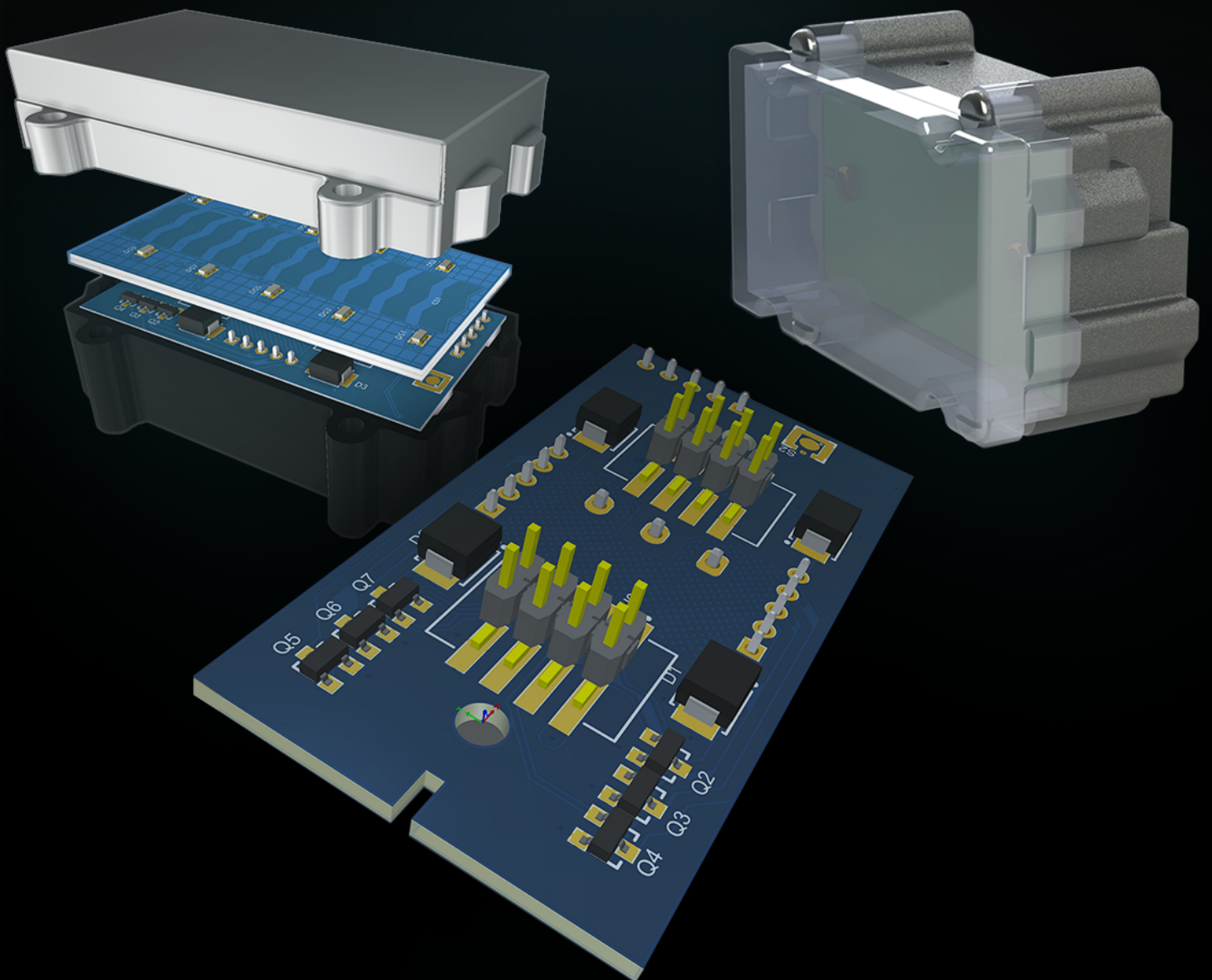


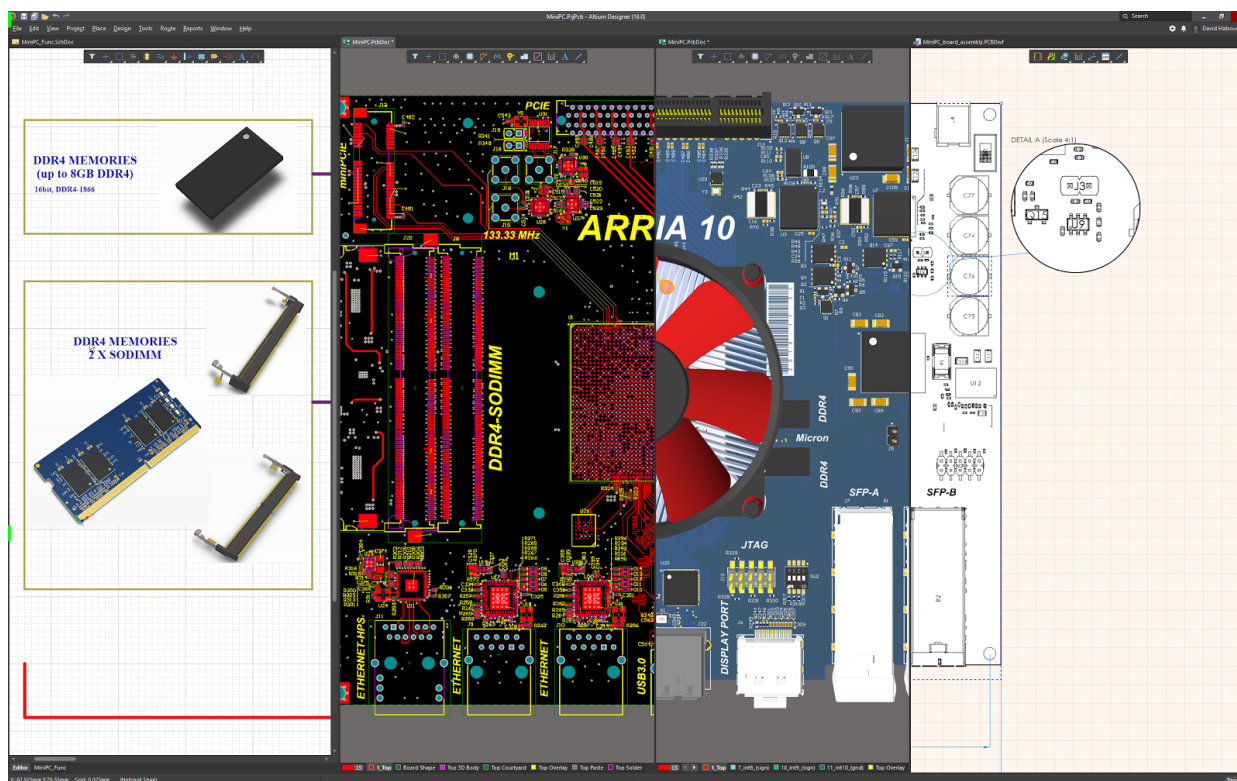
Altium[®]

ALTIUM EVALUATION GUIDE FOR OrCAD[®] USERS



PCB DESIGN IS A WORK OF ENGINEERING ART

When designing electronics, productivity and performance are two of your foremost concerns. Immovable time to market deadlines makes efficiency a must, as you work hard to create precise layouts and exact fittings, for the optimum performance. To achieve both quality and efficiency, you can't afford not to invest in a **complete PCB design platform**. Altium Designer® combines exhaustive scientific knowledge with a natural design intuition to create a unified, **cohesive design environment**. It can meet all of your design challenges, with a proven track record of delivering the features you need in the timeframe you've been given.



Altium Designer - The most cohesive, Easy Modern and Powerful PCB design solution

WHAT IS GETTING IN THE WAY OF YOUR NEXT MASTERPIECE?

