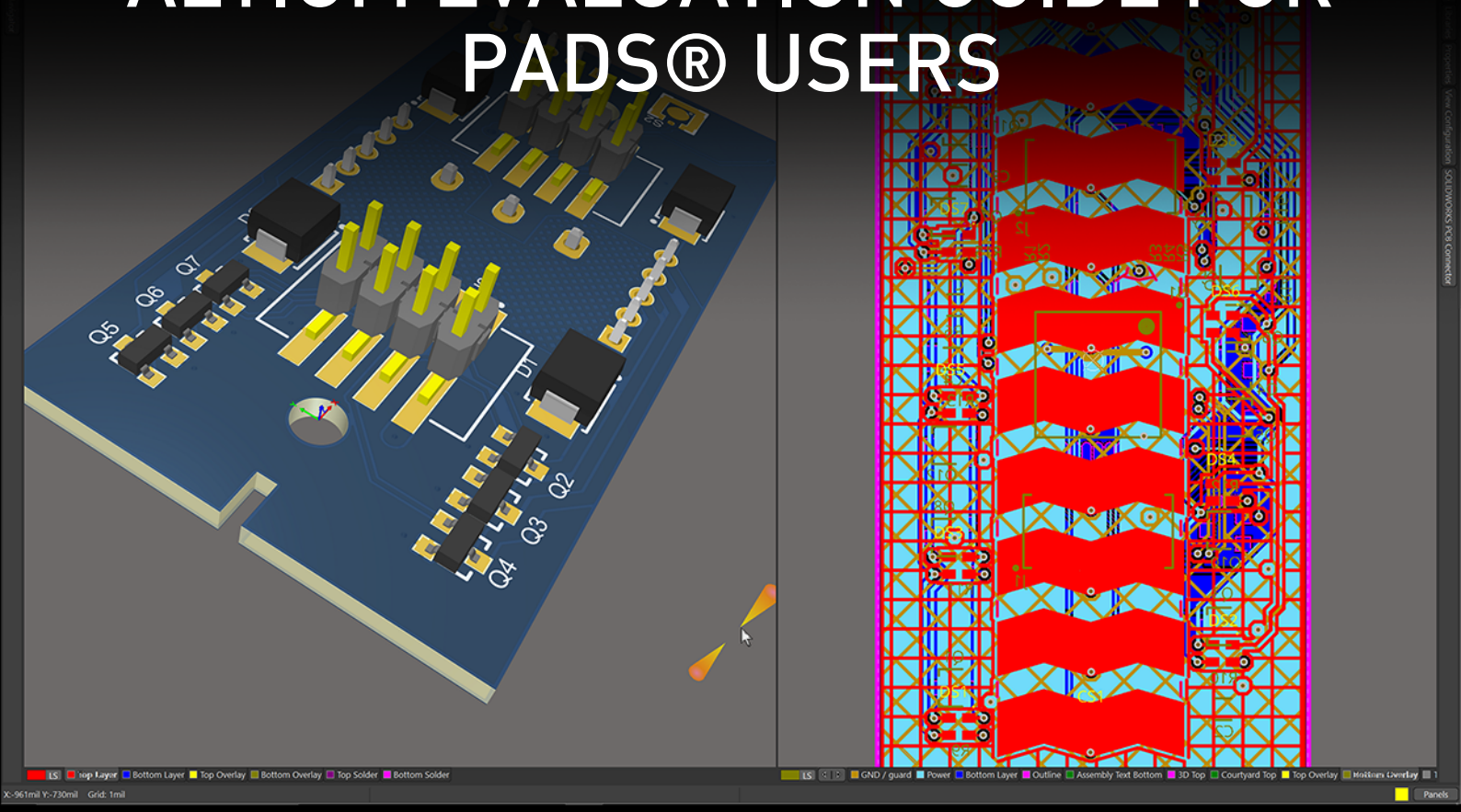
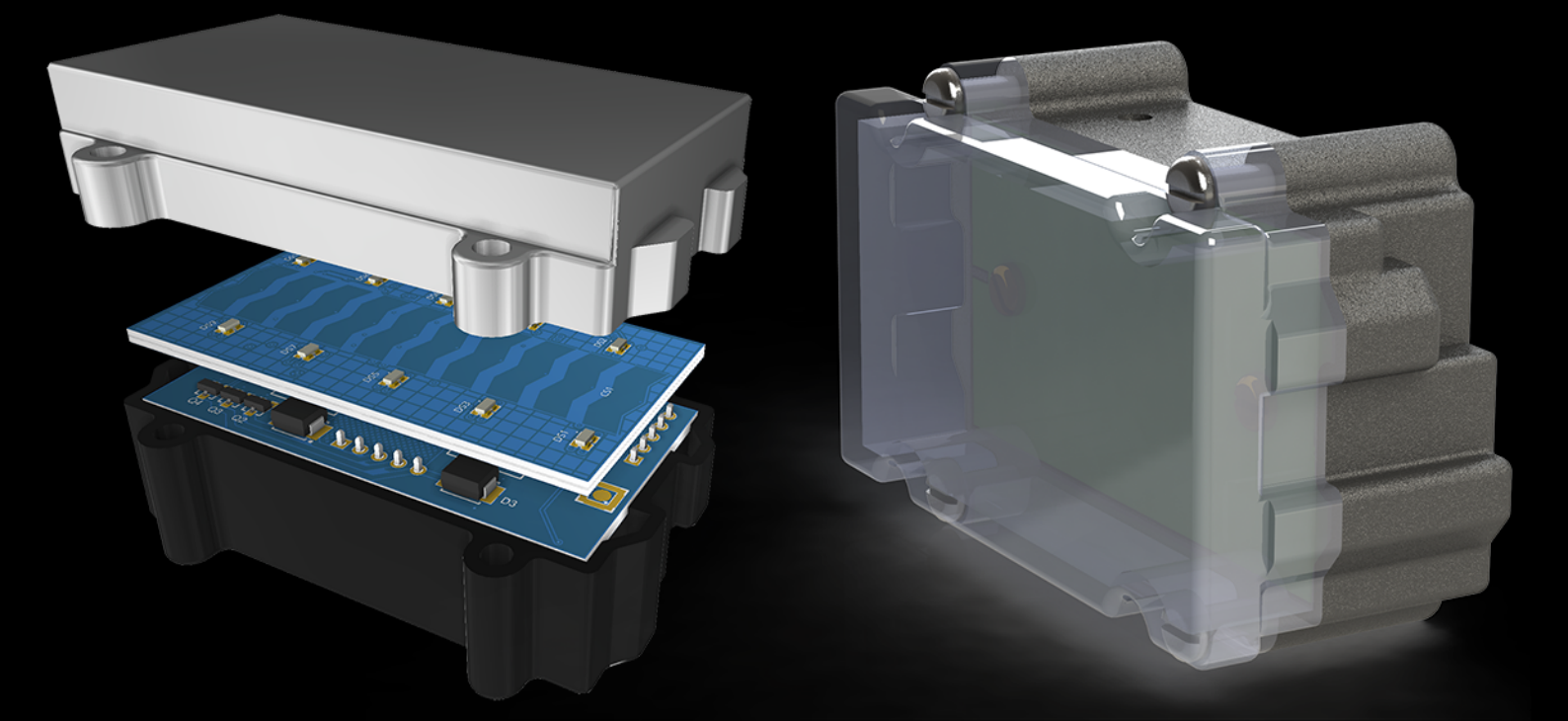
