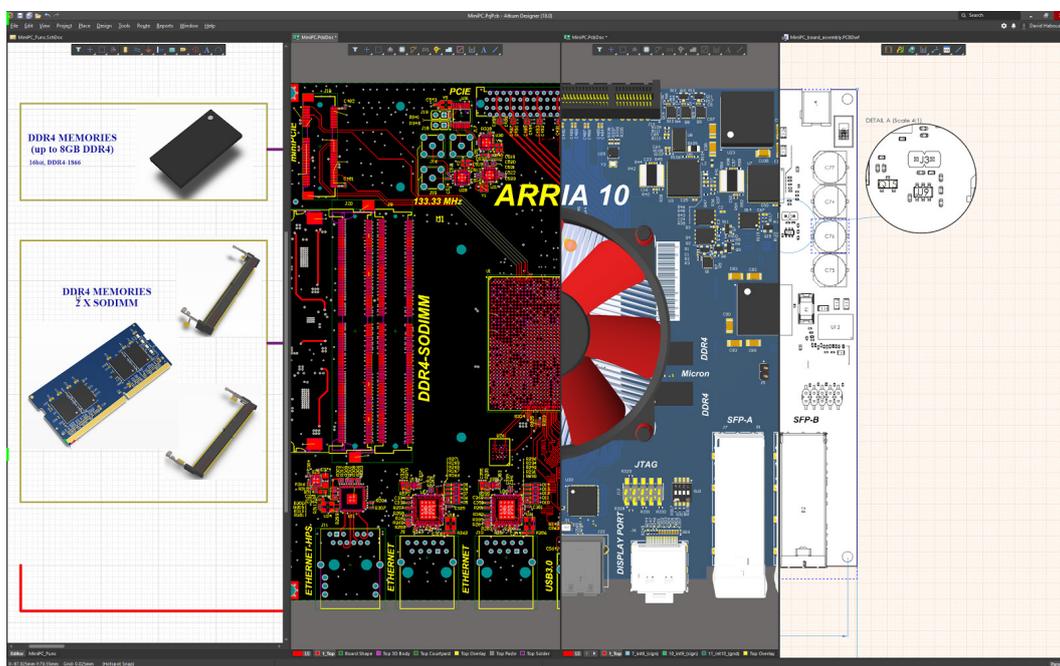


FEATURES AND BENEFITS

- Unified Design Environment with synchronized schematic, PCB, and ECAD data for every project — No data translation needed
- Easily import your legacy projects from various ECAD tools into Altium Designer
- Integrated tools that accelerate the design process and reduce prototype respins
- Remove any performance, stability, and functional constraints with a powerful 64-bit architecture
- Modern design workspace with intuitive schematic entry features ensure a quick and easy design capture



UNIFIED PLATFORM

Traditional concepts of an electronics design system haven't changed much, at least for most EDA vendors. This can be apparent in the way they struggle to meet even the current challenges. The fragmented approach of trying to glue separate solutions together eventually breaks down as more challenges are thrown at them. It's just unsustainable and difficult to innovate on top of inconsistent design applications that rely on constantly unmatched data exchanges. That's not how Altium Designer is built.

Altium Designer has been built from the ground up as a powerful, single-application electronics development environment that contains all the advanced design tools you'll need to capture and simulate hardware, design PCBs, collaborate with MCAD, manage your ECAD data and documentation, as well as manage your designs from concept to production. It's all there in a single, unified design platform.

Altium Designer also unifies your design data, from component models to subassemblies. Unlike conventional design systems based on a collection of linked tool applications, the Altium Designer unified platform allows each design application to access and modify a single, centralized model of the design data. That's one design project; one model of the design data; one development environment, visible to everyone. With this unified and coherent source of design data being available to all the design applications at the platform level, design synchronization and data management becomes a simple process.

