



KEY BENEFITS	HIGHER PERFORMANCE MADE SIMPLE
<p>Easy</p> <p>Experience the most cohesive, user-friendly interface for you to achieve productivity right “out of the box.”</p>	<p>Altium Designer is the most widely-used PCB design solution and the only cohesive, end-to-end design environment that delivers consistent, easy-to-use upgrades all in one single-price license to ensure electronic design professionals are always supported and equipped for productivity.</p>
<p>More Powerful</p> <p>Attain the power you need to complete large, complex designs - quickly, and with exactitude.</p>	<p>The unified data model in Altium Designer enables rapid, efficient design of new electronic products with a synchronized, rules-driven approach. Streamlined uniformity of the user interface across all editors (Symbol, Footprint, Schematic, PCB, Documentation, etc.) makes the design process highly productive, removing traditional bottlenecks and errors caused by manual design synchronization across tools.</p> <p>Altium Designer leverages over 30 years of EDA software development to provide innovative technologies for PCB design. Additional products can be added to enhance functionality through the Altium Extensions platform. Extensions are analogous to apps on your cell phone and provide specialized capability, such as DC power integrity analysis with PDN Analyzer.</p>
<p>Modern</p> <p>Get the latest design tools today with the guarantee of continuous innovation in the future - year after year.</p>	<p>Automation helps minimize the tedious parts of the job and powerful NATIVE 3D™ MCAD integration makes your digital designs look as real and accurate as the finished product. With Altium Designer, PCB design is the main focus, so you can spend less time worrying about process and more time on design.</p>



Schematic Capture

Every PCB design depends on accurate schematics. Whether you are starting from scratch or working on existing designs, schematic capture is fast and intuitive with our modern user interface. You can import designs from PCB design packages and get started on existing designs immediately. Altium Designer maintains a two way connection between schematics and PCB to provide a unified interface throughout the design process, improving productivity and enabling cross-referencing between schematic and PCB layout.

Design Import

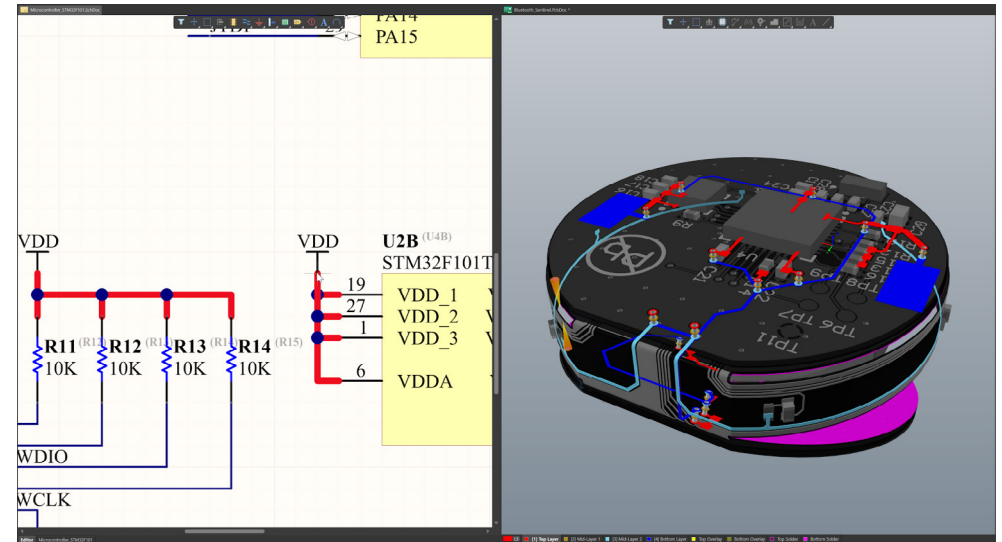
Leverage previous designs reducing time re-creating schematics, board layouts, and associated design data with an automatic importer of project files from P-CAD®, EAGLE®, OrCAD®, PADS®, Xpedition® xDX Designer, Xpedition® Enterprise, CADSTAR®, and Allegro®.

You don't have to start from scratch facilitating switching, enabling learning about Altium Designer with a familiar project, and allowing switching during any stage of a project to Altium Designer.

Unified Schematic and PCB layout

With cross probing and 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.

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



Cross-Probe from schematic to PCB layout

- ¹Xpedition® and PADS® are registered trademarks of Mentor Graphics Corporation and Altium claims no rights therein.
- ²EAGLE® a registered trademark of Autodesk Inc. and Altium claims no rights therein.
- ³OrCAD® and Allegro® are registered trademarks of Cadence Design Systems, Inc. and Altium claims no rights therein.
- ⁴CADSTAR® is a registered trademark of Zuken and Altium claims no rights therein.



Schematic Design Rules

You can add design rule “Directives” to the schematic - nets, wires, busses, harnesses, any component or sheet or document parameter. These are used to drive automated rules for correctly laying out the PCB to help get the board design right the first time.

Examples of this are differential pair definitions and length matching rules for DDR memory routing. Created design rules drive the routing and layout, saving time and also providing guidance from the schematic. You benefit by reducing the number of possible errors and helping identify existing errors, for example, collision errors with an enclosure. You will experience fewer errors and find them quicker reducing manufacturing and respin costs.

Electrical Rule Checks

The Electrical Rule Checks (ERC) in schematic alert you to problems in the design. While Design Rule Checks (DRC) help layout the board correctly and meet manufacturing requirements, the Electrical Rule Check helps prevent you from making design mistakes at an engineering level.

For instance, connecting two output driving sources together will cause a rule violation and associated error message, so you prevent an electrical fault in the final, assembled circuit. You will experience fewer electrical errors and find them faster. ERC also reinforce the design will function correctly once manufactured.

Hierarchical & Multi-channel Design

Electronic devices are generally complicated systems within systems. It's a natural desire to break up the design into pieces like blocks or modules, to “divide and conquer” the design. Also, it is often desirable to re-use specific circuit blocks in different designs, or as multiple channels within the same design. Altium Designer provides a Hierarchical Design

Environment enabling creation of designs at a block diagram top-level, and allowing design projects to be split into manageable logical chunks (ie. Power Supply, Analog Front-End, Processor, IO, Sensors etc.) Hierarchical design also allows you to instantiate multiple copies of the same block when you need multiple channels of an identical circuit (e.g. audio-visual mixing equipment).

You save time on the PCB side by allowing the circuit layout and routing to be automatically duplicated for identical circuits. When a change is made, it can be made to the base logical chunk and the results will propagate through the design. Overall your work and potential rework is minimized and design integrity increases by reusing blocks.

Design Designator Annotation

Annotation is a routine task that you have to perform to detail your work and maintain synchronization between different parts of your design. Annotation is the process of making critical or explanatory notes for the purpose of clarifying details. The most critical form of annotation is the systematic and methodical process for ensuring that each component is uniquely identified. Based on a component's designator, annotation is the primary means of referencing each component.

