The following collapsible sections list the release notes for publically released versions of the indicated historical version of Altium Designer. Documentation for these older versions can be found in Techdocs.

Release notes for all versions documented on this website can be found on the Release Notes parent page.

Release Notes for Altium Designer 14.3

Version 14.3.20

Build: 54863 Date: 30 November 2015

When using the Create Primitives From Board Shape command's Route Tool Outline option, you can now also select the Include Cutouts option to simulate tool paths that contour board cutout edges (BC:5185).

Arcs export as OutputJob file correctly for IDF (BC:4541).

The import of complex arc shapes from AutoCAD has been improved.

An issue where arcs were sometimes flipped when a DWG file was imported into a PCB, has been resolved.

Version 14.3.19

Build: 51423 Date: 6 October 2015

Error Message no longer occurs when Step-file are embedded to the PCB-document.

Resolved an issue for Preview in Print Job where selecting multiple PCB or Schematic documents caused Access Violation or empty preview.

Resolved an issue with Altium Designer hanging when attempting to import large Allegro design data.

Resolved an issue where DXP.EXE stopped when trying to configure outputs in AD 14.3.17 for some projects created in AD 15.0.

Resolved an issue where a new project being created from a template would point to the wrong OutputJob file.

Exception no longer occurs when choosing PCB font style in VariantManager (BC:4664).

PCB exception while re-building a net to an arc center point in a specific design no longer occurs.

Issues with opening older Protel/Altium designs have been resolved.
**Version 14.3.18**

**Build:** 45973  **Date:** 30 June 2015

6328 Resolved an issue when Pad names were incorrect and Net information was missing in ASCII file after PADS import.

6580 Fixed several issues with pre-defined stackups in layer stack manager.

6582 DRC units window has variable size now.

6604 AVS 1.1 is now supported by Altium Designer 14.3.

Fabrication output for a vault-managed, panelized PCB design now produces a warning message when the panel PcbDoc and referenced PcbDocs are not members of the same project.

Fabrication output for a vault-managed, panelized PCB design now produces a warning message when the panel PcbDoc and referenced PcbDocs are not members of the same project.

6332 Error after trying to generate A4-size for PCB has been resolved.

6334 Resolved an issue where a BOM is added to a publishing destination and the Excel file is created but the file size is zero bytes (BC:973).

6526 Issue with Schematic compiler producing intermittent crashes when compiling integrated library has been resolved

6534 SimView.DLL access violation error no longer occurs while simulating Mixed Sim.

6535 Allegro importer does not crash anymore on large designs.

6586 Resolved an issue with the BGA "Escape Routes" Fanout Feature whereby traces were added to the top layer only, and vias were placed, but there were no escape routes.

**Version 14.3.17**

**Build:** 42447  **Date:** 31 March 2015

5379 Resolved issue where Gerber generation of an embedded board referencing a non-existent PCB document would cause crashes.

5364 Timeout errors after releasing project documents to Altium Vault have been resolved.

5365 Improvements to releasing projects to Altium Vault with file-locking enabled.

5367 Support for AVS 1.1 in Altium Designer 14.3.

5369 Updating Altium Designer from an NIS no longer requires a current Portal connection.

5373 Resolved an issue with saving menu customization between sessions (BC:4957).

5375 BOM generation do not hangs anymore, when ActiveBOM is used as source with Variant switch.

5377 Resolved an issue where it was not possible for CmpLib folders to share the same name on different paths.

5380 Resolved exception error while working with split planes (BC:5059).

4032 Resolved issue where the Parallel Segment rule would produce an adjacent layer DRC violation when layers are separated by internal plane layers.
Resolved issue where deleting a free object inadvertently deletes nearby component-owned objects.

Resolved issues with Legacy 3D View's display and export to STEP model.

Testpoint component flipped in the PCB editor correctly shows as top or bottom, fabrication or assembly in the PCB inspector panel.

The correct lifecycle and naming schema is now being loaded during CmpLib file-less editing.

Resolved issue when deleting multiple sub parts of a component in the schematic library editor, the selected components aren't always the ones that are deleted (BC:4768).

Device Support Report no longer crashes.

A PDF generated from a schematic that includes an image pasted from the Windows clipboard now works correctly.

**Version 14.3.16**

**Build:**37051  **Date:** 16 December 2014

Changing PCB layer names could result in layer-related lists populating incorrectly and certain outputs not generating correctly, this has been fixed.

Improved performance of selection and zooming in Schematic in comparison with 14.3

STEP models from Inventor 2014 are now loaded without errors

Class generation settings are now stored for device sheets (BC:3840)

An AV could occur while placing a pin in a schematic library and pressing Esc to quit the command, this no longer happens.

With a specific combination of preferences, placing a component from the schematic libraries panel could cause Altium Designer to crash, this no longer occurs.

Performing a release after file-less editing an Item in the Planned state, no longer fails with the error (BC:4677, BC:4870).

**Version 14.3.15**

**Build:**35511  **Date:** 02 October 2014

The BomDoc now correctly includes all of the defined solutions.

Users without Admin permissions can now log into a Vault when there is no TC2 configuration defined.

Multi-selection in the Variant Management dialog followed by an editing action no longer causes occasional exceptions.

The PCB Hole Size Editor now correctly includes blind and buried layer pairs in the list of holes to be drilled.

Vault login now supports including a protocol prefix, such as http.

Updating a PCB footprint from a library now correctly retains all objects and only updates modified objects.

Comments can now be added to a newly created managed project via the Comments panel.
If a managed project had a space character in the name, it could not be opened on a remote PC. This fix allows new projects to be created and opened with spaces in their names, it does not support opening an existing space-in-name project.

Opening a large CmpLib via file-less editing no longer causes an overflow error.

OpenBus Ports can now be correctly linked to in an OpenBus document (BC:4669).

When a multi-part component is varied, each part in the component now retains the same part in the new component.

The designer can now decide if parameter values come the database components, or the variant management dialog.

Generating a Pick and Place file from a variant no longer results in the duplication of Alternate Parts.

Setting a component to Fitted via the right-click menu in the Variant Management dialog now functions correctly.

Generating ODB++ after running Remove Unused Pads on a design with blind vias, no longer causes the application to crash.

Version 14.3.14

Build:34663   Date: 25 August 2014

If a design which uses ComponentLink parameters is PDF'ed with the Include Component Parameters option enabled, you can now click on the link in the PDF to open the referenced document.

Comments are no longer posted with the wrong author name.

New Project dialog no longer prefixes each file with the project name, when the Create Project Folder option is enabled.

Switching a component from Alternate Part to Not Fitted now correctly clears the component parameters.

Creating a polygon from the Board Outline in the Polygon Manager no longer causes an exception.

When a copied polygon is pasted, it now retains the pour state of the original polygon.

In certain situations, changes made in the Parameter Manager were not applied to all objects, this has been resolved.

Generating a BOM from an Outjob now retains the Excel template formatting.

A schematic auto-junction at a 4-way + junction is now retained when a drag occurs.

An exception no longer occurs when the Class Generator is used to create a component class.

The Variant Manager now supports changing the order of variants.

Parameters varied in an Alternate Part can now be restored to the original values in the Alternate Part.

Alternate Parts no longer produce schematic compiler warnings stating that multiple pins are connected.

Alternate Parts now show correctly as fitted in the assembly drawing.

The number of clicks required to choose an alternate part has been reduced.
Hatched polygons now switch to the unpoured state correctly.

Pick and place data generated from a variant now displays the correct component data (BC:4344).

Creation of components with identical names in a Schematic library is now prevented.

Alternate parts now display the correct layer information in the BOM.

Random crashes no longer occur when working in the PCB Panel.

Performance while deleting schematic wire segments has been improved.

Document Insight now works for all schematic documents in the project.

The Include not fitted components BOM option now works correctly.

The Comment is now displayed correctly in the BOM for a component that has varied parameters.

ECO no longer detects unmatched nets for alternate components.

The order of variants can be changed by clicking and dragging the column header, this new order is retained between edit sessions.

Comment is now displayed correctly in an Assembly drawing for a varied component.

The values for multiple variant parameters can now be edited in a single editing action.

