

Using an IPC-2221 Calculator for High Voltage Design



Zachariah Peterson | January 17, 2020



Unsure of your conductor spacing requirements in high voltage PCBs? An IPC-2221 calculator can help.

PCB design and assembly standards aren't there to constrain your productivity. Instead, they're there to help create uniform expectations of product designs and performance across multiple industries. With standardization comes the tools for compliance, such as calculators for certain design aspects, processes for auditing and inspection, and much more.

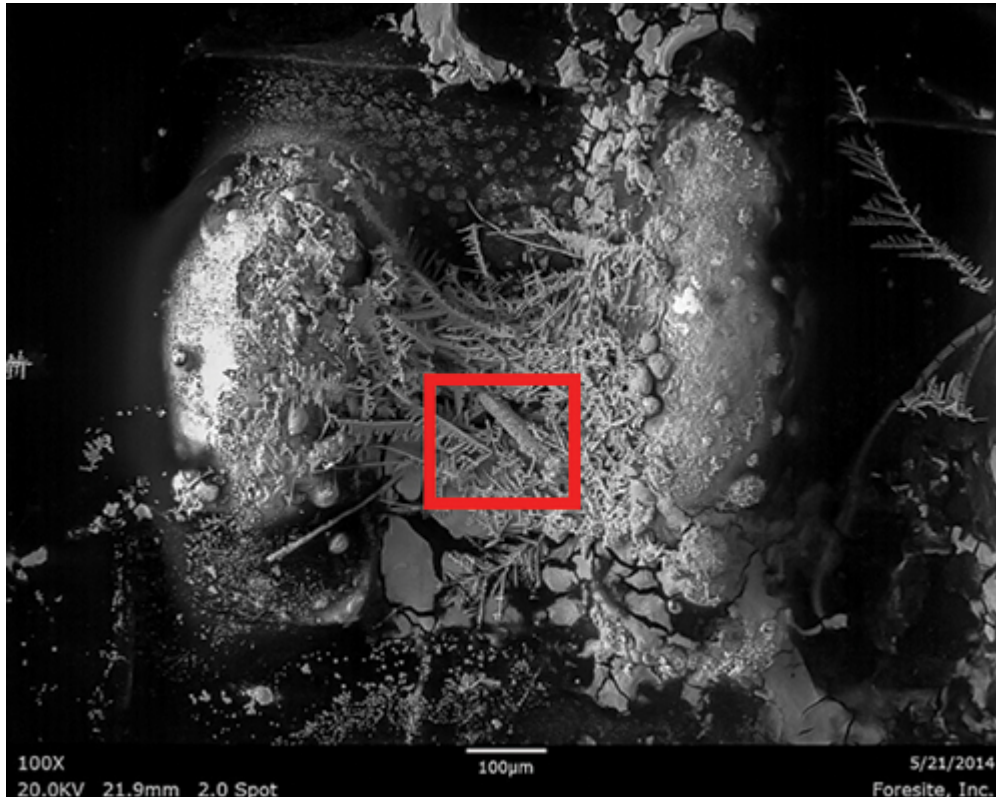
In high voltage PCB design, the important generic standard for PCB design is IPC-2221. Many important design aspects are summarized in this design standard, some of which boil down to simple mathematical formulas. For high voltage PCBs, an IPC-2221 calculator can help you quickly determine appropriate spacing requirements between conductive elements on your PCB, which helps ensure your next high voltage board will remain safe at its operating voltage. When your design software includes these specifications as automated design rules, you can remain productive and avoid making layout mistakes as you build your board.

What is IPC-2221?

IPC-2221 is a generally accepted industry standard that defines a multitude of PCB design aspects. Some examples include design requirements on materials (including substrates and plating), testability, thermal management and thermal reliefs, and annular rings, to name a few. In high voltage PCB design, important design requirements are specified in the IPC-2221B standard. Note that IPC only specifies minimum conductor spacing values for voltages up to 500 V. Simple formulas are provided for calculating minimum conductor spacing values for voltages above 500 V.

