

**Altium**<sup>®</sup>

# Guidelines for PCB Design



## GUIDELINES FOR PCB DESIGN

PCB Design is a complex field and every step you take to design and develop a board has its own set of best practices to help you determine the best solutions for your pending puzzle. Read on for more information about guidelines from across the spectrum of PCB design disciplines. We hope these PCB design guidelines will help you in your next project.

Join us as we explore a variety of topics to help you decide if guidelines for PCB design is for you, including:

- [Conducted Emissions Test Equipment and Reduction Guidelines](#)
- [High Speed PCB Design Guidelines: An Overview for Getting Started](#)
- [PCB Design Guidelines for Designing Solar Powered Embedded Systems](#)
- [PCB Design Guidelines for High Frequency Circuits in Car Radar and 5G Applications](#)
- [Stripline vs Microstrip: Understanding Their Differences and Their PCB Routing Guidelines](#)

# CONDUCTED EMISSIONS TEST EQUIPMENT AND REDUCTION GUIDELINES



One of my classes in college was so difficult that the professor would give us the test questions a week in advance. Even knowing exactly what to study for the exam, lots of students still failed. Conducted emissions analysis for electromagnetic compatibility (EMC) is similar. Devices need to be checked to see if they're putting too much noise back into the grid through their power supply. Otherwise, the FCC will think you're disrupting utility power. It's not difficult to pre-test devices for EMI conducted back into the power source through the power supply. However, lots of products still fail when it comes time for the final check. Failure at the last stage costs a lot in both time and money. All that can be avoided by preparing yourself with the right equipment and doing some pre-compliance testing. It's also a good idea to look at your PCB design and power supply to try and root out any conduction problems at the source.

## BENEFITS OF PRE-COMPLIANCE TESTING

We also had several classes back then with examinations that were open book. Lots of students wouldn't study beforehand because they thought if they had their books the test would be easy. They were wrong, and lots of them failed. Lots of people assume that the conducted emissions part of EMC is simple compared to radiated emissions, but they cause just as many failures.

If you fail final conducted emissions testing you'll have to do it again, which can cost thousands of dollars. Failing a college exam is bad, but a blunder in these tests is like failing a whole class. Pre-compliance equipment is expensive, but not as costly as repeat certification testing. Failing an EMC assessment can also delay getting your product to market. If you have major fixes that need to be

# GUIDELINES FOR PCB DESIGN

---

done it could significantly delay your project. Better to catch problems at the beginning when they're relatively easy to fix.

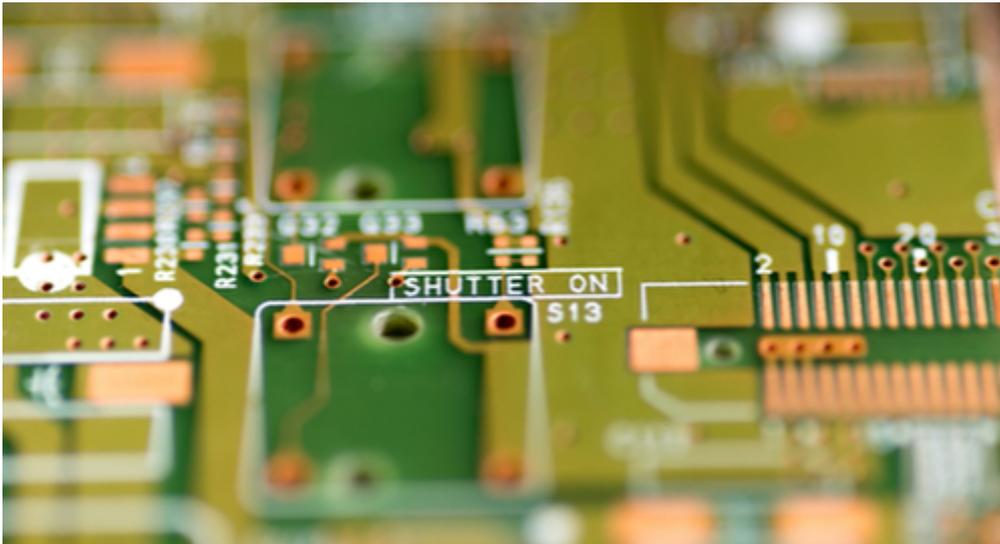


*And you thought college tests were hard.*

## PRE-COMPLIANCE EQUIPMENT

The equipment you'll need for conducted emissions is fairly similar to that required for radiated emissions. A kit like this will cost you several thousand dollars.

