



Altium Designer 6.0 – Why upgrade now?

Altium Designer 6.0 – Why upgrade now?

Altium Designer 6.0 is a single, unified application that incorporates all the technologies and capabilities necessary for unified electronic product development. Altium Designer 6.0 integrates board- and FPGA-level system design, embedded software development for FPGA-based and discrete processors, and PCB layout, editing and manufacturing within a single design environment. This, combined with modern design data management capabilities, makes Altium Designer 6.0 the only solution for unified electronic product development.

What's new?

The development and release of Altium Designer 6.0 brings a host of new and enhanced features to improve PCB design productivity. This release includes a mix of major new features and technologies, combined with numerous smaller enhancements. Many of the enhancements are based on feedback from our users, the engineers and designers developing electronic products in Altium Designer.

To create this document, we reviewed the features and enhancements within Altium Designer 6.0, and then looked at those features that provide the strongest benefits for upgrading from Altium Designer 2004. Since the last release, considerable advancements have been made, and we've seen through customer feedback that even some of the simple improvements are really making a big difference to the design experience.

Feature highlights

Altium Designer 6.0 has been upgraded to support the high board densities and high-speed signaling found in today's designs. Differential pair, smart connection and BGA escape routing, backed up by pin/part swapping with dynamic net assignment brings a new level of power to the routing process.

With this release it is also significantly easier to integrate Altium Designer with company engineering and business management systems commonly found in medium and large-scale organizations. Place components directly from your company database using the new database-driven part libraries, and include database information directly in the bill of materials (BOM).

Understanding that your development tools need to improve as the technology and the industry changes, Altium Designer 6.0 now incorporates web updating capabilities, making it easy to apply the latest features and enhancements to your installation. Updates are developed in a continuous cycle – as soon as one is released the next one moves into beta, which goes through rigorous testing by your peers in the Altium Designer beta test group. In fact, since the release of Altium Designer 6.0 in December 2005, there have been two updates.

Read on for a brief summary of just a few of the new and enhanced features in Altium Designer 6.0.

Software, hardware, documentation and related materials:

Copyright © 2006 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, 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.

Keeping you and your software up-to-date

Live Web Updating

Software stability is essential for design software. With the release of Altium Designer 6.0, Altium has moved to a continuous stability improvement process, delivered through the new web update feature.

The web update feature allows you to easily keep your Altium Designer software, libraries and documentation up-to-date.

Checking for updates is under your control, when you click to accept an Update it will then download and install automatically. Updates can also be rolled back if required.

As well as checking for available updates on the Altium Website, you can also configure it to check a network location – ideal if your company has multiple installations of Altium Designer.

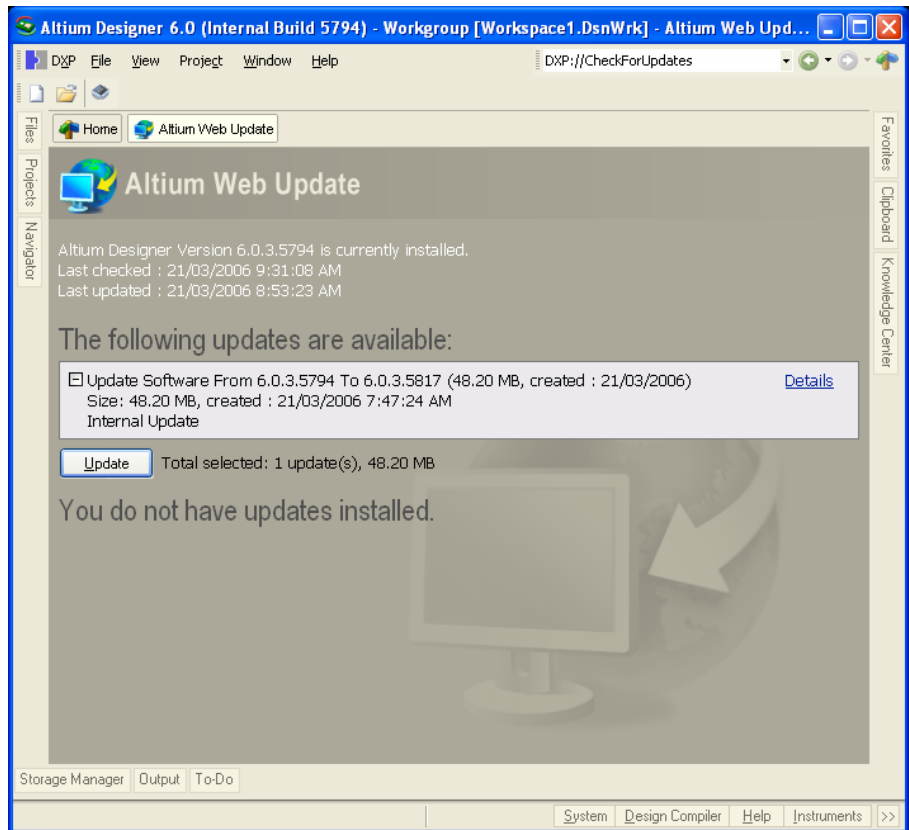


Figure 1. Keep your installation of Altium Designer up-to-date with the new Web Update feature.



Figure 2. Visit the DEMOCenter to learn more about Altium Designer 6.0

The Altium DEMOCenter

One of the challenges of learning about new features is finding the time.

The new online Altium DEMOCenter demonstrates many of the new features in Altium Designer 6.0 in short on-demand product demonstration videos, allowing you to learn a new feature when it is convenient for you.

Each series in the DEMOCenter contains an extensive range of individual demos, each only taking a couple of minutes, making this a quick and easy way for you to browse the areas of most interest and importance to you.

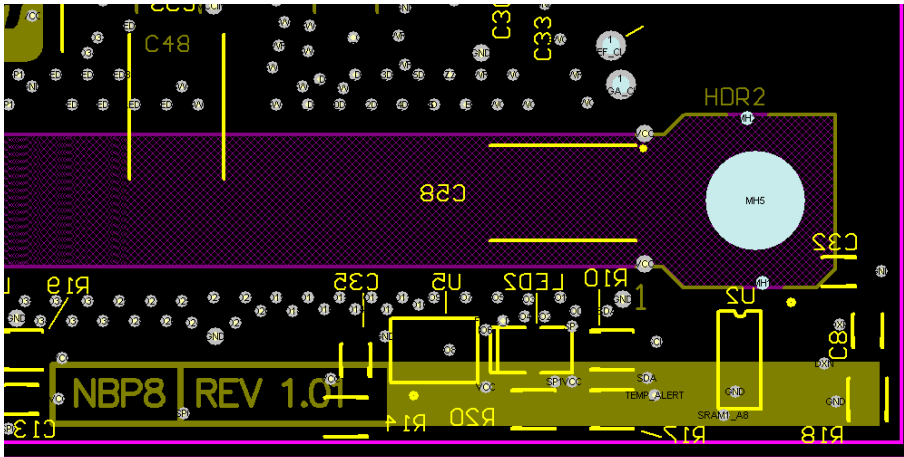
[Visit the DEMOCenter now.](#)

A better PCB design space

Flip and edit the board

Today most boards have components on both sides. Designing on the bottom of the dense board is not a trivial task, planning routing patterns and positioning designators, logos, and free text strings requires the designer to work with mirrored components and strings, visualizing the normal view of the bottom of the board in their mind.

In Altium Designer 6.0 you can work on the bottom of the board as easily as you work on the top. Use the new **View » Flip Board** command to turn over the entire workspace, just as if you were turning over the board in your hand. All standard actions and editing commands are supported, including routing, positioning components, and positioning text. Simply select the command again to flip the board back over.



Bottom Overlay

Bottom Overlay
Bottom Overlay
Bottom Overlay

Figure 3. View the board from below by selecting View » Flip Board.