An Out of Resource error no longer occurs when printing a schematic document with an embedded Excel sheet.

Version 14.3.13

Build: 34012   Date: 09 July 2014

Layer Stack Manager could be started now if .Net 4.5 is not installed

Version 14.3.12

Build: 33957   Date: 01 July 2014

Invalid pointer error fixed in cmplib editor when working with component libraries created in older versions

Fixed error where Objects were accessed after being destroyed when working with the Filter panel

Resolved application error on Windows XP when a PCB was reopened during Altium Designer start up

Alternate part information no longer erased when original library is not accessible anymore

Manually adding a Vault in the Preferences dialog would sometimes causes an invalid handle error, this no no longer occurs

Select Next command could cause a crash in certain conditions, this has been resolved

Variant Management dialog, Set Selected As Alternate no longer causes an error

Footprint changes done through the Footprint Manager are now preserved

Errors caused by the XSpice Model Wizards have been resolved
Version 14.3.11

**Build:** 33708  **Date:** 13 June 2014

3241 Exception no longer happens when copy/pasting component in cmplib editor.
3280 Selecting an object in the PCB workspace now correctly selects that object in the PCB panel (Nets mode).
3296 Alternate parts are no longer removed when opening an older design that includes variants.
3301 Model and parameter inheritance in CmpLib files is now correctly handled when opening an older version CmpLib in AD14.3.
3273 Handling of duplicate UIDs has been strengthened, to ensure that the compiled project correctly synchronizes with the PCB.
3304 Editing a Part Choice List with insufficient rights no longer causes an exception.

Version 14.3.10

**Build:** 33625  **Date:** 4 June 2014

3219 PCB Release process now continues correctly when non project documents are encountered.
3221 Component variant parameters can now be changed back to their original values.
3222 BOM documents can now update alternate variant components from database parameters.
3234 Schematic connection highlighting no longer cause incorrect results after deleting wire segments.
3244 The polygon pour order is now used correctly when rebuilding polygons. [BC:4283]
3245 Varied components can now be restored to their original Fitted state.
3249 PCB Design View layer visibility is now working correctly.
3256 PCB string formatting issues when pasting from MS Excel documents has been fixed.
3278 Polygon Pours now use the correct clearance rule values when rebuilding polygons.

Version 14.3.9

**Build:** 33548  **Date:** 29 May 2014

1053 RS Components and Future Electronics Suppliers have been added as new extensions.
1113 STEP Export now supports folded state of the PCB design.
1290 Opening OutJob BOM report no longer generates "Could not convert variant of type" errors.
1510 New front end VHDL and Verilog compilers have been included in Altium Designer.
1583 Smart PDF paper sizes are now maintained regardless of Default Printer settings. [BC:2009]
1595 A problem in the Signal Integrity extension causing an access violation when parsing certain comment fields has been fixed.
1623 Objects are now pushed even if the object being moved has a violation. [BC:1067]
1628 PCB Polygons now pour correctly when remove necks setting is greater than Electrical clearance. [BC:678]
Schematic Multi-Part components are now displayed using the Alpha Numeric Suffix preference in the Libraries and SCH Library Panels. BC:291

P-CAD and PADS PCB Importer fixes incorrect component rotations.

Schematic wire dragging has been greatly improved. BC:3216

Altium Designer now supports file-less editing of vault data. Right-click on Item in vault to edit model, component or managed sheet.

Vaults panel right-click menu enhanced to support logical behavior throughout the panel, for example right-click in search results to place, or add to content cart.

Generate Board Edge Rout Paths option (NC Drill Setup dialog) now supports complex and curved board shapes. BC:821 BC:705

Solder and Paste Masks are now rendered correctly with Embedded Board Arrays. BC:3790

Variants now support the ability to specify a different (alternate) component. BC:1921

ActiveBOM document memory management has been improved.

Schematic wire segments can now be individually deleted. BC:829 BC:3410 BC:1501 BC:1223

PADS Logic Importer has been updated. Fixes to Port Sizes and Designator naming.

Via Shielding tool has been added. Tools>>Via Stitching/Shielding>>Add Shielding to Net.

Unused Pad Removal tool has been added. Tools>>Remove Unused Pad Shapes.

Polygon Pour thermal connections now correctly connect to the corners of pads, rather than the center.

PCB Interactive Length Tuning tool (and gauge) has been improved and patterns can now be modified after placement.

PCB track chamfering has been added. Tools>>Convert>>Convert Selected Tracks to Chamfered Path.

Differential Pair rule now displays metric values correctly for Gap values.

Moving Rooms has been greatly improved.

PADS Importer has been improved and supports version 9.5 files.

PCB object selection popup can now be disabled in PCB Preferences. Pressing TAB will cycle through overlapping objects.

PCB component placement now uses the Polar Grid to allow automatic rotation to the origin of the grid.

Memory leaks while debugging discrete CPUs have been fixed.

PCB Nets Panel has been updated with Min, Max and Estimated Length columns and uses color coding (green, yellow, red) to indicate range values.

Moving Vias with connected tracks has been improved.

Refactor clearance rule into grid form.

Now the used part of the via has been taken into account when calculating the routed length of net.

PCB Teardrop tool has been improved providing more control over generated teardrop shapes.
Interactive Routing now supports automatic neck down while routing through rooms.

IPC-7351B-Compliant IPC Wizard added packages: CAPAE, DFN, LGA, PQFN, PSON, SODFL, SOF, SON, SOTFL.

ActiveBOM documents now support Variants and includes a Variants toolbar.

CmpLib editor has been extensively updated to improve usability & release process, including: simplified presentation with column-based filtering, grouping, horizontal scrolling, multi-select, drag and drop models from Vault, and right-click model editing.

Å± character is now included in PDFs for all language regions.

Schematic text drawing of Japanese characters has been improved.

Top and bottom paste masks are displayed correctly in Layer Stack Table.

Access violations associated with switching between Sch and SchLib documents and making library edits has been fixed through investigation of crash reports.

ActiveBOM now supports components with different values but using same Design Item IDs.

New PCB SMT Design Rule - "SMD Entry".

Schematic Fonts can now be copied/pasted in the Inspector Panel.

Schematic Power Objects now include Font property.

Schematic Text Frames now support text margins.

Schematic Update From Libraries now correctly ignores parameter changes that have been excluded from ECO.

'PCB Density Map' can be displayed successfully in 2D Mode.

PCB Component "Flipped On Layer" property has been added to Inspector, FSO and List Panels. BC:3864

PCB Polygon Pour Vertices editing has been improved.

PCB GDI support has been removed.

Gerber output of octagonal pad shapes now accurately matches the PCB octagonal pad shapes.

PCB Polygon Pour Shelving has been replaced with Unpoured. Commands added to Polygon menus and Polygon Manager dialog. Loading earlier version designs will change shelved polygons to unpoured state.

PCB Clearance Rule introduces a new "minimum clearance matrix".

Length tuning tolerance is now applied to the length of the longest scoped net target.

Placing multiple Vias now retains the net names.

Libraries panel now supports installing a Vault folder (or tree of folders) as a library, from which components can be browsed and placed.

3D PCB Flex rendering has been improved.

During Release to Vault process Project no longer asks to be saved.

File>>New>>Project commands have been replaced with a New Project dialog.

Schematic Update From Libraries now preserve Designator, Comment and Parameter locations if "Choose Component" is used.

Loading time for components in the Libraries Panel has been significantly improved.
PCB track dragging now preserves non-orthogonal angles.

DirectX memory limits have been increased, improving stability for larger designs.

3D PCB Variants now display correctly.

PCB Object Class Explorer dialog now remembers column position and size.

Compiling large Integrated Libraries no longer cause exception and out of memory errors.

Schematic Text Frames no longer display invalid characters when empty. BC:3941

Schematic .PNG and .SVG graphic images with transparency can now be printed/saved to PDF.

Schematic Update From Library no longer changes justification/mirrored settings when preserving parameter positions.

FPGA Project options have been updated to support Verilog Include Paths and Defines.

PCB tracks and vias now default to drag mode regardless of selection state.

PCB DRC uncoupled length check no longer reports false length calculation errors.

PADS PCB Importer option "Do not translate hidden net names" has been fixed.