Annotation makes it possible to ensure that all the Schematic components remain related to their physical PCB implementation. PCB layout changes can result in a reassignment of designators or re-annotation, and these changes must be passed back to the Schematic environment.

Altium Designer automates how annotations are handled, tracked and verified, to keep design data in sync. Design integrity is boosted with synchronization between designators on the schematic and PCB level reducing errors.

Component Management

Last-minute supply issues derail design timelines and increase costs. Component management leverages complete control over the component selection process to avoid any delays when manufacturing your board. Integrated supply chain information combined with specified alternate parts directly in the bill of materials (BOM) minimize the possibility of supply issues.

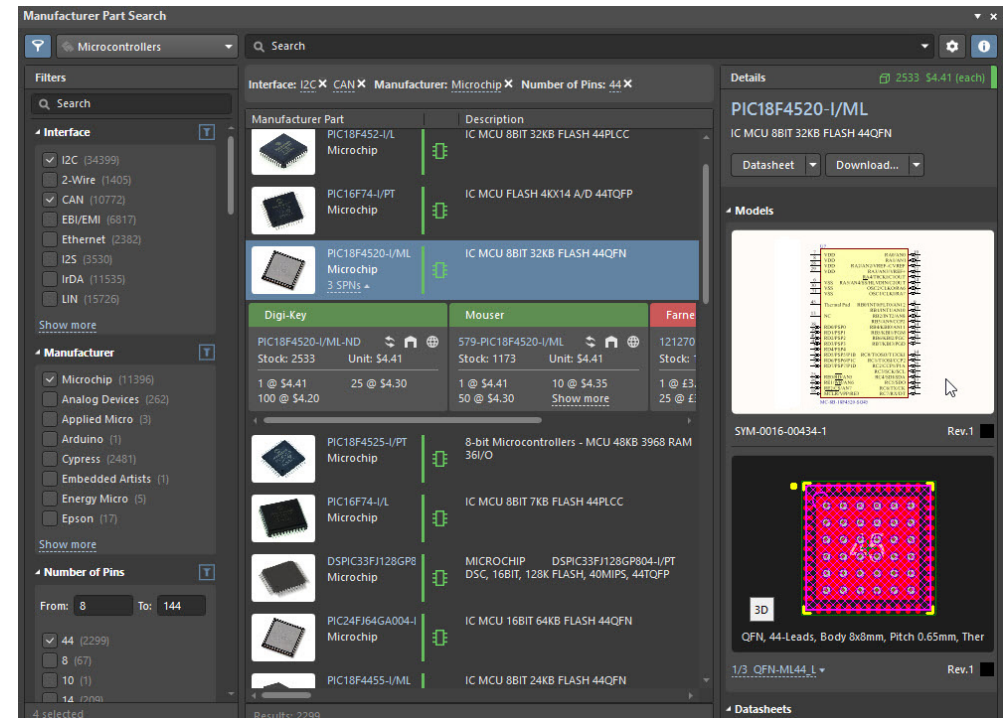
Unified Component Model

Altium Designer uses different Library Types (Schematic, PCB, Database, etc) to define different aspects of a component (Symbol, Footprint, etc) unified by a component library. Unified component models combine all defining information into a single, placeable design part. Maintained libraries exist in the same ecosystem as the design environment allowing direct placement on a schematic and PCB layout level.

Manufacturer Part Search and Components Placement

You can link parts with real-time pricing and availability data from personal parts suppliers and over 100 Altium verified suppliers. You can leverage price, availability, life-cycle status and real-time supplier information to meet their design goals. Utilizing the information reduces risks of last minute costs due to component supply chain issues. In turn, you can plan manufacturing reducing time to market and minimizing unexpected costs and design changes.

With the Manufacturer Part Search, rapidly find which parts are available and at what costs - right at design time. And why waste time making the parts every time? If symbol and footprint models are available in the Altium content ecosystem, you can either acquire them from the parts search panel to your own library for customization, or simply place them directly in your design.



Streamlined part search

Placing parts into your design from anywhere - libraries, data management servers, or the cloud, is a snap. Through a single unified Components panel, all the parts and rapid parametric search and filtering will let you find exactly the part you need, right when you need to place it.

From Altium Designer 19 on, you can connect your engineering desktop to Altium's new cloud based collaboration and data management solution, Altium 365, to keep components organized, gather usage statistics, and update components to the latest revision with the click of a button.



Real-Time BOM Management

The Bill of Materials (BOM) is a list with the necessary parts used in a design for manufacturing.

ActiveBOM provides you with automation by supplying part information such as availability and price from selected suppliers.

Altium Designer allows specifying pin-compatible backup part choices directly in the Bill of Materials (BOM) referred to as Alternative Part Choice. Having pin-compatible backup part choices nearly eliminates supply chain issue risks for manufacturing. In turn, you can design taking into account potential manufacturing blowouts reducing time to market and minimizing unexpected costs and design changes.

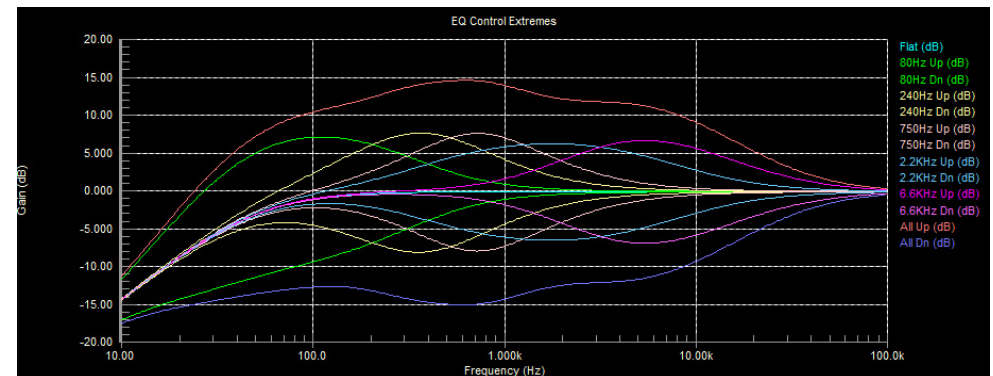
IPC Compliant Footprint Wizards

No need to spend hours copying and pasting pads on BGAs or rummaging for component 3D models. And no need to pay for expensive 3rd party software wizards for building IPC-7351 compliant footprints. All that is built-in to Altium Designer, and with each release new footprint types and updates are added. Any part you need can be rapidly and accurately built to match your own library standards while also meeting the fabrication and assembly quality and layout set forth by IPC-7351, SM-782 and JEDEC (through hole). You can even batch produce whole families of footprints from an Excel sheet or CSV file.

