ALTIUM UNDERSTANDS THAT PCB DESIGN IS A WORK OF ENGINEERING ART

Altium Designer® combines exhaustive scientific knowledge with a natural design intuition to create **a single, unified design environment.** Your electronic designs demand the highest grade of productivity and performance. When your efficiency is measured against immovable deadlines, precise layouts, and exact fittings, you can't afford to not invest in a **complete PCB design platform**.

Altium Designer has all the features you need to meet your demand challenges and a track record of delivering more features and differentiating features in predictable and reliable releases. With Altium Designer **Links to Supply Chain**, you always get the best pricing and availability on components from your most trusted and reliable parts supplier. You work from a **single source of trusted design data** available to your entire design team with centralized library management tools. You have **advanced and easy to use schematic capture, placement and routing technology**.

Collaborate with mechanical design in real time, in parallel, with real data to ensure that your electronic designs fit right the first time with powerful **Native 3D** visualizations and clearance checking. Unite all of the separate yet intimately connected details of your design process with **powerful data management and design reuse** tools. Control the consistency and reliability of your project throughout its entire design journey with **flexible release management tools**.

WHY CHANGE & WHY NOW?

With your current design environment and PADS®, are you falling short of meeting minimum design specifications, 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? Have you heard the same story from PADS® for years and have decided that it is time to change?

With rapidly increasing PCB product complexity, larger circuits and decreasing sizes, you can no longer accept the inability to meet your goals as normal and acceptable. You require 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. Now is the time to look more closely at Altium Designer.
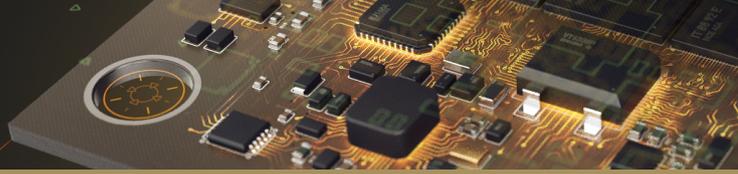
OVERVIEW OF PADS® PRODUCTS

PADS® sells products and options. The products include independent applications, different data models, design environments, and user interfaces.

PADS Standard

- PCB schematic and layout

PADS Standard Plus

- PCB schematic and layout
- Analysis and verification
- Advanced layout and high speed constraints

PADS Professional

- PCB schematic and layout
- Analysis and verification
- Advanced layout and high speed constraints
- FPGA co-design
- Signal/power integrity and thermal analysis
- RF, sketch routing, and 3D

OVERVIEW OF ALTIUM DESIGNER

A unified environment for your design, data and release management process

Altium Designer provides a unified PCB design environment within an all-in-one product that is easy to learn and use. You have the features you need to make design decisions early in the design process, 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's **advanced supply chain management** system, you will have real-time visibility into your internal and external supply chain data, and access to all information about sourced components. This means you can make intelligent part choices earlier in your design, reducing the likelihood of costly and time-consuming rework late in the design cycle.

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

Altium Designer's **Output Job Editor** is a central location for you to configure and save your publication outputs. You can publish selected outputs to PDF, print or generate files and add them to your project. The same Outjob can have any number of publications configured, **ready for reuse** everytime you update your project.

Altium Designer provides a **release management** capability that provides checks and balances for your entire PCB design process. You can search and release accurate output data on your designs. You will have a visual representation of your project's 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.
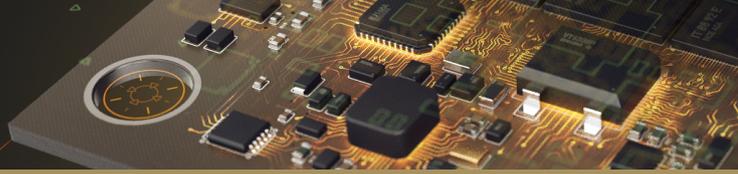
ALL WITHIN ONE, MODERN USER INTERFACE

All Altium Designer features are presented within one, **modern user interface (UI)**. If you are responsible or not for every aspect of the design process, 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. This gives you 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 to quickly become proficient as you take on additional design tasks.

