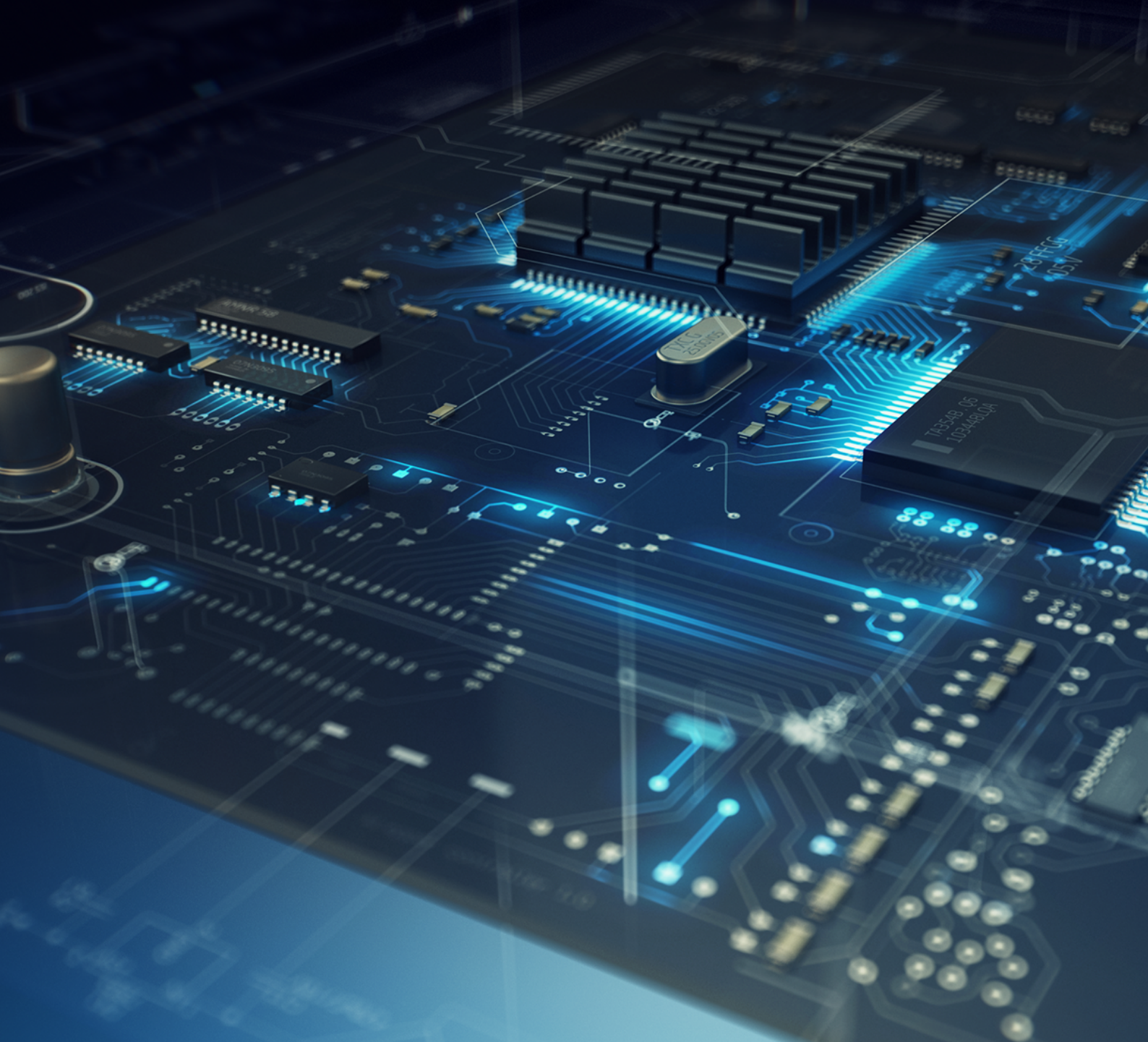
Altium®

ALTIUM 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 collaboration across the entire ECAD-MCAD design process. Now is the time to take a closer look at Altium Designer.

COVERVIEW OF EAGLE™ PRODUCTS

Eagle™ sells its PCB Design tool in three different stages of expansion:

Standard	Premium	Ultimate
PCB Schematic & Layout		
Autorouter		
2 Schematic Sheets	99 Schematic Sheets	
2 Signal Layers	6 Signal Layers	16 Signal Layers
100x80 Routing Area	160x100 Routing Area	4000x4000 Routing Area

For Premium and Ultimate, there is also an LS Version available which does not include the Autorouter.
Along with these commercial options there are also some Non-Commercial Licenses to choose from.

