COMPARISON REPORT
ALTIUM DESIGNER® 16 vs. ALTIUM DESIGNER® 17.1

Wherever you work and whatever you create, you can always use the help of the latest technology to remain competitive. Altium Designer 17.1 provides a valuable upgrade to your design workflow including guided routing with ActiveRoute®, backdrilling, and object specific keepouts. The Altium Designer 17.1 release emphasizes stability and design automation improvements to enable new design methodologies with the latest technology. When you have the latest PCB design software innovations at your fingertips, you can focus on making your design innovations a reality. What if you could leverage everything new Altium Designer has to offer as soon as it is available? Altium Subscription enables you to stay at the top of your productivity and design potential with the latest Altium Designer functionality?

Leverage
New Resources

Maintain your competitive edge with access to the newest design methodologies and resources.

Leverage
New Design Automation

Deliver the best products to market first at the lowest possible cost and continue to drive your industry forward.

Leverage
New Technology

Equip yourself to design the technologies of the future with the latest EDA innovations in Altium Designer.

VERSION FEATURE COMPARISON

Feature Name	What it Does	How It Benefits You	Altium Designer 16	Altium Designer 17.1
DRC Flexibility	Exclude design rule violations from further checks and add them to a violations report with author info and reason for violation exclusion.	Exclude formal DRC errors from future checks to enable work-in-progress project release in Altium Vault and combine them in a violations report.		✓
Object Specific Keepouts	Create object specific keepout regions to prohibit and allow the objects you want.	Automatically ensure object types don't get placed in restricted board regions, so that you don't have to manually.		✓
Draftsman® Documentation Platform	Create complete fabrication and assembly documentation in Altium Designer.	Easily communicate design intent right the first time with a complete documentation workflow in Altium Designer.	✓	✓
ActiveRoute®	Fully control your routing workflow with guided routing technology that adheres to design constraints.	Design the highest quality PCB layouts in a fraction of the time with high-performance, guided routing technology that routes on one or multiple layers simultaneously.		✓
Precise 3D Measurements	Measure all board objects while in 3D view.	Precisely measure your board layout in Native 3D PCB and clearly communicate design intent to manufacturing.	✓	✓
Track Glossing	Automatically optimize the length and quality of PCB nets.	Automatically align routing paths without ever having to waste time manually adjusting individual nets.		✓
Dynamic Selections	Define selection areas for PCB objects with any freeform shape or gesture.	Quickly select specific areas or objects on your PCB layout to easily group and edit design objects.		✓
Technology-Aware xSignals Wizard	Automatically detect and setup constraints for common high-speed technologies.	Design accurate high-speed board layouts with automatic configuration for DDR3/4 and USB 3.0 class and match length rules.	✓	✓
Dynamic Copper	Customize copper borders and add/subtract overlaying copper.	Save time customizing your copper polygons with easy-to-use editing modes and customizable borders.		✓
Backdrilling	Create rules for drill sizes, max stub lengths, and start/stop layers for drill holes.	Reduce signal integrity disturbances on high-speed PCBs with complete control over every drill hole.		✓

COMPARISON REPORT
ALTIUM DESIGNER® 16 vs. ALTIUM DESIGNER® 17.1

Feature Name	What it Does	How It Benefits You	Altium Designer 16	Altium Designer 17.1
Real-Time Licensing Reporting	Manage and track design team's license usage.	Easily optimize your design team's licensing usage with real-time reporting and metrics.	✓	✓
Auto Cross-Probing	Automatically cross-reference every net, pin and component on your PCB.	Quickly navigate between multiple files in your design project with cross-referencing for every design object.		✓
3D STEP Model Generation Wizard	Automatically generate 3D STEP models for components.	Create the most realistic, accurate, and data-rich 3D models and get an exact representation of your board in NATIVE 3D™ PCB.	✓	✓
Alternative Part Choice System	Select alternative part choices for obsolete or unavailable components.	Have complete control over your component selection process and include pin compatible backup part choices directly in your BOM.	✓	✓
Draftsman Enhancements	Add new PCB measurements between datums and design objects.	Precisely dimension objects and measurements with improved workflow efficiencies in Draftsman.		✓
Net Color Synchronization	Synchronize nets between schematic and PCB layout.	Ensure documentation accuracy and visual color notifications with net color synchronization in a managed ECO.	✓	✓
Visual Clearance Boundaries	Visually see clearance boundaries while routing.	Clearly understand the impact of routing decisions in real-time with visual clearances between traces and components.	✓	✓
Hole Tolerance Definitions	Add hole tolerances to pads and vias.	Guarantee reliability when manufacturing your PCB by specifying precise hole tolerances in your documentation.	✓	✓
PDF/A Support	Generate ISO-standardized PDF files with embedded annotations and fonts.	Maintain the long-term integrity of your documentation with PDF files that work on any device.		✓
Component Placement System	Automatically align component foot-prints in the PCB editor.	Design the most organized and efficient board layouts with components that push, avoid, and snap-to alignment with other components.	✓	✓
Offline Design System	Specify internet connectivity preferences for Altium Designer applications	Remain in complete control of what network data you share with the outside world with connectivity options for licensing, suppliers, and more.	✓	✓
PCB Component Parameters	Automatically synchronizes component parameters between schematic and pcb.	Easily define specific design rule scopes that can be used for part filtering, design rules, scripts and design variants.		✓
Advanced Component Search	Find components in libraries with customizable query fields and favorites.	Quickly find trusted components for your designs with advanced query options and search favorites.	✓	✓
Advanced Pin Length Definitions	Calculates pin lengths to include internal bond wire.	Accurately route your high-speed design layouts with included pin lengths without having to perform manual calculations.	✓	✓

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®, DXP™, 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.