

A UNIFIED SYSTEM FOR PCB DESIGN

Altium Designer® provides a unified PCB design environment to implement your entire PCB design process from concept through manufacturing. A modern user interface makes it easy to learn and enjoyable for daily use.

The unified workflow in Altium Designer provides a consistent user experience throughout the entire product creation process while maintaining a single, secure dataset. You get uninterrupted workflow from schematic capture, board construction, routing, ECAD data management, component management, and project release management without the error-prone practice of transferring files between design tools.

Additional products can be added to enhance functionality through the Altium Extensions platform. Extensions are analogous to apps on your cell phone and provide specialized capability, such as power integrity analysis.

Automation helps minimize the tedious parts of the job and powerful NATIVE 3D®™ MCAD/ECAD integration makes your digital designs look as real and accurate as the finished product. With Altium Designer, PCB design is the main focus, so you can spend less time worrying about process and more time on design.

Altium Designer leverages over 30 years of EDA software development to provide innovative technologies stemming from schematic capture to NATIVE 3D™ rigid-flex board layout and design.

SCHEMATIC CAPTURE

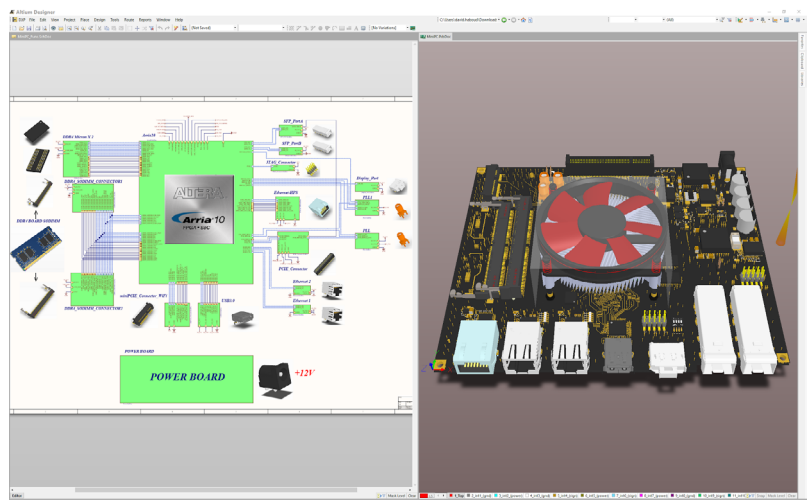
Every PCB design depends on accurate schematics. Whether you are starting from scratch or working on existing designs, schematic capture is fast and intuitive with the modern user interface. Import designs from PCB design packages and get started on your existing designs immediately. Altium Designer maintains a two way connection between your schematics and PCB to provide a unified interface throughout the design process, improving productivity and enabling cross-referencing between schematic and PCB layout.

Design Importers

Being able to leverage previous designs is important for every company. Save time re-creating schematics, board layouts, and associated design data with an automatic importer of project files from P-CAD®, EAGLE®, OrCAD®, PADS®, Xpedition® xDX Designer, Xpedition® Enterprise, CADSTAR®, and Allegro®.¹²³⁴ You don't have to lose any of your hard work and you can finish working on designs in a unified system.

Dynamic Cross Probing

With cross probing, when you select an object on schematic that same object is selected on your PCB and vice versa. Cross probing automatically cross-references every net, pin, and component on your PCB to give you the clearest insight into the implementation of your schematics. Quickly navigate between multiple files in your design project with cross-referencing for every design object and see the results in real time, in the same design environment.



Cross Probe from schematic to NATIVE 3D PCB layout

Mixed Simulation

Altium Designer allows users to easily create and manage multiple simulation profiles. Separate profiles allow designers to run different types of analyses with different simulation engines (Mixed Sim, SIMetrix, SIMPLIS). You can also have multiple runs of the same simulation type (e.g., AC analysis) with different parameters and options (e.g., different frequency ranges). Active profiles can easily add, remove, edit, run, and/or generate netlists. The Profile Manager organizes profiles and uses probes or active nets to select waveforms to display. All simulation results can be saved with other manufacturing outputs for conveyance to manufacturing.

