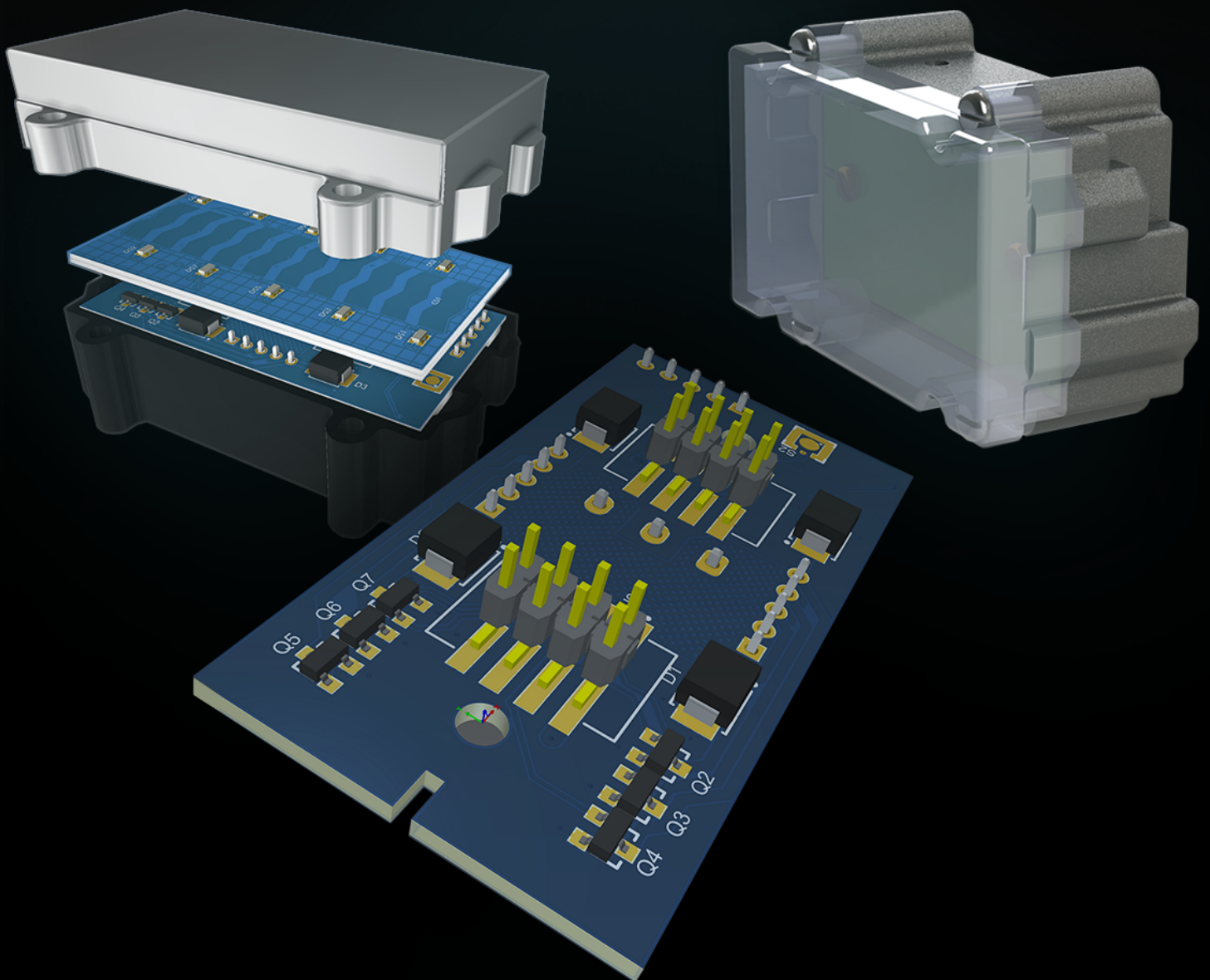


Altium[®]

ALTIUM DESIGNER EVALUATION GUIDE FOR AUTODESK EAGLE™ USERS



WHY CHANGE & WHY NOW?

Are you falling short of meeting minimum design specifications, or missing release dates and product cost targets? Do you have the expertise to design the perfect board, but are unable to achieve your “feature elegance” targets due to the limitations of your design environment? Have you decided that it’s time to change?

With the rapidly increasing complexity of modern products, all with larger circuits in smaller packages, you can no longer accept the inability to meet your goals as normal and permissible. You require a complete solution that offers feature-rich products, state-of-the-art automation technology, intelligent analysis tools, and efficient time-tested collaboration across the entire ECAD-MCAD design process. Now is the time to take a closer look at Altium Designer.

OVERVIEW OF EAGLE™ PRODUCTS

Autodesk sells its PCB Design tool (Eagle) in three different stages of expansion:

Free	Standard	Premium
PCB Schematic & Layout		
Autorouter		
2 Schematic Sheets	99 Schematic Sheets	999 Schematic Sheets
2 Signal or Plane Layers	4 Signal or Plane Layers	16 Signal or Plane Layers
80 cm ² (12.4 in ²) board area	160 cm ² (24.8 in ²) board area	Unlimited board area
Free after online activation Secure and Centralized Design Data	Monthly/Yearly Lease	Monthly/Yearly Lease

WHY ALTIUM DESIGNER?

Your electronic designs demand the highest grade of efficiency and performance. When your productivity is measured against immovable deadlines, precise layouts, accurate documentation and exact fittings, you can't afford to not invest in a complete PCB design platform. Altium Designer has everything you need to meet your design challenges and a proven track record of delivering innovative, differentiating features in predictable and reliable releases.

