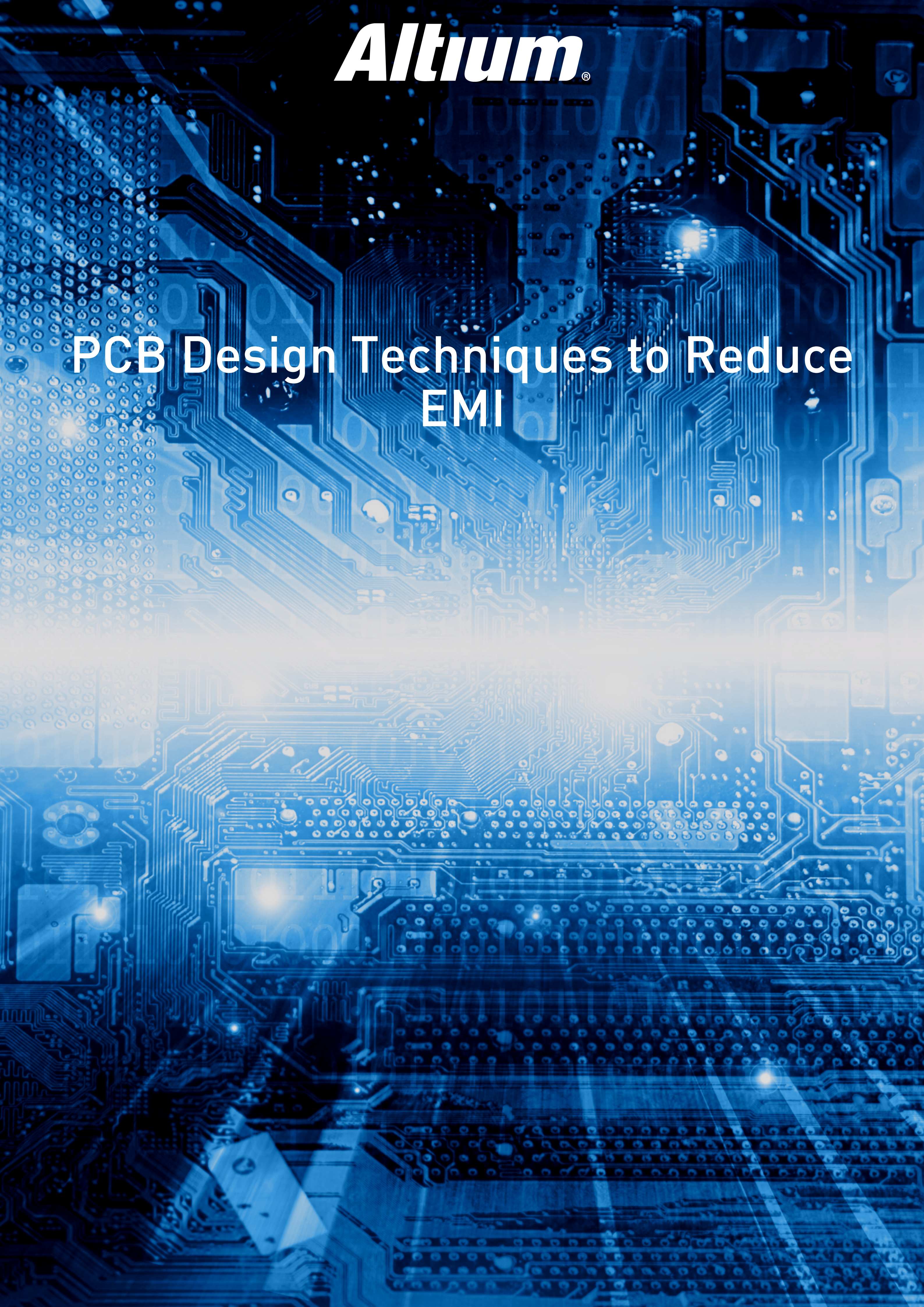
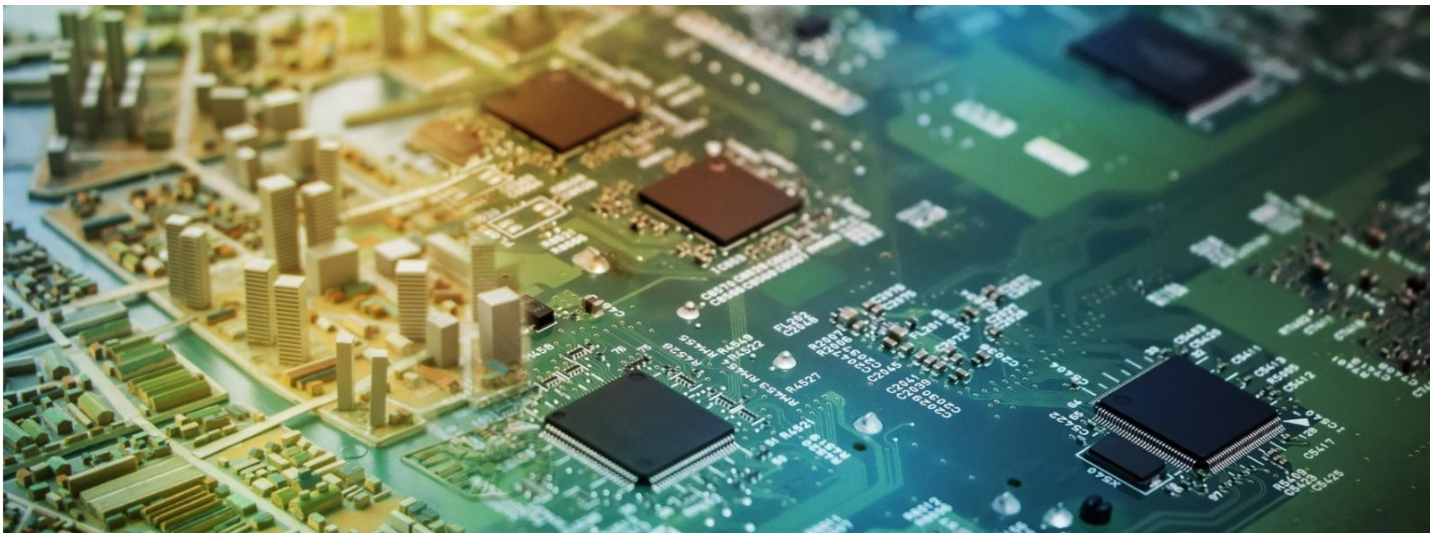


