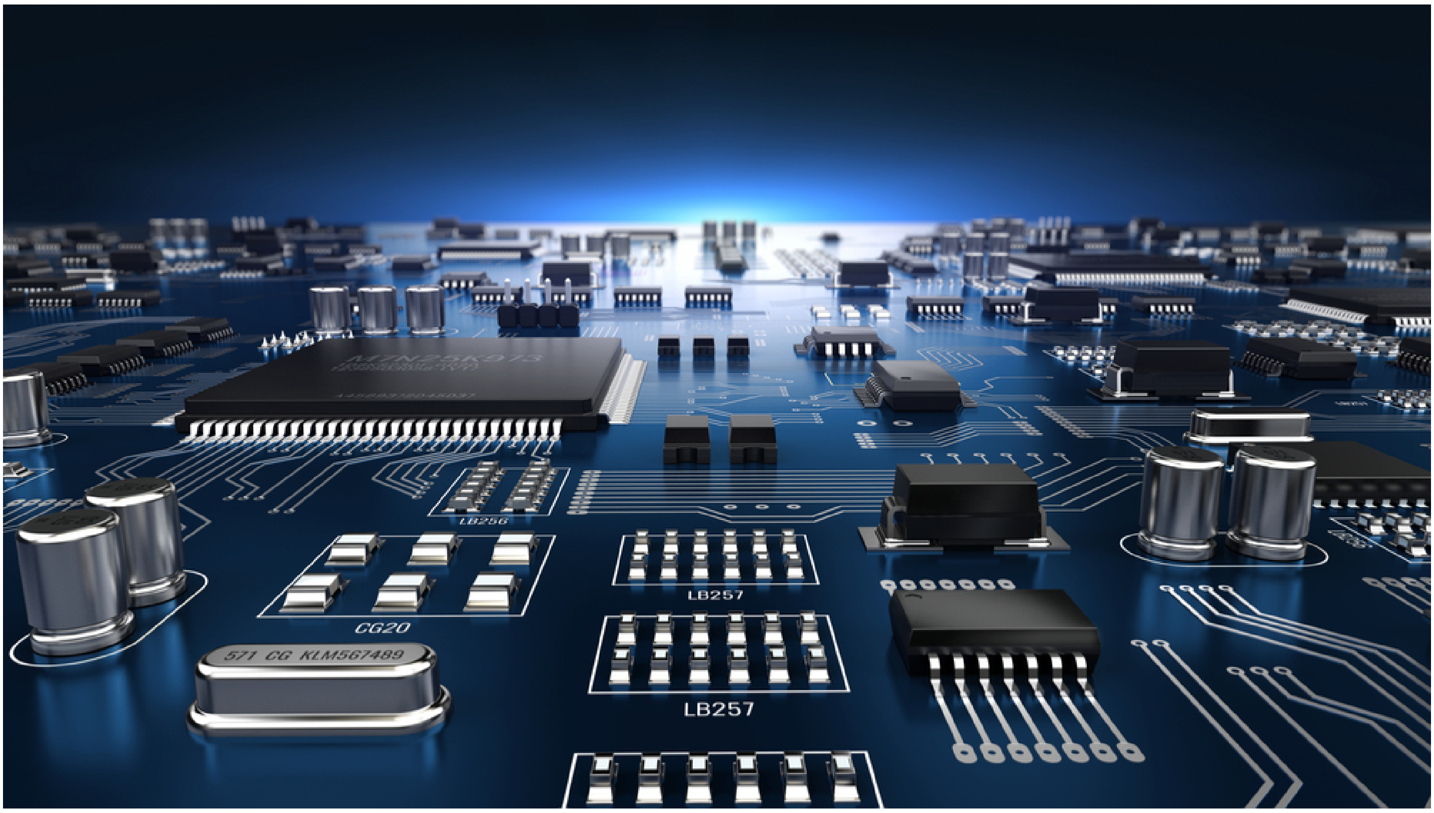
Altium

Protect Your PCB From ESD



PROTECT YOUR PCB DESIGN

From computers and televisions, to automobiles and kitchen appliances, PCBs are found in nearly every aspect of our lives-- many of our devices would not be possible without them. They are the brains of our electronics, and as such, it is of the utmost importance to do everything in our power to ensure our designs are protected from potentially catastrophic threats. Proper routing and schematics, component placement, and special attention to grounds to minimize the risk of ESD damage goes a long way to ensuring your PCB design maintains its efficiency and lifespan.

Join us as we explore topics about protecting your PCB design, including:

- The Correct PCB Routing and PCB Layout to Help Protect Your board from ESD
- Protecting your PCB from ESD using transient voltage suppressors
- How parasitic inductance can impact your ESD protection
- Using grounding to protect your PCB from ESD damage
- How Component Placement and Routing Helps Protect Your PCB from ESD

THE CORRECT PCB ROUTING AND PCB LAYOUT TO HELP PROTECT YOUR BOARD FROM ESD

When I started running, I went with a friend who had finished several ultramarathons. She would run miles from her house, do a short loop with me, and then carry on without me. When I got faster, instead of getting to go home sooner, she made our loop longer. She was tricky about it, too. She always planned a route where it would be boring for me to turn around early, or on new trails so I'd get lost trying to take a "shortcut" back.

