The 3D Visualization panel.

Summary

The 3D Visualization panel displays up to three simultaneous 3D views including cross-sections and a floating Board Insight. Using this panel, you can examine all aspects of your design inside and out in 3D while still working on it in either 2D or 3D modes in the design editor window. The 3D Visualization panel displays are shown based on the position of the cursor in the main window. You can freeze or unfreeze the position of the 3D panel displays whilst moving around in the design editor window by pressing Shift+F3.
Panel Access

- To display the PCB Inspector panel, click the PCB button at the bottom-right of Altium Designer when the PCB Editor is active and select the 3D Visualization entry from the pop-up menu.
- Alternatively, you can access the panel through the View » Workspace Panels » PCB » PCB sub-menu.
- The panel can also be accessed using the F12 shortcut key.

Panels can be configured to be floating in the editor space or docked to sides of the screen. If the 3D Visualization panel is currently in a group of panels, use the PCB Filter tab located at the bottom of the panels to bring it to the front.

Function

The 3D Visualization panel displays up to three simultaneous 3D views including cross-sections and a floating Board Insight. Using this panel you can examine all aspects of your design inside and out in 3D whilst still working on it in either 2D or 3D modes in the design editor window. The 3D Visualization panel displays are shown based on the position of the cursor in the main window. You can freeze or unfreeze the position of the 3D panel displays whilst moving around in the design editor window by pressing Shift+IF3.
Content and Use

Use the controls to show/hide the various sections of the panel.

The **Configuration** region of the panel has a dropdown list that shows the standard Altium Designer 3D color configurations and any saved custom 3D configurations (set in View Configurations dialog [shortcut L]). The **Section Opacity** slide control varies the appearance of the cutting edge of sectional views - the greater the opacity, the less 'light' passes through the surface, reducing the level of visible detail and makes seeing the cutting edge easier. The **Thickness Scaling** slide control alters the vertical scale of the board to make it easier to differentiate layers when viewing the PCB internally. You can select between 1 and 100 times the vertical scaling. The **Show Section Lines In Main View** option turns on or off the cross-section cursor crosshairs in the main view. These crosshairs represent the scaled lengths and the axes of the **Section A-A** and **Section B-B** views.

The **Section A-A** region displays the Section A-A (X-axis) view. The cross-section shown is based on the position of the cursor in the design editor window. You can zoom the view by left-dragging the mouse - zoom changes take place in both sectional views simultaneously. Right-drag the mouse to rotate the view about the sectional cutting edge.

The **Section B-B** region displays the Section B-B (Y-axis) view. The cross-section shown is based on the position of the cursor in the design editor window. You can zoom the view by left-dragging the mouse - zoom changes take place in both sectional views simultaneously. Right-drag the mouse to rotate the view about the sectional cutting edge.

The **3D Board Insight** region displays the fully floating 3D Board Insight perspective. You can zoom the view by left-dragging the mouse. Right-drag the mouse to rotate the view in any direction. Rotation is about the cursor position in the design editor window.

**Tips**

- You need to have DirectX 9.0c or later installed and enabled and a graphics card that supports Shader Model 3 or later in order to use the 3D Visualization panel. To enable DirectX in Altium Designer, open the **Preferences** dialog from **DXP » Preferences**, navigate to the **PCB Editor - Display** page and enable the **Use DirectX if possible** option under **Display Options** in the right panel.
- You may notice that, when traveling inside the PCB in the 3D Board Insight display, you can still be inside the board while your cursor is off the board in the design editor window. This can occur because of the differing perspective, 3D positioning, and zoom levels between the 3D Board Insight and the design editor window views.
- The cursor position is not shown internally to the PCB in the 3D Board Insight display, however, it is always the position from where the internal PCB perspective is based around.

**Source URL:** https://www.altium.com/documentation/display/ADES/PCB_Pnl-3DVisualization((3D+Visualization))_AD