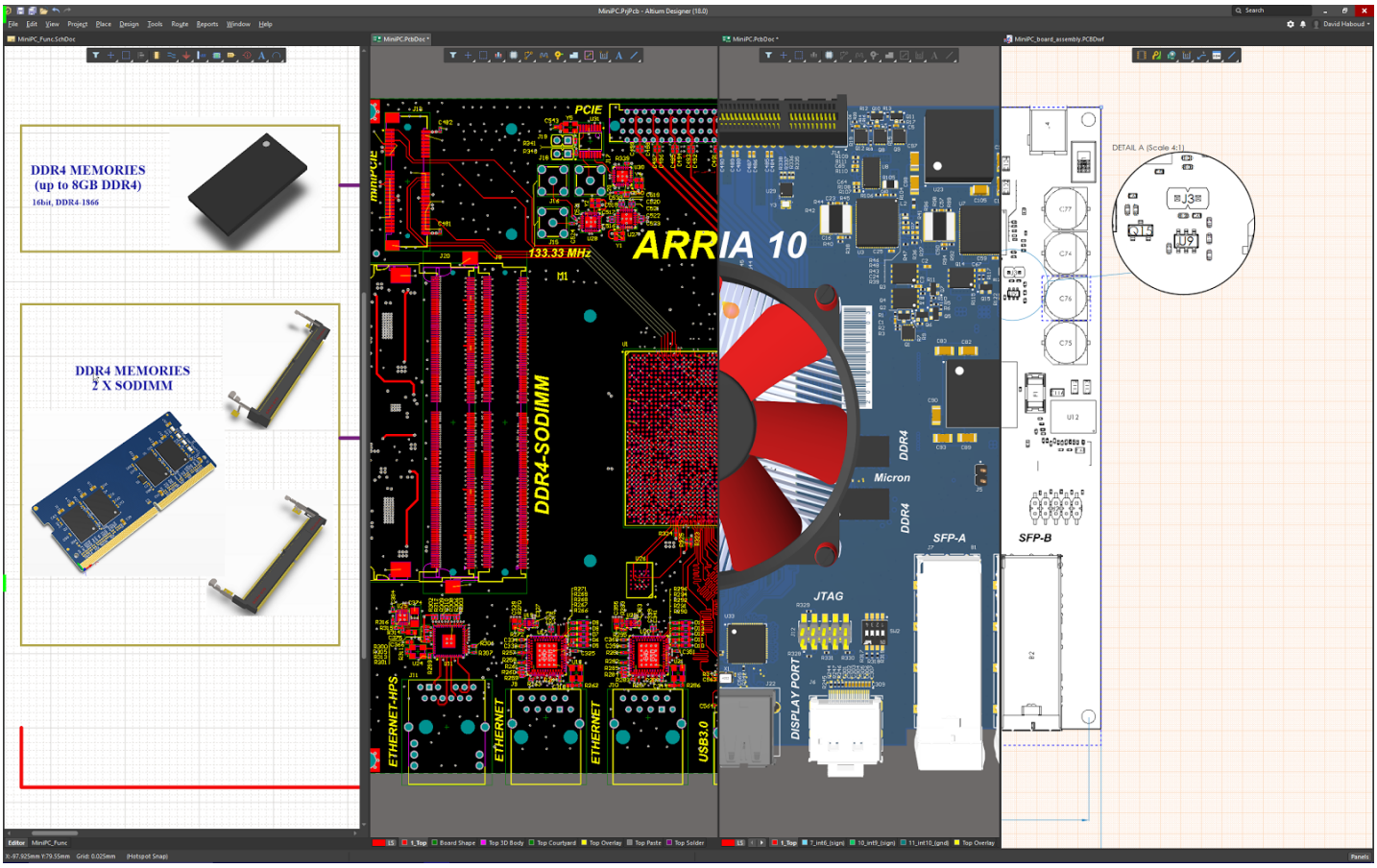
Altium®

