What's New in Altium Designer 6.6

Summary

Altium Designer 6.6 brings significant refinements to Variants combined with a number of smaller enhancements and improved system-wide support for existing technologies. Many of these improvements are based on feedback directly from you, the engineers and designers developing designs in Altium Designer.

Altium understands that improving the electronic product design platform isn't just about adding new features, but also about refining and strengthening existing ones. Electronics technology moves fast and your design systems need to keep up.

Altium Designer 6.6 delivers a number of enhancements designed to increase your effectiveness, improving features you depend upon to deliver better designs faster. Variants has been upgraded to support stronger support for printing and varying parameter values by updating from your libraries. In addition, a host of new commands and capabilities have further expanded the support of Variants system-wide.

With this release, support for a new pad shape Rounded Rectangle has been added. Recently-introduced 3D STEP has been expanded to include import from your preferred mechanical CAD system. Brand new example design and tutorial for Signal Integrity using one of Altium's own daughterboard designs show you how to determine optimum slew and drive settings for specific pins on an FPGA device.

These are just some of the new enhancements delivered by this significant new release of Altium Designer. To learn more about the new capabilities and productivity benefits offered in Altium Designer 6.6, read on!

Seeing is believing – read more and watch demos of Altium Designer 6.6

Altium's DEMOcenter gives you the opportunity to walk through the extensive design capabilities of Altium Designer featured as individual demos, each only taking a couple of minutes, making this a quick and easy way for you to browse the areas of most importance to you.

If you'd like to read more about updates in Altium Designer 6.6, as well as watch short videos about some of the exciting new features, then visit the **What's New in Altium Designer 6.6** page on the website and enjoy the action. Click the link below to read more and watch the videos.

http://www.altium.com/WhatsnewinAltiumDesigner/

Improved – Variants

Appreciating the importance of working with variants in the design process, Altium Designer 6.6 delivers new commands and capabilities to the *Assembly Variant Management* dialog as well as improvements system-wide:

- Ability to vary parameter values by updating from a library – browse the libraries to select a component and all parameters are updated from the library component.
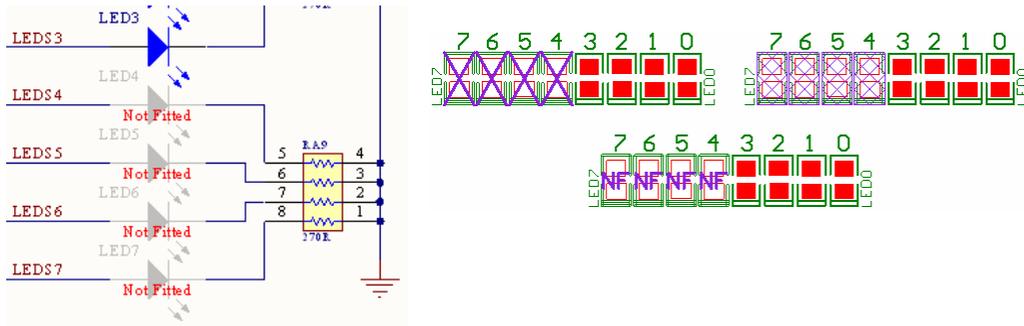
- Stronger support for printing Variants.

Figure 1. New options for handling schematic and PCB variant printing.

