



What's New in Altium Designer 6.9

Summary

Article

AR0146 (v1.0) March 6 , 2008

Altium Designer 6.9 brings significant refinements to 3D PCB Visualization 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 6.9 delivers a number of enhancements designed to increase your effectiveness, improving features you depend upon to deliver better designs faster. PCB Visualization has been further enhanced with new 3D STEP technologies. 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.

Embedded tools support for FPGA development continues to improve, enabling you to take advantage of new core technologies and optimize your power efficiency. Expanding options for peripherals, Altium Designer 6.9 delivers three new peripheral cores, a power monitor, and context-sensitive help support for C keywords and standard functions.

With this release, new web updating through the web portal will be introduced as a means of taking advantage of a wide range of future online services yet to come!

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 6.9, read on!

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

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

<http://www.altium.com/Evaluate/DEMOcenter>

New – STEP Models into 3D Bodies

When visualizing PCBs and components in 3D, it's now possible to import 3D STEP models to represent component bodies to provide further realism. STEP models can be loaded directly into component footprints in the PCB Library Editor.

The associated *3D Body* dialog now features a Model section allowing you to load, update or remove STEP model files quickly and easily. New commands in the PCB Library Editor **Tools** menu and *3D Body* dialog have also been introduced to accurately manipulate and orient STEP models in the 3D workspace.

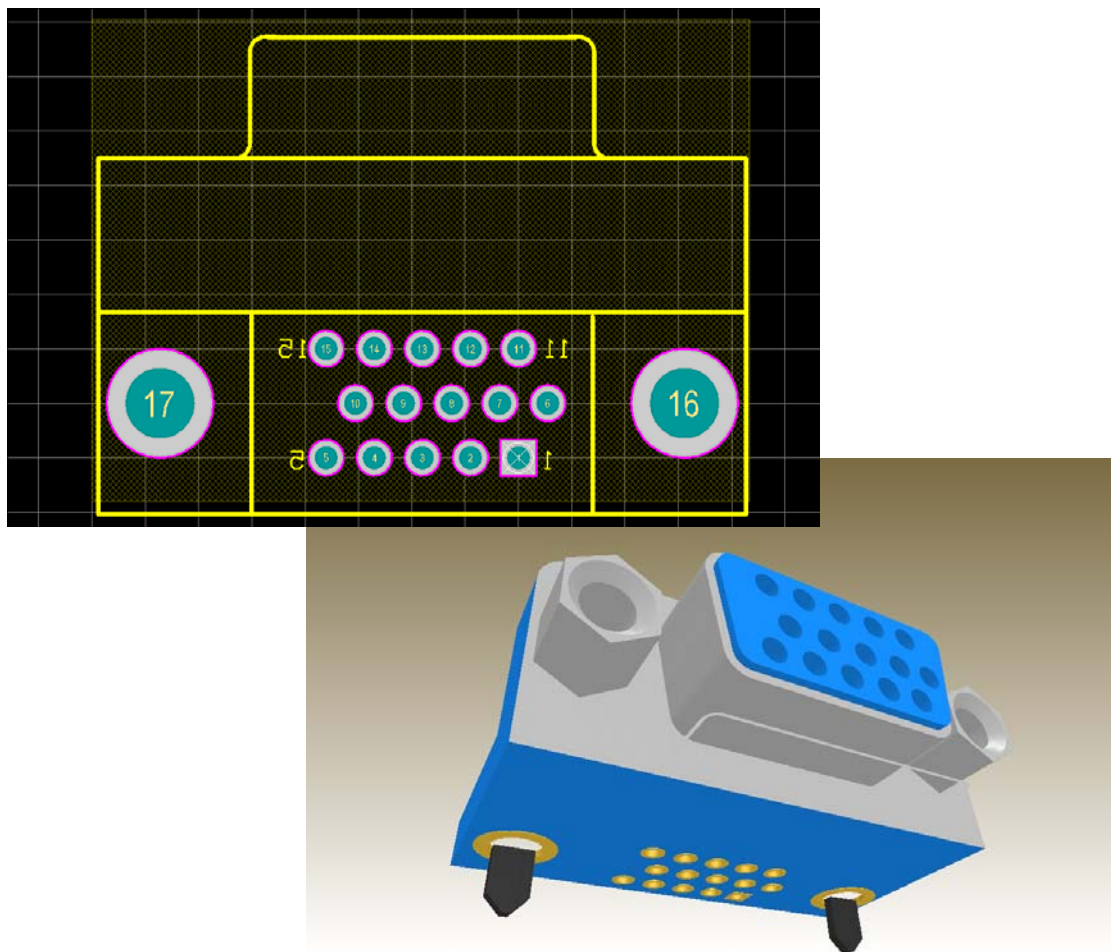


Figure 1. Amazing transformation of your component bodies in 3D using STEP models are now possible.