As an urban planner, she was very intentional in choosing what she wanted out of a running route. As an engineer, I wanted something very different, usually to run by the smoothie place, but I understand her deliberation. After all, the same thing is true when I design PCBs. I want my routing to accomplish something very specific in terms of cost and performance. Routing is especially important in ESD protection and helps you to keep your components safe from the induced EMI of ESD events.

MINIMIZE CIRCUIT LOOPS

Although our running routes were often circuitous loops, you want to do the opposite on your PCB. If you can minimize circuit loops, this will go a long way towards reducing damage from ESD events propagating across your PCB. This is because a loop that encloses a changing magnetic flux will experience induced current. If that flux is a response to an ESD event, the amount of induced current flowing unexpectedly into your components can cause some pretty catastrophic damage.

Sometimes there's just no other option, and your layout is going to require a loop. In that case, minimize the area of the loop. The magnitude of the induced current will be proportional to the loop size.

USE GROUND PLANES

When you are designing a multilayer PCB, you should definitely use a ground plane. The most common loops formed on PCBs are power to ground traces. Those loops are so ubiquitous that they're easy to overlook.

For designs that can't implement a ground plane, you should use a grid pattern of vias to connect to power and ground. This basically emulates a ground plane with traces. You can think of it like a trellis, with power connections occurring at points along one line, and ground traces connecting along orthogonal lines. Semtech has a really nice illustration, and they recommend connections every 6 cm.

They also recommend keeping power and ground traces close together. However, this can cause etching to occur in your board, especially with AC power.

Using a grid pattern, similar to a trellis, to connect to power and ground can help minimize circuit loops if you're unable to use a ground plane.

OPTIMIZE ROUTING PATHS

Besides minimizing loops, you should try to eliminate traces that run parallel to each other. This is particularly true of parallel traces between interconnected devices. Parallel traces can easily couple to each other. If you have a line protected by a TVS parallel to an unprotected trace, EMI damage can easily propagate through the system despite the protection circuit.

A next level ESD suppression method is to use guard traces. If you simply cannot avoid using a long signal trace (Semtech considers this to be 30 cm or longer), then you can use a guard trace instead. A guard trace runs parallel to a high speed signal. I know I just told you to avoid parallel traces, but a guard trace is essentially sacrificial, providing a return path that cancels much of the radiating field pattern. Thus, it "guards" the nearby traces from crosstalk from your signal line.

TRANSPOSE LONG TRACE LENGTH

If you're designing for high-speed applications, you probably already know you should keep your traces short. You want to avoid creating unintentional radiating antennas across your PCB, or long linear loops. However, once again the fates may have it in for you, and there's no way around a long signal or power line. However, you can break those loops up by transposing the signal and ground line. Now, you have lots of smaller loops, like using twisted pair wiring, but inside your PCB layers.

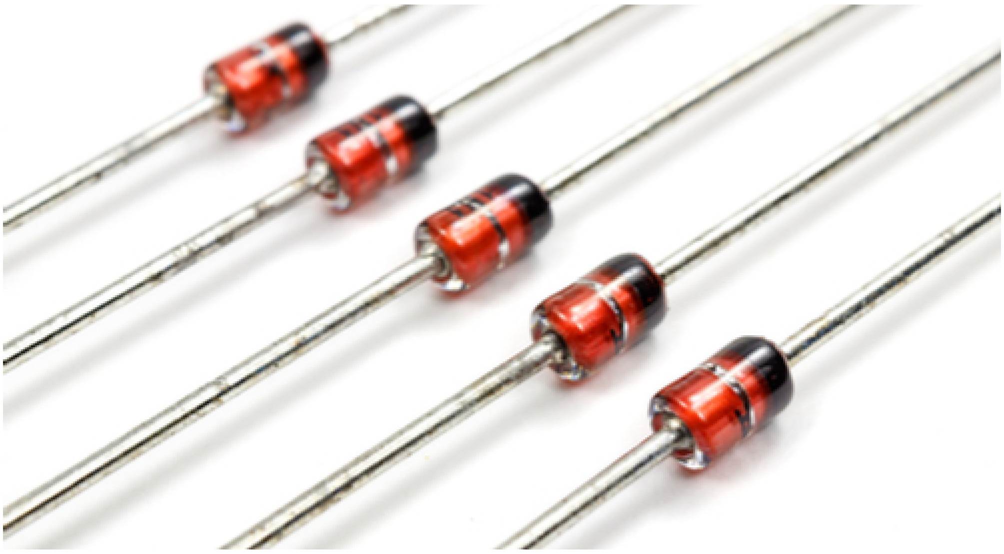
Transposing PCB traces is like using twisted pair wiring to reduce EMI.

Whether it's for a training run or for PCB traces, planning your routing in advance is worth the extra time. Improving your speed, performance, and protection doesn't come easily, but the results stay with you for the long haul. Even if you don't have a friend to do routing for you, there are excellent tools that can help. Software for PCB design, like Altium Designer, provides a unified design and schematic environment that can help you get your PCB design just right. Altium support can help you get started now!

