Altium Designer

Essentials Training with Altium 365

Module 16: PCB Viewing and Navigation

Software, documentation and related materials:

Table of Contents

Module 16: PCB Viewing and Navigation

1 Purpose

In this exercise, users will gain a better understanding of the PCB Editor interface and learn about basic navigation and viewing tools.

This exercise is designed to familiarize you with the PCB design environment. We will explore the various panels used for navigation within the PCB, examine different viewing modes, and introduce essential tools for effective utilization of the PCB design environment.

2 Shortcuts

Shortcuts used when working with Module 16: PCB Viewing and Navigation

1	Board planning mode
2	2D mode
3	3D mode
8	3D Orthogonal Rotation view.
9	3D 90 Degree Rotation
0	3D Zero Rotation
Shift+right click	3D Navigation
or T » P	Preferences
ALT key and left-click	Highlight Net
Shift+C	Clear Masking and/or Selection
V » D	View » Fit Document
V » F	View » Fit Board
J » C	Jump to Component
J » L	Jump to Location
Hovering	Show Object information (Head Up Display)
Mouse Wheel	Scroll up - Scroll down
Mouse Wheel+Ctrl	Scroll left - Scroll right
Mouse Wheel+Shift	Zoom in - Zoom out
Mouse Wheel+Shift+Ctrl	Layer change
“ + ” , “ - ” , or “ * ”	Layer Change (on the numerical keypad)
Press Mouse Wheel+ Mouse Move	Zoom in - Zoom out
Page Up	Zoom in
Page Down	Zoom out

T » C	Activate Cross probe
Cross Probe + Ctrl	Jump to Document
Shift+Ctrl+X	Cross select mode
Ctrl+Z	Undo
Ctrl+S	Save Document

3 Preparation

For this module, we can use the example project from the last module, `Module 15 PCB Introduction`.

3.1 Using an existing Project

1. If you have closed the Project `Module 15 PCB Introduction – [Your name]` from the last module, please reopen it.
   This project may also be available in the **File** menu, under **Recent Projects**.

4 Exploring the Jump Command

The Jump command is a commonly used command, which can be found from the **Edit** menu, or by using the keyboard **J key.**

2. Navigate to the *Projects* panel and open the PCB document `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PcbDoc`.
3. Hit the **2** key to ensure you're in 2D mode.
4. Hit the **J** key, to see all the different sub-menus available, see Figure 1.

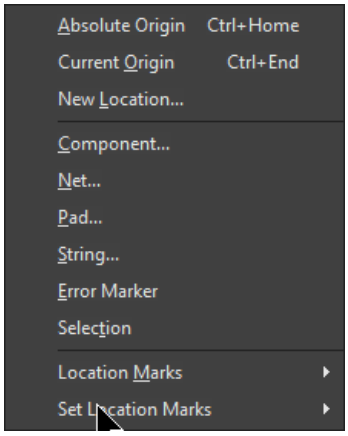


Figure 1. Jump Menu Options

5. Hit the **Esc** key to close the Jump menu.
6. To jump to the current origin: Press the **J » O** keys in succession.
7. To jump to a specific location: Press the **J » L** keys in succession, then enter the X, Y coordinates within the *Jump to Location* dialog as shown in Figure 2 below.

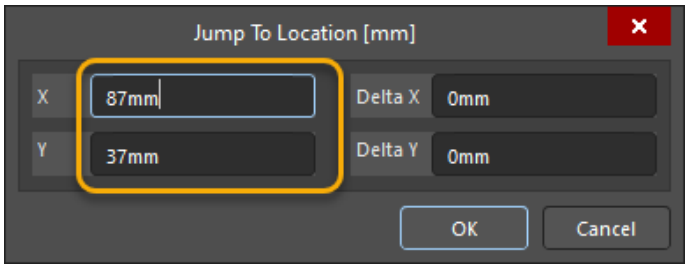


Figure 2. Jump to Location dialog

8. To jump to a component using its reference designator, press the **J » C** keys in succession. Enter `U1` into the resulting *Component Designator* dialog as shown in Figure 3 below.

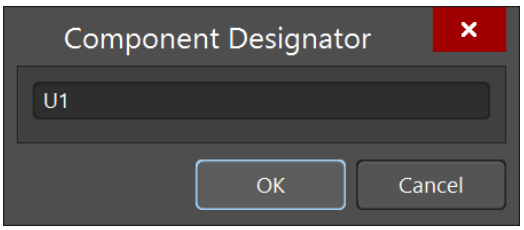


Figure 3. Jump to Component dialog

9. Press **OK** and the cursor will jump to the component `U1`. It will zoom and pan only if it's necessary to reach the location of the component in the event it is currently outside the visible area.