Viewing the models is easy – once you're in 3D mode, hit the 'L' shortcut key to bring up *View Configurations* dialog and turn on the **Show Step Models** checkbox.

New – 3D STEP Export

Altium Designer 6.9 expands new STEP support to include exporting a 3D STEP format file to your preferred mechanical CAD system. With the introduction of 3D Visualization in the PCB Editor, complete boards can be exported as 3D STEP AP214 file format.

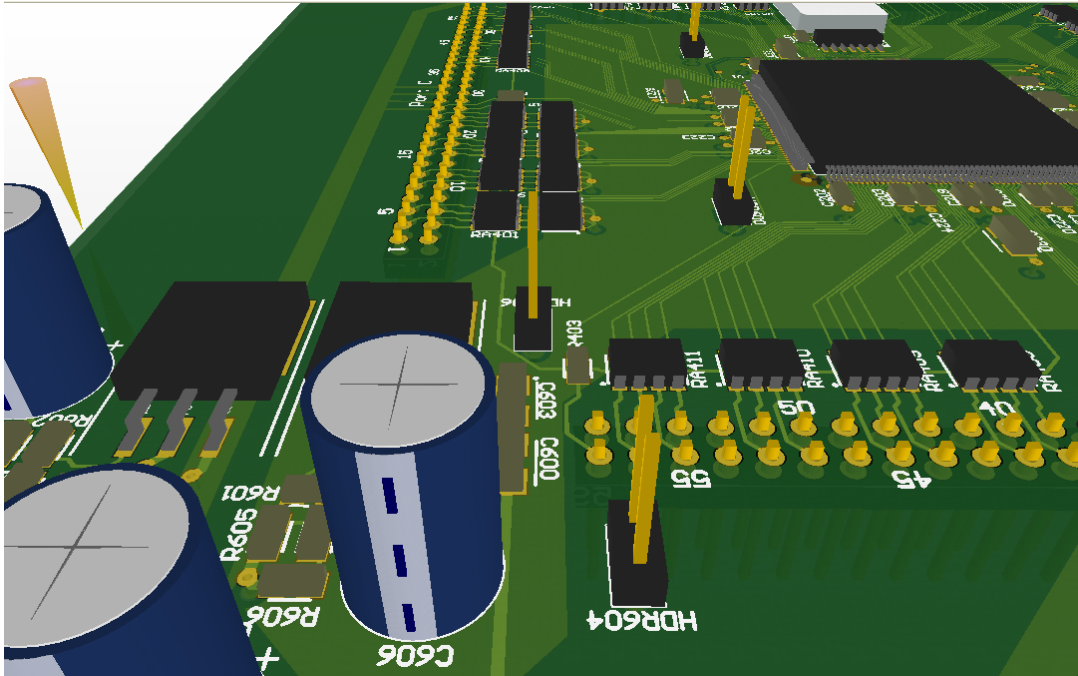


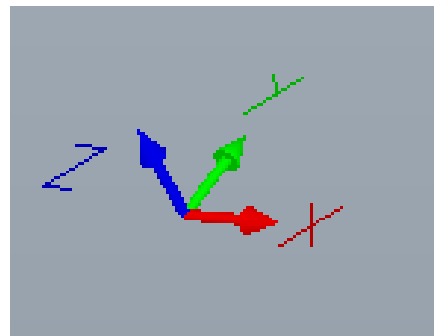
Figure 2. Export options allow you to customize output according to all or selected components, STEP models or Extruded Component Shapes, and All or selected Holes.

The new 3D export feature is launched from the PCB Editor, **File » Save As**. Simply change the Save As file type to Step File (*.step, *.stp) and press **Save**.

New – 3D Origin Marker

New tools in Altium Designer 6.9 help accurately orient and position imported STEP Models in the 3D workspace, the first of these being the 3D Origin Marker.

The **Physical Materials** page of the *View Configurations* dialog has options to display an origin marker when in 3D view mode. The marker represents the 0, 0, 0 positions for the X, Y, and Z-axis respectively.



New – Transparent Layer Mode options

Further complementing the capabilities of PCB Visualization, additional options in the **PCB Preferences** give added flexibility when working with Transparent Layer Modes.