PROTECTING YOUR PCB FROM ESD USING TRANSIENT VOLTAGE SUPPRESSORS

Way back in undergrad circuits class, our TA loved to give us little "black box" circuits. We'd measure the voltage and current characteristics, then have to determine what components made up the circuit. Our TA particularly loved to throw diodes in there. At the time I couldn't figure out why, as I didn't see much purpose in diodes. However, I did come to appreciate diodes when I was looking for ways to protect my PCBs from electrostatic discharge. Diodes are a crucial component for managing voltage levels, which is a critical effect of electrostatic discharge (ESD).

ESD can come from many sources, even just an inadvertent touch, and cause a voltage discharge that wreaks havoc on your board. One of the most common sources of ESD is at inputs, like cables being plugged in or buttons being pressed. Here, you'll want to minimize the voltage spike that will reach your sensitive components.



Despite the underwhelming appreciation of diodes from my misspent youth, they are a critical component to a lot of circuit design and protection.

HOW TO APPLY ESD PROTECTION ON INPUTS

The first line of defense in ESD protection is to minimize the impedance of the path to ground. It's a way to minimize the voltage (VESD) that the board will experience during a discharge. When you consider Ohm's law, VESD=IESDR, IESD isn't something you can control, so the only way to minimize VESD is to minimize R. The problem is that you can't just short the path to ground, because the ground plane will provide a direct path to all of your sensitive components.

Instead, you can add a protection circuit at the input in the form of a transient voltage suppressor. Transient voltage suppressors, or TVS, consist of two diodes combined with an avalanche diode, and they also come as a single component. This significantly reduces the chances of your components being damaged without significantly increasing the cost or complexity of your design.

The TVS subcircuit basically creates a current divider between the TVS and the IC or components that you want to protect. It presents a high input impedance to the input and thus doesn't interfere with normal operation. However, when the input current spikes dramatically under an ESD event, the TVS reaches its breakdown voltage and shunts the current to ground, instead of the sensitive components you are trying to protect.

Each time something is plugged into your device, it introduces an opportunity for an electrostatic discharge.

HOW TO USE TVS PROTECTION EFFECTIVELY

When you add a TVS to your design, you want to smooth out your traces as much as possible to minimize any EMI generated at corners and propagating across the board. The Texas Instruments recommendation is to use large radius curves instead of corners. If you can't because of the PCB technology, then use a maximum corner angle of 45°. To me, it looks like a 135° angle, where instead of a right angle, you cut off the corner, like a stop sign, and have two very obtuse angles instead. Also, I highly recommend TI's Technical Resources section when you really need to get down in the weeds to solve a problem.

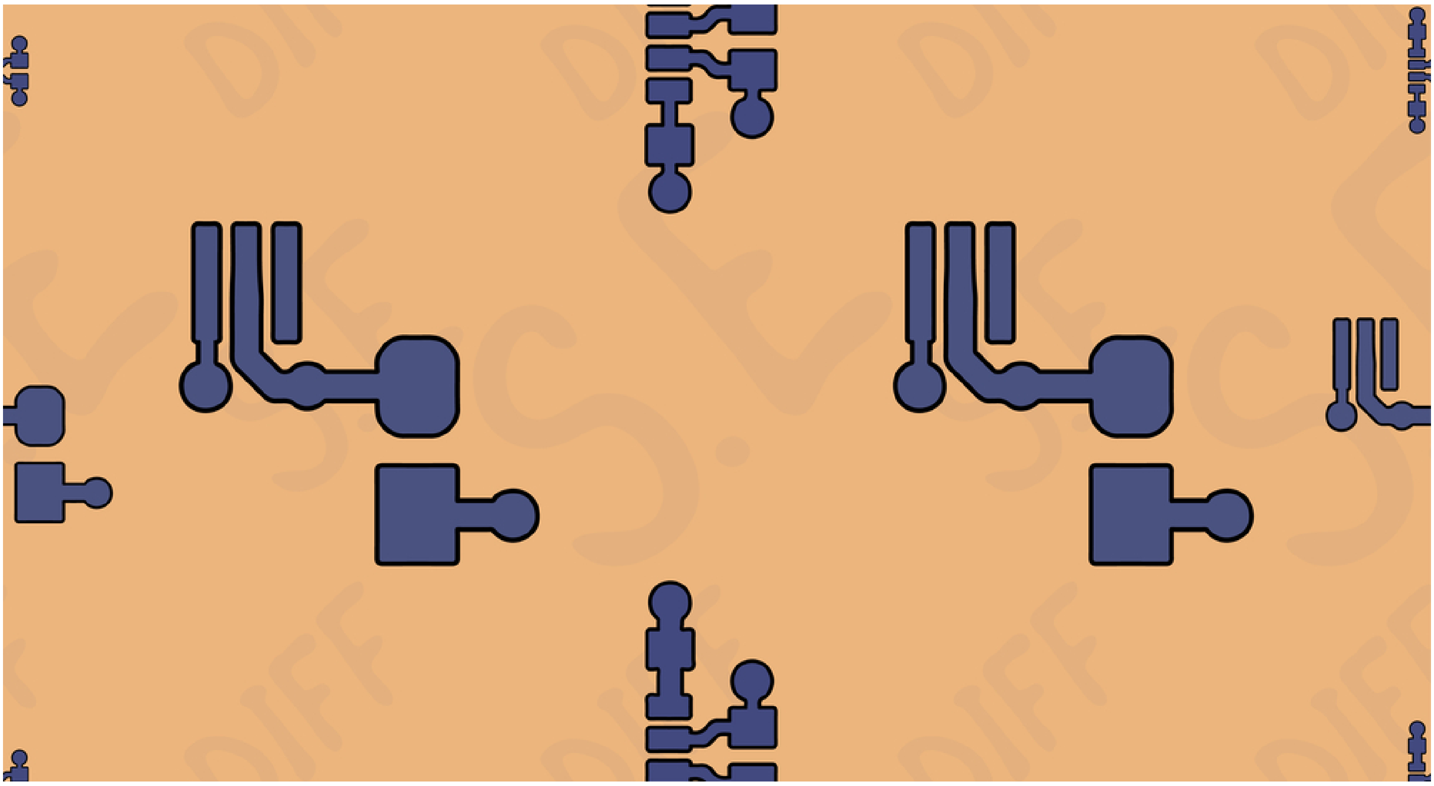
You should use wide angle turns in your traces near ESD protection circuits. Sharper corners are more likely to generate EMI as the voltage spike dissipates.