Improved Interactive Routing

One of the most focused and intense phases of board design is routing. Altium Designer is known for its excellent interactive routing capabilities, intuitively giving you the right track width on that layer, easing the path finding process with the look-ahead feature, and instantly removing any redundant track segments when you re-route a section as you explore possible route-path options. Altium Designer 6.0 includes a number of improvements to interactive routing, which are outlined below.

Pick from your favorite track widths

Altium Designer 6.0 brings a new level of control to interactive routing, while still giving you the confidence and security of the boundaries specified in the design rules. Press **Shift+W** to pick a track width from a pop up list, which you can configure in the *Preferences* dialog.

Protect intentional routing loops

Altium Designer supports re-routing existing routing by a feature called **automatic loop removal**. To use it you simply select **Place » Interactive Routing** and reroute the existing route along a new path – when you terminate routing any redundant segments are automatically removed.

There are times when this behavior can work against you, for example when you are routing a power net with a complex distribution pattern. In Altium Designer 6.0 you can disable Loop Removal selectively for any net, simply double click on a net name in the panel and clear the **Remove Loops** checkbox in the *Edit Nets* dialog.

PCB routing completion detector

The cursor is automatically released from the current route when a connection is completed in Altium Designer 6.0.

Imperial		Metric	
Width	Units	Width	Units
5	mil	0.127	mm
6	mil	0.152	mm
8	mil	0.203	mm
10	mil	0.254	mm
12	mil	0.305	mm
20	mil	0.508	mm
25	mil	0.635	mm
50	mil	1.27	mm
100	mil	2.54	mm
3.937	mil	0.1	mm
7.874	mil	0.2	mm
11.811	mil	0.3	mm
19.685	mil	0.5	mm
29.528	mil	0.75	mm
39.37	mil	1	mm

Apply To All Layers

Figure 4. Select from the list of favorite routing widths by pressing **Shift+W** during routing.

Preserve track angles during dragging

During board design you will often want to move existing routing, to free up a routing channel or create space for an additional component. Altium Designer 6.0 includes a track dragging mode that *slides* the track segment, preserving the angles to adjoining track segments and maintaining the routing quality.

Hold **Ctrl** as you click and drag on a segment to preserve the angles. This works in harmony with the design rules, maintaining specified distances from other objects, even hopping over existing objects where possible.

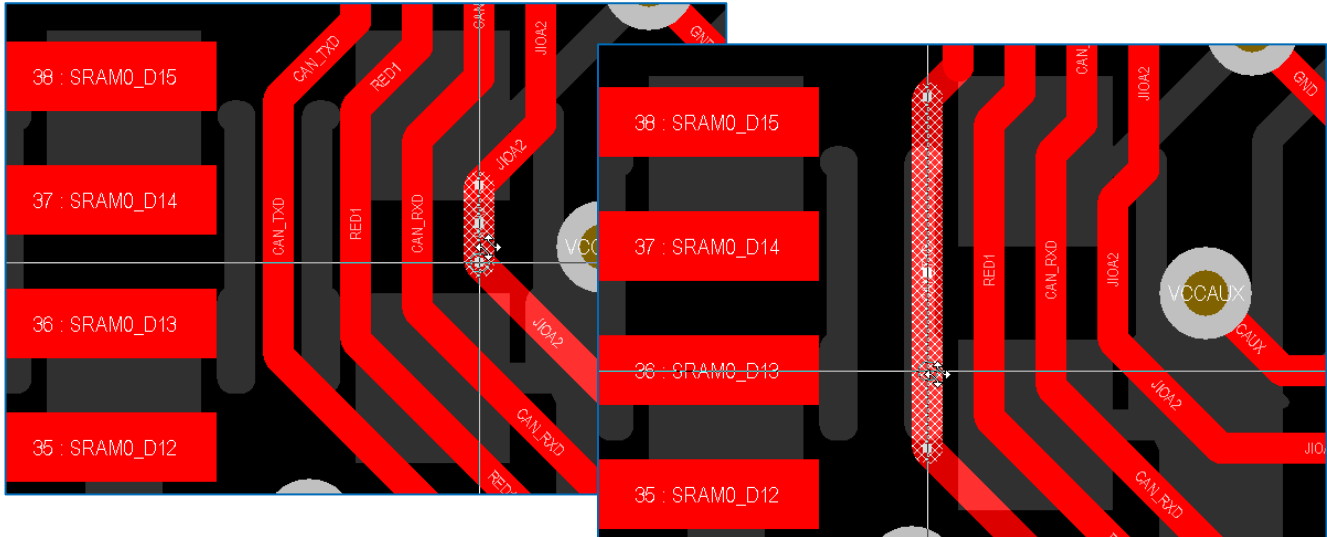


Figure 5. Quickly dragging a number of routes to open up another routing channel, while preserving the corner angles.

BGA Escape Routing

Altium Designer 6.0's BGA fanout capabilities have been enhanced by the addition of BGA escape routing.

The escape routing engine will attempt to route each pad out to just beyond the edge of the device – making the remaining routing challenge much easier.

Figure 6 shows the escape routing from a 1mm pad-pitch BGA. Inner pads are first fanned out using the traditional dog-bone track+via to access another layer, then from the via they are escape routed out just beyond the edge of the device.

Simply right-click on a BGA and select **Component Actions » Fanout Component** from the context menu to escape route the device, in accordance with the applicable design rules. A report of all pads that could not be escape routed will be generated, click on an entry in the report to cross probe to the PCB and examine that pad.

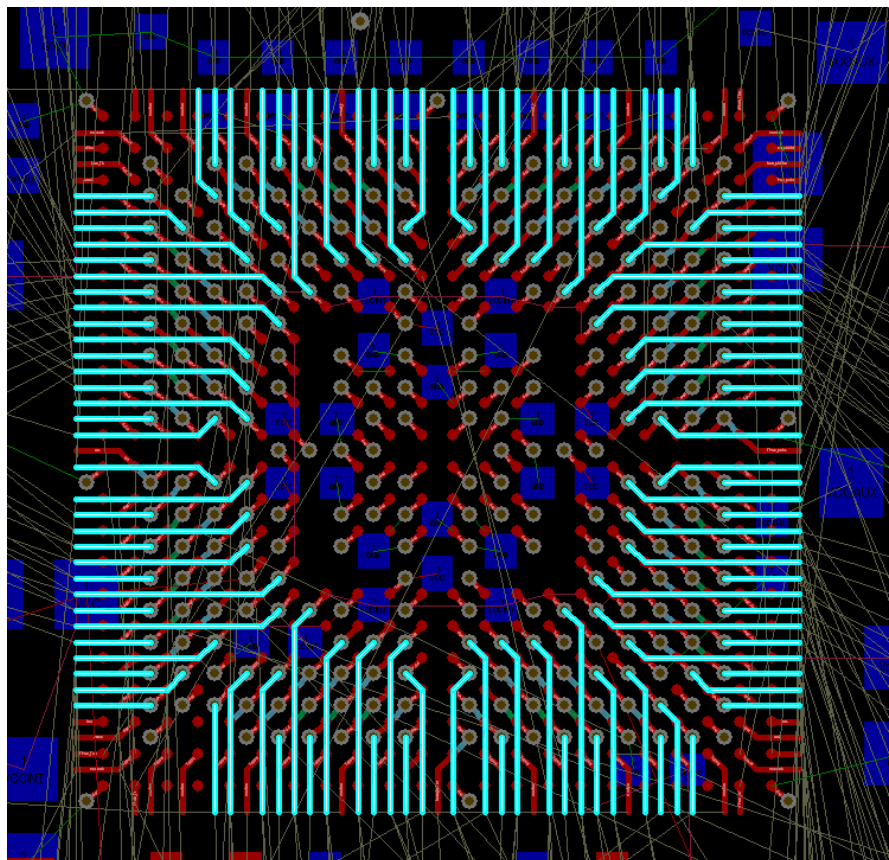


Figure 6. Note how the escape route feature presents each connected pad as an accessible route outside the edge of the BGA.