ALTIUM EVALUATION GUIDE FOR PADS® USERS





ALTIUM UNDERSTANDS THAT PCB DESIGN IS A WORK OF ENGINEERING ART



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

Altium Designer® combines exhaustive scientific knowledge with a natural design intuition to create a single, cohesive 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 tools you need to meet your design demands, and a track record of delivering more features and differentiating features in predictable and reliable release cycles. 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™ visualization 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.

Multi-Board Assembly Projects let you combine multiple child PCB designs into the overall assembly, with intelligent connection management to ensure correct connector and cable pin assignments, electrical rules checking, and net naming. The Multi-board Assembly editor allows perfect alignment of child PCBs in photorealistic 3D including import of enclosures and other hardware 3D models, to guarantee right-the-first-time PCB positioning and component placement

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?

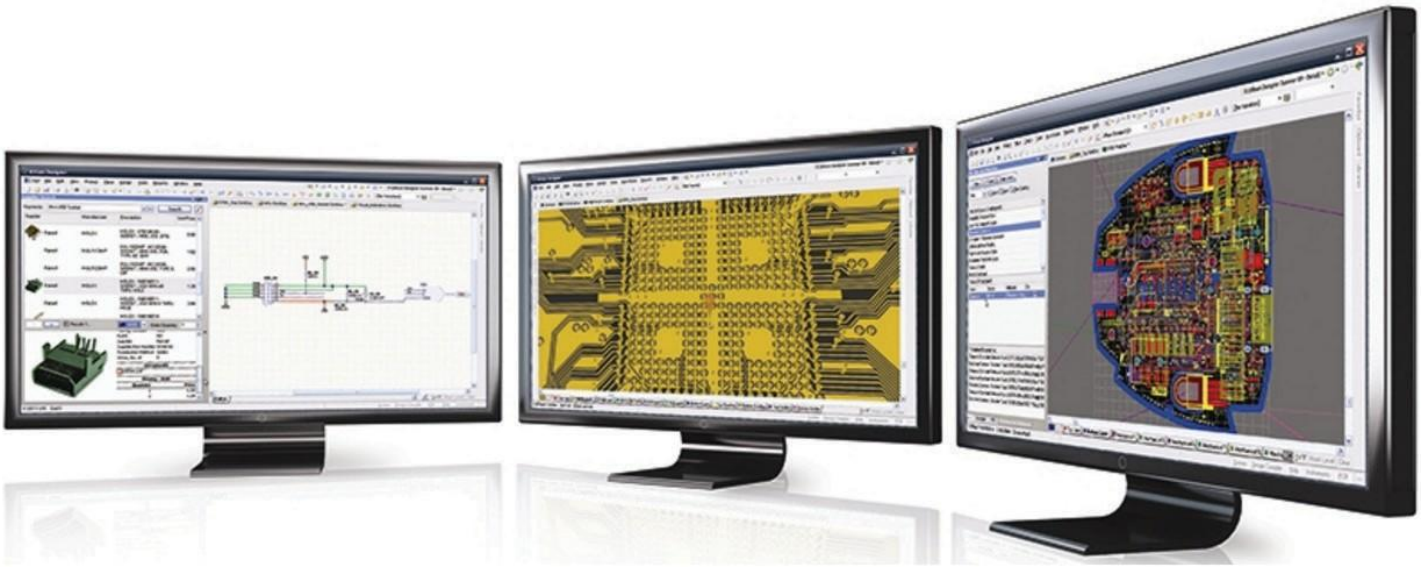
With rapidly increasing PCB product complexity, larger circuits and decreasing sizes, you can no longer accept the inability to meet your goals as normal and acceptable. You require a complete solution: feature rich products, product differentiation, real-time connection to your supply chain, effective 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

