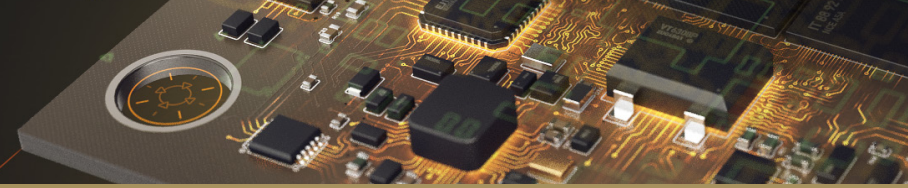


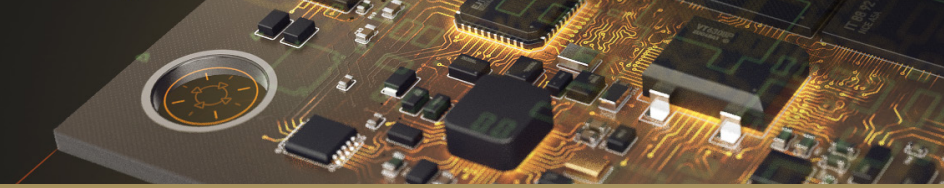
## ALTIUM DESIGNER® KEY FEATURE SUMMARY

| Feature  | Detail  | Benefits  |
|--|---|---|
| <b>Intuitive Interactive Routing</b>                                       | Interactively route your board with several powerful routing options including autoroute, walk around, push, hug and push, and ignore obstacle.                         | Save time when routing your board with powerful and intuitive interactive routing options that allow to precisely control your the organization and flow of your board layout.                      |
| <b>High-Speed Routing Tools for DDR/2/3</b>                                | Define speed signal paths for modern, multi-topology high-speed PCB designs with the xSignals wizard and differential pairs.  | Easily plan and constrain your high-speed designs with xSignals. Route your high-speed design with fully configurable differential pair routings that carry precise signal lengths across your PCB. |
| <b>Powerful Native 3D PCB Editing with Support for Rigid-Flex Circuits</b> | 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 with extended coverlay support. Confirm that your board fits your mechanical enclosure right the first time with real-time clearance checking in Native 3D.  |
| <b>Customizable Design Rules &amp; Constraints</b>                         | Set specific manufacturing guidelines with a customizable design rule system, including specifications for board outlines, solder mask expansions, and drill placement. | Keep your board layout error-free and in line with your specific manufacturing guidelines with a fully customizable design rule system.   |
| <b>Enhanced Layer Stack Manager</b>  | Define and manage all of the layer stacks in your design and account for rigid-flex and complex high-speed stackups.  | Easily manage the layers stacks for your rigid-flex or high-speed design in one central location. Add multiple layer stacks using subsets of materials used in your primary layer stack.            |
| <b>Unified Design Environment</b>  | Unites your design process together between schematic capture and PCB layout with a unified interface and design environment.   | Improve your design efficiency and productivity in schematic capture or PCB layout with the same unified interface and design environment.  |
| <b>Seamless ECAD/MCAD Native 3D Integration</b>                            | Import 3D STEP models and check board clearances with mechanical enclosures in Native 3D.   | Ensure your board fits your mechanical enclosure right the first time without the need for costly prototypes. Check fitting with real-time clearance checking in Native 3D.                         |



## ALTIUM DESIGNER® KEY FEATURE SUMMARY

| Feature   | Detail  | Benefits  |
|---|---|---|
| <b>Cross Probing</b>  | Cross probe between objects on your schematic document and corresponding objects on your PCB.                             | Easily locate and edit similar objects between your schematic and PCB layout, with full cross-probing support for components, buses, nets, pins, and pads.                            |
| <b>Real-Time Supplier Links</b>                             | Link directly to part suppliers with included data for pricing and availability.  | Pick parts for your design that provide the best price and exact availability you need for your budget and time-to-market goals.  |
| <b>Flexible Design Variants</b>                             | Create multiple versions of a board design with modifications to objects and other design elements.                       | Save time producing variations of your original design by creating multiple versions of the same design with different components and other version-specific design elements.         |
| <b>Automated Design Reuse Tools</b>                         | Reuse design elements on future designs with snippets, multi-channel designs, and pad & via libraries.                    | Shave hours off of future projects with several design reuse options, allowing you to easily reuse sections of circuitry or create templates for pad and vias on your design.         |
| <b>Automated Output Job Files &amp; Documentation Tools</b> | Automatically document and configure all required manufacturing outputs with flexible and powerful documentation tools.   | Keep your design data organized and updated with automated output job files and documentation tools. Communicate your exact design intent with Smart PDFs and 3D PDFs.                |
| <b>Integrated Version Control</b>                           | Check design files in and out of a central repository and track changes made to your design.                              | Know exactly who made changes to your design and when with integrated version control. Detailed change logs allow you to compare changes and keep your design on track.               |
| <b>Comprehensive Library Management</b>                     | Add components and schematic templates to your library for later design reuse, or link to your existing company database. | Easily reuse design assets and avoid re-creating components and schematics with an integrated library. Link to your existing company database to use approved objects on your design. |