While we're on the subject of traces - you should avoid using a VIA to connect the ESD source (your input) to the TVS if it is at all possible. Stick to traces for the TVS, since VIAs can really complicate ESD protection. When you're so close to a likely source of ESD, it's a mess you don't want to get into.

Finally, you should also include a buffer resistor in your protection circuit. You add it in series between the possible ESD source and the IC you're trying to protect. That helps decrease the peak current that will reach the IC from the current divider you've added at the input. Some TVS diodes can "reset" themselves from a massive voltage spike, but not all. If they've been cooked from a discharge, they might short to ground and eliminate any protection you were getting from the TVS. Diodes are mighty, but it's good to have a second line of defense.

Choosing the right TVS for your application requires careful consideration of the inductance on your board and the voltage range you need to protect against. Texas Instruments goes into exquisite detail if you're ready to start your design. While you're designing, you can minimize the work you'll need to repeat in your next product by using modular designs and internally managed requirements. A great tool for this is with Altium Vault, which can be used in conjunction with PCB software, Altium Designer. While it won't pick the components for you (or do your circuits labs), it makes your design flow so much easier. Altium representatives can help you get started today!

HOW PARASITIC INDUCTANCE CAN IMPACT YOUR ESD PROTECTION



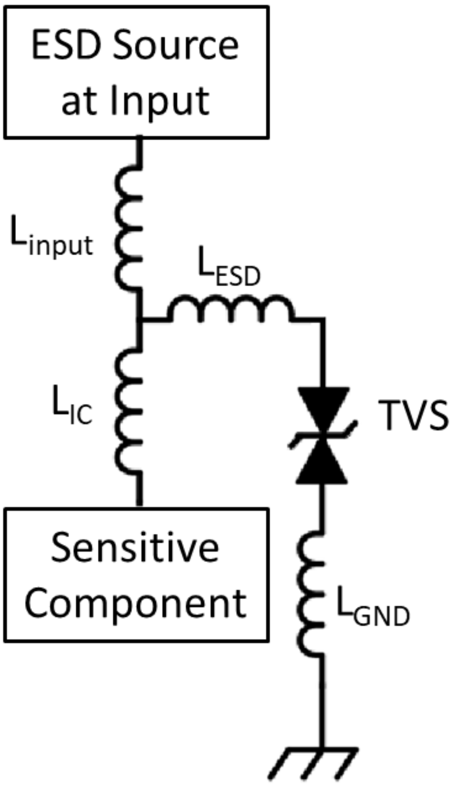
When I was a kid, I was certain I was going to be a biologist. I collected all kinds of lizards, tadpoles, and insects, and spent most of my allowance on aquariums for my various creatures to live in. However, one thing really held me back. I am super grossed out by parasites. I will let a mantis or snake crawl all over me, but even looking at a tapeworm gives me dry heaves. Eventually, I went into engineering, where nothing needs weekend feedings. While parasitic circuit parameters are bad news, they've never made me throw up at work.

In particular, parasitic inductance (L) can have a significant impact on how effective your electrostatic discharge (ESD) protection is. Managing your interfaces and using transient voltage suppressors (TVS) at your inputs are critical first steps. However, if you don't minimize parasitic inductance all that work can go to waste. This is especially true when you're using a TVS. If a TVS diode experiences high parasitic inductance, in the event of an ESD pulse the voltage might be allowed to overshoot dramatically and not protect your components at all.

I kept lots of little creatures, like anoles, but a strenuous aversion to parasites kept me from a career in biology.

WHAT DOES PARASITIC INDUCTANCE DO TO MY PROTECTION CIRCUIT?

We can work this out if you look at the inductance in a TVS protection circuit and think way back to your introductory circuits class.



The parasitic inductance in a TVS protection circuit.

