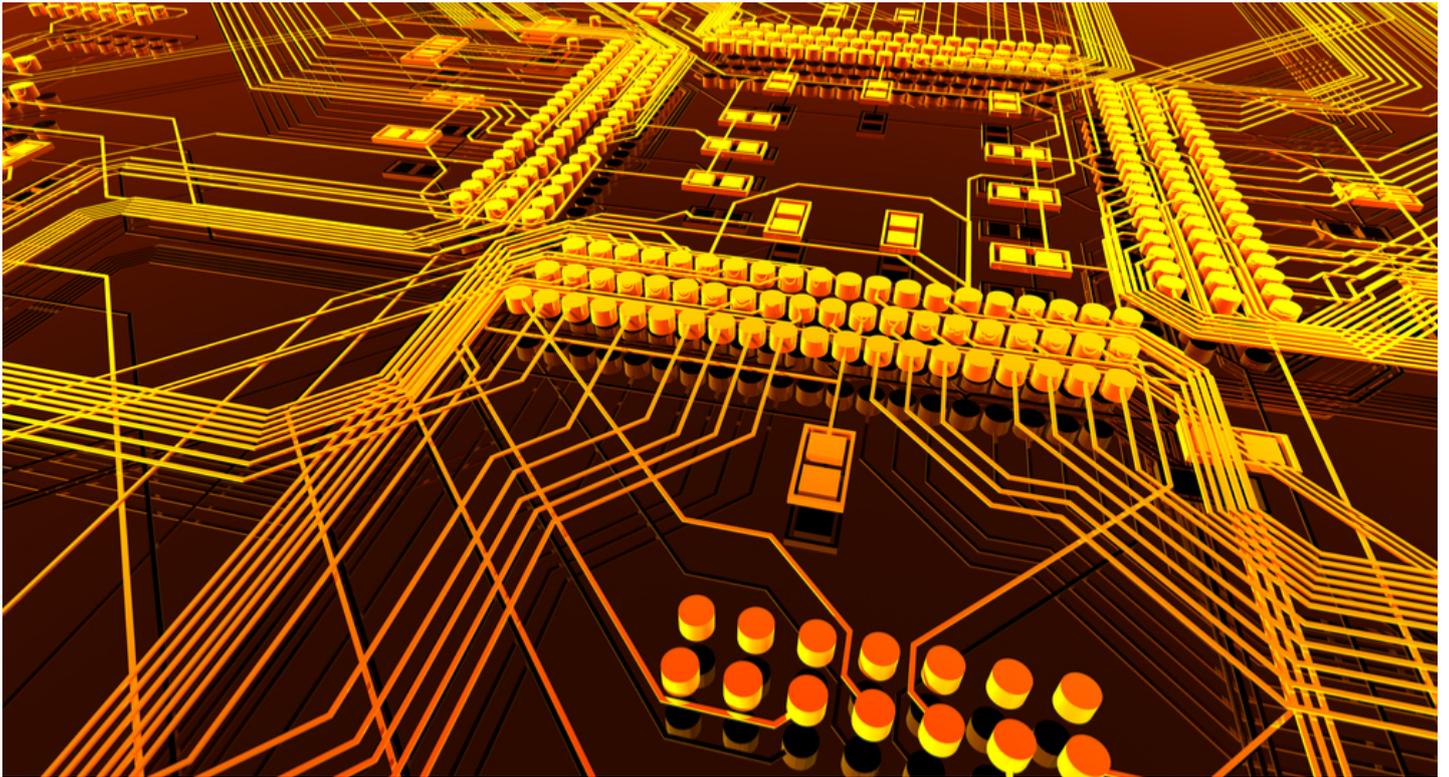


Altium[®]

Advanced PCB Layout



ADVANCED PCB LAYOUT

Simultaneously consider the big picture and the very details that impact your projects' success. That is the point of view that will help you become successful at any project no matter the difficulty. But why stop with just solving the problem. Become an expert in PCB Layout and utilize the Unified Data Model Approach, a smart way to manage design. Benefits include not just time savings, but the potential of simultaneous design that is available with the integrated schematics and PCB layouts offered in some of the most advanced tools, like Altium Designer.