- **Smart PDF** and schematic output jobs support printing of physical documents that include variant information.
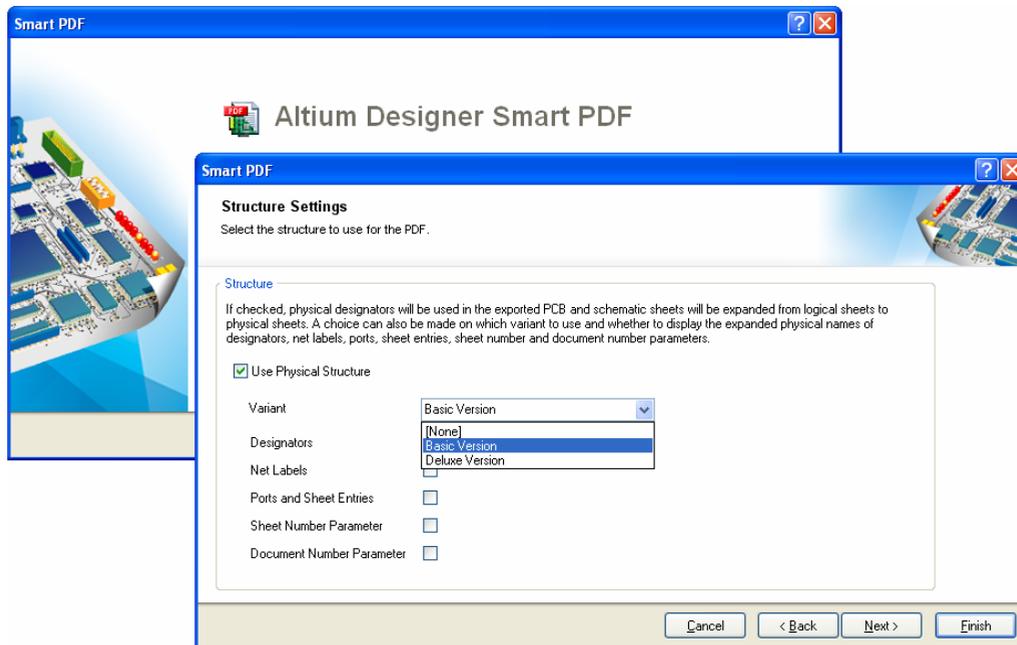
Figure 2. Specifying use of an assembly variant as part of the Smart PDF setup options.

- Multiple variant fittings can be edited simultaneously.

- A whole variant can be copied and pasted.

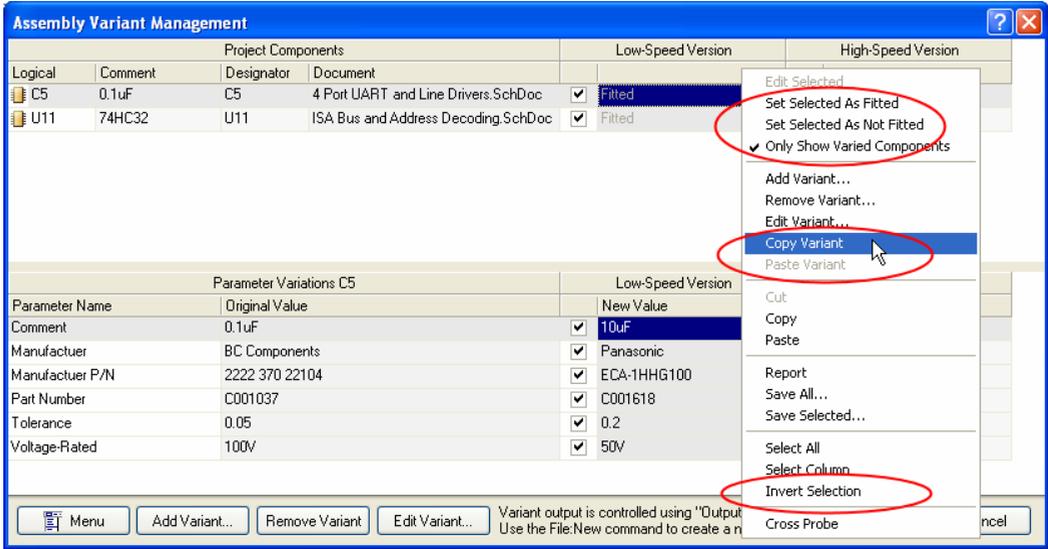- An **Invert Selection** command, as well as other new commands, are available.

Figure 3. Many new commands now available from the right-mouse menu in the Assembly Variants Management dialog

- Component and parameter variation grids can be configured to show only values that are different from the original.

- BOM file name format supports display of variant names in the following format: <Bill of Materials> <Project Name> (<Variant Name>). You can also make a choice of which variants to use with your BOM.

- Parameter variation values can be generated in a detailed report.

- Variant information is preserved during re-annotation.

New – Rounded Rectangle pad shape

Altium Designer 6.6 expands support for pad shapes to include new *Rounded Rectangle*. Rounded Rectangle pad shapes are defined in the redesigned PCB *Pad* dialog, giving you immediate visual feedback on the design of the pad.