There are any number of problems that can hinder your PCB design. Maybe you find you're constantly falling short of meeting the minimum design specifications within the current design environment. Maybe you've missed some of your release dates or exceeded your product cost targets. You have the expertise to design the perfect board, but for some reason, you're still unable to achieve your feature elegance targets. It's not just you, either. Your entire design team is experiencing similar problems and have been for years. So now, you've finally decided, it's time for a change.

Electronic products are rapidly increasing in complexity, with larger circuits but decreasing overall board sizes. As designs continue down this path, you can no longer accept the inability to meet your goals as a normal part of the job. There has to be a solution. What's more, it needs to be a complete solution: feature-rich products, product differentiation, real-time connection to your supply chain, effective design data management, reuse, change order management, and efficient collaboration across the entire ECAD-MCAD design process. How can you achieve all of this? Let's take a closer look at the solutions that Altium Designer provides.

ALTIUM DESIGNER EVALUATION GUIDE FOR ORCAD® USERS

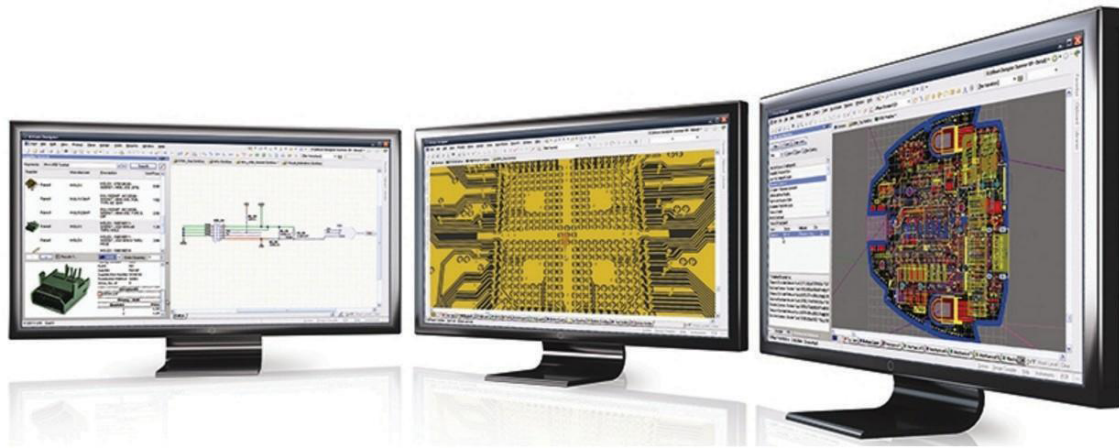
ORCAD® PCB PRODUCTS

No matter what Cadance® product you were using, be it OrCAD® PCB Designer Standard, Professional, or an upscaled Allegro® PCB Designer, you'll find Altium Designer well-suited to meet all of your design challenges. With our world-class research and development team, the Altium goal has always been to produce best-in-class software for every PCB designer.

OVERVIEW OF ALTIUM DESIGNER

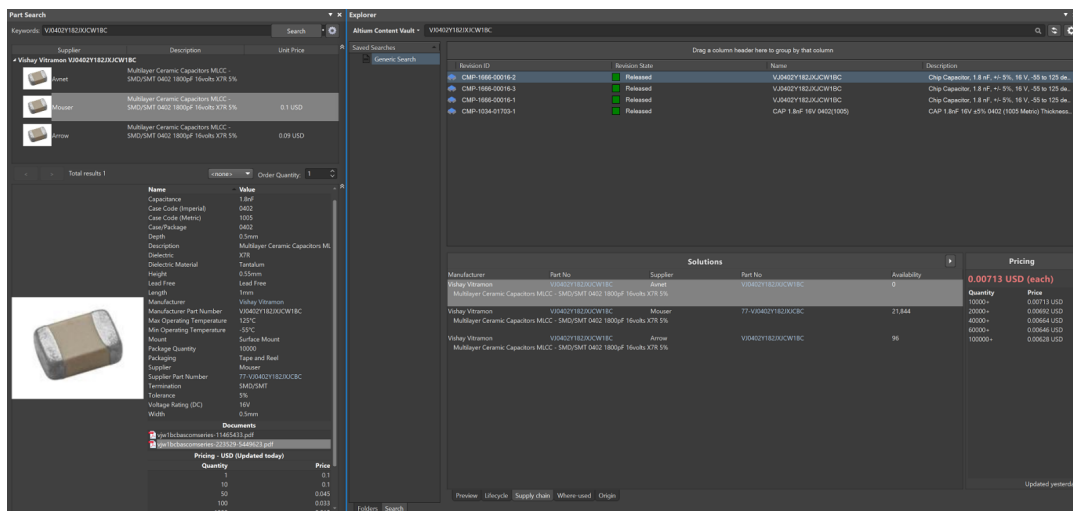
A cohesive environment for your design, analysis and release processes

Altium Designer provides the most cohesive PCB ECAD design environment that's modern, and easy to learn and use. You have the features you need at your fingertips to make key decisions early in the design process. You can perform tasks more efficiently and implement checks and balances throughout the process to keep everything in check. Altium Designer also interfaces seamlessly with third party analysis, synthesis and 3D mechanical software.



Native3D™ powered Unified Design Environment

With the **advanced supply chain management** system in Altium Designer, you have real-time visibility into both your internal and external supply chain data, as well as access to all available information about sourced components. This means, rather than including parts that turn out later to be unavailable, with long lead time, or cost-prohibitive, you can make more intelligent part choices during any stage of the design process, reducing the likelihood of costly and time-consuming rework late in the design cycle.

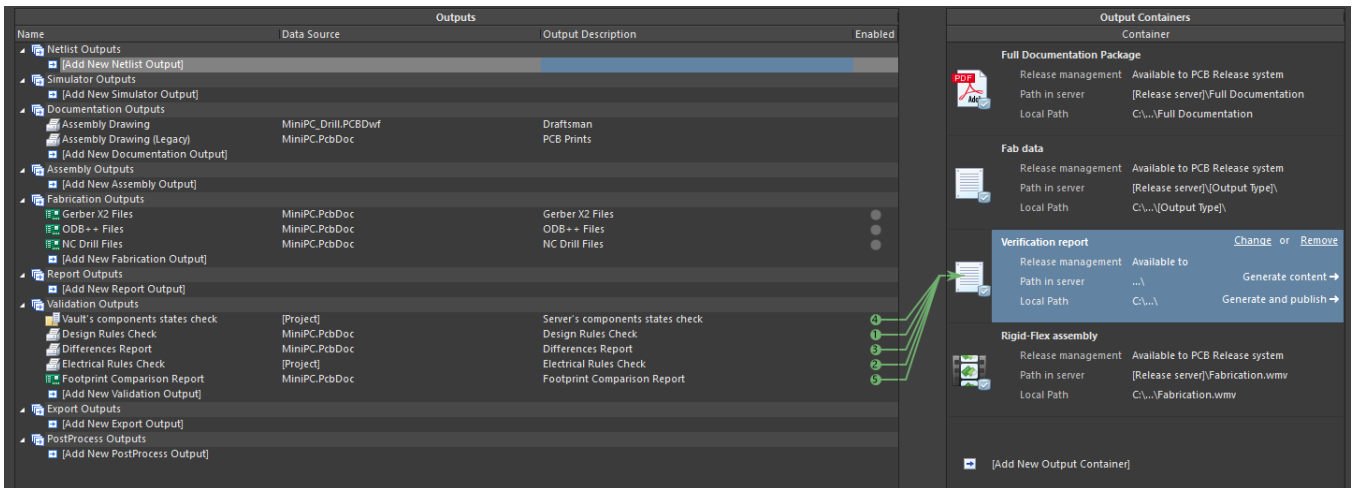


Integrated Real-Time Supply Chain

ALTIUM DESIGNER EVALUATION GUIDE FOR ORCAD® USERS

Components, design specifications, documentation, and revisions are all stored in a central repository. You can attach **native built-in or external version control** capabilities within the system, to keep track of everything you and your team have been doing, and graphically and logically verify the work. You then synchronize data and use the **built-in ECO** capabilities to ensure that your entire team will **automatically see changes** and work off of the most up-to-date files and information available.