Layer ordering allows more control over the layer drawing order for transparent layer modes.

Full Brightness and Ordered Blending (shown below) enables the current or selected layer to appear on top of other layers (according to the Layer Drawing Order) in transparent layer mode and also for the colors of objects in transparent layer modes to have improved brightness and contrast.

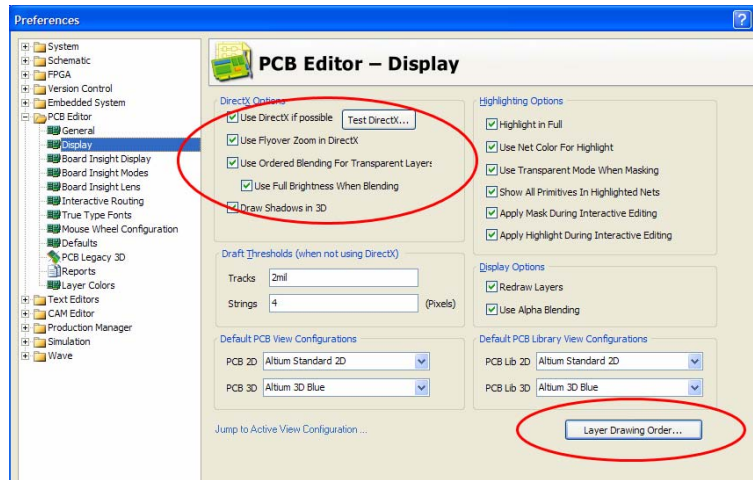


Figure 3. New options for Transparent Layer Modes are found in **Tools » Preferences PCB Editor** page.

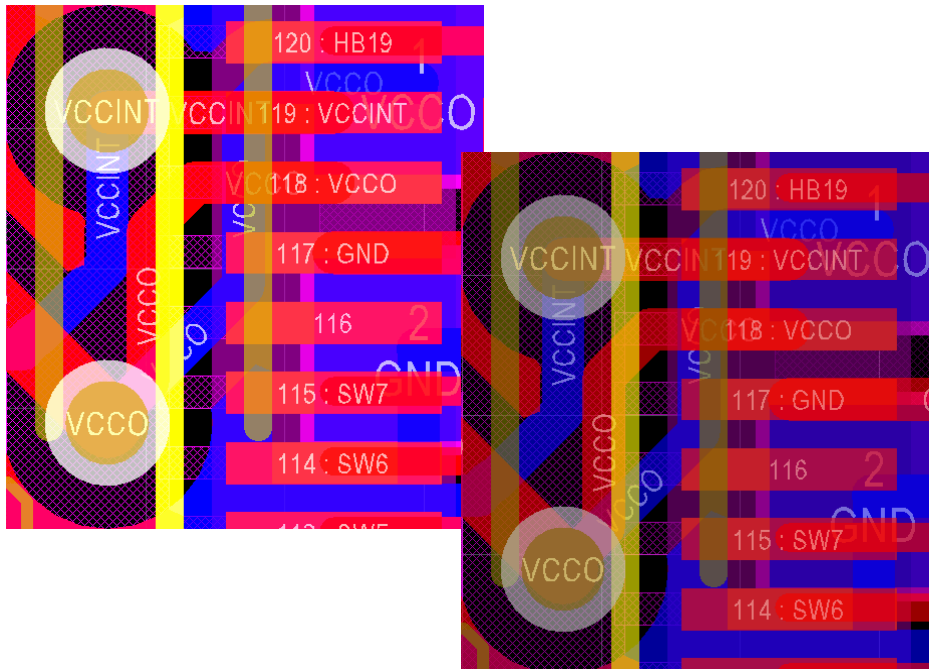


Figure 4. Transparent Mode is shown with **Full Brightness** on the left and in **Ordered Blending** on the right.