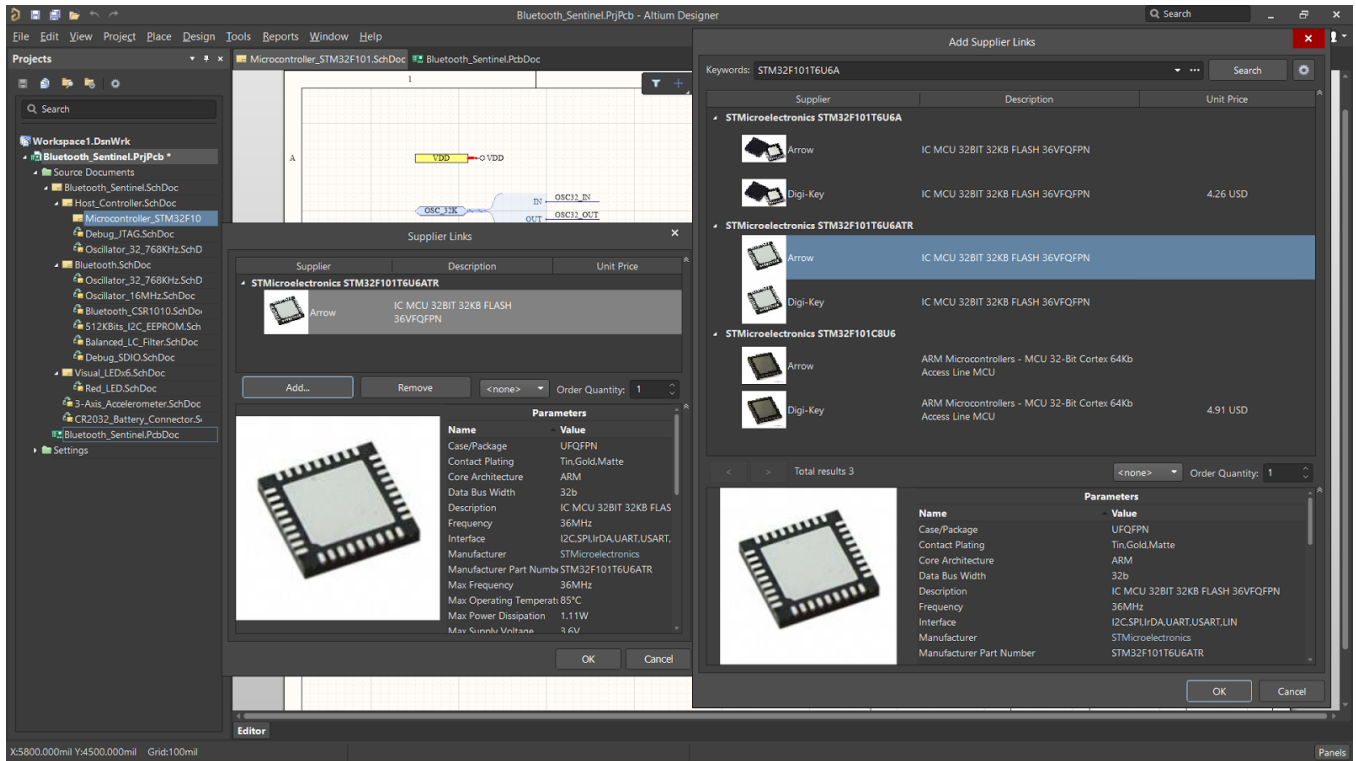
OVERVIEW OF ALTIUM DESIGNER

A cohesive environment for your design, data and release management process.

Altium Designer provides a cohesive PCB design environment that is easy to learn and use, following the Windows standard behavior. You have the features you need to make design decisions early, perform tasks efficiently and implement checks and balances throughout your design process. Altium Designer also interfaces seamlessly to third-party analysis, synthesis and 3D mechanical software.

ALTIUM DESIGNER EVALUATION GUIDE FOR AUTODESK EAGLE™ USERS

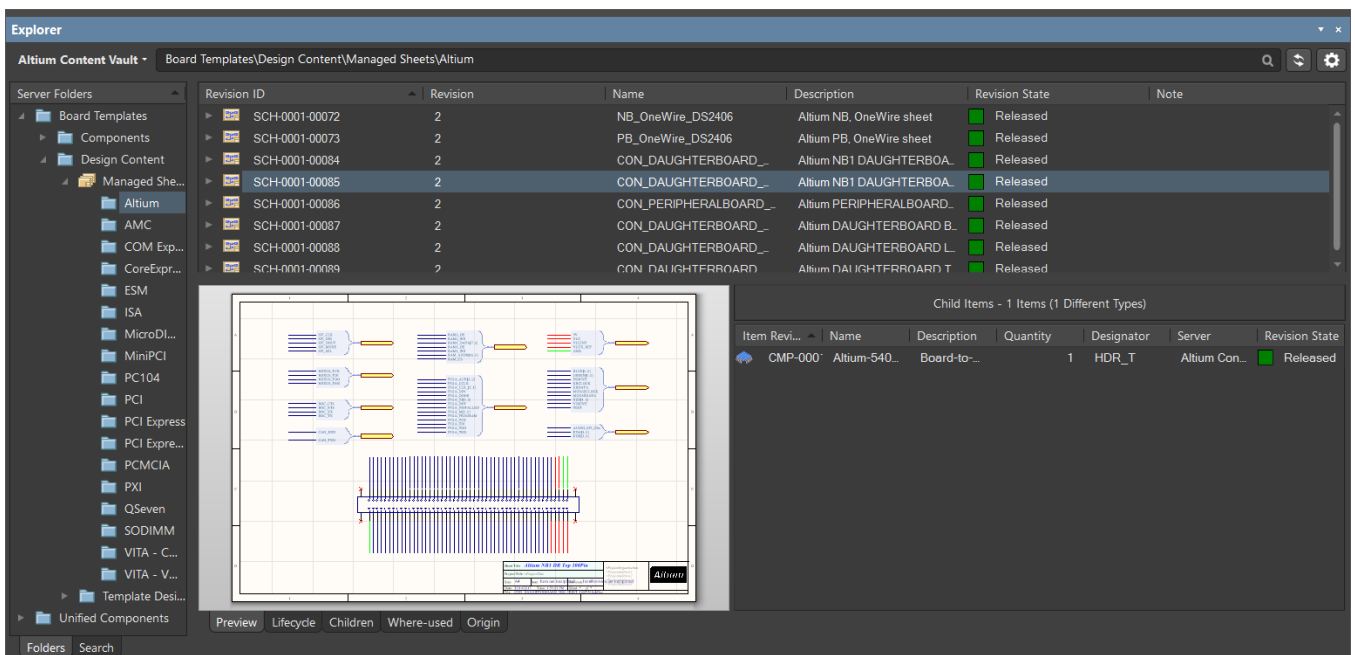
With Altium Designer, you're equipped with everything you need to solve even the most complex design challenges, including:



Supply Chain Integration

Advanced Supply Chain Management

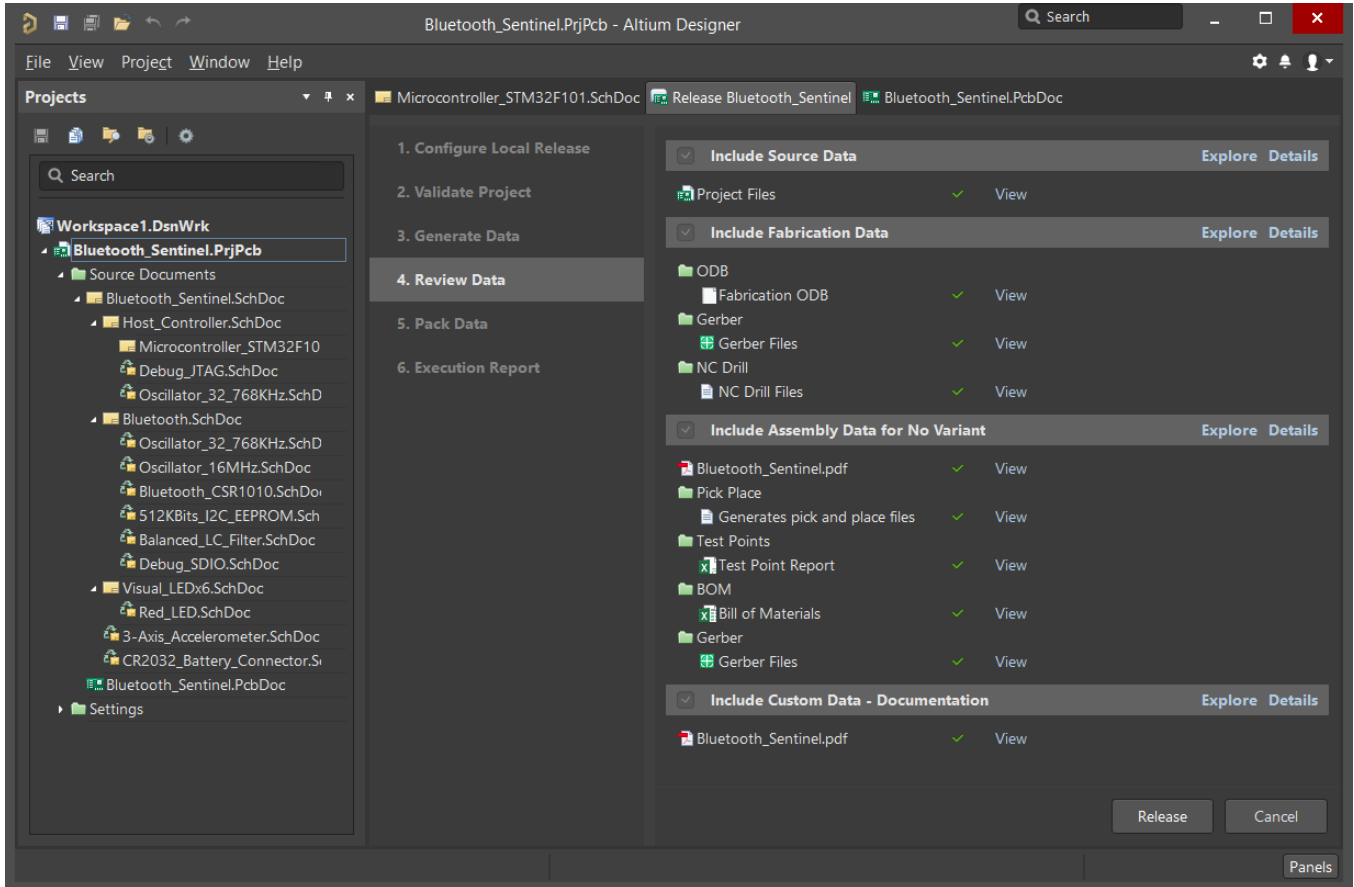
Always ensure you make intelligent part selections with real-time pricing and availability data from your most trusted and reliable suppliers. Reduce the likelihood of costly and time-consuming design re-spins with complete visibility over supply chain data early in the design process.