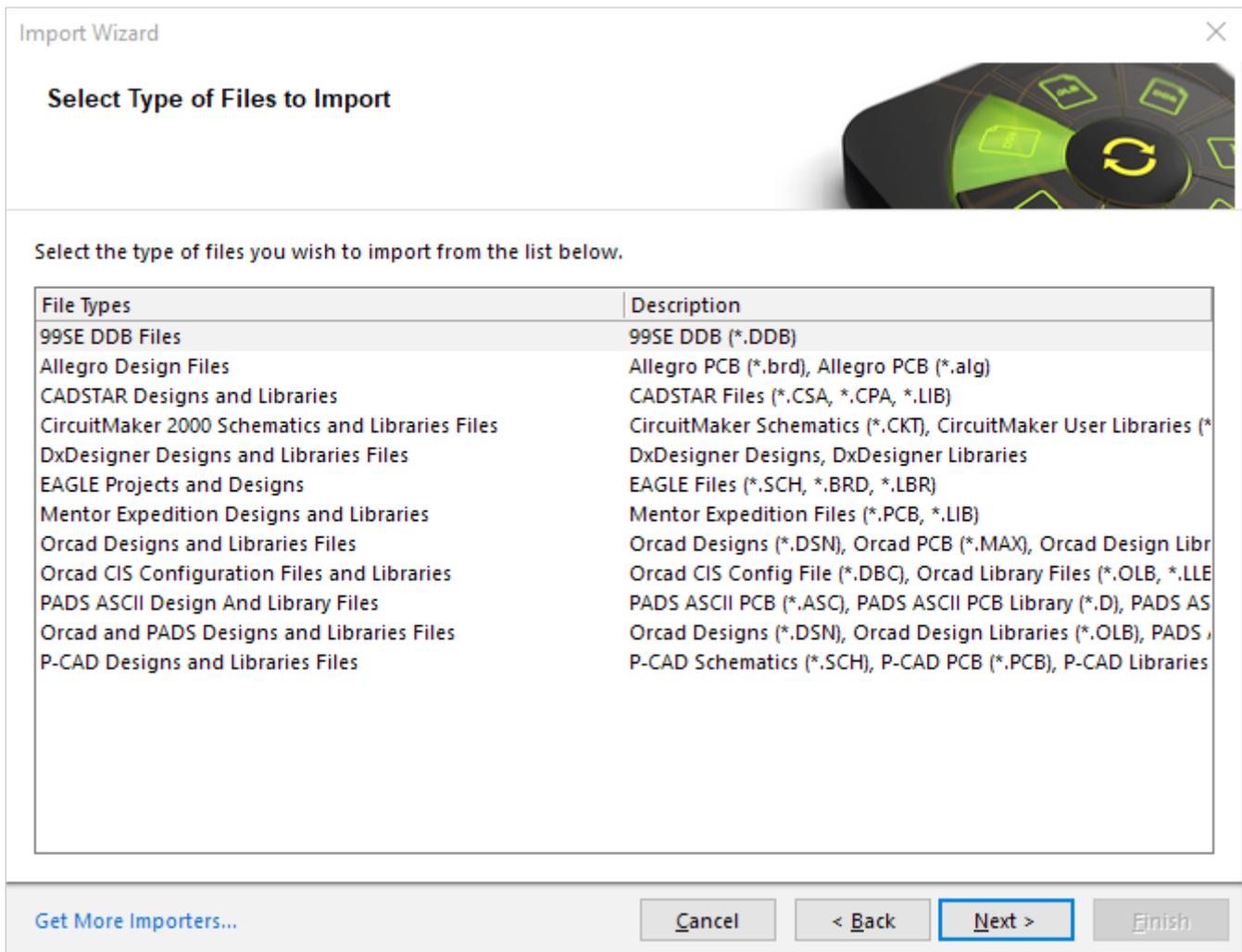


Excited but new to Altium Designer? No need to worry. Altium Designer enables you to import projects from other PCB design packages so that you can start taking advantage of Altium Designer by leveraging your existing designs immediately.

Design Import

While you might think that your switching cost is high because your legacy projects were packaged with an older tool, worry not. Altium Designer includes an easy-to-use importer wizard that can translate most design projects from various ECAD tools. Access your project files without the hassle of re-designing 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}.

Leveraging your old libraries and designs by migrating them into Altium Designer through our versatile importer will give you a head start as you'll be working with your own familiar projects. This will streamline your transition, ease your learning curve, and enable you to switch at any phase of your project without having to worry about starting design projects from scratch.



Project Designs Import Wizard

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.