New – 3D Shadow and Object Color by Layer

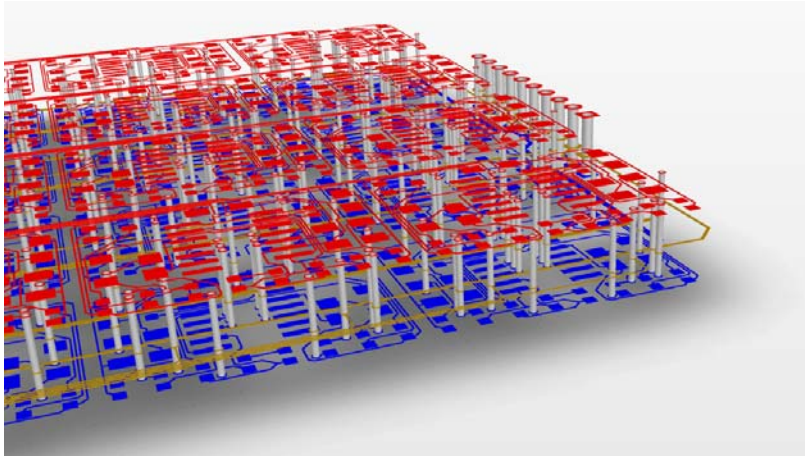


Figure 5. Enabling Shadowing and Object Color by Layer makes it easier to discern the layers of objects from the 3D view.

Layer option in the **Physical Materials** page of the *View Configurations* dialog that allows display of objects either in their system-based 2D layer color or using the colors specified in the dialog

This effect can be easily added to your 3D design through the *Preferences* dialog giving enhanced visibility in dense designs. Shadowing can only be seen when viewing the PCB from above which helps prevent confusion between the viewing the top and bottom of the board.

As a complement to shadowing, there is also Object Color by

New – 3D Image onto clipboard

3D image to clipboard takes a snapshot of the main design window view in 3D and stores it as a bitmap on the Windows clipboard for pasting into other applications. Whatever is displayed in the main window of Altium Designer (bar panels and dialog boxes) is what will be placed on the clipboard. You even have control over the resolution of the bitmap image.

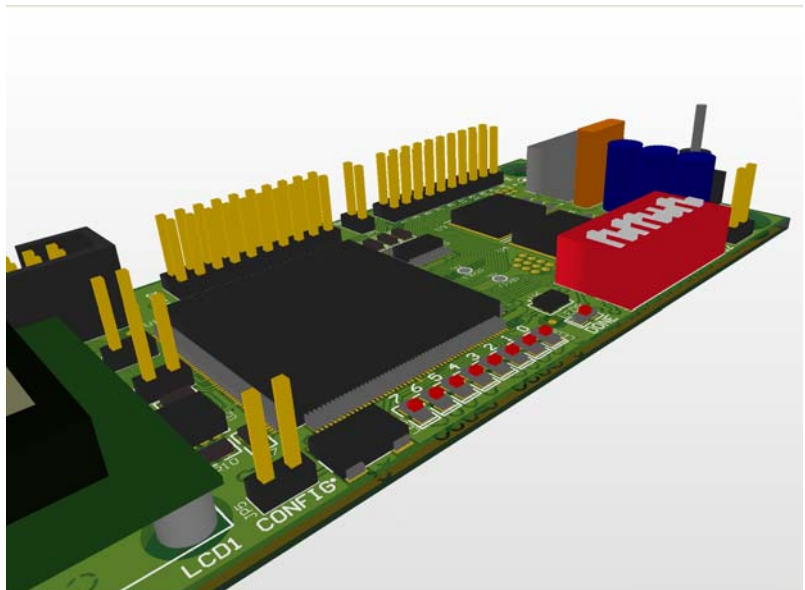


Figure 6. A representative bitmap pasted from Altium Designer to an external application.

New – Jumper connections

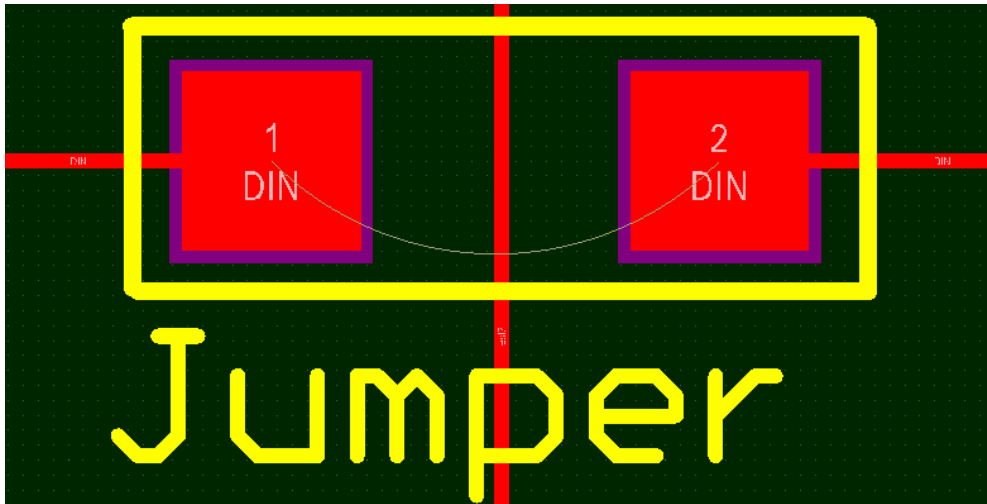


Figure 7. Jumper connections are shown as curved connection lines in the PCB Editor.