Mixed Simulation

Altium Designer allows you to easily create and manage multiple simulation profiles. Separate profiles allow designers to run different types of analyses with different simulation engines (Mixed Sim, SIMetrix, SIMPLIS). This allows multiple runs of the same simulation type (e.g., AC analysis) with different parameters and options (e.g., different frequency ranges). Active profiles can easily add, remove, edit, run, and/or generate netlists. The Profile Manager organizes profiles and uses probes or active nets to select waveforms to display.

All simulation results can be saved with other manufacturing outputs for conveyance to manufacturing. You can convey design intent to the contact manufacturer minimizing errors.





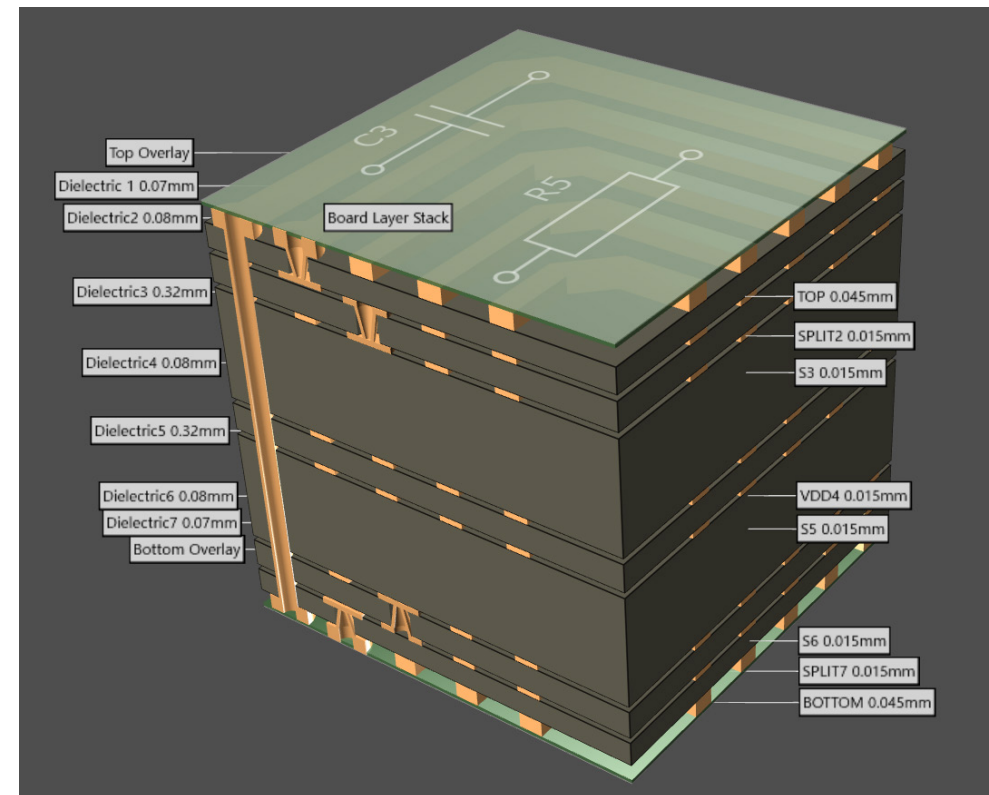
Board Layout

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 and pads on your board layout. These features make laying out dense boards much easier, and maintain design rule compliance as well. Signal integrity disturbances are reduced on high-speed PCBs with complete control over every drill hole with hole tolerances and back drilling capabilities.

Visual Layer Stack Management

Layer stack management allows you to define the material composition and specialized regions on the board. For flex circuitry, rigid-flex and embedded technology PCB designs, you can control the entire stackup, including all rigid and flex portions with bending angles and individual layer definitions. You can visualize layer stacks using subsets of materials used in the primary layer stack. Each layer has an individual definition and corresponding parametric data from the Materials Library.

Complex boards with multiple stackups can be defined side by side to facilitate board construction. You define and manage all of the layer stackups in a central location to facilitate tracking of layer stack details and minimizing errors and miscommunication on layer details.

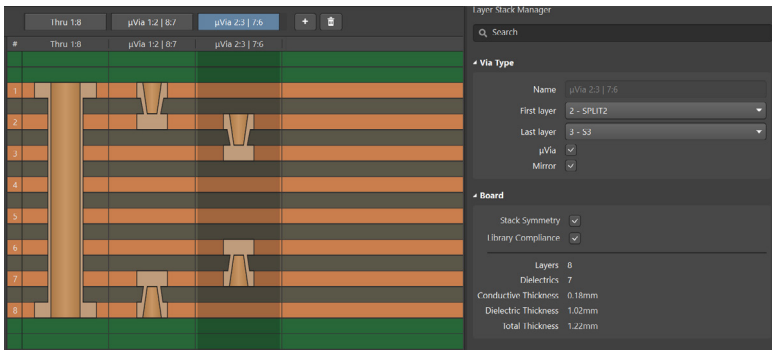


Layer Stack Manager



Materials Library

The Materials Library contains system-wide parametric data on any materials you may specify in the construction of your layer stack. From dielectric cores and prepregs with different base thicknesses and glass weaves, to electro-deposited and rolled coppers, to glue pastes, conductive inks, and films - it's all there. And you can create any new materials you intend to use, or use the set of generic ones already there, and the materials library can be saved and loaded to XML - so you can share it with the entire team.



Fearless HDI™

Fearless HDI (High Density Interconnect) is our technology for enabling accurate design and visualization of HDI structures, including laser-drilled and mechanical microvias, stacked microvias, skip-vias.

A dedicated Layer Stack Manager tab allows you to define what µVia, blind/buried and other via structures you can allow in your project, and control the layer stack symmetry.

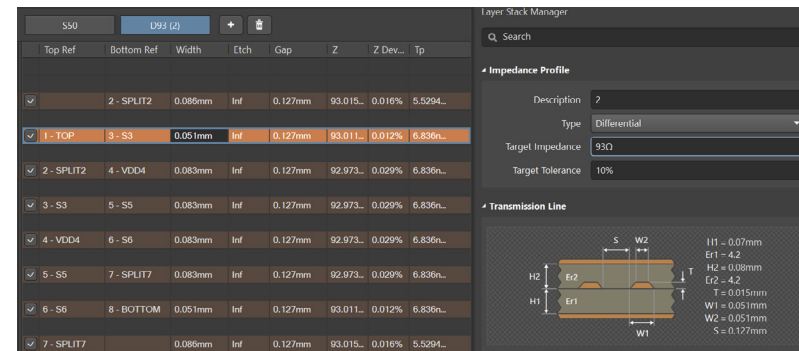
During routing of the board, the HDI via structures can be chosen interactively or used for fanning out parts on surface layers. Changing layers can automatically stack them as defined in the layer stack manager.

Impedance Profiles

More and more modern designs require accurate control of trace impedances (single and differential pair) to maintain signal integrity. USB 3.0, Type C, DDR4/5 and RF/µwave designs all require accurate planning of the trace widths, gaps, and layer stack in order to maintain correct transmission line impedances across the board.