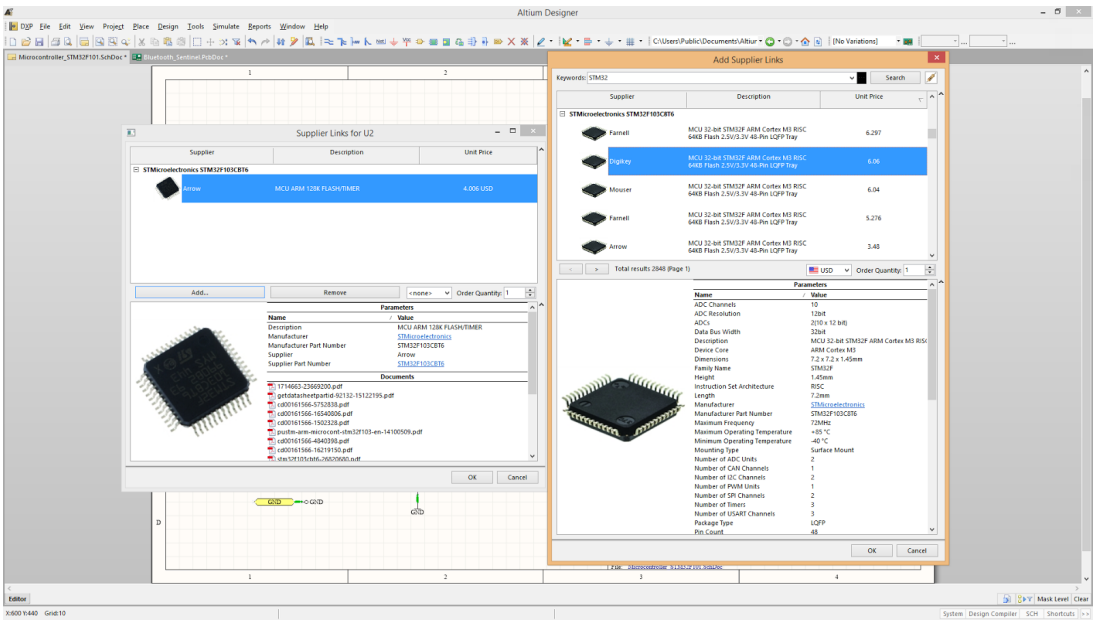
WHY ALTIUM DESIGNER?

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 unified environment for your design, data and release management process.
Altium Designer provides a unified PCB design environment that is easy to learn and use, following the Windows standard behaviour. 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.

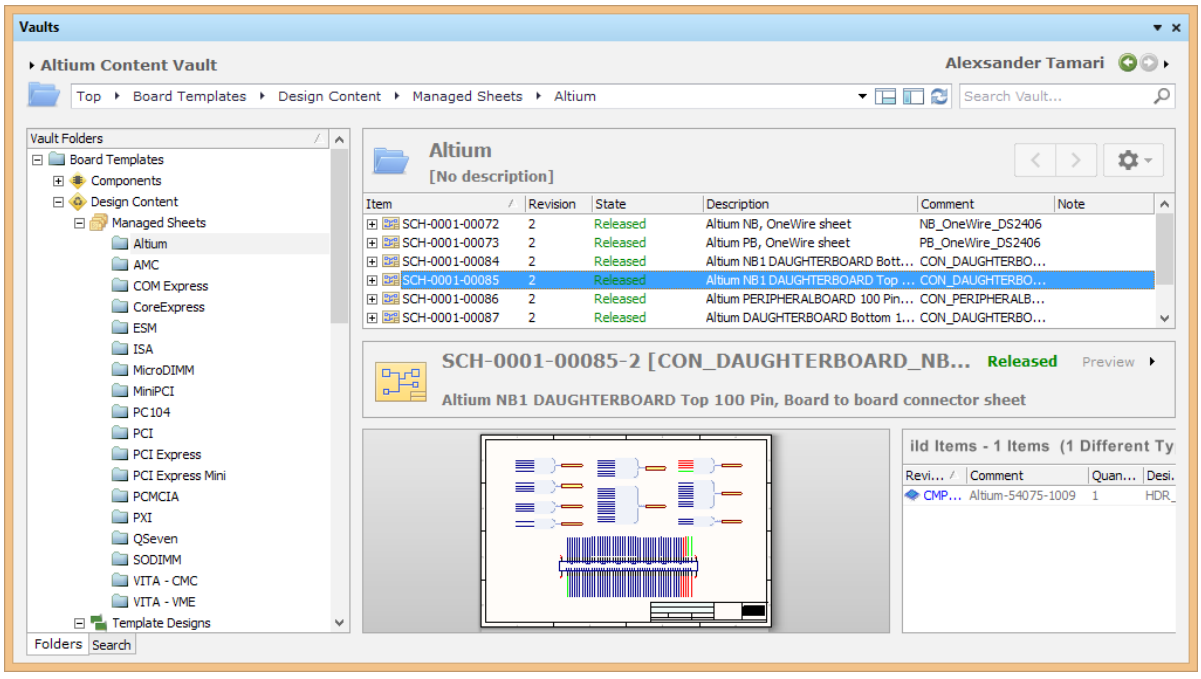
With Altium Designer, you're equipped with everything you need to solve even the most complex design challenges, including:



Supply chain

Advanced Supply Chain Management

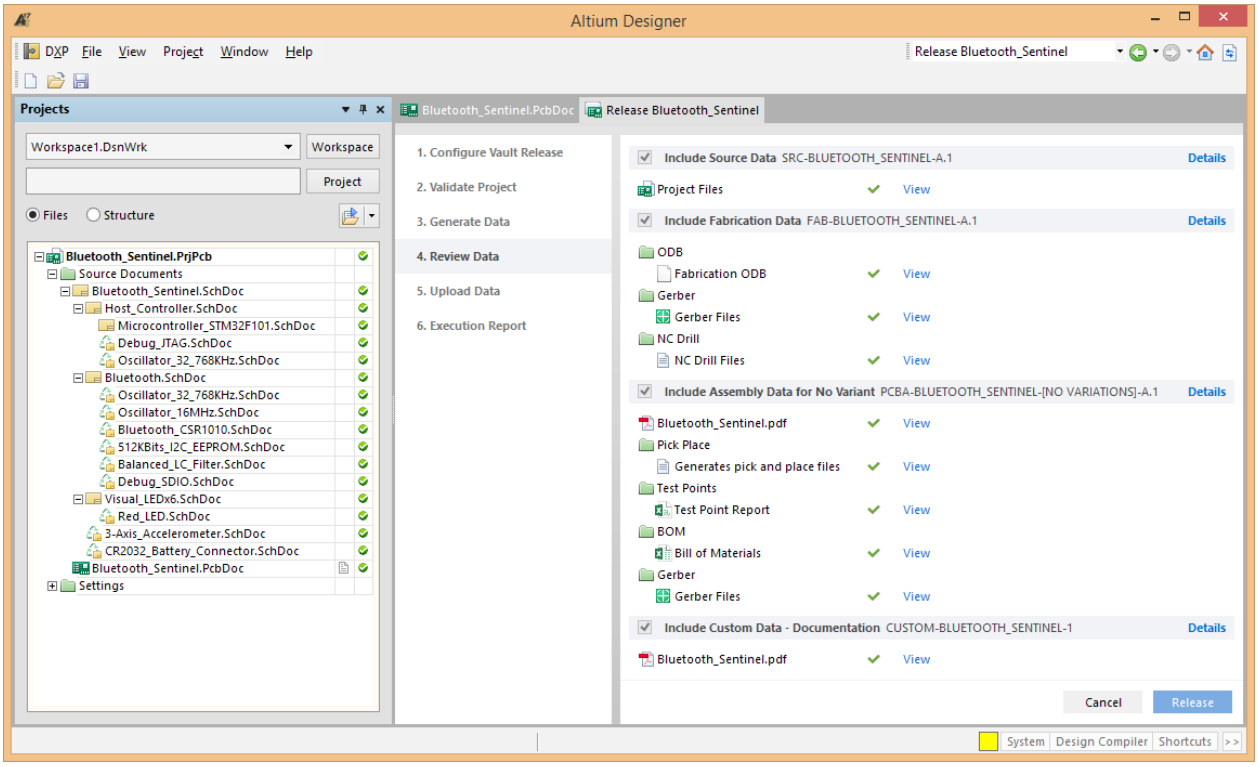
Always ensure you make intelligent part selections with real-time pricing and availability data from your most trusted and reliable suppliers. Reduce the likelihood of costly and time-consuming design re-spins with complete visibility over supply chain data early in the design process.



