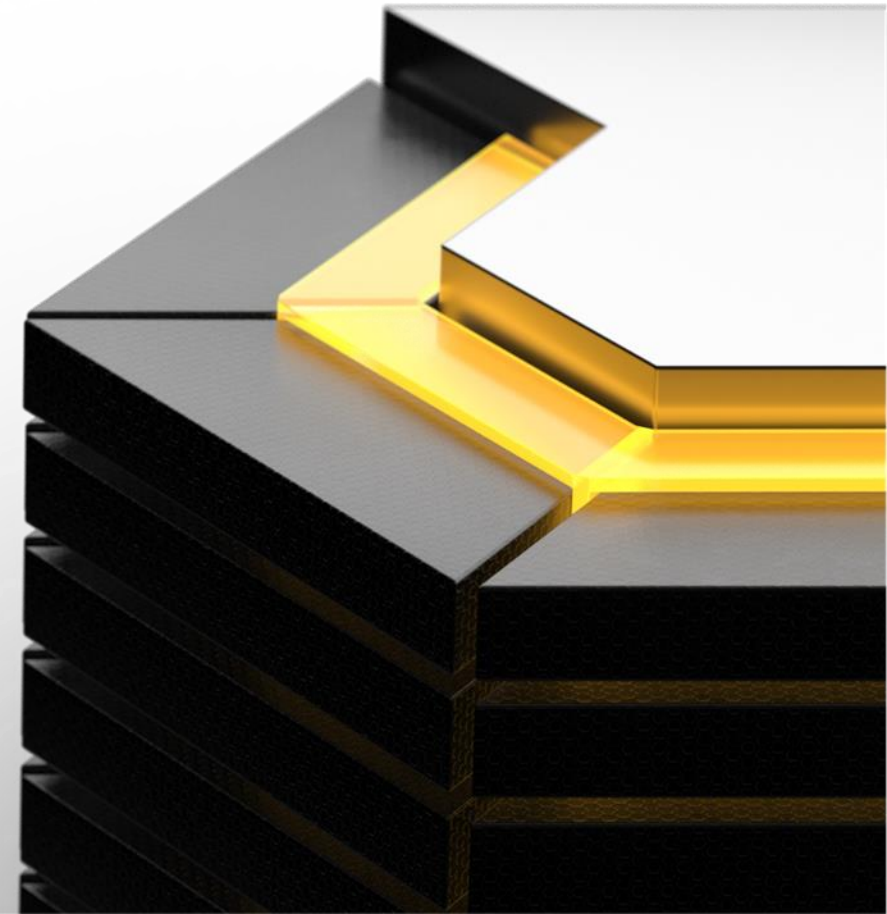


## Creepage and Clearance Distance

There are board level standards and protection required for high voltage products. Some common questions and answers about these standards include:

- **When does safety require specific spacing rules?** If the normal operating voltage of your product meets or exceeds 30 VAC or 60 VDC, then you should be very diligent about spacing rules in your board design
- **What is clearance?** Clearance is the shortest distance through air between two conductors. If the clearance anywhere on a PCB is too small, an over-voltage event can cause an arc between neighboring conductive elements on the board
- **What is creepage on a PCB?** Creepage measures distance between conductors on a PCB. But instead of distance in air, it measures the shortest distance along the surface of the insulation material
- **How do I know what material and spacing to use?** Because there are so many variables in PCB design and material selection, your best bet for meeting the safety requirements and standards is to go directly to the source

[See original content here](#)



1. HOW TO PLAN YOUR PCB LAYOUT FOR A HIGH VOLTAGE DESIGN
2. WHY DESIGNING A HIGH VOLTAGE PCB FOR RAPID PROTOTYPING IS MORE EFFECTIVE THAN RUSHING TO FORM FACTOR
3. HIGH VOLTAGE PCB DESIGN: CREEPAGE AND CLEARANCE DISTANCE
4. **CONSIDERATIONS FOR DESIGNING A HIGH VOLTAGE PCB**
5. HIGH VOLTAGE PCB DESIGN FOR ARC PREVENTION: HOW TO MANAGE PADS AND TRACES
6. HIGH VOLTAGE PCB DESIGN TIPS: MATERIALS FOR HIGH VOLTAGE PCBs