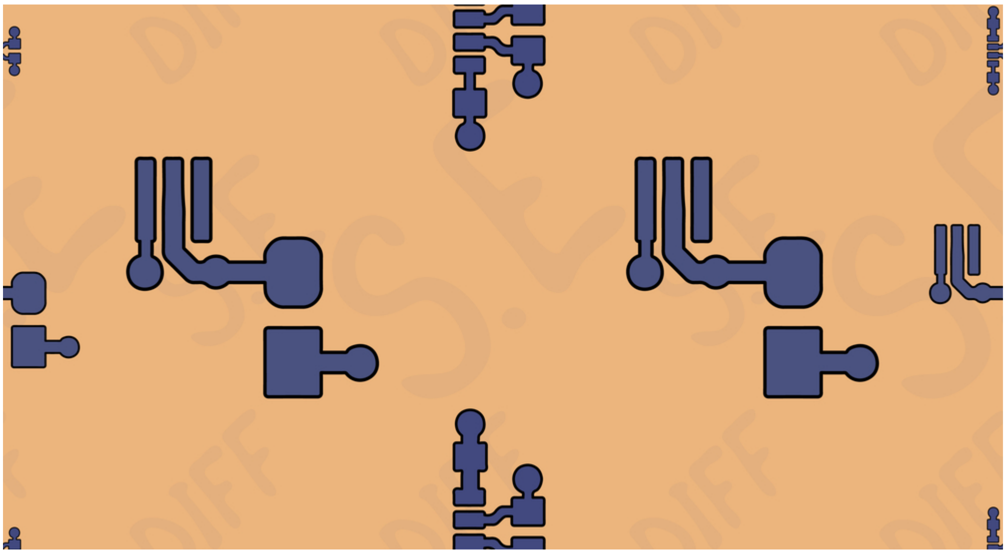
The voltage of the ESD pulse (VESD) can be considered: VESD = VBREAKDOWN(TVS) + RDYNAMIC(TVS)*IESD + LESD(dIESD/dt). If you really want to dig deep into the math, Texas Instruments has an amazing walkthrough. The key takeaway for those of us skimming this over lunch is the last term: LESD(dIESD/dt). Because t is very tiny, dIESD/dt will be huge. Even if the inductance LESD is very small, you can still have a huge voltage spike in the system.

HOW DO I MINIMIZE MY PARASITIC INDUCTANCE?

So what do you do if the problem is inherent in the traces of your PCB? The key is smart component placement to minimize shunt paths and the resulting parasitic inductance.

Minimize any inductance from the TVS to ground by keeping the trace short and by using direct routing. Don't use a stub or via to connect to the ground plane, so there's no additional path length or material to contribute to LGND.

The same is true of the input to the TVS: keep it short, don't use vias or stubs. LESD can also contribute negatively to your parasitic inductance and protection capability. Keep the TVS close to the input connector, too. In addition to keeping parasitic inductance low, it will help prevent transient coupling of the ESD pulse into neighboring traces.



Minimize trace length and don't use vias to connect to your TVS.

HOW DO I POSITION MY SENSITIVE COMPONENTS?

Keep the sensitive components that you are protecting further away from from the TVS. You don't just want to keep the inductance LESD small, but you'll also want maximize the ratio of LIC to Linput on the protection line. As Machine Design explains it, "The nonlinearity of LIC acts as a buffer to the initial peak of the ESD current pulse. This creates a substantial voltage drop toward the IC. This inductance gets smaller the closer the ESD device gets to the IC, and the voltage drop shrinks to the point where it provides no additional advantage."

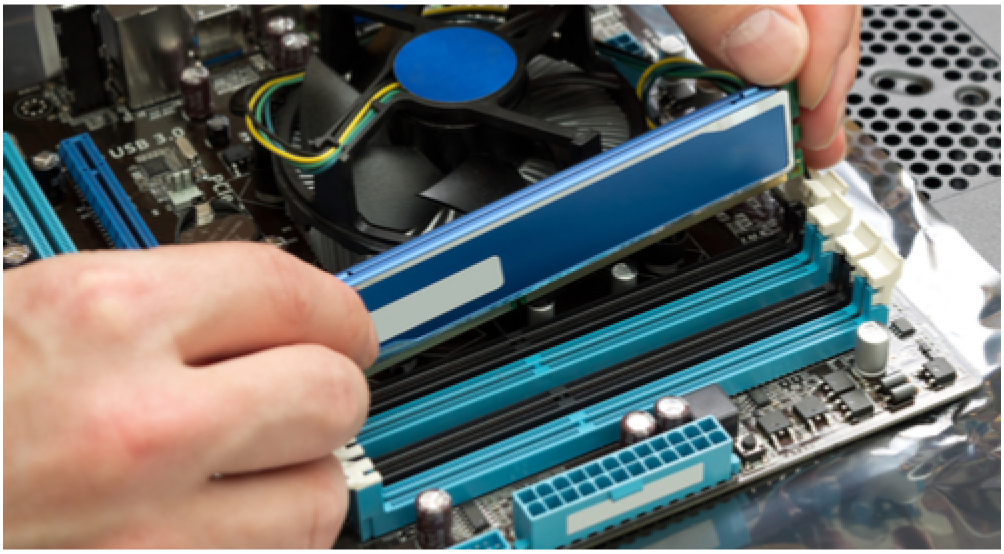
Basically, by putting your sensitive components further away from the input and TVS protection, you can get some of your parasitic resistance to work for you by decreasing the voltage spike of the ESD pulse that your components experience.