5 Exploring the PCB Panel

Just like the Navigator Panel in the Schematic, the PCB panel serves as a valuable tool for searching, finding, and jumping to various objects associated with the PCB. The PCB panel simplifies tasks such as locating components, tracing signal paths, and accessing various design-related items, including 3D views. To explore the different sections of the PCB panel, navigate through the drop-down menu options.

10. If not already opened, open the *PCB* panel by selecting the **Panels** button Panels in the lower right corner and then selecting **PCB.**
11. The PCB panel has six different areas, see Figure 4:
    a) The Panel Mode
    b) Filter Option
    c) Object Classes
    d) Objects
    e) Primitives
    f) Panel Mini-viewer

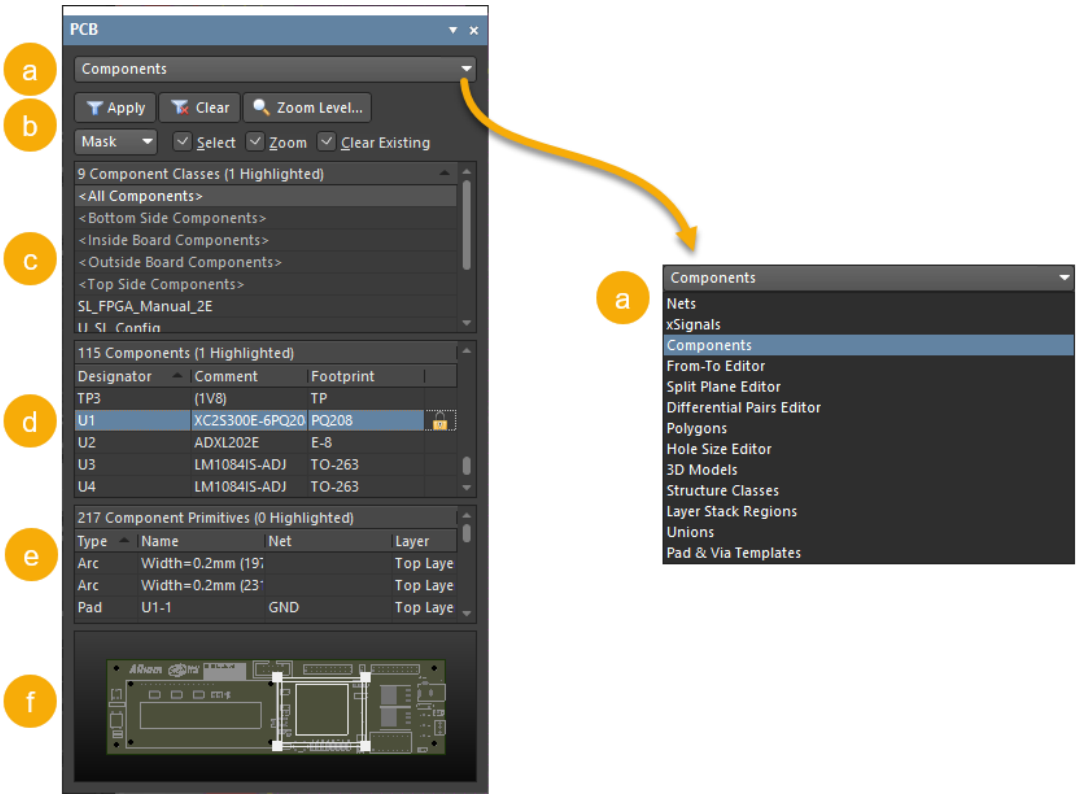


Figure 4. PCB Panel

12. Set the Filter Options as shown in Figure 5.
    a) **Mask (feel free to try the two other options, Dim or Normal).**
    b) Activate the options **Select.**
    c) Activate the options **Zoom.**
    d) Activate the options **Clear Existing.**