Designing with Differential Pairs

Differential signaling is fast becoming the preferred signaling interface method, driven by the ever-increasing signal speeds in electronic products. By their very nature FPGAs are ideally suited to high-speed designs, and in support of this FPGA vendors are including differential signaling capabilities (LVDS), from their lower-cost devices right through to their high-end 1500+ pin mega-gate devices.

Altium Designer 6.0 has strong support for differential signaling – from defining pairs on the schematic, through to interactive differential pair routing on the PCB. PCB routing is backed up by full support for pair swapping using Altium Designer's new dynamic net assignment capabilities, an exciting concept that can swap not only unrouted net pairs, but also partially routed net pairs, allowing you to harness the full benefits of the FPGA's reconfigurable design capabilities throughout the routing process.

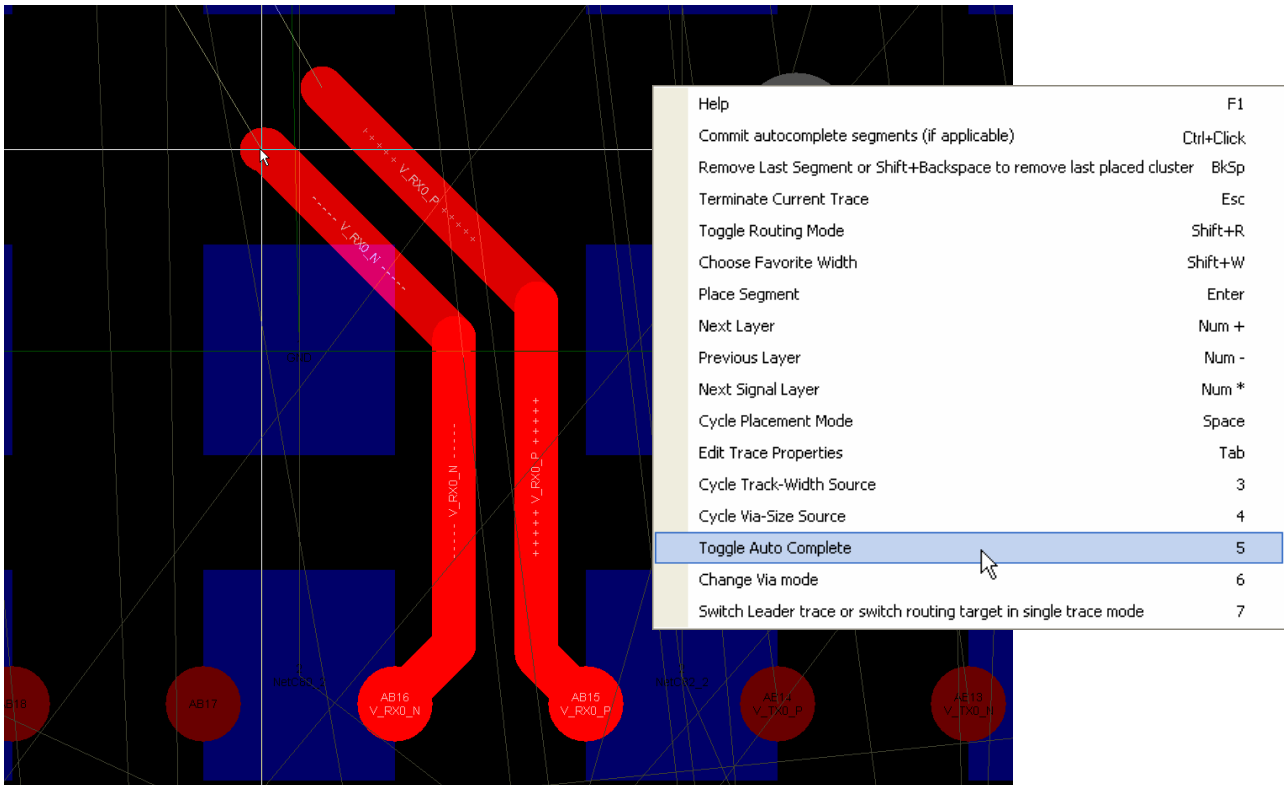


Figure 7. Both nets in the differential pair are routed simultaneously, press ~ (tilde) to see the shortcuts.

Pin and Part Swapping with dynamic net assignment

Working in harmony with the new differential pair routing and BGA escape routing capabilities is the new pin and part swapping capabilities. While providing all the benefits of traditional pin-swapping systems, it also takes advantage of Altium Designer's intimate understanding of the net assignments in the design.

During a pin swap operation Altium Designer analyses the net assigned to the chosen pin, and dynamically reassigns the net on all connected routing. This level of functionality means that partially routed nets and pre-routed multilayer escapes from complex BGA devices can now be swapped.

Differential pairs can also be swapped, taking advantage of the knowledge about differential pin-pairs on an FPGA.

The system also includes a powerful automatic optimizer that can dynamically re-assign nets to improve routability.

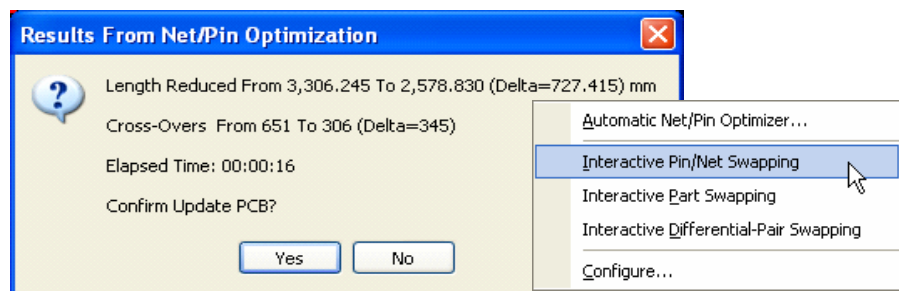


Figure 8. The 2-stage automatic pin/net optimizer minimizes connection length and crossovers.

Better views of your board with the new Board Insight™ system

A complex-multi layer board makes for a visually dense and often difficult to interpret workspace. The new Board Insight system makes it easier to view and understand the objects in your design.

The Board Insight system is an integrated set of features developed to meet your view management needs. For this release Board Insight includes an *Insight Lens*, heads-up cursor information, floating graphical views, simplified net highlighting, and enhanced net labeling on objects.