PCB Design Techniques to Reduce EMI



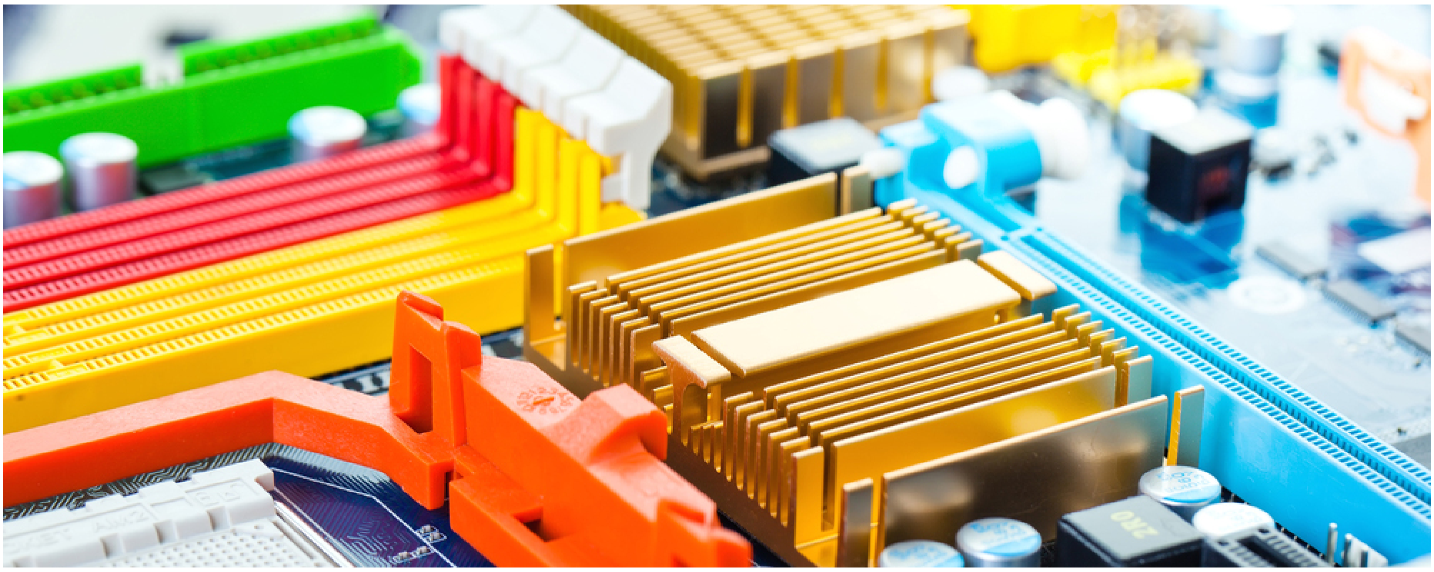
PCB DESIGN TECHNIQUES TO REDUCE EMI

Digital circuits are getting faster and more powerful each year, and as a result, it's more important than ever to utilize proper design techniques and materials to reduce electromagnetic interference (EMI) in our PCB designs. Poor planning can lead to increased and unnecessary radiation, as well as reduced system immunity, which can wreak havoc on our devices. Approach the challenge like a pro, with a multi-pronged plan of attack involving choosing proper PCB ground designs, isolating circuits, routing differential pairs to preserve signal integrity, and more.

Join us as we explore a variety of design techniques to reduce EMI in your PCBs, including:

- EMI/EMC Design: PCB Noise Reduction Through Isolation of AC and DC Signals
- How to Reduce EMI in Mixed-Signal Systems Using Proper PCB Ground Designs
- Using cans for EMI shielding on your PCB
- High-Speed PCB Differential Pair Routing to Preserve Signal Integrity
- High-Speed PCB Design Principles: Keep Traces Short and Direct

EMI/EMC DESIGN: PCB NOISE REDUCTION THROUGH ISOLATION OF AC AND DC SIGNALS



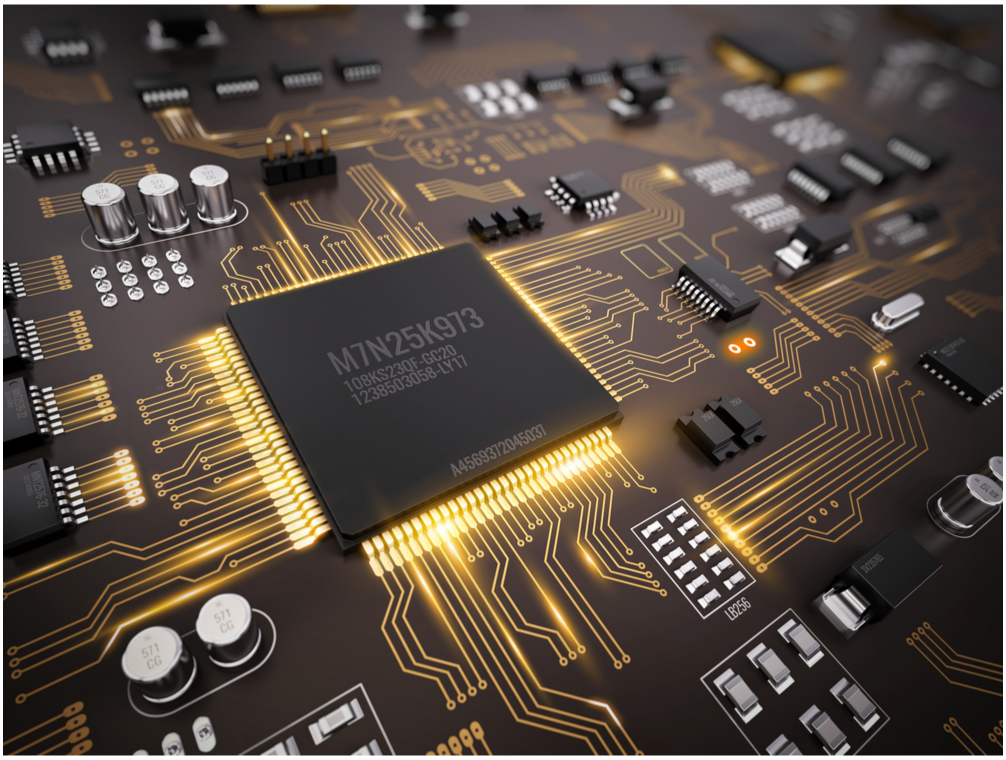
Using AC and DC components on the same board can cause EMI issues, however, these issues can be resolved by isolating the AC and DC systems. There are several straightforward solutions to help manage interference between AC and DC circuits, mainly; shielding components, separating systems, dedicated power supplies, good grounding, and not bridging isolation. Read on to find out more!

In high school, my physics teacher always used to talk about the "War of Currents" between Edison and Tesla. Back then I never really cared much why Edison preferred DC power for distribution and use, while Tesla favored AC power. That changed when I began to see the "War of Currents" play out firsthand between AC and DC systems on PCBs. Many PCBs today use both AC and DC circuits, and where they meet, there is often trouble in the form of electromagnetic interference (EMI). Luckily, just as the "War of Currents" eventually gave way to peace, the AC and DC signals on modern PCBs can coexist in harmony. The key is isolation.

ISOLATION FROM ELECTROMAGNETIC INTERFERENCE

There are several straightforward solutions to help manage interference between AC and DC circuits, mainly; shielding components, separating systems, dedicated power supplies, good grounding, and not bridging isolation.

Shielding: As you are aware, there are many components in a PCB that can radiate EMI. This includes power supplies, IC clocks, and oscillators, all of which can interfere with AC components. One way to limit EMI from "noisy" DC components, or protect AC components that are susceptible, is simply to shield them. Shielding will essentially guarantee that no EMI radiating through the air in the enclosure will interfere with the shielded circuit. So if you have something you need to protect or constrain, do what humans have been doing for centuries, and put it in a metal box. If you have a multi-layer board, you can also use a ground plane layer as a shield. While effective, shielding will increase the weight and cost of a board, so be careful to weigh EMI reduction against other concerns.



No matter how simple or complex your board, it can benefit from AC/DC separation.