To configure and save your publication outputs in a central location, use the Altium Designer **Output Job Editor**. You can publish selected outputs to smart linked PDF and **3DPDF**, print or generate files, and add them to your project. The same Output Job can have any number of publications configured, ready for reuse everytime you update your project.



Easy to Generate Documentation Outputs, in Multiple Formats

Altium Designer also has a streamlined **project releaser** that provides design rule checking and **validation for your entire PCB design**, allowing you to release accurate output data knowing that the sources are fully up to date. In addition, you will have a visual representation of your project history, so you can go back at any point to retrieve, modify, and re-release your designs quickly and easily. This accelerates the design process and eliminates the need to reproduce data and documentation. An extensive **design reuse** capability reduces errors and re-spins by using previously validated and proven designs.

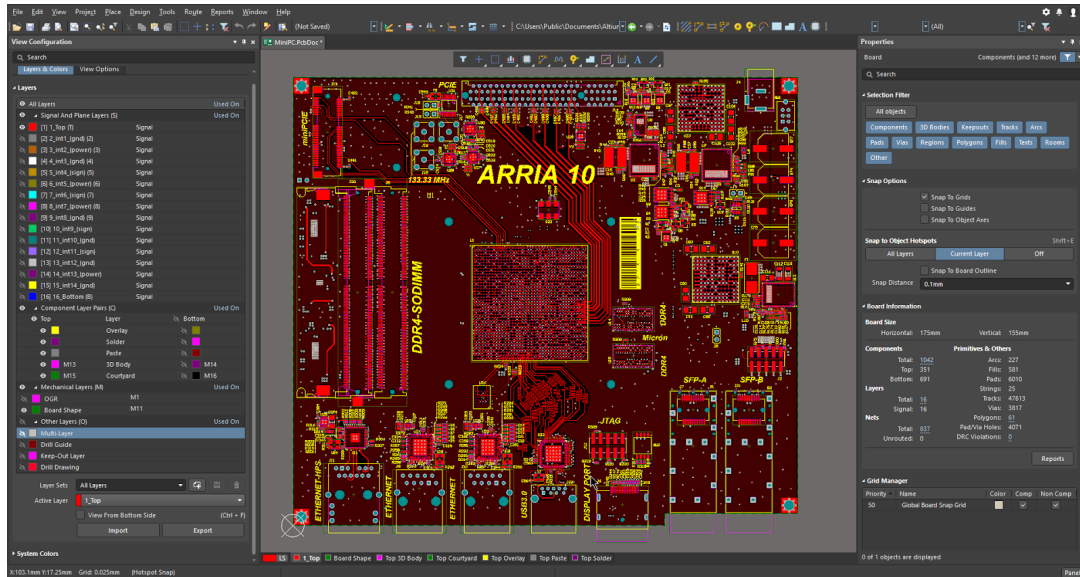
ALL WITHIN ONE, MODERN USER INTERFACE

Most if not all modern EDA tools don't look that modern, as their user interface has not changed much in the last decade. They are still not oriented toward simplicity and productivity; with disorganized views, cluttered information, and really old graphics. Altium Designer now delivers a fresh, **modern user interface (UI)** that is more logical and streamlined. With updates that greatly improve and consolidate many functions enabling designers to reach optimal productivity quickly and efficiently, without having to go through countless menus, toolbars and dialogs.

All Altium Designer features are presented within the same modern user interface. Whichever aspects of the design process you're responsible for, a consistent selection and editing paradigm allows you to move quickly between design tasks. The **context sensitive** UI changes when you switch from one aspect of the process or document to another, to provide you with the most relevant and intuitive selections. If you focus on one element of the design process, the UI can be configured to match your preferences. The **consistent look & feel** allows you to become proficient quickly, as you take on additional design tasks.

ALTIUM DESIGNER EVALUATION GUIDE FOR ORCAD® USERS

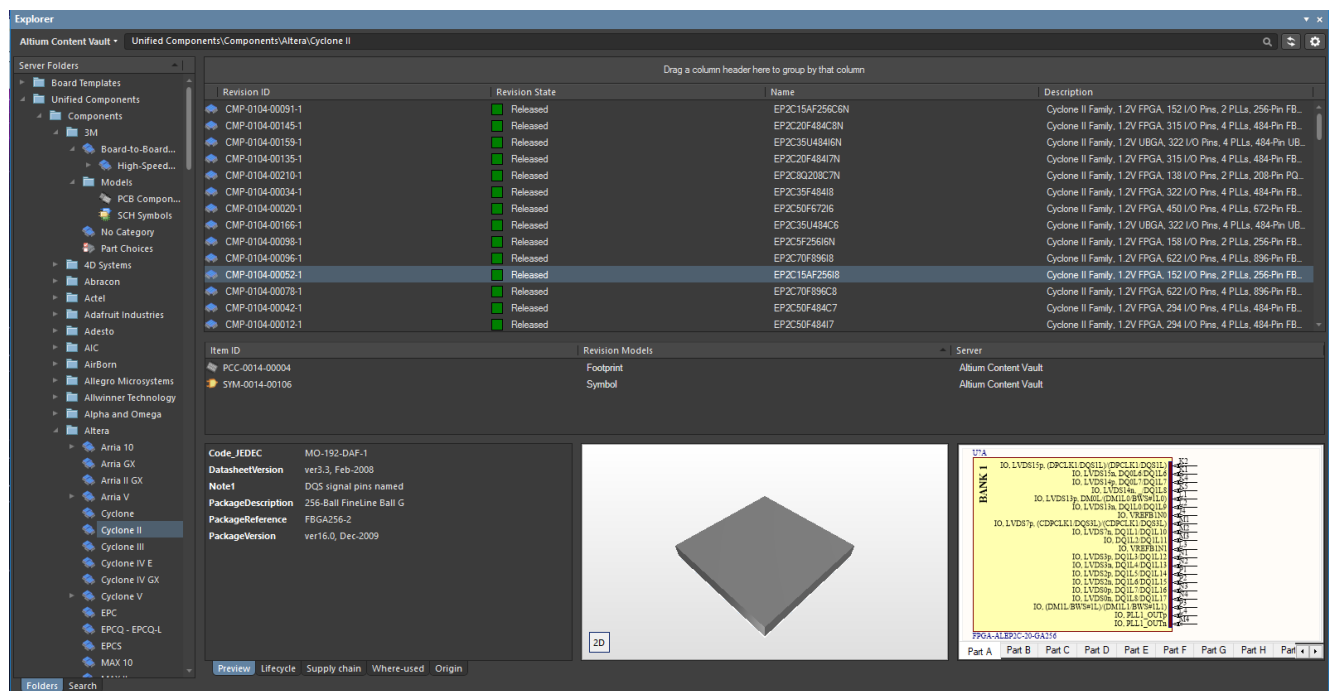
With a single, unified, adaptive properties panel, whether in PCB, schematics, or library, you can quickly access document options and their objects. The projects panel now puts all the design files and navigation data within one place, making it easy to find and access the right information about your design structure whenever you need it. And with a powerful graphics engine, the layers and colors panel now gives you unprecedented control over how you view and mask your board, layers, 3D components, and even system level colors. And if you can't remember where to find something, simply perform a global search to find any object or command. The new interface in Altium Designer provides a design environment that is cleaner, leaner, and more visually appealing - with all the powerful features and functionality you have come to rely on, made ever more accessible.