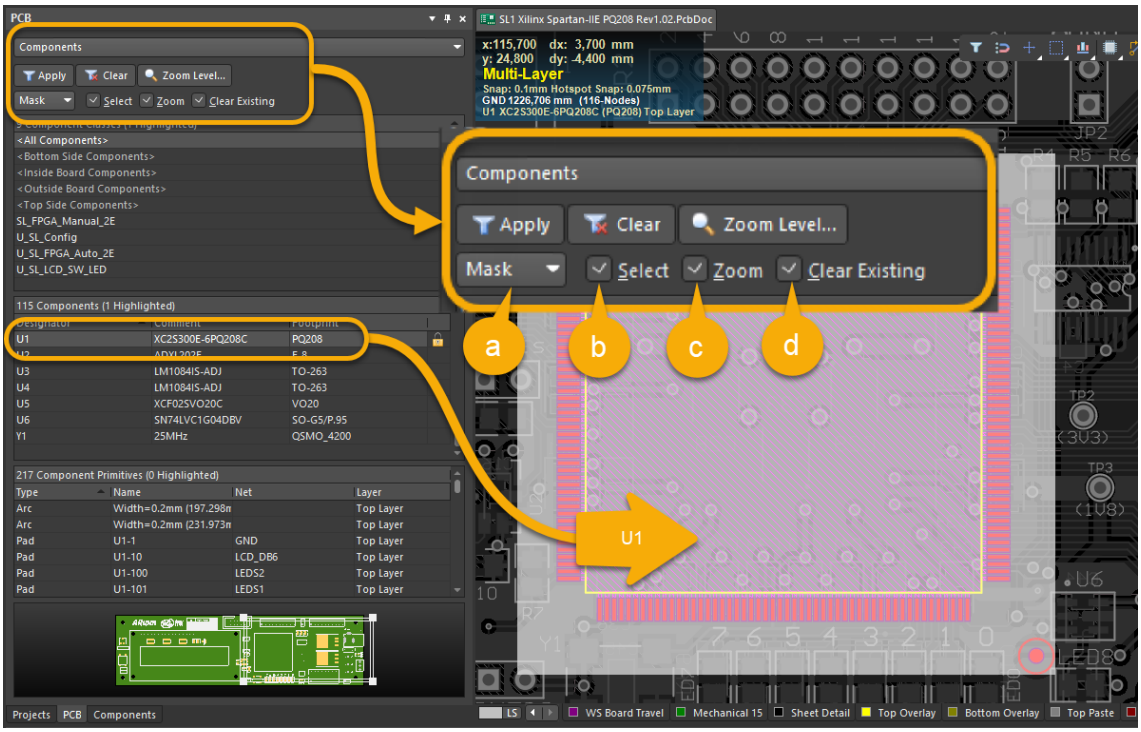


Figure 5. Filter Options for PCB Panel

5.1 Search for Components

13. Select **Component** Mode from the *Panel Mode* drop down list.
14. Select the Class **<All Components>.**
15. Select the Component **U1** in the Objects list to highlight (a), select (b) and jump to (c) U1.

5.2 Search for Nets

16. Select **Nets** from the *Panel Mode* drop down list.
17. Select the Class **<All Nets >**.
18. Select the net **5V** in the objects list to jump and highlight the net.
19. Press the button **Clear**, or press **Shift+C**, to remove the masking and selection.

6 Cross Probe Mode

20. Keep the PCB document `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PcbDoc` open.
21. From the *Projects* panel, open the `SL_FPGA_Auto_2E.SchDoc`.
22. Go to **Tools » Cross Probe** (alternatively use right mouse button menu and select **Cross Probe**)**.** The mouse cursor should now be a crosshair.
23. Select component `U1`. The PCB should briefly pop up before returning to the schematic.
24. Press **ESC** to stop the **Cross Probe** mode (alternatively right mouse click to exit an active command).
25. Switch to the PCB to check that U1 is now also masked in the PCB.

Hint: Holding the **CTRL** key before selecting the component during the Cross Probe command will maintain the view of the selected component in the PCB, instead of just a brief pop-up.

Hint: Feel free to move (Drag and Drop) the PCB document to the second screen for Cross Probing. If you have the two documents on two screen you can always see the Cross Probe information, instead of just a brief pop up.

26. Press **Shift+C** to clear the mask.

7 Cross Select Mode

Hint: **Tools » Cross Select Mode** allows a PCB component to be selected directly from its schematic symbol counterpart. Used in conjunction with the PCB's **Tools » Component Placement » Reposition Selected Components** command, it enables a very powerful and effective way to go about component placement.

27. Verify that cross-select mode is enabled within the *Preferences*, Figure 6.
    a) Open the *Preferences* and head to the *System branch,* then to the *Navigation page*.
    b) Verify that the **Cross Selection** option is enabled. The *Cross Select Mode* section also controls the selection scope.
    c) Click **OK** to close the preferences dialog.