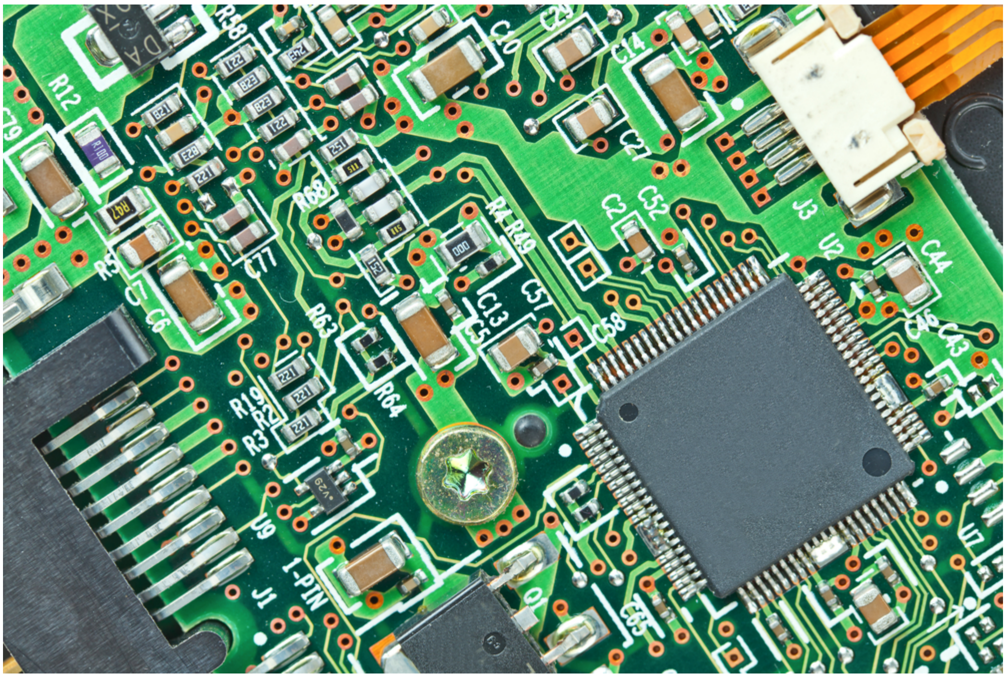
Separation: If Edison and Tesla had been in a room together they certainly would have had some words for each other, if not some more physical rebuffs. Luckily they mostly stayed separate, and your AC and DC components should follow their lead. Keeping AC and DC systems, both chips and traces, far away from each other on the PCB will help ensure there is no “crosstalk” between the systems. If you don’t have enough space on the board to put physical distance between AC and DC systems, you can also cut gaps in the ground plane between components that need to be separated. Gaps in the ground plane will force any currents flowing through the plane to go around the gaps, which, used strategically, can help you reroute currents around sensitive systems. In short, don’t cross the wires! Getting good separation can be easy in a simple circuit, but quite difficult in more complicated circuits. Do your best to separate AC and DC, but know that it may be difficult to reach an optimal solution.

Power: In every AC/DC PCB there should be separate power rails for AC and DC components. DC components can draw spikes from their power supply, creating a voltage transient. While your AC components may (or may not) be able to operate with this voltage transient, they will certainly not operate at their highest capacity. If the voltage transients are too large they can produce errors in AC components, or stop the circuit from working altogether. Having separate power supplies may not be convenient, but it is better than having a chip that doesn’t work.

Grounding: As you already know, grounding in AC/DC circuits is a complex issue, too complex for this article to delve into fully.

However, I can give you this advice; check current return paths in grounding grids or planes. Remember that while DC currents will follow the path of least resistive impedance, AC return currents will follow the path of least reactive impedance. For AC return currents, the path of least reactive impedance is always below the trace. Return paths are important to remember because they're easy to forget. Check your ground plane and trace out the current return paths. The previous "separation" recommendation also applies here, don't cross the wires, even if the wires are invisible!

(Not) Bridging: If you've followed all the (excellent) advice I've had to offer, you should have two fairly isolated AC and DC systems. Now, if you have gaps in any of your planes and are thinking about bridging them, please do not. This whole article is about AC/DC isolation, and that would defeat what we've taught you!



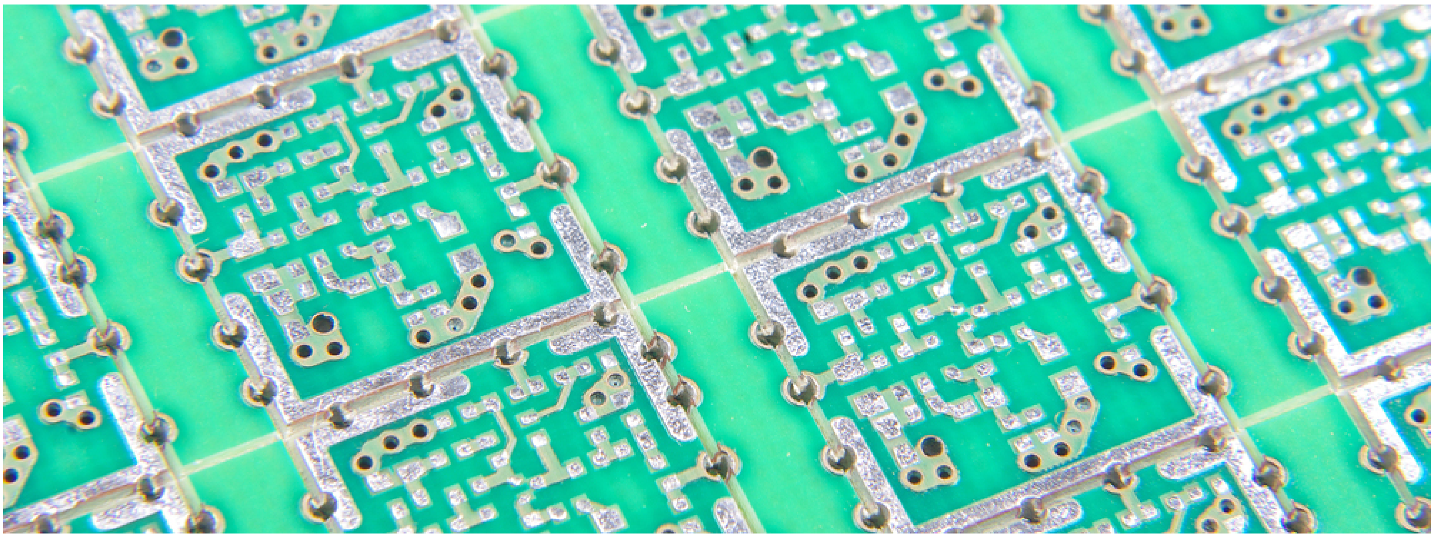
Your PCB design software can help you design your board with separate AC and DC signals.

HOW SOFTWARE CAN HELP YOU SEPARATE AC FROM DC

These rules of thumb may be good advice, but they can't be utilized without a plan for implementation. This is where your PCB board design software comes into play. Isolation can be achieved by color coding your PCB designs. You can track your AC and DC systems by assigning them distinct colors to ensure that they are separate from one another, both physically and electrically. Check out documentation for Altium's Circuit Studio for instructions on how to do this. For more advanced users, color coding is also available in Altium Designer.

Whenever you think back to the good old days of yore, when genius inventors went head to head in battles for electrical dominance, remember that the AC/DC signals on your PCB are doing the same thing. Though, instead of one side winning this war, let's sue for peace on the PCB.

HOW TO REDUCE EMI IN MIXED-SIGNAL SYSTEMS USING PROPER PCB GROUND DESIGNS



Traces and pours can even look like roadways.

Back when I was living in Philadelphia, I used to drive for an hour to and from work. Those of you who have done long commutes know the sort of road rage even a 5-minute delay can set off. Luckily, Philadelphia's city planning is particularly efficient as a result of William Penn's careful designs. Were it not for his efforts in traffic efficiency, I'd be spending more time making hand gestures than steering. Sometimes designing a mixed-signal PCB can feel like city road planning. If it is done without forethought it can cause PCB "road rage", and put out as much electrical "noise" as my car horn. Keeping your board grounded will get you started on the road to an electromagnetic interference (EMI), and rage-free PCB. Here are some suggestions for keeping your board calm and well grounded.

