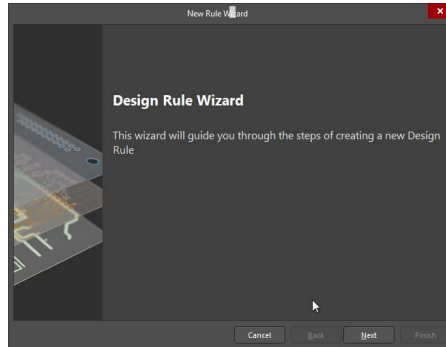


The PCB Rules and Constraints editor in Altium Designer

The rules that you do use, especially for manufacturing, should be in line 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.



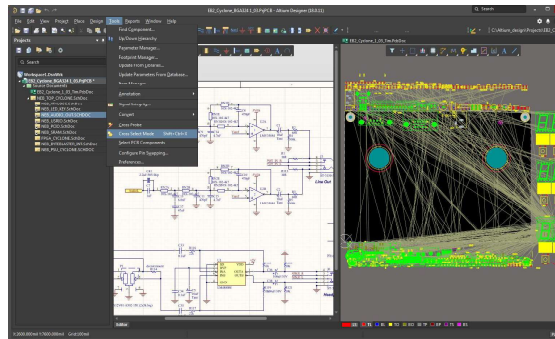
PCB Design Rule Wizard in Altium Designer

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.

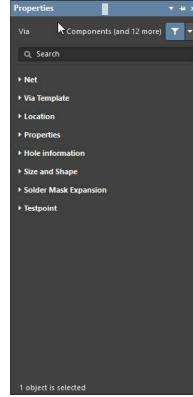
- [Learn more about PCB layout best practices](#)



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.



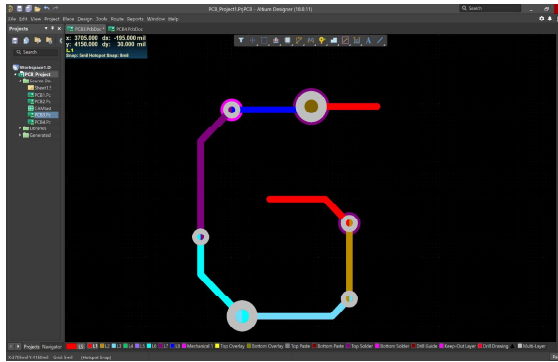
Drill Hole Options Dialog

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 silkscreen (overlay layers)

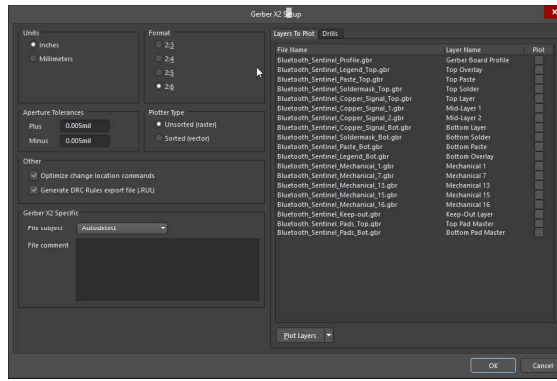
Step 10: Generate Design Output Files

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

will be able to use a set of Gerber files as shown below; however, some manufacturers prefer other manufacturing file formats (IPC-2581 or ODB++).

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



Set of Gerber files

By following the above PCB design steps, the process of creating a comprehensive PCB board design is as easy as counting to ten. Using a systematic approach such as this ensures that all aspects of your circuit board design are accounted for inherently during the process, with minimal need to retrace your steps.

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](#) →



About Author

Zachariah Peterson has an extensive technical background in academia and industry. He currently provides research, design, and marketing services to companies in the electronics industry. Prior to working in the PCB industry, he taught at Portland State University and conducted research on random laser theory, materials, and stability. His background in scientific research spans topics in nanoparticle lasers, electronic and optoelectronic semiconductor devices, environmental sensors, and stochastics. His work has been published in over a dozen peer-reviewed journals and conference proceedings, and he has written 2500+ technical articles on PCB design for a number of companies. He is a member of IEEE Photonics Society, IEEE Electronics Packaging Society, American Physical Society, and the Printed Circuit Engineering Association (PCEA). He previously served as a voting member on the INQITS Quantum Computing Technical Advisory Committee working on technical standards for quantum electronics, and he currently serves on the IEEE P3186 Working Group focused on Port Interface Representing Photonic Signals Using SPICE-class Circuit Simulators.

[More content by Zachariah Peterson](#)

Related Resources

The Peak of Engineering Efficiency is Now Available in Altium Designer 17

The Peak of Engineering Efficiency and PCB Design Technology is Now Available In

[Read Article](#) →

For Your DC Power Application, Which Switching Power Supply Topology is Right?

With so many switching power supply topologies to choose

[Read Article](#) →

Heat Sink PCB Mounting: Maximizing Cooling with Best Design Practices for a Surface Mount

Image Source: Flickr user beignre (1/20/17 7:01) Just Blue

[Read Article](#) →

How to Use Approach PCB Design

Modular PCB taking more defined out

[Read Article](#)

Related Technical Documentation

QuickNav - Panels in Altium Designer

Explore Altium Designer 24 technical documentation for QuickNav - Panels and related features.

[Read Documentation](#) →

Managing Project Documents in Altium Designer

Explore Altium Designer 24 technical documentation for Managing Project Documents

[Read Documentation](#) →

Navigating a Document in Altium Designer

Explore Altium Designer 24 technical documentation for Navigating a Document and related features.

[Read Documentation](#) →

Saving Print Documents in Altium Designer

Explore Altium Designer 24 technical documentation for Saving Print Documents

[Read Document](#)

[← Back to Home](#)

Platform-based Solutions

Discover
Develop
Agile

Tools

Altium Designer
Octopart

Platform

Altium 365

Product Extension

Power Analyzer
Signal Analyzer

Company

Our Customers
Trust Center
Newsroom

Resources & Support

Status
Downloads

Learning Hub

Support Center
Bug Crunch
Forum
Documentation
Ideas
Education
Professional Training / Certification
Webinars

Education

Student Lab
Educator Center
Altium Education Curriculum

Careers

Career at Altium