Data MGMT

Secure and Centralized Design Data

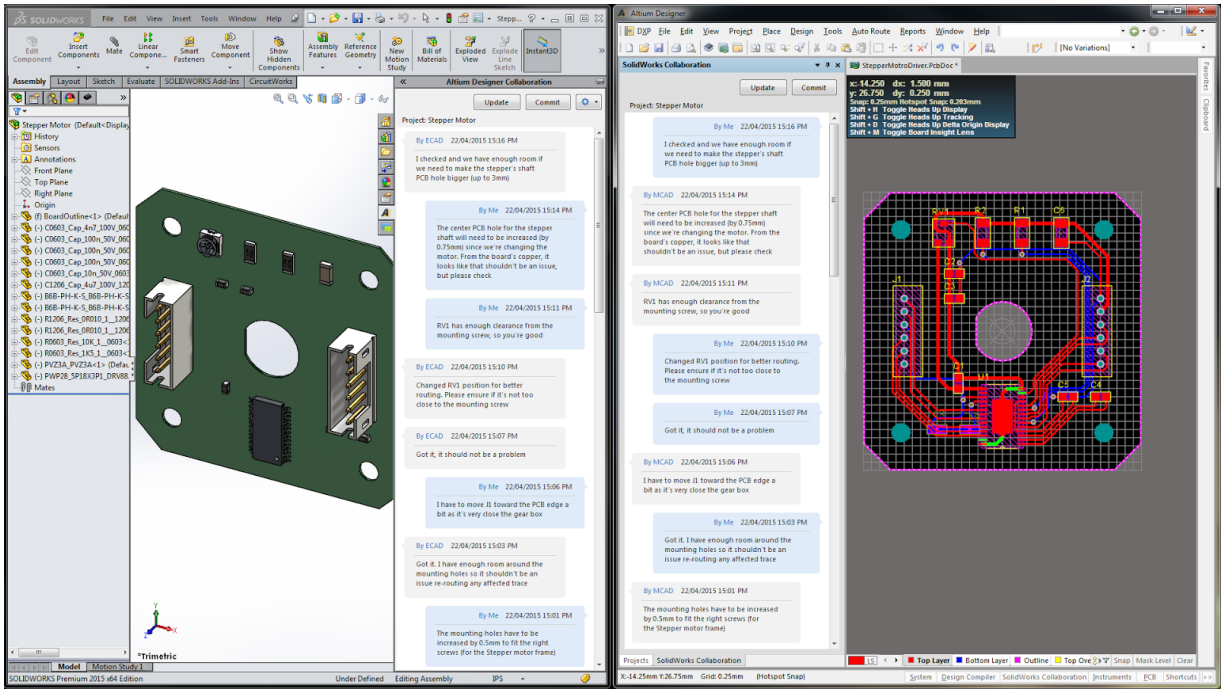
Store all of your valuable design assets in a securely centralized location accessible by your entire design team. Ensure that your team is working with trusted design data with centralized library management tools.



Release MGMT

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.