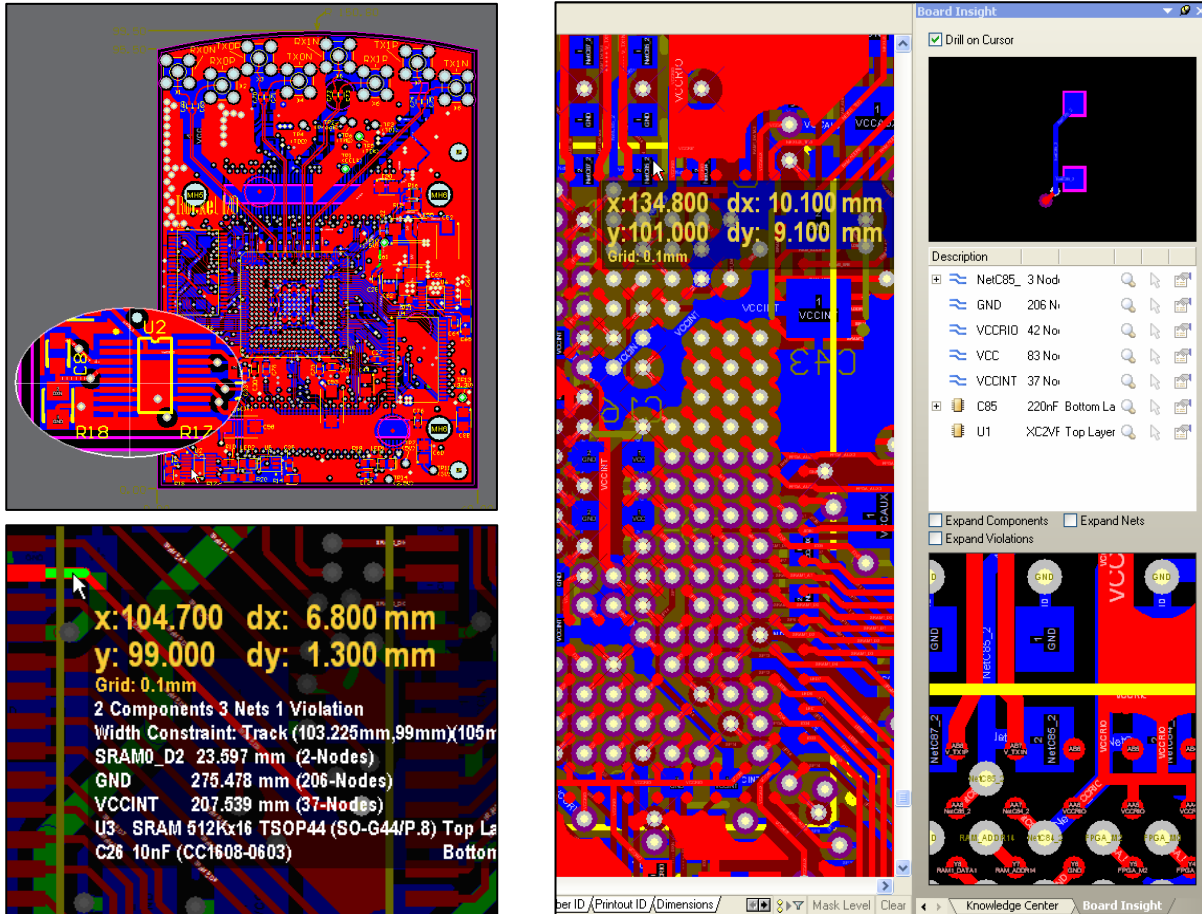


Figure 9. The Board Insight system is an integrated set of features to you more easily interpret the workspace. It includes: an Insight lens, ideal for close, visual examination of what is currently under the cursor; a configurable heads up text display; and the Board Insight panel, which includes a magnified view, as well as detailed object-level information.

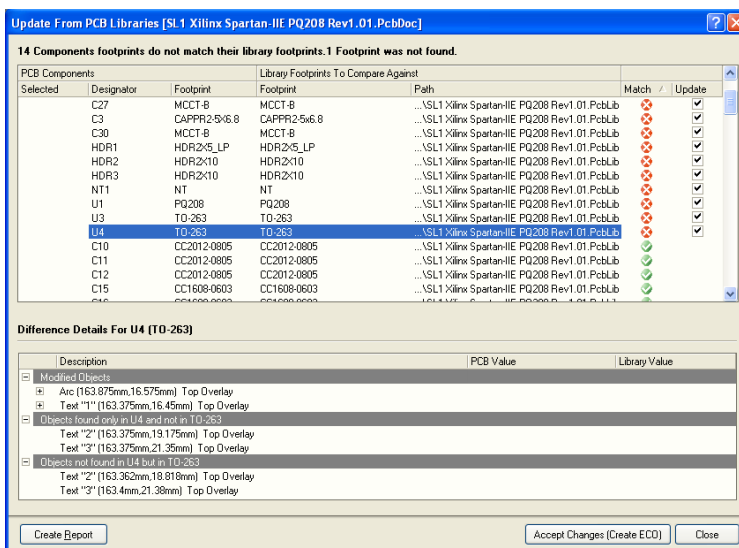


Figure 10. The new Update from PCB Libraries feature does a full comparison of every footprint used on the board and allows controlled updating from the source libraries.

Compare and update PCB footprints from libraries

Altium Designer 6.0 introduces a new **Update from PCB Libraries** feature, developed to give you complete confidence that the footprints on the board exactly match those in the source libraries.

It performs a full analysis and comparison of all objects in both the board and library version of each footprint, and details every difference. For any footprints that do not match you can individually select if that footprint is to be updated or not.



Figure 11. Use the new TrueType font support to display text in your preferred font face. Invert the string if you need it etched in a copper layer.

Use TrueType fonts for PCB text

In Altium Designer 6.0 the PCB editor has full support for TrueType® fonts. This gives you access to all the TrueType fonts available on your PC, including Unicode-character sets, such as Japanese. Place your company or product name in your preferred font and give your board the high-quality finish you require.

All PCB text strings can be set to one of the PCB editor's 3 built-in fonts, or to a TrueType font available on the PC. As well as Bold and Italic, TrueType strings can also be inverted, ideal when you need a string etched into a solid copper layer.

Fonts can be embedded in the PCB file by enabling the **Embed TrueType fonts** option in the *Preferences* dialog. If a TrueType font is not embedded and the font is not available when the file is reopened on a different PC, the specified alternate system font will be used in its place.

TrueType characters are rendered as region objects when Gerber or ODB++ output is generated, giving full support through to board fabrication.

Move an object by an offset

A common design task is to move a set of objects by a precise amount. Rather than relying on a steady mouse hand and a close eye on the status bar coordinates, Altium Designer 6.0 supports moving an object or a selection by entering an offset amount. Use the new **Move Selection By X,Y** command to move the selection set by the specified X and Y amount. Include the units if you need to move by non current grid units. You can also offset-edit an object's location in the **Inspector** panel, using the new **!+** offset syntax. For example, **!+100** will increase the edited coordinate by 100 current units.

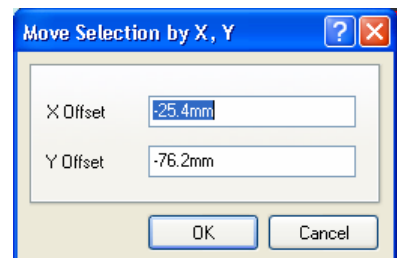


Figure 12. Define the X and Y offset amounts in the Move Selection dialog.

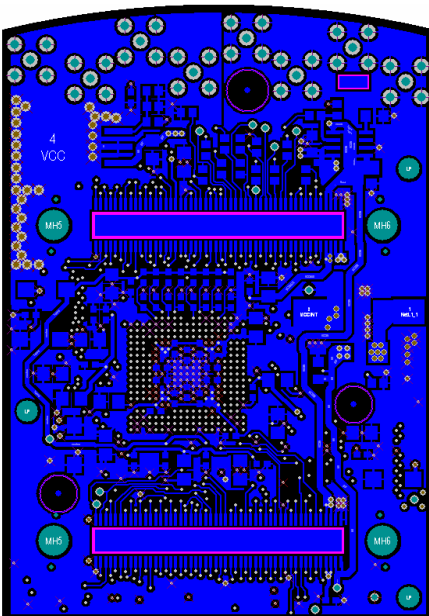


Figure 13. Polygons are an essential part of most board designs.

Improved Polygon Pour performance

Polygons are an essential part of most boards designed today. Solid polygons were introduced in Altium Designer 2004, reducing the object count as well as the design and output file sizes.

The pouring speed for solid polygons has been substantially improved in Altium Designer 6.0, making working with solid polygons faster and easier.

Separate Drill files for plated and non-plated holes

In Altium Designer 6.0 separate ASCII format drill files are automatically generated for both plated holes and non-plated holes, simplifying the preparation of the fileset for board fabrication.

Pick and Place Output Generation

Pick and place output is now available for multi-PCB boards created with the PCB editor's embedded board array feature.

Configurable mouse wheel

The mouse wheel, in combination with keyboard keys, is a standard zoom and pan control feature in most Windows applications. In Altium Designer 6.0 you can now configure your preferred wheel+key combinations for zooming and panning. It also supports changing PCB layers via the mouse wheel, hold **Ctrl+Shift** while rolling the mouse wheel to move through the currently enabled PCB layers.