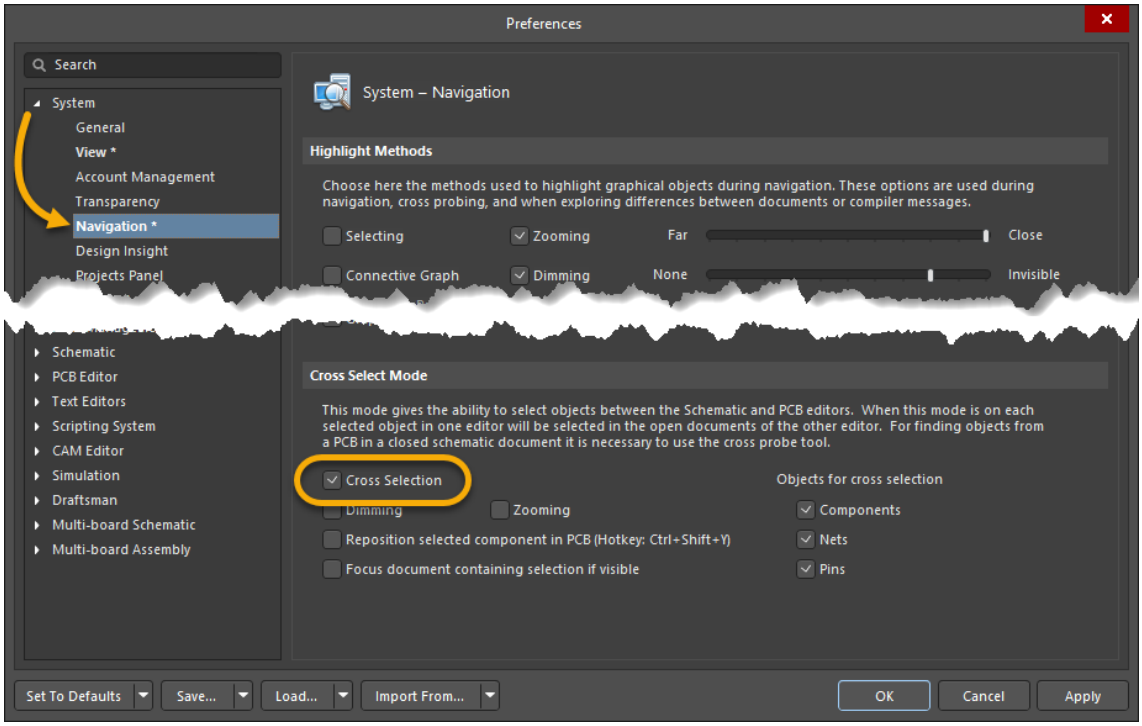


Figure 6. Preferences - Cross Selection

28. To toggle the Cross Selection Mode, go to **Tools » Cross Select Mode**. If the icon has a blue outline around it, then *Cross Select Mode* is enabled as shown in Figure 7 below.

Figure 7. Cross Select mode enabled

29. Navigate to the *Projects* panel and double-click on `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc` to open the schematic document.
30. Use **Jump » Component** to find U2. Left-click on U2 to select it.
31. Switch back to the `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PcbDoc` PCB document to see the `U2 – ADXL202E` (Footprint `E-8`) now selected.