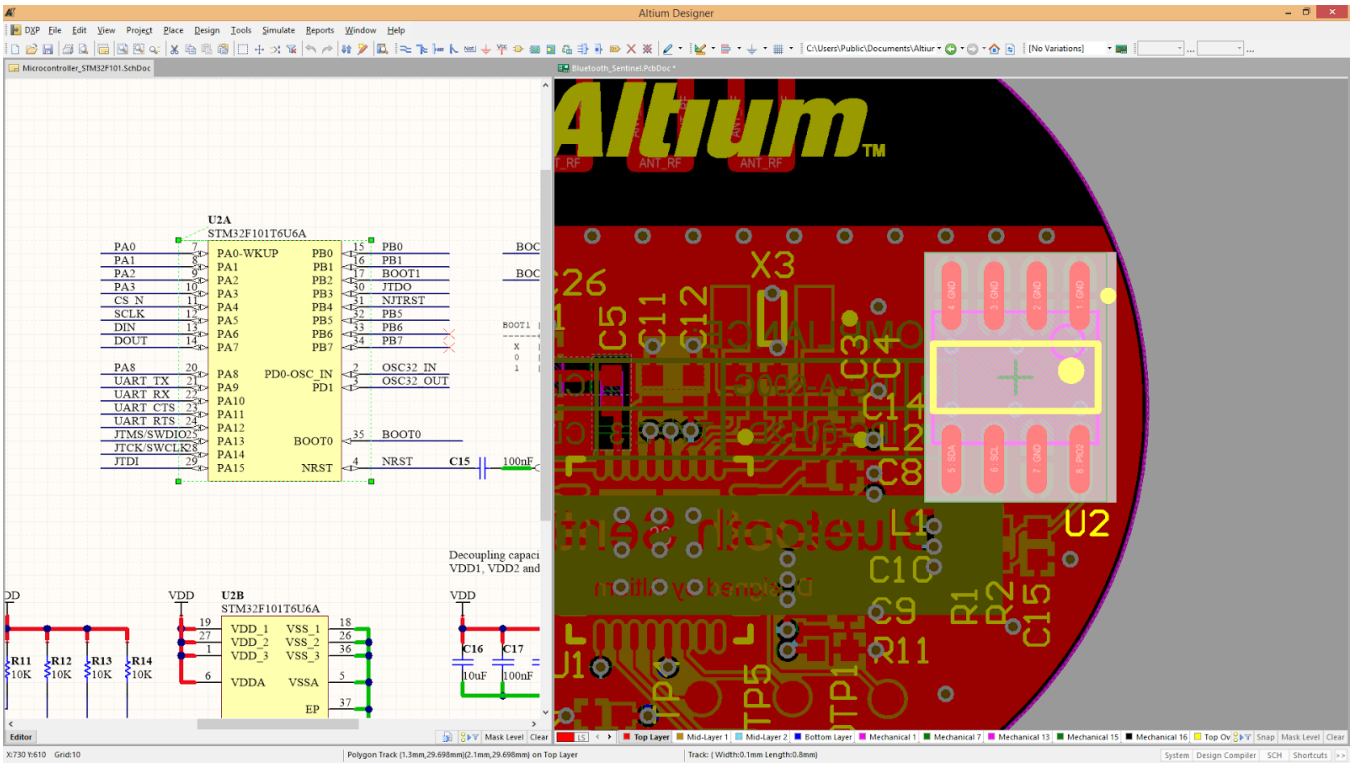
ECAD/MCAD collaboration

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 Native 3D 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.



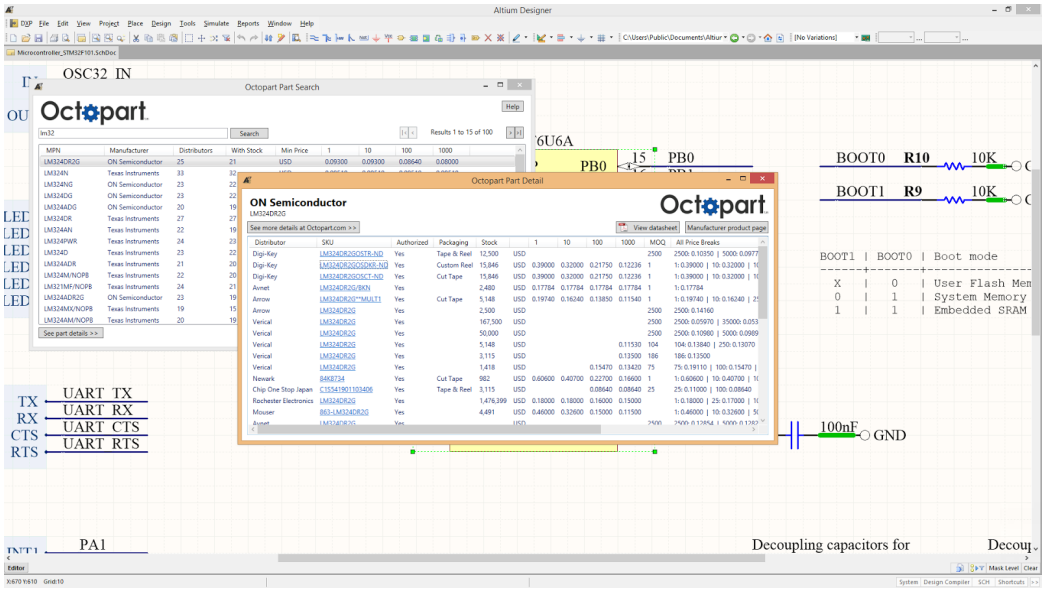
Unified UI, show schematic/layout side-by-side

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. This information is drawn exclusively from Farnell.