- **Spectrum Analyzer (Required)** - A spectrum analyzer is the backbone of pre-compliance testing. This will allow you to analyze any EMI coming out of your board. This will likely be the most expensive piece, costing upwards of \$1,000.
- **Software (Required)** - An open book test isn't helpful if you don't know how to read, just like a spectrum analyzer without software. Some spectrum analyzers come with software included. If yours doesn't, make sure you choose one that is compatible with free programs.
- **Line Impedance Stabilization Network (LISN)** - This is one thing you need for conducted emissions that you don't for radiated emissions. A LISN isolates your device from any noise in the power source and matches impedance so that your spectrum analyzer can operate accurately. If you want your tests to be really accurate, you might need two of these.
- **Pre-Amplifier (Optional)** - A pre-amp will boost any signals coming out of your device, making them easier to read. It's not necessarily required, but might be a good idea.



*Careful routing and grounding will help you avoid conducted EMI.*

## POSSIBLE PROBLEMS

Now that you have all your equipment it's time to test your product. If everything turns out to be fine, you might resent me for getting you to do pre-compliance, but I'm okay with that. Alternatively, if there's a problem, it's likely that the source of it is in your power supply or PCB. Things like **crosstalk** and **ground loops** can both amplify noise in your circuit, which can then be conducted out through your power supply.

While your chosen power supply may supposedly already be compliant, the **reality can be different**. A specific supply may have come from a bad batch. Power supplies are also sometimes only tested with DC current, and AC noise coming from your circuit can force them out of compliance. Regardless of the reason, if your supply is non-compliant, your device probably will be as well. Pre-compliance testing can help you determine if you chose the right supply, or if you need to make a change before finalizing your designs.

Crosstalk happens when signals on your board go where they're not supposed to. There are many causes of crosstalk including: **mixed AC/DC signals**, **poor differential pair routing**, and **poor high speed routing**. Generally you'll want to keep interfering signals far away from other circuits. You should also keep traces short to keep them from radiating EMI like antennas.

Ground and power planes can also be sources of trouble for your PCB. Generally ground and power planes help attenuate EMI, but if you create ground or power loops in them you'll have a problem on your hands. To avoid doing that you should **carefully design your ground planes**.

You should always try to pass EMC certification tests the first time. Repeat assessments will cost you time and money and are easily avoided. Performing tests yourself and catching problems early is the way to go. Even better, you can further prevent issues by designing a PCB for EMI reduction. Try to reduce crosstalk and eliminate ground loops so that your circuit passes the initial test.

## GUIDELINES FOR PCB DESIGN

---

The theory of EMC PCB design is all well and good, but the practice can be another story. Great PCB design software like [CircuitStudio](#) can help you put into practice all these principles. It has lots of [advanced tools](#) that will make your job a breeze.

Have more questions about conducted emission? Call an [expert at Altium](#).

# HIGH SPEED PCB DESIGN GUIDELINES: AN OVERVIEW FOR GETTING STARTED



No matter where you are in your career as a PCB designer, if you haven't had an opportunity to work with high speed designs yet then you may be in for a surprise. There is a lot to consider when you start working in high speed. Most of the techniques you will need to learn are an expansion of the basic circuit board design rules and methods that you are already familiar with. However, you will also find that there are different ways to approach basic design techniques, for example routing, that you may have never considered before.

Here we'll explore an overview of the concepts that you will need to consider when you are laying out high speed circuits. Future articles will hone in these areas to give you a more in depth understanding of each topic. There is a great [need for skilled high speed designers](#) to work on the next generations of PCB designs. Applications in the growing fields of communications, aerospace, and IoT are just a few of the product areas that are requiring high speed circuits. Our hope is that this series of articles will help you in becoming that designer.



