

# Key Feature Summary: Altium Designer 13.2

## Altium Designer 13.2

Feature	Detail	Benefits
<b>ActiveBOM</b>	ActiveBOM is a tool to effectively use supply chain solutions (costs, availability) in a project's design constraints.	Providing engineers with a clear picture of BOM cost during design empowers them to: <ul style="list-style-type: none"><li>■ implement a design-to-cost development approach,</li><li>■ pinpoint parts of the design that excessively contribute to overall cost,</li><li>■ model cheaper and more supply-robust component choices,</li><li>■ investigate design changes with an understanding of cost implications.</li></ul>
<b>Live Drill Table</b>	A highly configurable and real-time, WYSIWYG PCB drill table.	By clearly showing design intent and manufacturing requirements, production errors through misunderstanding are avoided.  When designers can see a live view of drill size they will naturally see opportunities for drill optimization before committing to release to manufacture.
<b>PCB Design View</b>	A Design view is a PCB documentation tool that allows a board layout or a section of a board layout to be displayed at a specific scale using specific set of layers.	PCB design can be scaled to fit company or industry standard page layouts.  Design intent and manufacturing requirements can be clearly and professionally documented.  Details of a specific section of a design can be highlighted to ensure correct manufacturing and assembly of those features.
<b>Interactive Net Length Calculation</b>	Current routed length of a net is displayed in the PCB HUD and Nets PCB Panel during interactive routing	This feature significantly improves productivity when routing or modifying tracks that require length matching or control for high speed signals.
<b>Expression Based Output Filenames</b>	Filenames for important design outputs can be named using simple rules, special strings, and project parameters.	Consistent naming standards for design output file avoids human error and assists in locating files and understanding what they contain at a glance.
<b>Via Stitching Enhancements</b>	Via Stitching tools have been greatly enhanced: <ul style="list-style-type: none"><li>■ Stitching Vias are now created as part of a union</li><li>■ Stitching Vias are now identified in the PCB editor using a string suffix</li><li>■ The stitching pattern can now be offset</li></ul>	Users have more control over geometrically separated and overlapping groups of stitching vias, and are able to globally edit each set of vias quickly and easily. In addition, it is easier for PCB designers to see which vias are stitching vias, and which are regular vias, when routing a board.
<b>Separator Character for Designator Suffix</b>	Multi-part symbols can now include a . (dot) or : (colon) to separate the designator and the part suffix.	

# Key Feature Summary: Altium Designer 13.2

## Altium Designer 13.2 Vault Server Support Features

Feature	Detail	Benefits
<b>Component Library (CmpLib) Editor Enhancements</b>	The Component Library Editor now allows more control of the display of required parameter columns, release status, required models; as well as the ability to copy and paste into the cell array.	In Component Libraries with large parameter arrays it is now much easier to manage editing and creation of parametric data.  External spreadsheets such as Excel can be used in conjunction with the Component Library Editor for dealing with complex parameter arrays, using standard copy and paste to extract and replace data within the Component Library Editor's table view.
<b>Inplace reporting of out of date Design Items</b>	Schematic and PCB component models, Managed Components, and Managed Sheets with a superseded revision are now flagged as out of date.  The Item Manager, Component Library Editor, and individual managed item properties (Component, Model and Sheet) properties indicate when the currently placed item in the design is out of date.	Users of Managed Items placed in the design from a vault can now be more clearly notified that a newer revision of such an item is now available for use.  As the design progresses the team using components and other managed sheets will more easily be able to keep their designs completely up to date with the centrally managed content.
<b>Enhanced Update from Vault flow (Item Manager)</b>	Update of out of date items in the Item Manager is now easily accessed from the right-click menu.	Users spend less time when performing batch updates to designs when multiple vault managed source elements have new revisions available for use.
<b>Batch Download of Vault Items</b>	Multiple Schematic, PCB and Simulation Models, and unified components, can be group selected and downloaded from the vault into a single destination folder.	Obtaining and working with Vault-managed unified component sources no longer requires access to the original source files, as the released revision of those sources can be obtained directly from the vault server.
<b>Drag and Drop Placement from Vault</b>	Components and re-use elements can be placed by drag and drop directly from the Vaults panel.	Placing components and managed sheets is more intuitive with common mouse gestures.
<b>Other Enhancements</b>	A number of other Vault support feature enhancements including: <ul style="list-style-type: none"><li>■ Ability to Rename Lifecycle definitions and Item Revision Naming Schemes</li><li>■ Ability to Set ID for Initial Revision of Vault Item</li></ul>	