Data Management

Secure and Centralized Design Data

Store all of your valuable design assets in a securely centralized location accessible by your entire design team. Ensure that your team is working with trusted design data with centralized library management tools.



Release Management

Flexible Release Management Tools

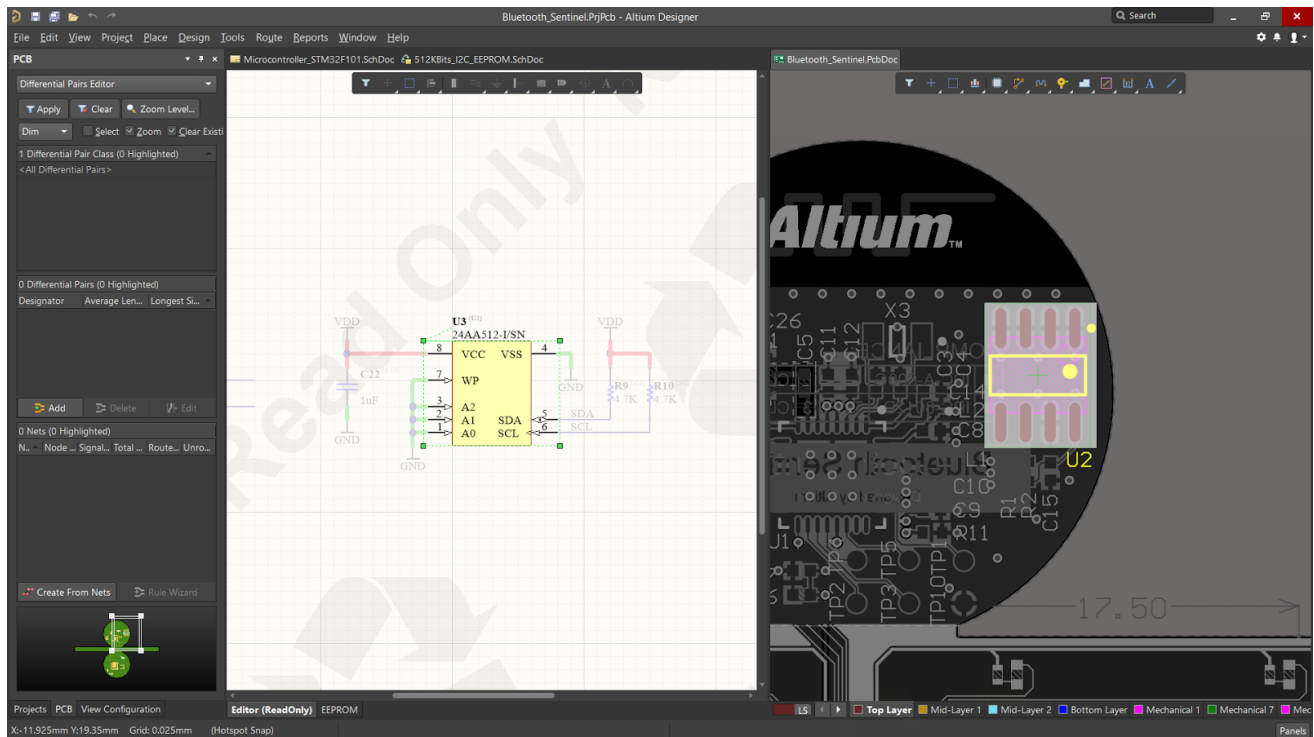
Control the consistency and reliability of your project with the ability to search and release accurate design data. Accelerate your design process by eliminating the need to reproduce data and documentation.

Seamless ECAD/MCAD Collaboration

Easily collaborate with your mechanical team in real-time with automatic data synchronization and tracking. Get your board manufactured right the first time with powerful Native3D™ visualizations and clearance checking.

ALL WITHIN ONE, MODERN USER INTERFACE

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



Unified UI, show schematic/layout side-by-side

LIBRARY AND COMPONENT MANAGEMENT IN EAGLE

Eagle offers a file-based library which incorporates common file access limitations to share library content with other team members. There's also a feature that allows you to add design links to components to maintain the synchronization across your schematic and PCB layouts.

LIBRARY AND COMPONENT MANAGEMENT IN ALTIUM DESIGNER

Altium is the leader in providing a complete solution for the PCB design, development and production process. One of the fundamental aspects of this process includes a close connection to supply chain and real-time component management. In Altium Designer, there are different library concepts available. Starting with easy file-based structures to database-driven libraries supporting collaboration as well as managed libraries which give you the means of maintaining lifecycles and revisions. Whichever option you choose the setup and use is fast and easy.

ALTIUM DESIGNER EVALUATION GUIDE FOR AUTODESK EAGLE™ USERS

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. Altium Designer 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 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.