Join us as we discuss a variety of topics and learn more about advanced PCB Layout, including:

- High Speed PCB Layout Guidelines: Placement Tips and Strategies
- How PCB Software Performance Affects the Layout of Power Planes and Ground Planes
- PCB Design Tutorial: Tips for Using Optocouplers in Your PCB Layout

HIGH SPEED PCB LAYOUT GUIDELINES: PLACEMENT TIPS AND STRATEGIES



In real-estate, the buzz word is; “location, location, location”. Interestingly enough, the same can be said of high speed PCB design parts placement. Although all aspects of the [high speed PCB design process](#) are important, the placement is especially so. Placement methods that work without a problem in a standard PCB design may not satisfy the critical signal flow requirements of a high speed design. For the design to work, it really is all about “location, location, location”.

Here are some tips and strategies to consider for high speed designs. First, we will look at some basic placement considerations for high speed designs followed by the benefits of creating a floor-plan of the placement before any actual parts are put on the board. Lastly, but certainly not the least in importance, we will discuss termination resistors and where they should be placed.

HIGH SPEED PCB DESIGN PART PLACEMENT

Let’s start with part placement in general. Standard [PCB design component placement practices](#) will serve you well here, although there are a couple of high speed considerations to keep in mind.

You should avoid placing sensitive high speed devices close to the edge of the board. This is because the edge of the board has different impedance characteristics, and there is a greater chance of electromagnetic interference (EMI). Additionally, cables use

ADVANCED PCB LAYOUT

'connectors' to connect to your board, and these can radiate EMI. As a result, you'll want to place connectors close to the board edges and sensitive high speed devices close to the board center, to reduce EMI on your design.

Thermal effects are another aspect of high speed design placement to consider. This is because high speed devices may run at higher temperatures than standard components. To ensure that your hot components placement stay cool, plan your placement so that these components receive an unrestricted airflow. For example, don't place taller components, like connectors, in the direction of airflow to a hot BGA.



PCB circuit flow

PREPARING FOR PLACEMENT BY FLOORPLANNING YOUR PARTS

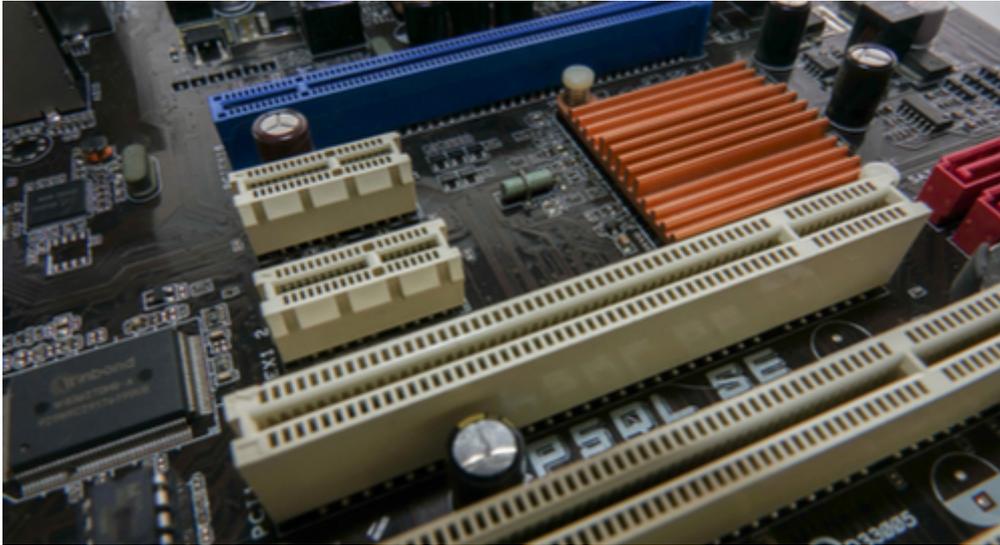
Creating a floor-plan of your parts placement is an effective way to prepare. By planning ahead, you can account for small parts like termination resistors and bypass capacitors. This way they will get placed appropriately instead of being squeezed in at the end of your design. Additionally, a pre-placement floor-plan will also allow you to optimize the placement for signal integrity purposes.

