

TABLE OF CONTENTS

- [Circuit Board Design in 10 Steps](#)
- [Step 1: Schematic Capture](#)
- [Step 2: Create a Blank PCB Layout](#)
- [Step 3: Synchronize Schematics to Your PCB Board Design](#)
- [Step 4: Designing Your PCB Stackup](#)

[Expand Table of Contents](#)

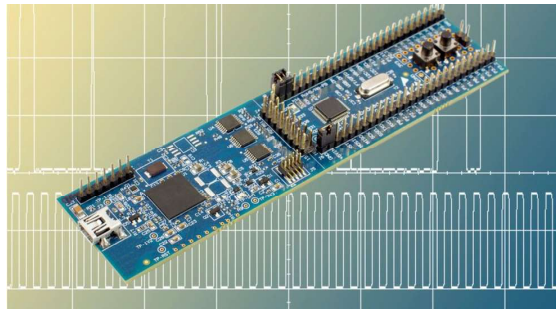
Altium is transforming the electronics industry so thoroughly that our web pages need a minute to catch up. For a short time, some information on this page may be outdated. We appreciate your patience. It will be worth the wait!

[Learn More](#)

How to Design a Printed Circuit Board in 10 Easy Steps



Zachariah Peterson | Created: August 31, 2018 | Updated: November 3, 2025



Printed circuit board (PCB) design and layout is both an art and a science, and it can be difficult to get started designing a new circuit board from scratch. If you're new to electronics and circuit board design, and you're still learning about designing a custom circuit board in [Altium Designer](#), available within [Altium Develop](#), we've compiled the 10 important steps you can use to create modern PCB layouts. In this tutorial, we'll show all the steps involved in creating your circuits and schematics, then we'll transition into creating a circuit board design that can actually be manufactured.

There is a lot that goes into any engineered [circuit board design](#), from a basic printed circuit to a complex non rigid PCB. Any new electronic device will start as a [block diagram](#) and/or a set of [electronics schematics](#). Once you have your schematics finished and validated, you can follow the steps [below](#) to create modern PCB designs and [layouts](#) in [Altium Designer](#). Here's the full list of PCB design steps:

1. [Create the Schematic](#)
2. [Create a Blank PCB Layout](#)
3. [Schematic Capture: Linking to Your PCB](#)
4. [Designing Your PCB Stackup](#)
5. [Defining Design Rules and DFM Requirements](#)
6. [Place Components](#)
7. [Insert Drill Holes](#)
8. [Route Traces](#)
9. [Add Labels and Identifiers](#)
10. [Generate Design Files](#)

Printed Circuit Board Design in 10 Steps

When designing a circuit board it may sometimes seem as though arriving at the final design is going to be a long and arduous journey. Whether it's the basics of micromanaging your copper and solder, or trying to ensure that your circuit board ends up printed after all, or going into more specific design problems such as through-hole technology or layout design with vias, pads, and any number of signal integrity issues, you'll want to make sure you have the right circuit board design software.

If you've been doing this for decades, you don't need me to tell you how valuable knowing your circuit board design software is to getting your printed boards designed right. Laying out traces for routing and copper placement, or managing the layer needed for solder can become difficult without an accurate and reliable integration from schematic capture to layout.

Though a deep program, Altium Designer's user-experience is highly rewarding for new and experienced PCB board designer experiences alike. It offers a design environment built from the ground up for a streamlined custom circuit board design process within a single, unified printed circuit board layout environment. Let's go over the general PCB design steps.

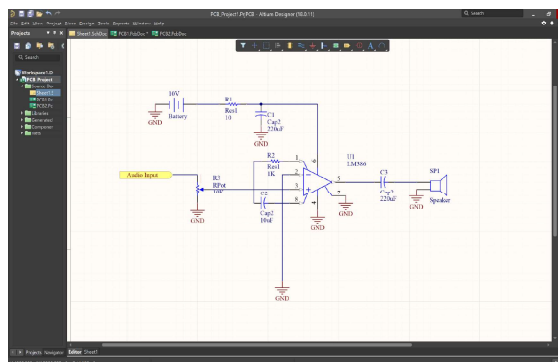
Step 1: Schematic Capture

Whether you are generating your circuit board design from a [template](#) or creating your printed circuit board from scratch, it is probably best to begin with the schematic. Your schematic is similar to the blueprints for your new device, and it is important to understand what is shown in your schematics. First, your schematics show you the following:

- Which components are used in your circuit board design
- How components are connected together
- The relationships between groups of components in different schematics

The last point above is quite important, as complex designs may use [hierarchical schematics](#). You can enforce significant organization in your new board if you take a hierarchical approach to your circuit board design and place different circuit blocks in different schematics.