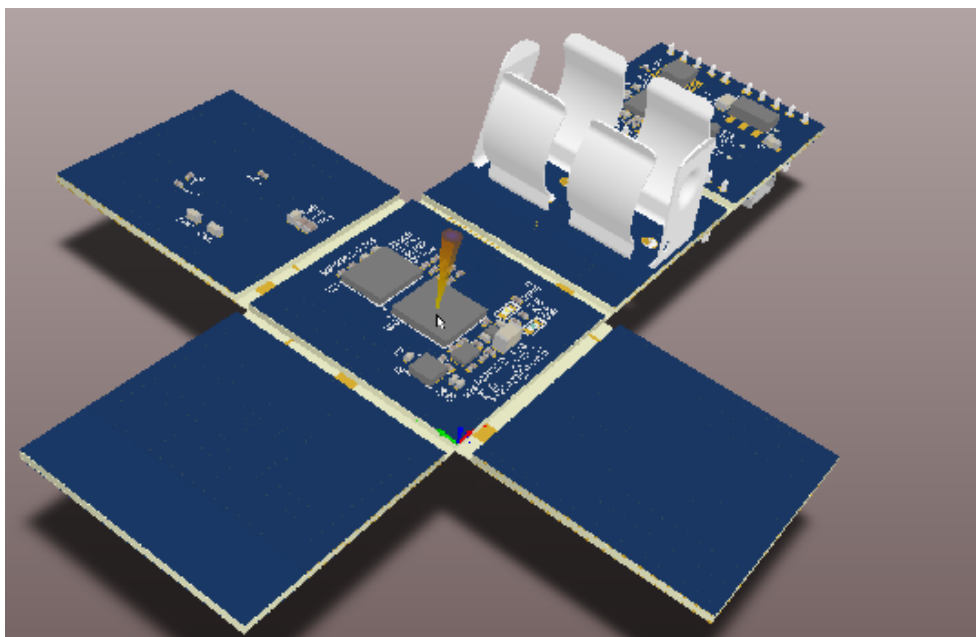
Board Construction

Finding out your board interferes with a connector at the prototype phase is a costly error. Ensure your board fits the mechanical enclosure right the first time with powerful ECAD/MCAD collaboration tools including NATIVE 3D clearance checking. This lets you view, rotate, and translate your design so that you can visually check all clearances.

Layer stack management allows you to define the material composition and specialized regions of your board. For flex circuitry and rigid-flex PCB designs, you can control the entire stackup, including all rigid and flex portions. All your design content creation is guided with design rules and regions to make it easy to convey your design intent to manufacturing and create accurate designs that can be readily fabricated.

NATIVE 3D MCAD/ECAD INTEGRATION

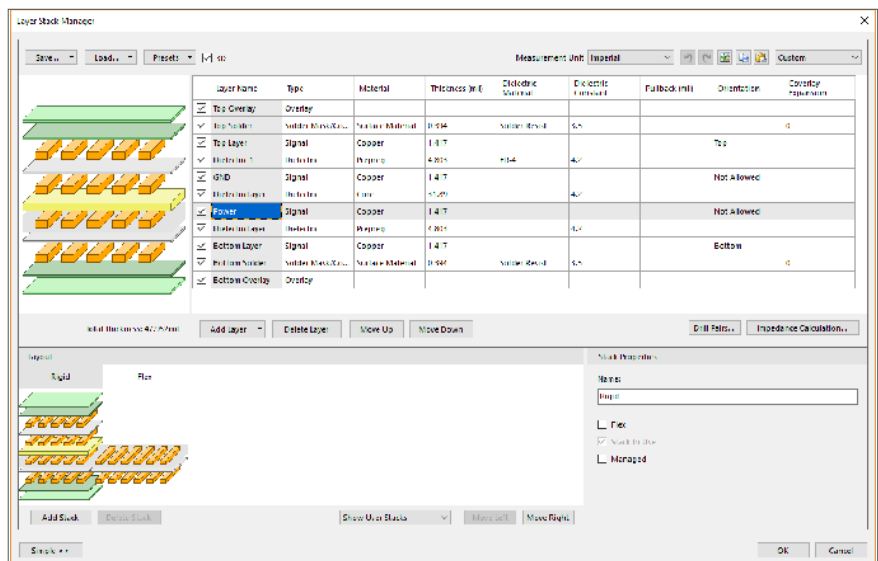
Altium Designer connects your PCB design workflow to the complete engineering ecosystem with powerful ECAD/MCAD collaboration features and fully managed library systems. You can ensure your board fits your mechanical enclosure right the first time without costly prototypes by check-fitting with real-time clearance checking in NATIVE 3D. Import 3D models and mechanical enclosures from SOLIDWORKS® and other MCAD systems to have the most realistic, accurate, and data-rich 3D models to get an exact representation of your physical board. The IPC-compliant PCB Component Wizard handles any other model needs with a guided process for custom component creation.