While the getting placement right is certainly not as terrifying as a worm that controls a cricket's mind (warning: you can't unread this), it may not be something you want to do repeatedly. If you are using similar protection methodologies in multiple products, you can design your circuits once and use modular designs to make re-use easy. PCB software, like Altium Designer, makes modular designs simple to implement and helps you protect your PCBs. You can contact an Altium representative to help you get started.

USING GROUNDING TO PROTECT YOUR PCB FROM ESD DAMAGE

Editorial Credit: Willrow Hood / Shutterstock.com

I was in high school before I considered engineering as a career outside of the engine room on any Star Trek show. Although, subconsciously I was definitely interested in it. The signs were there early considering I could name all the chief engineers from Star Trek. The deal was sealed though when I was ridiculously excited to receive an ESD grounding bracelet as a gift. Don't have one in your jewelry box? Well, it's composed of elastic webbing with a wide metal strip that goes against your skin and a cable with an alligator clip that you attached to ground. We only had dial-up internet at the time, but that didn't stop me from spending hours waiting for pages to load so I could figure out how to ground myself. Armed with my bracelet, I convinced my friends to hire me to upgrade their computers' RAM, or just let me open up their computer.



Although grounding can be as simple as touching something metal and not scuffing your feet, it's best to be grounded before handling anything ESD sensitive.

Using grounding to protect from electrostatic discharge is necessary at many stages in your product development. When you're handling sensitive products, like RAM cards, good practice includes using an ESD mat and grounding yourself. You can also provide grounding protection for your products by designing them properly. It's better to apply good grounding practices to your PCB design and reduce the dependence on the safe handling practices. As Spock would say, it's illogical to assume that all future customers or installers will obsessively ground themselves when handling sensitive products.

USE GROUND PLANES

There are a number of ways to utilize grounding for ESD protection, but first and foremost is using a ground plane. Although it's not always feasible to use a multilayer design, the ground plane can really help you out if you're concerned about ESD protection. As you know, a sudden voltage discharge will induce electromagnetic fields. A properly connected ground plane can reduce the damage from this by routing the current away from sensitive components.

Using a ground plane gives you another way to reduce the area of circuit loops in power to ground traces. When you reduce the area of circuit loops, you'll decrease the total EMI induced within the loop area. That, in turn, decreases the corresponding current that can flow into components where it shouldn't.

PROTECT YOUR GROUND PLANES

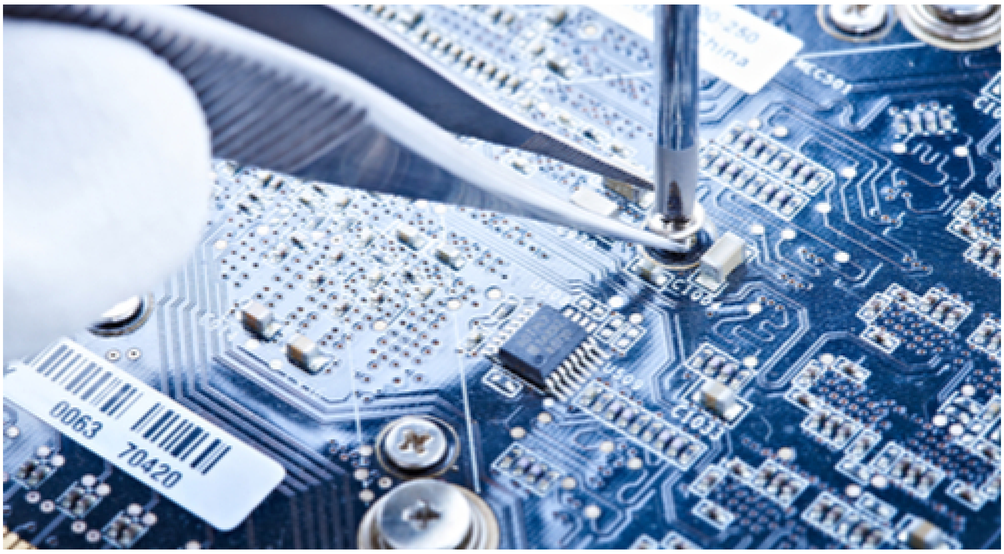
For all the good a ground plane can do, it can also act as a direct path to your sensitive components if an ESD pulse discharges directly into it. To prevent this sort of damage, remember to use TVS circuits between the power and ground on sensitive components to divert the induced currents. When implemented correctly, the voltage differential experienced by components will be held to the clamping voltage of the TVS.

You can also use high-frequency bypass capacitors between power and ground on sensitive components. The capacitors will reduce the charge injection and voltage differences between power and ground. Keep those capacitors and your TVS close to the components you are worried about protecting.

Additionally, you should use a copper land when you are attaching connectors to your PCB. Make sure the land is separate from the PCB ground, or you just introduced a nice, low resistance path for ESD to reach all your components, even though you put all that other protection in place. And, in general, you should minimize path lengths whenever you can.

USE A CHASSIS GROUND

Similar to grounding yourself and a computer tower case before you start pulling things out, you can ground the external casing of your product. By allowing your board and chassis to share a ground, you can improve the grounding of the entire system. One of the easiest ways to implement a chassis grounding is to include a "chassis screw" that connects the ground plane to the chassis. However, you need to make sure you're using adequate standoffs so other components aren't crushed or shorted to the casing when the PCB is screwed in.