BEST PRACTICES FOR MIXED-SIGNAL GROUNDING

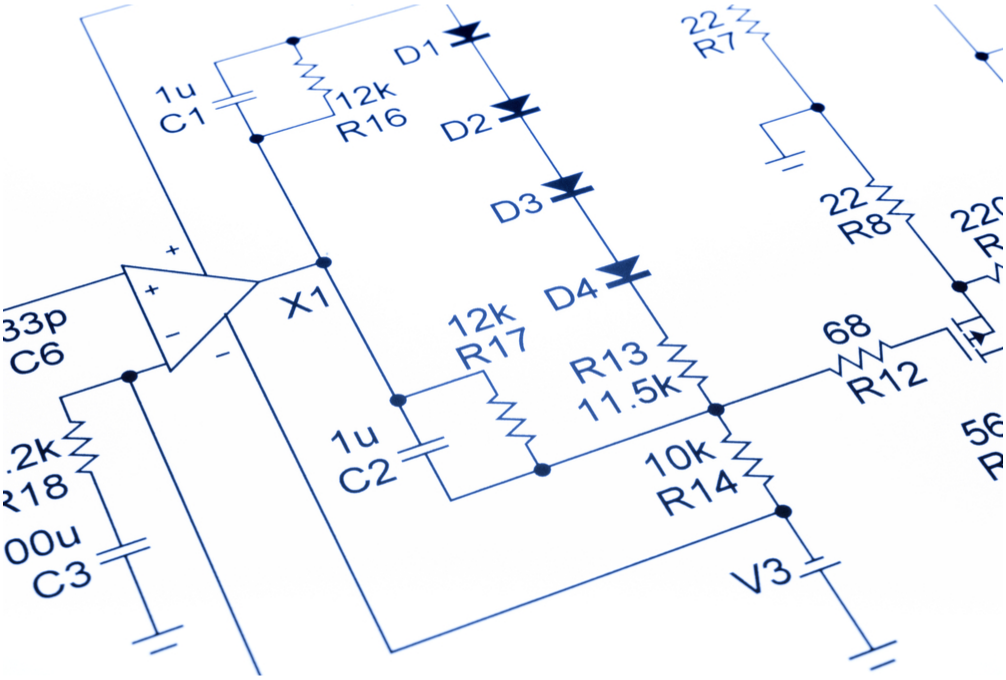
You know that grounding solutions will often need to be specifically tailored for mixed-signal PCBs, but nevertheless, there are several "best practices" that can get you part of the way.

When designing, remember to choose the optimal grounding system; bus wire, grid, or plane.

Bus Wire: For most PCBs with mixed signal systems a bus wire is not a good solution. A bus wire's impedance can be quite large at common system frequencies and this will cause large voltage drops.

Grounding Grid: The grounding grid can be a balanced solution for two layer PCBs that don't have enough space for a full grounding plane, but can't accept the voltage drops associated with a bus wire. The grid does not have to be a square mesh but should have as much area as possible. A larger area will reduce grid impedance and allow for more trace connections, which in turn shortens current return paths. If you use a grid, take special care to make certain AC/DC return current paths do not cross.

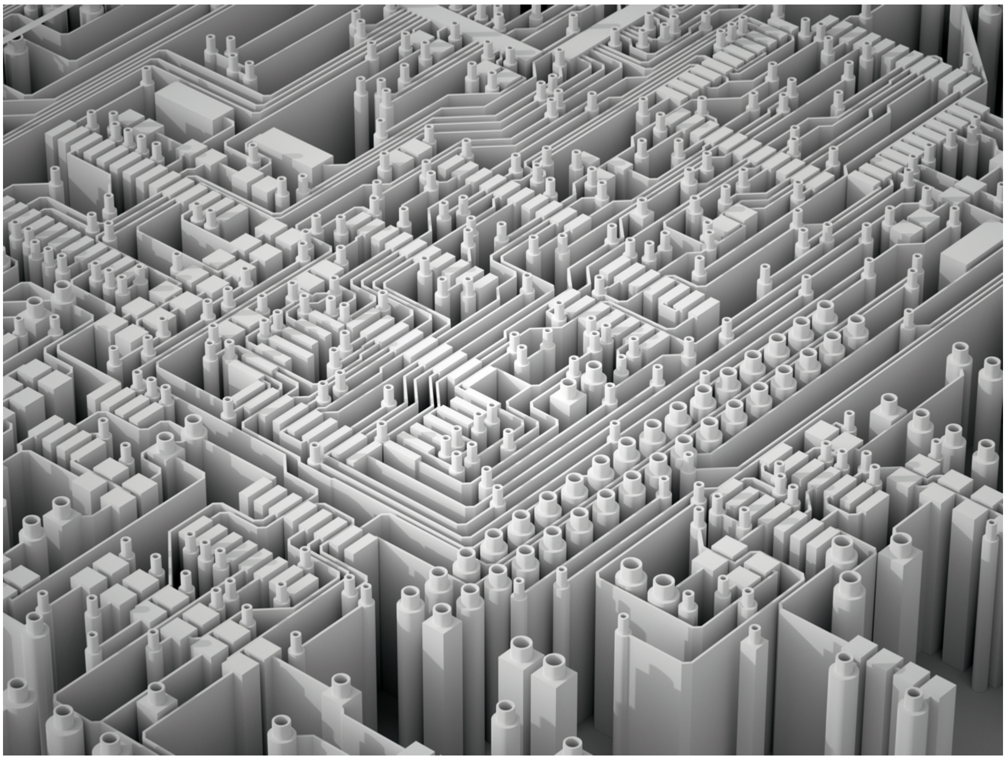
Grounding Plane: Normally the best solution for grounding in a PCB is a full ground plane. Ground plane geometry ensures the lowest possible impedance and often offers the most direct current return paths. Additionally, a full ground plane will shield a PCB more than a ground grid. Just like with a grid, care must be taken when connecting integrated circuits (ICs) to the ground plane to ensure that return current paths do not cross. As is often the case, the "better" the solution, the more costly it will be. If you're as frugal as a Quaker, calculating the impedance associated with using a bus wire or grounding grid may be worth your while.



Real grounding is not as easy as on schematics.

Some mixed-signal designs may use two separate grounding planes in an attempt to separate analog and digital circuits in order to reduce EMI. However, having two completely separate grounding planes will mean references for certain parts of the circuit could be different. This can introduce errors into your circuit. In circuits with one data converter, separate ground planes can be connected using a ferrite bead and Schottky diodes. For more general systems, though, the two grounding planes can be merged, or one plane can be used. This is as long as AC/DC current return paths do not cross each other, which will result in "crosstalk" (like the "crosstalk" I yell when someone cuts me off). I recommend using one solid ground plane, preferably, with carefully planned current return paths. If a solid plane is not possible, cut gaps into the plane to separate AC/DC components and their current return paths. A ground plane with gaps is still contiguous and will be a good reference, but will introduce small amounts of noise to circuits. If you do cut gaps, do not run traces over them, as the gaps can act like antennae. Regardless of the method you use, try to keep current return paths as short as possible. This will help reduce EMI that can be radiated by current loops.

If you have any bypass or decoupling capacitors in your PCB design, connecting them directly to the ground plane will help reduce EMI. Grounding these capacitors quickly ensures that return currents have a very short path to complete their loop. If the capacitor connection path to ground is too long, your return currents might take a shortcut and end up where they're not supposed to be.



Both cities and ICs need good ground.

PRACTICAL TIPS FOR MIXED-SIGNAL ICS

As you know, mixed-signal systems have complications, but mixed-signal ICs can introduce further complications. Different combinations of these require different solutions, here are some tips when dealing with mixed-signal ICs.

PCBs with a Single Mixed-Signal IC

If you've designed audio circuits before you might be familiar with star grounds. When designing a PCB that has only one mixed-signal IC, a star ground can be a great solution. The star ground uses a single point as a reference instead of an entire plane. For converters, and some other mixed-signal chips, the manufacturer normally recommends that AGND and DGND pins be connected together outside of the chip, this connection can act as your star ground. If you're using two separate ground planes with a single mixed-signal IC, you can also tie the two ground planes together at that point. One complication with star grounds is that the connections to the star ground need to be as close to the same length as possible. If that configuration is not practical for your circuit, it's better to use another ground type.