Support for Altium web applications, such as Vault browsing, in Internet Explorer 11 has been improved.

P-CAD and PADS Logic Schematic importers now resizes Port objects correctly.

P-CAD PCB import now translates clearance rules into new clearance grid matrix.

Performance when changing component state of variant "fitted/not fitted" on large opened designs has been greatly improved.

It is now possible to change the lifecycle state of multiple Items in a single edit action.

PCB Layer Stack Internal Layer Pairs setting can now be selected and used.

Recursive folder deletion in Vault Explorer has been improved.

BOM now supports including Vault Item revision and lifecycle state.

Navigation preview in PCB Panel now works correctly in 3D view.

Non Altium Designer documents that are stored in a Vault can now be opened in the appropriate editor directly from the Vaults panel.

Additional columns added in the Vaults panel: Item section now includes Note column, and Where-used section now includes State column.

Display state of Vaults panel now retained between Altium Designer sessions.

Vault web interface has been upgraded, including new Where-Used and Children features and improved browsing capabilities.

Octagonal Pad selection issue when x-size is smaller than y-size has been fixed.

Using "=VariantName" parameter in expressions is now interpreted correctly.

PADS PCB Importer has improved translation of Matched Length and Diff Pair rules.

OpenBus Signal Manager retains settings when invoked from processor configurator dialog.

PCB Teardrop tool has been improved providing more control over generated teardrop shapes.
Schematic wire dragging has been improved when connecting to other wires. New Junction display and color settings and cursor warning symbol have been added.

Schematic .png graphic images can now be rotated.

Vaults panel Preview display mode now presents ComponentLink parameters intelligently, combining them to display as ComponentLink1Description = ComponentLink1URL in the parameter list.

Browser-based Vault access substantially enhanced to include all of the browsing and searching functionality available in the Vaults panel, as well as a broad range of Vault management features.

Libraries panel no longer changes the selected library when the Refresh command is used.

Smart PDF and Printer Page setup dialog now includes a Quality setting for DPI (75, 150, 300 and 600).

"Variants..." and "Choose Alternate Part" commands have been added to the right mouse popup menu, when Variants are selected.

Schematic Rubber Stamp command now maintains component rotation and mirroring after initial placement.

The PCB Board Outline area is no longer generated in output if the multi-layer or a mechanical layer is enabled.

During interactive dragging of a wire end with multiple wires selected, all moving wire ends are displayed.

Generic [ActiveBOM Document] can now be used as a source in Report Outputs job in an OutputJob file.

Tiling documents and reopening Altium Designer no longer causes tiled blank space or access violations.

Opening or compiling schematic documents could result in an access violation due to invalid expression references. This has been fixed through investigation of crash reports.

Vaults panel, multiple selected Items can be moved to another folder using drag and drop.

After generating a BOM, exporting to Excel, then closing the Excel BOM, Excel no longer remains running as a Process.

Session validation cache added to the Vault and the Part Catalog.

New Dragging options for selected and unselected vias/tracks have been added to the PCB Interactive Routing Preferences.

Vault Explorer browsing control has been improved.

Current Polygon editing style is now displayed on the Status bar during interactive polygon editing.

"Retrieve Layer Colors" command now works correctly when setting up preferences for printing documents.

"IsFree" function has been added to expression and scripting engine.

The issue where the DXP menu would appear on all toolbars has been fixed.

PCB Differential Pair routing now allows for proper corner exits.

An exception that would occasionally occur during PCB collaboration has been resolved.

PCB Inspector and List panels now display the layer order correctly for the Layer drop down.
In Board Planning Mode, Move Board Vertices command has been replaced by Edit Board Shape command, which uses new polygon editing behaviors.

Issue where Altium Designer would lose the project's SVN data, resulting in a project becoming unversioned, has been resolved.

PCB special string .Application_BuildNumber now displays correct software version.

Editing objects via the Inspector or List panel no longer causes the objects to de-select.

PCB design objects can be locked via the right-click menu.

Additions for new Jumper type component, including: automatic net name changes for Jumper pins, and new IsJumperComponent keyword added.

Importing a specific PADS library no longer causes an exception.

Smart PDF from schematic now supports Wingdings font. BC:4206

Net name inheritance now occurs correctly when routing from a PCB fill object.

To support new Jumper component type, new View » Jumper submenu added in PCB Editor.

Application_BuildNumber special string added to the Schematic editor.

Support for a new Jumper component Type added to the Schematic editor.

Importing a specific Texas Instruments WEBENCH design no longer results in incorrect object placement (requires WEBENCH Altium Connector Extension).

Board units now toggle correctly when Q shortcut is pressed in Board Planning Mode.

Custom page sizes for PDF output can now be defined in the DXP>>>Preferences>>>System>>>Printer Settings dialog.

Schematic objects incorrectly associated with a union while placing and using the insert key has been fixed.

The Pad property for "Hole Width" has been changed to "Hole Length" in the Inspector and List Panels.

The word Width now spelt correctly in the Width Constraint design rule.

Message displayed during update installation reworded to clarify what will happen, now says: Altium Designer must be closed to complete the update process. Close Altium Designer now?

Altium Designer status bar now includes indicators to show state of memory and GDI resource usage.

FPGA Signal Manager no longer causes an exception when the Export To Do Item button is clicked after selecting an Unmatched Signal and adding ports.

Updating PCB from Vault based components no longer cause "Invalid or Deleted Item Revisions" error when the symbol name contains special characters.

Schematic redraw speed has been improved when using large blanket objects.

License Management view has been adjusted to show at least three licenses on displays with lower resolution screens.

Variants now support parameter inheritance. BC:2232

PCB multi-layer objects can now be properly filtered when in Single Layer Mode. BC:4008

InViaShielding query keyword added, use this to scope an applicable polygon connect style design rule for shielding copper.

Release Notes for Altium Designer 14.2
### Version 14.2.5

**Build:** 32823  **Date:** 5 May 2014

- A crash when database libraries experience connection problems while placing parts or generating BOM reports has been fixed. 
- Access violations caused by PCB editing, routing and undo operations have been fixed through the investigation of crash reports. 
- Pads are now generated correctly in Gerber files for flipped Embedded Board Arrays.  
- The "Flipped on Layer" option is no longer being set during ECO component placement due to default PCB primitive settings.  

### Version 14.2.4

**Build:** 31871  **Date:** 5 March 2014

- Grouping Vault Search results no longer causes an exception error.  
- Variant values are now displayed correctly for Multi-Channel and Complex Hierarchy schematic projects.  
- The Preferences option in the Subversion update repository dialog now works correctly and no longer causes an exception error.  
- PCB footprint names that use special characters (such as * or /), no longer cause footprint not found errors and other editing, browsing and synchronizing issues.  
- Rotating schematic .EMF graphic files no longer causes an exception error.  

### Version 14.2.3

**Build:** 31718  **Date:** 19 February 2014

- Schematic drawing line object now supports "dash dotted" line style.  
- Schematic Pin object now includes Line Width attribute for Symbols.  
- Schematic arcs now always display correctly at different zoom levels.  
- Schematic documents now render non-English text correctly when saving between AD10 and AD14 formats.  
- Box.Net publisher has been updated and now uploads files correctly.  
- The "Create Primitives From Board Shape" command now correctly uses arcs instead of line segments.  
- DBLib and SVNLib using MySQL will now attempt to reconnect automatically to the server after PC hibernation or lost connection.  
- Subversion commit comments now support quotation marks.  
- Subversion commit comments now properly support different language input text.  
- Subversion 1.8 support has been added. This also includes support for the new working folder format, as well as updating the built-in Subversion Client to version 1.8.4.  
- OrCAD Capture export now correctly handles Power, GND and test point objects.
PCB Drill Table has been updated to include new Text Alignment and Column Width settings (available from the right-click menu).

Schematic now supports vector graphic images (.wmf and .svg).

PCB Polygon repour speed has been significantly improved. Speed increase up to 20x depending on number and complexity of polygons.

There is no longer a hang when running the automatic loop removal tool.

PCB footprint primitives are no longer removed from the footprint during interactive routing.

Loop removal will no longer remove incorrect track.