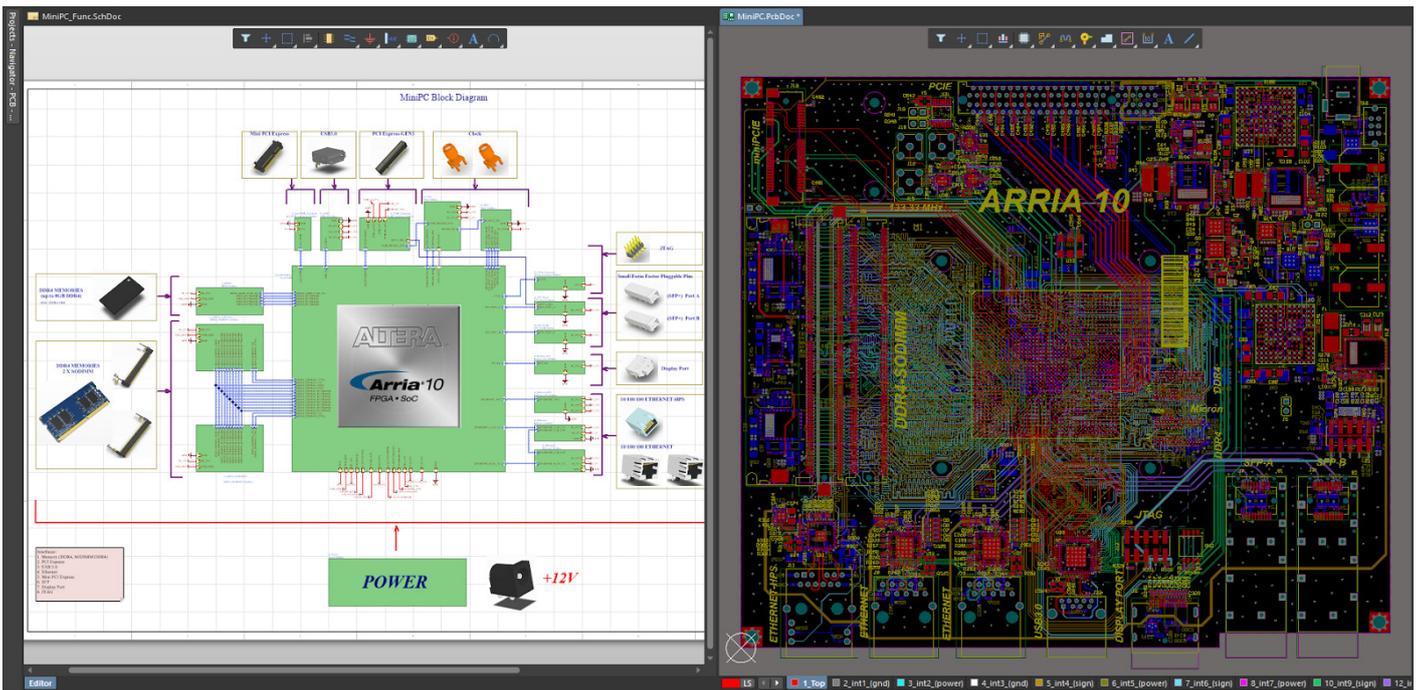
⁴ CADSTAR[®] is a registered trademark of Zuken and Altium claims no rights therein.



Unified Schematic and PCB layout

With the help of cross-probing and a unified dataset, 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 the PCB to give the clearest insight into the implementation of schematics. Altium Designer maintains a two-way connection between schematic and PCB to provide a unified interface throughout the design process, improving productivity and enabling cross-referencing between schematic and PCB layout.

You can place related circuitry quickly and make informed decisions on placement to easily get a successful layout on the first try. Plus, finding specific design aspects is easy with dedicated panels that include all design details.