Functional blocks of circuitry such as power conditioning, RF, digital, analog, etc., should be organized and placed as groups in order to minimize signal crossing. A pre-placement floor-plan will allow you to see what the signal flow between functional blocks is and how best to plan for it. For instance, group your power conditioning parts together as much as possible in order that their signals do not have to cross through sensitive areas of RF circuitry. Also, plan your placement to keep connection lengths as short as possible. Additionally, components that are part of high speed signal paths (multiple nets that connect a series of components together), should be placed as close together as possible and not spread apart. You should also consider your routing channels when planning your placement to make sure that you have the necessary space.

When planning out the of placement functional blocks of circuitry, keep in mind the power and ground needs as well. Using continuous power planes is usually preferred, but in the event that the needs of the design necessitate a split power plane for

ADVANCED PCB LAYOUT

multiple voltages, exercise caution placing connected components across the split. High speed transmission lines should not cross splits in power planes as that will break up the **return path** for those signals. Also, avoid placing other components that are not part of a circuit between components of that circuit. This too will affect the return path for that circuit.



PCB parts placement. Editorial credit: I AM CONTRIBUTOR /Shutterstock.com

TERMINATION RESISTORS

The final and most specific placement strategy is considering the placement of your **termination resistors**. These resistors are often treated as an afterthought drop-in once the main portions of the circuitry have already been placed. Since these resistors are part of the circuit as a whole, their placement is extremely important for it to function correctly. That is why they need to be placed along with the rest of the components in the high speed circuitry instead of after. Although there are various schemes for placing termination resistors based on the needs of the circuit, the following two examples are the most widely used.

Simple Parallel Termination: This scheme puts one side of a termination resistor on the end of the circuit closest to the receiver while the other side is tied to power or ground. The greater the trace length from the load pin to the resistor, the more the circuit is prone to signal reflection resulting in signal degradation. This is why you should place the resistor as close as possible to the load pin of the receiver to minimize the stub length of the connecting trace.

Series Termination: With this scheme, the resistor is placed inline immediately after the driver pin of the circuit. Since the resistor is an inline part of the circuit, it doesn't have the same stub length concerns as a parallel termination resistor. However, placing multiple series termination resistors for a large scale device will take up a lot of board space around the device. This will require planning ahead to make sure that adequate space is available without having to rip up and replace parts.

Once you've decided on how to organize and place your termination resistors, you can add it to your placement floorplan. With your circuitry paths as well as your power and ground needs accounted for, you can create a well thought-out floor plan that will guide

ADVANCED PCB LAYOUT

your high speed PCB parts placement.

Floorplanning and the other strategies that we've talked about will give you a great start on your parts placement. Your [high speed design tools](#) can help you with the placement as well as many other aspects of the design. Professional PCB design software, like [Altium Designer](#), comes with the right tools for the job.

Would you like to find out more about the different high speed capabilities that Altium offers its users? [Talk to an expert at Altium.](#)

HOW PCB SOFTWARE PERFORMANCE AFFECTS THE LAYOUT OF POWER PLANES AND GROUND PLANES



The detective on your favorite police television show will often be looking for the plain facts of the case. In PCB design tools, we have some “plane” facts to look for. One of those would be getting a better understanding of what it takes to create the power and ground planes.

As designers, we know that planes provide power and ground to the components on our circuit boards. We also know how to manipulate our tools in order to create those planes. But do we know why certain terminology is used, or why our tools create the planes the way they do? You may have heard the term “D-code” or even used it yourself, but do you know where it came from and what it is for?