# 4. CONSIDERATIONS FOR DESIGNING A HIGH VOLTAGE PCB

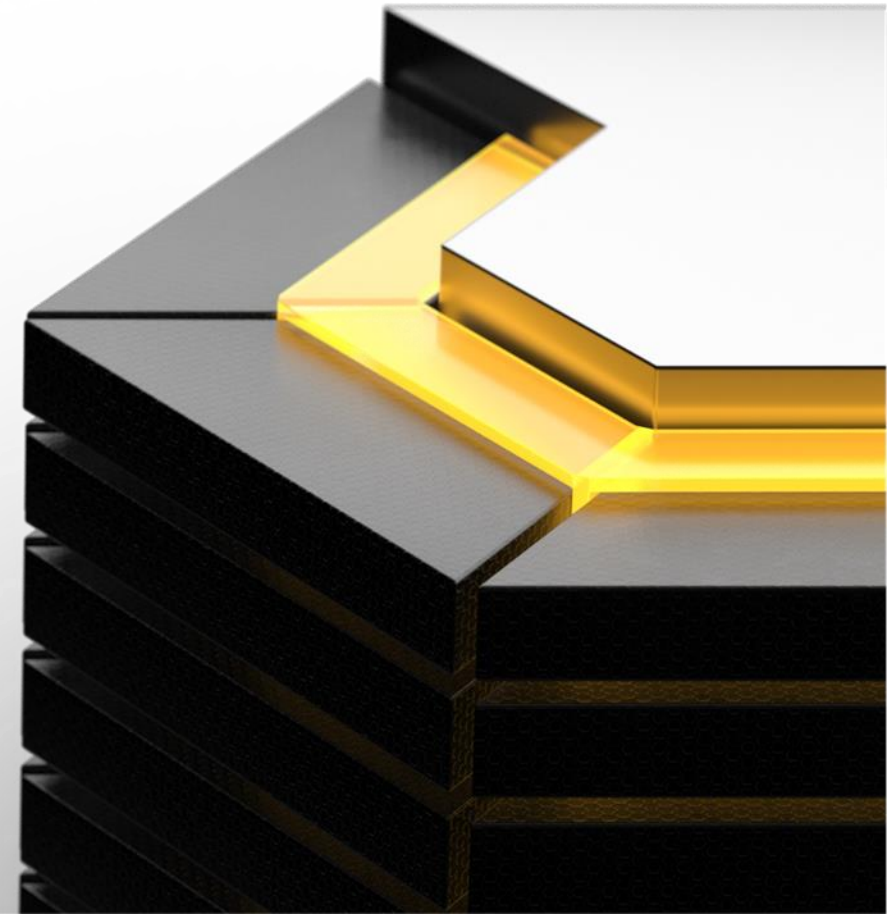


## Considerations for Designing a High Voltage PCB

Designing a PCB for high voltage applications requires consideration of a host of details at every level of your design and fabrication. The board will be subject to harsh operating conditions and will be much more sensitive to the lifetime of the parts and materials. There are a few design considerations before you start your board layout:

- **Consider your operating frequency:** The operating frequency of your product affects high voltage design as much as ESD and noise management affect the board. To minimize risk, don't use pure tin or silver as a finish on your PCBs
- **Derate your components:** You should derate to the maximum voltage, rather than the average, to improve the survivability of your product during an over-voltage event
- **Select your components:** After derating, you may find some of the initial components you've selected aren't viable for your product's operating environment. Even if they all pass, you should review each part again

[See original content here](#)



1. HOW TO PLAN YOUR PCB LAYOUT FOR A HIGH VOLTAGE DESIGN
2. WHY DESIGNING A HIGH VOLTAGE PCB FOR RAPID PROTOTYPING IS MORE EFFECTIVE THAN RUSHING TO FORM FACTOR
3. HIGH VOLTAGE PCB DESIGN: CREEPAGE AND CLEARANCE DISTANCE
4. CONSIDERATIONS FOR DESIGNING A HIGH VOLTAGE PCB
5. **HIGH VOLTAGE PCB DESIGN FOR ARC PREVENTION: HOW TO MANAGE PADS AND TRACES**
6. HIGH VOLTAGE PCB DESIGN TIPS: MATERIALS FOR HIGH VOLTAGE PCBs