## ALTIUM DESIGNER® KEY FEATURE SUMMARY

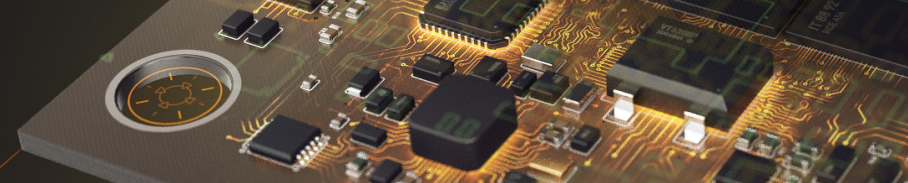
| Feature                                     | Detail  | Benefits  |
|---|---|---|
| <b>ActiveBOM Interactive BOM Management</b> | Get real-time cost estimation and part tracking throughout your entire board design process.  | Make the most informed part selections and meet your project budget with ongoing cost estimations and availability directly in your BOM.  |
| <b>Altium Vault® Integration</b>            | Provides a centralized platform for design data management.   | Take the guesswork out of managing and organizing your design workflow, with a centralized platform for managing design data, projects, and infrastructure.   |
| <b>Design Importers</b> <sup>1 2 3 4</sup>  | Import schematics and PCBs from P-CAD®, EAGLE™, OrCAD™, PADS®, xDxDesigner®, Xpedition®, CADSTAR®, and Allegro® to Altium Designer® | Save time re-creating schematics, board layouts, and associated design data with an automatic importer of project files from P-CAD®, EAGLE™, OrCAD™, PADS®, xDxDesigner®, Xpedition®, CADSTAR®, and Allegro®. |

<sup>1</sup> xDxDesigner®, Xpedition® and PADS® are registered trademarks of Mentor Graphics Corporation and Altium claims no rights therein.

<sup>2</sup> EAGLE™ is a registered trademark of the CadSoft Computer GmbH and Altium claims no rights therein.

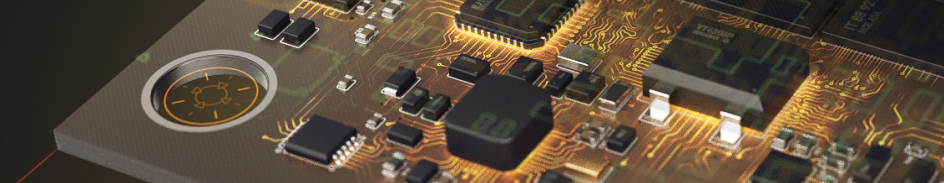
<sup>3</sup> OrCAD™ and Allegro® are registered trademarks of Cadence Design Systems, Inc. and Altium claims no rights therein.

<sup>4</sup> CADSTAR® is a registered trademark of Zuken and Altium claims no rights therein.



## WHAT'S NEW IN ALTIUM DESIGNER® 16.1

| Feature  | Detail   | Benefits   |
|--|--|--|
| <b>High-Speed Routing Wizard for USB 3.0</b>     | Define speed signal paths for modern, multi-topology high-speed PCB designs with the xSignals wizard for USB 3.0.    | Easily plan and constrain your high-speed designs with USB 3.0 with xSignals. Route your high-speed design with fully configurable differential pair routings that carry precise signal lengths across your PCB. |
| <b>Draftsman™: Integrated Documentation Tool</b> | Create complete documentation for fabrication and assembly in your design workspace.                                 | Save time creating and updating assembly and fabrication documentation with a set of powerful and easy-to-use documentation tools integrated in Altium Designer®.  |
| <b>Precise 3D Measurements</b>                   | Measure board layout dimensions in Native 3D for all primitives.   | Clearly communicate design intent to manufacturing by importing 3D layouts and defining measurements between mounting holes and all primitives on your board.  |
| <b>3D STEP Model Generation Wizard</b>           | Generate data-rich 3D STEP models with included parametric data.   | Easily generate the most realistic, accurate, and data-rich 3D models and get an exact representation of your physical board in real-time Native 3D.   |
| <b>Alternative Part Choice System</b>            | Specify pin compatible backup part choices directly in your BOM and automatically substitute part numbers as needed. | Have complete control over your component selection process and avoid any delays when manufacturing your board by specifying backup part choices directly in your BOM.   |
| <b>Streamlined Design Rule Query Editor</b>      | Create advanced design rule queries with a streamlined and intuitive query editor interface.                         | Understand exactly how your design rule queries interact together and avoid any conflicting rules with an organized and intuitive query management interface.  |
| <b>Integrated TASKING Pin Mapper</b>             | Share design data between your PCB and embedded software project.  | Save time translating data between your PCB and embedded software project by instantly sharing pin assignments, processor chip identifiers, and symbol names.  |



## WHAT'S NEW IN ALTIUM DESIGNER® 16.1

| Feature                                | Detail   | Benefits   |
|--|--|--|
| <b>Offline Design System</b>           | Specify network connectivity for specific applications in Altium Designer®.                      | Always remain in complete control of what network data you share with the outside world by specifying connectivity for specific applications including licensing servers, part supplier connections, and more. |
| <b>Enhanced Pin Length Definitions</b> | Instantly calculate pin length definitions to include the internal bond wire of your components. | Accurately and efficiently complete your high-speed design layouts with enhanced pin length definitions in your component pin properties.  |
| <b>Visual Clearance Boundaries</b>     | Visually see clearance boundaries between traces and components as you route your board.         | Clearly understand the impact of your routing decisions in real-time with visual clearance boundaries between traces and components on your board.   |
| <b>Hole Tolerance Definitions</b>      | Define specific hole tolerance values for pads and vias on your board layout.                    | Manufacture the most cost-efficient board by specifying precise hole tolerance values for pads and vias.   |
| <b>Component Placement System</b>      | Dynamically place and drag components that align with other objects on your board.               | 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 on your board layout.                            |

### 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 Altium Designer®, Altium Vault®, CircuitStudio®, PCBWorks®, CircuitMaker®, Octopart®, Ciiva® 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 and Zug and Asia-Pacific locations in Shanghai, Tokyo and Sydney. For more information, visit [www.altium.com](http://www.altium.com). You can also follow and engage with Altium via [Facebook](#), [Twitter](#) and [YouTube](#).