PCB Polygon Pour quality has been improved when using hatched style and polygon cutouts.

PCB Design Rule dialog will now grey out disabled rules making it easier to distinguish between enabled and disabled rules.

The PCB Library Report can now support non english characters.

Schematic documents now render text correctly when using non english language system settings.

Smart PDF now supports an option to include "Global Bookmarks for Components and Nets".

PCB Copy Room Formats command will no longer copy the board region.

PADS Library importer has been updated to fix specific cases where components were not created.

DxDesigner importer has been updated to fix connection issues with off grid objects.

PCB Via Stitching has been improved and will now properly use the clearance rule settings.

Schematic Formatting Toolbar can now change text font and size settings.

STEP model clearance errors between top and bottom layers has been fixed.

Selecting PCB components in Single Layer Mode will now only select components which are on that layer.

Schematic directives no longer produce false error reports when single documents are compiled.

The PCB Printout Properties dialog now opens correctly even if the PCB is not the top document window.

Use of the backslash character (\) in an Output Job container file name no longer inserts an underscore character (_).

The PCB Drill Table is now correctly re-positioned when using the Move»Move Selection by X,Y command.

Scripts using SchServer.RobotManager.SendMessage can now be run successfully.

PCB Undo now works correctly when primitives are deleted from a component.

The PCB "Update Free Primitives From Component Pads" command no longer removes vias from their nets.

The Locked attribute is no longer updated when changing the Pad or Via test point settings.

Deselecting the Preview top - bottom layer command for a drill table, will no longer cause an Access Violation.
PCB polygon thermal relief calculation causing incorrect DRC violations in some situations has been fixed.

IDF Export has been updated to fix a hang when exporting from specific files.

The Libraries Panel now remembers the layout when the application is restarted.

There is no longer a discrepancy in the display of top and bottom SM layers when opacity is set to 50%.

Smart PDF now displays the correct component parameters when using Variants.

BOM generation now supports variant supplier and supplier part number parameters.

Variants now support the .COMMENT and .REFDES special fields.

The Schematic "Update From Libraries" command has been updated. The "Preserve Parameter locations" option now works correctly and a new option "Preserve parameter visibility" has been added.

Automatic adjustments to track end-points when moving a selected track can now be reliably undone using the Undo feature.

The Schematic "Break Wire" command now works with wires that have smallest width setting.

Soldermask now correctly remains as defined for regions and polygons after saving and reloading.

Net Antennas with a terminating via are now correctly detected by the DRC.

Chinese and Russian text is now correctly encoded when exporting a Schematic to DXF.

Via Stitching by constrained area no longer causes shorts with polygons on internal layers.

The issue of incorrect From-To lengths has been partially resolved. Length evaluation no longer counts primitives twice when modifying an existing net. Note: Incorrect evaluation of a complex signal path between points could still occur.

The Schematic Library Parameter Manager now displays selected pins when "Selected Objects Only" option is enabled.

The query InFromToClass will return all primitives in the from-to, and will no longer miss track segments.

Vault now supports batch download of selected items.

There is no longer an Access Violation when generating a Gerber from a PCB containing an active drill table.

The legend in a generated Drill Drawing now works correctly when using Characters for the .Legend symbols.

FPGA to PCB Project Wizard now uses installed FPGA integrated libraries to correctly place schematic symbols in generated PCB project.

DXF Exporter now correctly supports slot holes (zero line width and donut), and square holes.

Debugging embedded project targeting discrete CPU no longer causes a crash on exit.

There is no longer an error when using the parameter "=VariantName" as the folder name in an Output Job file.

An issue with the Mixed Simulation engine failing to create the correct netlist has been resolved.
There is no longer an issue when exporting files for route using SPECCTRA. BC:3833

There is no longer an Access Violation when printing a schematic containing a large graphic image.

IDF Exporter has been fixed to resolve the issue with duplicated board outlines. BC:3696

The Layer Stack Manager dialog now correctly opens in active display space, in accordance with monitor configuration.

The "ssl_error_unsafe_negotiation" error no longer occurs when opening URLs inside Altium Designer.

There is no longer a crash in Supplier_TME when closing Altium Designer.

An access violation caused by an undo operation in PCB has been fixed through the investigation of a crash report.

The correct board height is now used when exporting to STEP.

An access violation associated with importing preference files has been fixed.

Version 6.3 and 6.5 Eagle files can now be successfully imported to Altium Designer without error or Access Violations being generated.

There is no longer a crash when accessing SchServer.RobotManager.SendMessage through a script.

The graphical preview of the schematic part is once again available when accessing the Port Map tab for a Sim Model.

Parameters with spaces in their names can now be used indirectly through a part's Comment field. BC:1129

Issues with the import of EAGLE files have been fixed. EAGLE frames have been implemented, the un-named sheets issue addressed, and schematic loading for version 6.5.0 also resolved.

The version number is now correctly checked for the Lattice Diamond toolchain.

There is no longer an Access Violation when clicking on All Folders after an initial Vault search.

Vias are no longer removed from existing Via Stitch objects when cancelling the Via Stitch by Area command.

A crash with the SCH List panel when changing scope is now resolved.

TMDS33 constraint for Xilinx Transition Minimized Differential Signaling IO standard is correctly translated to UCF TMDS_33 value.

CoreGenerator targeting Xilinx devices now correctly generates the FIFO memories for data widths other than multiples of 8.

Editing the properties of a component from the SCH Library panel no longer causes the location of that component to change in the panel's listing of components.

The layer for a placed via can no longer be changed through the PCB List panel.

The PCB Signal Integrity design rules are now analyzed by the Signal Integrity extension.

An issue with incorrect library selection in the Libraries panel has been resolved. The library selected rather than the one under the cursor is now correctly used. BC:2868 BC:3079

An issue whereby the "Flipped on Layer" option was being erroneously set during component placement, has been resolved. BC:3796 BC:3843
The Process Flow after successful synthesis is no longer reset when performing a manual refresh (F5).

FPGA build status in Devices View is correctly updated when Altera Synthesizer is used.

A new FPGA compiler option has been added to the FPGA General Preferences to "Clear compiler output messages".

FPGA projects are now linked properly to PCB projects when components are placed from the Vault.

Embedded extension adds support for TI Stellaris and NXP LPC2000 families of discrete microcontrollers.

Cortex-M3-based Stellaris Support extension enables programming and debugging for Texas Instruments Stellaris family of microcontrollers.


Release Notes for Altium Designer 14.1

Version 14.1.5

**Build**: 30772  **Date**: 5 December 2013

- PCB object transparency levels are now displayed correctly.
- AVS and PVS now support prefixed wildcard searches.
- The graphical selection issue with the Layer Stack Table has been fixed.
- The slow editing performance when selecting pins in the schematic Library Editor has been fixed.
- An access violation with the Silk to Board Region DRC check has been resolved through investigation of a crash report.
- The dielectric layers have been removed from Layers drop down list in the PCB inspector and PCB list panel.
- A "memory error" when opening the Layer Stack Manager dialog has been resolved through investigation of a crash report. [BugCrunch #3493](https://bugcrunch.altium.com/issue/3493).
- Solder Mask expansion now updates correctly when updating a footprint from a PCB library.
- An access violation when Board Regions do not have a defined layer stack has been fixed through the investigation of a crash report.
- PCB Collaborate, Compare & Merge panel is now showing / matching layers correctly.
- The differential pair rule width now displays "N/A" instead of "0" when mixed widths are defined.
- PCB Polygons can now be placed on solder mask layers.
- Mixed Sim and SIMetrix no longer cause exception errors if models not found or other schematic problems encountered. Mixed Simulation extension update v1.1.0.30712. SIMetrix/SIMPLIS Interface extension update v1.01.30765.
- TME Supplier search has been improved to support different currency selection.
- PCB net name and pad number display issues have been resolved.
STM32 Write to Flash dialog is enabled now.

An access violation in Core3D.dll has been fixed through investigation of a crash report.

Schematic editor local language resource files have been updated.

Xilinx Spartan 3AN internal flash programming is working correctly now.

The slow performance issue with some designs when moving PCB components has been fixed.