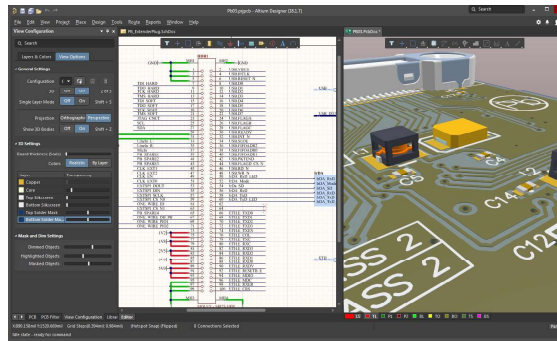
- [Learn more about schematic capture and how to create readable schematics](#)



The schematic editor in Altium Designer

Not only is circuit interconnectivity easier to define and edit, but converting a schematic to a circuit board layout is much easier than designing directly on the board. For components, Altium Designer has an extensive database of parts libraries. Additionally, you can utilize the [Altium 365 platform](#), which provides access to thousands of component libraries and adds flexibility to your project management and product development. However, you can also design your own schematic symbols and [create footprints](#). Or, if you would like to take advantage of pre-existing component footprints, try using the [Manufacturer Part Search Panel](#).

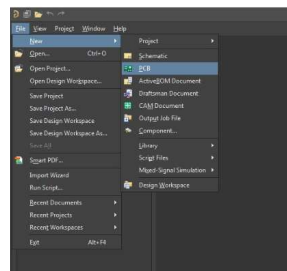
It's also important to annotate your schematics, as well as name nets when schematics get large and complex. As higher pin count devices are added to schematics, you can use net name assignments to keep things organized and easily track connectivity between components. Using net names is also helpful once you're in the PCB layout as you'll be able to quickly identify nets during placement and routing. This will also help once you go to [test your finished prototype](#) as you'll be able to quickly track nets in the corresponding PCB layout in Altium Designer.



Net names can be assigned in the schematics to help you stay organized. Take advantage of the view configuration features to quickly find what you need.

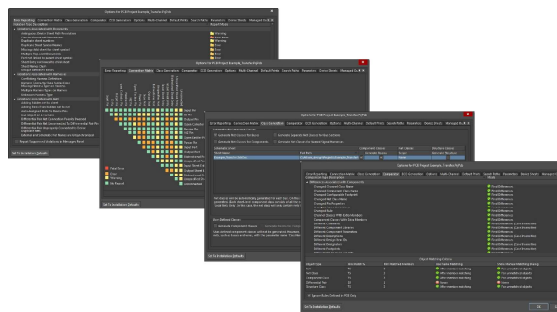
Step 2: Create a Blank PCB Layout

After you've created your schematic, for the next PCB design step you'll need to use the [schematic capture](#) tool in Altium Designer to import components into a blank PCB layout. First, create a blank printed circuit board document which will generate a PcbDoc file. This is done from the main menu in Altium Designer, as shown below.



Starting a new PCB project in Altium Designer

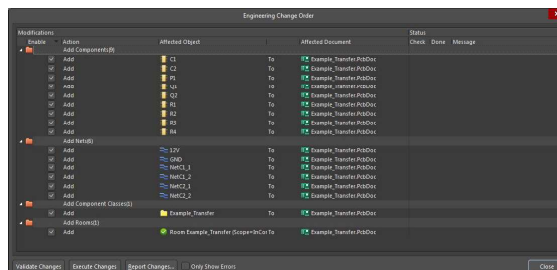
If the printed circuit board shape, dimensions, and [layer stackup](#) for your board have already been determined, you can set them now. If you don't want to do these tasks now, don't fret, your board shape, size, and layer stackup (see Step 4 below) can be changed later. Schematic information is made available for the PcbDoc by compiling the SchDoc. The compilation process includes verifying the design and the generation of your several project documents that allow you to inspect and correct the design prior to transfer to the PcbDoc, such as those shown below. It is highly recommended that you review and update the Project Options at this point, which are used to create the PcbDoc information.