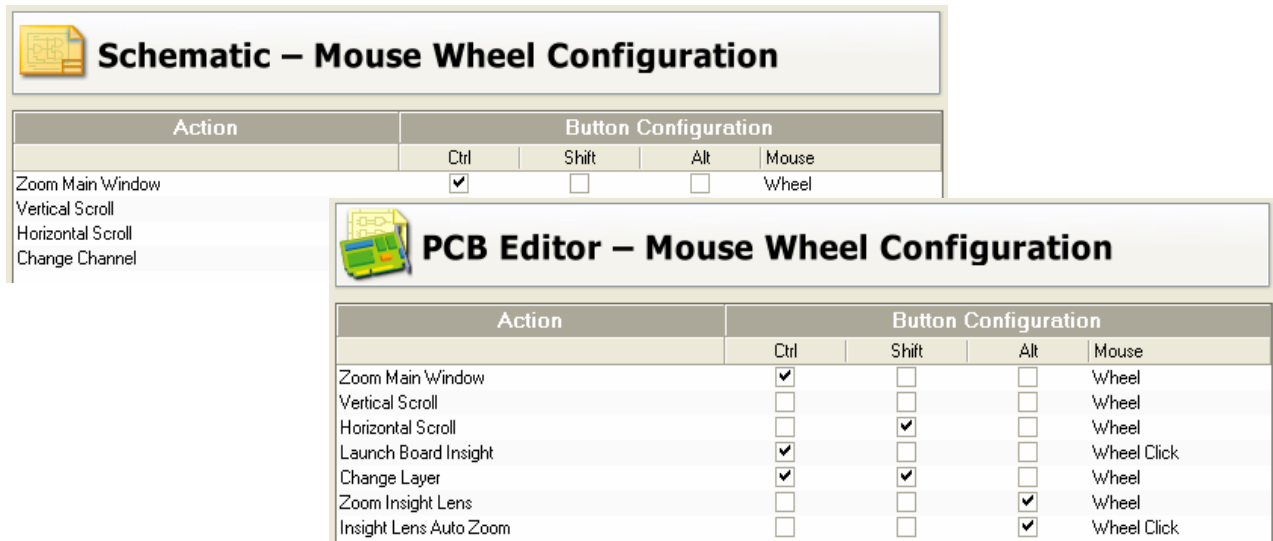


Figure 14. Reconfigure the standard mouse wheel / key combinations.

What was that shortcut?

Remembering shortcuts, particularly the command-specific ones, takes time and frequent usage.

Pressing the tilda key (~) during any interactive board design command will pop up a list of shortcuts that apply to that command. Use this as a visual reference to read off the shortcut, or use it as a menu and click to select.

Alternatively, display the new **Shortcuts** panel, this presents a dynamic list of current shortcuts, updating automatically as you select different commands.

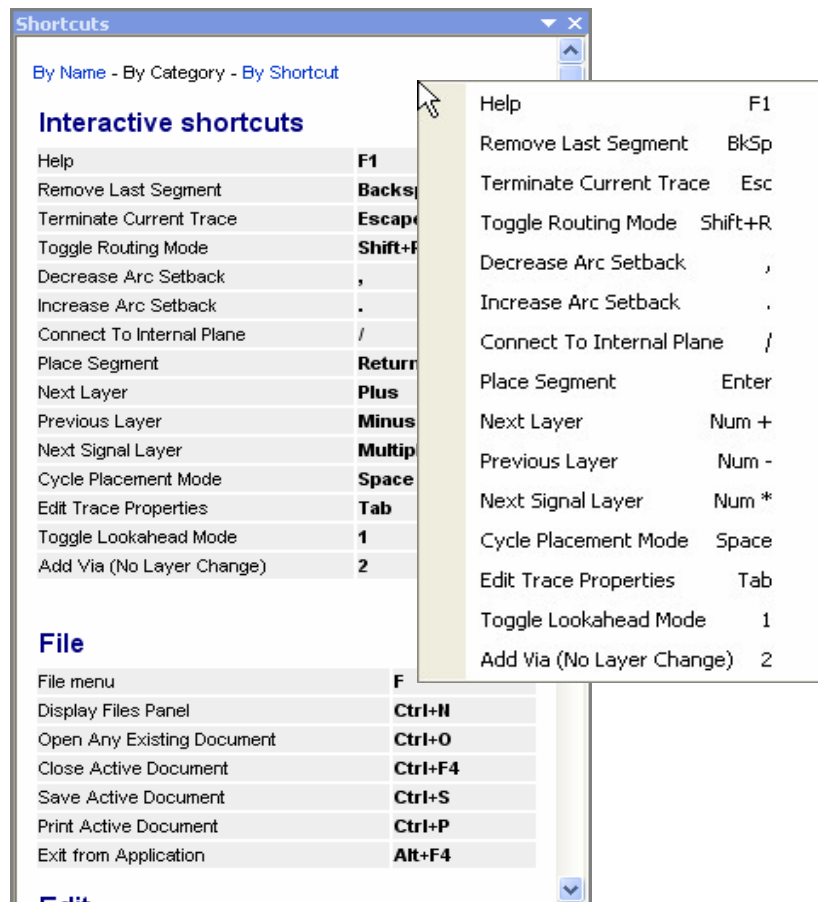


Figure 15. The keyboard shortcut panel and popup shortcut menu for Interactive Routing.

Stronger mechanical CAD interface

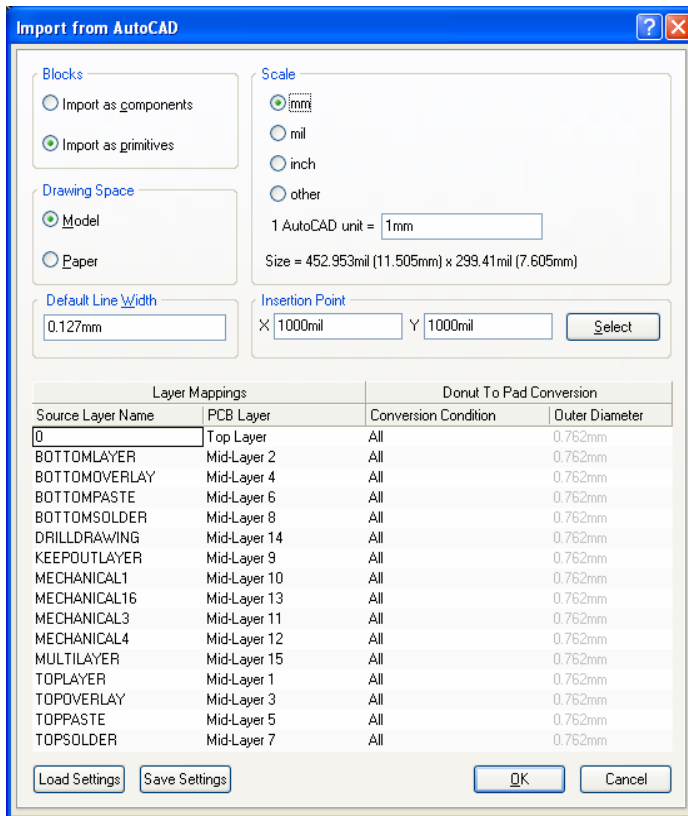


Figure 16. the improved DXF/DWG (AutoCAD®) import options.

Enhanced DXF/DWG Interface

DXF and DWG are popular file formats for transferring data between design tools. DXF/DWG export has been enhanced in Altium Designer 6.0 by the addition of new controls that allow exporting selected objects with zero line widths, ideal when the data is to become part of a dimensioned mechanical drawing.

DXF/DWG import has also been enhanced, you can now apply scaling, define the default line width, and specify the insertion location.