PCB design in NATIVE 3D

Layer Stack Manager

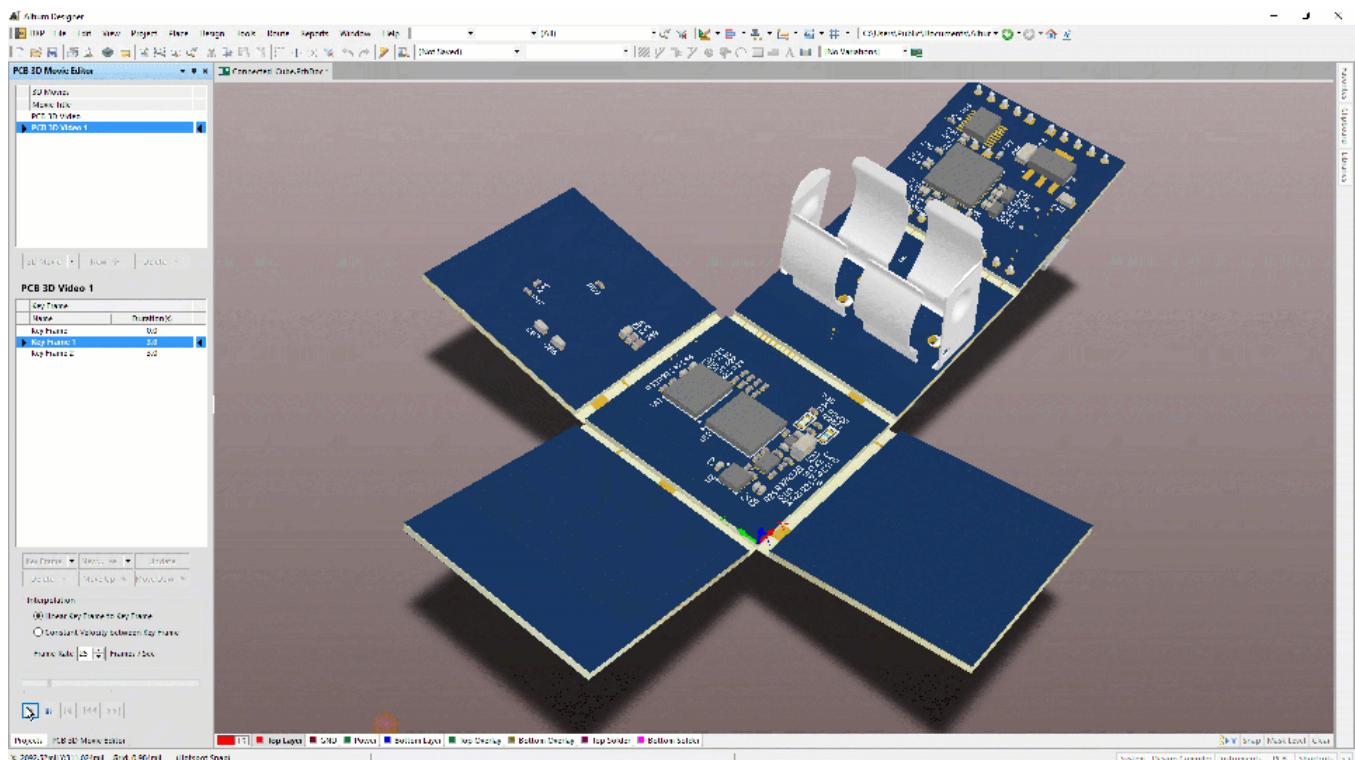
Define and manage all of the layer stackups in your design and account for rigid-flex and complex high-speed stackups in a central location. Visualize your layer stack anywhere from 2 to 32 layers for your designs using subsets of materials used in your primary layer stack. Complex boards with multiple stackups can be defined side by side to facilitate board construction.



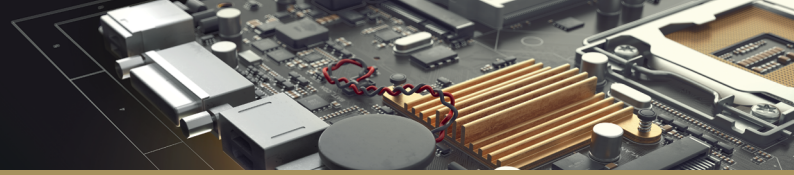
Layer Stack Manager

Rigid-Flex

You can add rigid-flex sections to your PCB design with bikini coverlay support and check clearances in NATIVE 3D. Easily design flex and rigid-flex PCBs and confirm that your board fits your mechanical enclosure right the first time with real-time clearance checking in NATIVE 3D. Gain control of all your board movement with definable angle, radius, and fold index for rigid-flex regions. One of the most valuable features lets you create full-motion video clips from board 3D snapshots to convey design intent to manufacturing. This is critical in designs that use multiple folds to fit in tight spaces or allow movement of the flex circuitry.

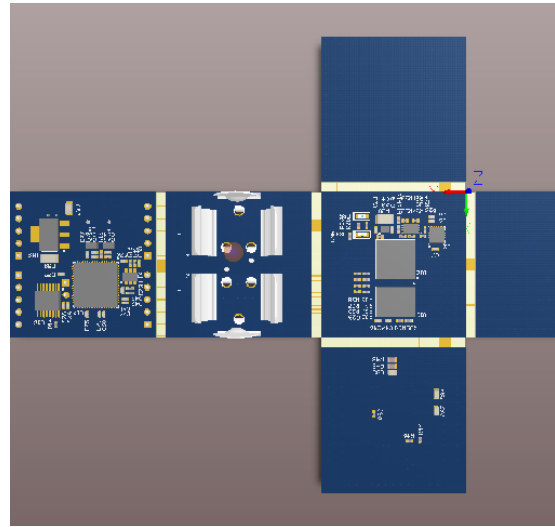
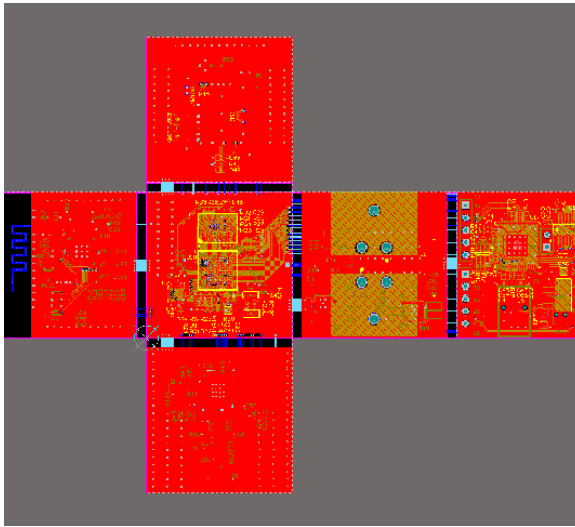