The "Drill symbol limit exceeded, switching to letter generation" message is now displayed in the Messages Panel instead of a popup dialog. BugCrunch #652.

PCB split planes no longer lose their net assignments when opening and closing the Layer Stack Manager dialog. BugCrunch #3571.

An issue with DRC Report that affected some boards resulting in additional error counts has been fixed.

The PCB Width rule now correctly calculates the impedance-driven width value.

OrCAD Capture export now correctly handles Power, GND and test point objects.

DXF/DWG Export from an OutJob no longer causes an exception error.

The Release Manager no longer causes an access violation when releasing footprint models to AVS.

IDF export now associates the correct component footprints when using Integrated Libraries.

P-CAD and PADS importers have been updated to fix a crash or lockup with some files.

An access violation when editing union objects has been fixed through investigation of a crash report.

Internal Plane layers are now added correctly to the design when placing snippets or pasting internal plane objects.

ODB++ Export no longer produces a crash when using embedded components.

DXF/DWG Export has been updated to fix a layer mapping problem with arcs.

DXF/DWG Import no longer causes an exception error when "Paper" option is selected.

Step Model cache handling has been improved and crashes associated with "Cannot load 3D model from file" errors have been fixed. BugCrunch #2733.

Hyperlynx export now supports 6 digit precision. BugCrunch #2899.

Generated Pick and Place files now ignore Design View objects.

The Copy Room Formats command now moves objects to the correct layers. BugCrunch #3566.

Mirrored Design Views now generate pad objects in outputs.

Camtastic "File»Export»Export to PCB" command no longer generates an exception error. BugCrunch #3556.

An offline version of the Altium Installer is now available for Altium Designer 14.1. More Information

FPGA Synthesis options panel correctly shows all options for Altera Quartus synthesizer.

Release Notes for Altium Designer 14.0
**Version 14.0.9**

**Build:** 30380  **Date:** 21 October 2013

872 Diff Pairs now use the Diff Pair Gap Rule instead of the Clearance Rule.
889 When starting a Diff Pair route it now picks up the correct track widths.
890 The initial gathering of traces in the differential pair router has been improved.
895 Connection lines never get dimmed if "Show All Connections in Single Layer Mode" is on.
924 NanoBoard 3000 firmware update is now working correctly.
962 The PCB Drill Table can no longer be selected or edited when the Drill Drawing layer is hidden.
970 The 'L' shortcut key to change PCB routing layers is now working correctly.
988 The "File In Previous Format" report is no longer added to the PCB Project when opening older files.
989 PCB DRC Reports now display correctly when using medium sized (125%) fonts.
990 The PCB DRC dialog now opens correctly when using medium sized (125%) fonts.
1002 PCB Polygon Pour "Remove Islands Less Than" threshold no longer rounds to nearest whole number.
1022 PCB Polygon Pours now follow the Min conductor width settings correctly.
1025 IPC Footprint Wizard QFN and BGA silkscreen and sizing issues have been fixed. [View BugCrunch report #2682.](#)
1133 STEP Export now allows Polygon Pours to be shelved.
1192 ODB++ now exports Drill Pairs correctly regardless of the defined layer order. [View BugCrunch report #1011.](#)
1207 Film Size checks are now ignored when generating Gerber files and using the "Separate file per layer" Batch Mode option.
1214 A new option "Snap to Arc Centers" has been added to the PCB Board Options dialog. [View BugCrunch report #1512.](#)
1418 Retrieving Supply Chain information from the Part Catalog has been fixed.
1648 Clicking on Blog entries within the Home Page no longer opens a blank document.
1671 PADS Logic Importer: Fixed incorrect Port orientations.
1767 Schematic Library loading issues with some files have been fixed.
1834 Schematic Library saving issues with some files has been fixed.

[Release Notes for Altium Designer 13.3](#)

**Version 13.3**

**Date:** 13 Jun 2013

**System Components: Altium Designer Base**

339 Fixed memory leak leading to application crash with "runtime error 216" while debugging embedded code.
Searching Solutions in Vault and ActiveBOM now uses the Keywords defined in Supplier Preferences.

The Supplier Search dialogs no longer list currencies twice.

The File in Previous Format html report now uses the correct document tab name.

ActiveBOM Production Quantity no longer looses it's focus when typing values.

The Design>>Update ECO process now continues to executes all valid changes regardless if an error is found.

A crash that could occur while matching items within the Item Manager has been resolved.

Floating Panels no longer disappear when File>>Import command is used in PCB Editor. View BugCrunch report #2640.

Expressions can now be used to define output folder names in OutJobs.

BOM Supplier Unit Price is no longer rounded to 2 digits.

ActiveBOM Supply Risk is now updated correctly when Production Quantity is changed.

A crash when removing or renaming documents in a project has been resolved.

OutJob filenames are now generated correctly when using expressions that contain "." characters. View BugCrunch report #2892.

Variant support has been added for ActiveBOM documents in OutJob.

ActiveBOM solutions are now ranked automatically (if no ranking exists) based on order of existing supplier links contained within schematic components.

ActiveBOM now supports currency selection.

ActiveBOM custom solution prices have been increased from 3 to 5 decimal places.

ActiveBOM actual pricing is now calculated correctly when item quantity is below the minimum quantity price.

SUPPORTcenter link in License Management page now opens Support section in AltiumLive.

ActiveBOM is no longer case sensitive when matching Supplier Names in supplier links.

DBLib column visibility and grouping is now remembered when application is closed. View BugCrunch report #3045.

Schematic URL paths and links to directories now open correctly.

Releasing PCB Projects to a Vault now works correctly and does not produce incorrect missing files errors.

ActiveBOM now correctly groups items when updating and Refresh.

ActiveBOM now allows deletion of invalid auto generated solutions.

BOM reports are now being updated correctly when using database parameters.

OutJob now supports the 'Do not Include any output folder' option for Release Managed Folder Structure.

'OutputType' parameter has been added to OutJob expressions.

OutJob now includes both the "Custom prefix_[OutputType]" and "Custom Folder Name" options for Folder Structure Containers.

Opening files (referencing file://) and HTML reports (containing special characters) now open correctly within the internal browser.
System Components: Data Management

423 Altium Vault Properties dialog now displays license details.
1023 Vault SimModels have been extended to support Simetrix/Simplis models.
1517 Vault search results are now displayed correctly if order of columns have been changed.
1546 Placing managed sheets from a Vault now uses the correct revision.

System Components: Schematic System

1362 Additional styles for Power Ground, Earth and Bar have been added to the Power object in the schematic editor. View BugCrunch report #2246.
1502 Schematic "out of resources" error has been fixed through the investigation of crash reports.
1503 Schematic now supports Unicode text allowing any combination of characters regardless of current locale.
1524 The focus now goes to the Name control when opening the Schematic Port properties dialog. View BugCrunch report #2980.
1546 Placing managed sheets from a Vault now uses the correct revision.

System Components: PCB System

906 Initiating the Differential Pair Routing command in PCB no longer highlights only differential pairs.
1334 Room boundaries are now displayed correctly when they contain arc segments.
1419 PCB dimension objects are now loaded correctly when using Hungarian regional settings. View BugCrunch report #2885.
1405 The PCB display mode no longer changes when unshelving polygons.
1455 Embedded Board Array no longer swaps the column and rows when generating output.
1470 The arc center point is now included as a movable hot spot when moving arcs in the PCB Editor. View BugCrunch report #1512.
1481 When modifying a PCB document the slow updating of Power Planes Connections has been fixed.
1483 PCB expression that use aggregate functions no longer cause "Incorrect definition error".
1484 A crash when saving Show Top Positive option in PCB View Configurations has been resolved.
1486 Fixed slow analyzing nets issue when synchronizing PCB in some situations.
1489 Drill Tables are now displayed correctly in the PCB List Panel.
1516 Changing the Drill Table defaults no longer causes an access violation.