LIBRARY AND COMPONENT MANAGEMENT IN ALTIUM DESIGNER

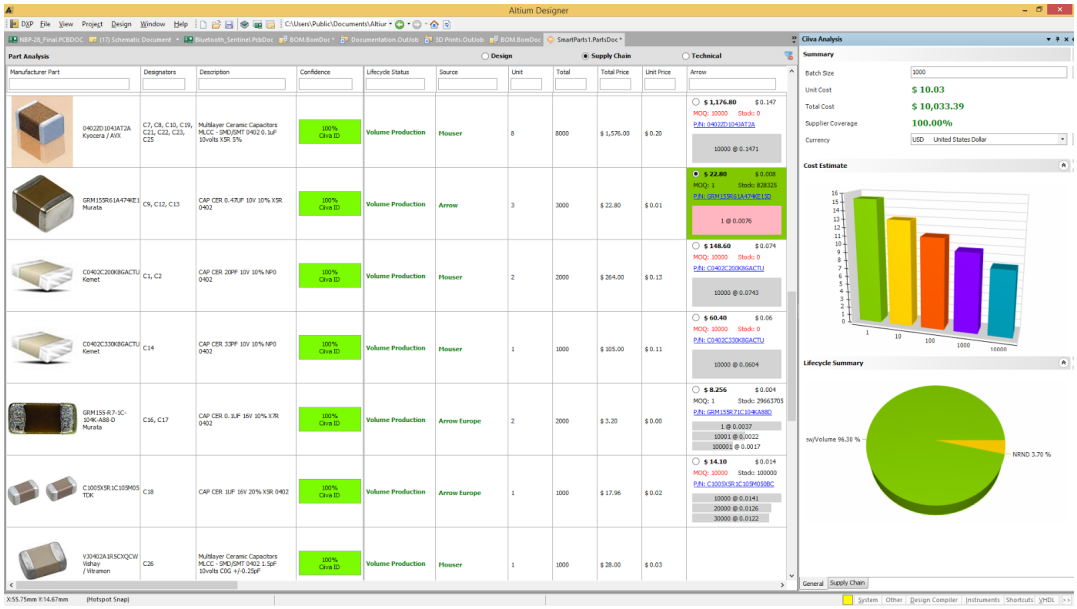
Altium is the leader in providing a complete solution for the PCB design, development and production process. One of the fundamental aspects of this process includes a close connection to supply chain and real-time component management. In Altium Designer, there are different library concepts available. 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. But that's not all:



Octopart search engine

Altium provides **Octopart, a component search engine that provides access to hundreds of distributors, thousands of manufacturers, and millions of parts**.

- Automatically **verify part numbers for every component in your Bill of Materials** and get real-time pricing and availability information with an advanced BOM Tool.
- Make the most of your prototypes in the **Common Parts Library for Prototyping**, a collection of components, tools and equipment for prototyping new connected parts.



Ciiva

Altium also provides **Ciiva, a cloud-based electronic component management system** that brings together supply chain intelligence from sources that traditionally have been fragmented and places this intelligence at your fingertips.

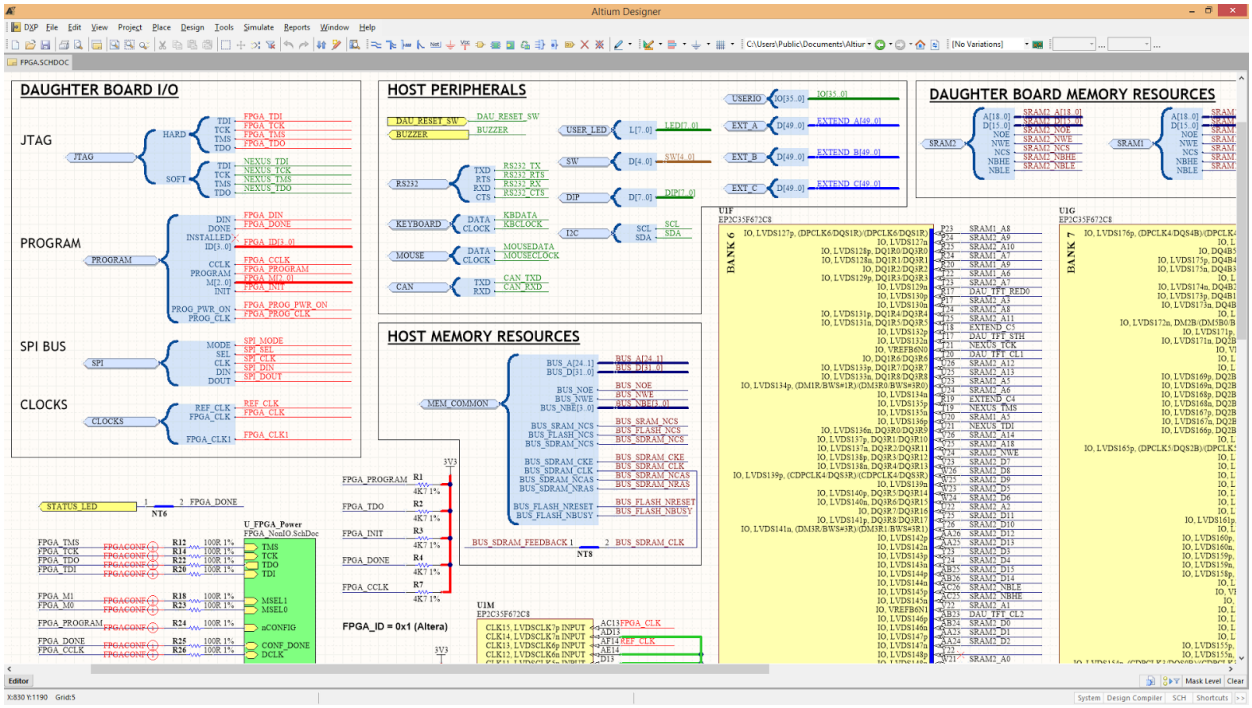
- Easily search for components and datasheets, check lifecycle and alternatives, and compare stock and pricing.
- Automate your Bill of Materials with **a fully-traceable, version-controlled BOM management feature** that links every component used in a BOM to an access controlled, centralized component library.