Video of Rigid-Flex Design



Layout

Design the most organized and efficient board layout with the ability to place and drag components that push, avoid, and snap-to alignment with other objects and pads on your board layout. These features make laying out dense boards much easier, and maintain design rule compliance as well. Signal integrity disturbances are reduced on high-speed PCBs with complete control over every drill hole with hole tolerances and back drilling capabilities.



2D & NATIVE 3D PCB Layout

Design Rules & Regions

With Altium Designer, you can easily define design rules and rule hierarchy. Design rules compliance as you progress keeps the layout manufacturable and lets you communicate intent to manufacturing based on your part parameters. Design rules let you check for issues as you are designing, so that you can prevent issues before they stack up. Set specific manufacturing guidelines with a customizable design rule system including specifications for board outlines, solder mask expansions, drill placement, and an advanced query editor to create non-standard rules. With design rules, you can complete right-first-time designs.

Routing

Design the highest quality PCB layouts in a fraction of the time with an advanced routing engine that includes push and shove, hug, walk around, and interactive length tuning modes for single and differential pair routes. xSignals delivers fully configurable differential pair routings that carry precise signal lengths across your PCB for high-speed design. ActiveRoute® assisted routing lets you control where and how much automated routing assistance you want on a net connection level up to your whole design. Visual clearance boundaries between traces and components on your board let you visualize design rules and understand your layout at a glance.

Interactive Routing Modes

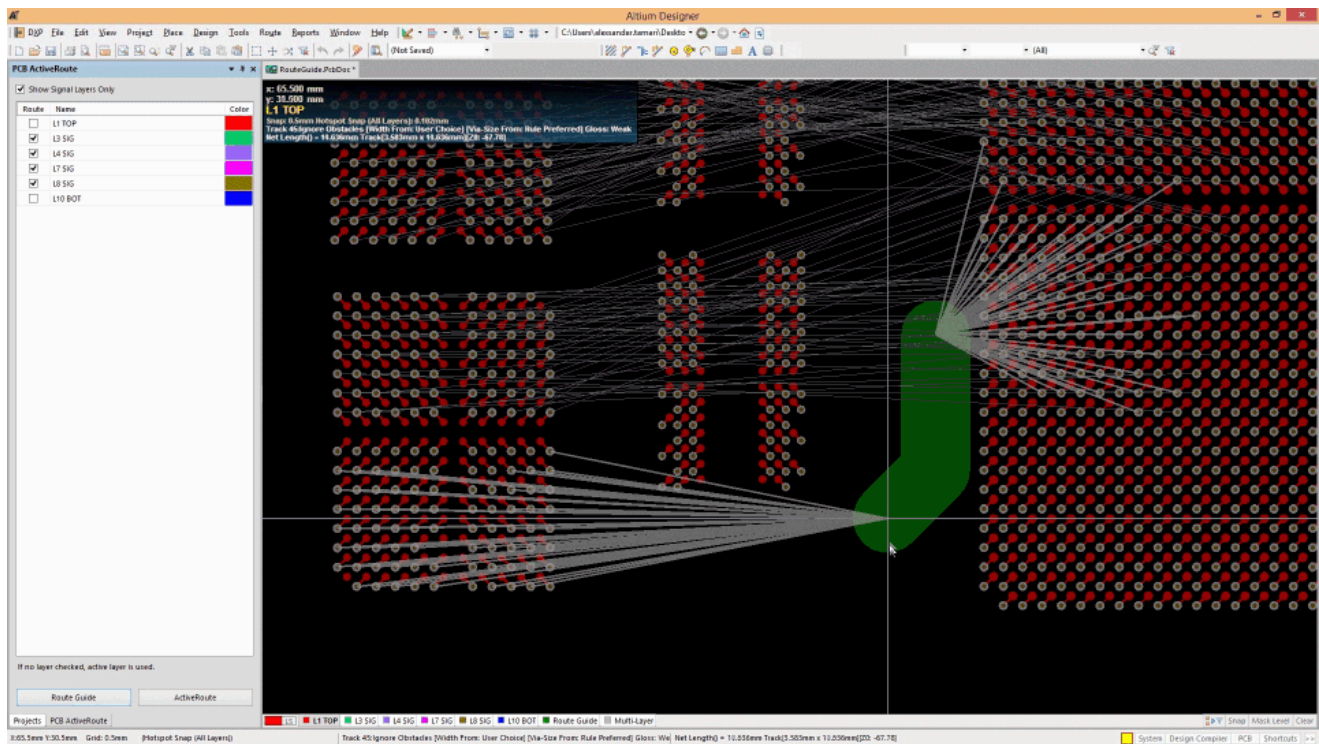
The interactive routing modes in Altium Designer help you lay out boards faster than ever with precise control over the organization and flow of your board layout. Interactively route your board with several powerful routing options including autoroute, walk around, push, hug and push, ignore obstacle, push & shove, and differential pairs. You can even automatically align routing path lengths without ever having to waste time manually adjusting individual nets with interactive length tuning.