Impedance profiles in the Layer Stack Manager use a fast, accurate 2D field solver to calculate impedances from trace widths, and trace widths and spacings from impedance. Single-ended and differential microstrip or striplines including asymmetric planes can be specified using plane layers or signal layers for references.

Impedance profiles are then used in design rules to automate the correct trace or pair characteristics during interactive routing, or to test for impedance within the desired tolerances. The hardware designer can drive it either way for accurate results.



Panelization

Panelization is the process of placing several designs on a single board for manufacturing. The boards break apart and can be used separately. The process is generally used for smaller, less complex designs and the panel can contain the same design or various small designs.



Rules Driven Design

You can easily define design rules and rule prioritization hierarchy. Design rules compliance as you progress keep the layout manufacturable and enable design intent communication to manufacturing based on part parameters. Design rules check for issues as you are designing, preventing issues before they stack up. Specific manufacturing guidelines with a customizable design rule system include specifications for board outlines, solder mask expansions, drill placement, and an advanced query editor to create non-standard rules.

With design rules, you can complete right-first-time designs. Created design rules drive the routing and layout, saving time and also providing guidance for the PCB designer. You benefit by reducing the number of possible errors and helping identify existing errors, for example, collision errors with an enclosure. You will experience fewer errors and find them quicker leading to a reduction in time to market, reducing manufacturing and re-spin costs, and increase in design integrity.

Copper Pour Management

Copper regions are used to make connections in a PCB. They are commonly used to create power planes and signal planes to connect to components and can be used to help with heat distribution. Stand-off regions are required distance from one copper region to another to prevent interference between the connections. PCB designers generally use filled copper regions to cover the remaining area outside those tracks, pads, and stand-off regions. You can control the placement (pour) order and temporarily disable (shelve) regions to make it easier to see underlying PCB layout.

In Altium Designer areas of copper can be defined using three different design objects: Fills, Solid Regions and Polygon Pours. The advantage of a Polygon Pour is that it automatically creates stand-off regions to copper objects that belong to another net based on Design Rules. The automation of copper region placement increases design integrity and productivity and minimizes design errors.

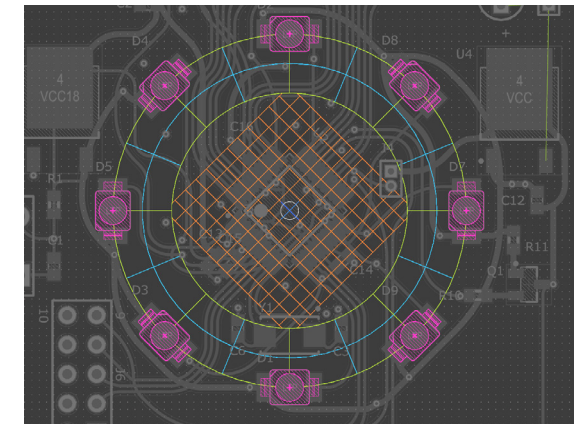
Precise Object Placement

There are always board areas that require special considerations and rules to maintain a viable design. Altium Designer incorporates the use of rooms, keepouts, and polygon regions to aid you when designing. Rooms are regions that assist in the placement of components by grouping them inside the designated area. On the other hand, keepout regions act as an 'interference' object that prevents other copper objects from intersecting its area, as specified by the global clearance rule. Lastly, polygons act as regions in the board to dictate allowable copper pour areas, as well as defining copper pour order.

Precise placement defines the PCB layout to facilitate routing and avoid physical collisions. You can minimize errors, complete designs faster, and get the product to market faster. The added benefit of precise fit to minimize risk of re-spin and maximize first pass manufacturing.

Advanced Snap Points and Grids

Grids define spacing for the entire board. Snap points are defined locations in 3D space that make it easy to align 3D bodies (ie enclosure to board or mounting holes). Combining customizable grids, definable snap points, and 3D body/Pad alignment, every aspect of a user's design is placed with complete precision.



Precise placement defines the PCB layout to facilitate routing and avoid physical collisions. You can minimize errors, complete designs faster, and get the product to market faster. The added benefit of precise fit to minimize risk of re-spin and maximize first pass manufacturing.

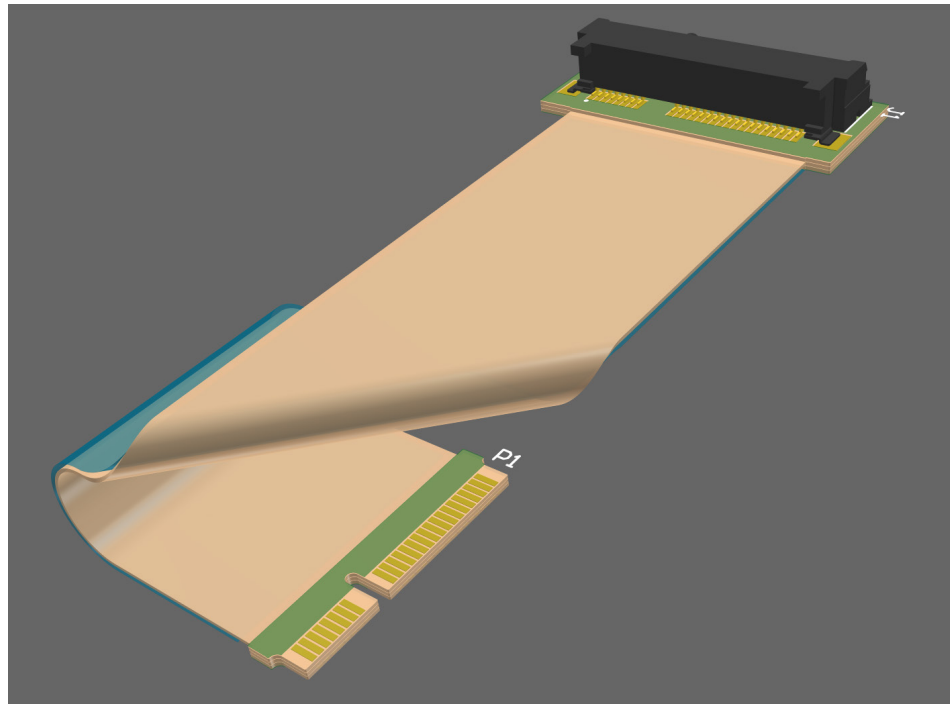


Rigid-Flex and Multi-Board

Altium Designer makes it simple to define and modify the shape of a board.

NATIVE3D™ gives a realistic view of what a manufactured board will look like when produced. This lets you view, rotate, and translate your design so that you can visually check all clearances.

Layer Stack Regions allow a single board to be composed of multiple materials, flexibility, and thickness, enabling design of Rigid-Flex-Rigid PCB assemblies.