PCBs with Multiple Mixed-Signal ICs

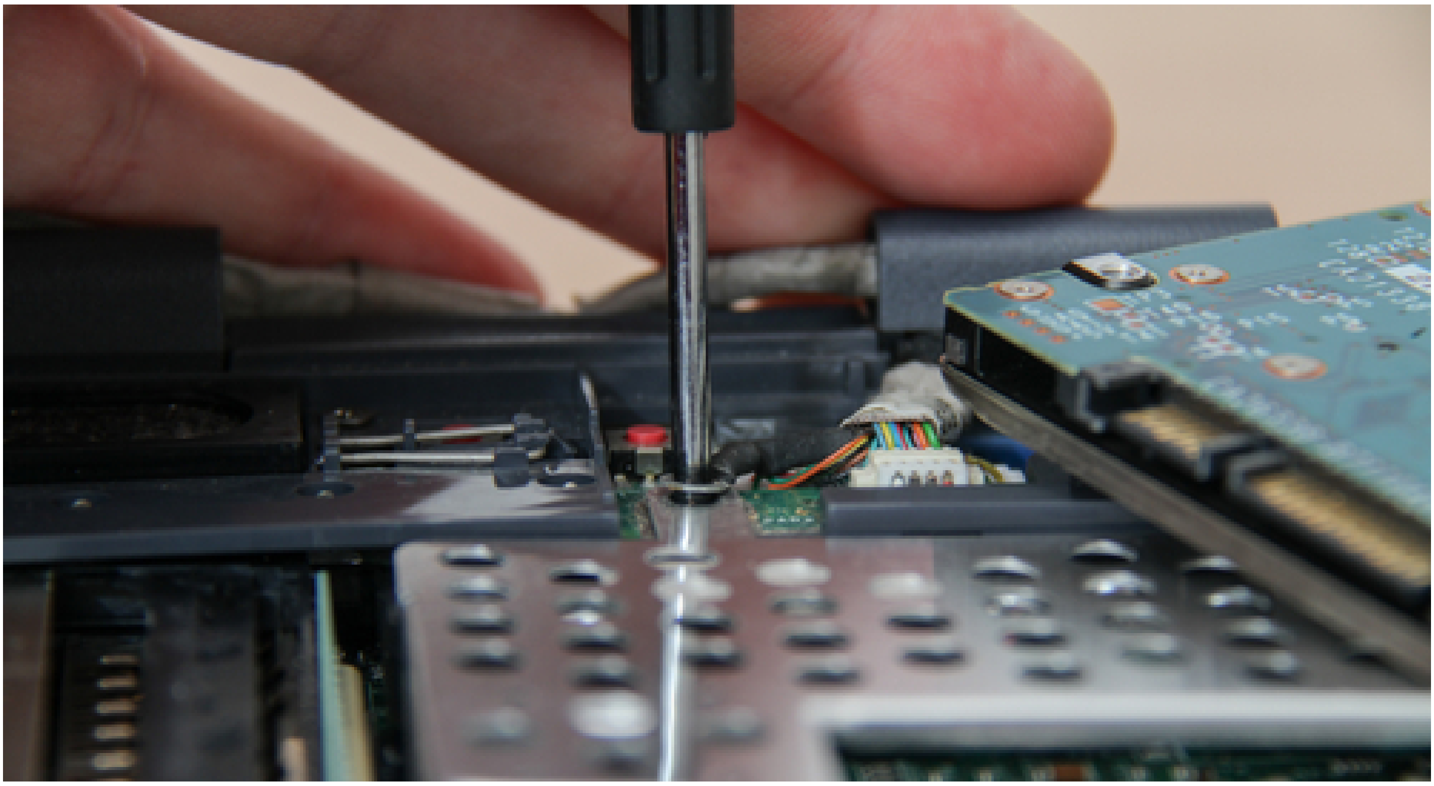
If your PCB is using more than one mixed-signal IC, the star ground is not practical. This is because you would need to tie the AGND and DGND from each IC together right outside each IC case, all at the exact same point. Not even William Penn could plan that intersection. If you're building a PCB that has multiple mixed-signal ICs on it, I recommend using a ground plane with gaps to separate return currents, or without gaps and very carefully checked return current paths.

Mixed-signal system grounds require careful layout planning, as return paths need to be checked to reduce EMI and "crosstalk." Layout planning is very difficult if everything looks the same. I recommend using your PCB design software to color code your components and nets so that you can visually separate AC and DC systems, or pins if you're working with mixed-signal ICs. CircuitStudio has documentation showing how to change colors on your PCB.

Sometimes congested highways and road rage are unavoidable, but congested PCBs are not. A good grounding layout can reduce EMI on your board, and keep you anger free.

Have more questions about grounding design? Talk to an expert at Altium.

USING CANS FOR EMI SHIELDING ON YOUR PCB

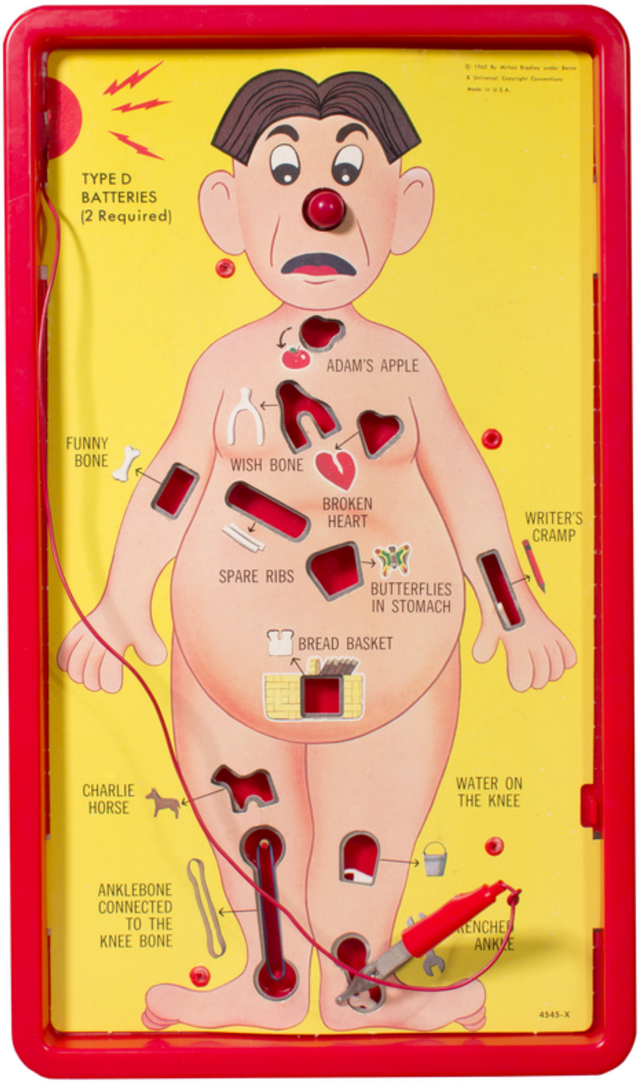


Make sure your can has enough clearance for the components it's shielding.

Preventing electromagnetic interference is critical for proper functioning of your PCB design. In addition to good design practices, can shields can also be used to isolate sensitive components.

Did you ever play Operation? The game where you had to fish little pieces out of an awkward looking body without touching the edges of the slot. When you did, your failure was broadcasted to everyone as a horrible buzzer went off. Obviously, it was one of my favorites.

Since then, in every engineering and IT job I've held, I've been the designated "surgeon" in "PCB Operation" (Grown-up Operation for electrical engineers). However, in this game, there is no buzzer, just dead components if they are incorrectly removed from a module, board, or out of tight casings. This seems to be one of the side effects of having dainty (some would even say elfin) fingers in a field primarily staffed by more senior guys.