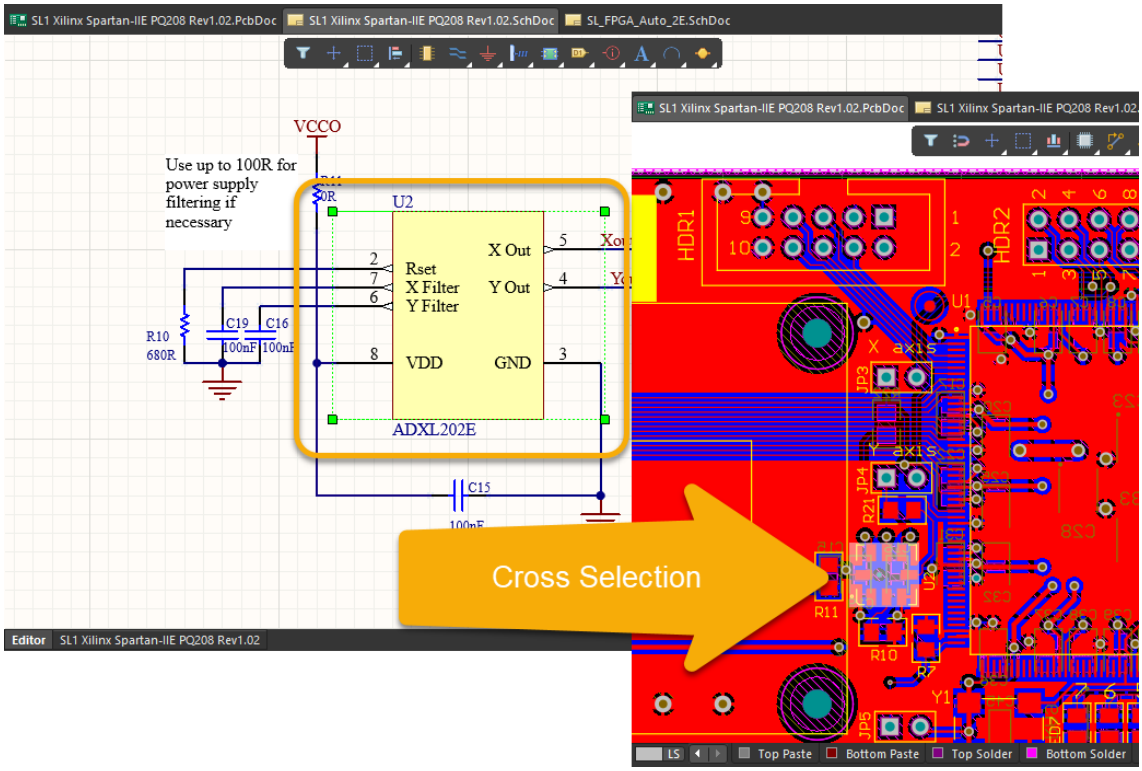


Figure 8. Cross Selection from SCH to PCB for U2

8 PCB Layers

The visible layer tabs will be displayed at the bottom of the workspace. These are the layers that are a part of your PCB design.

32. We can toggle through the different layers in multiple ways. Notice the Heads Up Display (HUD) in the top left corner will display the current layer name.
    a) Click the two small arrows beside the layer tabs to scroll through the tabs, similar to what is shown in Figure 9.
    Click directly on a layer tab, to change it to the active layer.
    b) Press the "+", "-", or "*" shortcut key on the numerical keypad to change the active layer.
        - "+" Next Layer
        - "-" Previous Layer
        - "*" Next electrical Layer
    c) A layer change can be done without a numerical keypad, or by holding **Shift+CTRL** and scrolling your mouse wheel forward or back.