*High speed designs will include many new design techniques including measured trace lengths*

## WHEN IS A PCB DESIGN CONSIDERED TO BE HIGH SPEED?

Before you start designing high speed circuits, let's define what is considered to be "high speed". When a printed circuit board is intended to operate at frequencies high enough to significantly degrade circuit performance, that design is considered to be high speed. There are many other factors that are also considered when determining what constitutes high speed such as the circuit, the materials, the environment, the size of the board and the length of the traces. In general though, 50MHz and above is typically the point where a PCB is considered to be a high speed design.

## VARIABLES TO CONSIDER FOR HIGH SPEED DESIGNS:

High speed designs impose a lot of restrictions on you as the designer. This is because you will need to meet the different signal speeds and other design specific requirements. Here are some of the areas that you will need to consider in order to successfully complete a high speed design:

Schematic considerations: Depending on whether you're an electrical engineer (EE) or a board designer, you might treat the schematic differently. Schematics are typically seen as a way of communicating connectivity to board. However, the schematic can be a great help with organizing and presenting your high speed design.

Board materials and stack-up requirements for high speed: What materials your board is built from and how the layer stack-up is structured will make a difference in your high speed design. We will discuss some of these materials and stack-up tactics.

High speed placement strategies: There are ways that your component placement can be optimized for high speed design and there are also component footprint enhancements for high speed. This is because altered component clearances and pad sizes can help minimize high speed connection lengths.

Understanding stripline and microstrip: High speed designs will often require a different approach to routing traces. We will look at

## GUIDELINES FOR PCB DESIGN

---

how having a better understanding of stripline and microstrip routing techniques will help you with high speed routing.

Crosstalk, impedance control, and parallelism considerations: In a high speed design there are a lot of different forces at work that can adversely affect your design. It is important to look at ways to minimize their influence on your design.

Routing topologies and best routing practices: High speed routing often requires specific shapes or “topologies” be used in order to achieve the desired circuit path. It is beneficial to explore different ways to route in via escapes, trace lengths, return paths, etc. in order to accomplish this.

Differential pair and trace length routing: In a high speed design, the routing of differential pairs is critical so that paired signals arrive at their destinations at the same time. Single line signals also may have length requirements, and future articles will look at tips for routing them as well.

Trace length tuning: Once those measured trace lengths have been routed in, they often have to be matched in length to other signal traces. It’s important to look into tuning those trace lengths to get the desired lengths within a group of signals.

Simulators: High speed designs benefit greatly from simulation both before the layout starts, during and afterward. Familiarize yourself with your circuit design software to learn the tips and tricks used to simulate your designs.



*High speed: a whole new world of PCB design awaits you*

## PUTTING THEORY INTO PRACTICE

As you can see, there’s a lot to consider when working on a high speed design. Fortunately, there are many functions within your [CAD software for PCB design](#) that can help you. There are impedance calculators, diff pair routers, trace length reporting options, as well as a host of others tools that are just a menu click away.

## GUIDELINES FOR PCB DESIGN

---

We've only scratched the surface of high speed PCB design techniques here. There's so much more that we'll be covering in future articles. As they say in this high-speed world of PCB design; "stay tuned".

Would you like to find out more about high speed PCB design considerations? Talk to an [expert at Altium](#).

# PCB DESIGN GUIDELINES FOR DESIGNING SOLAR POWERED EMBEDDED SYSTEMS



Have you ever gone on a vacation and felt like you needed another one immediately after? I sure do, my last beach holiday was completely ruined by constant thunderstorms. Unpredictable weather always is always a dilemma when I'm planning ahead for my next vacation, especially if it involves outdoor activities.

I take the same cautious approach when I'm designing solar powered embedded systems intended for outdoor applications. It is a totally different beast than embedded systems that run off of a regulated power supply. As usual, I learned to be cautious the hard way, since my first solar powered prototype didn't even last a day in the rain.

There are many aspects to consider and plan to ensure that your solar powered embedded system continues to work for days without sunlight.

## VARIABLES TO CONSIDER WHEN DESIGNING SOLAR POWER EMBEDDED SYSTEMS

### 1. Solar Panels

It goes without saying that the solar panel is the most critical part of a solar powered system. Monocrystalline is the preferred choice

# GUIDELINES FOR PCB DESIGN

---

of solar panels since it is more efficient than polycrystalline or thin-film, and it performs well in hot weather. There are panels that can convert up to 22% of sunlight to electricity. That being said, the efficiency of monocrystalline and polycrystalline may differ depending on their supplier, so it is good to confirm these details in advance.