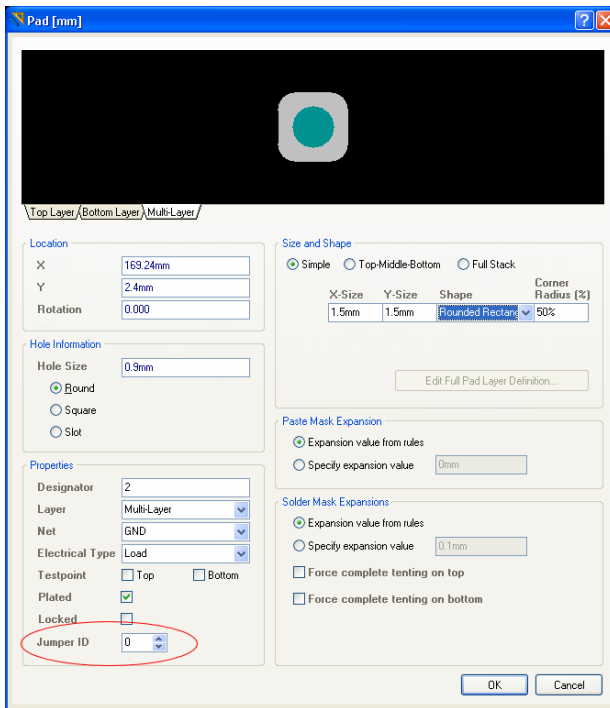



Figure 8. Jumper IDs can be configured from within the **Pad** properties dialog

A new jumper feature allows you to set a jumper connection (a physical connection between pads but not using tracks on PCB, a separate wire instead) between pads on a component. This is especially useful when crossing tracks on a single layer board.

Jumper connections define electrical connections between component pads that are not physically routed with primitives on the PCB. These are especially useful on single layer boards, where a wire is used to jump over tracks on the one physical layer, or even as complex as designing with a 'crossover' switch. The **Design Rules Checker** will not report jumper connections as unrouted nets.

Pads within a component can be labeled with a Jumper ID value from within the *Pad* dialog. Pads that share the same Jumper ID and electrical net tell the system that there is a legitimate, although physically unconnected, connection between them.

New – Units Toggle

Many dialogs now feature a units toggle control  in the top-left corner that will change the units of measurement currently used in the dialog between the metric and imperial. Shortcut **CTRL + Q** can also be used in dialogs where this control is present. Toggling units at any time does not affect system accuracy as all numerical calculations are carried out at system resolution.

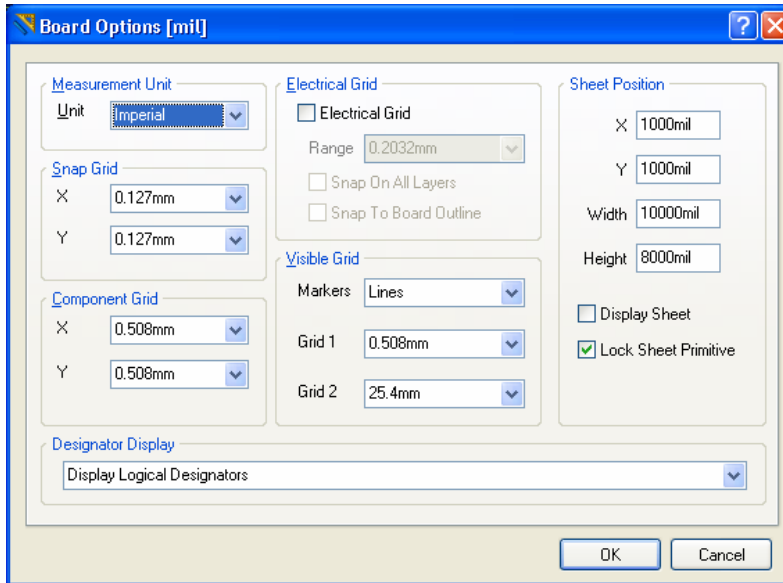


Figure 9. The current unit of measurement displays in the dialog title area as shown here. You can also control the display precision and rounding of metric units through the **Tools » Preferences** dialog.