Mentor sells three PADS® product bundles. The bundles include independent applications, different data models, design environments, and user interfaces for schematic versus PCB design.

PADS Standard	PADS Standard Plus	PADS Professional
• Schematic Capture • PCB Layout	• Schematic Capture • PCB Layout • Analysis and Verification • Advanced Layout and High-Speed Constraints	• Schematic Capture • PCB Layout • Analysis and Verification • Advanced Layout and High-Speed Constraints • FPGA-PCB Synchronization • Signal/Power Integrity and Thermal Analysis • Sketch Routing • 3D and STEP model support

OVERVIEW OF ALTIUM DESIGNER



Native3D™ powered Unified Design Environment

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.

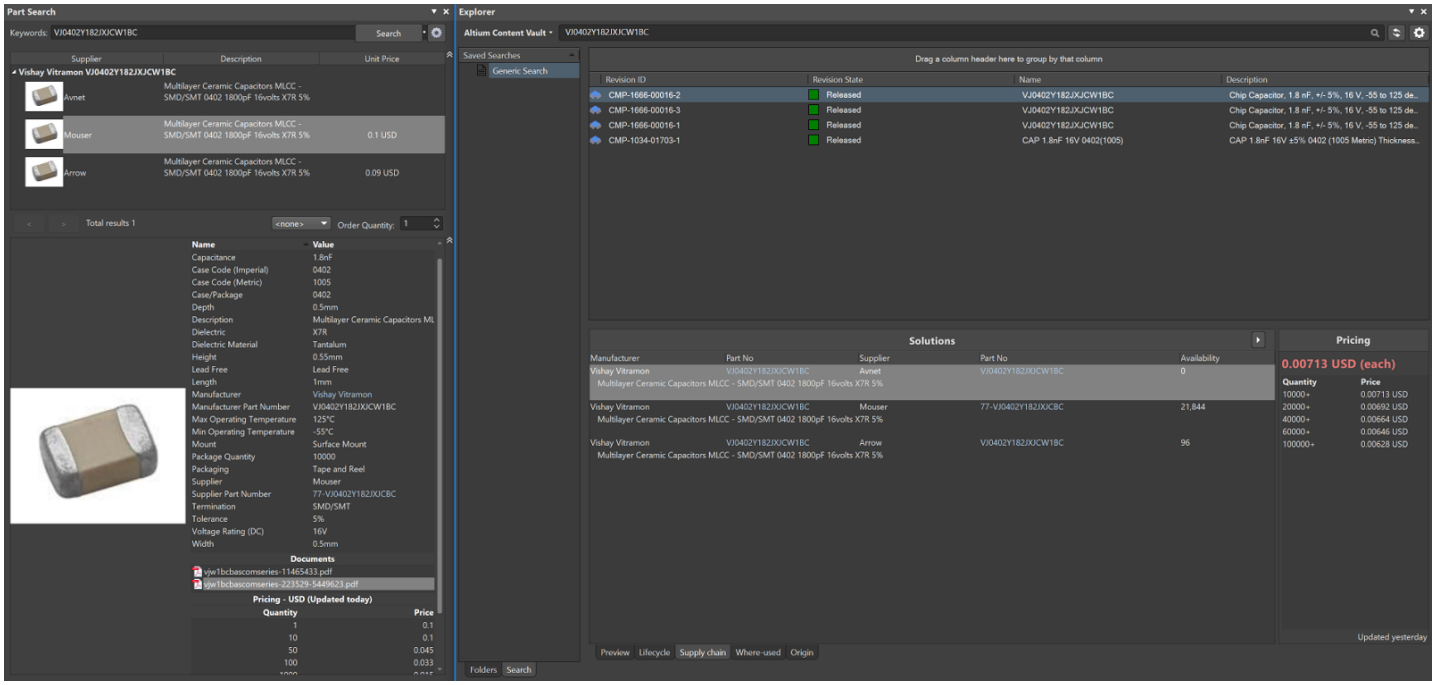
ALL WITHIN ONE, MODERN USER INTERFACE

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

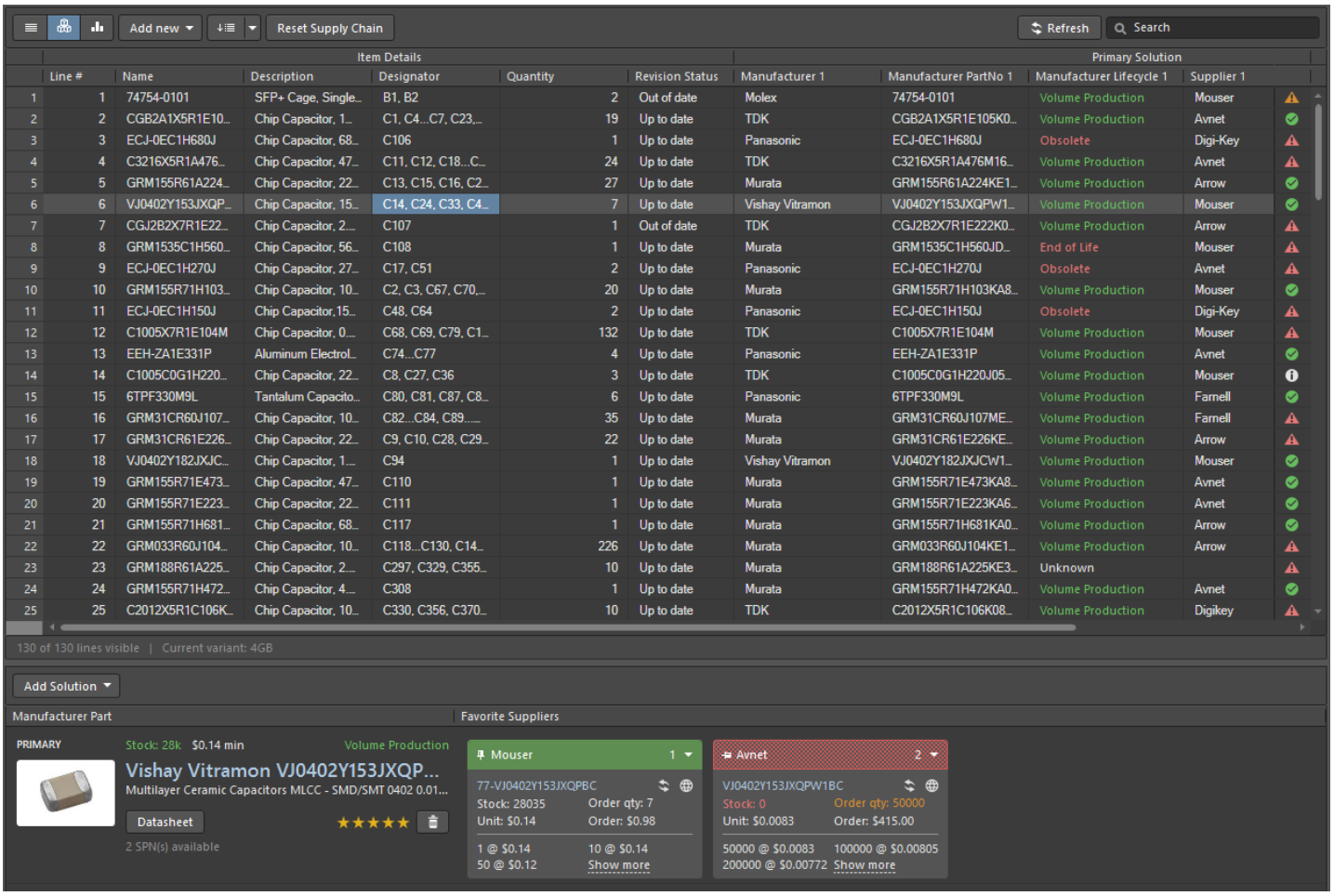


Integrated Real-Time Supply Chain

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.

The Octopart search and linking is fully integrated to your schematic, libraries, and ActiveBOM BOM editor in Altium Designer, working behind the scenes to provide real-time supply solutions and a series of BOM rule checks to prevent you ever going to prototype or production without having the parts you need. And it's all automatic.