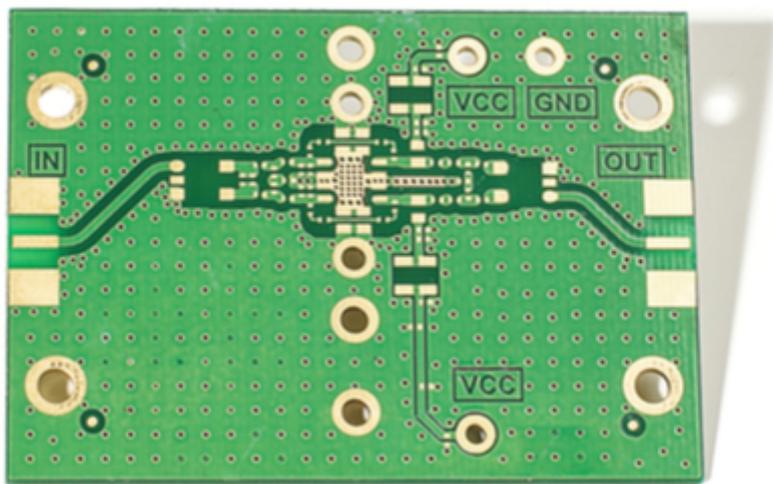
Hang on, we’re about to take a trip down memory lane that will give you some background history on PCB planes and how software performance has affected their creation. Our goal is to expose some of the plain facts about planes so that you can plainly see how all of these planes work together today. Are you with me? Good, then I’ve got some ‘splaining to do.

A BRIEF HISTORY OF THE PLANE

There are two ways that we create power and ground planes in PCB design tools, as a positive image and as a negative image. There are lots of names associated with planes such as area fills, copper pours, and floods. For clarity though, let's stick with positive and negative planes so we are all on the same page.

Positive Planes: A positive plane is usually created by designating a polygon shape on the board, and the layout tool will fill it in giving you a solid plane image. There is a lot going on under-the-hood of the layout tools in order to do this. Object clearances must be calculated, pads must be connected to the plane, and proper net connectivity must be maintained to prevent shorts to other net objects. As a result, creating a positive plane is one of the more complex functions performed by your layout tools.

Negative Planes: Because of these complexities, the first generation of CAD layout tools had difficulties creating positive planes. It was too much data for them to process. To overcome this, the negative plane was invented. A negative plane is a reverse image of a positive plane. Polygons are not filled to create metal, instead pads and other shapes are used to create clearances in the metal.



A positive plane on the exterior of a board

POSITIVE PLANES REQUIRE HIGHER PERFORMANCE

The computers using those early CAD tools were nothing compared to what we have available today. At one point I was using Recal-Redac software on a PDP-11 with a 16-bit processor. Each day I would have to install my personal disk platter in the disk drive, which was about the size of a dishwasher, to access my 10 megabytes of design storage.

Those systems simply didn't have the performance, the memory, or the disk space to handle positive planes like we can today. Even when they had the capability for positive planes, you ran the chance of crashing the system if your plane was too large. Because of that, designers depended on negative planes. These negative planes often wouldn't have actual contours or shapes. As well, connections to the planes would be represented by a simple "X". The designer would then be responsible for assigning these shapes

and characters to the correct photo plotter aperture position.

EARLY PHOTO PLOTTERS

Another deterrent to creating positive planes was the photoplotter itself. Early photo plotters used a bright lamp that was focused through a physical aperture in order to create an image on the film. This film was then used by the manufacturer to fabricate the board. Since these were **vector plotters** instead of raster plotters, it could take a lot of time to paint the images on the film, especially if you were trying plot a positive plane.