Improved MCAD Data Exchange

Exporting your board design to either IGES or IDF has been greatly improved. The exported files are now much smaller, making them easier and faster to import into your MCAD environment.

IGES export includes component body objects found in the footprints. You can use these as a simple method of defining the component shape, as shown by the grey shapes in the 3D view on the left below. For an exact definition VRML or IGES component models can also be imported into Altium Designer.

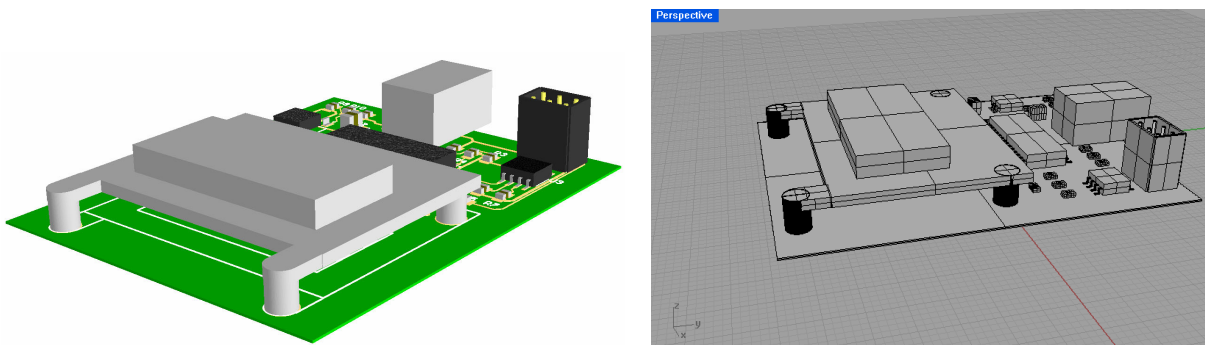


Figure 17. View the board in Altium Designer's 3D viewer and then export to IGES, ready for import to your MCAD application.

Full Database-driven Part Libraries

A common requirement today is that the components in your board design are connected back to your company database, allowing company information to be used during board design, and then propagate through to the Bill of Materials and other fabrication and assembly files.

This became possible in Altium Designer 2004 with the introduction of database linking, allowing you to reference a database record from your Altium schematic symbol.

Altium Designer 6.0 takes this capability much further, with the introduction of database-driven parts libraries. Component definitions can now be constructed in an ODBC or ADO based database, or an Excel spreadsheet. Each record in the database represents a component, storing all of the corporate component data, as well as design parameters and links to the required Altium Designer models, such as the symbol and footprint.

Components are placed directly from the database, via the Altium Designer **Libraries** panel. The new Database Library, or DBLib, is added to and presents like any other library in the **Libraries** panel – you can browse the list of components, examine the component symbol and its models, and place the component.

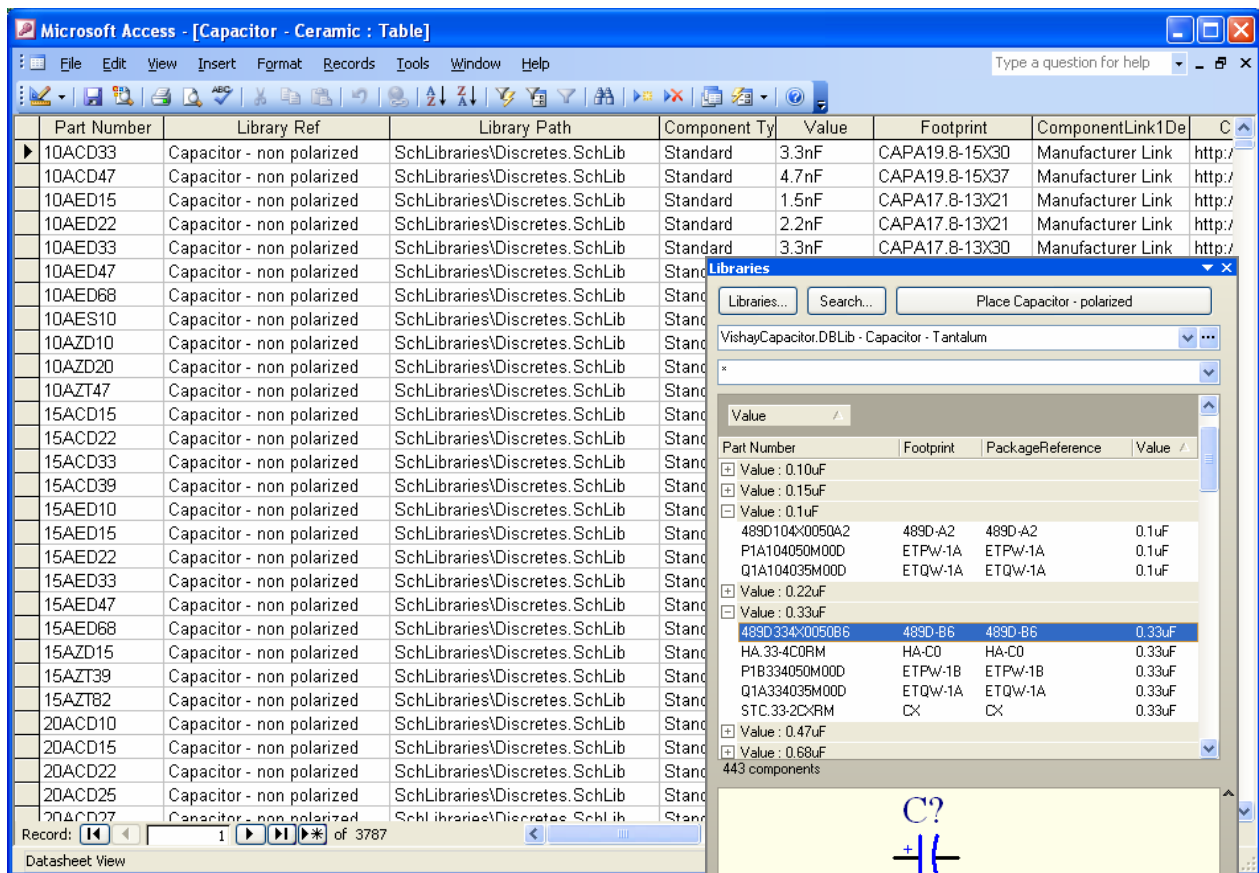


Figure 18. When you place from a DBLib all the component information comes directly from your database.

Behind the DBLib that you are browsing in the **Libraries** panel is your database, each component in the panel corresponding to a record in the database. As well as company-type data, such as cost or stock number, the database will also specify the Altium Designer schematic symbol, the footprint, and other models that are part of the component.

When you click the **Place** button in the **Libraries** panel the symbol is extracted from the specified schematic library, and then component properties, such as the footprint and parameters, are added as you place it on the sheet.

The DBLib is a document that you edit in Altium Designer 6.0. You can think of the DBLib as a set of mapping instructions, where you specify what data in each record is to be mapped to the Altium Designer component parameters and models. The link to the database can be established in a variety of ways, from simply browsing and selecting the Access database or Excel spreadsheet, right through to using a Data Link File (*.UDL).

And when it comes time to generate output such as the Bill of Materials, not only can you include any of the parameter information held in the schematic component, you can also extract any other BOM-pertinent information directly from that record in the database, and include it in your BOM.

Enhanced Design Capture

Design snippets

It is common practice, and good design sense to reuse proven sections of circuitry. The new Snippets feature in Altium Designer 6.0 allows you to save any selection of circuitry on a single schematic sheet, or any selection of a PCB design, including the components and the routing, as a Snippet.

Schematic and PCB selections are saved as snippets in the new **Snippets** panel, right-click in the panel to add the current selection as a snippet.

Snippets can be organized into standard Windows® folders on your network, click the **Snippets Folders** button to add an existing folder to the list of available folders in the Snippets panel. Each snippet is stored in a standard schematic or PCB file.