High-Speed Routing Tools for DDR3/4 and USB3.0 (xSignals)

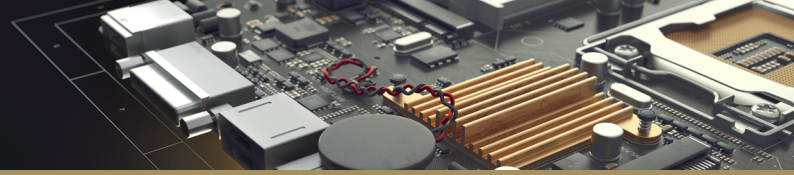
Working with high-speed designs can be time-consuming. The xSignals wizard lets you easily plan and constrain your high-speed designs by defining high-speed signal paths for modern technologies. Route your high-speed design with fully configurable differential pair routings that carry precise signal lengths across your PCB. DDR3/4 and USB3.0 signals are automatically identified by the wizard creating rules to keep all signals in sync and tuned to the correct length.

ActiveRoute

ActiveRoute is a tool that lets you select where and how much automation you want to employ. The technology in ActiveRoute coupled with your guidance produces high-quality layouts in seconds. ActiveRoute lets you breakout and route large, fine-pitch BGAs by instructing it where to route them and letting it route for you. Unlike other routers, ActiveRoute works on multiple layers simultaneously. By routing on multiple layers simultaneously, routing is faster, traces can be evenly distributed, and the ability to complete the routes increases significantly.

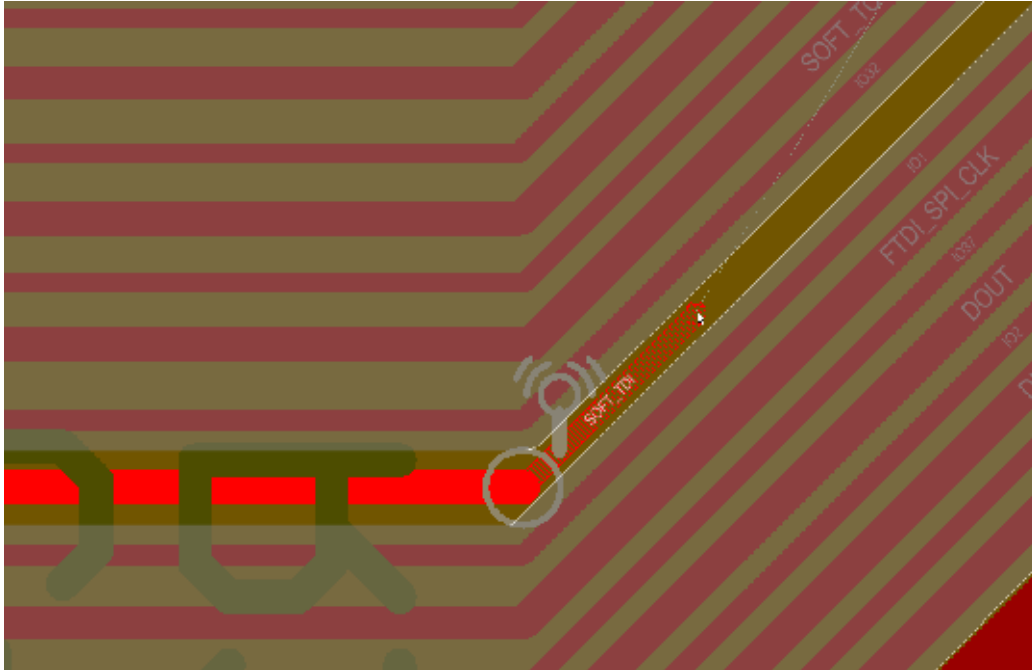


ActiveRoute with Guided Path for BGA



Visual Clearance

You can visually see clearance boundaries between traces and components as you route your board. Understanding the impact of your routing decisions in real-time alleviates the stress from unclear obstacles in the design process. Route through high-density areas with certainty that traces fit where you need them.



