Altium Designer Updates

Table of Contents

Altium Designer Version 14.0 (Build: 30380) – 17 October 2013

Key highlights

Support for Flex and Rigid-Flex Design

Enhanced Layer Stack Management

Support for Embedded Components

Differential Pair Routing Improvements

Via Stitching within a User-Defined Area

AutoCAD Importer/Exporter Enhancements

CadSoft EAGLE Importer

Ibis Model Implementation Editor

Supplier Support for TME

New Installation System

Altium Designer Extensions

Preferences-based Control over Vendor Tool Usage

Support for Xilinx Vivado Toolchain

Browser-based F1 Resource Documentation

Version 14.0

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.

Altium Designer Version 14.1 (Build: 30772) – 5 December 2013

As well as numerous new features and enhancements to the Altium Vaults, this update brings the following improvements that deliver improved stability and performance.

Version 14.1

1111	PCB object transparency levels are now displayed correctly.
1281	AVS and PVS now support prefixed wildcard searches.
1282	The graphical selection issue with the Layer Stack Table has been fixed.
1285	The slow editing performance when selecting pins in the Schematic Library Editor has been fixed.
1291	An access violation with the Silk to Board Region DRC check has been resolved through investigation of a crash report.
1292	The dielectric layers have been removed from Layers drop down list in the PCB inspector and PCB list panel.
1298	A "memory error" when opening the Layer Stack Manager dialog has been resolved through investigation of a crash report.
1299	Solder Mask expansion now updates correctly when updating a footprint from a PCB library. BugCrunch #3493.
1301	An access violation when Board Regions do not have a defined layer stack has been fixed through the investigation of a crash report.
1304	PCB Collaborate, Compare & Merge panel is now showing / matching layers correctly.
1314	The differential pair rule width now displays "N/A" instead of "0" when mixed widths are defined.
1315	PCB Polygons can now be placed on solder mask layers.
1318	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.
1319	TME Supplier search has been improved to support different currency selection.
1322	PCB net name and pad number display issues have been resolved.
1323	STM32 Write to Flash dialog is enabled now.
1328	An access violation in Core3D.dll has been fixed through investigation of a crash report.
1334	Schematic editor local language resource files have been updated.
1341	Xilinx Spartan 3AN internal flash programming is working correctly now.

1342	The slow performance issue with some designs when moving PCB components has been fixed.
1350	The "Drill symbol limit exceeded, switching to letter generation" message is now displayed in the Messages Panel instead of a popup dialog. BugCrunch #652.
1365	PCB split planes no longer lose their net assignments when opening and closing the Layer Stack Manager dialog. BugCrunch #3571.
1385	An issue with DRC Report that affected some boards resulting in additional error counts has been fixed.
1386	The PCB Width rule now correctly calculates the impedance-driven width value.
1392	OrCAD Capture export now correctly handles Power, GND and test point objects.
1395	DXF/DWG Export from an OutJob no longer causes an exception error.
1402	The Release Manager no longer causes an access violation when releasing footprint models to AVS.
1403	IDF export now associates the correct component footprints when using Integrated Libraries.
1406	P-CAD and PADS importers have been updated to fix a crash or lockup with some files.
1407	An access violation when editing union objects has been fixed through investigation of a crash report.
1410	Internal Plane layers are now added correctly to the design when placing snippets or pasting internal plane objects.
1414	ODB++ Export no longer produces a crash when using embedded components.
1419	DXF/DWG Export has been updated to fix a layer mapping problem with arcs.
1429	DXF/DWG Import no longer causes an exception error when "Paper" option is selected.
1433	Step Model cache handling has been improved and crashes associated with "Cannot load 3D model from file" errors have been fixed. BugCrunch #2733.
1434	Hyperlynx export now supports 6 digit precision. BugCrunch #2899.
1479	Generated Pick and Place files now ignore Design View objects.
1487	The Copy Room Formats command now moves objects to the correct layers. BugCrunch #3566.
1512	Mirrored Design Views now generate pad objects in outputs.
1515	Camtastic "File»Export»Export to PCB" command no longer generates an exception error. BugCrunch #3556.
1523	An offline version of the Altium Installer is now available for Altium Designer 14.1. More Information
1524	FPGA Synthesis options panel correctly shows all options for Altera Quartus synthesizer.