Support for Rounded Rectangle pad shapes include:

- User-defined **Corner Radius** allows you to specify the corner radius and allows a range of 0 – 100%.
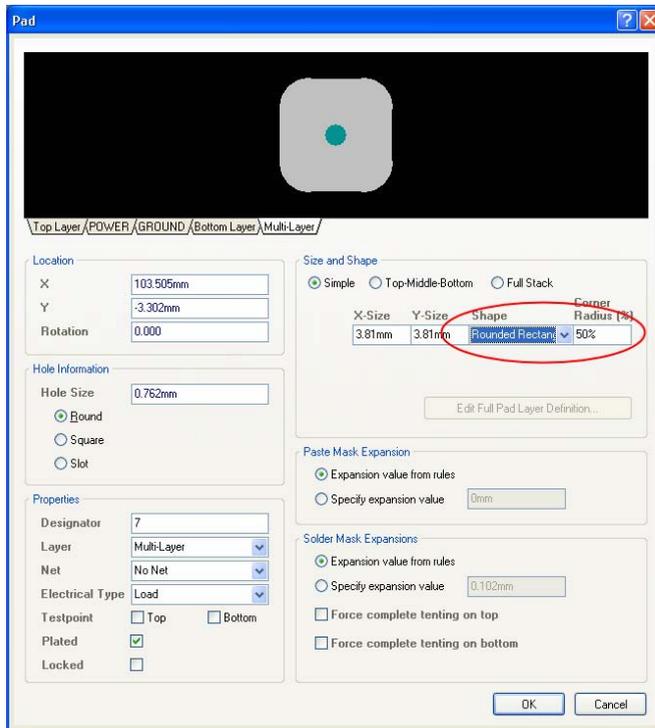
- Updated PCB *Pad* dialog giving instant visual feedback on the pad design.

Figure 4. Create Rounded Rectangle pad shapes in your PCB.

- Updated PCB *Pad Layer Editor* dialog shows graphic representation of pad design.
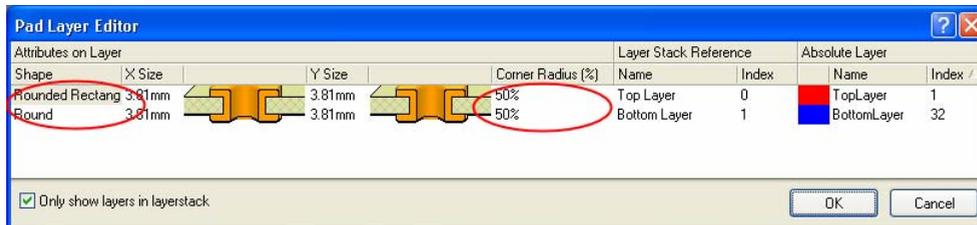
Figure 5. Define your pad size and shape for each layer in the pad stack, on a layer-by-layer basis.

New – STEP (3D File Format) import

STEP, the **ST**andard for the **E**xchange of **P**roduct model data is becoming a preferred standard for ECAD to MCAD data exchange – allowing transfer of 3D models between CAD applications. Altium Designer 6.6 expands new STEP support to include importing a 3D STEP format file from your preferred mechanical CAD system.
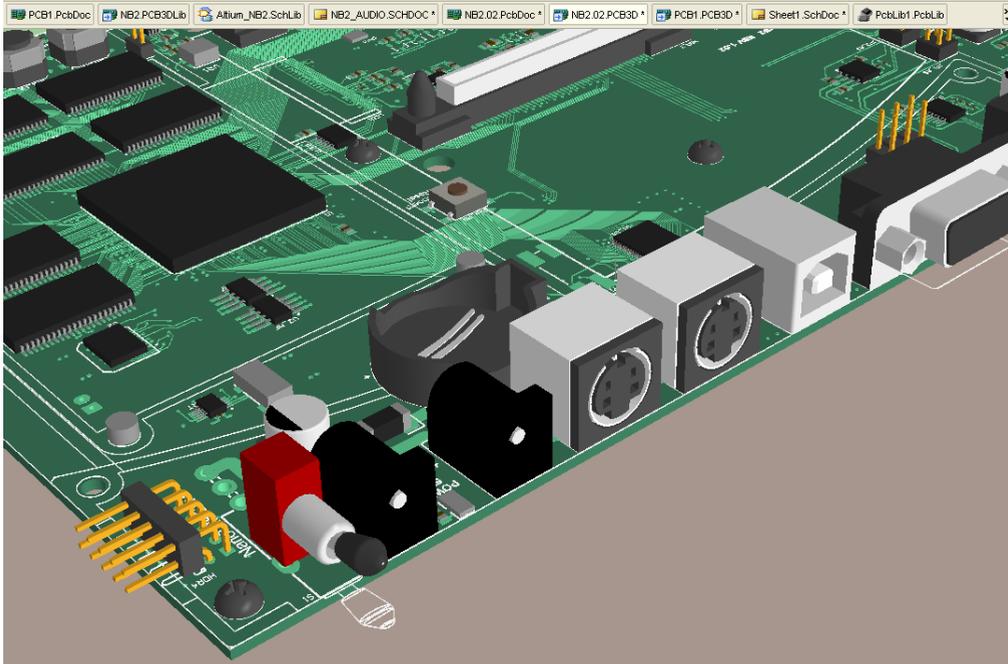