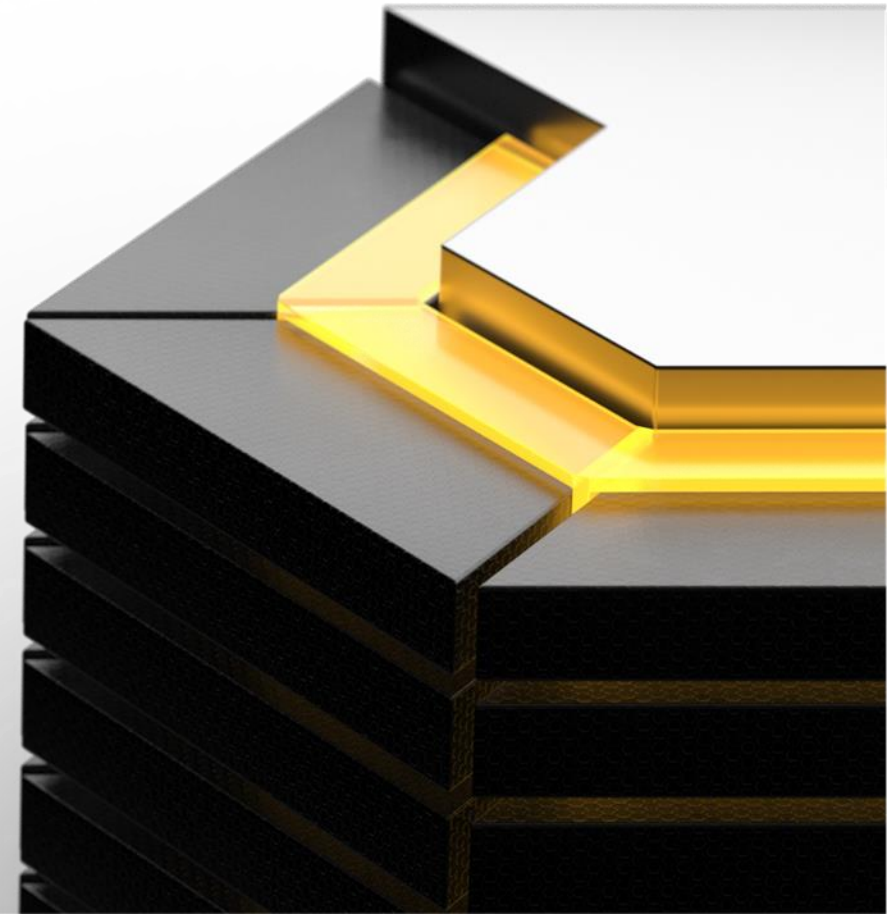
# 5. HIGH VOLTAGE PCB DESIGN FOR ARC PREVENTION: HOW TO MANAGE PADS AND TRACES





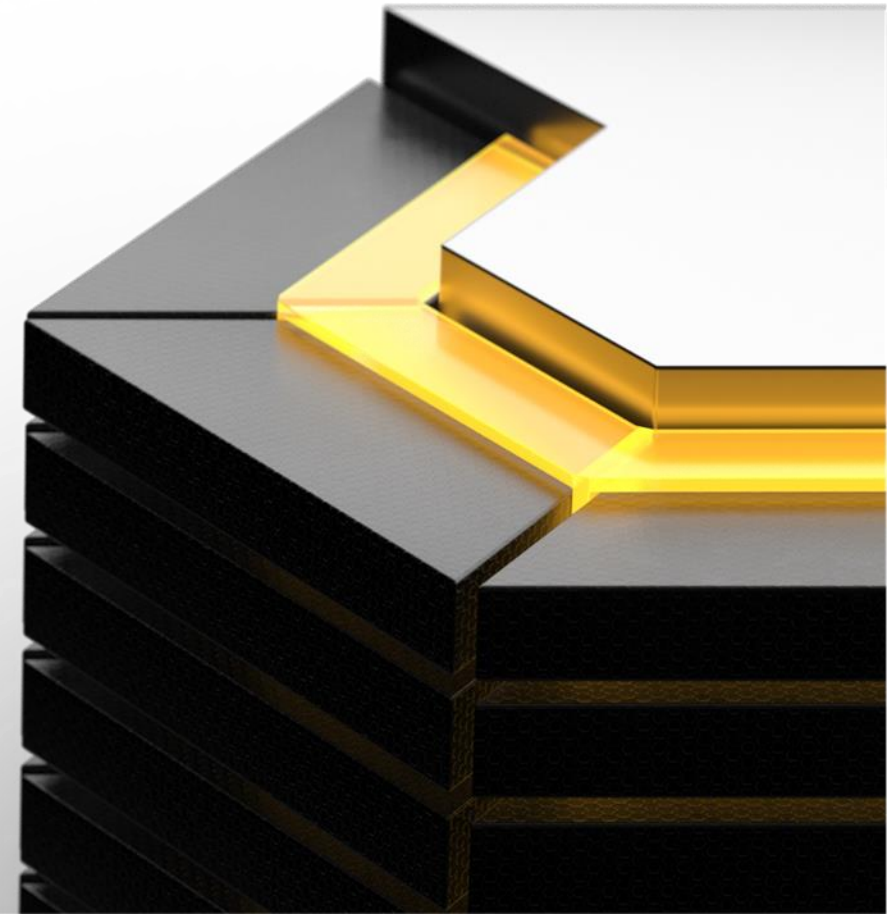
## Best Practices for Designing Corners

- You want to avoid any corners and sharp edges or turns on high voltage PCBs. For pads, use a smooth, round curve that cuts off corners. Design the curve with the largest radius you can without losing necessary surface area
- You also want to avoid sharp corners at turns. Instead, use a curved track with the biggest possible radius