System Components: Soft Design System

384 FPGA_PCI_CLAMP constraint is correctly generated for Altera devices.
1508 Unknown Status code error upon startup with DT05 and other FTDI based dongles connected has been fixed. View BugCrunch report #2674.
System Components: Soft Design Support

383 Core Generator now accepts hex memory images larger than 256 kB.

System Components: Altium Designer Localization

1536 Traditional Chinese localization of Altium Designer has been updated.

System Components: Altium Designer Support

1485 Internet connection errors due to firewall or SSL have been fixed.

Importers and Exporters: Importer_Accel

1538 P-CAD Schematic Importer : Can now optionally generate netlabels for hidden net names.

Importers and Exporters: Importer - PADS

1431 PADS Logic Importer : Fixed incorrect Pin and Port orientations.
1435 PADS Logic Importer : Fixed Pin Designator and Name visibility.
1438 PADS Logic Importer : Fixed Footprint model not defined in schematic symbols.
1442 PADS Logic Importer : Fixed positioning of free text, component parameters and designators.
1445 PADS Logic Importer : Fixed polygon fill drawing.
1446 PADS Logic Importer : Fixed Power Port and Net label placement.
1451 PADS Logic Importer : Fixed designator, parameter and text string fonts.
1538 PADS Logic Importer : Can now optionally generate netlabels for hidden net names.

Importers and Exporters: Importer - PCAD ASCII

1433 P-CAD Schematic Importer : Fixed incorrect model names.
1436 P-CAD Schematic Importer : Fixed wire and component primitive widths.
1437 P-CAD Schematic Importer : Fixed text that is inserted into Description field.
1444 P-CAD Schematic Importer : No longer incorrectly changes the schematic grid settings.
1450 P-CAD Schematic Importer : Fixed polygon fill colors.
1452 P-CAD Schematic Importer : Fixed designator, parameter and text string fonts.
1538 P-CAD Schematic Importer : Can now optionally generate netlabels for hidden net names.

Output Generators: Printer - PCB

1387 Design Views now print correctly when the Mirror option is enabled in the Printout Options dialog.
1458 The .legend is now generated correctly when used with an Embedded Board Array. View BugCrunch report #2924.
Design Views that are disabled are no longer included in the printable area for Fit to Page printouts.

Printing Design Views now print correctly if the top layer is removed from the printout.

Variant Markers are no longer offset when scaled Design Views are printed. View BugCrunch report #2982.

Full arcs are now printed correctly with mirrored Design Views.

Output Generators: Output - Simple BOM

OutJob filenames are now generated correctly when using expressions that contain "." characters. View BugCrunch report #2892.

Output Generators: Output - ERC

OutJob filenames are now generated correctly when using expressions that contain "." characters. View BugCrunch report #2892.

Output Generators: Output - NC - Drill

NC Drill output generation now ignores placed Design Views.

Output Generators: Output - Testpoint

Testpoint output generation now ignores placed Design Views.

Data Management Packages: Supplier - Farnell

ActiveBOM currency prices for Farnell solutions are now calculated correctly. Release Notes for Altium Designer 13.2

Version 13.2

Date: 10 May 2013

System Components: Data Management

Searching facility in the vault explorer has been enhanced to facilitate the narrowing down of keyword based search results.

The enabled state of connected vaults is now properly taken into account across the system.

It is now possible to drag and drop components from the vault explorer onto a schematic sheet.

The CmpLib Editor user interface has been improved.

The CmpLib editor user interface has been improved with additional editing functions.

Lifecycle definition and revision naming schemes can now be modified.

When creating cmplib files out of dblibs, the different tables activation statuses are now taken into account.
It is now easier to identify items that are out of date in a design or in a cmplib, and to update them to the latest revision.

Component definition checkboxes in cmplib editor are now properly taken into account when proceeding with the release of selected components.

The item manager now saves and reloads its settings and column sort order.

The item manager has now an Apply function which generates the necessary ECOs, executes them and then refreshes the item manager, allowing a steadier workflow.

Can now release simulation data/models to a Vault.

The choose item revision dialog always displays all revisions appropriately.

The intermittent error "failed to move temporary files for revision [revision name]" while releasing footprints, symbols or components does not occur anymore.

It is now possible to download from the vault a selection of symbols, footprints and components as libraries.

A hang no longer occurs when creating items in a folder with the naming scheme that does not contain curly brackets.

Can now successfully opening .cmplib documents when naming scheme does not contain curly brackets.

The ‘ISC ERROR‘ when accessing the AVS server license has been fixed.

Vault lifecycle definitions and revision schemes are now always visible regardless of system login rights.

No error occur when migrating an empty Satellite vault to Personal Vault or Altium Vault Server.

It is now possible to set the initial revision ID of an item first revision when it is in the planned state.

Vaults now support ODBC connections providing database access to supply chain information.

Can now generate SimModel files from Schematic Libraries and Database Libraries.

The Release Managers now issues a warning about read only files, and does not attempt to process them.

Components containing the # character are now released without issue.

The last selected search result in the vault explorer is now persistent.

Altium Content Vault now benefits from the new search engine.

Altium Shanghai Vault contents have been merged with Altium Content Vault.

**System Components: Altium Designer Base**

New live component catalog and BOM capabilities providing ‘real time’ design cost estimation and availability.

Expressions can now be used to define output file names, for PDF outputs.

The stability when closing the application has been improved resulting in fewer runtime errors.

It is now easier to identify items that are out of date in a design or in a cmplib, and to update them to the latest revision.
Item Manager now provides a Report command in the Right Click menu.

An error related to _Release.log no longer occurs when trying to release a project that is under version control.

Output job generation of print, pdf, and ODB outputs no longer produce unexpected condition for files that are absent from the project.

Can now release simulation data/models to a Vault.

New revisions of vault based component footprints now update correctly when synchronizing a PCB design.

Can now successfully opening .cmplib documents when naming scheme does not contain curly brackets.

Synchronizing a design that uses designator names like AUX, COM1, COM2 no longer causes a lock up or crash.

The Embedded Project Output paths are now saved correctly.

Managed sheets are now properly added to version control from the PCB Release view, if needed at release time.

Improved loading and navigation of URL based pages.

Vaults now support ODBC connections providing database access to supply chain information.

A crash occurring with the Navigator panel when application is closing has been resolved through the investigation of crash reports.

Can now generate SimModel files from Schematic Libraries and Database Libraries.

It is now easier to choose target folders in the Library splitter wizard and the Library splitter tool of the release manager.

The popup notification indicating a newer version is installed has been removed.

A crash when using the Libraries panel when no libraries are installed has been fixed.

Variant support has been enabled to Step Export option in Output Job. View BugCrunch report #2506.

SmartPDF documents now produce PCB component bookmarks correctly. View BugCrunch report #2792.

The following labs features have been enabled. "AutoRoute On Current Layer", "AutoRoute On Multiple Layers", "Extended C-to-Hardware Features" and "Visual designer for embedded GUI".

The Help menu links for User Forums and SUPPORTcenter have been updated.

System Components: PCB System

A new Drill Table object has been added that provides a full display of the drill legend at design time. Use Place>>Drill Table command in the PCB Editor. View BugCrunch report #155.

Transparency now works correctly for Via and multi-layer Pad holes.

It is now easier to identify items that are out of date in a design or in a cmplib, and to update them to the latest revision.

PCB loop removal quality has been improved and no longer ripped up unrelated tracks in some situations.
Can now release simulation data/models to a Vault.

New revisions of vault based component footprints now update correctly when synchronizing a PCB design.

STEP file export now produces the correct output regardless of computer regional settings.

A crash that could occur when analyzing nets containing zero sized primitives has been resolved. View BugCrunch report #2030.

Re-ordering the columns in the PCB Rules and Constraints editor no longer causes incorrect rule types to be created.

Configure 3D Video output job no longer crashes.

A crash in the PCB 3D Video Output Job configuration dialog has been fixed through investigation of a crash report.

A crash when cancelling the muti-selection popup in the PCB editor has been fixed through investigation of a crash report.

Footprint previously placed from Altium Content Vault can now always be found.

Fixed a crash when using the Add/Remove Net to Class commands and the class is not selected in the PCB editor.

Via Stitching placement and editing has been improved with new pattern x,y offset control, added [VS] name visibility and auto-regeneration when via stitch union is re-positioned.

The HUD and PCB Nets panel are now being updated when resizing tracks in the PCB Editor.

The current net length now updates in the PCB Panel and HUD when interactively routing a net, and when interactively modifying an existing route.