Improved – Special Strings

Especially useful for design documentation, special strings act as placeholders for PCB design or system-based information and are only populated with information when generating manufacturing output data.

To use a special string on a PCB, place a string object and set its text to be one of the special string names. Special strings provide manufacturing records for the software build number of the version of Altium Designer that was used during output generation and for the current revision number of the document.

- **.Application_BuildNumber** – places the version of Altium Designer on the PCB document that the PCB is currently loaded in. When generating Gerber output, this string will record the software build that the design was created on.
- **.VersionControl_RevNumber** – once placed, displays the current revision number of the document. Version control must be used for this string to contain any information.
- **.Computer Name** – places the name of the machine that the PCB is currently loaded on.

New – Design Refactoring

Making changes on the fly and restructuring your schematic design is more flexible than ever with Design Refactoring. Design Refactoring converts existing Schematic Sheets into Device Sheets, and converts Device Sheets into Schematic Sheets. Parts can also be converted to ports, and parts can be converted to sheet symbols although it's worth mentioning in this instance that the Unique ID isn't maintained as one part may be converted into many ports.

Subcircuits can be moved easily between schematic sheets as well. Refactoring of sheets is performed at the Sheet Symbol or Device Sheet Symbol level. Design Refactoring automatically maintains synchronization with the rest of your design, ensuring that subcircuits in the schematic are always linked to their physical instances in the PCB through the maintenance the Unique ID of all subcircuits.

Description: MONITOR

MONITOR

Monitor.SchDoc

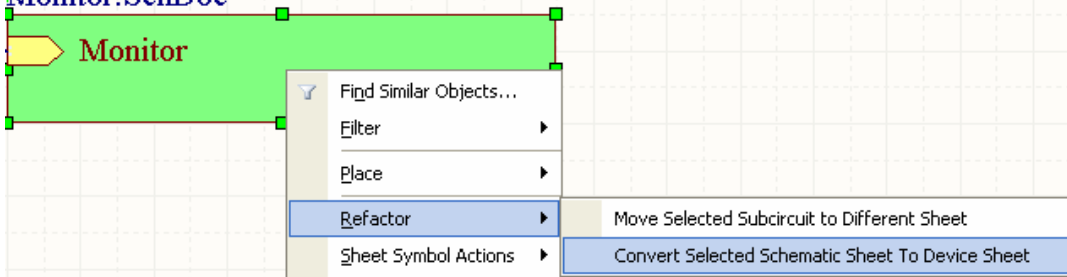


Figure 10. Converting a Schematic Sheet into a Device Sheet using the Refactor feature

Found in two convenient locations, Refactoring commands can be launched from both the **Edit** and right-click menu in a schematic document.

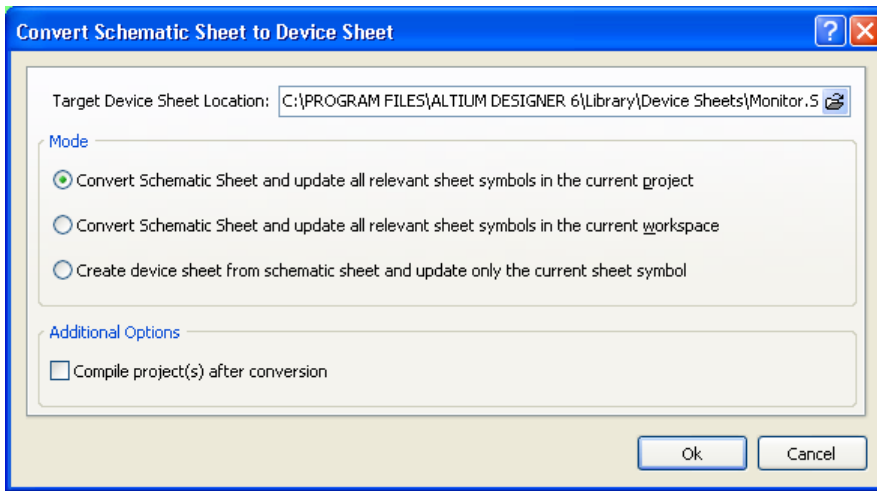


Figure 11. Converting a Sheet into a Device Sheet Symbol using the Refactor feature

Improved – Publish to PDF

If you publish a specific set of documents to PDF on a regular basis, you can now use a Batch flag to specify your document set and run the **Batch Publish to PDF** command. **Batch** preferences are saved to the Output Job File. You can create multiple Output Job Files with different batch preferences for the same project to suit your needs.