- If using solder balls at connection points, you should also specify a minimum radius for the solder balls



## Spacing Considerations

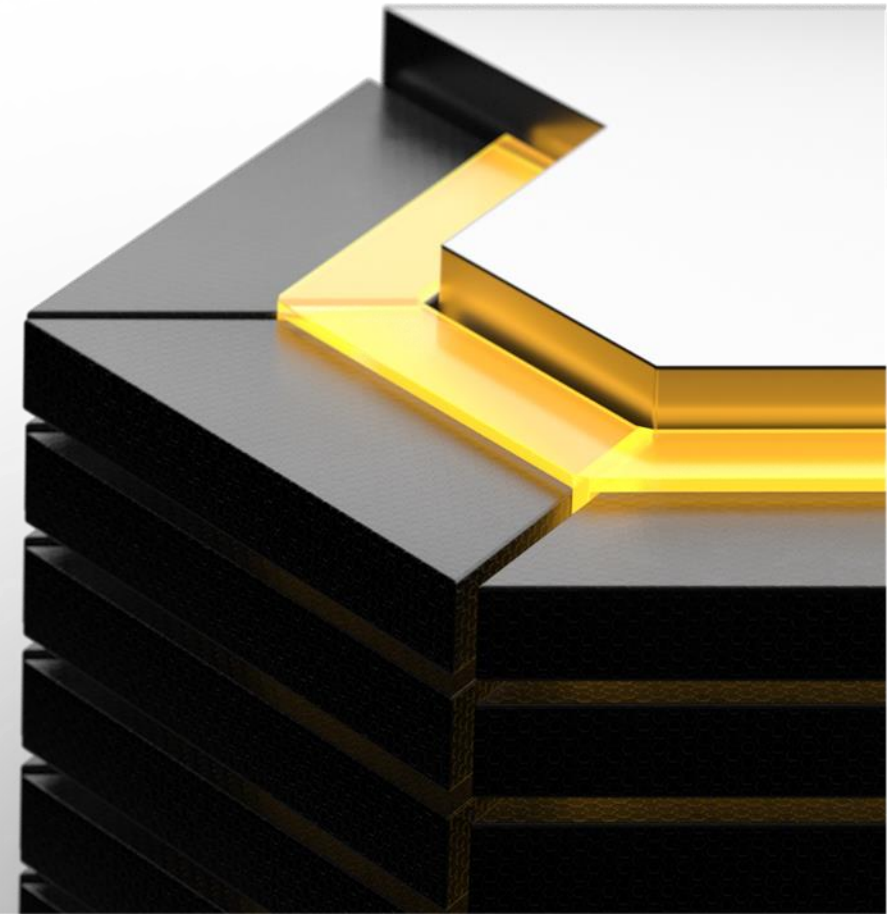
- By using curved edges instead of corners, you can use a much smaller spacing between features
- High frequency will arc more easily, so your spacing increases by a recommended factor of 1.12X
- Once you have your board, make sure that the surface and edges are clean, and all of the conductors are smooth before you power it up. Dirt, rough edges, and scratches can all decrease the arc-over voltage by 70-80%