Fishing PCBs out of an awkward casing is like playing Operation, except your components might actually die if they get shocked.

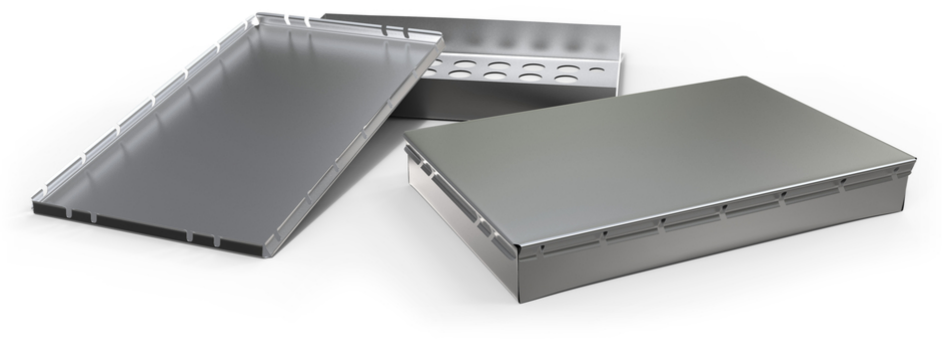
Editorial credit: digitalreflections / Shutterstock.com

Being able to fish out our fallen components was a particularly valuable ability for one prototype product. We were losing about a third of our boards in field installations when a particularly ESD (electrostatic discharge) sensitive component was accidently touched. With tiny fingers, you could slide the PCB into the casing while only holding the edges, but normal sized hands were hopeless and always mashed into the components. We were working to reduce EMI (electromagnetic interference) in the next version and put a can over these components. This also happened to also protect the board from inadvertent fingers.

ESD shields are not the same as EMI shields, so we were very lucky to get two birds with one stone when we added protective shielding to our board. To protect your PCBs, it's important to understand how a can shield works when you're planning board level shielding and certification testing.

WHAT IS A CAN?

EMI can shields may also be called cans, cages, covers, or lids, depending on where you look for parts. They are basically metal boxes that attach to your PCB to enclose part of the circuitry on the surface.



RF can shields are metal enclosures that help provide board level shielding after your initial design procedures to optimize any EMI and emissions.

The metal used depends on your application and the price you're willing to pay. I recommend talking directly to a manufacturer to be sure you'll get the right material for your product. The usual options are:

- Steel: Tin or zinc plated steel, stainless steel
- Aluminum: Usually tin-plated aluminum
- Brass
- Copper alloys: Copper beryllium is especially common
- Nickel
- Silver
- Tin plated plastic

The metal enclosure of the can keeps EMI from entering or leaving the covered region. Many can shields have a grid of holes in the metal that helps with thermal management, while still providing the shielding effect of a conductive cover over your components.

An ideal shield would completely encapsulate your components. Unfortunately, that leaves no options for inputs and outputs, or power and grounding, so a can will still have some leakage. If necessary, you can supplement the can with additional shielding, like gaskets, mesh, and films.

WHEN SHOULD I USE A CAN?

When you're designing a PCB, you should first design your board to minimize emissions with good design practices like short traces, proper grounding, and component placement to minimize emissions. Using shields should never take precedence over good board design for EMI management, but they are a great next step.

Shields are particularly appropriate when you need to isolate components from EMI that might occur elsewhere on the PCB. They are most often used over the RF output, or inputs and amplifier stages, since these are the sections of a circuit most likely to be affected by/cause EMI. In our design, the shield covered both the RF module and amplifier to isolate them from other noise in the circuit.

EMC sensitive components can also be shielded to prevent interference. Finally, you might want to shield high-speed components like oscillators to prevent them from creating interference across the board.

HOW DO I INCORPORATE IT INTO MY DESIGN?

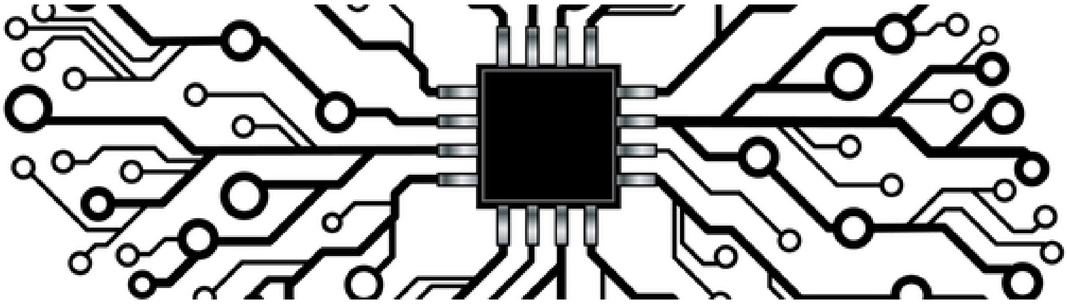
When you've identified the components or subassemblies that can most benefit from can shielding, you can calculate the shield size you need. This can be tricky because most manufacturers give you external dimensions, even though you need to know internal clearance to fit around your components. I speak from experience when I say that height is especially critical and easily overlooked.

To attach the can to the board, you add solder pads around the necessary components on your PCB and ground them appropriately. There are also clips or frames that hold the can onto the PCB. If you are in an early version of the design, I recommend a removable option. It makes rework under the can much easier. The shield or clips can be attached during automated fabrication processes by soldering the shield to surface mount pads.

You can have custom shields made, if necessary. It's more expensive, but if you have a non-standard area that you need to be shielded, and don't want to waste board space, it's worth it. Also, be aware if you have traces on the surface of the board that the can would contact. You should have recesses cut into the edge of the can to prevent shorting on the board. I know someone who got around that by always using clips, but I worried that large mashing fingers might push the edge of the can into contact with the trace anyway.

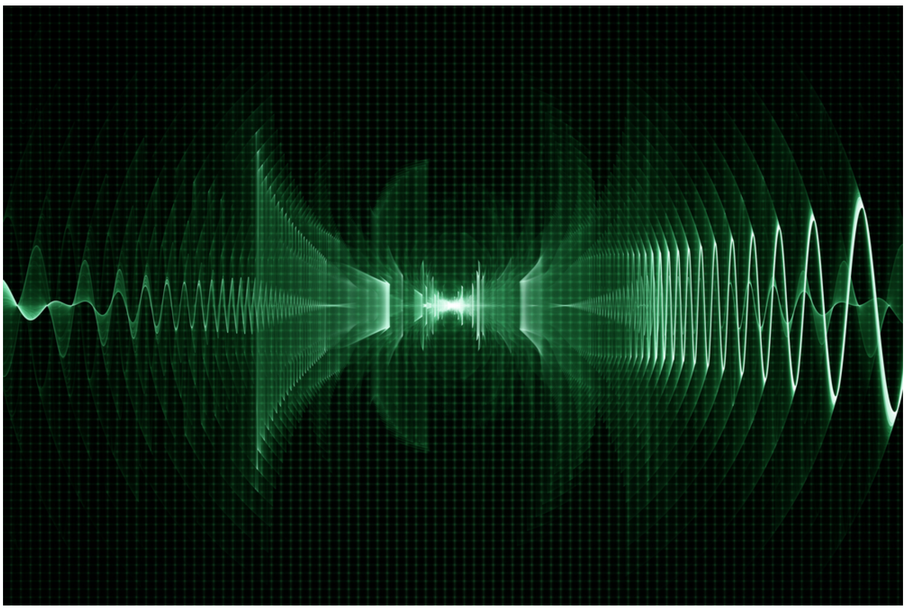
While you can't protect your PCBs from every accident that comes along, you can certainly shield your boards from themselves (and hopefully some probing fingers). I really like having modular RF designs that you can reuse in future products, and minimize your non-recurring design effort. Altium Designer is a tool that makes this much easier. It can also save you from my sizing mistakes by using built-in 3D clearance checking to make sure the cans are the right height for your components. To start protecting your PCBs, reach out to an Altium specialist today!