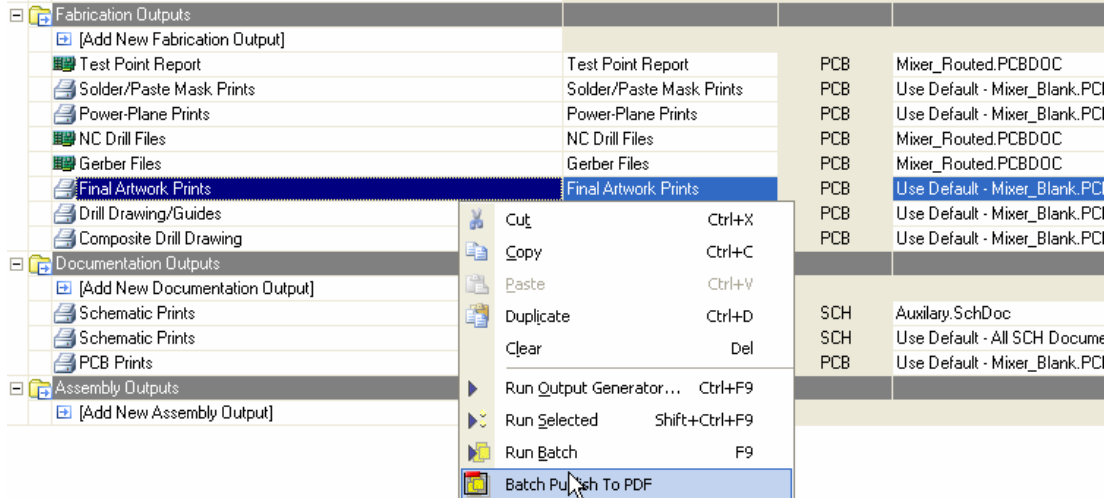


Figure 12. Access the **Batch Publish to PDF** command from the Tools menu or by right clicking in the Output Job Editor and selecting it from the pop-up menu that appears.

New – Web updates

Web updates are now obtained via your SUPPORTcenter account. From the **Home Page**, simply log into your account using your SUPPORTcenter credentials (available through either your local Altium Sales and Support Center or Reseller, or you can view contact details listed at <http://www.altium.com/contacts>). Then, use the **Check for Updates** link to get the latest updates.



Figure 13. Getting your SUPPORTcenter contact details for your account are important for obtaining web updates.

New – Power Monitoring for the Desktop Nanoboard

Power consumption is a critical issue in all appliances and particularly portable devices. Power requirements for FPGAs can be complicated and it's essential to be able to accurately determine power drawn in real time in order to assess the appropriateness of the devices for battery-powered applications. The newest Desktop Nanoboard now includes a new optimization tool **Power Monitoring** for maximizing power efficiency. On-board current sensing circuitry enables you to analyze and monitor power consumption in your design and allows you to make direct comparisons between different devices and circuit configurations.

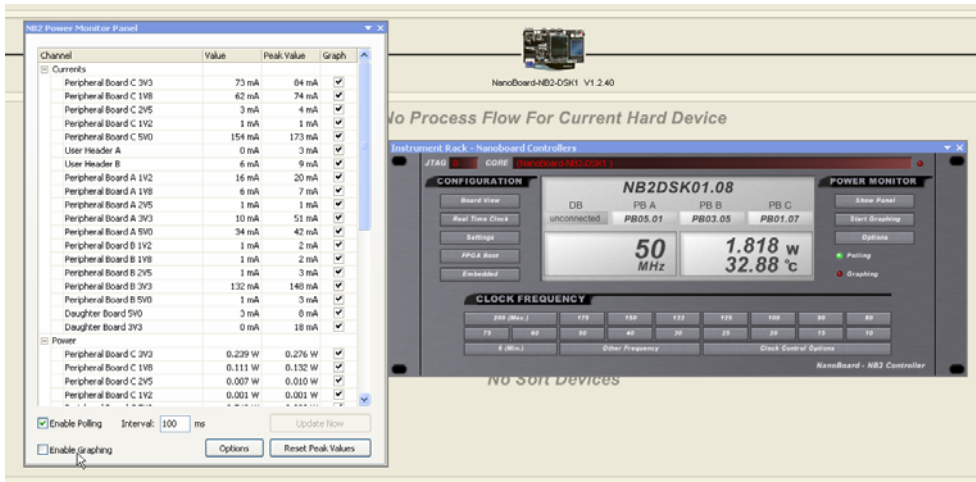


Figure 14. In the **Power Monitor** panel, values for **Current** and **Power** are displayed for individual channels and the aggregate of the power consumption across the voltage rails is also calculated.