Multi-Board and Rigid-Flex Design

Rigid-flex regions with bend angles

You can add rigid-flex sections to the PCB design with bikini coverlay support and check clearances with NATIVE 3D. All board movement is controlled with definable angle, radius, and fold index for rigid-flex regions. One of the most valuable features lets you create full-motion video clips from board 3D snapshots to convey design intent to manufacturing.

This is critical in designs that use multiple folds to fit in tight spaces or allow movement of the flex circuitry. You can gain confidence in flex and rigid-flex PCBs by confirming board fits in a mechanical enclosure without the financial and time cost of a physical prototype. Reduced prototyping costs resulting in reduced time to market and manufacturing/repin costs.

Interconnected Multi-board Assembly

Multi-board design means design of unitary electronic systems that have multiple boards or modules housed together and electrically interconnected in some way sometimes referred to as modular design.

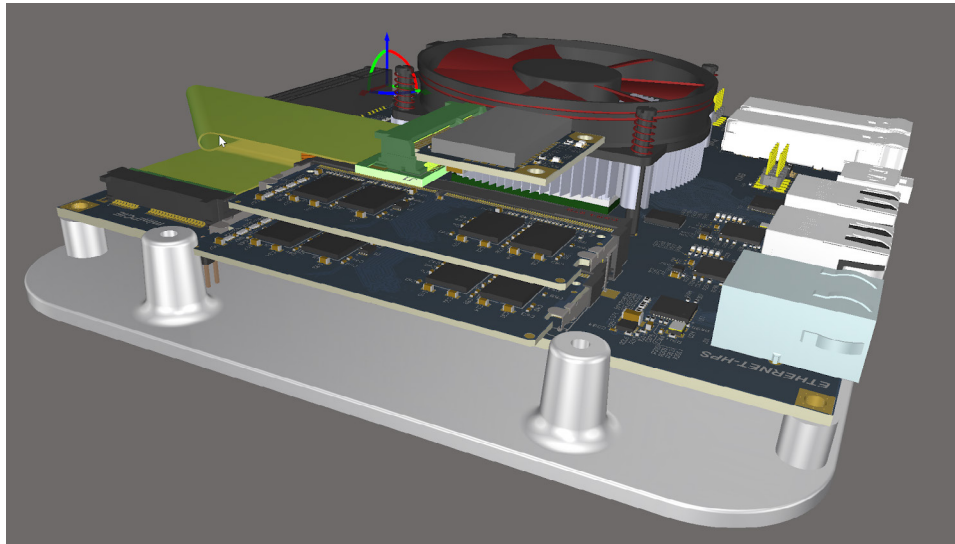
Modular design effectively means developing each board in the multi-board system as a complete functional unit. Each module forms a distinctly reusable object (logical and physical) within the broader system. A good example of this would be the DDR4 SDRAM memory modules within a server or desktop PC.

You gain efficiency and have fewer design errors when developing multi-board PCB based electronic systems in Altium Designer. Automation helps define the way boards will interconnect and the signals that go between them. Reduced errors, time to market, and costs and increase in design integrity.



MCAD Collaboration

Collaboration is essential for a successful design process. Altium Designer connects your PCB design workflow to the complete engineering ecosystem with powerful MCAD collaboration features and fully managed library management systems. Finding out your board interferes with a connector at the prototype phase is a costly error. Ensure your board fits the mechanical enclosure right the first time with NATIVE 3D clearance checking.



Multi-Board and Rigid-Flex Design

Perfect fit mechanical enclosures

The main offering to ensure fit, NATIVE 3D capabilities, allow clear 3D modelling with real time clearance checking. ECAD/Mechanical CAD collaboration is simple because we can use popular mechanical models directly in the design environment.

You import 3D models and mechanical enclosures from MCAD systems to have the most realistic, accurate, and data-rich 3D models for an exact representation of a physical board without the need for costly prototypes. The IPC-compliant PCB Component Wizard handles any other 3D model needs with a guided process for custom component creation. You get reduced prototyping costs resulting in reduced time to market and manufacturing/repin costs.

Mechanical Model Import

Altium Designer Libraries, PCB Documents, and Multiboard Assemblies all support import and creation of 3D solid models using STEP. Parasolid is also supported with the optional MCAD Collaboration Plugin.

Even the 3D PCB shape can be imported from a mechanical CAD (MCAD) model, via STEP, DXF/DWG, IDF and IDX Baselines. IDF and IDX are not only used for modifying the board shape, but can relocate and re-orientate moved components, allowing bi-directional synchronization between mechanical and PCB layout.

MCAD import into Altium Designer provides more than just a slick visualization of your PCB before you've made it. It allows the PCB design rules engine to immediately inform you of component body collisions and clearance violations in the 3D space of the assembly.

MCAD import of enclosure models allows the PCB designer to use key features of the 3D enclosure and other mechanical objects for alignment and for creation of the board shape from a projected surface.

Interactive Routing

You can design the highest quality PCB layouts in a fraction of the time with an advanced routing engine that includes push and shove, hug, walk around, and interactive length tuning modes for single and differential pair routes. Fully configurable single and differential pair routing in Altium Designer also provides phase and delay tuning across terminations and complex topologies using xSignals™. ActiveRoute® lets you control where and how much automated routing assistance you want on a net connection level up to your whole design. Visual clearance boundaries between traces and components on your board let you visualize design rules and understand your layout at a glance.