HIGH-SPEED PCB DIFFERENTIAL PAIR ROUTING TO PRESERVE SIGNAL INTEGRITY



Proper routing of differential pairs in high-speed digital circuits is necessary for preserving signal integrity. Follow these guidelines for differential pair routing and leave your EMI troubles behind.

I once went on a blind date with a woman who, unbeknownst to me, was always late. I got to the restaurant right on time and waited for 20 minutes before I figured I had been stood up. As I was about to leave, my date arrived. If she'd been 5 minutes later we would've missed each other altogether. The same kind of thing can happen on high-speed PCBs when differential pairs aren't routed correctly. One signal will arrive where it's supposed to be, find its partner missing, then go home. On PCBs, though, a missed date doesn't meen hurt feelings. It means the circuit will have poor signal integrity or won't work at all. Be a good matchmaker for your high-speed signals and route them so that they both arrive on time.



This is your high-speed signal with bad routing.

DIFFERENTIAL ROUTING TIPS AND TRICKS

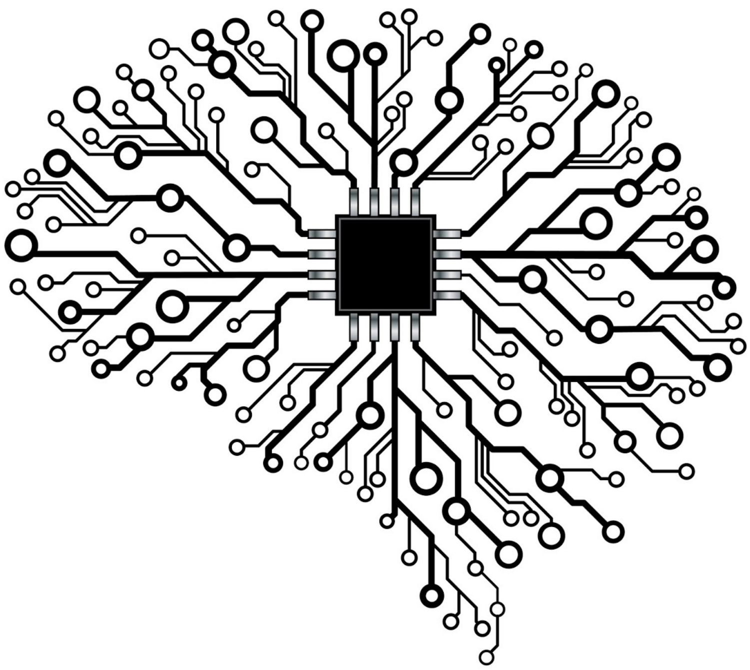
On later dates, I used a few tricks to try and get my date to arrive on time. The ethics of tricking my date into arriving on time are debatable, but tricking differential pair signals into punctuality will ensure signal integrity. Use these tips to make sure differential pairs are routed with timing in mind.

Trace Length Matching: Trace length matching should be a top priority when routing differential pairs. Don't make one signal go all the way across the PCB while the other one just has to go next door. When differential pair trace lengths are mismatched the timing difference will cause destructive interference and degrade signal integrity. Similar to how my height preferences for a date may vary from yours, different circuits have different trace length mismatch tolerances. Make sure your differential pairs see eye to eye, check their mismatch tolerances before beginning design.

Parallel Routing: Whenever routing a differential pair, try your best to keep their traces parallel. Routing a differential pair in parallel helps cancel out any radiated EMI, and assists with trace length matching.

Electrical Clearance and Creepage: Like current and ex-girlfriends, separate differential pairs should always be kept as far from each other as possible. When multiple differential pairs are routed in close proximity they will always interact with each other in negative ways. Keep fights for dominance and EMI to a minimum with lots of distance.

Differential pairs will also need to be routed far from components susceptible to EMI. This distance is measured in both clearance and creepage. There are many different approaches for meeting your circuits' clearance and creepage requirements.



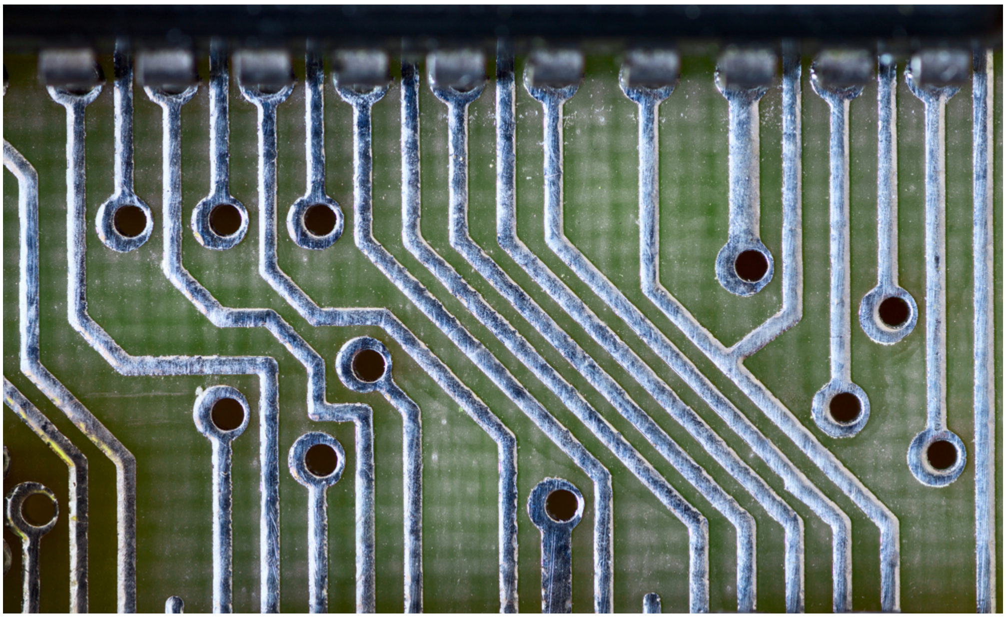
Be smart, don't route your differential pairs like this.

No Sharp Turns: It is best to route a differential pair straight, with no turns at all. However, your PCB layout may require turns. Some women like smooth men, but differential pairs always like smooth curves. Sharp edges on turns will radiate much more EMI than a smooth curve. Any changes in direction in a differential pair should not deviate more than 45 degrees. This is important at both the inside and outside edges, as both can radiate EMI.

Vias: Just like it's not a good idea to have lots of girlfriends, it's not a good idea to use lots of vias. Via geometry guarantees at least a small amount of signal degradation. When used too often vias can significantly degrade signal integrity and cause destructive reflections in the differential pair.

If you do end up using vias on your PCB make sure to either shorten via stub length or back drill the stubs. A via stub will act as an open-ended transmission line, which means lots of signal reflection. Depending on the length of the stub, the signal can even be reflected back into the differential pair at 180 degrees and cancel out the useful signal. The best way to reduce the negative effects of a stub is to minimize the stub's length. Stub length can be minimized by using blind or buried vias, or by back drilling via stubs. All of those options can increase manufacturing cost, so if you're on a tight budget, you can simply make the via connections on distant board layers. In an 8 layer board, a 1-7 connection will have a shorter unused stub than a 1-2 connection.

It is also important to match any amount of signal delay caused by a via. This can be done by either using the same number of vias in both legs of the differential pair or by adding some serpentine routing on the leg without a via. No one likes to be the third wheel on a date, so make sure everything is matched evenly.



Look at those beautiful parallels.

HOW TO MAKE YOUR COMPUTER DO IT

In order to optimize brain power, let your computer do some of the work. Your PCB design software should be able to automatically check some of these rules, such as electrical clearance. More advanced software will also help you with the actual routing of differential pairs. Altium Designer may not have solutions for your love life, but it does have features to help users with differential pair routing.