## 2. Battery Capacity

When it comes to a solar powered embedded system, an important parameter is the duration of the system when the solar panel is reduced to 0% efficiency. Environmental factors can result in your solar panel not receiving any sunlight for days or weeks. You'll need a battery that has an adequate capacity, and you'll also need to ensure that the solar panel's charging rate is higher than the usage rate of the battery. It's not very efficient if it takes 5 hours to charge the battery and only 2 hours for it to be drained by the system.

## 3. Sunlight Exposure

In a way, solar technology is pretty straightforward. Without sunlight, no electricity is generated. However, having 8 hours of daylight does not necessarily mean that your solar panel is producing electricity efficiently for 8 hours. There is another term called "peak sun hours" where the sun is at its highest in the sky and when your solar panel is at its most efficient. It is good to be aware of this variable and to calculate what your peak sun hours are.

## 4. Blockage and Dust

There was one time when one of our solar powered open space parking machines kept running out of power. After hours of checking out every single piece of hardware, we realized that the machine was installed under a tree and the shadows overlapped a part of the solar panel. The efficiency of a solar panel can be drastically reduced if a small part of it is blocked by dust, shadows or a fallen leaf. That is why it is good to plan your design specifically for the location it will be used.



*A single leaf can reduce a solar panel's efficiency close to zero percent.*

# GUIDELINES FOR PCB DESIGN

---

## 5. Power Intensive Module

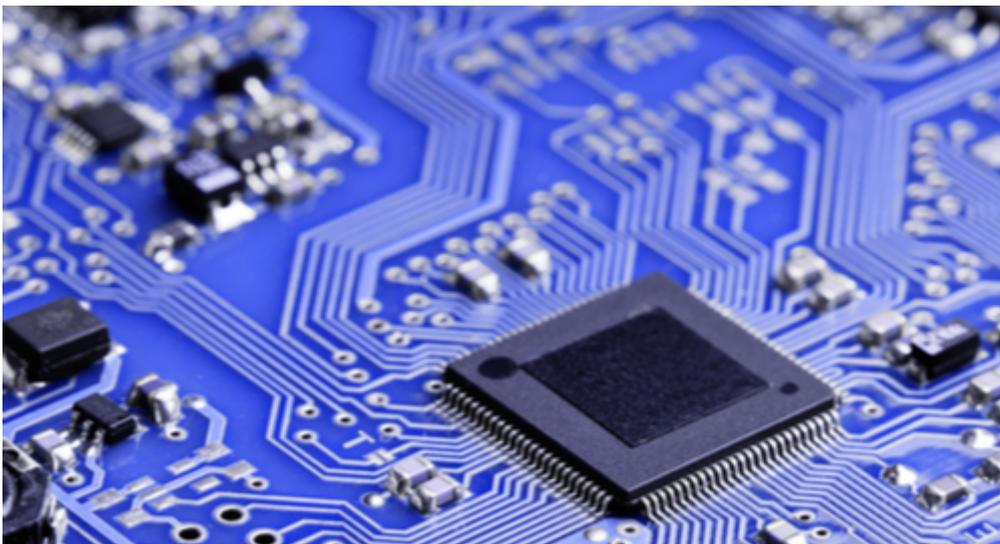
Power intensive modules will put a faster drain on your battery. Yet certain applications require power intensive modules like thermal printers, WiFi or a GSM module. If this is the case, it is necessary to understand and predict the power usage of the module so that you can budget the capacity of the solar panel and battery. For example, a system may only need to activate the GSM module twice a day to transmit the information to the data center. Proper calculation of the estimated size of data and the speed of transmission will provide a figure of how much power will be consumed during the transmission.

## 6. Firmware Architecture

While firmware programmers have the luxury of pushing microcontrollers to the limit in non-solar powered applications, solar power makes this a more delicate process. Take the time to get the right firmware structure. It can result in your solar powered embedded system lasting for weeks instead of days in cloudy weather. The best approach in developing firmware for a solar powered system is to put the microcontroller in deep sleep mode whenever it is not in use. The microcontroller will only wake up from its deep sleep mode by selected interrupts or scheduled timers.

