



Published on *Online Documentation for Altium Products* (<https://www.altium.com/documentation>)

[Home](#) > New in Altium Designer

---

Using Altium Documentation

Modified by Phil Loughhead on Oct 24, 2018

**Release Notes**

[Release Notes for Altium Designer 15.1](#)

[Release Notes for Altium Designer 15.0](#)

[Historical Release Notes](#)

**QuickLinks**

[Exploring Altium Designer](#)

[Altium Designer - Features](#)

## Altium Designer 15.1

*Released: 8 May 2015 - Version: 15.1.9*

*Released: 2 June 2015 - Version: 15.1.12*

*Released: 2 July 2015 - Version: 15.1.13*

*Released: 22 July 2015 - Version: 15.1.14*

*Released: 14 September 2015 - Version: 15.1.15*

*Released: 02 June 2016 - Version: 15.1.16*

This update to Altium Designer continues to deliver new features as well as enhancements to the core Altium Designer 15 technologies, while also addressing many issues raised by customers through the AltiumLive Community's BugCrunch system. As well as delivering a range of new and exciting features that develop and mature the existing technologies - including 3D PDF generation, high-speed design and rigid-flex board development, it also brings a large number of fixes and enhancements across the software as a whole, helping designers continue to create cutting-edge electronics technology.

[Release Notes for Altium Designer Version 15.1](#)



Alongside the development of Altium Designer 15.1, the Altium Vault continues to receive detailed and focused development attention to improve the ease of use, functionality and performance.

The release of Altium Vault 2.5 includes: storage of any file type, user-definable component templates, major improvements to the control of lifecycle transitions and permissions, Item cloning, the ability to create vault components directly from a supplier search, live supply chain data, plus a host of other performance and vault administration enhancements.

## 3D PDF

PCB designs can now be exported in the universal 3D PDF format that allows interactive 3D viewing in Adobe® Acrobat® Reader. External stakeholders now have the means to examine a physical design in three dimensions at high detail, without the need for specialized software. The foundation of this new 3D capability is the industry-developed **Universal 3D** (U3D) format for computer graphic files, which has been standardized by the European Computer Manufacturers Association (ECMA) as ECMA-363. U3D is natively supported by the PDF format and can be simply embedded (and compressed) in a PDF file for interactive 3D viewing in Acrobat Reader. The result is a modestly sized, cross-platform 3D file format that does not require specialized 3D viewing software. In the case of PDF 3D files exported from Altium Designer, the embedded content includes the majority of the data viewable in Altium Designer's own 3D view - including all tracks, pads, vias, polygons etc, all components and the full set of board layers.

[Feature in-depth...](#)

## Enhanced 3D Model and Export Support

Tight collaboration between the ECAD and MCAD design environments is essential for many designers today. Altium Designer supports collaborating with SolidWorks, and other MCAD tools that can import Parasolid-format files.

[Feature in-depth...](#)

## Multi-line PCB Text

The PCB Layout Editor's String object has been enhanced to support multi-line text, in both Stroke and True Type font modes. The string object is now resizable, offers dynamic word wrap and accepts carriage returns to force a new line.

[Feature in-depth...](#)

## xSignal Wizard

xSignals allow the designer to cluster multiple nets into a single signal, referred to as an xSignal. The xSignal Wizard greatly simplifies the process of defining xSignals, xSignal classes, and suitable Matched Length design rules.

[Feature in-depth...](#)

## Polygon Pour Improvements

Simple improvements can bring a profound change to usability - with automatic naming and intelligent dependency monitoring, the job of identifying, managing and modifying polygons becomes much easier.

[Feature in-depth...](#)

## Custom Coverlay Support

A common feature on rigid-flex boards is the selective use of coverlay material. This insulation layer is cut and laminated onto specific areas of the board, and because of this selective use, this custom coverlay is often referred to as *bikini coverlay*.

[Feature in-depth...](#)

## Union Enhancements

Unions are a type of group object that is created as part of a design process, such as placing via stitching or placing a drill table. the PCB panel now includes a Unions mode, select it to list all of the Unions present in the workspace, right-click on a specific union to rename it.

[Feature in-depth...](#)

## Pad & Via Templates and Libraries

The management Pads and Vias used in PCB designs has been greatly improved with the addition of Pad & Via Libraries, which contain collections of user-defined Pad/Via templates that can be reused in new and existing board designs. This is supported a new Templates Editor and PCB workspace Panels for managing both local and library based Pad/Via templates.