Modern Consolidated Interface for a More Productive Workflow

LIBRARY AND COMPONENT MANAGEMENT

Whether you are using standard Orcad® libraries, CIP, CIS, or PartLink, you will feel right at home using Altium Designer to bridge the gap between library components and supplier data.



Unified Component Model Management

INTEGRAL LINK TO SUPPLY CHAIN MANAGEMENT WITH ALTIUM DESIGNER

When it comes to providing a complete solution for the PCB design, development, and production process, Altium Designer has proven to be the leading PCB design platform. One of the fundamental aspects of the process includes a **close connection to supply chain** and **real-time component management**.

Altium Designer provides a live supplier search engine from which you can search across multiple distributors and add links directly to your schematic components. In order to optimize the process, you then automatically verify part numbers for every component in your Bill of Materials (BoM) and receive real-time pricing and availability information with an advanced BoM Tool.

You can also place components directly from the Altium Content Vault into your design. The Altium Content Vault is a cloud-based source with a wealth of design content, including access to template designs and thousands of unified components, to help streamline your design process. The Unified Component incorporates supply chain information with links to suppliers, quantities, and pricing, which can then be included on your BoM in real time.

Additionally, Altium Designer includes an integrated powerful parametric components search engine (through Ciiva SmartParts™ and Octopart™), that provides you instant access to information, including real-time pricing and availability, second sources, and lifecycle states for tens of millions of components. Documentation, including datasheets and application notes, can also be accessed with a single click, with direct links to supplier websites.

Real-Time BOM Management in Altium Designer - ActiveBOM

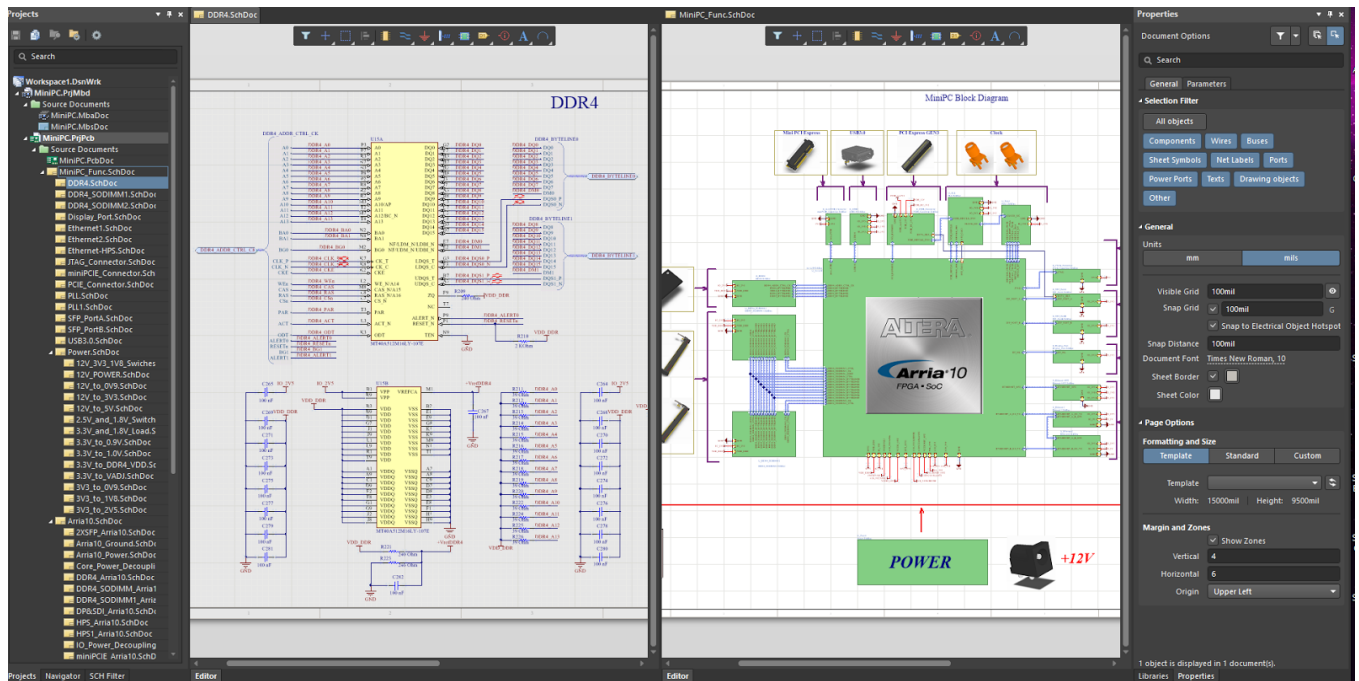
ActiveBOM® offers a live presentation of the design from the outset, providing early and ongoing critical supply chain information, such as availability and pricing. ActiveBOM facilitates real-time cost estimation and tracking for your board design, by bringing to the table a system that effectively and efficiently aides you in managing costs and availability of items used in your design. It allows you to define target pricing at the individual item level. You can then track how actual costing fares against these estimates, and gives you a timely flag if any cost or lead-time blow-outs are on the near horizon. In addition, you can quickly assess item availability, complete with notification if there is a risk in the supply of a chosen part (i.e a part going into End-of-Life state). ActiveBOM also allows specifying pin-compatible backup part choices directly in the BOM referred to as alternative part choice. Having pin-compatible backup part choices nearly eliminates supply chain issue risks for manufacturing. In turn, you can design while taking into account potential manufacturing blowouts, reducing time to market and minimizing unexpected costs and design changes.