## 7. Power Efficient Hardware Design

It is essential to minimize idle current when you're designing hardware for a solar powered embedded system. Saving 1mA might be insignificant in a non-solar powered system, but in a solar powered system, it can extend the operational time on a cloudy day. A good strategy is to provide a separate power channel to logics and peripheral ICs that are controlled by the microcontroller. This eliminates unnecessary power consumption when the system is not in use, regardless of the microcontrollers operational mode.



*The best solar powered embedded systems consume minimal power when they're idle.*

Why you'll need to Analyze your power delivery network

When you're trapped in the desert, you'll realize how precious water is, especially when you're almost at your last drop. The same

## GUIDELINES FOR PCB DESIGN

---

principle applies to power efficiency in solar powered embedded systems. A power delivery network (PDN) analysis allows you to evaluate if the copper traces on the PCB are sufficient for delivering power efficiently to the load. You'll want to avoid narrow spaces of copper or vias that are too small between copper planes. This will result in resistive losses and generates unnecessary heat. You can prevent this potential waste of power in the design phase, so it's worth taking advantage of this feature if your software provides it.

If you're designing a solar powered embedded system, built in tools like Altium's [PDN Analyzer](#) will help ensure that your design does not exceed its power budget before it is manufactured.

Need more help designing a solar powered embedded system? Contact an expert at [Altium](#).

# PCB DESIGN GUIDELINES FOR HIGH-FREQUENCY CIRCUITS IN CAR RADAR & 5G APPLICATIONS



One of the latest evolutions in next generation PCB design is being driven by two emerging technologies, 5G networks and advanced driver assistance system (ADAS) enabled cars. Both of these technologies use something long feared by PCB designers, the extremely high-frequency (EHF) band. Read on and prepare your boards for a high-frequency future.

This morning I saw a very strange sight while walking down my street. A long, tangled rope of magnetic VHS tape was twisting down the road, carried by the wind. It took me back to a simpler time of Blockbusters and rewind machines. If you thought those rewind machines were fast, today's electronic breakthroughs will make your head spin. One of the latest evolutions in PCB design is being driven by two emerging technologies, 5G networks and advanced driver assistance system (ADAS) enabled cars. Both of these technologies use something long feared by PCB designers, the extremely high-frequency (EHF) band. You'd best prepare your boards for a high-frequency future before they go the way of BetaMax and boomboxes.



*Thank goodness we've gotten rid of this stuff.*

## WHY ARE WE USING MILLIMETER WAVES?

Weren't RF and Microwave frequencies good enough, why are we moving up into the EHF Spectrum? There are two advancements forcing us into higher frequencies, 5G and ADAS radar.

- **5G** - Telecommunication companies are looking to take us from today's 4G/LTE speeds and latency to the faster, brighter tomorrow of 5G. Current cellular networks can give us download speeds of somewhere on the order of 10's of megabits per second and latencies around 70 ms. 5G will be a big jump, up to 10 Gbps download with latencies under 10 ms. All of this is possible because 5G will operate in the EHF spectrum. Wider frequency bandwidths give us lower latencies, and faster frequencies give us better data rates. The industry expects 5G to start being implemented somewhere around 2018. At that time, you'll need to be ready to deal with millimeter (mm) wavelength signals.
- **ADAS Radar** - One technology that's already here is radar for ADAS enabled vehicles. Collision detection radars used to operate below 30 GHz, but recently the standard has moved up to 77 GHz. As manufacturers build more cars with ADAS capabilities, we can expect to see more radar systems driving around our streets. If you want to design PCBs that deal with any kind of car radar, you should be ready to work with EHF signals.

# GUIDELINES FOR PCB DESIGN

---

As both of these things grow, you'll need to know more and more about how to deal with their operating frequencies. This is the part where I give you some material and design guidelines to help you cope with the rapidly changing PCB design ecosystem.

## MATERIAL GUIDELINES

I actually just wrote about how to **choose which material** to use for your high-frequency boards. However, the frequencies we're talking about are a bit higher than normal, so I'll reiterate a few points.