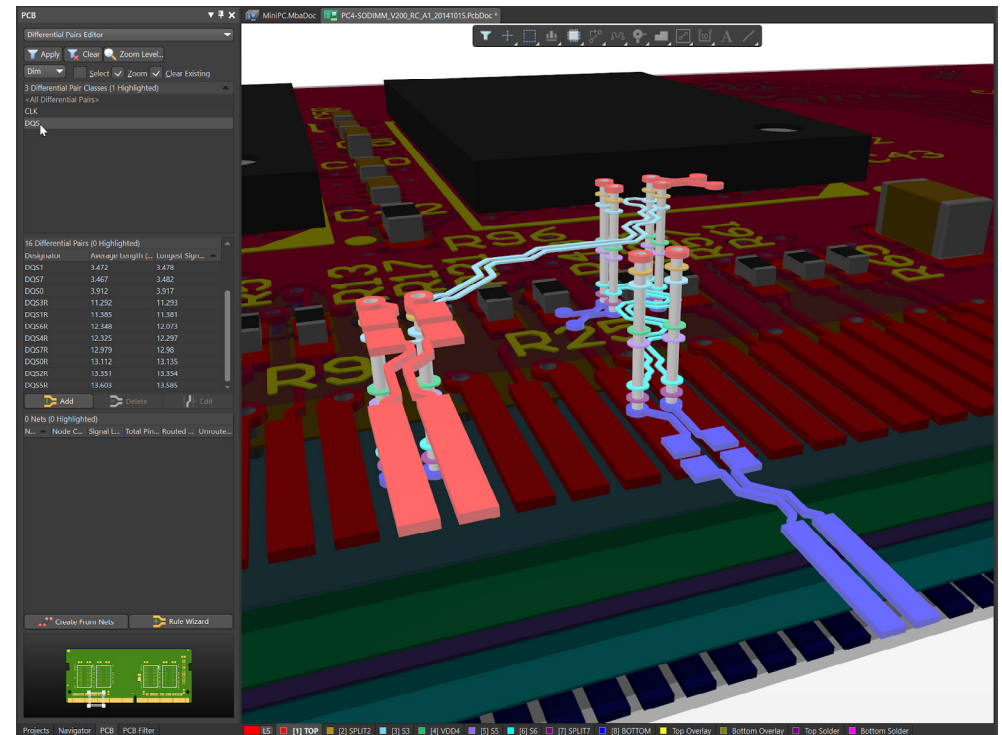
Single and Multiple Length Tuned Traces

The interactive routing modes in Altium Designer help You lay out boards quickly with precise control over the organization and flow of your board layout. They can interactively route their board with several powerful routing options including autoroute, walk around, push, hug and push, ignore obstacle, push & shove, and differential pairs. They can even automatically align routing path lengths without ever having to waste time manually adjusting individual nets with interactive length tuning.

You can leverage routing automation to route their designs quickly and accurately reducing the repetition of the task. You can spend more time perfecting their design leveraging the speed enhancement and implemented design rules of the interactive routing. Combined with design rules and design objects (rooms, keepouts, etc) you can reduce errors, achieve first pass manufacturing, and get to market faster.

Automated High-Speed Signals for High Speed Topologies

Working with high-speed designs can be time-consuming. The xSignals wizard lets you easily plan and constrain your high-speed designs by defining high-speed signal paths for modern technologies. They can route high-speed designs with fully configurable differential



PCB design using NATIVE 3D Technology

pair routings that carry precise signal lengths across their PCB. DDR3/4 and USB3.0 signals are automatically identified by the wizard creating rules to keep all signals in sync and tuned to the correct length.

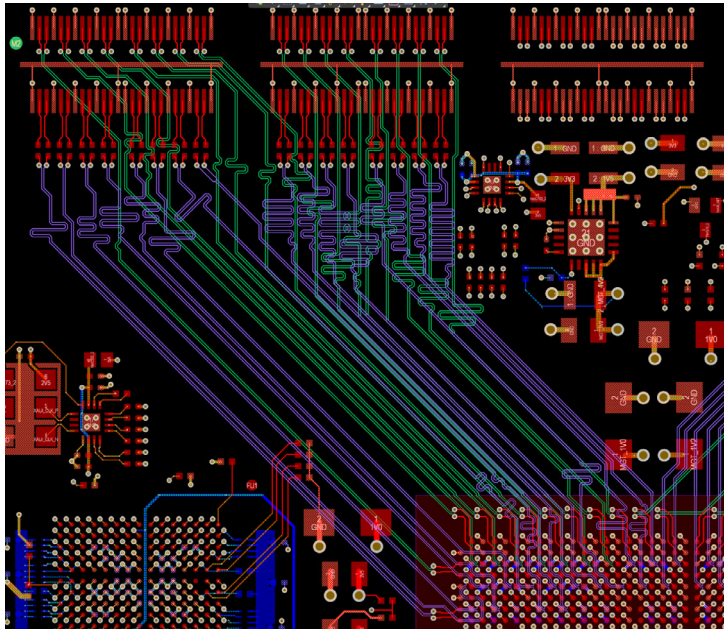
You can minimize timing errors for their signals. The signals are grouped together to ensuring organization and traceability while facilitating error correction and precise length tuning. The risk for design respins is minimized and the product design can go to market faster.



Fast and High-Quality Routing

ActiveRoute is a tool that lets you select where and how much automation you want to employ along selected nets. The technology in ActiveRoute coupled with user guidance produces high-quality layouts in seconds. ActiveRoute lets You breakout and route large, fine-pitch BGAs by instructing it where to route them (ie select layers, draw a guide), and letting it route for you. Unlike other interactive routing technologies, ActiveRoute works on multiple layers simultaneously while using all design rules.

By routing on multiple layers simultaneously, routing is faster, traces can be evenly distributed, and the ability to complete the routes increases significantly. Errors are minimized, time to market is reduced, and first pass manufacturability is promoted.

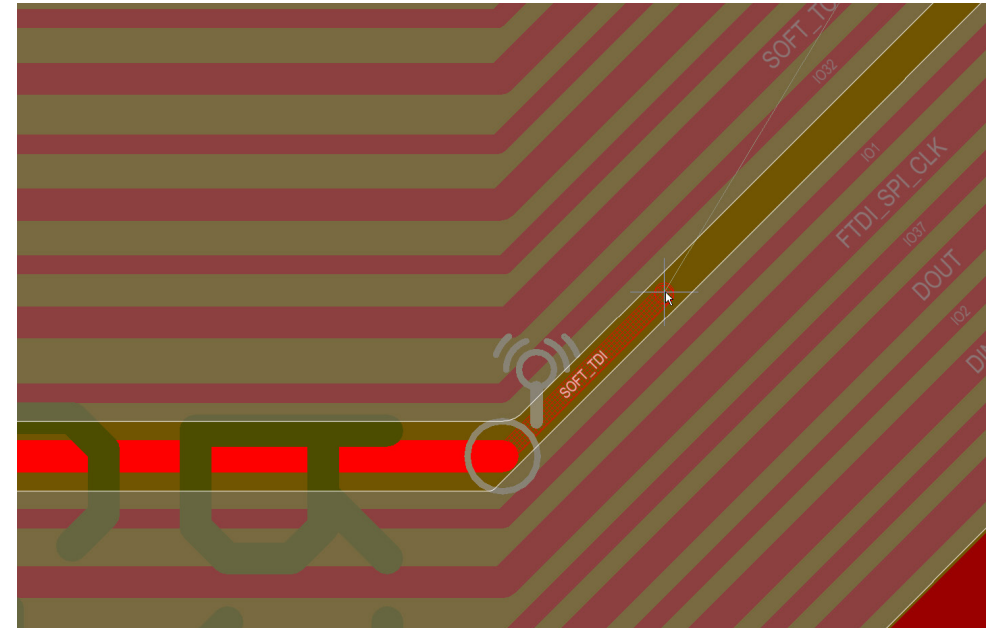


ActiveRoute with Guided Path and Tuning for High Speed Design

Visual Clearance Boundaries

You can visually see clearance boundaries between traces and components during route placement. Understanding the impact of your routing decisions in real-time alleviates the stress from unclear obstacles in the design process.

You can route through high-density areas with certainty that traces fit where they need them. You have immediate feedback to how route placement changes their layout and how it will change future routes. Errors are minimized, time to market is reduced, and first pass manufacturability is promoted.