Line #	Name	Description	Designator	Quantity	Revision Status	Manufacturer 1	Manufacturer PartNo 1	Manufacturer Lifecycle 1	Supplier 1
1	74754-0101	SFP+ Cage, Single...	B1, B2	2	Out of date	Molex	74754-0101	Volume Production	Mouser
2	CGB2A1X5R1E10...	Chip Capacitor, 1...	C1, C4, C7, C23...	19	Up to date	TDK	CGB2A1X5R1E105K0...	Volume Production	Amet
3	ECJ-0EC1H680J	Chip Capacitor, 68...	C106	1	Up to date	Panasonic	ECJ-0EC1H680J	Obsolete	Digi-Key
4	C3216X5R1A476...	Chip Capacitor, 47...	C11, C12, C18, C...	24	Up to date	TDK	C3216X5R1A476M16...	Volume Production	Amet
5	GRM155R61A224...	Chip Capacitor, 22...	C13, C15, C16, C2...	27	Up to date	Murata	GRM155R61A224KE1...	Volume Production	Arrow
6	VJ0402Y153JXQP...	Chip Capacitor, 15...	C14, C24, C33, C4...	7	Up to date	Vishay Vitramon	VJ0402Y153JXQPW1...	Volume Production	Mouser
7	CGJ2B2X7R1E22...	Chip Capacitor, 2...	C107	1	Out of date	TDK	CGJ2B2X7R1E22K0...	Volume Production	Arrow
8	GRM1535C1H560...	Chip Capacitor, 56...	C108	1	Up to date	Murata	GRM1535C1H560J0...	End of Life	Mouser
9	ECJ-0EC1H270J	Chip Capacitor, 27...	C17, C51	2	Up to date	Panasonic	ECJ-0EC1H270J	Obsolete	Amet
10	GRM155R71H103...	Chip Capacitor, 10...	C2, C3, C57, C70...	20	Up to date	Murata	GRM155R71H103KA8...	Volume Production	Mouser
11	ECJ-0EC1H150J	Chip Capacitor, 15...	C48, C64	2	Up to date	Panasonic	ECJ-0EC1H150J	Obsolete	Digi-Key
12	C1005K7R1E104M	Chip Capacitor, 0...	C68, C69, C79, C1...	132	Up to date	TDK	C1005K7R1E104M	Volume Production	Mouser
13	EEH-ZA1E331P	Aluminum Electrol...	C74, C77	4	Up to date	Panasonic	EEH-ZA1E331P	Volume Production	Amet
14	C1005CG1H220...	Chip Capacitor, 22...	C8, C27, C36	3	Up to date	TDK	C1005CG1H220J06...	Volume Production	Mouser
15	6TFF330M9L	Tantalum Capacito...	C80, C81, C87, C8...	6	Up to date	Panasonic	6TFF330M9L	Volume Production	Farnell
16	GRM31CR60J107...	Chip Capacitor, 10...	C82, C84, C89...	35	Up to date	Murata	GRM31CR60J107ME...	Volume Production	Farnell
17	GRM31CR61E226...	Chip Capacitor, 22...	C9, C10, C28, C29...	22	Up to date	Murata	GRM31CR61E226KE...	Volume Production	Arrow
18	VJ0402Y182JXJC...	Chip Capacitor, 1...	C94	1	Up to date	Vishay Vitramon	VJ0402Y182JXJCW1...	Volume Production	Mouser
19	GRM155R71E473...	Chip Capacitor, 47...	C110	1	Up to date	Murata	GRM155R71E473KA8...	Volume Production	Amet
20	GRM155R71E223...	Chip Capacitor, 22...	C111	1	Up to date	Murata	GRM155R71E223KA6...	Volume Production	Amet
21	GRM155R71H681...	Chip Capacitor, 68...	C117	1	Up to date	Murata	GRM155R71H681KA0...	Volume Production	Arrow
22	GRM033R60J104...	Chip Capacitor, 10...	C118, C130, C14...	226	Up to date	Murata	GRM033R60J104KE1...	Volume Production	Arrow
23	GRM188R61A225...	Chip Capacitor, 2...	C297, C329, C355...	10	Up to date	Murata	GRM188R61A225KE3...	Unknown	Amet
24	GRM155R71H472...	Chip Capacitor, 4...	C308	1	Up to date	Murata	GRM155R71H472KA0...	Volume Production	Amet
25	C2012X5R1C106K...	Chip Capacitor, 10...	C330, C356, C370...	10	Up to date	TDK	C2012X5R1C106K08...	Volume Production	Digikey

ActiveBOM - Real-Time Cost Estimation & Part Availability

ALTIUM DESIGNER SCHEMATIC CAPTURE: A TECHNOLOGY DIFFERENTIATOR

Altium Designer schematic capture technology has been long recognized as a technology differentiator. Engineers and designers alike find that the Altium Designer schematic features are **easy to learn**, vastly improving productivity on all designs, from relatively simple single-sheet schematics to complex multi-sheet hierarchical projects. **Starting a schematic is fast and easy** with intuitive dialogs, e.g., editing workspace and establishing sheet design parameters, preferences and associated documents. This allows you to manage versions immediately. You can quickly select and place qualified components from integrated libraries and real-time links to component suppliers. Wiring, bus creation, and net labeling are all incredibly fast.



Modern and powerful Unified Schematics Capture Editor

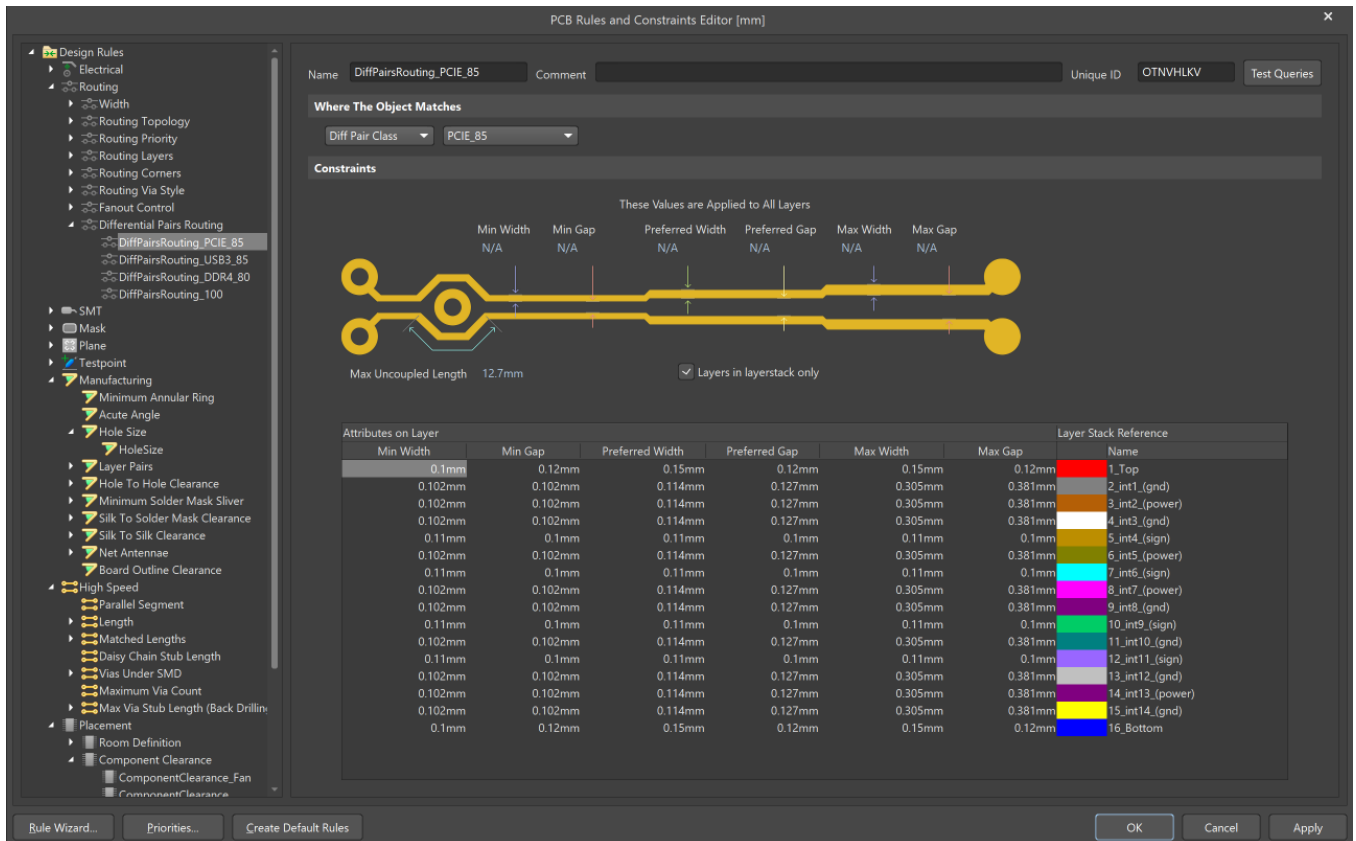
A key feature of Altium Designer is the intuitive method of setting component classes, net classes, and placement rooms. When the design is transferred to the PCB, this information can then be generated automatically - a significant benefit of creating a well-structured, hierarchical design up-front. Using this feature, you can compile your project efficiently and check to ensure that circuits are drawn and wired correctly. Altium Designer also provides powerful ECO features, to let you transfer a captured design to a new PCB, make changes to an existing design on either the schematic or PCB, **synchronize the schematic and board**, and compare and resolve any differences.