Item Details						Primary Solution				
Line #	Name	Description	Designator	Quantity	Revision Status	Manufacturer 1	Manufacturer PartNo 1	Manufacturer Lifecycle 1	Supplier 1	
1	1	74754-0101	SFP+ Cage, Single...	B1, B2	2	Out of date	Molex	74754-0101	Volume Production	Mouser
2	2	CGB2A1X5R1E10...	Chip Capacitor, 1...	C1, C4...C7, C23...	19	Up to date	TDK	CGB2A1X5R1E105K0...	Volume Production	Avnet
3	3	ECJ-0EC1H680J	Chip Capacitor, 68...	C106	1	Up to date	Panasonic	ECJ-0EC1H680J	Obsolete	Digi-Key
4	4	C3216X5R1A476...	Chip Capacitor, 47...	C11, C12, C18... C...	24	Up to date	TDK	C3216X5R1A476M16...	Volume Production	Avnet
5	5	GRM155R61A224...	Chip Capacitor, 22...	C13, C15, C16, C2...	27	Up to date	Murata	GRM155R61A224KE1...	Volume Production	Arrow
6	6	VJ0402Y153JXQP...	Chip Capacitor, 15...	C14, C24, C33, C4...	7	Up to date	Vishay Vitramon	VJ0402Y153JXQPW1...	Volume Production	Mouser
7	7	CGJ2B2X7R1E22...	Chip Capacitor, 2...	C107	1	Out of date	TDK	CGJ2B2X7R1E222K0...	Volume Production	Arrow
8	8	GRM1535C1H560...	Chip Capacitor, 56...	C108	1	Up to date	Murata	GRM1535C1H560J...	End of Life	Mouser
9	9	ECJ-0EC1H270J	Chip Capacitor, 27...	C17, C51	2	Up to date	Panasonic	ECJ-0EC1H270J	Obsolete	Avnet
10	10	GRM155R71H103...	Chip Capacitor, 10...	C2, C3, C67, C70...	20	Up to date	Murata	GRM155R71H103KA8...	Volume Production	Mouser
11	11	ECJ-0EC1H150J	Chip Capacitor, 15...	C48, C64	2	Up to date	Panasonic	ECJ-0EC1H150J	Obsolete	Digi-Key
12	12	C1005X7R1E104M	Chip Capacitor, 0...	C68, C69, C79, C1...	132	Up to date	TDK	C1005X7R1E104M	Volume Production	Mouser
13	13	EEH-2A1E331P	Aluminum ElectroL...	C74...C77	4	Up to date	Panasonic	EEH-2A1E331P	Volume Production	Avnet
14	14	C1005C0G1H220...	Chip Capacitor, 22...	C8, C27, C36	3	Up to date	TDK	C1005C0G1H220J05...	Volume Production	Mouser
15	15	6TPF330M9L	Tantalum Capacito...	C80, C81, C87, C8...	6	Up to date	Panasonic	6TPF330M9L	Volume Production	Farnell
16	16	GRM31CR60J107...	Chip Capacitor, 10...	C82...C84, C89...	35	Up to date	Murata	GRM31CR60J107ME...	Volume Production	Farnell
17	17	GRM31CR61E226...	Chip Capacitor, 22...	C9, C10, C28, C29...	22	Up to date	Murata	GRM31CR61E226KE...	Volume Production	Arrow
18	18	VJ0402Y182JXJC...	Chip Capacitor, 1...	C94	1	Up to date	Vishay Vitramon	VJ0402Y182JXJCW1...	Volume Production	Mouser
19	19	GRM155R71E473...	Chip Capacitor, 47...	C110	1	Up to date	Murata	GRM155R71E473KA8...	Volume Production	Avnet
20	20	GRM155R71E223...	Chip Capacitor, 22...	C111	1	Up to date	Murata	GRM155R71E223KA6...	Volume Production	Avnet
21	21	GRM155R71H681...	Chip Capacitor, 68...	C117	1	Up to date	Murata	GRM155R71H681KA0...	Volume Production	Arrow
22	22	GRM033R60J104...	Chip Capacitor, 10...	C118...C130, C14...	226	Up to date	Murata	GRM033R60J104KE1...	Volume Production	Arrow
23	23	GRM188R61A225...	Chip Capacitor, 2...	C297, C329, C355...	10	Up to date	Murata	GRM188R61A225KE3...	Unknown	Arrow
24	24	GRM155R71H472...	Chip Capacitor, 4...	C308	1	Up to date	Murata	GRM155R71H472KA0...	Volume Production	Avnet
25	25	C2012X5R1C106K...	Chip Capacitor, 10...	C330, C356, C370...	10	Up to date	TDK	C2012X5R1C106K08...	Volume Production	Digikey

130 of 130 lines visible | Current variant: 4GB

Add Solution

Manufacturer Part: Vishay Vitramon VJ0402Y153JXQP...
Stock: 28k \$0.14 min
Volume Production
Multilayer Ceramic Capacitors MLCC - SMD/SMT 0402 0.01...

Favorite Suppliers

Mouser 1
77-VJ0402Y153JXQPBC
Stock: 28035 Order qty: 7
Unit: \$0.14 Order: \$0.98
1 @ \$0.14 10 @ \$0.14
50 @ \$0.12 Show more

Avnet 2
VJ0402Y153JXQPW1BC
Stock: 0 Order qty: 50000
Unit: \$0.0083 Order: \$415.00
50000 @ \$0.0083 100000 @ \$0.00805
200000 @ \$0.00772 Show more

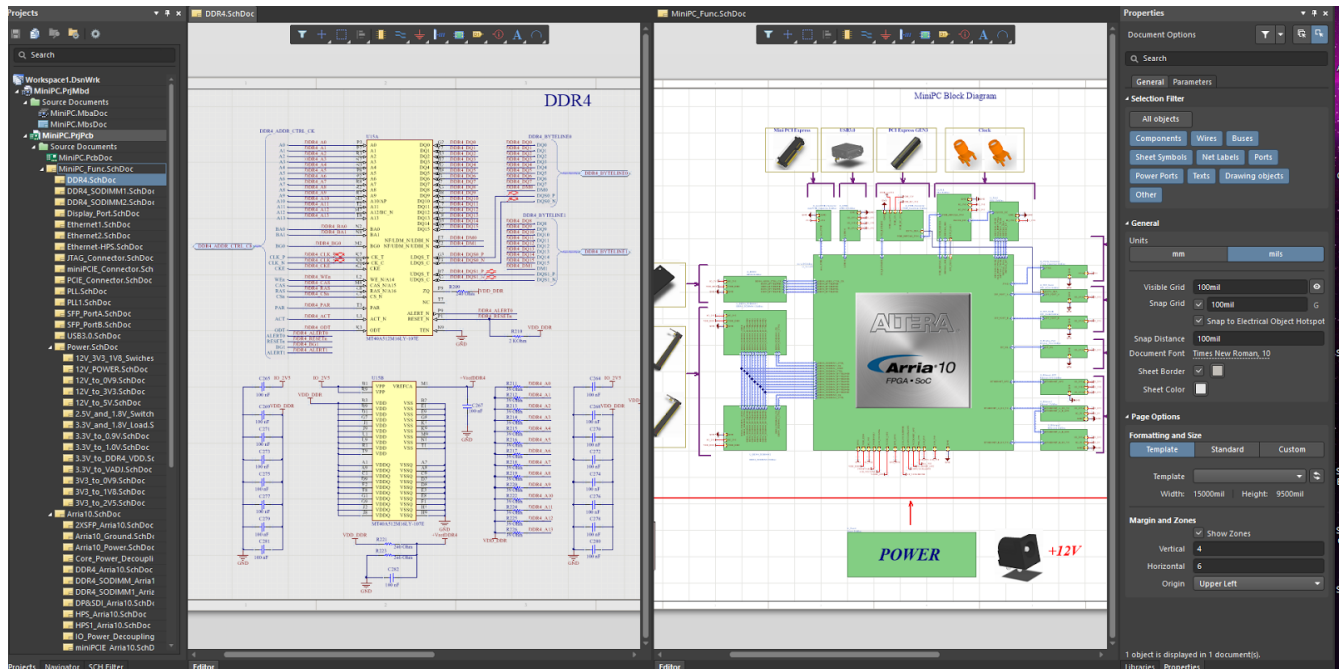
ActiveBOM - Real-Time Cost Estimation & Part Availability

EAGLE SCHEMATIC CAPTURE

Eagle offers a nice schematic entry that provides the most commonly needed features. However, there are some obvious limitations when it comes to bottom-up development of hierarchical designs, controlled reuse and snapping outside the standard grid. With the user-interface being quite different from the Windows world, there can be a learning curve that could result in one or two gray hairs.