Despite our timing differences, I continued going out with my blind date. The timing issues persist, but our relationship has retained its integrity. If you have problems with differential pair routing on your PCB, your board may not be as lucky. Make sure to follow the above guidelines to preserve signal integrity on your high-speed PCB.

Want more advice on differential pair routing? Talk to an expert at Altium.

HIGH-SPEED PCB DESIGN PRINCIPLES: KEEP TRACES SHORT AND DIRECT

Your signals travel at the speed of light, and so does their EMI.

As you know, high-speed PCBs can often have issues with EMI. This EMI risk can come from long traces that form transmitting or receiving antennas. Longer traces also cover more distance, which increases the chances that sensitive circuits are affected along the way. Using short traces that take direct paths will help reduce EMI risk from both antennas and trace proximity.

I never really believed in the idea of “less is more” until I experienced it for myself at a cookout with some friends. I was the “grillmeister” which meant I was in charge of seasoning and grilling the steaks. I decided to go with a simple salt and pepper seasoning, and being a salt lover, thought that the sky's the limit. Well, the steaks turned out to be too salty to eat for most of my friends. Salty steak will leave a bad taste in your mouth, and so will a high-speed PCB with lots of long traces. Long, winding, traces on your PCB can increase EMI and cost you in real estate. Practicing “less is more’ by keeping traces short and direct will ensure that no one finds your designs unpalatable.

LONG TRACES INCREASE EMI RISK

As you know, high-speed PCBs can often have issues with EMI. This EMI risk can come from long traces that form transmitting or receiving antennas. Longer traces also cover more distance, which increases the chances that sensitive circuits are affected along the way. Using short traces that take direct paths will help reduce EMI risk from both antennas and trace proximity.

Much like how no one intentionally over-salts a steak, no one intentionally routes a trace so that it is an antenna for EMI. Maybe a high frequency altered a capacitor’s impedance, or a return current made an unexpected loop in the ground plane. Whatever the reason, the best way to mitigate antenna radiation is to shorten traces. Antennas can transmit or receive, so traces in both “noisy” and “quiet” circuits should be shortened. A smaller antenna will always radiate or receive less than a larger antenna.

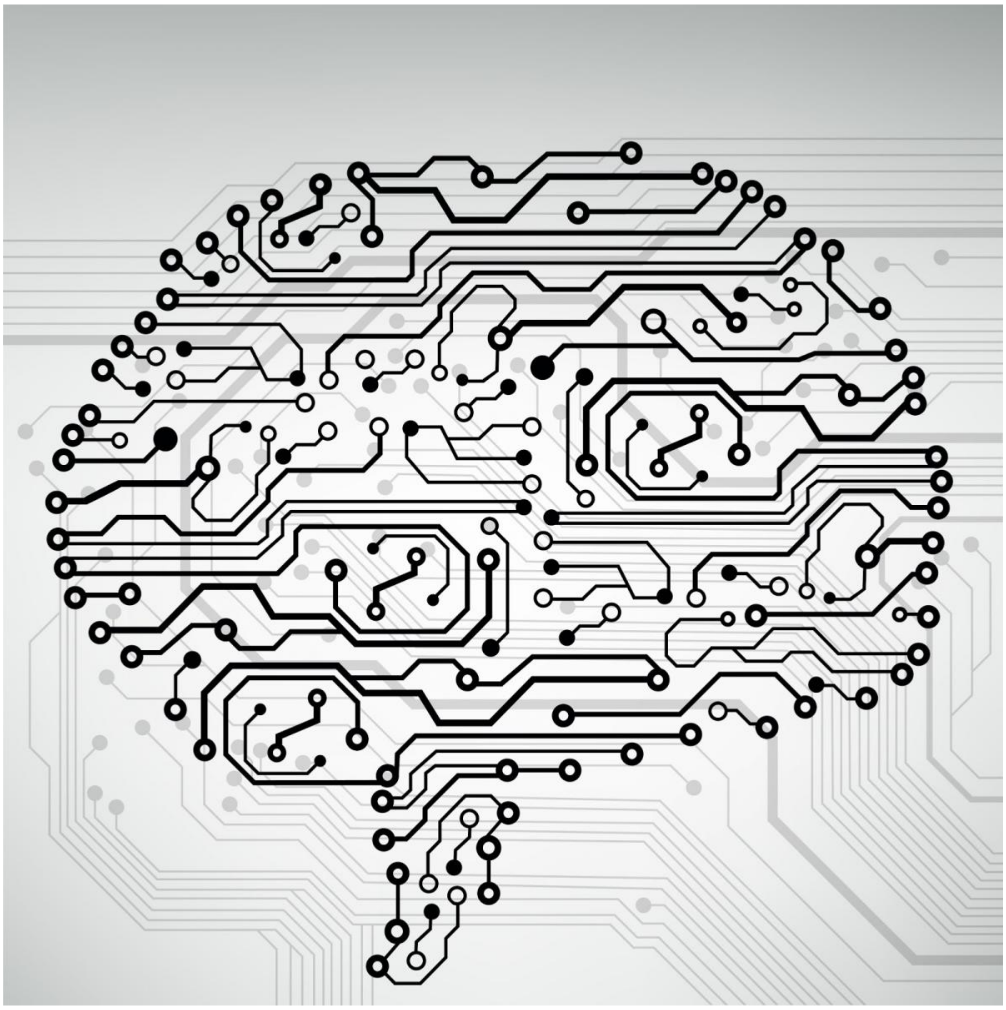
Traces themselves aren’t the only things on the PCB that can radiate EMI. Gaps in the ground plane will also act as antennas when crossed by a trace. Keep traces short and direct, but not at the cost of crossing gaps in the ground plane.

A transcontinental trace would definitely be too long.

LONG TRACES COST MORE

The longer the trace, the more real estate it will take up. Less trace means more money.

PCB designers are always being pushed to make things smaller, so I'll add to the problem by pushing you to make traces shorter. Not only will the trace itself take up space, but it will also require more space for clearance around the trace. Your company's accountants might already be, as the kids say, "salty." Don't oversalt them by making them pay for an excessively large PCB.



Trace Direct and Think before you route.

HOW TO SHORTEN TRACES

The best way to shorten high-speed trace length is to carefully lay out your PCB. Keep high-speed systems near each other and keep receivers near their respective inputs.

A good PCB layout can help solve problems before they even happen. Setting up your layout so that high-speed systems are close to one another will allow you to initiate your design with short trace lengths. It is especially important to keep the signal input to receiver path as short as possible, as this line can inject the most "noise" into a system. Additionally, keeping high-speed components in close quarters will lessen the effect of any radiated EMI on any other circuits. Close the gap before drawing the first trace.

These design principles may be almost common sense, but implementing them can be tiresome. Fortunately, your PCB design software can help you. Try color coding your high-speed systems to help keep track of components and traces. Software with good trace routing features will also make it easy for you to route, or reroute, traces to the shortest possible path. It just so happens that Altium Designer already has documentation showing how to color code components, and how to arrange traces.

After that "less is more" cookout fiasco, I've been much more careful when seasoning steaks. Learn from my mistakes and apply "less is more" to your high-speed PCB, before long traces leave a bad taste in your mouth. Trace direct the right way with expert tools and resources.

Have more questions about high-speed PCB design? Talk to an expert at Altium.

ADDITIONAL RESOURCES

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

- EMI/EMC Design: PCB Noise Reduction Through Isolation of AC and DC Signals
- How to Reduce EMI in Mixed-Signal Systems Using Proper PCB Ground Designs
- Using cans for EMI shielding on your PCB
- High-Speed PCB Differential Pair Routing to Preserve Signal Integrity
- High-Speed PCB Design Principles: Keep Traces Short and Direct