

Altium NEXUS

Welcome to Altium NEXUS, which is a complete, end-to-end design environment for electronic printed circuit board design. Altium NEXUS enables you to bring your ideas to life with the most efficient and collaborative PCB design environment available. From a tiny, foldable rigid-flex board that goes into a hearing aid, to a large, 20+ layer high-speed network router, Altium NEXUS works with you to deliver a successful design.

New to Altium NEXUS?



The way you work in Altium NEXUS is much like other Windows applications in that commands are accessed through familiar menus, graphical views can be zoomed and panned using standard Windows keyboard and mouse actions, and many of the commands and features can be accessed through keyboard shortcuts.

Where Altium NEXUS differs from other Windows applications is the way that it brings all of the editing tools that you need into the one environment. That means you edit the schematic and lay out the printed circuit board in the same software application. You also create the components, configure the various output files, and can even open the ASCII outputs in that same environment.

The 64-bit application that you launch is referred to as the X2 platform. Each different document type opens inside the X2 application with the appropriate editor-specific menus, toolbars and panels appearing automatically as you move from one document-kind to another.

Why are all the tools inside the one environment, you ask? Doing this allows you to shift your focus from being tool-oriented to being design-oriented. Working in a design-oriented environment delivers you, the designer, significant advantages including:

- Easily access any document in the project from the *Projects* panel. All of the project documents are displayed and the schematics are also organized to reflect the design structure.
- Provides the ability to easily move back and forth between the schematic and PCB. Tasks such as moving design changes from the schematic to the board, or the board back to the schematic, are quick and non-intrusive.
- When you select a set of components on the schematic and they are also selected on the board, they are ready to be added to a PCB component class or to be repositioned and aligned or flipped to the other side of the board.
- You can flip back and forth from a 2D view of the board to a highly realistic 3D view, detect a mistake, switch to the schematic and make an edit, update the PCB, and you are back on track.
- You can add a new component on the schematic and see it appear immediately in the BOM document, ready to have its supply chain details finalized.
- Provides the ability for the software to work with a single, unified model of the entire design in memory - delivering the advantages detailed above, along with a host of others.

These are just a few examples of the many advantages a design-oriented environment delivers. Regardless of whether you work as a solo designer, or as a member of a large, geographically dispersed team, Altium NEXUS delivers an easy-to-use, immersive design space on which you will enjoy crafting your next great idea.