ALTIUM DESIGNER SCHEMATIC CAPTURE

Altium Designer schematic capture technology has long been recognized as a technology differentiator. Engineers and designers will find that Altium Designer schematic features are easy to learn and quickly improve productivity on all designs ranging between relatively simple single-sheet schematics to complex multi-sheet hierarchical projects. Here's why:



Modern and powerful Unified Schematic Editor

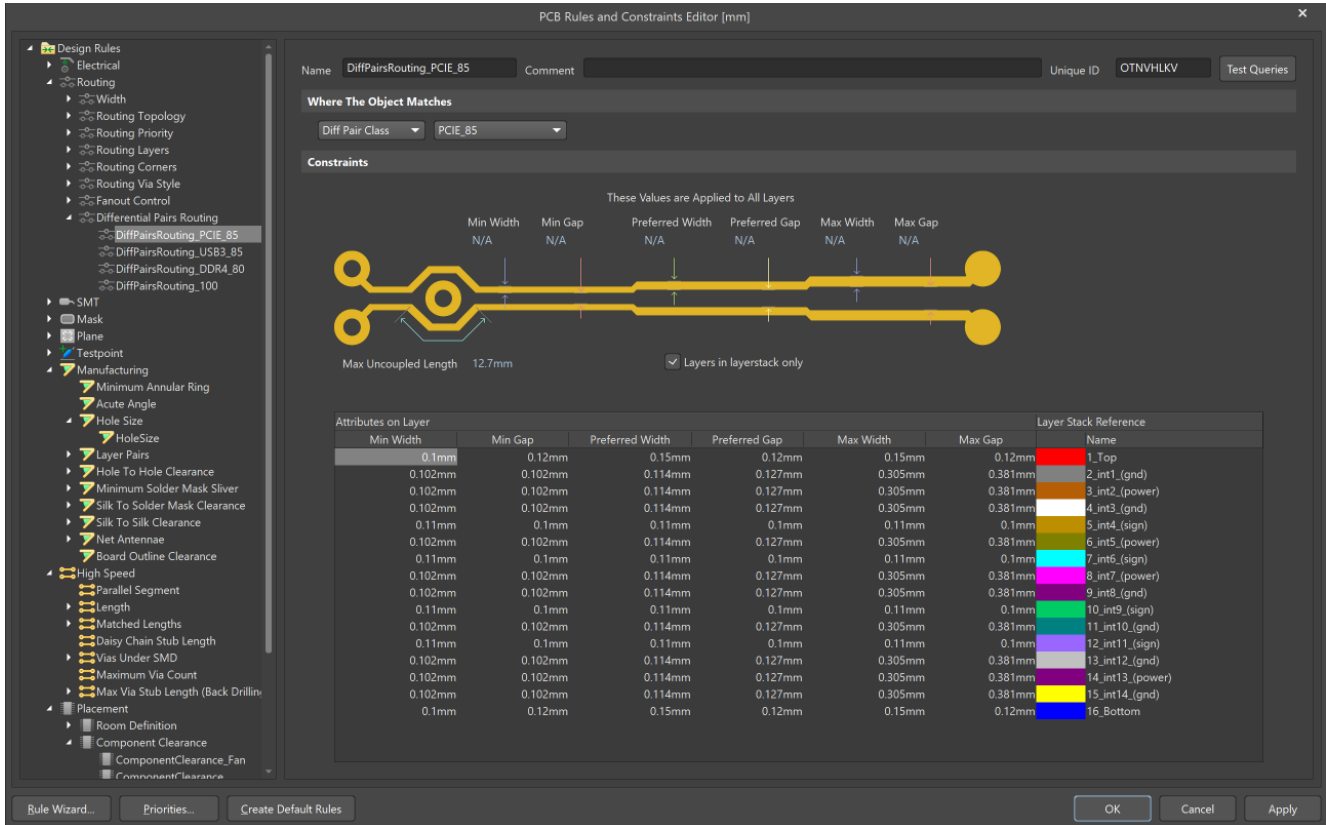
- Starting a schematic is fast and easy with intuitive dialogs, e.g., editing workspace and establishing sheet design, parameters, preferences and associated documents. Immediately manage versions.
- Easily set component classes, net classes, and placement rooms in an intuitive environment that boosts productivity.
- Quickly select and place qualified components from integrated libraries with real-time links to component suppliers.
- Leverage powerful ECO feature to 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 differences.
- Verify your circuits using the built-in XSPICE/SPICE3F5 (PSpice® model compatible) circuit simulator without leaving the environment.

RULES AND CONSTRAINTS IN EAGLE

In Eagle, you'll find a basic, matrix-driven way to configure minimum clearances and width definitions for elements and net classes. More complex rulesets that take different Layers or specific areas on the PCB into account cannot be defined. This also includes length definition sets for partial nets, which are needed in high-speed designs like DDR3 or 4 technology, or differential pairs with serial terminations.

RULES AND CONSTRAINTS IN ALTIUM DESIGNER

True to its cohesive and easy-to-use nature, Altium Designer provides a streamlined PCB Rules and Constraints Editor, with more control over the entire design process:



Constraint Driven PCB Design with Design Rule Checking

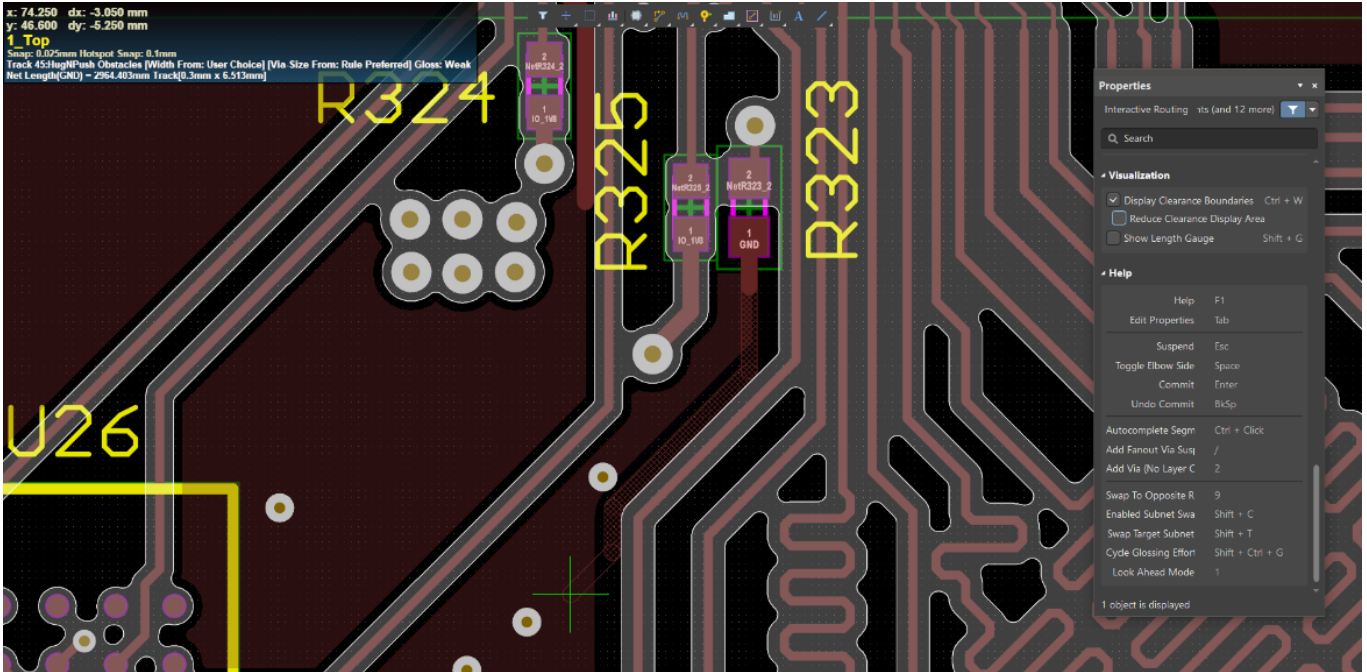
- Easily browse, create, prioritize, define the scope, edit, duplicate and delete rules all in one editor.
- Evaluate your rules in a table-based summary that provides straightforward review.
- Define multiple rules of the same type, but each targeting different objects.
- Decide exactly what a rule's precedence will be and how it will be applied to target objects through a query.
- Write your own, more complex queries using Advanced Query options.
- Create new rules in a step-by-step manner with the New Rule Wizard guiding you along the way.