Project Options for Conversion to PcbDoc

Step 3: Synchronize Schematics to Your PCB Board Design

All the tools in Altium Designer work in a united design environment, where the schematic, printed circuit board layout, and BOM are interlinked and can be accessed simultaneously. Other programs force you to manually compile your schematic data, but Altium Designer does this for you automatically while you create your circuit board design. To transfer SchDoc information to the newly created PcbDoc, click on Design > Update PCB (Filename of your new PCB).PcbDoc. An Engineering Change Order (ECO) dialog will open listing all components and nets from the schematic, similar to the one below.



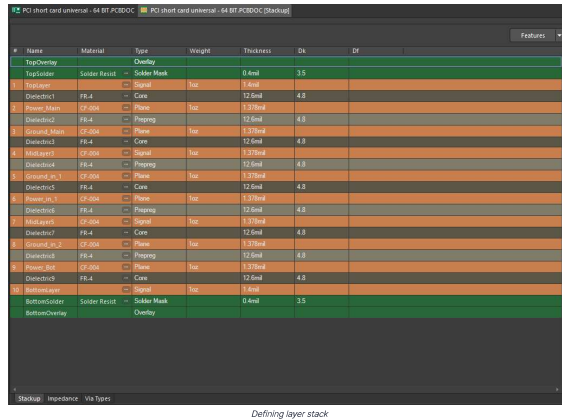
Engineering Change Order example

Verify the changes (addition of the SchDoc information to the project without error) by clicking on the Validate Changes tab. If the status for all items is green, then click on the Execute Changes tab. To complete the process, close the dialog. Altium Designer can save you a substantial amount of time during this PCB design step.

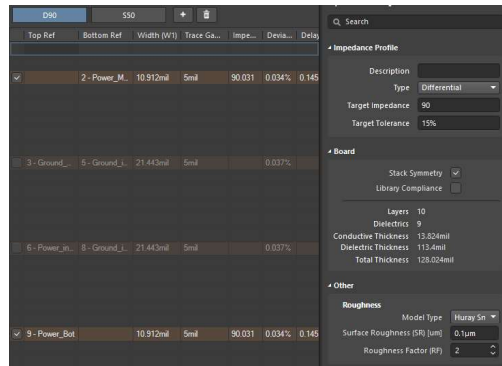
Step 4: Designing Your PCB Stackup

When you transfer your schematic information to the PcbDoc the component footprints are shown in addition the board outline specified. Prior to placing components you should define the PCB layout (i.e. shape, layer stackup) using the Layer Stackup Manager, shown below.

If you're new to the printed circuit design world, most modern PCB board design concepts will start with a 4-layer board on FR4, although you can define any number of layers you like in Altium Designer. You can also take advantage of the Materials Stackup Library; this lets you choose from a range of different laminates and unique materials for your printed circuit board.



If you're working on a high speed/high frequency circuit board design, you can use the built-in impedance profiler to ensure impedance control in your board. The impedance profile tool uses an integrated electromagnetic field solver from Simberian to tailor the geometry of your traces to meet a target impedance value.



Make sure you decide your routing style before you start calculating impedances. For example, will you be using a coplanar line on a thicker dielectric layer, such as in this example, or will you be using a standard microstrip/stripline on a thinner dielectric? These choices are important because they affect the trace width you can use in the circuit board design; different routing styles will enforce different trace widths in order to hit your required impedance.

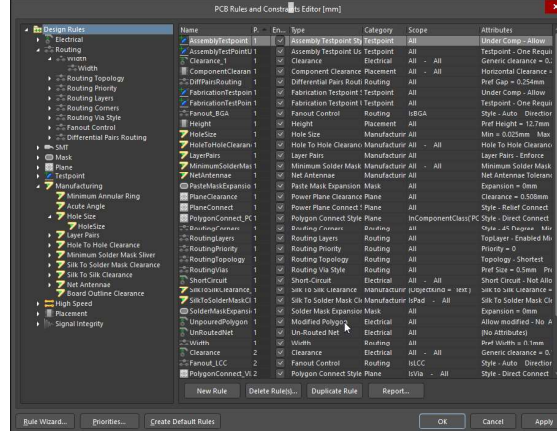
Also note whether you'll be using any differential pair routing in the design. The impedance solver in the Layer Stack Manager enables differential pair solutions as well as single-ended, so you can determine both requirements and use these when routing. After the layer stack is created and any impedance profiles are determined, it's time to set up your circuit board design rules so that you can start placing and routing components.