ActiveBOM - Real-Time Cost Estimation & Part Availability

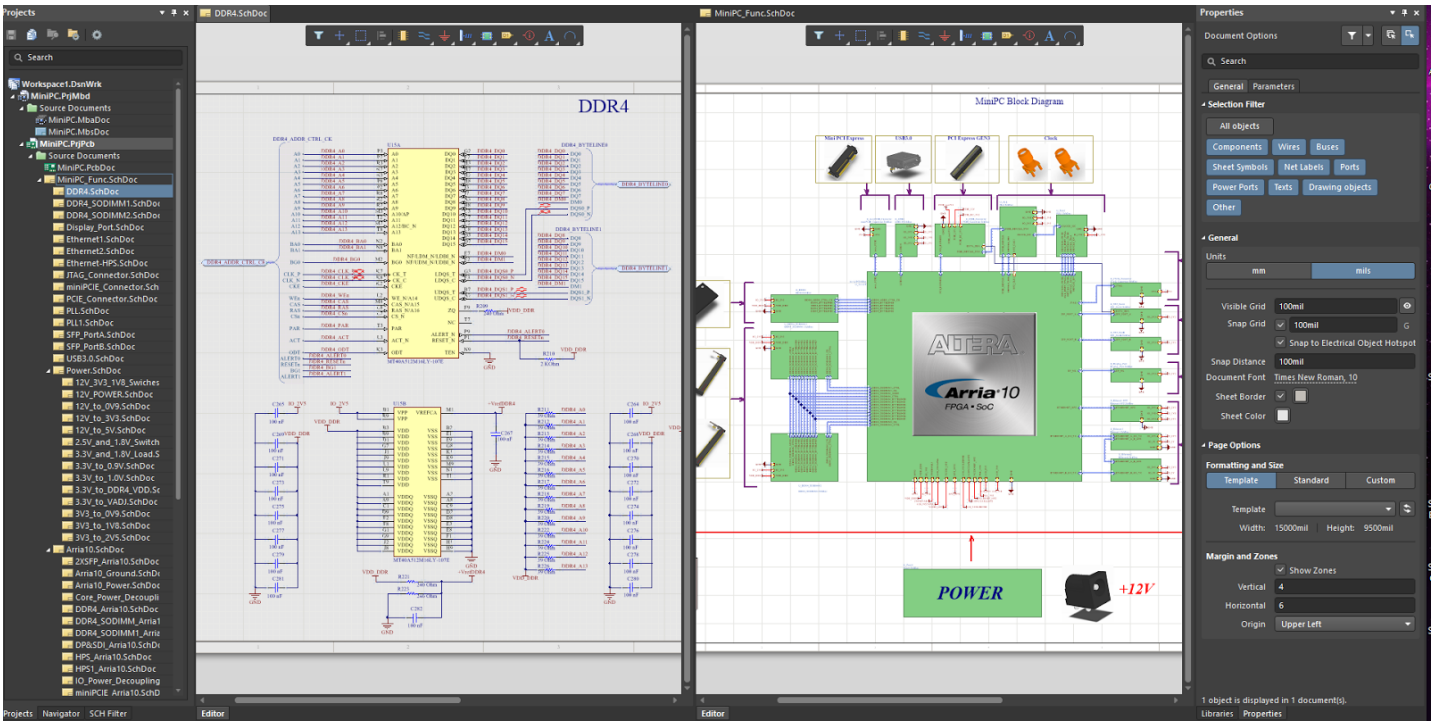
PADS SCHEMATIC CAPTURE APPLICATIONS

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

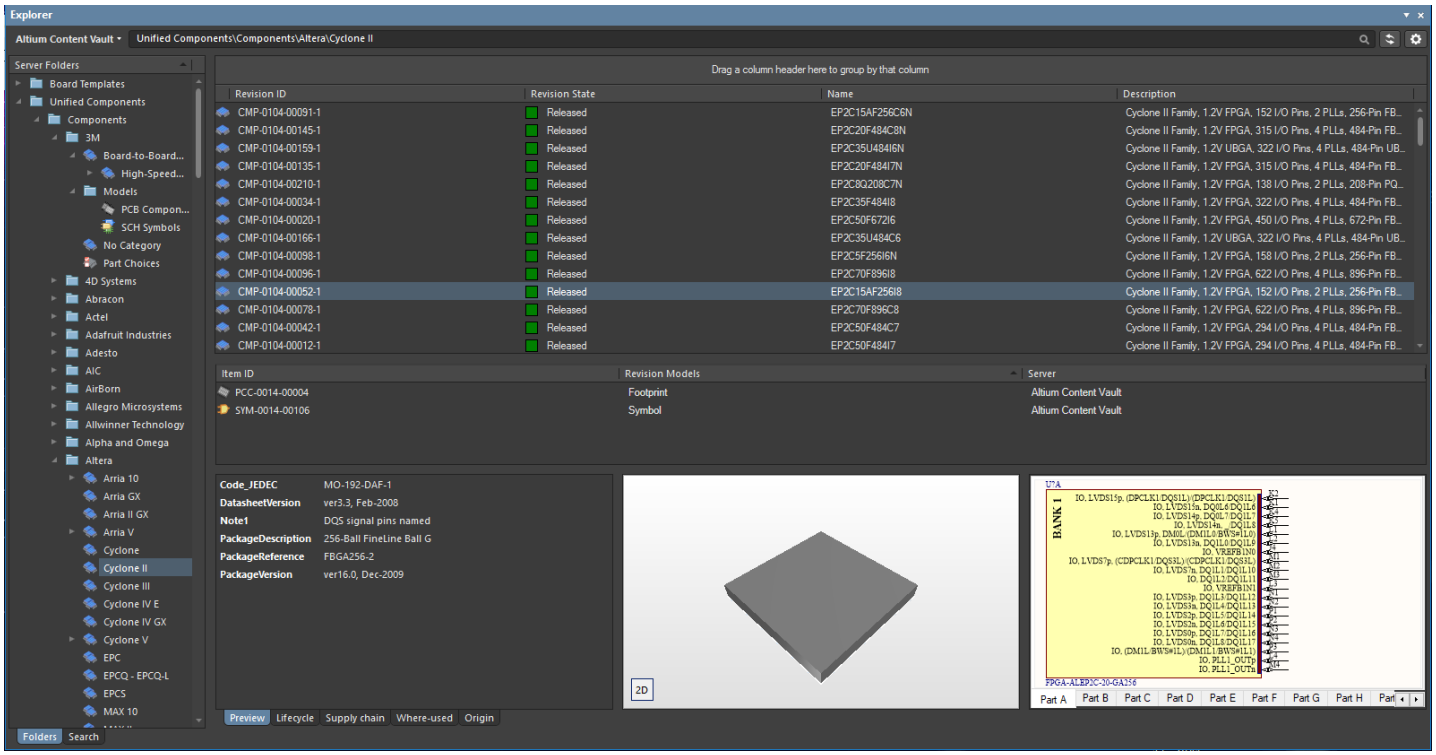
Altium Designer schematic capture technology has been long recognized as a technology differentiator. Engineers and designers will find that Altium Designer schematic features are easy to learn and they will become productive quickly on all designs ranging between relatively simple single sheet schematics 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.



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 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.

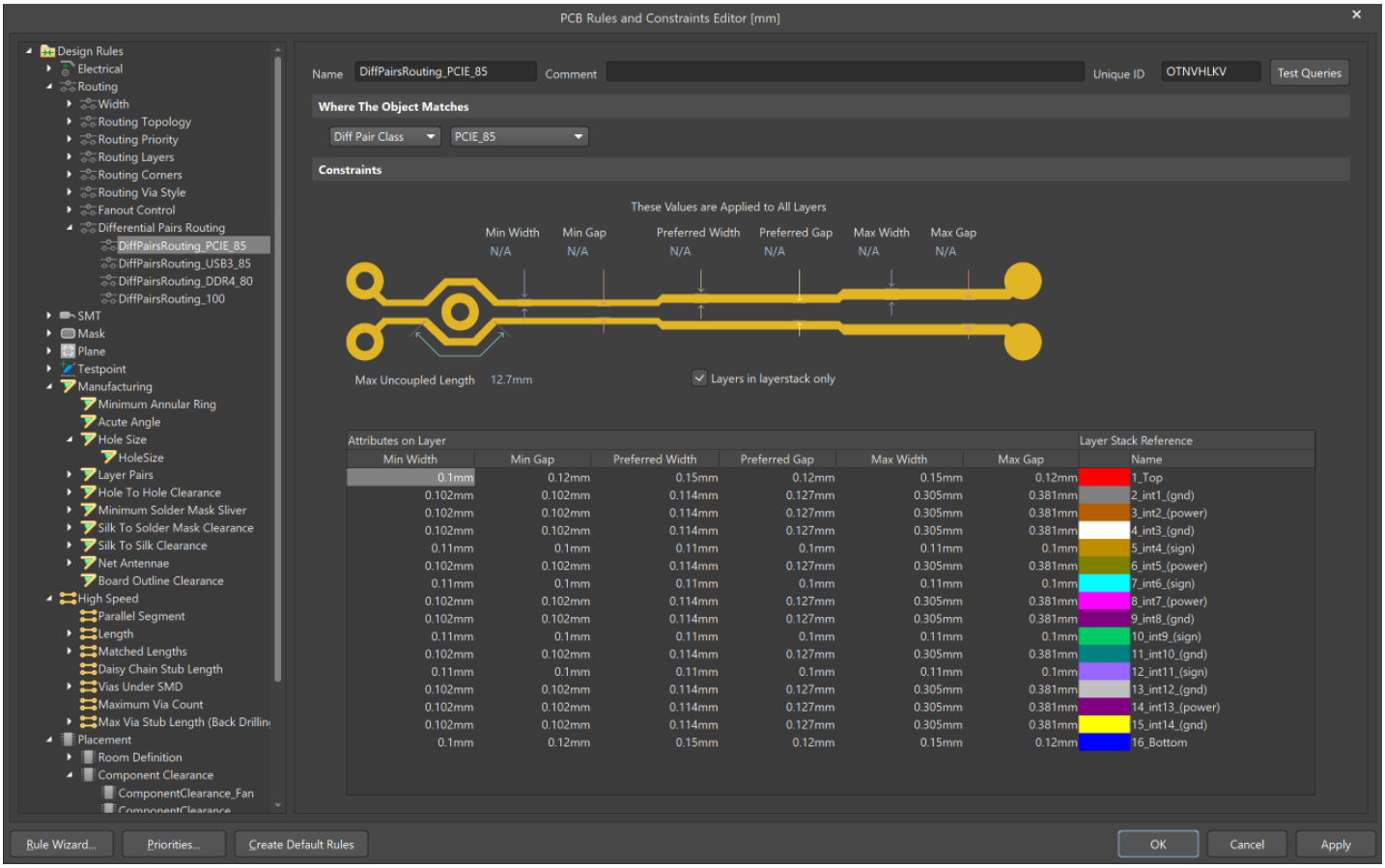


Unified Component Model Management

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



Constraint Driven PCB Design with Design Rule Checking

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.