PLACEMENT AND ROUTING IN EAGLE

Being able to place components solely within the boundaries of a PCB limits the developer when cleaning up the mess of components caused by the automatic, uncontrollable and disorganized schematic-to-PCB synchronization. The idea of sorting and placing components according to their destination task and possibly routing those modules is completely impossible in this way. Even more annoying is making adjustments when most of the routing work has already been done, as routing the last set of tracks often requires pushing other tracks away. An alternative to deleting and redrawing those tracks manually should exist to facilitate the routing process for designers.

PLACEMENT AND ROUTING IN ALTIUM DESIGNER

It's crucial to have an organized and efficient placement of components on your PCB. Altium Designer offers enhanced capabilities to ensure proper component placement and create the most efficient board layout possible:



Routing in Altium Designer

- Dynamically place and drag components that push, avoid, and snap-to alignment with other components on your board layout.
- Easily align multiple components to keep your board layout organized and tidy.
- Mask or filter objects in the workspace to gain more visibility over your board.
- Optimize your routing layers with the Layer Stack Manager giving you full control over all layers.

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.

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 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.

ALTIUM DESIGNER EVALUATION GUIDE FOR AUTODESK EAGLE™ USERS

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.

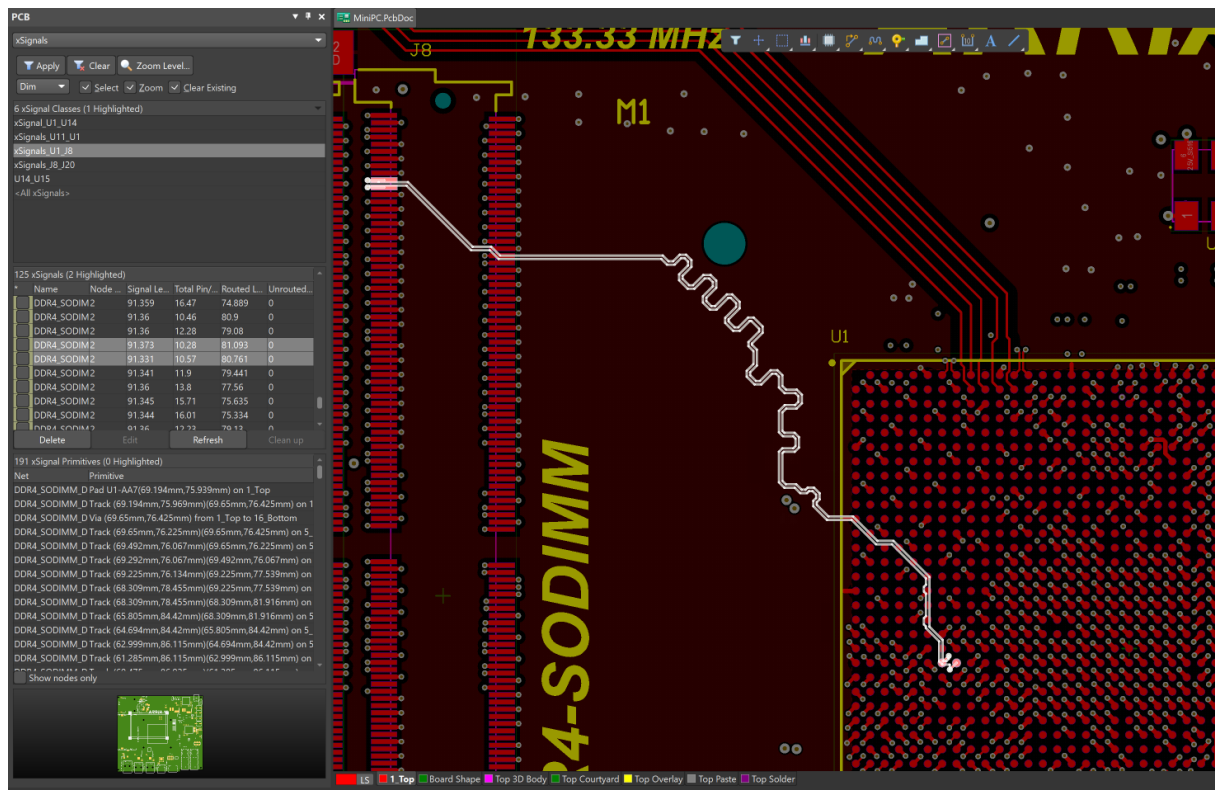


ActiveRoute With Length Tuning - Before and After (25 Seconds later!)

OTHER DIFFERENTIATING PLACEMENT AND ROUTING FEATURES IN ALTIUM DESIGNER

- Eliminate the stress of your manual routing process with powerful interactive routing modes and an intelligent routing assistant.
- Easily save, share, and reuse your most trusted design assets with smart, copy-and-paste managed schematic sheets and component library templates.
- Gain even more control over your clearance checking with enhanced test point clearance checks between test points, through-hole pads, and inter-test point spacing.
- Get even more precise with your solder mask expansions with user-definable expansion options from hole edge or pad edge.
- Intelligently route your rigid-flex board layout in Native 3D, then visualize your work of engineering art.

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.