Step 5: Defining PCB Board Design Rules and DFM Requirements

The number of PCB board design rule categories is extensive and you may not need to use all of these available rules for every design. You can select/deselect individual rules by right clicking on the rule in question from the list in the PCB Rules and Constraints Editor, below. Your PCB board design rules are divided into several categories, which includes:

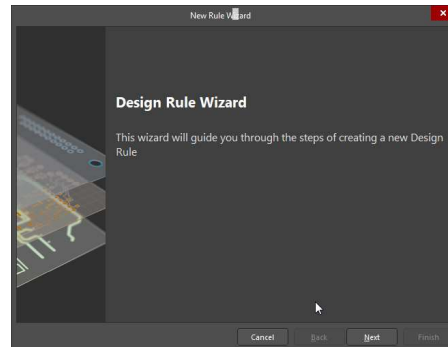
- Clearances between objects in the PCB layout, such as between traces and pads
- Copper or solder mask feature size limits, such as holes and solder mask slivers
- Routing rules, including trace width and length limitations that can be enforced on certain nets
- High speed and signal integrity limits, such as overshoot
- Board fabrication limits and clearances, such as board edge clearance

This is just a sample of the rules that can govern any PCB layout, but these rules are designed to help ensure a board is manufacturable at the required scale with your fabricator's standard capabilities. These rules can be applied to individual objects or to groups of objects using queries or using Net Class objects. Take a look at this article to see how you can set up a Net Class in Altium Designer.



The PCB Rules and Constraints editor in Altium Designer

The rules that you do use, especially for manufacturing, should be inline with the specifications and tolerances for your PCB board manufacturer's equipment. Advanced circuit board designs, such as impedance controlled designs and a number of high speed/high frequency designs, may require very specific design rules that need to be followed in order to ensure your product works properly. Always check your component datasheets for these design rules. If necessary, you can create new circuit board design rules by following the steps of Altium Designer's Design Rule Wizard.



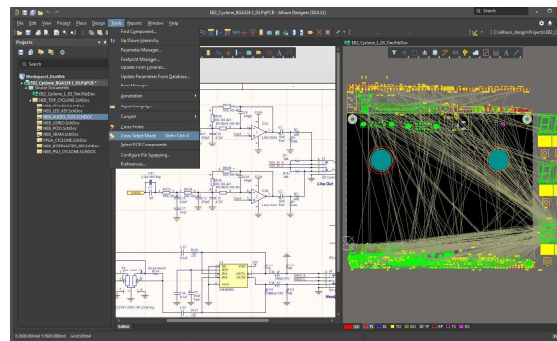
PCB Design Rule Wizard in Altium Designer

Altium Designer will treat your custom design rules just like the built-in design rules. As you place components, vias, drill holes, and traces, the unified design engine in Altium Designer will automatically check the layout against these rules and will flag you visually if there is a violation.

Step 6: Place Components

Altium Designer provides a great deal of flexibility and allows you to quickly place components on your circuit board. You can have your components automatically arranged or you can place them manually. You can also use these options together, which allows you to take advantage of the speed of auto-placement and ensure your board is laid out according to good component placement guidelines. An added advanced feature of this latest version of Altium Designer is the ability to arrange components as groups. You can define these groups in the PCB layout, or you can define groups on the schematic using Cross Select Mode, which is accessible from the Tools menu.

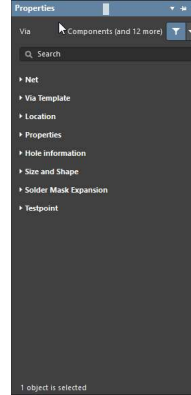
- [Learn more about PCB layout best practices](#)



Component Placement using Cross Select Mode

Step 7: Insert Drill Holes

Before routing your traces, it is a good idea to place your drill holes (mounting and vias). If your circuit board design is complicated you may need to modify at least some of the via locations during trace routing. This can be done easily from the via Properties dialog, shown below.