Access the **Power Monitor** panel through the **Instrument Rack - Nanoboard Controllers** panel and click the **Show Panel** button in the Power Monitor section. Values can also be graphed to show the power consumption over time for a particular hardware configuration and design as shown below. It's worth noting that power monitoring functionality is only available on the NB2DSK01.08 and later boards.



New – FPGA Peripheral Cores

Expanding your options for peripherals, Altium Designer 6.9 adds support for three new peripheral cores. For schematic-based design of your main processor system, all peripherals can be found in, and placed from, the FPGA Peripherals integrated library (`\Program Files\Altium Designer 6\Library\Fpga\FPGA Peripherals.IntLib`). For a design incorporating an OpenBus System, all components can be found in, and placed from, the **OpenBus Palette** panel.

Corresponding core reference documents can be found in the Help directory (`\Program Files\Altium Designer 6\Help`).

WB_USB

The WB_USB peripheral provides the interface between a processor in the FPGA design and an external USB Interface device, for subsequent communications over a Universal Serial Bus (USB). The peripheral has been built specifically to interface to the EZ-USB SX2™ device (CY7C68001, from Cypress Semiconductor). This high-speed (USB 2.0) USB Interface device has a built-in USB transceiver and a Serial Interface Engine (SIE), which automatically manages the USB protocol.

If you are using Altium's Desktop NanoBoard NB2DSK01, the USB-IrDA-Ethernet Peripheral Board PB03 features a CY7C68001 device.

WB_BOOTLOADER

The WB_BOOTLOADER peripheral provides the ability to automatically load (or bootstrap) from serial Flash memory on the Desktop NanoBoard NB2DSK01. The device provides the interface between the NB2DSK01'S SPI bus and independent SRAM on the daughter board. Provided the device is enabled for boot operation, then as soon as the design is programmed into the daughter board FPGA (or an external reset is issued if already programmed) the content of the serial Flash memory device will be copied into the SRAM.

The device is configurable in that you can specify where in SRAM the content copied from the serial Flash memory is to be written, and the size of the memory involved in the copy.

The WB_BOOTLOADER can also be used as a standard SPI Controller, with an optional interface to a processor in your design, enabling the processor to communicate with slave SPI-compatible devices external to the FPGA in which the design is running. Operating as an SPI Controller, the device is functionally identical to the SPI_W peripheral.

WB_I2S

Altium Designer's WB_I2S Configurable Wishbone Audio Streaming Controller is used to facilitate data transfers over the inter-IC sound (I²S) bus. The I²S bus – developed by Philips as a dedicated serial link for digital audio – allows a standardized communication medium for an ever-increasing array of digital audio devices.

The WB_I2S is an enhanced version of the legacy I2S_W peripheral. As part of its configurable nature, it allows you to enable, and define, a much larger hardware buffer. With larger FIFO buffers, interrupts can be disabled and simple periodic polling of the state of the Transmit and Receive buffers performed instead. Additional internal registers are used to store the values for the pointers to the Head and Tail of each buffer, enabling you to ascertain how many samples are currently available to be read in the

What's New in Altium Designer 6.9

Receive Buffer, and how much sample space is available for writing new samples in the Transmit Buffer.

Like the I2S_W, the WB_I2S provides for transmission and reception of data. Unlike the I2S_W however, you can configure the WB_I2S to just transmit data or just receive data.

Improved – Xilinx EDK Support

Altium Designer 6.9 now supports the following later versions of the Xilinx EDK (Embedded Development Kit):

- 8.1 and 8.2
- 9.1 and 9.2

New – Context-sensitive C language help

Altium Designer 6.9 brings with it **F1** context-sensitive help support for C keywords and standard functions. Simply highlight part of the keyword or function of interest, in the C coding editor, to access related information directly in the **Knowledge Center** panel.

The information can also be found in the document [TR0173 C Language Reference](#), located in the \Help folder of the installation.

Revision History

Date	Version No.	Revision
06-Mar-2008	1.0	Altium Designer 6.9 release

Software, hardware, documentation and related materials:

Copyright © 2008 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, Altium Designer, Board Insight, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, P-CAD, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.