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

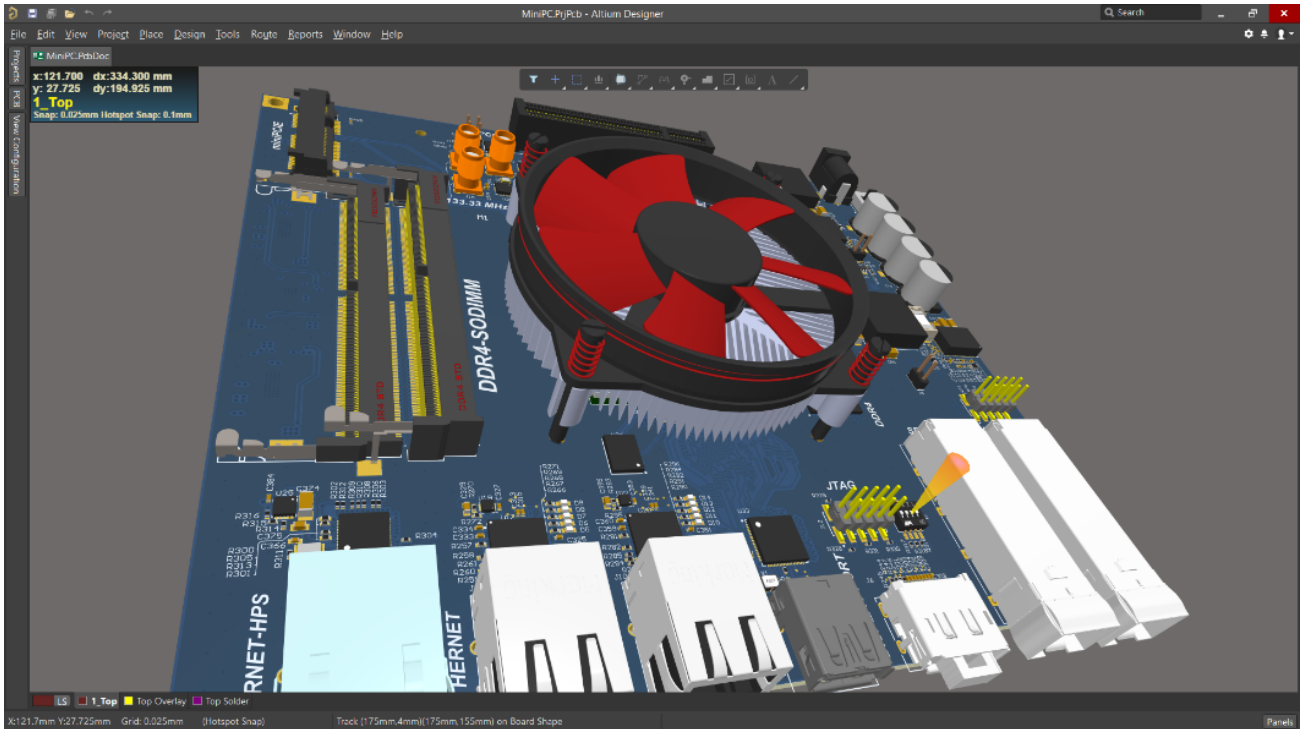
See the Altium Designer product website for examples of the many benefits you can reap and the new features you can explore: <http://www.altium.com/altium-designer/whats-new>.

DESIGN COLLABORATION IN EAGLE

Mechanical and 3D are not really part of the Eagle environment. Although first steps have been taken - IDF 3D export and scripts are available for generating 3D data - Eagle itself only provides a 2D PCB editor requiring export to Fusion 360™ or other MCAD tools for clearance checking making it slow and iterative to resolve placement issues.

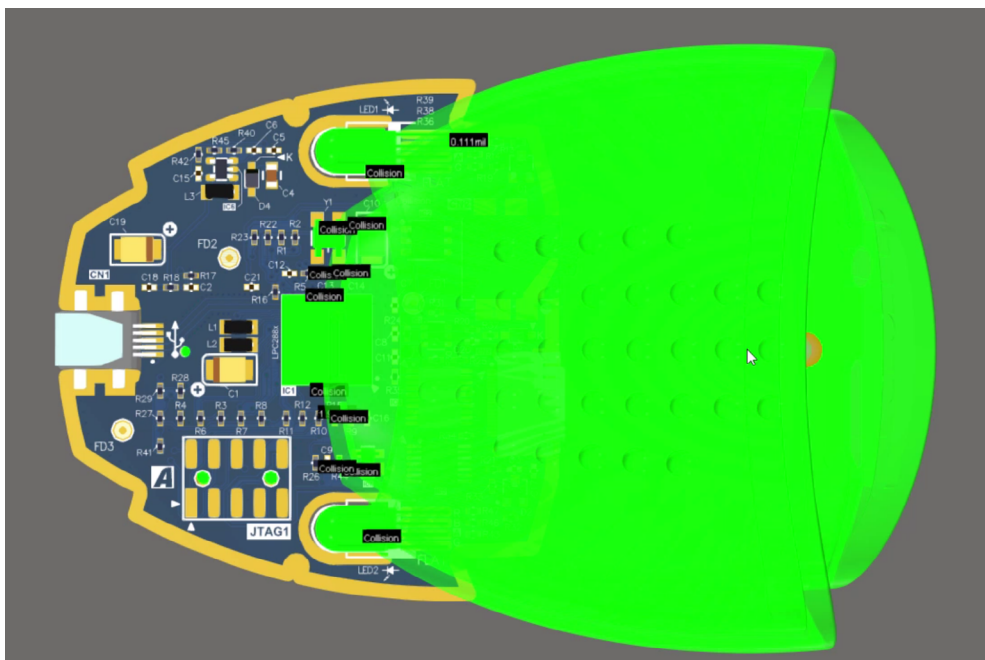
DESIGN COLLABORATION IN ALTIUM DESIGNER

Altium Designer was the first PCB design product to provide true ECAD/MCAD collaboration with 3D editing features powered by the Native3D™ PCB editor engine to visualize, compare, merge, track, and comment on design changes. Electrical and mechanical design data integrates seamlessly into your workflow with real-time visibility into incremental changes. This allows the electrical and mechanical engineering work to be done simultaneously and in parallel.



Native3D™ PCB

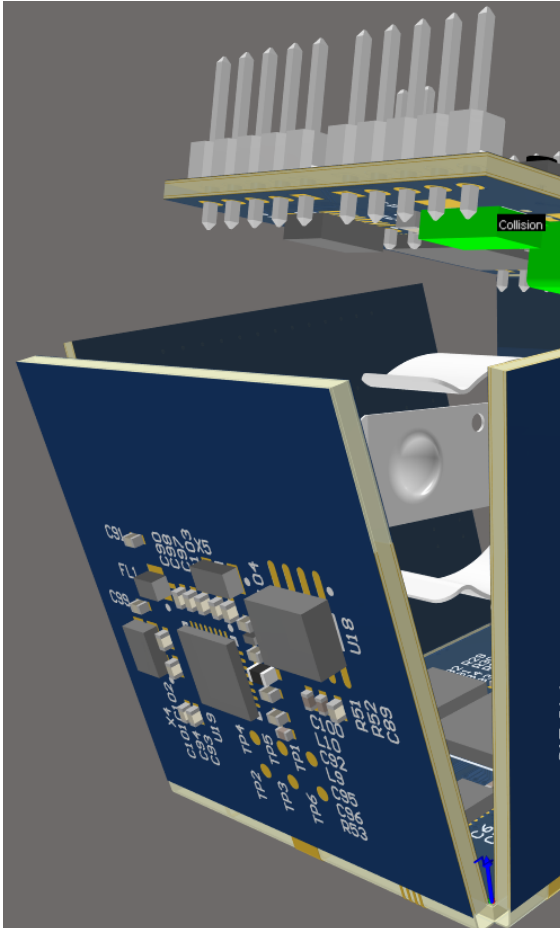
You can visualize exactly how your board will fit its mechanical enclosure and fix collision errors in seconds. You can perform real-time clearance checking for components and mechanical enclosures, and generate folded STEP models.