- **Very Low Dielectric Constant (Dk)** - We engineers often find something that works and then stick with it. Maybe you moved your high-frequency boards one level up from FR4 and figure that will be ok for EHF. For mm waves you need to use materials with the absolute **lowest Dks possible**. Dk losses increase proportionally with frequency. That means a moderately low Dk is no longer acceptable.
- **Very Little Soldermask** - You may ask your supplier about the moisture absorption of your substrate, but I doubt you ask about the soldermask. Most soldermasks have high absorption, allowing them to gorge on water, which has a Dk of 70. A damp soldermask will **introduce high losses** into your mm wave circuit. You should use as little soldermask as possible on these PCBs.
- **Very Smooth Copper** - Your copper needs to be as smooth as possible on these boards. The skin depth for current at these frequencies is **very shallow**. So shallow, that it's sometimes on the order of the mountains and valleys that make up a rough surface. Rough copper will give your current a longer path, increasing resistive losses. Use smooth copper.



*ADAS car radar will require us to learn new design techniques.*

## PHYSICAL GUIDELINES

Along with material considerations, you need to think about geometry and other physical specifications. Two important things to think about are laminate thickness and transmission line characteristics.

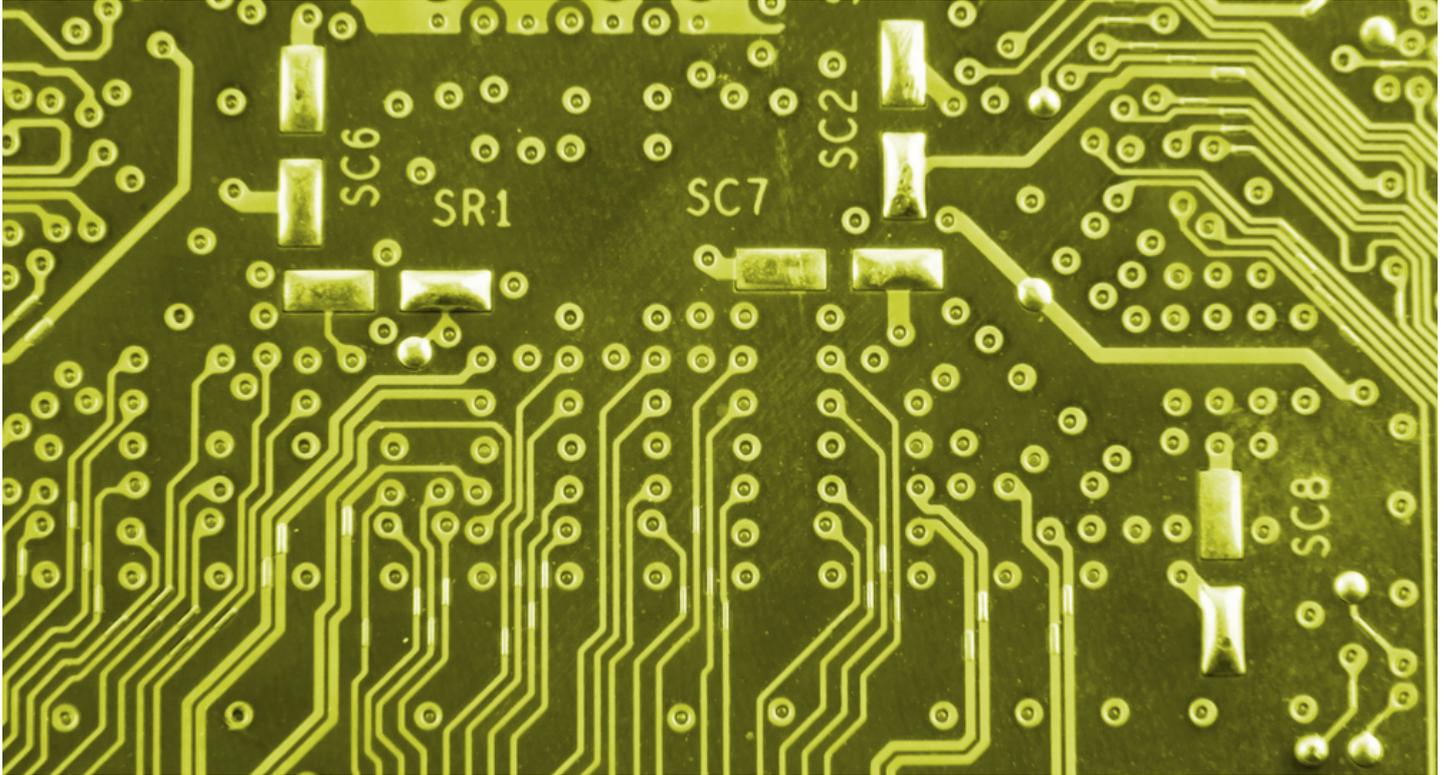
- **Laminate Thickness** - You not only need to select the **right kind of laminate**, you need to select the right laminate thickness. Generally you want your laminate thickness to be somewhere between  $\frac{1}{4}$  and  $\frac{1}{8}$  **wavelength** of the highest operating frequency. If your laminate is too thick, it can resonate and even propagate its own waves. Remember that laminate thickness can affect your conductor widths, so factor that into your decision.
- **Transmission Line Characteristics** - In regard to transmission lines, you'll need to decide which **type of conductor** you want: microstrip, stripline, or grounded coplanar waveguide (GCPW). Microstrip is probably the most familiar but has problems with **radiated losses and spurious mode propagation** above 30 GHz. GCPW is a good choice but will suffer **more conductor losses** than microstrip or stripline. Stripline is great but can be difficult to manufacture, increasing costs. In addition, you'll need to use microvias to connect the stripline to outer layers with **minimal signal reflections**. Microvias are difficult to fabricate, so if you choose this option work with your manufacturer to **reduce potential defects**.