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:



AD schematic

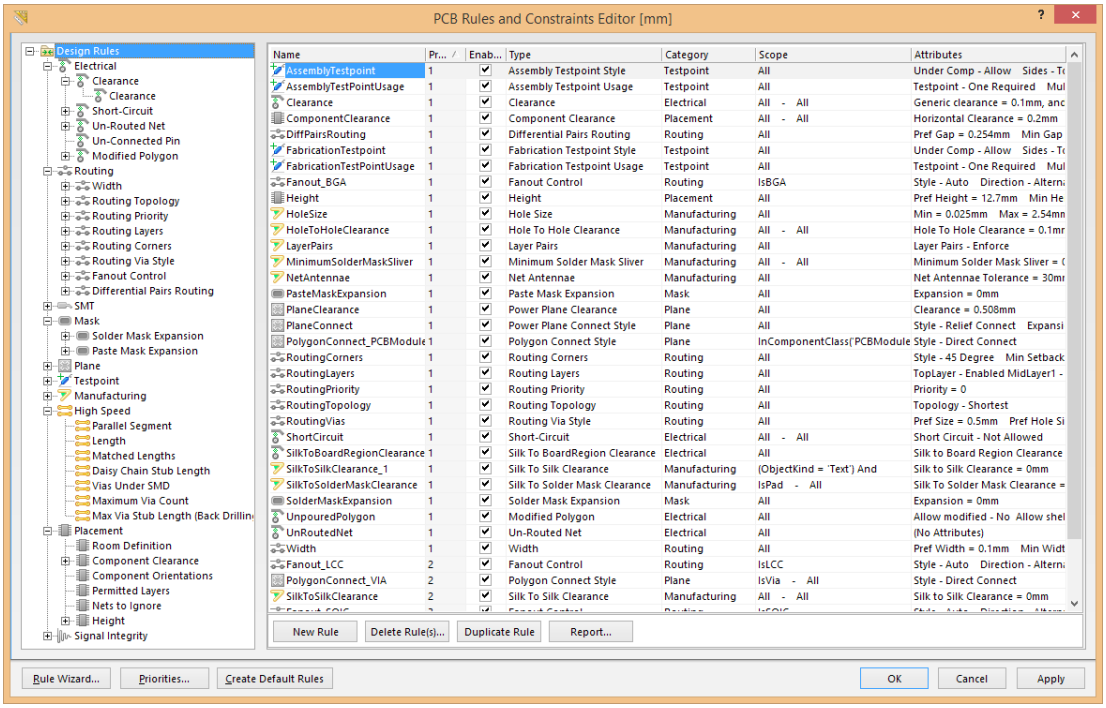
- **Starting a schematic is fast and easy** with intuitive dialogs, e.g., editing workspace and establishing sheet design, parameters, preferences and associated documents. Immediately manage versions.
- **Easily set component classes, net classes, and placement rooms** in an intuitive environment that 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.

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



Rules and constraints editor

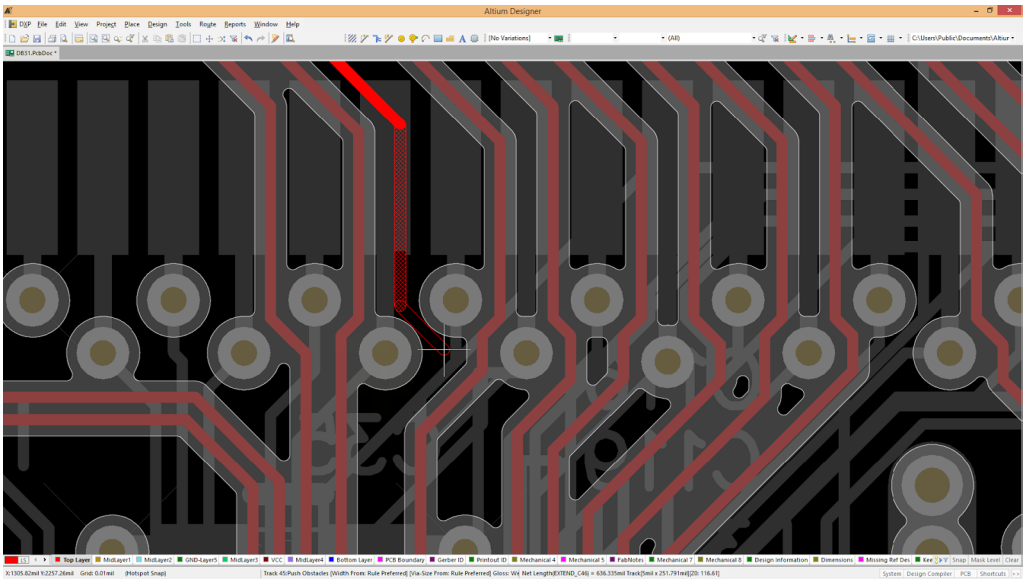
- **Easily browse, create, prioritize**, define the scope, edit, duplicate and delete rules all in one editor.
- **Evaluate your rules** in a table-based summary that provides straightforward review.
- **Define multiple rules** of the same type, but each targeting different objects.
- **Decide exactly 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.