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 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.

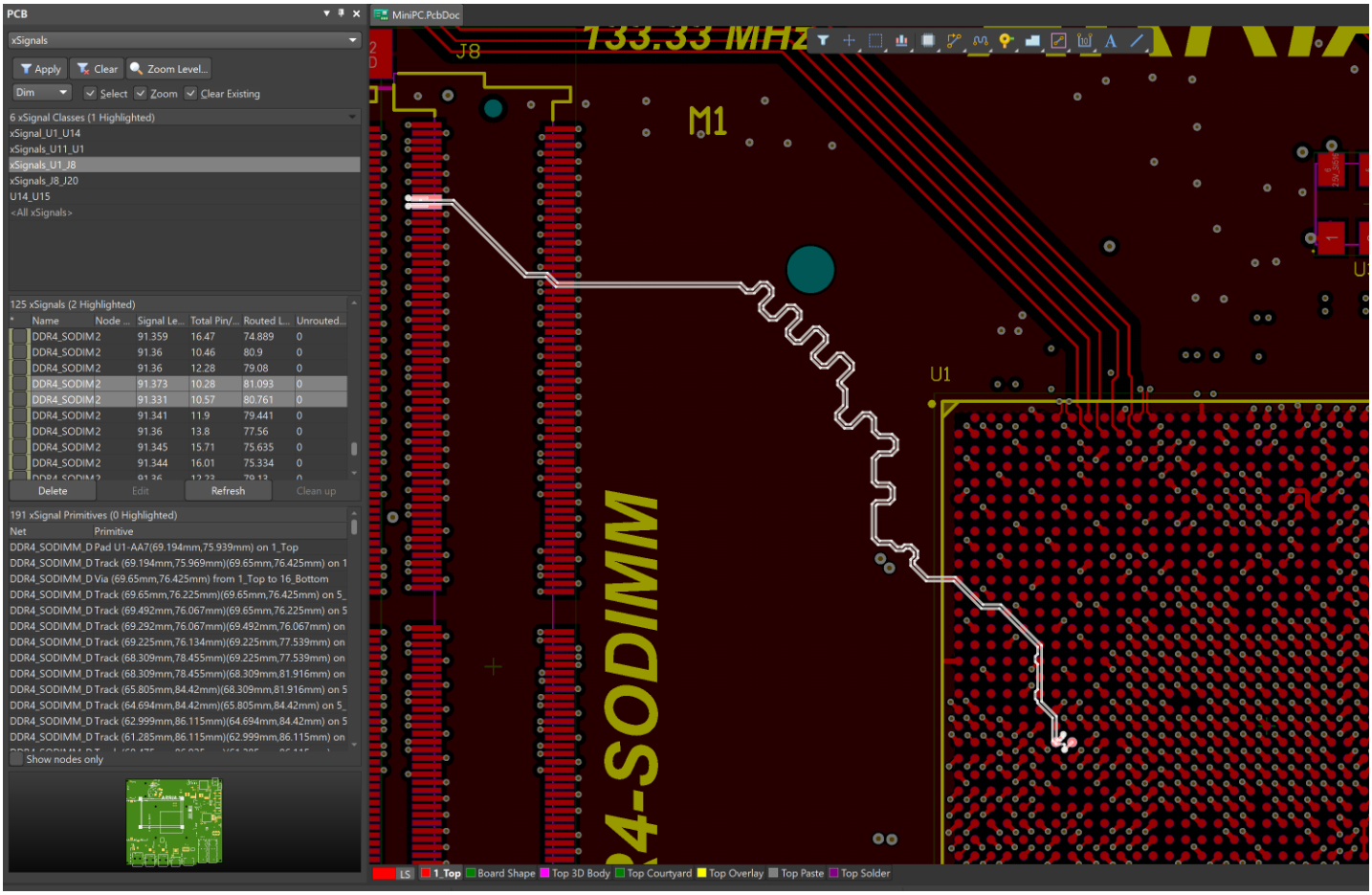
ActiveRoute With Length Tuning - Before and After (<25 Seconds later!)

OTHER DIFFERENTIATING PLACEMENT AND ROUTING FEATURES IN ALTIUM DESIGNER

Powerful 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.

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

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

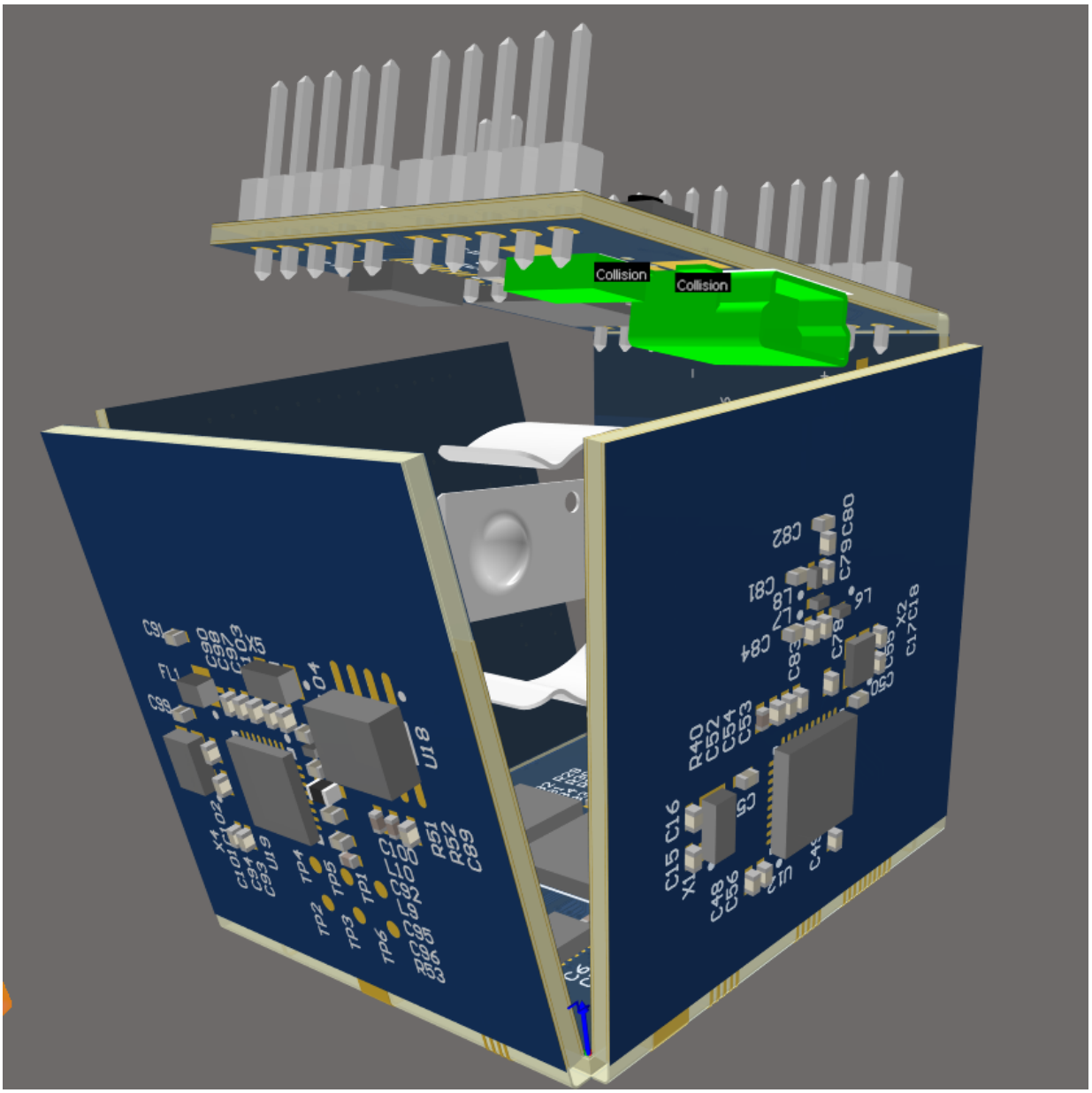
Designer product website for examples of the many benefits to you and the new features recently introduced: http://www.altium.com/altium-designer/whats-new.