CREATE THE MOST ORGANIZED AND EFFICIENT BOARDS IN ALTIUM DESIGNER

An organized and efficient placement is critical to your PCB. **Dynamically place and drag** components that push, avoid, and snap-to alignment with other components on your board layout. Features exist to allow you to **align multiple components** easily. One differentiating feature is the PCB editor's ability to mask or filter objects in the workspace. This will fade everything in your panel except the objects of interest. When you select a net name in the panel, the workspace display will change, zooming to show the nodes in the net and fading out everything except the pads and connection lines in the net. As an alternative to masking, you can also completely hide one, several, or all of the connection lines. Interactive routing includes **multi-route, any-angle, push-and-shove, follow-me, high-speed diff-pairs**, and an **auto-complete** feature that will speed you to complete the design. Another key feature of Altium Designer is the Layer Stack Manager in which routing layers, also referred to as signal layers, are set up. The display of all layers, and the addition of mechanical layers, is controlled in the View Configurations dialog.

PCB RULES AND CONSTRAINTS IN ALTIUM DESIGNER

True to its unified nature, Altium Designer provides you with a streamlined **PCB Rules and Constraints Editor**. In one editor, you can browse, create, prioritize, define scope, edit, duplicate, and delete rules, all with **immediate visual feedback**. You can also export a rule template for reuse. Rules are divided into an easily manageable number of categories. Within the editor, rule types are defined and their attributes assigned. Rules are organized hierarchically and viewed in a tree. A report feature provides a table-based summary for a straightforward review.



Constraint Driven PCB Design with Design Rule Checking

When evaluating, note that a rule scope is a **query that you build**, in order to define all the member objects that are governed by that rule. Scoping allows you to decide exactly what a rule's precedence will be and how it will be applied to target objects through a query. You can even define multiple rules of the same type, each targeting a different object or class. Queries are easy to access for any rule. Advanced Queries can be created with an intuitive Query Builder or directly typed in to the rule's scope.

In addition to scoping, there is also a user-defined priority setting. The combination of rule scoping and priority is very powerful and gives you an unprecedented level of control. This allows you to target the design rules for your board precisely. Finally, a new rule can also be created using the New Rule Wizard. The wizard will guide you step-by-step in creating the rules, scope, and priority.

FAST AND HIGH-QUALITY ROUTING IN ALTIUM DESIGNER - ACTIVEROUTE®

ActiveRoute, included in Altium Designer, brings a new approach to interactive routing - select the connections and ActiveRoute them to produce high-quality routes, in a fraction of the time it would take to manually route them. Rather than allowing an autorouter to do its best at routing the entire board, ActiveRoute acknowledges the reality that board design is a highly interactive process, where the best results are produced by skilled designers using powerful tools, under their control. Altium Designer enables this by giving you easy intuitive control over the selection of the connections or routes of interest.

ALTIUM DESIGNER EVALUATION GUIDE FOR ORCAD® USERS

Unlike other interactive routing technologies, ActiveRoute works on **multiple layers simultaneously** while **adhering to your design constraints** so you don't have to worry about breaking any rules. It also has strong support for modern design techniques, including differential pairs and room-based width requirements. ActiveRoute lets you break out of and route large, fine-pitch BGAs by instructing it where to route them (i.e. select layers, **draw a guide path**), and letting it do the heavy lifting for you.

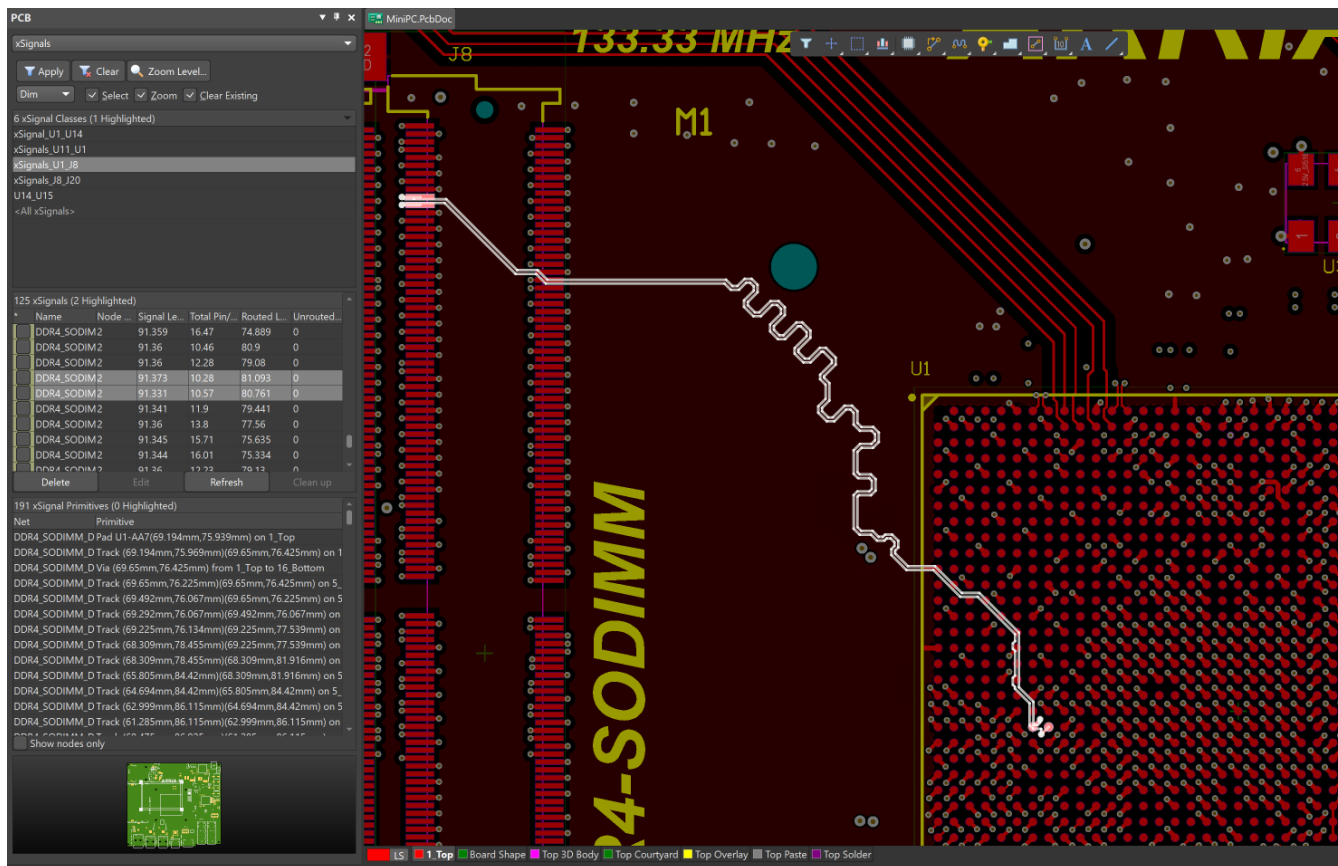
Complementing ActiveRoute, the **Glossing engine** carefully analyzes selected routes, neatening and shortening them. The Glossing engine also delivers a Retrace Selected command, which can be used to update the selected routes to the current routing rule settings - this enables you to fatten up that existing power routing, or update that differential pair to new width and gap settings. By routing on multiple layers simultaneously, routing is faster, traces are evenly distributed, and the ability to complete the routes increases significantly. The result: a beautiful, expert, manual-like, glossed routing, without the hours of manual work.



ActiveRoute With Length Tuning - Before and After (<25 Seconds later!)