Another challenge with positive planes was giving the photoplotter the correct commands to work with. Early photo plotters used the original Gerber format for their input, and the apertures were assigned to “D-codes”. This was before the aperture information was **embedded in the Gerber file**, and as a designer you were responsible for creating an accurate aperture list. You had to know the proper D-code assignments, and then assign those codes to specific functions such as filling in positive planes. I have seen a two-hour photo plot turn into an overnight ordeal because a 10 mil D-code was specified as the aperture instead of a 100 mil D-code.



Today's high performance PCB software does the hard work of creating planes for us

PLANES IN HIGH PERFORMANCE PCB SOFTWARE

Fortunately, in today's PCB design tools we don't have to worry about D-code assignments, or how our design will affect the photoplotter. When we work on a positive or negative plane, all of the work to create the fills and assign the D-codes is handled automatically for us. Today's raster laser plotters are also vastly superior and will plot large positive planes in minutes instead of overnight.

Now you should have a better idea of what a D-code is and why it is important, as well as some historical context for negative planes. There is a lot more that can be said about this subject, but we are plain out of time.

ADVANCED PCB LAYOUT

Altium Designer is leveling up to 64 bit. Its high performance and powerful functionality let you choose whether you want to work with both positive and negative planes. This will help you to create the power connectivity that you need for your design.

Would you like to find out more about how Altium can help you design your power and ground planes? [Talk to an expert at Altium.](#)

PCB DESIGN TUTORIAL: TIPS FOR USING OPTOCOUPPLERS IN YOUR PCB LAYOUT



Are you guilty of hitting the snooze button at least twice before you reluctantly wake up from your blissful sleep? My wife claims that I probably have the world record time for hitting the snooze button since I do it without even opening my eyes. But there are times when the third snooze alarm mysteriously doesn't go off and it is a losing battle to get my day started on time.

In electronics, embedded systems often rely on optocouplers to receive input signals from external sensors or switches. In a way, they are like the microcontrollers' alarm clocks. Ideally, all the signals are relayed accurately to the microcontroller. However, when the optocouplers are not properly implemented, the microcontroller can sometimes miss inputs signals or falsely detect signals when no inputs are triggered. In this PCB design tutorial, we will discuss how to set your optocoupler up for success in your PCB layout. But first, let's remind ourselves how an optocoupler works.

PCB DESIGN TUTORIAL: BASIC PRINCIPLES OF AN OPTOCOUPLER

Optocouplers are electronic components that isolate input signals through an optical interface. The most basic form of optocoupler consists of an infrared LED and a photo-transistor within a single integrated circuit. The infrared LED is turned on when electrical current passes through and the intensity depends on the amplitude of the current. The phototransistor is activated by the LED light, which causes a short connection on between its collector and emitter.

ADVANCED PCB LAYOUT

The infrared LED and the phototransistor are often separated by glass or air. This gives an electrical isolation of <math><10\text{kV}</math> across the optocoupler. As a result, optocouplers are an ideal choice for isolating embedded systems from electrical interference originating from the input signal's environment.

Besides protecting the embedded system from electrical noise, optocouplers are also used to keep low voltage and high voltage systems apart. For example, photo-triac, which is a variation of optocouplers, can be used to control high AC voltages devices. For example, an AC motor. This eliminates the risk of circuitry faults that can cause damage to the microcontroller and its accompanying components.



An optocoupler can reduce the risk of high voltage damage.

PCB DESIGN TUTORIAL: COMMON MISTAKES WITH OPTOCOUPERS

The optocoupler is a simple passive component that most designers encounter. Getting an optocoupler to work is not rocket science; however, there are a couple of design mistakes that defeat the purpose of using one or that causes unstable input signals.

1. Failing To Separate Optocoupler Ground Connections.

In the basic optocoupler, the integrated circuit (IC) consists of two ground pins. One is connected to the infrared LED and the other connected to the photo-transistor. A common mistake is to **connect both grounds** together when routing the PCB. In my engineering experience, I've encountered this even in electronics controller used in machines.