Running a trace through a congested area



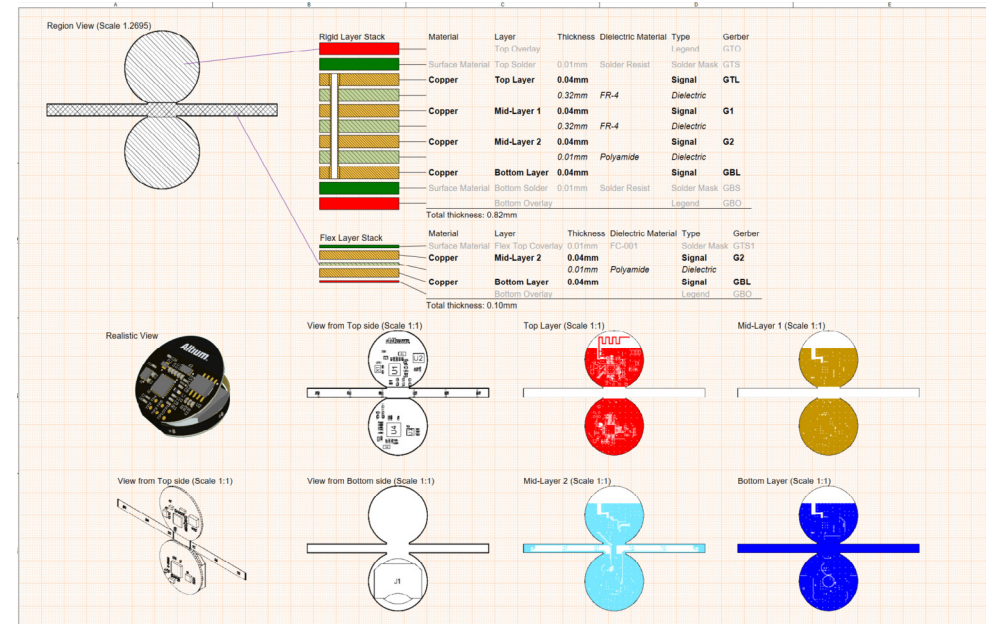
Manufacturing Documentation

When you are ready to move finished design into manufacturing, clear communication is mandatory. Output jobs act as an organized, reusable container for all necessary design outputs. Clear design intent communication to manufacturing is simple with powerful release management and automated documentation tools built into Altium Designer.

Automated Documentation Outputs and Project Release

Altium Designer enables you to add verified project snapshots into your workspace, or desired folder structure with the streamlined project releaser. You can dynamically create customized project outputs for design variants linking fabrication and assembly outputs with the latest design source files.

With an organized release process, output generation gains consistency and accuracy while ensuring you don't use out-of-date design files. The result is conveyance of design intent to the contract manufacturer of the completed design. Reduced time to market, first pass manufacturability, and increased design integrity.



Fabrication Documentation generated in Draftsman



Visual Manufacturing Outputs

Altium Designer's CAM Editor (CAMtastic!®) offers a variety of tools, the most basic of which are for viewing and editing CAM data. Once image and drill files have been imported, the CAM editor can extract physical connectivity netlist and compare it with an IPC netlist generated from the original PCB design software. These netlists will handle not just through-hole components, but blind and buried vias as well. The CAM Editor also offers Design for manufacturing Rule Checking, copper thieving, panelization and NC-Routing (plus milling) tools.

You can view the manufacturing outputs of their design to gain insight to what the contract manufacturer will receive. This insight allows you to make changes at identified problem areas. It also allows you rudimentary reverse engineering capabilities to create a skeleton PCB from manufacturing files. Increased design integrity, design intent communication, and first pass manufacturability.

Unified Professional Documentation Process

You can create assembly and fabrication documentation directly linked to source designs with Draftsman® and update their entire documentation at the click of a button. They can create templates for documentation that only require minimal customization across designs. They can add PCB dimensions, measurements, notes, and callouts between points of interest (datums) and design objects to customize documentation workflow.

The set of powerful and easy-to-use features integrated in Altium Designer automate documentation ensuring consistency. You benefit from guaranteed updates to documentation and a unified workflow. The possibility for data mismatch is practically eliminated, and there's no need to export your design to DXF/DWG or other file formats into archaic 2D drawing tools. Reduced time to market, increased design integrity, design intent communication, and first pass manufacturability are the result.

Data Management

Keeping track of design data is a complicated process with differing opinions of what is "correct." It is important to synchronize methods across a team to facilitate compliance, reduce the strain of maintenance, and ensure consistency across designs. Altium Designer leverages common software design techniques of version control and reusable design materials to synchronize design teams and allow you to get started on the right path.



Version Control

Version control allows you to save and maintain file revisions in an organized structure. Altium Designer integrates two important version control systems: GIT and subversion version control (SVN) support. Version control makes it possible to use design tracking, change permissions, and track collaboration. Designers can check-in, check-out, and visually compare differences between revisions of components, schematic documents, and PCB projects.

Version control helps with collaboration on projects and makes it easy to visually track changes across file revisions reducing time for error recognition and resolution reduces time to market and increase design integrity.