All of us technology fanboys like to argue. First, it was Betamax vs VHS, then Blu-ray vs HD DVD, Firewire vs USB, etc. Unfortunately for us, we won't be able to argue against high frequencies in our next gen PCBs. Rising data rates and changing technologies like 5G and ADAS enabled cars are raising frequencies as well. Now we just have to learn how to deal with it. You'll need to be careful about which PCB materials you use for these new high-frequency designs. You'll have to keep an eye on the physical characteristics of your circuits as well.

The future can sometimes be a bit ominous, but luckily you don't have to face it alone. Great **PCB design software**, like **Altium Designer**, can help you master the techniques of tomorrow. **Altium Designer** has a **great range of advanced tools** to make design easy for engineers like you.

Have questions about EHF circuit design? Call an **expert at Altium**.

# STRIPLINE VS MICROSTRIP: UNDERSTANDING THEIR DIFFERENCES AND THEIR PCB ROUTING GUIDELINES



The first time I heard a presentation on high speed design techniques, it went straight over my head. Since this was at the beginning of my career as a designer, I'm sure that it was my inexperience that caused the confusion. The whole concept of stripline and microstrip routing didn't make any sense to me and I thought that the instructor was talking about a completely different type of PCB that I wasn't familiar with. Fortunately, my confusion was quickly straightened out when I learned that it isn't the PCB itself that is considered to be stripline or microstrip. Instead, stripline and microstrip are different methods of routing high speed transmission lines on a PCB.

Understanding stripline and microstrip can be difficult. So whether you are new to PCB design or if you are looking for a refresher on the topic, this basic review is for you.



*Understanding Stripline and Microstrip*

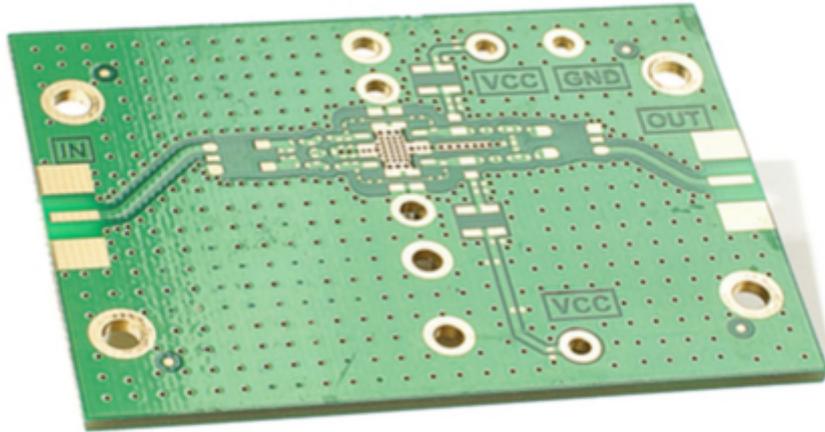
## STRIPLINE AND MICROSTRIP, WHAT ARE THEY?