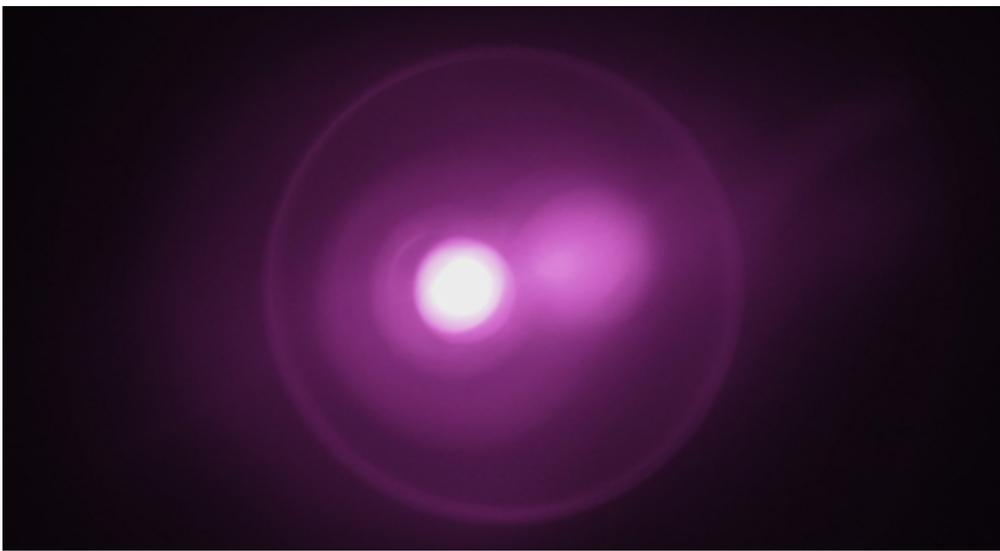
The main reason for using an optocoupler is to safely separate two circuits. When the external ground is connected to the PCB, any ground noises from the circuitry can directly couple to sensitive onboard circuits. Instead, create a separate signal connection for the external ground pins and allocate dedicated connectors for the input ground wires.

2. Using The Wrong Value For The Current Limiting Resistor

ADVANCED PCB LAYOUT

Besides applying the proper voltage, the optocoupler's infrared LED require adequate current to function properly. The value of the minimum forward current can be referred from the [Current Transfer Ratio](#) chart of the respective optocoupler. If the current limiting resistor performs at the optocoupler's minimum value, the phototransistor may behave irregularly. For example, out of 10 valid inputs from the switches, only part of it will be detected.

On the other hand, the value of the limiting resistor should not be too low. This is to prevent the infrared LED from breaking down. Like [regular LEDs](#), the infrared LED has a maximum forward current that should not be overshoot. This makes choosing the right current limiting resistor a crucial step in ensuring a reliable optocoupler operation.



As the name implies, enough current need is needed to light up the infrared LED

3. Choosing The Wrong Optocoupler

As general as it may seem, not all optocouplers are built the same. For example, the opto-triac is used for controlling an AC load and opto-Darlington is ideal for situations where only a small amount of input current is generated. Another consideration is the Collector-Emitter breakdown voltage, which can vary for different models of optocouplers.

But if you're just using optocouplers for normal input isolation, common models like PC817 will do the trick. You'll also spend less time in creating footprints since the generic optocoupler components in the library of your PCB design software will do the trick.

Speaking of [PCB design software](#), Altium's [CircuitStudio](#) and [Designer](#) are a great place to get started!

Have a question about optocouplers? Contact an [expert](#) at Altium.

ADDITIONAL RESOURCES

Thank you for reading our guide on Advanced PCB Layout. To read more Altium resources, visit the Altium resource center [here](#) or join the discussion at the bottom of each original blog post:

- [High Speed PCB Layout Guidelines: Placement Tips and Strategies](#)
- [How PCB Software Performance Affects the Layout of Power Planes and Ground Planes](#)
- [PCB Design Tutorial: Tips for Using Optocouplers in Your PCB Layout](#)