Figure 6. Once imported and associated to components, 3D STEP format models allow accurate representations of PCB boards to be created and transferred to Mechanical CAD applications.

3D STEP models are imported into PCB3D Library files (*.PCB3DLib) files and then associated with the component symbol in the same way that other models are associated with component symbols.
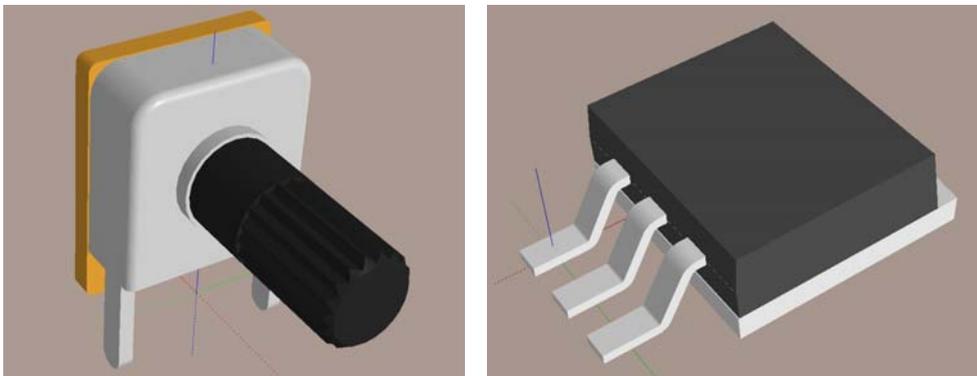
Figure 7. Imported 3D STEP components as seen in Altium Designer 6.6.

Improved – IPC Footprint Wizard

Available through the **Tools** menu when a PCB library is the active document, the new IPC Footprint Wizard creates IPC-compliant component footprints. Rather than working from footprint dimensions, the **IPC Footprint Wizard** uses dimensional information from the component itself in accordance with the standards released by the IPC. For Altium Designer 6.6, it has been enhanced by the addition of a preview window and support for additional package types.

Some of the **IPC Footprint Wizard** enhancements include:

- A variety of new footprint generators are included, tailored to suit your board's density – Chip Components (Capacitor, Inductor and Resistor), QFN, SOJ, SOT23 (3-Leads, 5-Leads and 6-Leads), SOT143/343 and SOT223.
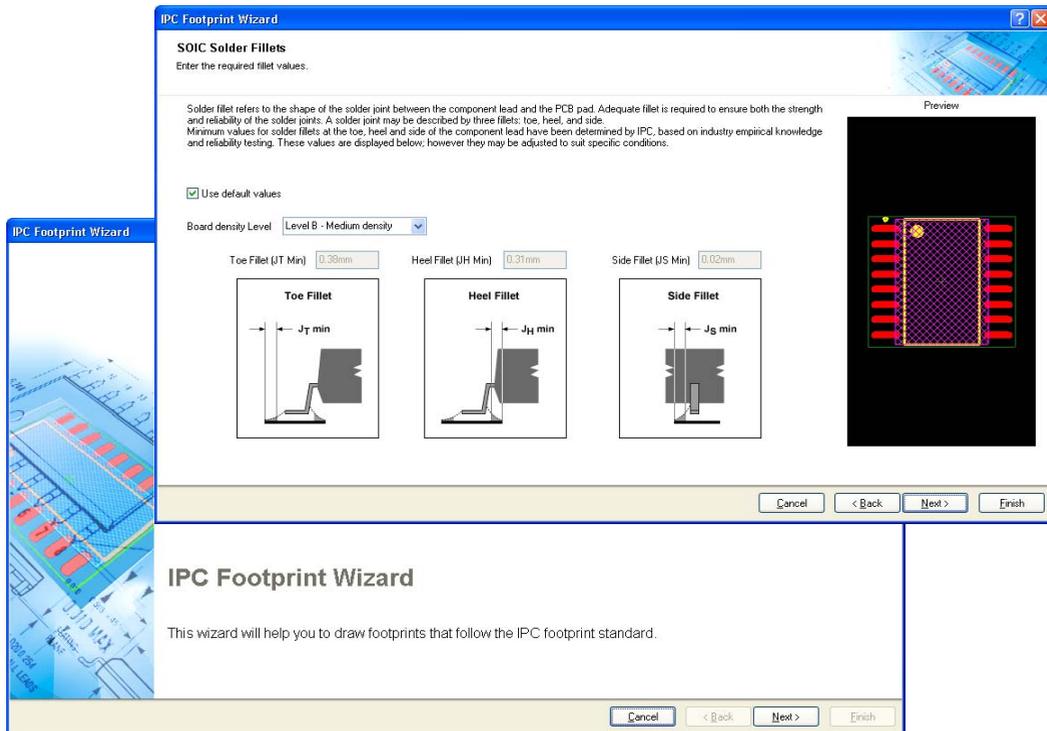
*Figure 8. Quickly create IPC-compliant component footprints based on component dimensions in the new **IPC Footprint Wizard**.*

- Overall packaging dimensions, pin information, heel spacing, solder fillets and tolerances can be entered and immediately viewed.

- Mechanical dimensions such as Courtyard, Assembly, and Component Body Information can be entered.

- Wizard is re-entrant and allows reviewing and making adjustments easy. Previews of the footprint are shown at every stage.

- The finish button can be pressed at any stage to generate the currently previewed footprint.

AR0140 (v1.0) November 07, 2006

New – Signal Integrity example

Determining how hard you can drive signals before ringing and crosstalk start to affect performance is directly related to finding optimum slew and drive settings for specific pins of an FPGA device. A new example has been added to the Signal Integrity examples folder (`\Examples\Signal Integrity`) that explores this. This example is based on one of Altium's own daughterboard designs, the NBP28, which features a Xilinx Spartan 3, a Sharp LH79520 incorporating an ARM 7 processor, SRAM and Flash RAM.

An accompanying tutorial - *TU0126 Checking Signal Integrity on an FPGA Design* - explores in detail how you can use Altium Designer's Signal Integrity Analyzer to determine optimum Slew and Drive settings for the data pins of the Spartan 3 device in this design.

Figure 9. Set criteria to identify coupled nets.

The tutorial includes:

- Setting up IBIS models for devices in your design.
- Running reflection analyses on data lines at different Slew and Drive settings.
- Identifying coupled nets and analyzing crosstalk.
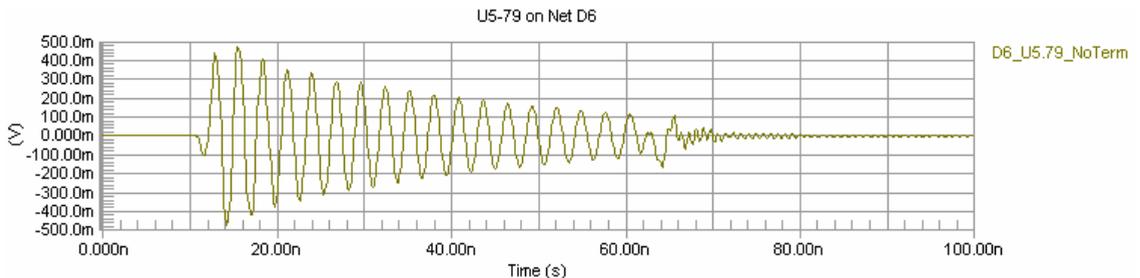
Figure 10. Experiment with Slew and Drive settings through the FPGA Signal Manager to see the effect on crosstalk.

A link to this tutorial can be found in the **Documentation Library » Design Verification » Signal Integrity** area of the **Knowledge Center** panel.