3D AND MCAD COLLABORATION IN PADS

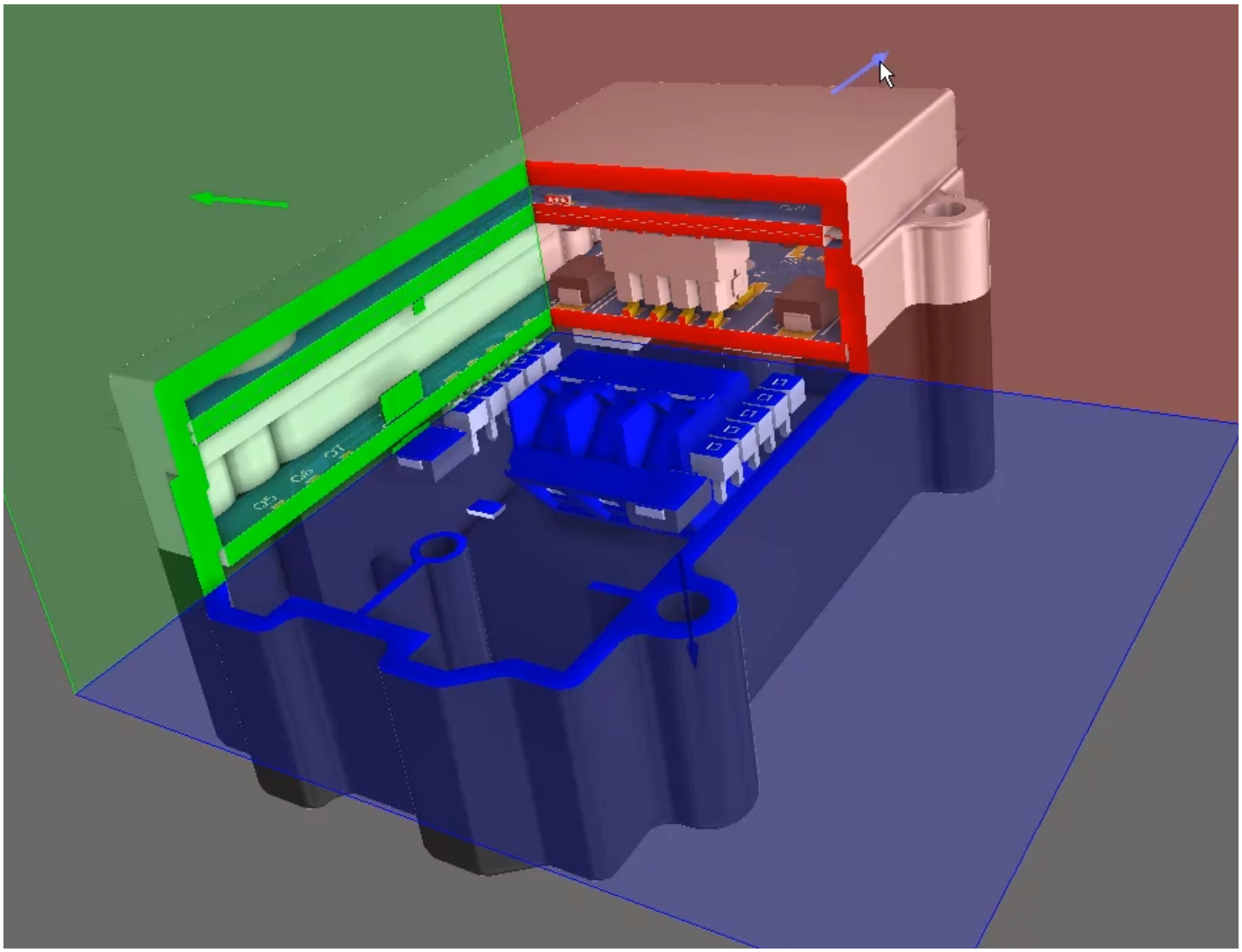
A recent release of PADS included 3D visualization, placement, design rule checking and PDF creation. The PADS application can import STEP models and export PCB assemblies in multiple formats. When evaluating, note that PADS 3D is another separate application with its unique look & feel and UI. Also, 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. PADS Professional recently introduced rudimentary rigid-flex support also, 4 years behind Altium's introduction of the technology.