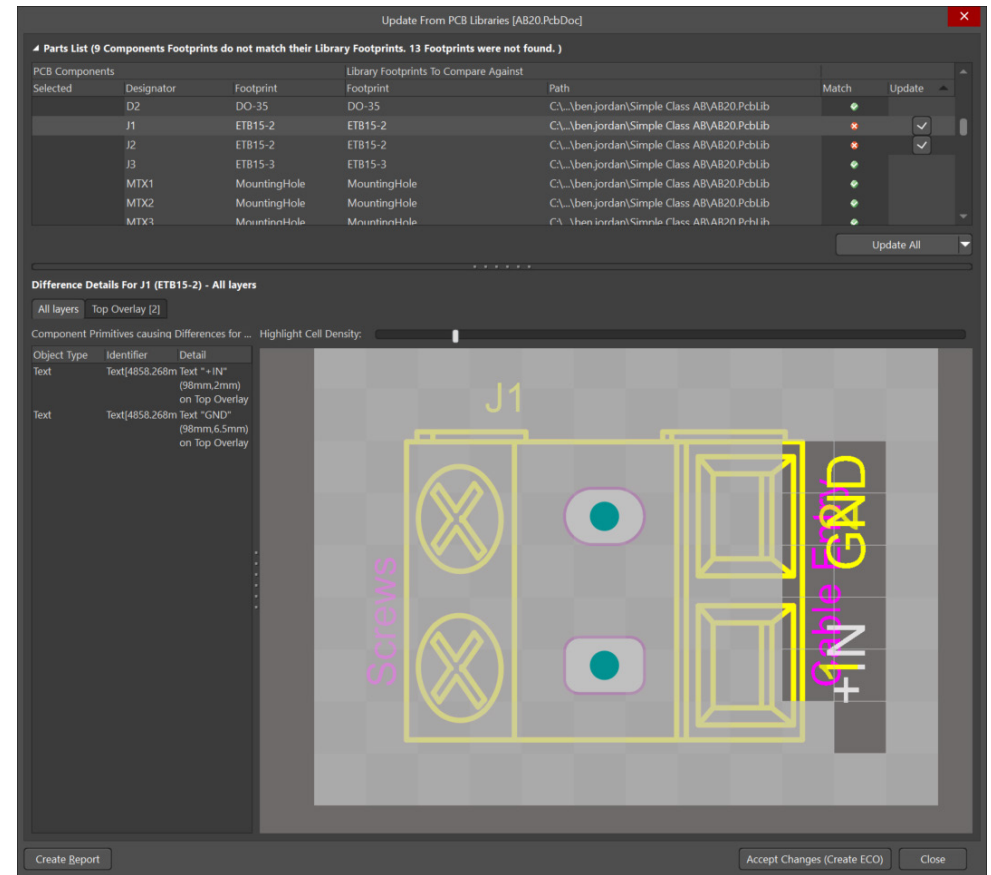
Design Reuse

Templates create a uniform design units to keep design information organized. The design units range from as small as pads to full project types to act as a common baseline for all new design materials.

Snippets are saved pieces of circuitry on a schematic and PCB level that can be used on any design to leverage common circuitry.

Device sheets allow you to create known circuitry blocks for reuse across designs. They differ from snippets by increased complexity and predefined interconnection to other parts of the design. For example, a power supply system that has a defined output of 5 Volts to power another circuit on the design.

You save time on the Schematic and PCB side by allowing the circuit layout and routing to be reused. When a change is made, it can be made to the base logical chunk and the results will propagate through the design. Overall, you minimize work and potential rework and increase design integrity by reusing blocks reducing time to market and minimizing errors.



Version comparison of component

Release Management

Whether you're using a formal managed process or working standalone, release management in Altium Designer helps hardware developers generate a complete package for going to prototype and production, lacking nothing.

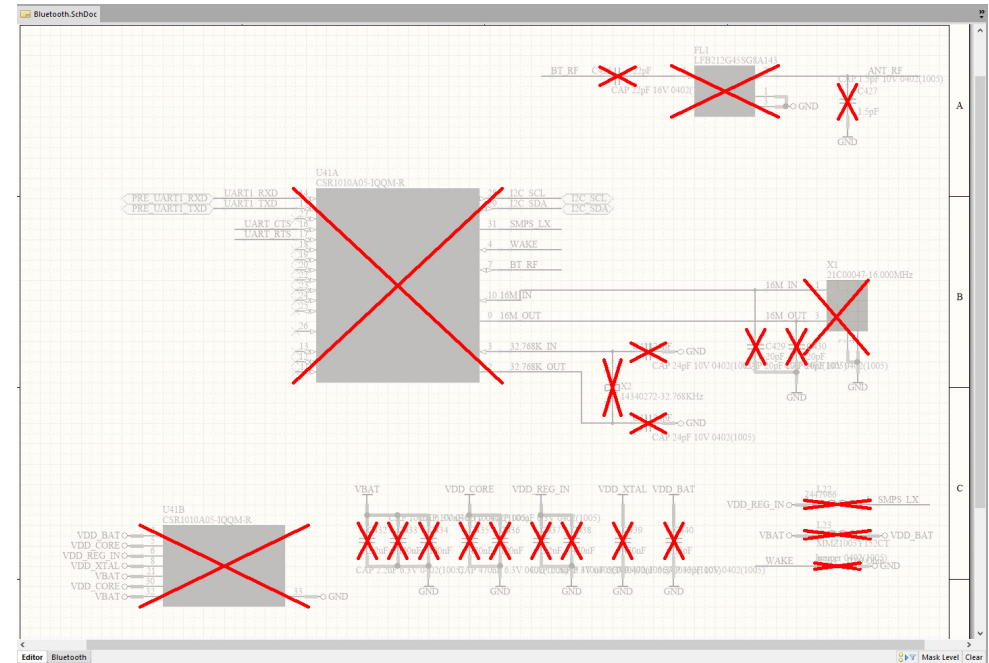
What's more, the release process is completely repeatable, letting you create templates and preset output jobs that guarantee the same formatting and collection of design CAM (Fabrication), Documentation, Validation, and Assembly outputs, every time.

The Project Releaser works with your predefined manufacturing outputs, combined with electrical and design rule checking and BOM generation, to achieve consistent results while allowing you to archive any production design snapshot in time, so you can always get back to the production restore point and compare changes. Release Management in Altium Designer gives you confidence.

Board Variants

You can create multiple versions of a board design with modifications to objects and other design elements to create unique products from the same base design (think iPod or phone variants with bigger hard drives, but some base design). Each variant stands as an original design with different components, version-specific design elements, and unique outputs to send to manufacturing.

You save time in duplicating a design for a variant creation while ensuring changes to the base design will not need to be duplicated. Sharing the same base design allows you to create multiple products with the same set of files and all documentation can be produced at the same time. You benefit with increased consistency, organization, and traceability minimizing time to market and potential respin and redesign costs.



This example shows a variant of a design with Bluetooth circuitry omitted



Easy | Powerful | Modern

High Performance Made Simple..... 1

Schematic Capture 2

 Design Import 2

 Unified Schematic and PCB layout 2

 Schematic Design Rules 3

 Electrical Rule Checks 3

 Hierarchical & Multi-channel Design..... 3

 Design Designator Annotation 3

Component Management 4

 Unified Component Model 4

 Real-Time BOM Management..... 5

 IPC Compliant Footprint Wizards..... 5

Mixed Simulation..... 5

 Analog and Mixed Signal Simulation 5

 XSPICE/SPICE3F5/PSPICE Model Support..... 5

 Simulation Profiles and Probe Management 5

Board Layout..... 6

 Visual Layer Stack Management 6

 Panelization 7

 Rules Driven Design 7

 Copper Pour Management..... 8

 Precise Object Placement..... 8

Rigid-Flex and Multi-Board..... 9

 Rigid-flex regions with bend angles..... 9

 Interconnected Multi-board Assembly..... 9

MCAD Collaboration 10

 Perfect fit mechanical enclosures..... 10

 Mechanical Model Import..... 10

Interactive Routing..... 11

 Single and Multiple Length Tuned Traces..... 11

 Automated High-Speed Signals for High Speed Topologies..... 11

 Fast and High-Quality Routing 12

 Visual Clearance Boundaries 12



Manufacturing Documentation	13
Automated Documentation Outputs and Project Release.....	13
Visual Manufacturing Outputs.....	14
Seamless PCB Documentation Process	14
Data Management	15
Version Control.....	15
Design Reuse	16
Release Management	16
Board Variants.....	16
Altium Designer	17

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®, DXF™, 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.