## Designing Edges

- Edges on a PCB will need special attention. When the edge of a board is machined or routed, burrs are often left behind on the raw edge. If this happens on a conductive trace, the burr will function like a needlepoint and dramatically increase the risk of arcing
- To be safe, your layout should set copper traces back from any edges. Use a minimum buffer distance of 0.02 inches between a conductor and the edge. Increase the distance by 0.001 inches for every 100 V the trace will experience at its maximum load
- If you need to shoehorn a trace in near the edge, you can use a conformal coat of a high voltage compatible material, like HVPF. This allows for additional protection to your PCB out to the edges of the board

[See original content here](#)



1. HOW TO PLAN YOUR PCB LAYOUT FOR A HIGH VOLTAGE DESIGN
2. WHY DESIGNING A HIGH VOLTAGE PCB FOR RAPID PROTOTYPING IS MORE EFFECTIVE THAN RUSHING TO FORM FACTOR
3. HIGH VOLTAGE PCB DESIGN: CREEPAGE AND CLEARANCE DISTANCE
4. CONSIDERATIONS FOR DESIGNING A HIGH VOLTAGE PCB
5. HIGH VOLTAGE PCB DESIGN FOR ARC PREVENTION: HOW TO MANAGE PADS AND TRACES
6. **HIGH VOLTAGE PCB DESIGN TIPS: MATERIALS FOR HIGH VOLTAGE PCBs**

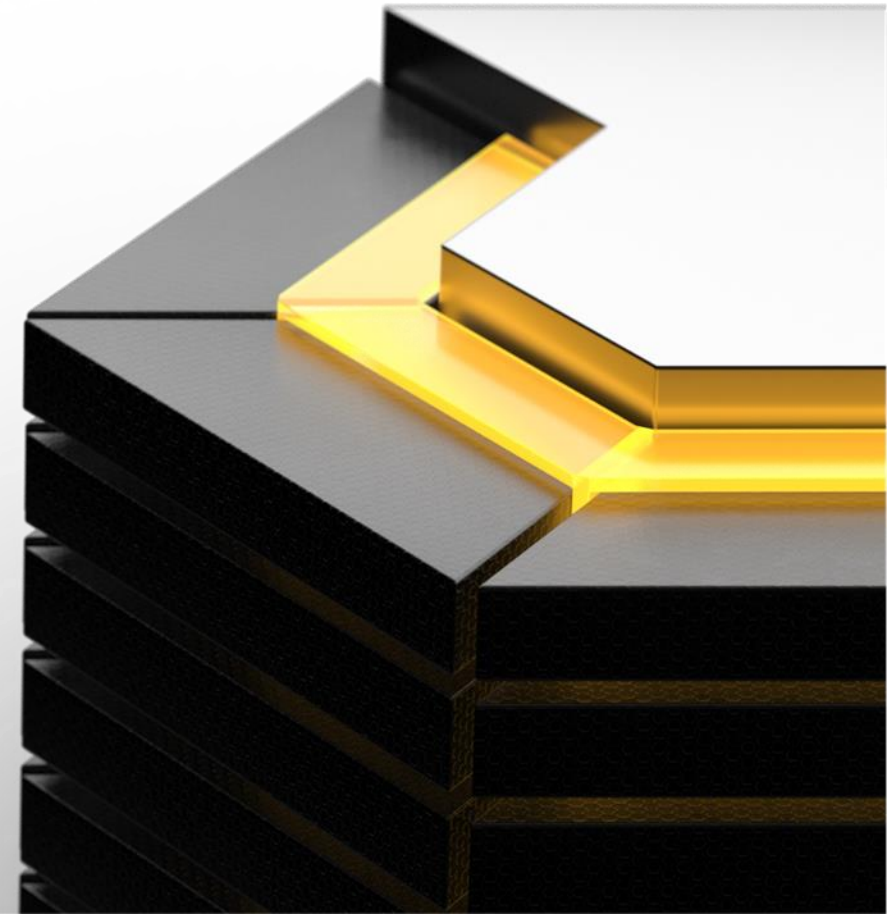
# 6. HIGH VOLTAGE PCB DESIGN TIPS: MATERIALS FOR HIGH VOLTAGE PCBs