[Feature in-depth...](#)

## Live Drill Drawing

Working together with the Live Drill Table and the new Drill Symbol editor, the PCB editor's Drill Drawing layer is now live! Place a Drill Table and the drill symbols are automatically displayed on the Drill Drawing layer. Drill-pairs are fully supported, and you can easily select which pair is currently displayed on the Drill Drawing layer.

[Feature in-depth...](#)

### **Drill Pair Reference**

Drill pairs are now defined in a single location - the *Drill-Pair Manager*. The required drill pair is then selected from a list of available drill pairs, for example in the *Via* dialog. The validity of drill pairs is also monitored - if a drill pair is removed in the manager, usage of that pair is automatically flagged as an error.

[Feature in-depth...](#)

### **Board Outline Clearance Check**

A new Board Outline Clearance rule has been added to the Manufacturing rules, use this to constrain the proximity of design objects to various types of board edges.

[Feature in-depth...](#)

### **Gerber X2 Improvements**

The Gerber X2 manufacturing data export format has been improved with the addition of the 2:6 (Inches) and 4:5 (mm) data precision options, and through enhanced compliance with the latest Gerber X2 file format specification.

### **Improved Jumper Support**

Expanding on the concept of wire Jumper links, Altium Designer now supports jumper-type electrical connections for all component types, rather than just those assigned as a Jumper Type. Component pads with the same (non-zero) Jumper ID index are regarded as 'jumped' and are autoassigned to the same Net. Also, a short-circuit violation is reported if different nets are assigned to jumped pads.

### **Tasking Pin Mapper**

The new Tasking Pin Mapper provides one-way pin data synchronization between the TASKING VX-toolset for ARM® and Altium Designer. The feature is enabled by installing the Tasking Pin Mapper software extension, and makes use of the pin configuration files produced by the VX-toolset's Pin Mapper. Launch the *Tasking Pin Mapper Wizard* from the Schematic editor **Tools** menu.

### **OrCAD 16.x Import Support**

The Altium Designer Import Wizard can now import and translate OrCAD® Capture™ schematic design and library files in the version 16.x format.

[Feature in-depth...](#)

### **xDX Designer Importer**

Support for the transfer of binary format designs captured using Mentor Graphics® Xpedition® xDX Designer® (formerly DxDesigner®), to Altium Designer, is available courtesy of the *xDX Designer Importer* extension. Not only has the binary importer interface been upgraded to support data transfer from the latest version of xDX Designer (version 7.9.4 (Expedition Enterprise 7.9.4, or simply EE7.9.4)), a wider range of object types are also now supported. Launch the *Import Wizard* to import an xDX Designer project.

### **Testpoint Clearance Check**

Fabrication and Assembly Testpoint Style rules now include properties for controlling the clearance between testpoints and neighboring Pad/Via object holes.

[Feature in-depth...](#)

### **Solder Mask Expansion from Hole**

Pad and Via Solder Mask Expansions can now be applied from the hole edge, as well as from the Pad/Via's copper shape edge, in the *Pad* and *Via* dialogs.

[Feature in-depth...](#)

### **User-Definable PCB Print Output Job Colors**

Previously all PCB print-type outputs in an OutputJob shared the same color scheme. Now the colors can be defined for each output in the OutputJob file, and that color scheme can be saved (and loaded) as the Default scheme in the *PCB Print Preferences* dialog.

### **Schematic Symbol Generation Tool**

The new Schematic Symbol Generation Tool simplifies and speeds up the symbol creation process for large scale devices. Pins are configured using grid based tabular data, and copy/paste/smart paste operations are supported. Note that this tool is delivered as the Schematic Symbol Generation Tool software extension.

[Feature in-depth...](#)

### **Support for Cypress Touch Controls**

Extending its support for the use of touch controls in designs, Altium Designer 15.1 provides support for creating planar capacitive sensor patterns on your PCB, for use with the range of Cypress® CapSense® controllers.

[Feature in-depth...](#)

### **Improved 2D - to - 3D Switching Behavior**

Bringing the best of both worlds, you can now switch between 2D and 3D views asynchronously (press **2** or **3**), or synchronously (press **Ctrl+Alt+2** or **Ctrl+Alt+3**). If you need to see the same location and orientation of the board as you switch then press **Ctrl+Alt+2** or **Ctrl+Alt+3**. Alternatively, if you're working on different areas of the board then use the **2** or **3** shortcuts.