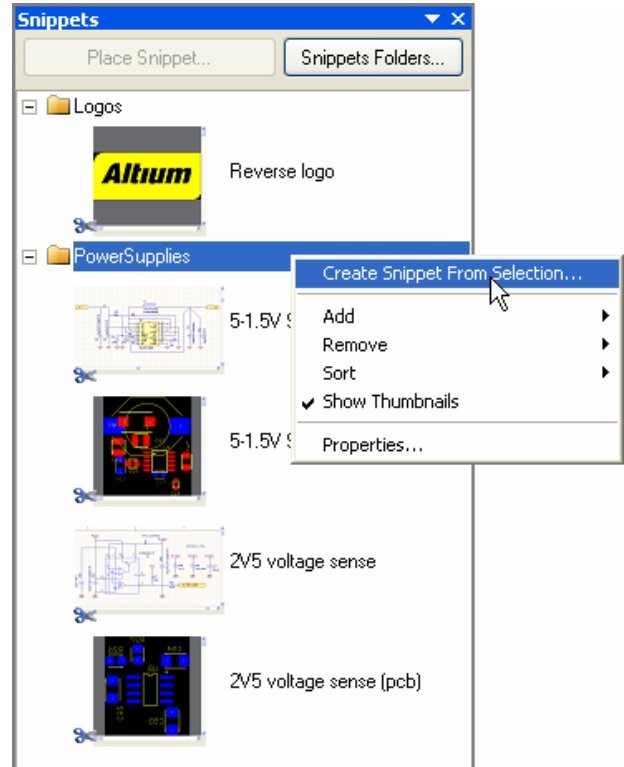


Figure 19. Right-click in the Snippets panel to add a snippet.

In place text editing

Text presented in a Schematic Note or Text Frame can be edited directly on the schematic in Altium Designer 6.0, speeding the editing process, and allowing you to see the layout of the text as you type.

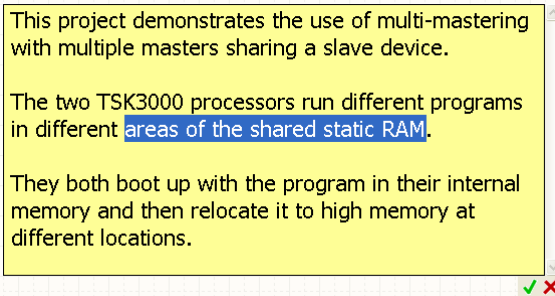


Figure 20. Editing text directly in a Note or Text Frame gives you immediate feedback about the text layout.

Schematic Smart paste

During schematic capture there are a large number of objects placed and connected to build up the design. A common technique to accelerate this process is to copy similar objects that have already been placed, paste them to where you are currently working, and modify them as required.

The schematic editor's new Smart Paste feature takes this approach to an entirely new level – using Smart Paste you can actually *transform* the copy of the selected objects into other objects as you paste them. For example the selected Net Labels can become Ports when pasted, or the selected Sheet Entries can become Ports+Wires+Net Labels, all in a single paste action.

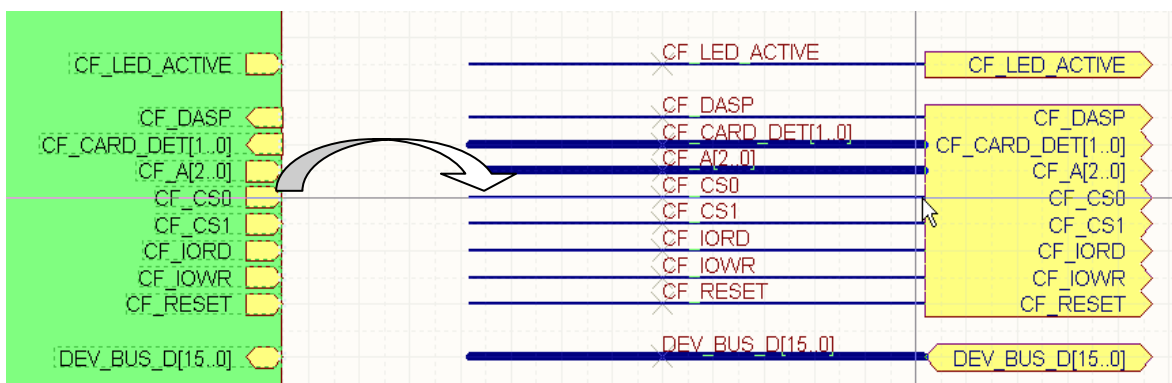


Figure 21. Sheet Entries being transformed into Wires+Ports+Net Labels as they are Smart Pasted.

Footprint manager – manage footprints across the entire design

Keeping track of the footprints across a large design is not a trivial task. The schematic editor in Altium Designer 6.0 includes a powerful **Footprint Manager**, developed specifically for this. The Footprint Manager makes it easy to review and detect problems with footprint assignments across the entire design, particularly useful when you are working on a legacy design, or one from another organization.

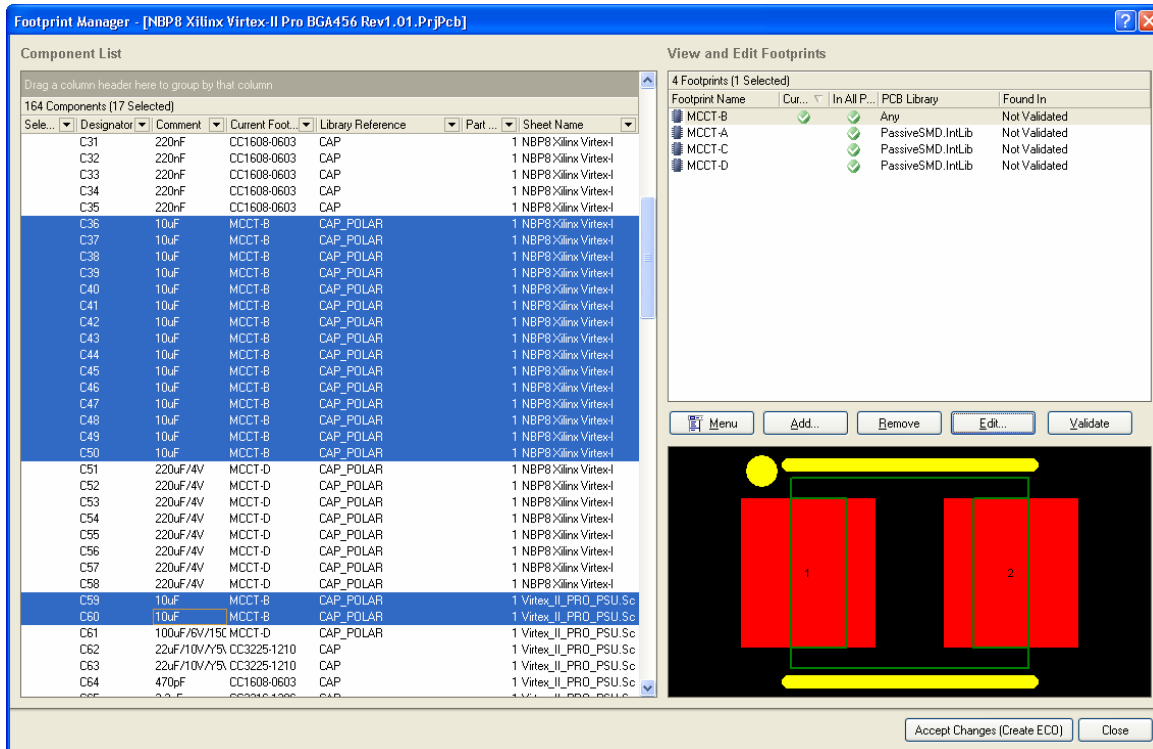


Figure 22. Manage footprint assignments across the entire project with the new Footprint Manager.