'DrillTable' has been removed as a Component Type. The Drill Table is a new PCB object.

A new Design View object has been added that provides a scalable (real time) display and output of a specified area of a PCB design. Use Place>>Design view command in the PCB Editor.

Drill Symbol assignment has been added to the Hole Size Editor.

Polygon pour issue with arcs when created from selected outline has been fixed.

PCB Silk to Solder Mask check performance has been improved when using large negative Solder mask expansions.

System Components: Altium Designer Installation System

Progress dialog now always displays when updating plugins. Will take affect once 13.2 update has been installed.

Updates are now only checked when opening the "Updates and plugins" page and user is signed in. The popup notification has been removed.

System Components: PCB Support

Can now release simulation data/models to a Vault.

Fixed 3D model loading issue that produced the "Cannot load 3D model from file! error.

Can now generate SimModel files from Schematic Libraries and Database Libraries.
System Components: Schematic System

571 It is now possible to drag and drop components from the vault explorer onto a schematic sheet.

736 It is now easier to identify items that are out of date in a design or in a cmplib, and to update them to the latest revision.

858 A new option in schematic preferences to control the character used for the designator/part id separator of multipart components.

890 A crash on close depending on arrangement of toolbars/panels and open documents has been fixed after investigation of crash reports.

913 Parameter editing using the Schematic Inspector panel has been fixed. View BugCrunch report #2629.

916 A crash with Schematic Library Report has been fixed after investigation of a crash report.

1003 A crash when selecting two schematic components at the same location has been fixed through investigation of a crash report.

1089 An issue when copying text to clipboard and Clip to Area setting is used in a schematic Text Frame has been fixed.

1168 PDF and print outputs of annotated compiled sheet numbers are now correctly generated. View BugCrunch report #2669.

1169 Saving schematic files that contain Hyperlink objects will now open in AD12 correctly.

System Components: Schematic Support

824 Can now release simulation data/models to a Vault.

926 A crash while using the schematic autowirer has been resolved through investigation of a crash report.

1010 Can now generate SimModel files from Schematic Libraries and Database Libraries.

Output Generators: Printer - PCB

298 A new Drill Table object has been added that provides a full display of the drill legend at design time. Use Place>>Drill Table command in the PCB Editor. View BugCrunch report #155.

1032 A new Design View object has been added that provides a scalable (real time) display and output of a specified area of a PCB design. Use Place>>Design view command in the PCB Editor.

Output Generators: Output - ODB

816 Output job generation of print, pdf, and ODB outputs no longer produce unexpected condition for files that are absent from the project.

Output Generators: Output - STEP

871 Status progress information is now displayed when exporting Step files in the PCB Editor. View BugCrunch report #2576.
Output Generators: Output - Gerber

A new Drill Table object has been added that provides a full display of the drill legend at design time. Use Place>>Drill Table command in the PCB Editor. View BugCrunch report #155.

A new Design View object has been added that provides a scalable (real time) display and output of a specified area of a PCB design. Use Place>>Design View command in the PCB Editor.

Gerber output no longer generates incorrect solder mask openings for arcs when the "Use software arcs" option is enabled.

Output Generators: Output - AdvSim

Can now release simulation data/models to a Vault.

Importers and Exporters: Importer_Accel

A crash when importing P-CAD libraries using the Import Wizard has been fixed.

Importers and Exporters: Importer - Allegro

Allegro importer now converts rounded SMD pads correctly. View BugCrunch report #1626.

FPGA Design Tools: Instrument - Custom

The Custom Instrument now displays correctly in the FPGA Devices View.

System Components: Soft Design System

Peripheral register viewing for STM32 is now available.

Hardware Support Packages: Device Support - STMicroelectronics STM32

Peripheral register viewing for STM32 is now available.

Release Notes for Altium Designer 13.1

Version 13.1

Date: 28 February 2013

System Components: Altium Designer Base

FPGA Project mapping to vault items dialog has been disabled.

The "Extruded 3D body conversion to generic STEP model" option has been enabled.

Unnecessary project compilation has been reduced in the following areas. Update from PCB Libraries, PCB Release View, Output Jobs, Harness Problem Finder, Annotate, Default Print Configuration, Component Links, View Channels, Parameter Manager, Footprint. View BugCrunch report #1183.
Saving constraint file paths in FPGA projects is now compatible with AD12.

Fixed an issue causing an incorrect or blank output path in both new and existing projects.

An option has been added to suppress output of component parameters when exporting to PDF.

A crash when using the All Off command to de-selected items in the Sch Item Manager has been resolved through investigation of a crash report.

A crash occurring during certain sequences of output generation has been resolved through investigation of a crash report.

Through investigation of crash reports a Access Violation occurring in version control that could occur upon opening of documents has been resolved.

A crash that could occur generating a report from the configuration dialog of BOM style outputs has been resolved through investigation of a crash report.

HTML based reports including DRC and File Format warnings now open in a new tab rather than replacing the home page.

A crash during opening of the DXP menu has been resolved through investigation of a crash report.

A crash occurring upon wake from system hibernate has been resolved through investigation of a crash report.

A crash when closing a modified PCB document with the automatic saving option enabled has been resolved.

A crash in copying data from various grids with a subset of columns selected has been resolved through investigation of a crash report.

A crash in wire optimization during drag operations in schematic has been fixed through investigation of several crash reports.

Fixed a "canvas does not allow drawing" error during print preview through investigation of a crash report.

AD12 Configurations are now correctly loaded into AD 2013. [View BugCrunch report #2460](#).

Crash reports generated within the PCB editor due to large floating values have been resolved.

Improved the generation of PDF reports when zero byte files where produced in some situations.

**System Components: Altium Designer Installation System**

Locate and install missing plugins now working properly.

An error that could occur when adding plugins has been resolved through investigation of a crash report.

Installing from Local and Network repositories has been improved.

**System Components: Data Management**

A Division by Zero error has been resolved when using copy/ paste operations in the CmpLib editor.

The Generate CompLib command was available when the incorrect document type was visible and caused a crash when run. The visibility has been corrected.
A cannot make a visible window modal error when attempting to move a folder in the vault explorer has been resolved.

The choose item revision dialog always displays all revisions appropriately.

**System Components: Altium Designer Localization**

Updated Japanese localization of Altium Designer.

**System Components: PCB Support**

Memory corruption that could occur when changing the color of component bodies through the PCB Inspector has been resolved through investigation of a crash report.

An AV occurring when Signal Integrity was run on a project containing duplicate designators has been resolved through investigation of a crash report.

An AV when saving or exporting CAM files has been resolved through investigation of crash reports.

Crash reports generated within the PCB editor due to large floating values have been resolved.

Memory allocation when working with polygons has been improved.

**System Components: PCB System**

The performance and reliability of the Silk to Soldermask rule has been improved.

A menu entry has been added to the Options Popup to jump directly to the Transparency Tab of the Board Layers and Colors dialog.

An AV when dragging parallel tracks when one of the selected tracks is not connected to any others has been resolved through investigation of a crash report.

A crash on close has been resolved through investigation of several crash reports.

A regression issue setting the selection when dragging multiple tracks has been corrected. The newly added tracks are now selected as before.

Invalid STEP files are now handled gracefully when placing 3D bodies.

Find Similar Objects now creates the correct expressions for match based on coordinates.

Memory corruption that could occur when changing the color of component bodies through the PCB Inspector has been resolved through investigation of a crash report.

An AV that could occur loading certain PCBDoc and PCBLib documents containing Component Bodies has been resolved through investigation of a crash report.

An AV occurring when creating a snippet from objects on internal planes has been resolved through investigation of a crash report.

Circles now display properly on PCB footprint previews.

An AV when saving or exporting CAM files has been resolved through investigation of crash reports.

Crash reports generated within the PCB editor due to large floating values have been resolved.

Memory allocation when working with polygons has been improved.