Altium Designer Version 14.2 (Build: 31871)   – 5 March 2014

This update continues the focus on fixes, enhancements and performance improvements to our core technologies.

Version 14.2

1151	Schematic drawing line object now supports "dash dotted" line style.
1152	Schematic Pin object now includes Line Width attribute for Symbols.
1279	Schematic arcs now always display correctly at different zoom levels.
1313	Schematic documents now render non english text correctly when saving between AD10 and AD14 formats.
1331	Box.Net publisher has been updated and now uploads files correctly.
1345	The "Create Primitives From Board Shape" command now correctly uses arcs instead of line segments.

1349	DBLib and SVNLib using MySQL will now attempt to reconnect automatically to the server after PC hibernation or lost connection. BC:2824 BC:1930
1357	Subversion commit comments now support quotation marks. BC:1661
1358	Subversion commit comments now properly support different language input text. BC:2133
1363	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. BC:3193
1392	OrCAD Capture export now correctly handles Power, GND and test point objects.
1457	PCB Drill Table has been updated to include new Text Alignment and Column Width settings (available from the right-click menu).
1458	Schematic now supports vector graphic images (.wmf and .svg).
1459	PCB Polygon repour speed has been significantly improved. Speed increase up to 20x depending on number and complexity of polygons.
1478	There is no longer a hang when running the automatic loop removal tool.
1482	PCB footprint primitives are no longer removed from the footprint during interactive routing.
1484	Loop removal will no longer remove incorrect track.
1485	PCB Polygon Pour quality has been improved when using hatched style and polygon cutouts.
1486	PCB Design Rule dialog will now grey out disabled rules making it easier to distinguish between enabled and disabled rules.
1516	The PCB Library Report can now support non english characters.
1529	Schematic documents now render text correctly when using non english language system settings.
1531	Smart PDF now supports an option to include "Global Bookmarks for Components and Nets".
1547	PCB Copy Room Formats command will no longer copy the board region.
1550	PADS Library importer has been updated to fix specific cases where components were not created.
1551	DxDesigner importer has been updated to fix connection issues with off grid objects.
1555	PCB Via Stitching has been improved and will now properly use the clearance rule settings.
1579	Schematic Formatting Toolbar can now change text font and size settings.
1580	STEP model clearance errors between top and bottom layers has been fixed.
1581	Selecting PCB components in Single Layer Mode will now only select components which are on that layer. BC:3545 BC:997
1582	Schematic directives no longer produce false error reports when single documents are compiled.
1584	The PCB Printout Properties dialog now opens correctly even if the PCB is not the top document window.
1585	Use of the backslash character (\) in an Output Job container file name no longer inserts an underscore character (_).
1586	The PCB Drill Table is now correctly re-positioned when using the Move»Move Selection by X,Y command.
1587	Scripts using SchServer.RobotManager.SendMessage can now be run successfully,
1588	PCB Undo now works correctly when primitives are deleted from a component.
1594	The PCB "Update Free Primitives From Component Pads" command no longer removes vias from their nets.
1601	The Locked attribute is no longer updated when changing the Pad or Via test point settings. BC:3758
1604	Deselecting the Preview top - bottom layer command for a drill table, will no longer cause an Access Violation.
1608	PCB polygon thermal relief calculation causing incorrect DRC violations in some situations has been fixed.
1612	IDF Export has been updated to fix a hang when exporting from specific files.
1616	The Libraries Panel now remembers the layout when the application is restarted.
1617	There is no longer a discrepancy in the display of top and bottom SM layers when opacity is set to 50%.
1618	Smart PDF now displays the correct component parameters when using Variants. BC:2517 BC:3201
1619	BOM generation now supports variant supplier and supplier part number parameters. BC:3222
1620	Variants now support the .COMMENT and .REFDES special fields. BC:2083