Among the list of minimum electrical conductor spacing values defined in IPC-2221, there is one set of requirements that is intended to prevent metal migration failure. Metal migration is one of many failure mechanisms in high voltage designs with high conductor density. When two conductors are brought up to a high potential, electrochemical growth of metallic dendrites can occur when the conductors contain residues with water-soluble salts; an SEM image of dendritic growth between two solder balls is shown below. These metallic dendrites can short out two points on a high density PCB. This is actually an electric field

effect, which explains why there is a minimum spacing requirement; increasing the spacing between conductors for a given potential difference reduces the field between the conductors, which inhibits dendrite growth.



SEM image showing dendritic growth between two solder balls. Image source.

In the case that your product is not covered by the more stringent UL or IEC standards (see more on this below), there is a simple set of data and a formula you can use to determine the minimum conductor spacing in your PCB. This data is a function of the peak voltage being used and whether a coating is present on your board, as shown in the following table. If you need to determine the minimum conductor spacing between two voltage values, you can use a linear approximation (interpolation) between two successive entries.

V _{peak} (V)	Internal (mm)	External, uncoated (mm)	External, coated (mm)
15	0.05	0.1	0.05
30	0.05	0.1	0.05
50	0.1	0.6	0.13
100	0.1	0.6	0.13
150	0.2	0.6	0.4
170	0.2	1.25	0.4
250	0.2	1.25	0.4
300	0.2	1.25	0.4
500	0.25	2.5	0.8
> 500	$0.25 + (V-500) \times 0.0025$	$2.5 + (V-500) \times 0.005$	$0.8 + (V-500) \times 0.00306$

IPC-2221B conductor spacing requirements.

If you look throughout the electronics standards landscapes for different industries and applications, you'll find a variety of safety standards that prescribe different conductor spacing requirements. IPC-9592B standard provides conductor spacing requirements for power conversion devices. These standards are quite consistent when graphed alongside the required conductor spacing specified in IPC-2221B. The table below specifies the spacing requirements under IPC-9592B. Note that this defines the minimum required trace spacing as a function of peak voltage values.

Minimum spacing (mm)	Voltage Range
0.13	$V_{\text{peak}} < 15 \text{ V}$
0.25	$15 \text{ V} \leq V_{\text{peak}} < 30 \text{ V}$
$0.1 + (V_{\text{peak}} \times 0.01)$	$30 \text{ V} \leq V_{\text{peak}} < 100 \text{ V}$
$0.6 + (V_{\text{peak}} \times .005)$	$V_{\text{peak}} \geq 100 \text{ V}$

IPC-9592B conductor spacing requirements for power conversion devices.

Both sets of standards include formulas that can be easily programmed into a calculator application. You can find many IPC-2221 calculator applications on the internet, including IPC-2221B trace spacing calculators.

Working With an Integrated IPC-2221 Calculator

When your design software includes an IPC-2221 calculator as part of your design rules, you won't have to manually calculate trace widths or clearances to prevent breakdown. This type of software encodes the required clearance as a design rule, which is automatically checked as you create your PCB layout. If you are looking to ensure a higher level of safety for your new product, you can encode a larger required trace and/or component spacing in your design rules.

As you place components and route traces in your high voltage PCB, clearances can be checked automatically as you build your board. The ability to automatically check clearances between traces and components saves you from making a mistake and reduces the extent of any required placement changes. Some design rules won't perform these important checks until you've finished your layout, putting you at risk of extensive redesigns when components are placed too closely together. Working with design tools that allow you to specify clearances reduces these risks and helps you remain productive.

Note that the IPC-2221 standards are totally voluntary. However, for products covered by legal safety standards, creepage and clearance requirements in the relevant UL or IEC standard are mandatory. As an example, the relevant set of safety requirements on IT and telecom products with AC mains and battery power can be found in the IEC-60950-1 standard (2nd edition). In terms of creepage, the spacing specified under IPC-2221B depends on RMS working voltage, pollution degree (numbered 1 through 3), and material group. The definitions of the latter two terms can be found in the UL 60950-1 standards. Whether you need to comply with IEC, IPC, or other required safety standards, you can specify your design requirements as design rules when you use the right PCB design software.

The CAD tools and routing features in Altium Designer® are built on a unified rules-driven design engine that automatically checks your layout as you create your board. Rather than using a manual IPC-2221 calculator, these design features help save time and sanity as you create your next high voltage PCB. You'll also have access to a full set of documentation features that help you prepare for manufacturing and assembly.

Now you can download a free trial of Altium Designer and learn more about the industry's best layout, simulation, and production planning tools. Talk to an Altium expert today to learn more.