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 Designer is continually adding more powerful and differentiating placement and routing features. These features will increase your productivity, streamline your core PCB design tasks, and reduce your time to market.

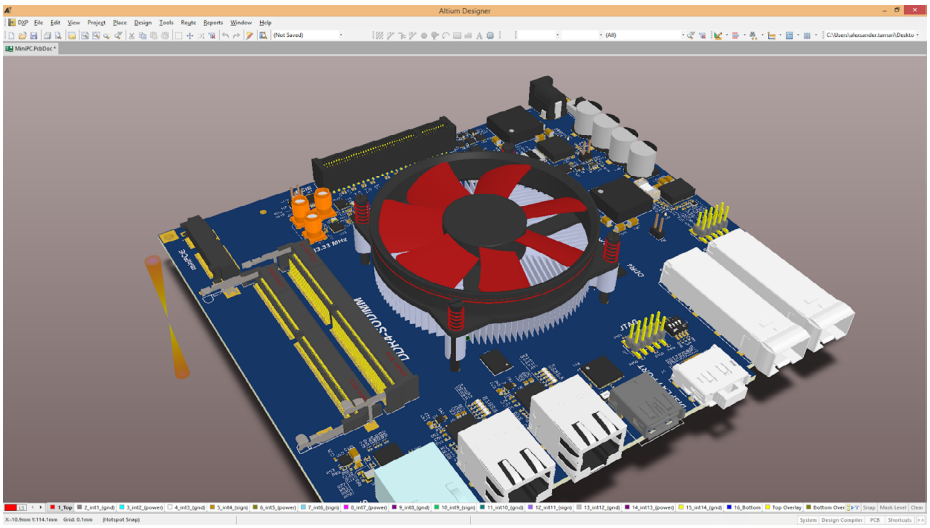
See Altium Designer product website for examples of the many benefits and new features introduced: http://www.altium.com/altium-designer/overview.

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 and offers very little insight into the third dimension.

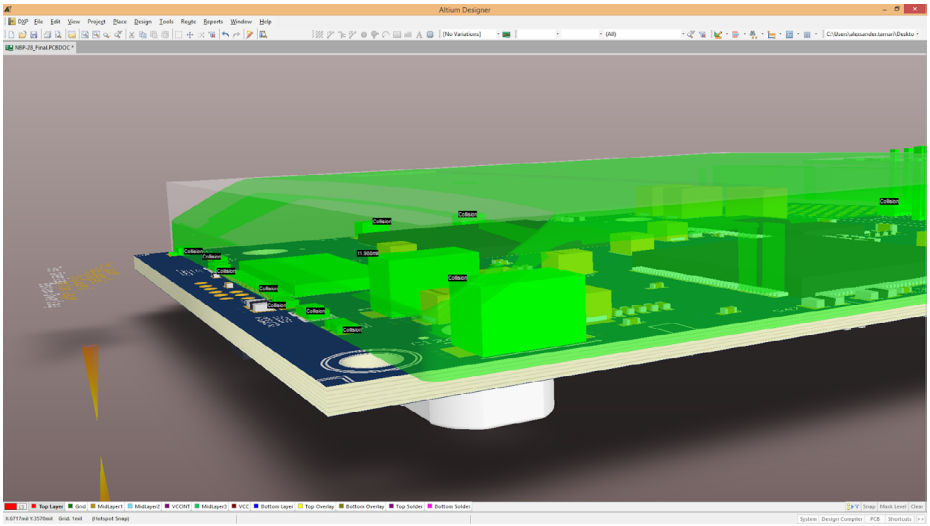
DESIGN COLLABORATION IN ALTIUM DESIGNER

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. 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.



Native 3d 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.



Clearance checking in 3d

FOR A FULL EVALUATION

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