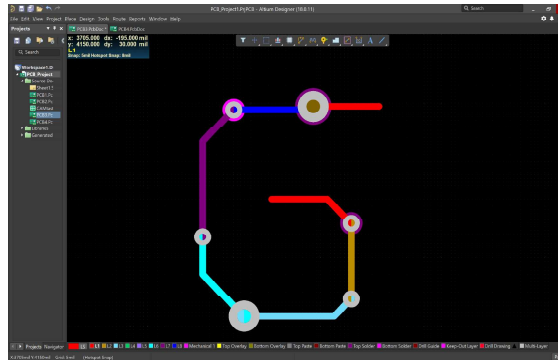
Drill Hole Options Dialog

Your preferences here should be guided by the design for manufacturing (DFM) specifications of your PCB manufacturer. If you already defined your PCB DFM requirements as design rules (see [Step 5](#)), Altium Designer will automatically check these rules as you place vias, drill holes, pads, and traces in your layout.

Step 8: Route Traces

Once you've placed your components and any other mechanical elements, you're ready to route your traces in this PCB design step. As you route your board, try to come up with a strategy to finish your important routes first, then fill in the gaps with the remaining connections as needed. Some of the important routes will include your power nets, any impedance-controlled nets, and any noise-sensitive nets like low-level analog signals. Be sure to utilize good routing guidelines and take advantage of Altium Designer tools to simplify the process, such as [highlighting nets](#) and interactive routing features.

- [Learn more about PCB routing and some best practices](#)

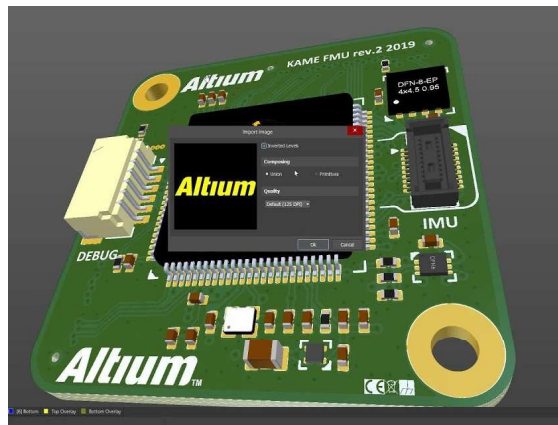


Color Coded Via Routing

Altium Designer includes a number of important tools to help make your routing experience easier and more productive. There is an autorouter engine that uses a modern algorithm to route traces, as well as traverse layer pairs with vias. The auto-interactive routing tools allow you to guide an automated routing feature so that you can speed up complicated routes between components, and the online DRC engine will automatically enforce design rules as you route. These tools will operate on multiple nets simultaneously, making it easy to route a large number of traces in tandem.

Step 9: Add Labels and Identifiers

With the circuit board layout verified you are ready to add labels, identifiers, markings, logos, or any other imagery to your board. It is a good idea to include reference designators for components as this will assist in PCB board assembly. Also, make sure to keep any polarity indicators, pin 1 indicators, and any other labels visible as these will aid PCB assembly and testing. It will also help if you ever need to debug the board while testing. You can also add a company logo and part numbers using the image tools and text tools in the PCB Editor. These elements need to be placed in the Top Overlay or Bottom Overlay layers in the PCB layout.



Logos and part numbers can be added to the silk screen (overlay layers)

Step 10: Generate Design Output Files

Before you create your manufacturer deliverables, it's always a good idea to verify your circuit board layout by running a design rule check (DRC). Altium Designer will do this automatically as you layout your components and route your circuit board design, but it never hurts to run another DRC manually. If your board checks out, then you're ready to release your manufacturer deliverables.

Once your board has passed the final DRC, you need to generate the design files for your manufacturer. The design files should include all the information and data necessary to build your board, including any notes or special requirements to ensure that your manufacturer is clear on what you require. For most manufacturers, you

- [Read more about design output files for manufacturing](#)



Whether you need to build reliable power electronics or advanced digital systems, Altium Develop unites every discipline into one collaborative force. Free from silos. Free from limits. It's where engineers, designers, and innovators work as one to create without constraints. [Experience Altium Develop today!](#)

Learn How Altium Supports PCB Designers →



[More content by Zachariah Peterson](#)



[← Back to Home](#)