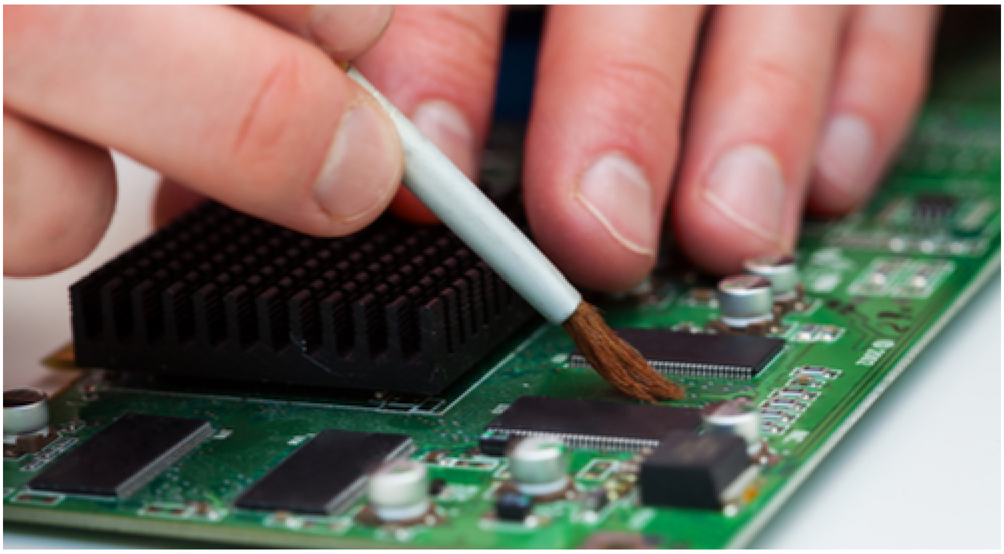


Using a chassis screw can help you ground your PCB to your external casing, and then to earth ground.

Additionally, grounding with a chassis screw makes ESD protection circuits more effective when you're using transient voltage suppression at inputs. Remember, you'll want to separate the chassis ground from the digital and analog ground by using inductive components. That way, a discharge into ground won't accidentally be shared with all your other components.

If you are designing high-speed circuits, you know that they are always more difficult to optimize for performance. This is especially true if you're routing across multiple ground planes, like a chassis and a PCB layer. The best case scenario is that you can connect the chassis ground directly to an earth ground. If that's not an option, you should keep all of the ground planes tightly coupled to each other. That will help to minimize any "ground shifts" around key components.

When you do use an earth ground to your chassis, Texas Instruments recommends that you should keep it "immediately adjacent" to both the ground on your TVS and to the ground of the expected ESD source, usually a connector shield on the input.



Well meaning fingers can do a lot of unintentional damage when installing, cleaning, or inspecting a PCB.

Incorporating good grounding into your design can save you from a lot of damage caused by awkward or inquisitive fingers; however well-intentioned. And while it adds an additional layer of complexity to plan for your casing and stand-offs, it can save you a lot of time respinning your design. One PCB tool that can help manage the added complexity is Altium's 3D clearance checking. By letting Altium manage the physical spacing of your board and case, you can focus on the rest of your design. You can get started with Altium now, and get it right the first time!

HOW COMPONENT PLACEMENT AND ROUTING HELPS PROTECT YOUR PCB FROM ESD

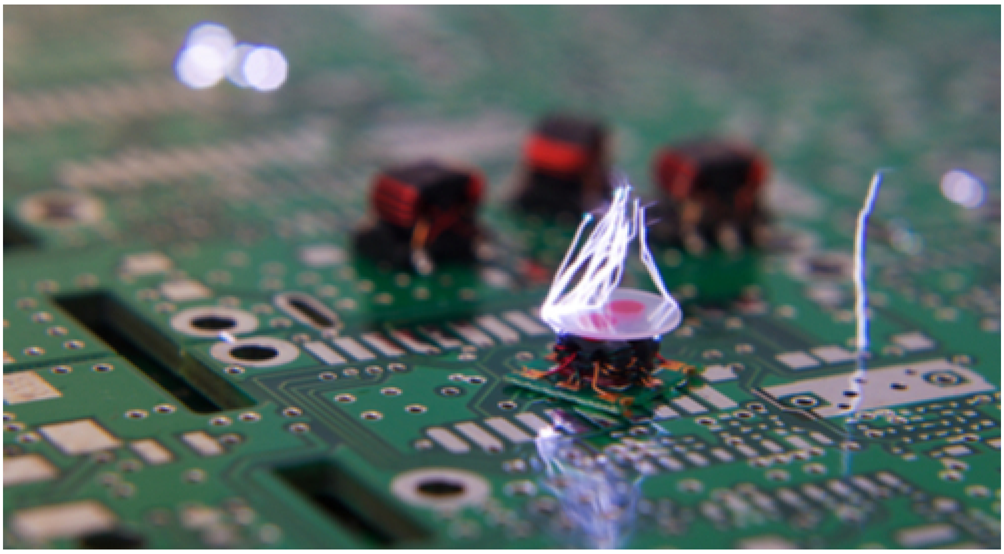
I organize certain things with a reportedly alarming neuroticism. In grad school, there was a line of demarcation between the edge of my side of the desk and the pile of sample containers and papers that began on my neighbor's side. This same tendency, though annoying to deskmates and boyfriends, especially since it doesn't extend to bathrooms, makes me an ace at optimizing component placement on PCBs. Not only does this keep things neat, it also improves the electrostatic discharge protection for the entire board.