Running a trace through a congested area

Design Objects

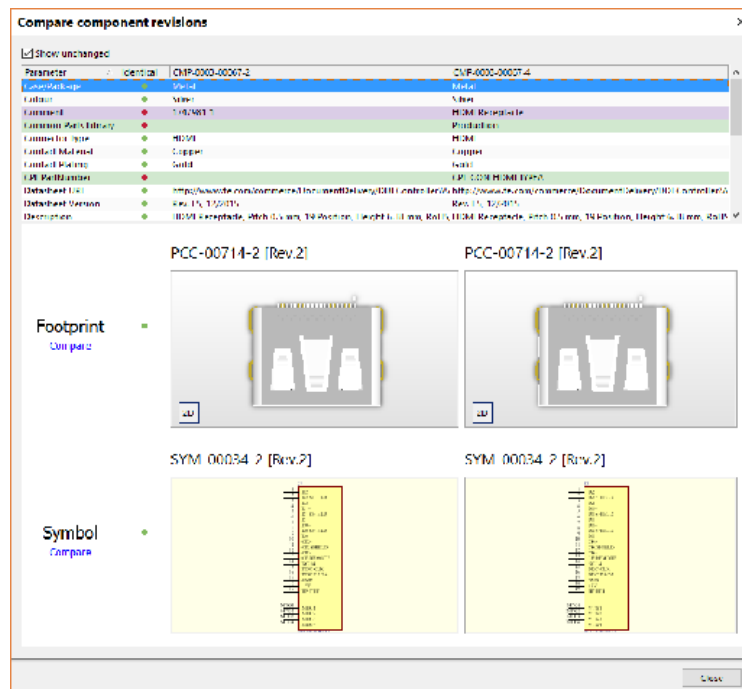
There are always areas of your design that require special considerations and rules to maintain a viable design. Altium Designer incorporates the use of rooms, keepouts, and polygon regions to aid you when designing. Rooms are regions that assist in the placement of components by grouping them inside the designated area. On the other hand, keepout regions act as an 'interference' object that prevent other copper objects from intersecting its area, as specified by the global Clearance Rule. Lastly, polygons act as regions in the board to dictate allowable copper pour areas, as well as defining copper pour order.

ECAD DATA MANAGEMENT

Knowing exactly when and what design changes were made and by whom is the cornerstone of ECAD data management. Integrated version control details compile logs to allow you to track, compare, and review changes and their sources to keep your design on track. Design reuse tools enable powerful logical and physical to give you a head start on your next design project as well as establish company-standard layouts, such as power supplies or connectors. Templates, comparison tools, and a centralized platform for managing design data, projects, and infrastructure facilitate ECAD data management the full length of the design process.

Version Control

Know exactly who made changes to your design and when with integrated version control. Check design files in and out of a central repository for design accountability and traceability. You can track changes made to your design with a detailed change logs and compare changes to identify exact modifications with revision comparisons for files, components, and text.



Version comparison of component

Design Reuse

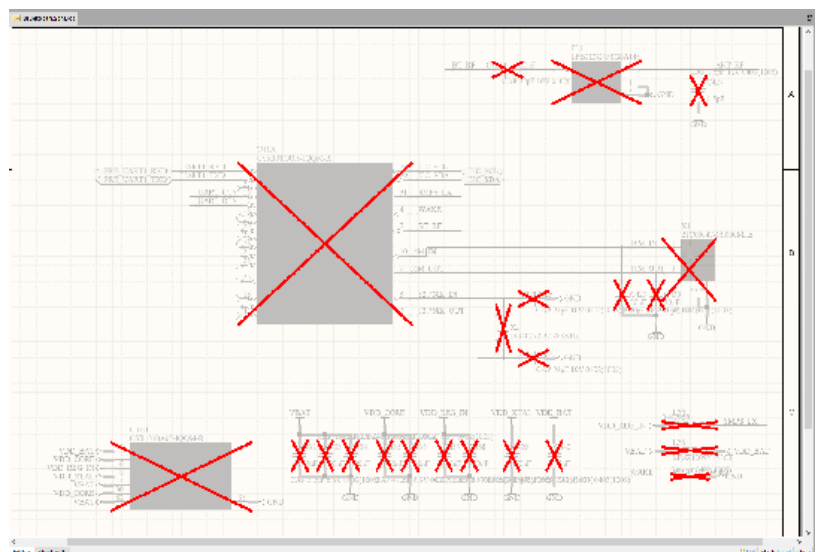
Efficiently manage your design work-in-process data with comprehensive data management options including a complete version control system and lifecycle management system. Shave hours off of future projects by reusing sections of circuitry known as snippets or creating templates for everything from pads to entire project types. Device sheets allow you to create known circuitry blocks for reuse across designs.

Design Variants

Create multiple versions of a board design with modifications to objects and other design elements to create unique products from the same base design. Each variant stands as an original design with different components, version-specific design elements, and unique outputs to send to manufacturing.

Component Management

A last-minute supply issue will derail your design timeline and increase costs. Component management leverages complete control over your component selection process to avoid any delays when manufacturing your board. Integrated supply chain information combined with specified alternate parts directly in your BOM minimize the possibility of supply issues.