Multi-select support makes it easy to edit the footprint assignment for multiple components, change how the footprint is linked, or change the current default footprint assignment for components that have multiple footprints assigned. Design changes are applied through Altium Designer's standard ECO system, updating both the schematic and the PCB if required.

Direct PSpice® model support

The PSpice simulation model format is the format of choice for many device manufacturers. Altium Designer's circuit simulator now has strong support for PSpice models. PSpice models are used in exactly the same way as Spice 3f5 models, simply hook them up in the graphical symbol-to-model editor, and start simulating. There is also excellent support for PSpice functions. Global parameter and equation support has also been added to the circuit simulator.

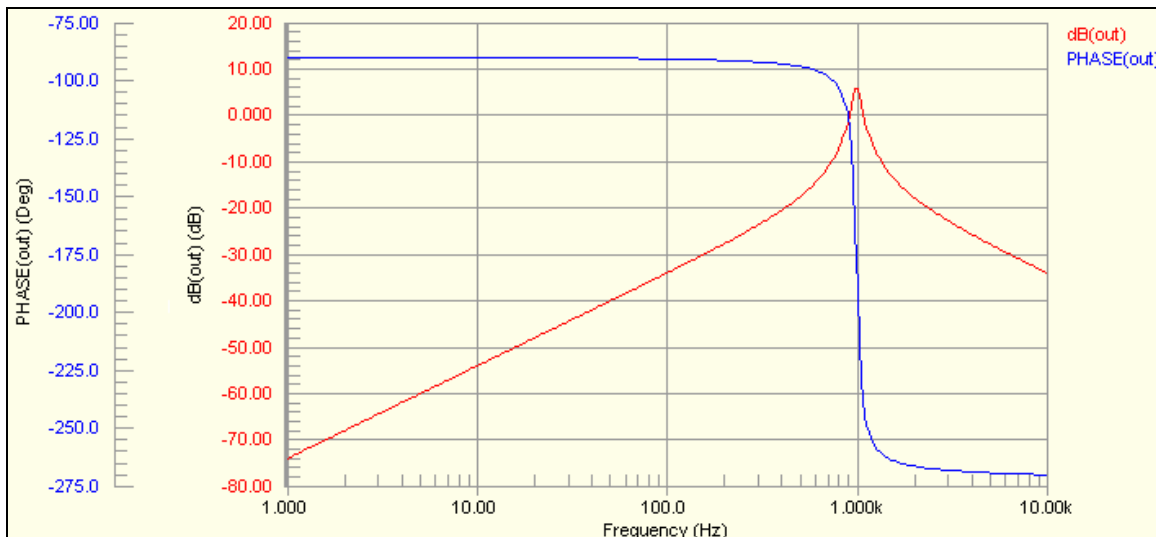


Figure 23. Use PSpice simulation models directly in Altium Designer 6.0

Enhanced Excel® BOM interface

The Microsoft® Excel format is commonly used for Bill of Materials generation. The BOM interface has been updated and a number of new Excel templates have been added, making it easier to use and deliver the output in the required format.

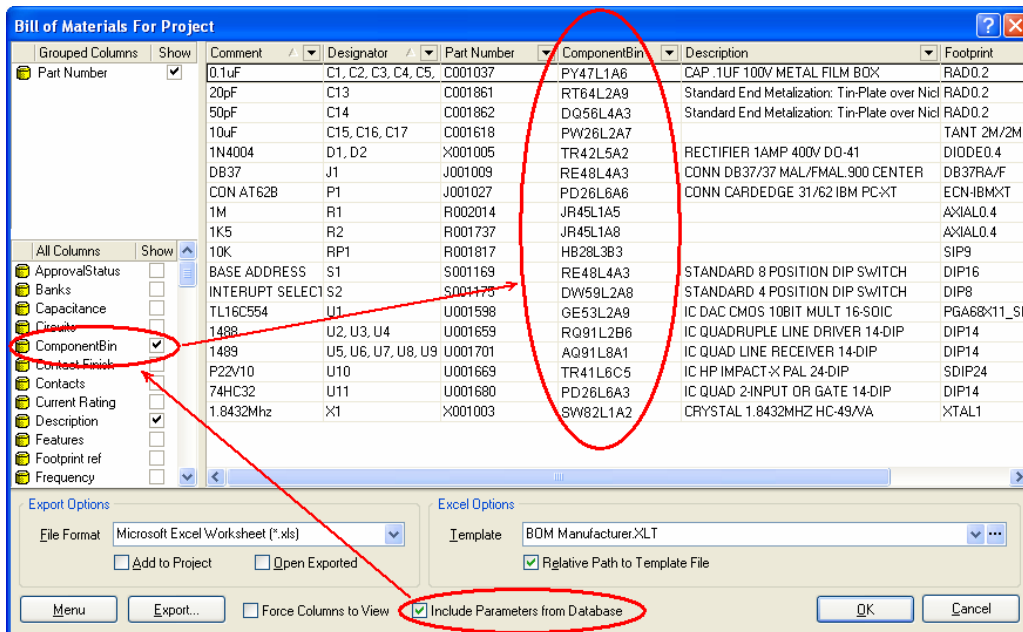


Figure 24. Include information in the BOM directly from a database, when the components are placed from a Database Library.

Include Document and Project Parameters in the BOM

Parameters are a universal feature of Altium Designer, you can add them to the project, a document, a component, almost any object in fact. Project and document parameters can now be extracted from the design and included in the Bill Of Materials. Document parameters are included with each component that comes from that document, and project parameters can be mapped to pre-defined Fields in your Excel template, as shown below.

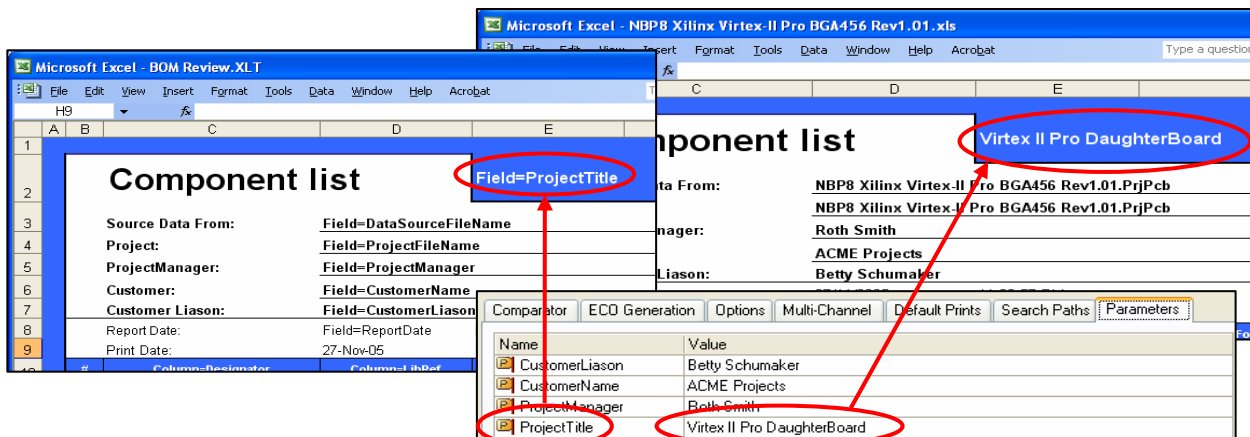


Figure 25. Include project parameters in your BOM by defining Fields in the Excel template.

Interested in learning more?

This document gives only a small sample of some of the valuable new features in Altium Designer 6.0. If you are interested in learning more about Altium Designer 6.0 visit the [DEMOcenter](#) and watch the on-demand product demonstrations, or [browse and download Learning Guides](#) from the Altium website.

For a focused overview of the new and enhanced features in Altium Designer 6.0, visit [Fasten your seatbelt and check out what's new in Altium Designer 6.](#)