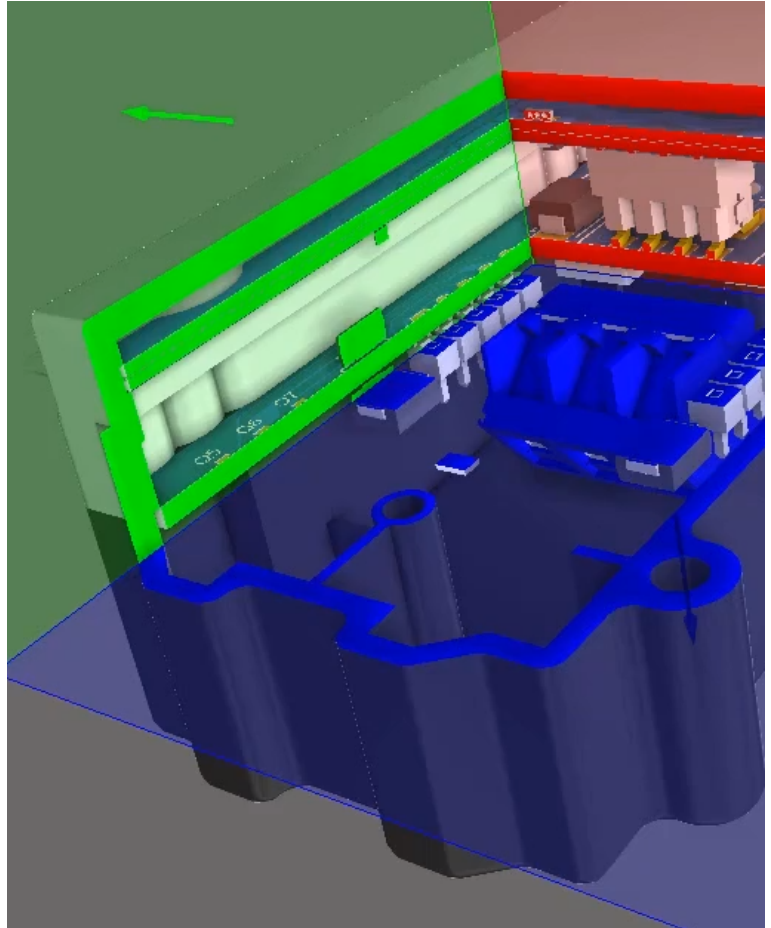
3D Mechanical Collision Detection & Clearance Checking in Real-Time

RIGID-FLEX 3D PCB MODELS

A significant differentiator, Altium Designer supports 3D rigid-flex design. With this feature, you can easily define material selections and intelligently route your rigid-flex board layout, then dynamically visualize your engineering work of art in 3D to make sure that the folding/folded board does not create violations in real time.



Detecting PCB Collisions in 3D as you Work



Enclosure Fit Check with Multi-Board Assembly Section View

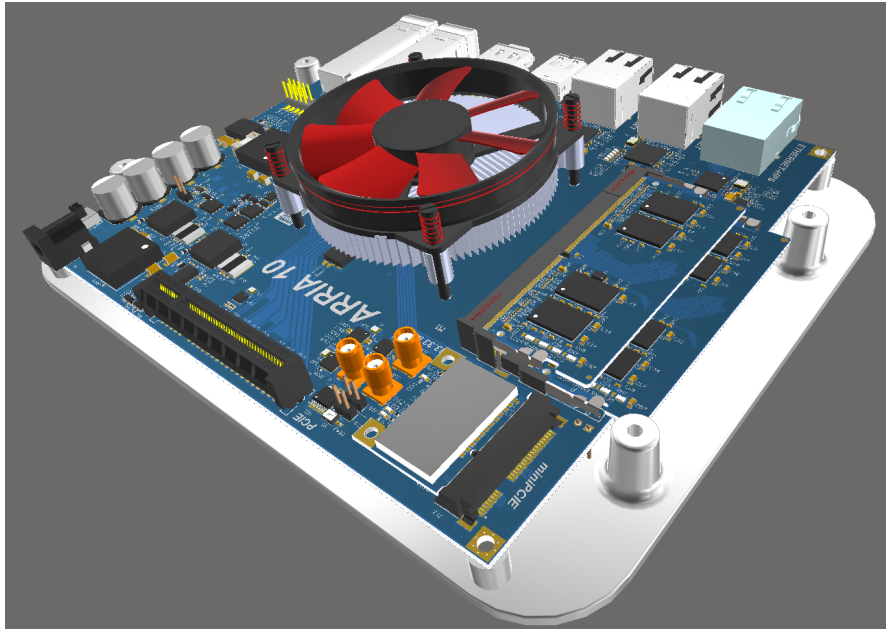
INTERCONNECTED MULTI-BOARD ASSEMBLY

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 design. 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 connections, resolve conflicts, and update system-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.

ALTIUM DESIGNER EVALUATION GUIDE FOR AUTODESK EAGLE™ USERS

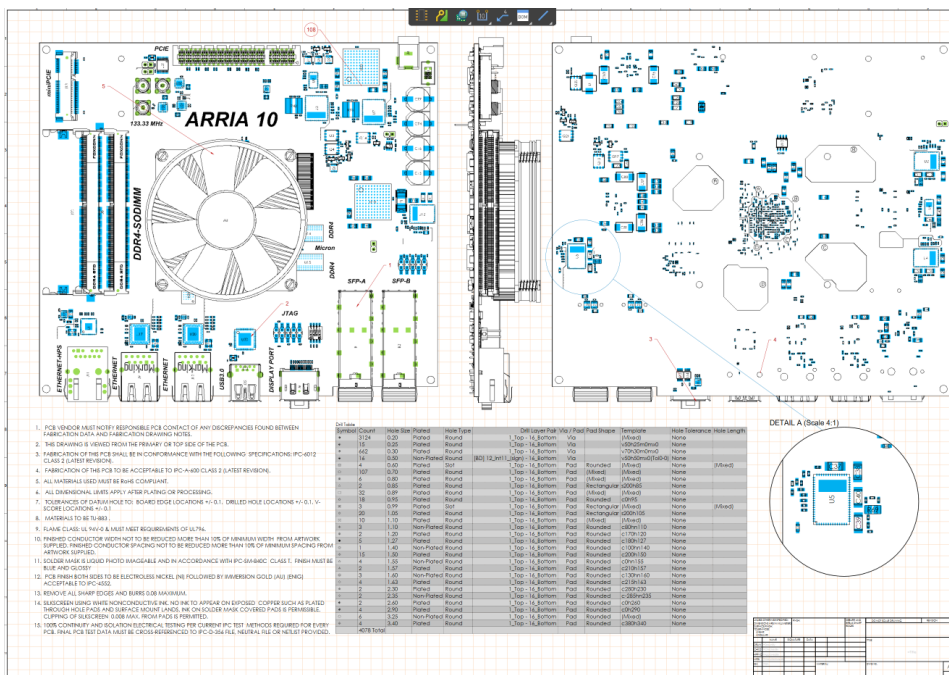
Altium Designer brings you system-level design capabilities to the electronic product development process so you can verify if nets have been assigned correctly, connectors are oriented correctly, plug-in boards fit together, and whether all the connected boards fit into the enclosure. This helps minimize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



Multi-board Assembly Management

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.



Multi-board Assembly Management

ALTIUM EVALUATION GUIDE FOR PADS® USERS

FOR A FULL EVALUATION

Obtain a 15-day full featured evaluation license at <http://www.altium.com/free-trial>.

EAGLE and Fusion 360 are trademarks of Autodesk Inc. and PSpice is a registered trademark of Cadence Design Systems Inc. and Altium claims no rights there within.

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](#).