Unified Design Data Model Environment

Powerful PCB Design

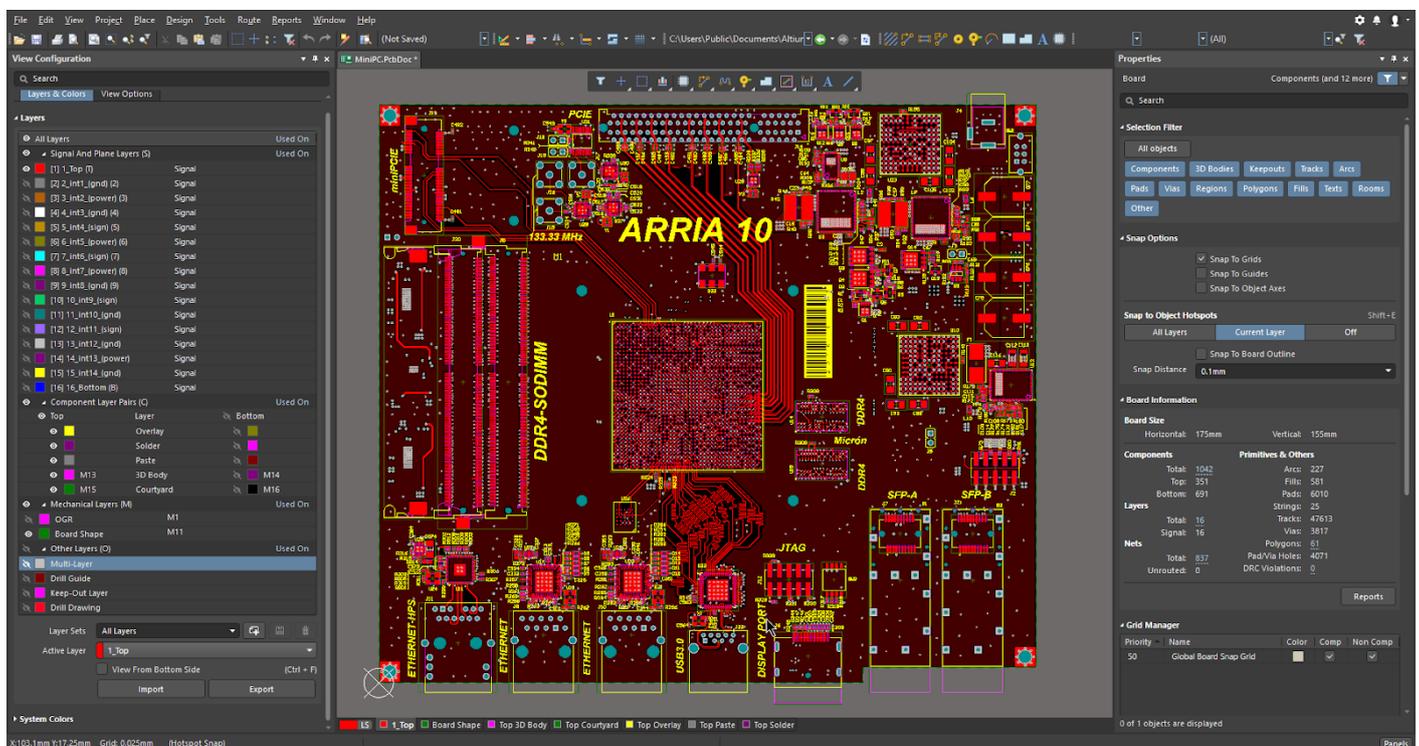
The Altium Designer single application platform coupled with a powerful 64-bit architecture and multithreading can handle any complex, large design with ease. It removes any constraints that might slow you down or get in the way of you completing your PCB design work. Whether you are opening a large file, pouring polygon, running online DRC (Design Rule Checking), or generating your documentation output, it's a simple, easy, and very fast workflow.

With everything you need at your fingertips to simplify your daily workflow, the Altium Designer graphic engine makes better use of GPU power and video card memory. With better rendering, it makes loading of 2D and 3D views much faster, no matter how large or complex the design is. The leveraged heavy lifting done by your GPU also enables instant scene changes from the workplace panels, further enhancing your design experience and productivity. With greater stability, speed, and functionality, you have more time and opportunity to pursue what counts - to create innovative electronics designs that are competitive and unique.

Modern Interface Experience

Most if not all modern EDA tools don't look that modern as their user interface has not changed much in the last decade. They are still not oriented toward simplicity and productivity, with disorganized views, cluttered information, and really old graphics. Altium Designer now delivers a fresh, modern interface that is more logical and streamlined. With updates that greatly improve and consolidate many functions enabling designers to reach optimal productivity quickly and efficiently, without having to go through countless menus, toolbars and dialogs.

With a single, unified, adaptive properties panel, whether in PCB, schematics, or library, you can quickly access document options and their objects. The projects panel now puts all the design files and navigation data within one place, making it easy to find and access the right information about your design structure whenever you need it. And with a powerful graphics engine, the layers and colors panel now gives you unprecedented control over how you view and mask your board, layers, 3D components, and even system level colors. And if you can't remember or find something, simply perform a global search to find any object or command. The new interface in Altium Designer provides a design environment that is cleaner, leaner, and more visually appealing - with all the powerful features and functionality you have come to rely on, made ever more accessible.



Modern Consolidated Interface for a More Productive Workflow