## Selecting an Appropriate PCB Materials

The foundation of your PCB is the board, so that's the first material specification you should consider. You want the material to be appropriate for the performance requirements, but also for the operating environment since that will have a large impact on how the PCB material ages. For a high voltage PCB, you'll need a material that's specifically designed to tolerate an overvoltage event. A few material options to consider:

- FR4 Laminate
- BT Epoxy
- Isola, High V Laminates

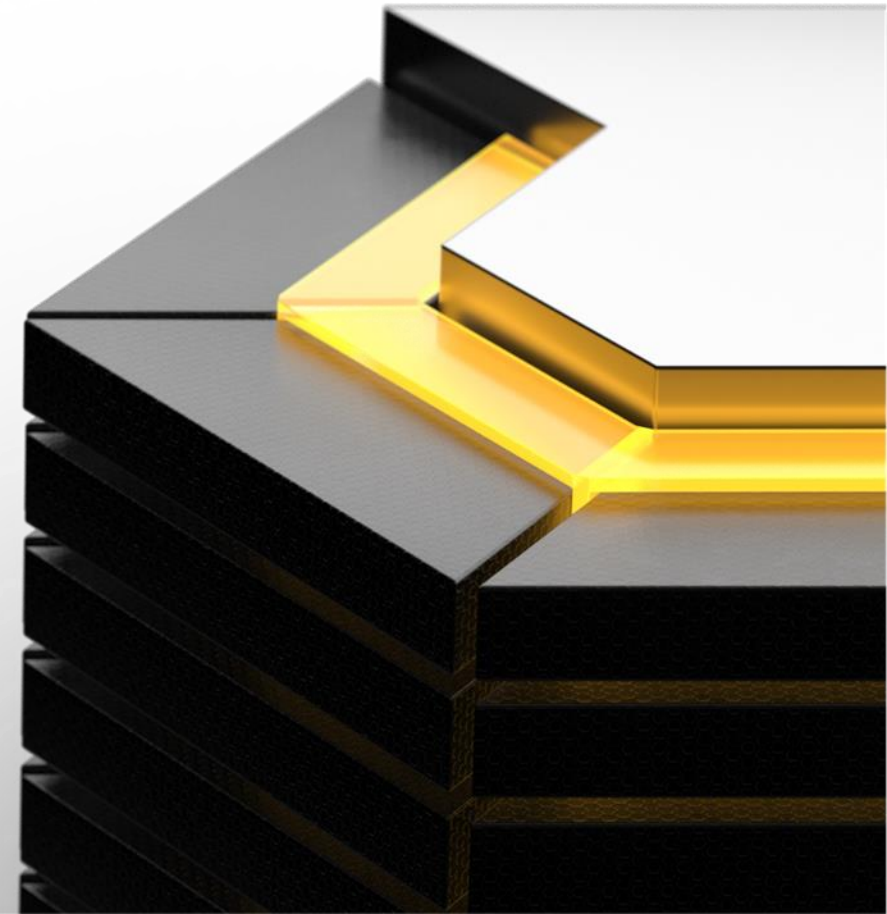


## Making Sure Performance Matches Requirements

When you first start discussing production, get the datasheets for all your options from the manufacturer and make sure the performance matches your requirements. Also, don't mix and match the insulators on your board which can cause issues in manufacturing and performance down the line. Some things you should make sure you do are:

- **Check glass and resin contents:** If the resin content and glass style used in the PCB aren't in the datasheets or manufacturing guides you receive, ask for them specifically. Having a high resin content is important as well as a small glass style to help the resin penetrate
- **Choose the right conductors:** The spacing between your traces and pads is dictated by safety and standards. However, some of these standards are outdated. You also want the quality of your copper to meet your requirements
- **Ask about finished surfaces:** Ask your manufacturer about the finishing on your board. Poor surface conditions on the board, like bumps, particles, inconsistencies, or contamination, make arching more likely and undo all expensive material selection you've done

[See original content here](#)



***Altium***

**Thanks for your attention!**