### **Improved Checking for Available Updates**

Altium Designer will now check for updates under all licensing and login conditions, keeping you better informed about available software improvements. The Updates tab in the Extensions and Updates page has also been improved, platform updates are clearly presented, all available platform updates are listed and more detailed information is available. [Feature in-depth](#)

---

## **Altium Designer 15.0**

*Released: 17 November 2014 - Version: 15.0.7*

*Released: 26 December 2014 - Version: 15.0.8*

*Released: 16 January 2015 - Version: 15.0.13*

*Released: 19 January 2015 - Version: 15.0.14*

*Released: 16 March 2015 - Version: 15.0.15*

This update to Altium Designer delivers a powerful array of new features and enhancements to the core technologies, while also addressing many issues raised by customers through the AltiumLive Community's BugCrunch system. In total, over 20 new features and over 120 fixes and enhancements across the software as a whole. Another sound release that reinforces Altium's ongoing commitment to providing you with a world class design solution.

### [Release Notes for Altium Designer Version 15.0](#)

Altium Designer 15.0 is a major update. It is not possible to update from Altium Designer 14.3, a new installation is required. You have the choice to continue with your current version or to install Altium Designer 15.0 to access the latest features. Alternatively, have both installed side-by-side, and use the version that best suits your needs.

### **PCB xSignals**

This new feature specifically enables the correct treatment of a high-speed signal path as just that - a path for a signal to travel between a source and destination, through termination components as well as y-splits. This means DDR3/4 memory routing and delay and phase tuning of the bus signals just got a whole lot easier.

[Feature in-depth...](#)

[Watch xSignals video...](#)

[Watch Length Tuning video...](#)

### **Solder Mask Expansion Enhancements**

With this new release, it is now possible to specify different values for solder mask expansions for top and bottom layers. The capability of defining separate layer expansions for pads and vias is available through the matching properties dialogs, the PCB Inspector panel etc, and can be applied on a Design Rule basis.

[Feature in-depth...](#)

### **Accurate Route Length Calculation**

The traditional approach to calculate signal length is to add up the centerline length of all segments used in a route, as well as the vertical distance due to the height of the vias, which was originally determined by the board thickness. This approach is not adequate for a high speed design though, for a number of reasons. The PCB editor's new length calculator in Altium Designer 15.0, returns the most accurate route length possible.

[Feature in-depth...](#)

### **Polygon Enhancements**

Based on extensive feedback from designers, there have been a number of improvements and changes to polygon pours. Shelving of polygons has returned and the display of modified polygons has changed to make it easy to identify an edited polygon. Vertex manipulation has also been simplified, with the default move mode for both corner and center vertex set to unconstrained movement.

[Feature in-depth...](#)

### **OLE Object Support in PCB Documents**

The PCB Editor now supports embedding OLE objects, such as Word or Excel documents, into a PCB document (**Place » Object from File**). As an OLE object it remains linked back to the source document, and an edit session can be launched from the PCB workspace.

[Feature in-depth...](#)

### **Separate 2D & 3D View Orientations**

2D and 3D view modes are now completely separate, each view mode retains the last-used orientation, zoom level, and layer configurations.

[Feature in-depth...](#)

### **Gerber X2 Support**

This release also includes the new Gerber X2 fabrication output format. Gerber X2 is an advanced extension of the existing Gerber RS-274X standard and provides an improved manufacturing output format that is backward compatible with existing workflows, software, and fabrication equipment.

[Watch video...](#)

[Ucamco Website](#)

### **Exporting to IDF in Unicode Format**

Altium Designer 15.0 sees an enhancement to the IDF Exporter, with support for generating files using Unicode (supporting the use of non-ASCII symbols). An example of the use of this aspect of the exporter, is when ensuring the generated files meet the compliancy requirements of the GOST standards.

[Feature in-depth...](#)

### **Output Job Editor Enhancements**

This release includes several enhancements to the Output Job Editor that make navigating and making selections within the OutJob generator listings much easier and more capable. The improvements include bulk OutJob enable/disable capabilities, scrollable window lists and drag & drop job reordering.

[Feature in-depth...](#)

### **Support for Rectangular Pad Holes**

Adding to Altium Designer's pad hole options of round, square and slotted definitions, this release includes enhanced hole support through the addition of Rectangular shaped holes.