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

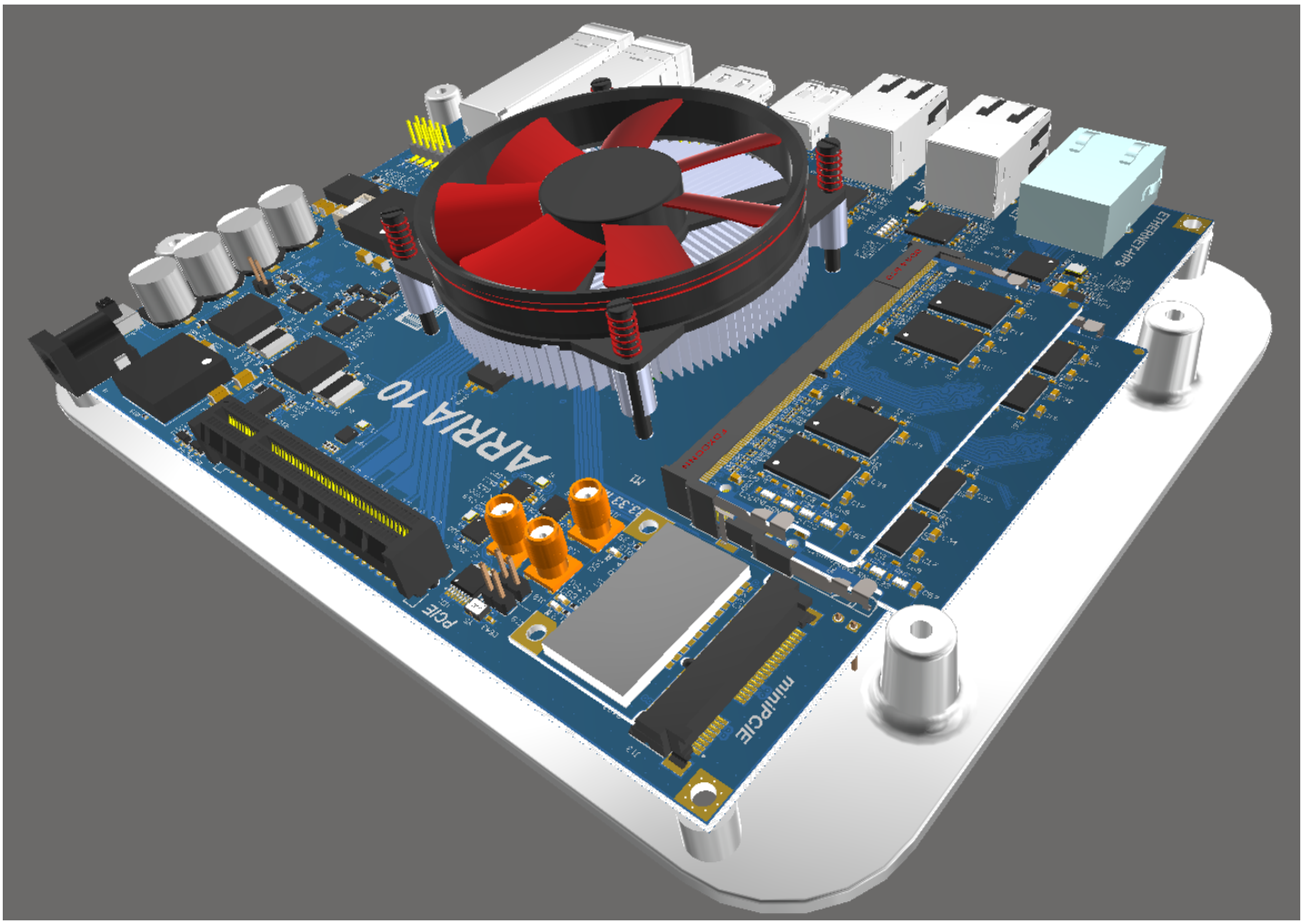


Multi-Board Assembly Enclosure Fit-Check with Cross-Sectional View

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.

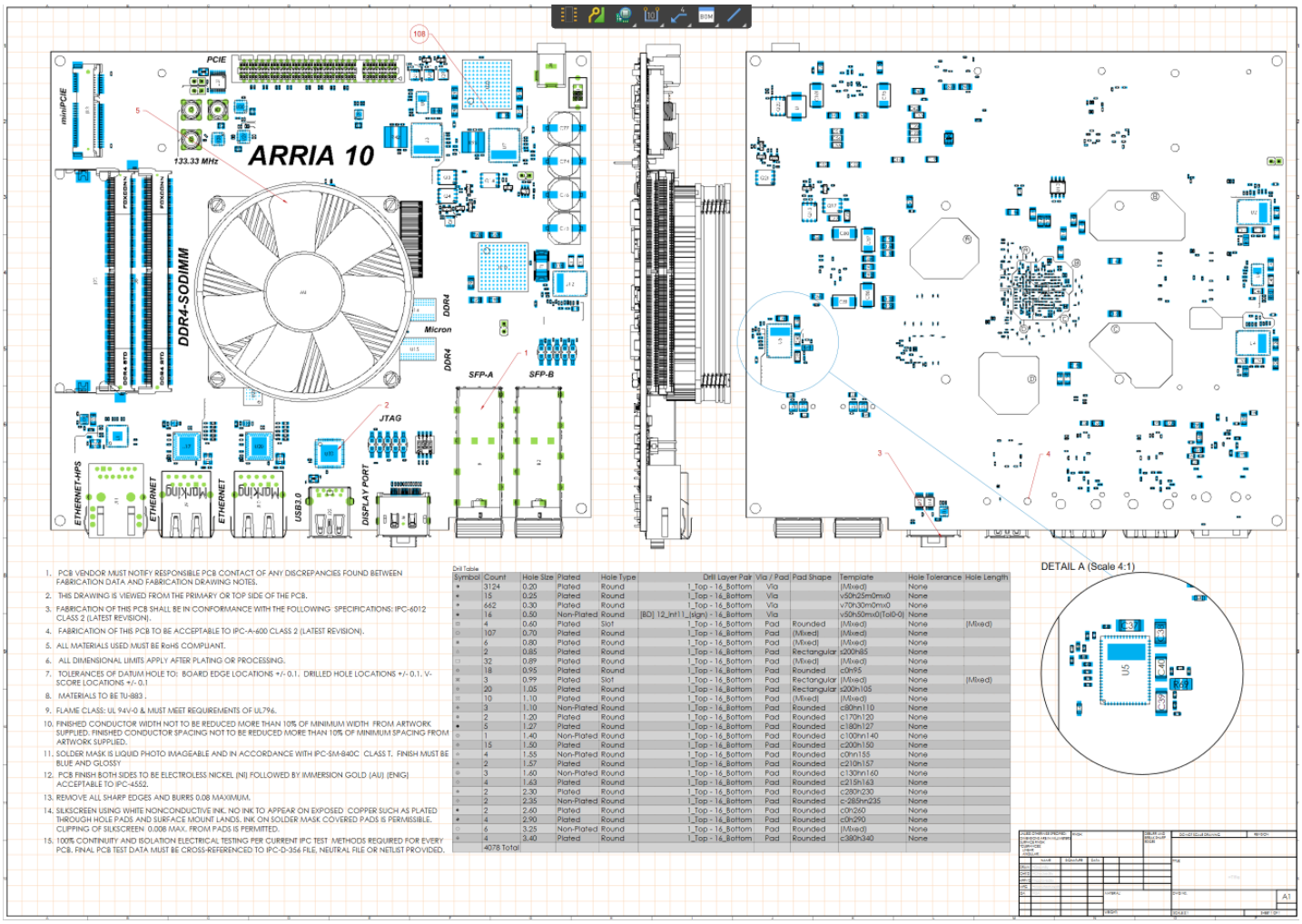
Altium Designer brings you system-level design capabilities to the electronic product development process so you can verify if nets have been assigned correctly, connectors are oriented correctly, plug-in boards fit together, and whether all the connected boards fit into the enclosure. This helps minimize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



Multi-board Assembly with STEP enclosure imported

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

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

PADS® is a registered trademark of Mentor, a Siemens Business.