Stripline and microstrip are methods of routing high speed transmission lines on your PCB. Stripline is a transmission line trace surrounded by dielectric material suspended between two ground planes on internal layers of a PCB. Microstrip routing is a transmission line trace routed on an external layer of the board. Because of this, it is separated from a single ground plane by a dielectric material.

With the transmission line on the surface layer of the board, microstrip routing has better signal characteristics than stripline. Board fabrication is also less expensive with microstrip since the layer structure of one plane and one signal layer makes the manufacturing process simpler. Stripline can be more complex to manufacture because it requires multiple layers to support the embedded trace between two ground planes. However, the width of a controlled impedance trace in stripline is less than an impedance trace in microstrip of the same value. This is because of the second ground plane. These smaller trace widths enable greater circuit densities, which in turn enables a more compact design. The internal layer routing of stripline also reduces EMI and provides better hazard protection.

Stripline and microstrip have different benefits. The decision of which is the better method should be based on the needs of the design. In a dense high speed design, a mixture of both methods on a multi-layer board is often used to accomplish the design goals.

Furthermore, it is extremely important to maintain controlled impedance across the design when routing transmission lines on a high speed design. The layer of the PCB that the transmission line is routed on, the physical characteristics of the transmission line trace, and the characteristics of the dielectric all need to be calculated together in order to give the correct impedance values for the circuit. There are many different impedance calculators with different stripline and microstrip models that are available for making these calculations.



*Stripline and microstrip routing are important to consider for your PCB design*

## EXAMPLES OF STRIPLINE AND MICROSTRIP ROUTING

The following are some examples of stripline and microstrip routing techniques and how some of their characteristics affect their impedance calculations:

1. Microstrip. Transmission lines that are routed on the external layers are considered to be microstrip. The model for these is based on the trace thickness and width, and the substrate height and dielectric type.
2. Edge-Coupled Microstrip. This technique is used for routing differential pairs. It is the same structure as regular microstrip routing, but the model is more complex with the addition of the trace spacing for the differential pair.
3. Embedded Microstrip. This structure is similar to regular microstrip except that there is another layer of dielectric above the transmission line. Soldermask can be considered as a layer of dielectric and must be accounted for in the impedance calculation.
4. Symmetric Stripline. Transmission lines that are routed on internal layers (between two ground planes) are considered to be symmetric stripline, or just plain “stripline” routing. Like microstrip, their model is based on the trace thickness and width, and the substrate height and the dielectric type with the calculation adjusted for the trace being embedded between the two planes.
5. Asymmetric Stripline. Although similar in structure to the symmetric stripline model, this model accounts for the transmission line trace that is not balanced precisely between the two planes.

# GUIDELINES FOR PCB DESIGN

---

6. Edge-Coupled Stripline. This technique is used for routing internal layer differential pairs. It is the same structure as regular stripline, but the model is more complex with the addition of the trace spacing for the differential pair.
7. Broadside-Coupled Stripline. This technique is also used for routing internal layer differential pairs, but instead of side by side, the pairs are stacked on top of each other. The model is similar as for edge-coupled stripline.

I hope that this tutorial on stripline and microstrip has been helpful in clearing up some of the confusion surrounding these concepts. Understanding what the different stripline and microstrip methods of routing transmission lines are will ultimately help you to design a better [high speed board](#).

Would you like to find out more about high speed design and how your [CAD layout software](#) can help you to achieve success? [Talk to an expert at Altium](#).

## ADDITIONAL RESOURCES

Thank you for reading our guide on Guidelines for 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:

- [Conducted Emissions Test Equipment and Reduction Guidelines](#)
- [High Speed PCB Design Guidelines: An Overview for Getting Started](#)
- [PCB Design Guidelines for Designing Solar Powered Embedded Systems](#)
- [PCB Design Guidelines for High Frequency Circuits in Car Radar and 5G Applications](#)
- [Stripline vs Microstrip: Understanding Their Differences and Their PCB Routing Guidelines](#)