The obvious implication of good component placement is that it affects routing on the board. That means your routing determines how any ESD effects will be spread across PCB and into your sensitive or unprotected components. As you arrange your components, there are several basic guidelines to help you improve your routing to best protect your PCB and sensitive ICs.

PLACE COMPONENTS IN THE SAFEST PLACE POSSIBLE

Sometimes, design requirements keep you from using protection circuits for all of your sensitive components. When that's the case, there are steps you can take to improve the odds for those ICs.

Keep unprotected circuits away from the traces between a TVS protection circuit and a connector input, or any other location where ESD is anticipated. This way, you minimize the risk of exposing the component to any current induced by rapidly changing EM fields resulting from an ESD pulse.



Component placement can help protect your ICs from ESD, even if you can't put them on a protected line.

Even devices that are on a protected can benefit from some forethought regarding their placement. Sensitive components that are on a protected line should be placed closer to the center of the board. That helps to balance the parasitic inductance for the best performance of the protection circuit.

MINIMIZE THE LENGTH OF YOUR LINES

Long traces and wires act like little antennas. They can both transmit and receive unintentional emissions. If you do have an ESD pulse, these guys can receive "output" from the spike, and pass it along their entire length.

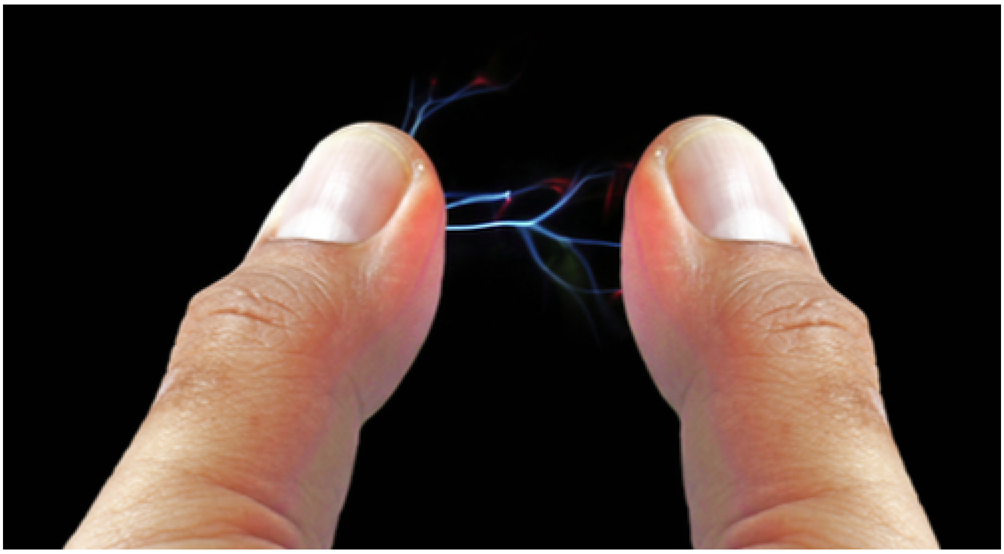
One of the easiest first steps for minimizing line length is to place all the components with lots of interconnects close to each other. This helps you to minimize the length, and hopefully the number, of interconnecting lines. And good organizers know you should keep similar stuff together.

I know I've said it before, but minimize circuit loops. Circuit loops over large areas increase the amount of your board that's exposed to EMI generated by an ESD pulse. They can undo all of the other protection you've implemented, and are really bad news. And while you're at it, use a properly designed ground plane.

MIND YOUR EDGES

Maybe you're a rebel and put books on different subjects next to each other on the shelf. Be warned, though, your rampant disregard for order can have serious repercussions for your PCB if you let different signal traces simply go where the autorouter sends them.

Don't run sensitive tracks along the edge of the board. This is particularly true for supply tracks. You want to minimize radiation from these tracks, as well as their exposure to crosstalk from other tracks, whether it's from a noisy signal, or ESD-induced interference. Keeping your noisy tracks away from everything sensitive is good design practice, anyway.



You don't want an ESD pulse from handling your board to go straight into your most sensitive components.

PCB layout can be incredibly tedious to get right. Using the right PCB tools, like Altium's unified design environment, can make the difference between a neat, functional board with good protections, and a rat's nest of routing that keeps you up a night. They can help you get started now, so you have more time to get everything else in order!

ADDITIONAL RESOURCES

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

- The Correct PCB Routing and PCB Layout to Help Protect Your Board From ESD
- Protecting your PCB from ESD Using Transient Voltage Suppressors
- How Parasitic Inductance Can Impact Your ESD Protection
- Using Grounding to Protect Your PCB from ESD Damage
- How Component Placement and Routing Helps Protect Your PCB from ESD