LIBRARY AND COMPONENT MANAGEMENT IN PADS®

PADS offers DxDataBook and PartQuest. DataBook is an older, spreadsheet-based application. PADS recently introduced PartQuest, a **component search engine for Digi-Key** part numbers with symbols and footprints. When evaluating, consider that **PartQuest only supports Digi-Key parts and does not output Logic symbols**.

INTEGRAL LINK TO THE SUPPLY CHAIN AND REAL-TIME COMPONENT MANAGEMENT WITH ALTIUM

Altium has been the leader in providing a complete solution for the PCB design, development and production process. One of the fundamental aspects of the process includes a **close connection to supply chain and real-time component management**.

Altium provides **Octopart, a search engine from where you can search across hundreds of distributors, thousands of manufacturers, and millions for parts**. As is typical with Altium Designer in which all aspects of the entire PCB design process are optimized, you 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.

Octopart is **integrated with Microsoft Excel**. With the Excel Add-In you can access pricing and availability data from within Excel. You can pull part information for your BOM without leaving your spreadsheet. Octopart also has a **Common Parts Library For Prototyping** which is a collection of electronic components, tools and equipment for prototyping new connected products.

Altium also provides **Ciiva, a cloud-based electronic component management system**. Ciiva brings together supply chain intelligence from information sources that traditionally have been fragmented and places this intelligence at your fingertips. The system includes a fully-traceable, version-controlled bill of materials (BOM) management feature, where every component used in a BOM is linked to an access controlled, centralised component library. You can search electronic components and datasheets, check lifecycle and alternatives, and compare stock and price.

PADS SCHEMATIC CAPTURE APPLICATIONS

PADS offers two schematic capture applications: PADS **Logic and DxDesigner. When evaluating, consider that Logic lacks more advanced schematic capture features, a component information system (CIS), support for hierarchical schematics and concurrent design, and integration with the spreadsheet based constraint editing system (CES)**. Also consider that **Logic has not been significantly enhanced** over the last couple decades. **DxDesigner's user interface and functionality are complex** and could be difficult to learn.
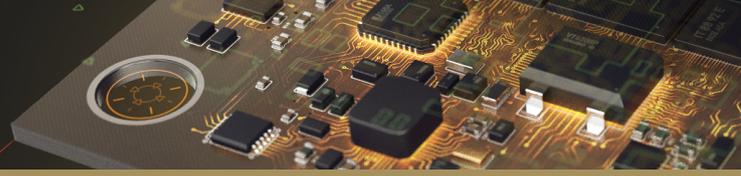
ALTIUM DESIGNER SCHEMATIC CAPTURE … A TECHNOLOGY DIFFERENTIATOR

Altium Designer schematic capture technology has been long recognized as a technology differentiator. Engineers and designers will find that Altium Designer schematic features are easy to learn and they will become productive quickly on all designs ranging between 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. Immediately manage versions. Quickly select and place qualified components from integrated libraries and real-time links to component suppliers. Wiring, bus creation, and net labeling is fast.

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 be generated automatically and a significant benefit of creating a well-structured, hierarchical design up-front. Efficiently compile the project, check that circuits are drawn and wired correctly. Altium Designer provides **powerful ECO features 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, compare and resolve differences.

RULES AND CONSTRAINTS IN PADS

PADS Layout and Router applications each have **separate systems for managing rules** and constraints, and use a **complex rules hierarchy** or pre-defined order. Recall that rules are different in the applications and **switching back and forth**, as you often do between Layout and Router, may cause rules to be overlooked and result in a large number of unnerving and time-consuming verification errors. When evaluating, consider that **area rules are not supported** by either PADS Layout or Router, and support for advanced rules may require you to purchase an option. **Constraint Editing System**, a separate **spreadsheet-based application**, was recently introduced in the PADS integrated flow. When evaluating, recall that your spreadsheet can quickly contain many hundreds of rules and the **interactions between rules can become quite complex**.

PCB 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**. In one editor, you can browse, create, prioritize, define the scope, edit, duplicate and delete rules. You can export a rule template for reuse. Rules are divided into a manageable number of categories (ten). 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.