OTHER DIFFERENTIATING PLACEMENT AND ROUTING FEATURES IN ALTIUM DESIGNER

Powerful Altium Designer routing technology includes differential pair routing, visualization of allowable routing areas, length tuning, definable rule areas for dynamic routing adjustment, adding/removing teardrops, **DDR3/4 and USB-3 xSignal wizard**, adding/removing via stitching and shielding, copy routing and placement for repeated circuitry, creating panelized boards with corresponding fabrication data, and more. Plus, you can easily save, share, and reuse your most trusted design assets with managed schematic sheets and snippets for design reuse. You can gain even greater control over your clearance checking with enhanced testpoint clearance checks between testpoints, through-hole pads, and inter testpoint spacing. And **solder mask expansions** provide even more precision, with user-definable expansion options from hole edge or pad edge.

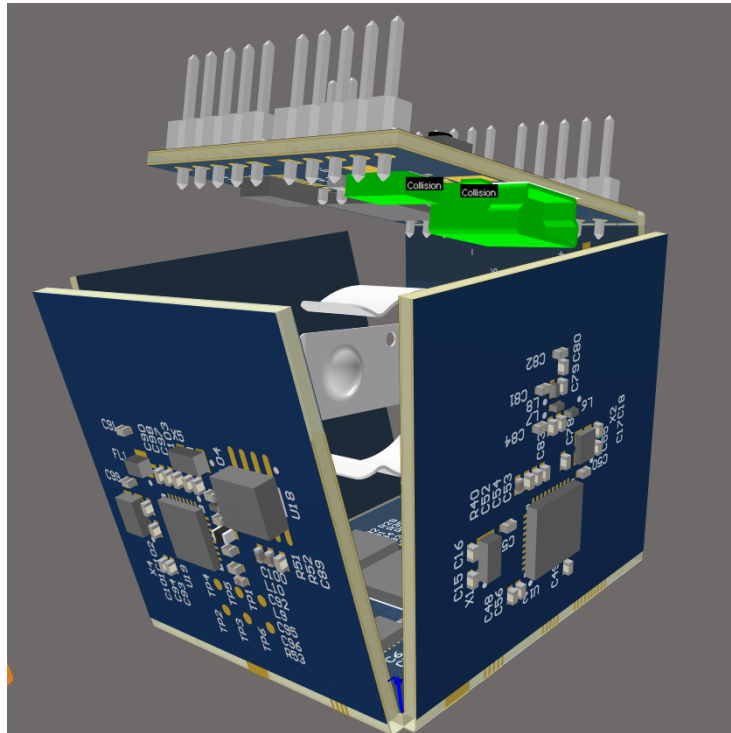


xSignals - Automated High-Speed Signals for High-Speed Topologies

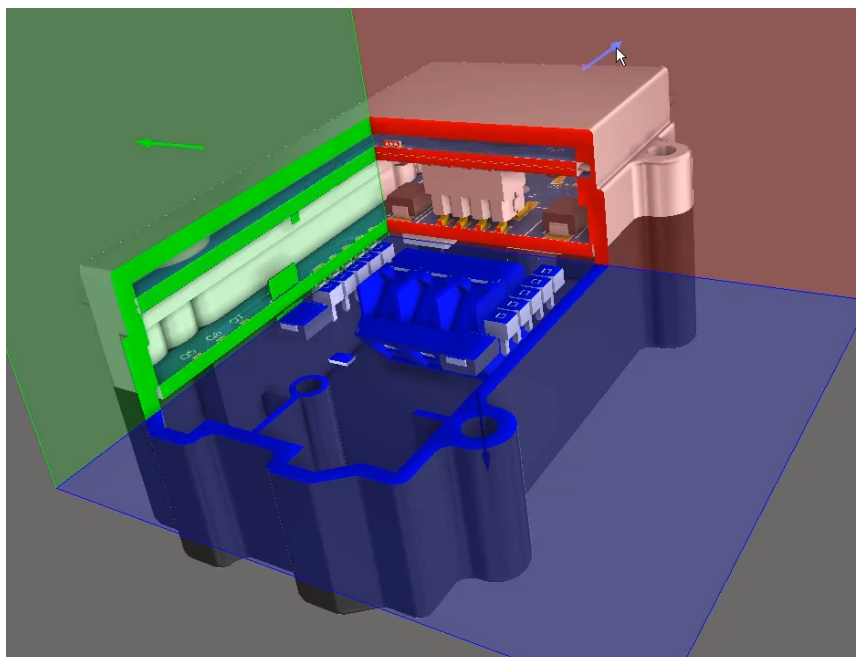
Altium Designer was first to support 3D rigid-flex design, introduced to the PCB tools market in 2014. With this powerful technology, you can easily define material selections and intelligently route your rigid-flex board layout, then dynamically visualize your engineering work of art in native 3D, to make sure that the folding/folded board does not create component clearance violations, in real time. Then, export the folded mechanical model of the assembly to MCAD to ensure perfect fit in the enclosure.

TRUE MCAD COLLABORATION WITH NATIVE3D IN ALTIUM DESIGNER

Altium Designer was the first PCB design product to provide true ECAD/MCAD collaboration with Native 3D editing features, to visualize, compare, merge, track, and comment on design changes. It seamlessly integrates **electrical and mechanical design data** into your workflow, providing designers with real-time visibility into incremental changes. This allows the electrical and mechanical engineering work to be done **simultaneously** and in parallel. You can represent your component footprints with extruded body, Step, SolidWorks, or Parasolid models. You can visualize exactly how your board will fit your mechanical enclosure and fix any collision errors in seconds. And you can perform real-time clearance checking for components and mechanical enclosures, and generate **folded STEP models**.



Detecting PCB Collisions in 3D as you Work



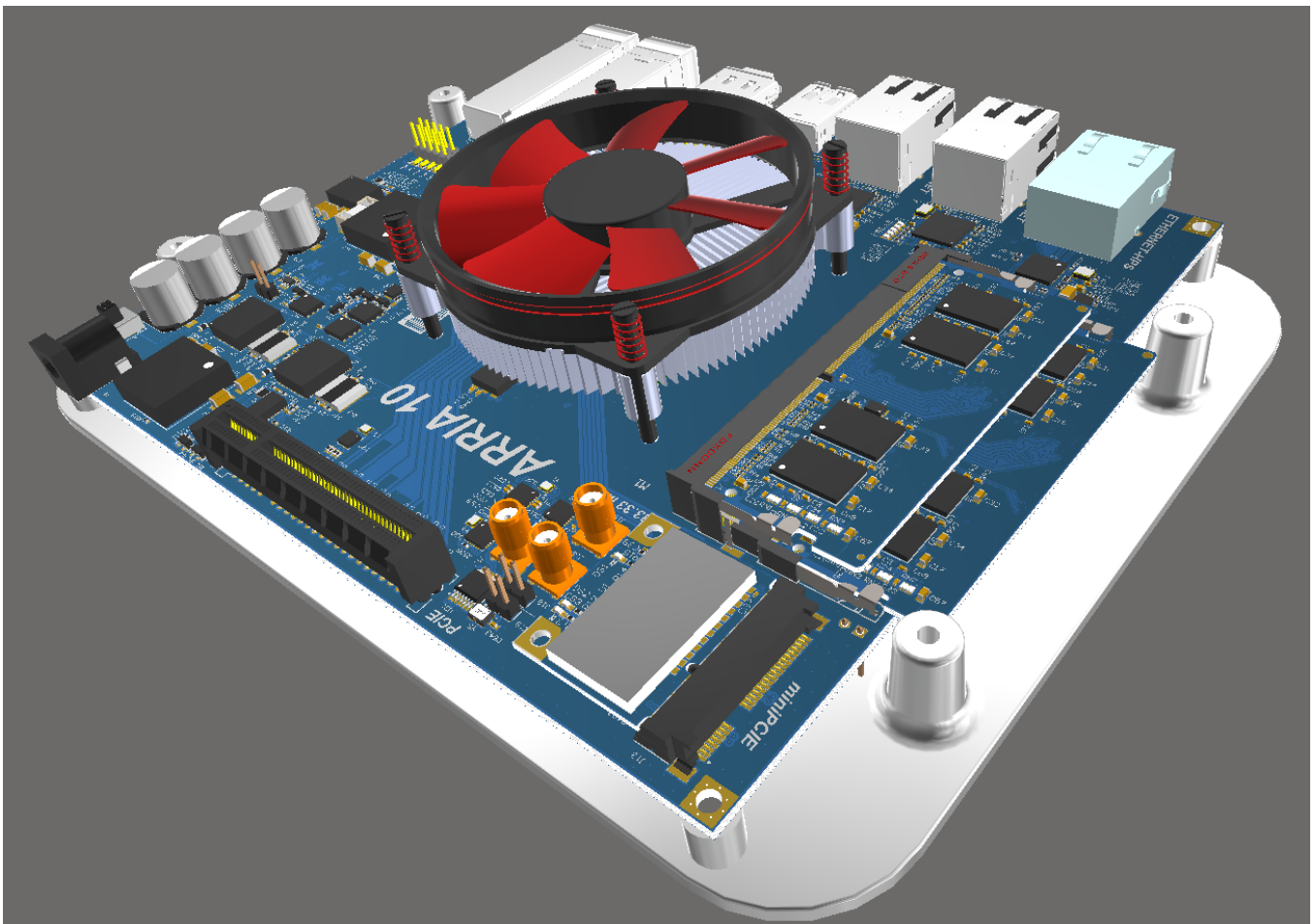
Multi-Board Assembly Enclosure Fit-Check with Cross-Sectional View