Rectangular pad holes are ultimately routed, punched or cut out, depending on board fabricator capabilities.

[Feature in-depth...](#)

### **IPC-2581 Support**

The IPC-2581 output format is a new standard from the IPC (Institute for Printed Circuits) that is now supported by Altium Designer. A significant advantage of IPC-2581 is that it generates a single XML file which is capable of including all information needed to fabricate and assemble the printed circuit board assembly.

[Feature in-depth...](#)

[Watch video...](#)

[IPC-2851 Consortium Website](#)

### **IDX Support**

Building Altium Designer's strong ECAD - MCAD integration, is the new IDX support in the Mechanical CAD Collaboration extension. This extension implements ProStep's Electrical Design, Mechanical Design (EDMD) standard for data exchange, based on the XML protocol. This collaborative standard is most commonly referred to by the name of the exchange file format – IDX (Incremental Design Exchange format). Import the IDX Baseline via the PCB editor's Import menu, then control the collaboration process through the *Mechanical CAD Collaboration* panel.

### **Reorganized Import & Export Menus**

Both the Schematic and PCB Editors now have all of their Importers and Exporters available through the **File » Import** and **File » Export** menu entries. The **Save As** and **Save Copy As** commands are now reserved for Altium formats only.

[Feature in-depth...](#)

### **Upgraded Duplicate UID Correction**

With this release, Altium Designer's schematic compiler now detects duplicate UniqueIDs for all common circuit elements. Duplicate UIDs can be corrected and reset through a single command during editing, and are also automatically checked and corrected during the schematic loading process.

[Feature in-depth...](#)

## System & Performance Enhancements

Altium Designer 15.0 delivers a number of system and performance-related improvements that collectively increase efficiency, while enhancing your overall design experience with the software.

[Feature in-depth...](#)

## Controlling Parameter Visibility for Vault Components

Altium Designer 15.0 brings you control over the visibility of parameters - or more specifically, their values - when placing vault-based components into your designs from the *Vaults* or *Libraries* panels. This facilitates the Visible On Add concept, long enjoyed by users of Database Libraries.

[Feature in-depth...](#)

## Parameter-based Name Templates

Parameter-based name templates allow you to precisely rename your components, based on the currently defined values of parameters available to those components in the CmpLib file. Include one or more parameters into the naming template using the format [`<ParameterName>`], for example CMP - [Part Number] - [Value] - {0001}.

[Feature in-depth...](#)

## Vivado Toolchain Improvements

The recent versions of the Xilinx Vivado toolchain are now supported by Altium Designer 15. These are detected and registered by the system, and their path can be manually entered in the FPGA *Preferences* settings. The support includes Vivado's updated Checkpoint behavior.

[Feature in-depth...](#)

## GOST Compliant BOM Templates

Building on the support for the GOST standards (technical standards maintained by the Euro-Asian Council for Standardization, Metrology and Certification (EASC)), three GOST-specific Excel BOM templates have been added. More information is available on Altium's [Russian website](#).

## Vault Connection Enhancements

Altium Designer 15.0 sees enhancements to the way in which you connect to your Altium Vault. The option for automatic future sign-in on starting Altium Designer is now more obvious, and you also have the ability to remember, and automatically reuse, a chosen environment configuration. Together, these features allow you to get back in to your vault as if you'd never left.

[Feature in-depth...](#)

## Parameter List Templates

Altium Designer 15.0 brings with it a stand-out enhancement to the Component Library Editor, by way of customizable parameter list templates. These simple, humanly-readable files can be built with a list of required parameters, that can then be loaded into the CmpLib editor with a single click.

[Feature in-depth...](#)

## True Variant Enhancements

Variants now support the selection of an entirely different component for a variant, referred to as an Alternate Part. A number of features have been added to support this capability, including DRC support for stacked alternate parts, improved output generation, and support for PCB re-annotation.

[Feature in-depth...](#)

## Schematic Smart Wire

Back by popular demand, when a perpendicular wire is placed across multiple component pins and then dragged away from the pins, individual wires are created between each pin and the wire being dragged.

## WEBENCH support for adding a new wave to SimView

The WEBENCH® Altium Connector Extension, which interfaces to Texas Instruments WEBENCH® Designer, has been enhanced by allowing a new wave plot such as that for a dB(Vout/Vin) plot to be passed (and updated) by WEBENCH for the design's SimView plot.