When evaluating, note that a rule scope is a query that you build 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, but each targeting different objects. Queries are easily accessed for any rule. Advanced (Query) options are also available to help you write your own, more complex queries.

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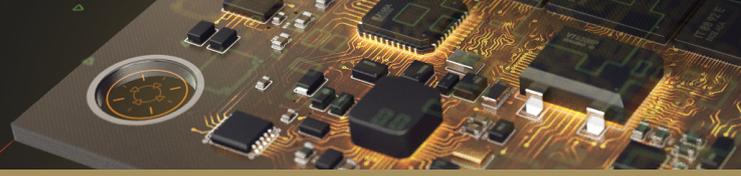
PLACEMENT AND ROUTING IN PADS

PADS Layout and PADS Router have been the core of the PADS product for a couple decades. PADS Layout is older technology and not significantly enhanced in the last decade or so. When evaluating, recall that frequent data corruptions may have forced you, as with many PADS users, to learn the "PADS ASCII Dance", i.e. output to ASCII followed by ASCII import and hope for a mysterious elimination of errors.

Router is more modern technology and has more features than Layout. However, engineers are often forced to **switch back and forth between the two applications**. The use **models, data constructs and the user interfaces are significantly different**. When evaluating, consider the impact of switching and potential loss of data or introduction of errors in your design process.

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 **easily align multiple components**. A differentiating feature is the PCB Editor's ability to **mask or filter objects** in the workspace. This feature 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 completely hide one, many, or all of the connection lines. **Interactive routing includes an auto-complete feature** that will speed you to routing complete. 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.

OTHER DIFFERENTIATING PLACEMENT AND ROUTING FEATURES IN ALTIUM DESIGNER

Powerful routing technology includes **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 greater 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 precision with your **solder mask expansions** with user-definable expansion options from hole edge or pad edge.

A significant differentiator, Altium Designer **supports 3D rigid-flex** design. Easily define material selections and intelligently route your rigid-flex board layout, then visualize your work of engineering art in native 3D.

When evaluating, consider Altium's investment in delivering capabilities to solve your biggest problems across the entire PCB design and production process. 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 to you and the new features recently introduced: http://www.altium.com/altium-designer/whats-new.
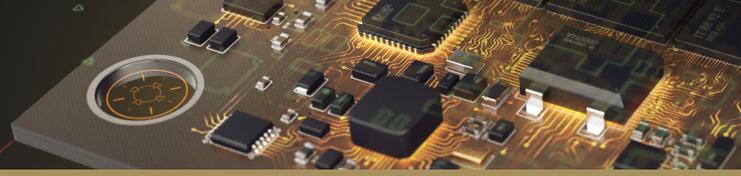
3D AND MCAD COLLABORATION IN PADS

A recent release of PADS included 3D visualization, placement, design rule checking and PDF creation. The PADS application can import STEP models and export PCB assemblies in multiple formats. When evaluating, note that PADS 3D is another **separate application** with its unique look & feel and UI. Also, the 3D application is **new and immature, and lacks some key capabilities** for effective 3D design, e.g. support for holes in the board including pad holes. To communicate between electrical and mechanical CAD systems, you use the PADS Collaborator option, available only with PADS Standard Plus. The option allows users to preview, accept, reject and counter-propose design proposals. The **option uses ProSTEP iViP and IDX data exchange files** to transmit data.

TRUE MCAD COLLABORATION WITH NATIVE 3D 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 is integrated seamlessly into your workflow with designers having **real-time visibility into incremental changes**. This allows the electrical and mechanical engineering work to be done simultaneously and in parallel. You can visualize exactly how your board will fit your 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.

Taking design collaboration between electrical and mechanical design to the next level, Altium provides **MCAD Co-Designer** for Altium Designer and SOLIDWORKS®. This extension takes all of the guesswork out of the design collaboration and data transfer process by providing a managed environment for design revisions between Altium Designer and your mechanical designer's environment.

FOR A FULL EVALUATION

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

Altium