1624	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. BC:59 BC:839
1626	Automatic adjustments to track end-points when moving a selected track can now be reliably undone using the Undo feature. BC:431
1627	The Schematic "Break Wire" command now works with wires that have smallest width setting. BC:829
1630	Soldermask now correctly remains as defined for regions and polygons after saving and reloading.
1636	Net Antennas with a terminating via are now correctly detected by the DRC. BC:1491
1647	Chinese and Russian text is now correctly encoded when exporting a Schematic to DXF.
1652	Via Stitching by constrained area no longer causes shorts with polygons on internal layers.
1662	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. BC:441
1664	The Schematic Library Parameter Manager now displays selected pins when "Selected Objects Only" option is enabled. BC:875
1665	The query InFromToClass will return all primitives in the from-to, and will no longer miss track segments. BC:1229
1668	Vault now supports batch download of selected items.
1697	There is no longer an Access Violation when generating a Gerber from a PCB containing an active drill table.
1700	The legend in a generated Drill Drawing now works correctly when using Characters for the .Legend symbols. BC:2929
1701	FPGA to PCB Project Wizard now uses installed FPGA integrated libraries to correctly place schematic symbols in generated PCB project.
1706	DXF Exporter now correctly supports slot holes (zero line width and donut), and square holes.
1711	Debugging embedded project targeting discrete CPU no longer causes a crash on exit.
1714	There is no longer an error when using the parameter "=VariantName" as the folder name in an Output Job file. BC:3826
1836	An issue with the Mixed Simulation engine failing to create the correct netlist has been resolved.
1837	There is no longer an issue when exporting files for route using SPECCTRA. BC:3833
1849	There is no longer an Access Violation when printing a schematic containing a large graphic image.
1850	IDF Exporter has been fixed to resolve the issue with duplicated board outlines. BC:3696
1857	The Layer Stack Manager dialog now correctly opens in active display space, in accordance with monitor configuration.
1874	The "ssl_error_unsafe_negotiation" error no longer occurs when opening URLs inside Altium Designer.
1911	There is no longer a crash in Supplier_TME when closing Altium Designer.
1913	An access violation caused by an undo operation in PCB has been fixed through the investigation of a crash report.
1934	The correct board height is now used when exporting to STEP.
1936	An access violation associated with importing preference files has been fixed.
1944	Version 6.3 and 6.5 Eagle files can now be successfully imported to Altium Designer without error or Access Violations being generated.
1945	There is no longer a crash when accessing SchServer.RobotManager.SendMessage through a script.
1947	The graphical preview of the schematic part is once again available when accessing the Port Map tab for a Sim Model.
1948	Parameters with spaces in their names can now be used indirectly through a part's Comment field. BC:1129
1954	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.
1961	The version number is now correctly checked for the Lattice Diamond toolchain.
1962	There is no longer an Access Violation when clicking on All Folders after an initial Vault search.
1963	Vias are no longer removed from existing Via Stitch objects when cancelling the Via Stitch by Area command.
1964	Grouping Vault Search results no longer causes an exception error.
1965	A crash with the SCH List panel when changing scope is now resolved.
1967	TMDS33 constraint for Xilinx Transition Minimized Differential Signaling IO standard is correctly translated to UCF TMDS_33 value.
1968	CoreGenerator targeting Xilinx devices now correctly generates the FIFO memories for data widths other than multiples of 8.

1981	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.
1985	The layer for a placed via can no longer be changed through the PCB List panel.
1988	The PCB Signal Integrity design rules are now analyzed by the Signal Integrity extension.
1991	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
1993	An issue whereby the "Flipped on Layer" option was being erroneously set during component placement, has been resolved. BC:3796 BC:3843
1996	The Process Flow after successful synthesis is no longer reset when performing a manual refresh (F5).
1998	FPGA build status in Devices View is correctly updated when Altera Synthesizer is used.
2067	A new FPGA compiler option has been added to the FPGA General Preferences to "Clear compiler output messages".
2070	FPGA projects are now linked properly to PCB projects when components are placed from the Vault.
2079	Embedded extension adds support for TI Stellaris and NXP LPC2000 families of discrete microcontrollers.
2080	Cortex-M3-based Stellaris Support extension enables programming and debugging for Texas Instruments Stellaris family of microcontrollers.
2081	ARM7-based LPC2000 Support extension adds programming and debugging for NXP LPC2000 seriesdiscrete microcontrollers.
2083	Variant values are now displayed correctly for Multi-Channel and Complex Hierarchy schematic projects. BC:3977
2104	The Preferences option in the Subversion update repository dialog now works correctly and no longer causes an exception error.
2106	PCB footprint names that use special characters (such as * or /), no longer cause footprint not found errors and other editing, browsing and synchronizing issues. BC:3952
2109	Rotating schematic .EMF graphic files no longer causes an exception error.