This example shows a variant of a design with Bluetooth circuitry omitted

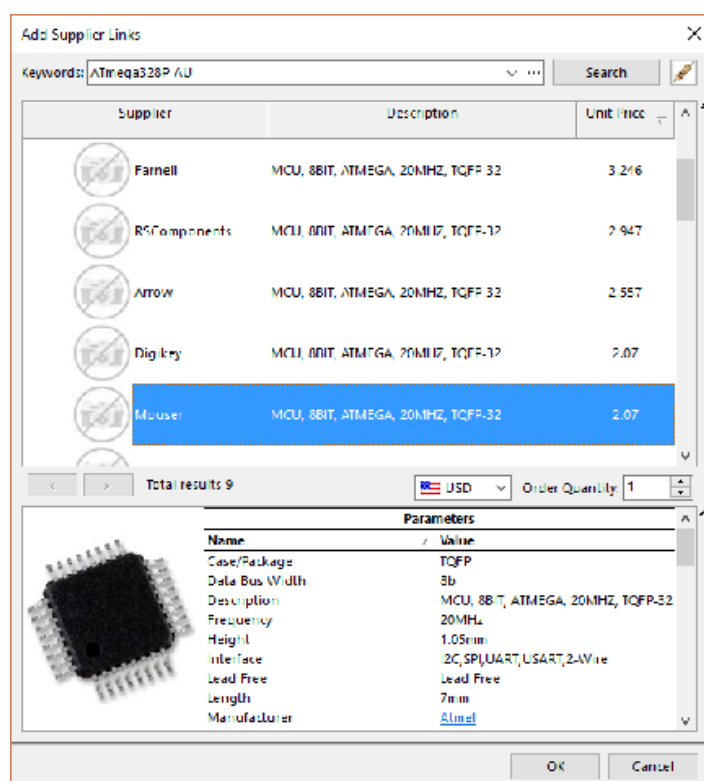
Plus, save time creating components by tapping into the extensive source of design content from the experts at Altium including over 300,000 ready-to-use parts with the Altium Content Vault.

Comprehensive Library Management

Avoid re-creating components with an organized component library. Altium Vault® lets you add components with schematic symbols and PCB footprints together with simulation information and user-defined parameters while preventing duplication. You can even link to your existing company database. Maintaining your libraries in the same ecosystem as your design environment allows direct placement on a schematic and PCB layout level. With the addition of Altium Vault, you can keep your components organized, gather usage statistics, and update components to the latest revision with the click of a button.

Supply Chain

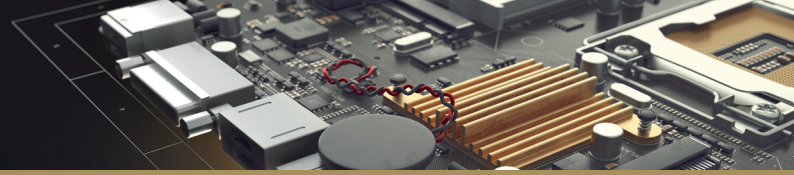
Always meet your time-to-market and budget goals with real-time pricing and availability data from your most trusted parts suppliers and over 100 Altium verified suppliers. Pick parts for your design by leveraging price, availability, and more real-time supplier information to meet all of your design goals.



Supplier link search

Alternative Part Choice System

Gain complete control over your component selection process and avoid any delays when manufacturing your board by specifying pin-compatible backup part choices directly in your BOM. You can automatically substitute part numbers as needed to get a design out without any replacement hassles. Informed part selections make it easy to meet your project budget with ongoing cost estimations and availability directly in your BOM.