**System Components: Schematic System**
Overlapping text in the Synchronize Ports to Sheets Entries has been fixed.
The No ERC and Schematic Component dialogs have been updated to accommodate scaled displays.
Sheet entries are now positioned correctly when using Smart Paste in Schematic.
Placing wires, lines and polygonal blankets in schematic now completes closed loops after right click.
Attributes missing from the Schematic Inspector for Rectangle, Text Frame, Compile Mask and Blanket objects are now visible.
Find similar objects can now correctly use the font property to match objects.
Memory corruption that could occur when changing the color of component bodies through the PCB Inspector has been resolved through investigation of a crash report.
A crash when attempting to exit with the Schematic inplace editor is active has been resolved through investigation of a crash report.
Crash reports generated within the PCB editor due to large floating values have been resolved.

System Components Soft Design System

- Rescanning JTAG chain is now much faster.
- Toolchain and device names are saved in the embedded project document to improve backward compatibility with AD12.
- Memory corruption that could occur when changing the color of component bodies through the PCB Inspector has been resolved through investigation of a crash report.

Data Management Packages Version Control - SVN

- The Use External SVN preference is now sticking correctly.

Output Generators: Printer - PCB

- Memory allocation when working with polygons has been improved.

Output Generators: Output - NC-Drill

- Memory allocation when working with polygons has been improved.

Output Generators: Output - Gerber

- Memory allocation when working with polygons has been improved.

Output Generators: Output - ODB

- Memory allocation when working with polygons has been improved.

Output Generators: Output - STEP

- A hang up when exporting some PCB documents to STEP has been resolved.
Memory allocation when working with polygons has been improved.

Embedded Design Tools: Software platform

SDHC driver in Software Platform now has a correct include and help files.

CANopen configurator is now enabled.

Embedded Design Tools: Software Platform R10

CANopen configurator is now enabled.

Embedded Design Tools: TASKING TSK3000

Embedded compilers issue warning W557 correctly.

Embedded Design Tools: TASKING MicroBlaze

Embedded compilers issue warning W557 correctly.

Embedded Design Tools: TASKING NiosII

Embedded compilers issue warning W557 correctly.

Embedded Design Tools: TASKING PowerPC

Embedded compilers issue warning W557 correctly.

Development updating various modules

Failure of several importers to run due to an interface not supported error has been resolved.

The Create Installation Repository command now uses the correct (AD 2013) version of the installer.

Release Notes for Altium Designer 13.0

Version 13.0

Date: 01 February 2013

System Components: Altium Designer Base

VHDL/Verilog netlisters have been improved. Port to sheet entry connection via bus wire on schematic generates correct hdl for buses of size [0..0].

A new Workspace View (View>>Workspace) has been added showing a graphical preview of all the documents contained within an open Project. (shortcut: CTRL+`).

Vault-defined supply chain information is now available for BOM outputs.

Color PCB PDF files are no longer wrongly created when Monochrome is selected.

When generating a component library from a schlibrary, duplicate footprint links are no longer created.
The performance of opening documents via https has been greatly improved. View BugCrunch report #1569.

Environment variables can now be used to define paths in Preferences.

Unicode characters are now properly supported in generated PDF files. View BugCrunch report #587.

The Component Library generator is now available directly from the Schematic and the DBLib library editors.

The error "Failed to add part to global catalog" when managing a part supply chain information does not occur anymore. View BugCrunch report #900.

Component Libraries can now be created from a database library.

An error no longer occurs when adding a Part choice list for an item the name of which previously existed in a vault.

The report spreadsheet generated from the ShowDifference command is now compatible with Office 2007.

Changes have been made to allow simulation models to be found within other integrated libraries when a specific location is not specified.

Clicking on a component in a PDF file produced by SmartPDF will display the parameters of that component. View BugCrunch report #945.

System Components: Altium Designer Installation System

Plug-ins and updates page; feedback is now provided when pressing the Update All button (this will take affect with subsequent updates).

System Components: Altium Designer Localization

Localization of Altium Designer updated: Simplified Chinese, Traditional Chinese.

Updated Korean localization of Altium Designer.

System Components: PCB Support

IPC Compliant Footprint Wizard now positions the component in the center of the screen.

System Components: PCB System

New outline vertices editor for Polygon Pour, Polygon Cutout and Solid Region objects.

A new "Silk To Solder Mask Clearance" rule with new checking modes (to exposed copper and to solder mask openings) has been added. SilkscreenOverComponentPads rules in existing designs are auto-converted with parameters set to match legacy behavior.


New pre-defined 3D rotations (Top, Ortho, Front, Left, Right, Back) have been added to the PCB toolbar. Orthogonal rotation has been added to the PCB View menu when in 3D mode. View BugCrunch report #813.
New PCB object and Layer transparency settings have been added to the View Configurations dialog.

Expression error no longer occurs when using FSO or Inspector without a leading whole value.

When rotating a solid region in the PCB Editor, the region's hole are now properly rotated.

Undeclared Identifier error no longer occurs when using FSO with Embedded Board Array. View BugCrunch report #517.

PCB Heads up Display now updates the X Y values when moving cursor with keyboard arrows.

3D Rotation mode drop down menu has been updated to with the following options; Top Side, Bottom Side, Front Edge, Back Edge, Left Edge, Right Edge and Isometric.

PCB Measure Primitives command now works with locked objects when protect locked objects preference is turned on.

Smart Grid Insert in PCB List Panel now inserts objects on the correct layers.

System Components: Soft Design System

HDL Project option "Use Verilog 95" is correctly passed to Synplify synthesizer. Deselecting option enables Verilog 2001 mode.

Creating sheet symbol from VHDL containing port declarations without mode (in, out, inout) specifier has been improved and created sheet entries are of input mode now.

System Components: Data Management

Vault-defined supply chain information is now available for BOM outputs.

The error "Failed to add part to global catalog" when managing a part supply chain information does not occur anymore. View BugCrunch report #900.

When updating a design with vault-based data using the Item Manager, the current footprint is now properly taken into account.

An error no longer occurs when adding a Part choice list for an item the name of which previously existed in a vault.

Characters used in components and models names are no longer restricted when releasing them to vaults.

System Components: Schematic System

Project parameters form has been improved. View BugCrunch report #1254.

An option is now available to use the part location as the reference for schematic annotation. View BugCrunch report #357.

Schematic Blanket objects can now be placed as polygon shapes.

The Component Library generator is now available directly from the Schematic and the DBLib library editors.

The report spreadsheet generated from the ShowDifference command is now compatible with Office 2007.
Port height, border width and text font can now be individually controlled. View BugCrunch report #1782. View BugCrunch report #1687. View BugCrunch report #1190. View BugCrunch report #544.

The pin name and designator position, font and color can now be customized individually. View BugCrunch report #981.

Hyperlinks are now supported in the schematic editor.

Schematic Inspector and List panels now display the font name, size and style instead of FontID.

The Change Font button in schematic object dialogs has been changed to an edit link that also shows the chosen font details.

PCB Design Tools: Microchip mTouch

116 Support for Microchip Touch Controls has been added.

Importers and Exporters: Importer - Expedition

601 Mentor Expedition PCB and library importers now support custom pad shapes.

Embedded Design Tools: Software platform

261 Software Platform now supports JPG encoding and JPG decoding in software.

280 Software Platform was upgraded to lwIP version 1.4.0.

282 Software Platform now supports HTTPS by means of MatrixSSL sources (GPL license).

298 The mkfs for FAT filesystem now releases the allocated entry from the file descriptor table.

Hardware Support Packages: Device Support - Xilinx Spartan-6

290 Build errors during PAR when using LAX and targeting Spartan6 devices have been resolved.

Embedded Design Tools: TASKING TSK3000

258 The TSK3000 debugger now supports hardware code breakpoints in Flash memory.

Embedded Design Tools: TASKING Core

281 Embedded project linker option for global type checking is switched on by default.

Development updating various modules

A new "Silk To Solder Mask Clearance" rule with new checking modes (to exposed copper and to solder mask openings) has been added. SilkscreenOverComponentPads rules in existing designs are auto-converted with parameters set to match legacy behavior.

TSK3000A CPU has been extended with support for hardware code breakpoints.

Wishbone SPI cores have been updated to enable Push Registers to IO blocks optimization during PAR.

Microchip MTouch configurable component support has been added.

Source URL: https://www.altium.com/documentation/altium-designer/altium-designer-historical-release-notes