Alternatively, the document can be found directly within the `\Help` folder of the installation.

Improved – Bill of Materials

Data from both the schematic and the PCB can now be included into a single Bill of Materials (BOM) report – source information is based on property information taken from the PCB in the event you need to customize and use the report generation for more than a BOM. An example would be for generation of a pick and place file where every placement machine wants the data (such as X, Y location) in a different column order and in different file formats.

With the required schematic or PCB documents open, select **Reports » Bill of Materials**. The *Bill of Materials for Project [project_name] (PCB_document)* dialog displays. In the parameter listing, the 🖳 icon distinguishes a PCB parameter for one or more placed components in the project.
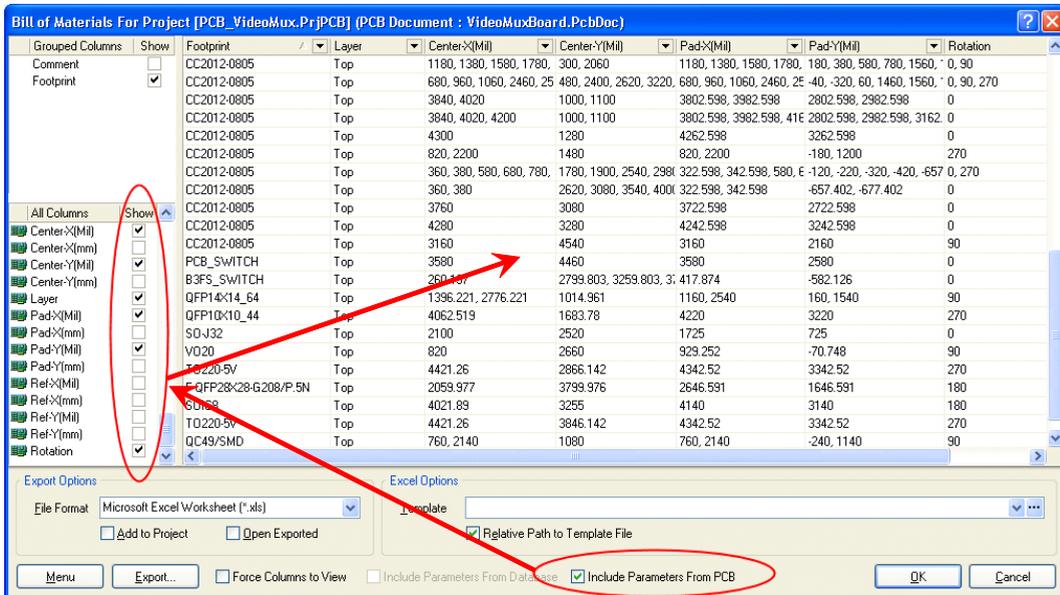
*Figure 11. When configuring the Bill of Materials report using the Report Manager dialog, simply enable the **Include Parameters From PCB** option. This option will only be available if there is a PCB document in the project file.*

Exporting your report

Parameters are a universal feature of Altium Designer and can be added to the project, a document, a component and other objects. Project and document parameters can be extracted from the design and included in the BOM report.

When exporting your data from the BOM, you can include two new PCB document fields in your Excel templates:

* **Field=PCBDataSourceFullName** – displays the full name of the PCB data source.
* **Field=PCBDataSourceFileName** – displays the file name of the PCB data source.

Improved – Drill Drawing Symbols table

Altium Designer 6.6 expands support for slotted holes in PCB pads with the addition of slot information added to the Drill Drawing Symbols table – providing more options for board fabrication and smoothing the process to manufacturing. Appropriate slot information is included at the time of output file generation.

Improvements for slotted holes in Drill Drawing include:

- Support for extended numbers of symbols improved to automatically switch to letters after the graphic symbols run out.

- Letter symbols now automatically allow an extended sequence (A…Z, AA, AB, etc.).

Figure 12. Enhancement for improved readability as well as slot information can now be viewed in the Drill Drawing.

- Reworked for greater overall presentation, the Drill Drawing Symbols table features the addition of headers and column separators. Symbols are drawn in the table at the same height as the rest of the table text for improved legibility. This allows for a clearer drill drawing utilizing small symbols.

Revision History

Date	Version No.	Revision
07-Nov-2006	1.0	Altium Designer 6.6 release