Project Release Management

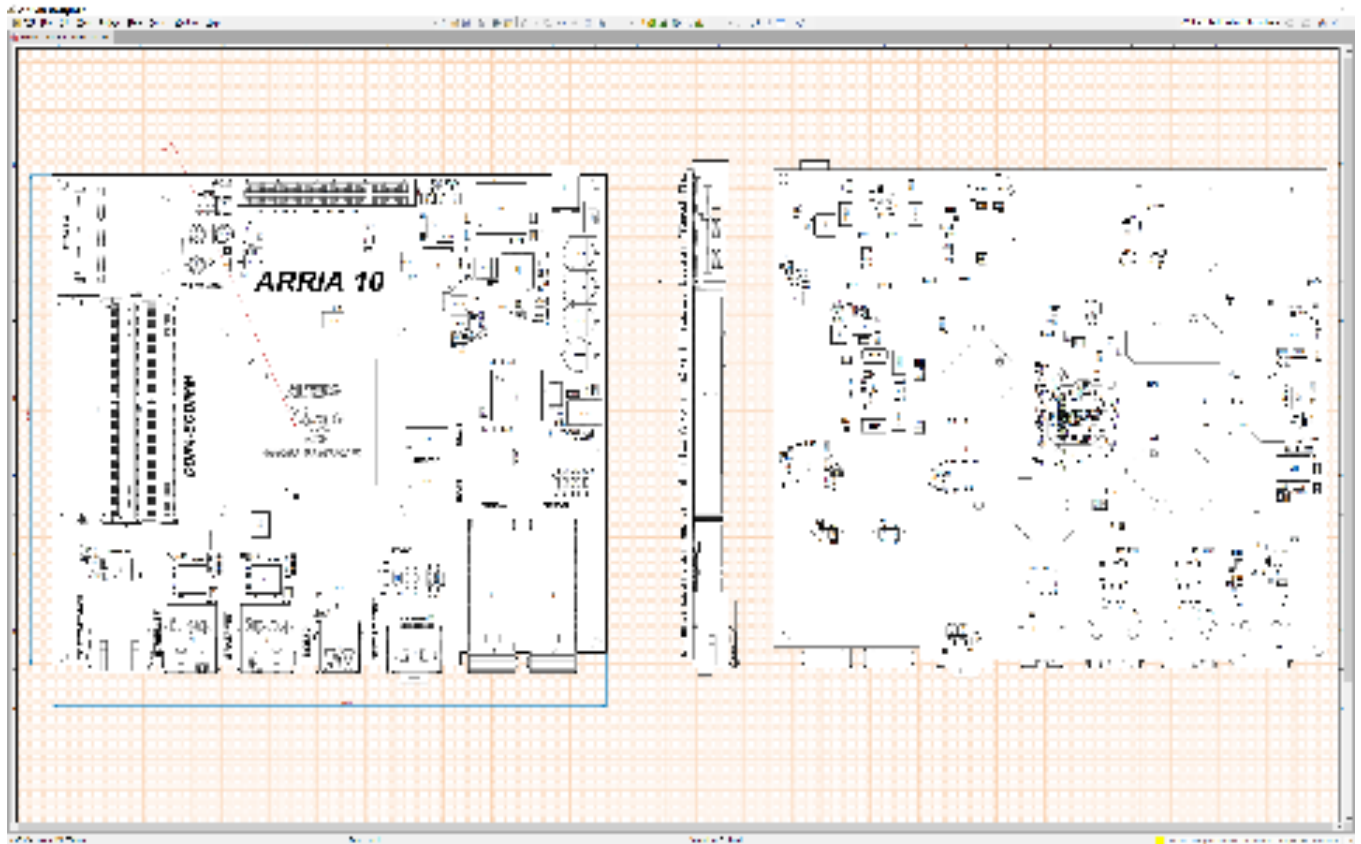
When you are ready to move your finished design into manufacturing, clear communication is mandatory. Output jobs act as an organized, reusable container for all necessary design outputs. Clear design intent communication to manufacturing with powerful release management and automated documentation tools built into Altium Designer.

Organized Release Process

Altium Designer enables you to add verified project snapshots into Altium Vault or desired folder structure with the streamlined project releaser. Dynamically create customized project outputs for design variants linking fabrication and assembly outputs with the latest design source files. With an organized release process, your output generation gains consistency and accuracy while ensuring you don't use out-of-date design files.

Documentation Automation with Draftsman

Create assembly and fabrication documentation directly linked to your source designs with Draftsman®. Update your entire documentation at the click of a button. The set of powerful and easy-to-use features integrated in Altium Designer automate documentation ensuring consistency. Create templates for documentation that only require minimal customization across designs. Add PCB dimensions, measurements, notes, and callouts between datums and design objects to customize your documentation workflow with Draftsman.



Assembly documentation generated from a Draftsman template

LICENSING VERSIONS

ALTIUM DESIGNER

Access to complete PCB design and manufacturing capabilities. Includes front-end engineering schematic design and verification system for both board-level and programmable logic design and NATIVE 3D PCB design/editing capabilities.

ALTIUM DESIGNER SE

Access to complete schematic design capture solution for both board-level and programmable logic design. Includes analog/digital mixed SPICE simulation.

ALTIUM DESIGNER VIEWER

Anyone on a design team can use the read-only Viewer to review design files created in Altium Designer including schematics, PCBs, and fabrication files.

LICENSING TYPES

STANDALONE LICENSE:

Ideal for a single workstation and solo designer. Work can be saved, copied, and backed-up as needed.

ON-DEMAND LICENSE:

Let us manage your licenses with the Altium On-Demand license server.

PRIVATE SERVER LICENSE:

Share licenses inside a personal network with the Altium Private License Server.

Refer to the **License Feature Set Matrix** for a full list features associated with each license type.

¹ **Xpedition®** and **PADS®** are registered trademarks of Mentor Graphics Corporation and Altium claims no rights therein.

² **EAGLE®** a registered trademark of Autodesk Inc. and Altium claims no rights therein.

³ **OrCAD®** and **Allegro®** are registered trademarks of Cadence Design Systems, Inc. and Altium claims no rights therein.

⁴ **CADSTAR®** is a registered trademark of Zuken and Altium claims no rights therein.

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®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXPTM, Easytrax®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™ and the TASKING® range of embedded software compilers.

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 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](#).