ALTIUM DESIGNER EVALUATION GUIDE FOR ORCAD® USERS

With circuit boards not existing in isolation, and often assembled together with other boards which are then housed inside an enclosure, Altium Designer now supports creating and managing multi-board assemblies. You can define the logical (schematics) structure of the system in a multi-board schematic, with each logical block in the multi-board schematic referring to a physical (PCB) design. Then the physical multi-board design is created by transferring the system design into a multi-board assembly. This enables designers to verify at the system level how their “child” PCBs are electrically and physically connected while maintaining the integrity of their pin and net connectivity.

Altium Designer gives you the design space where you can plug together multiple boards, and the tools to manage the whole system’s interconnects, resolve conflicts, and update child projects. And with the state-of-the-art 3D multi-board assembly editor, you can have the separate boards be rotated, aligned and plugged into each other. It also allows other parts, including other boards, assemblies, or STEP format MCAD models, to be imported and positioned in the assembly. If you need to move a connector to align with another or an enclosure hole, you can move it in the context of the multiboard assembly and then synchronize the child design, guaranteeing a right-first-time fit.

Altium Designer brings you system-level design capabilities to the electronic product development process so you can verify if nets have been assigned correctly, connectors are oriented correctly, plug-in boards fit together, and whether all the connected boards fit into the enclosure. This helps minimize any costly late development stage mistakes or time to market delays.

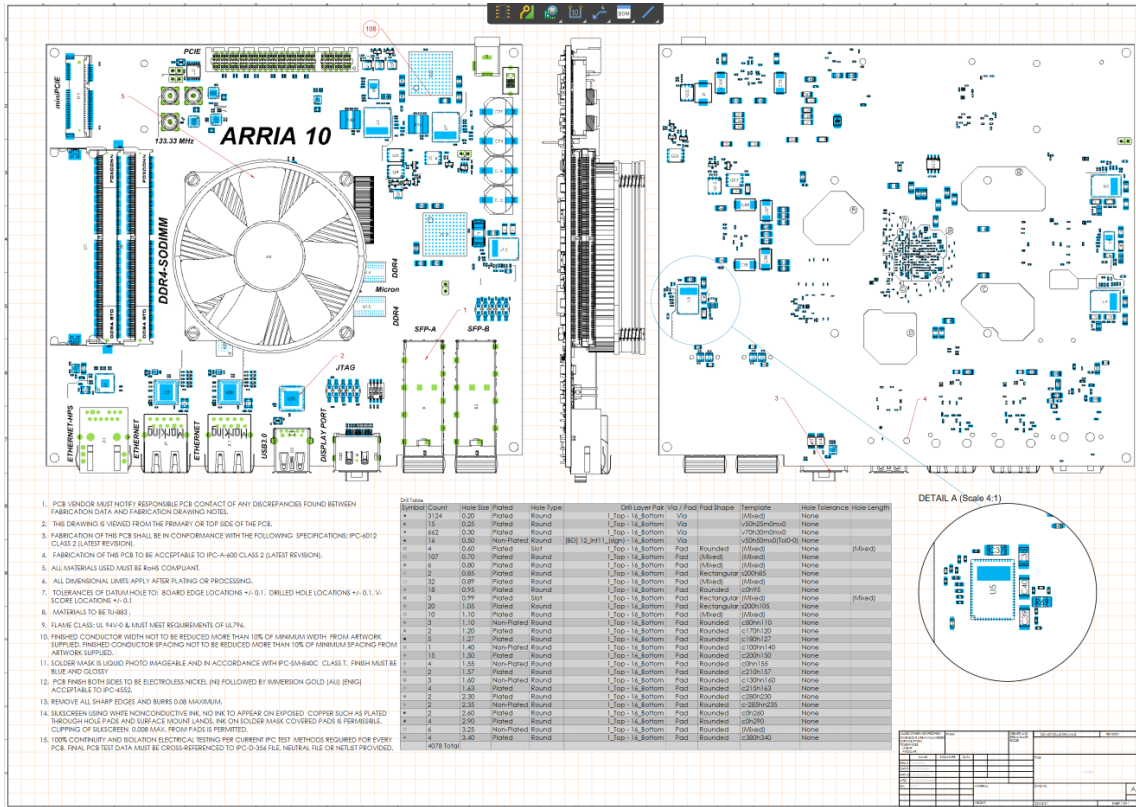


Multi-board Assembly with STEP enclosure imported

ALTIUM EVALUATION GUIDE FOR PADS® USERS

STREAMLINE ASSEMBLY AND FABRICATION IN ALTIUM DESIGNER

A powerful automated PCB design documentation tool is available directly within Altium Designer: Draftsman®. It automates the creation of tables, PCB design views, layer stack legend, and other details. The drawing document is linked to the source PCB document so they are always accurate and in sync.



Automated Fabrication and Assembly Documentation

FOR A FULL EVALUATION

Get a 30-day full featured evaluation license at <http://www.altium.com/free-trial>.

Altium is continually adding more powerful and differentiating placement and routing features to Altium Designer. These features will increase your productivity, streamline your core PCB design tasks, and reduce your time to market. See the Altium Designer product website for examples of the many benefits you can reap and the new features you can explore: <http://www.altium.com/altium-designer/whats-new>.

ABOUT ALTIUM

Altium LLC (ASX: ALU) is a multinational software corporation headquartered in San Diego, California, that focuses on electronics design systems for 3D PCB design and embedded system development. Altium products are found everywhere from world leading electronic design teams to the grassroots electronic design community.

With a unique range of technologies Altium helps organisations and design communities to innovate, collaborate and create connected products while remaining on-time and on-budget. Products provided are ACTIVEBOM®, ActiveRoute®, Altium Designer®, Altium Vault®, Altium NEXUS™, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, EE Concierge™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, the TASKING® range of embedded software compilers and Upverter™.

Founded in 1985, Altium has offices worldwide, with US locations in San Diego, Boston and New York City, European locations in Karlsruhe, Amersfoort, Kiev, Munich, Markelo and Zug and Asia Pacific locations in Shanghai, Tokyo and Sydney. For more information, visit www.altium.com. You can also follow and engage with Altium via Facebook, Twitter, LinkedIn and YouTube.