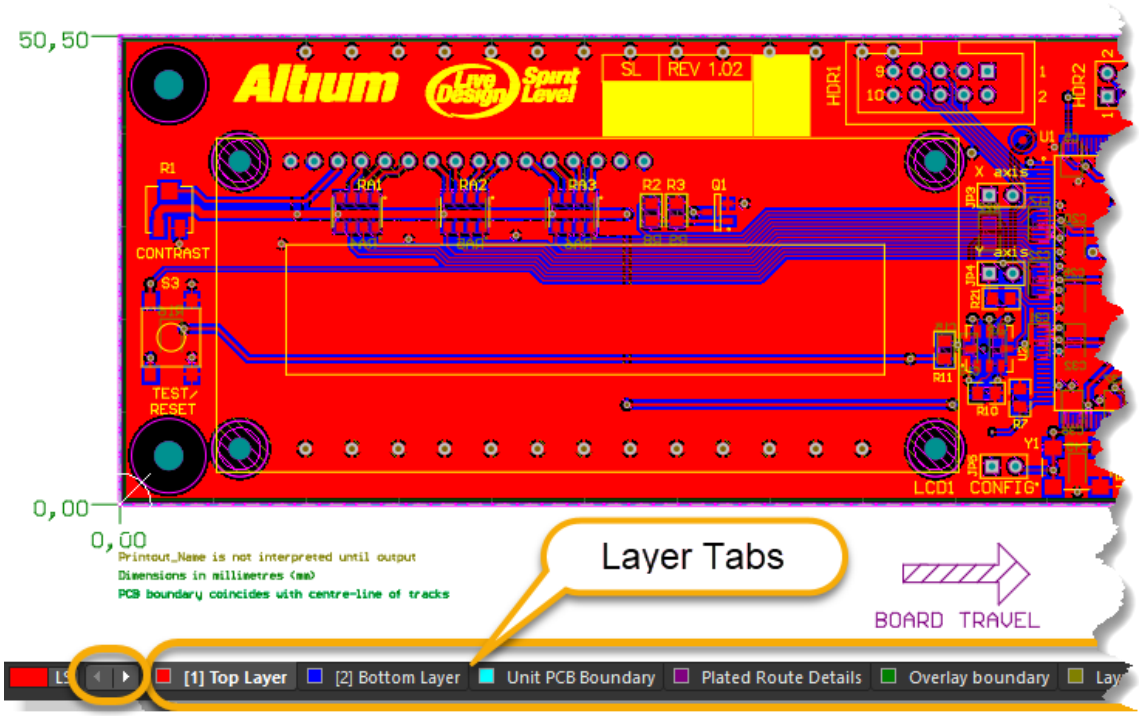


Figure 9. Layers tab showing all available layers

9 View Configurations

9.1 Layers & Colors

33. Open the *View Configuration* panel by clicking on the **Panels** button in the lower right corner and selecting **View Configuration,** or you can use the **L shortcut key**.
    a) The *View Configuration* panel will open, and the *Layer & Colors* tab will appear as shown in Figure 10 below.

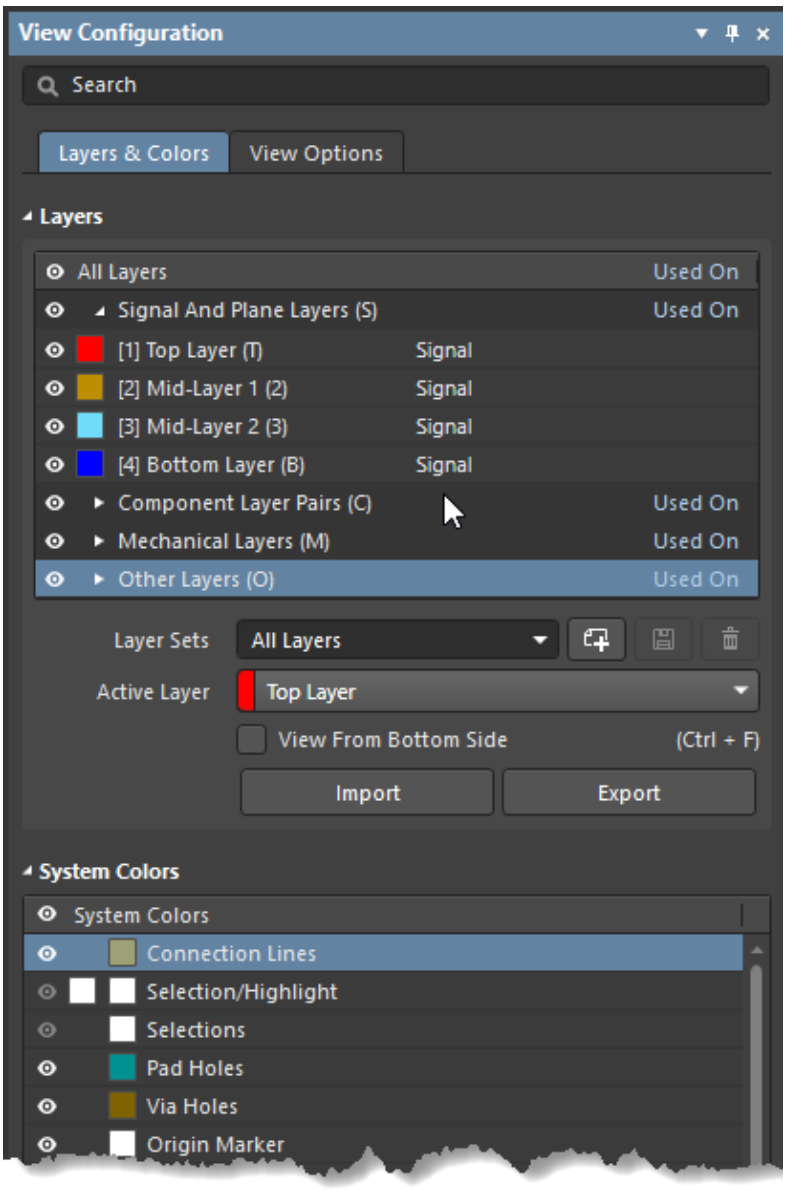


Figure 10. View Configuration panel

    b) Enable or disable the visibility for different layers by clicking on the icon next to the layer color and name.
    c) Click on a color box to change the color of a particular layer. Make note of the original layer color in case you want to go back to that color.

9.2 View Options

34. Select the *View Options* tab as shown in Figure 11 below.
    a) Under *Object Visibility,* click on the icon for polygons to hide the polygons.

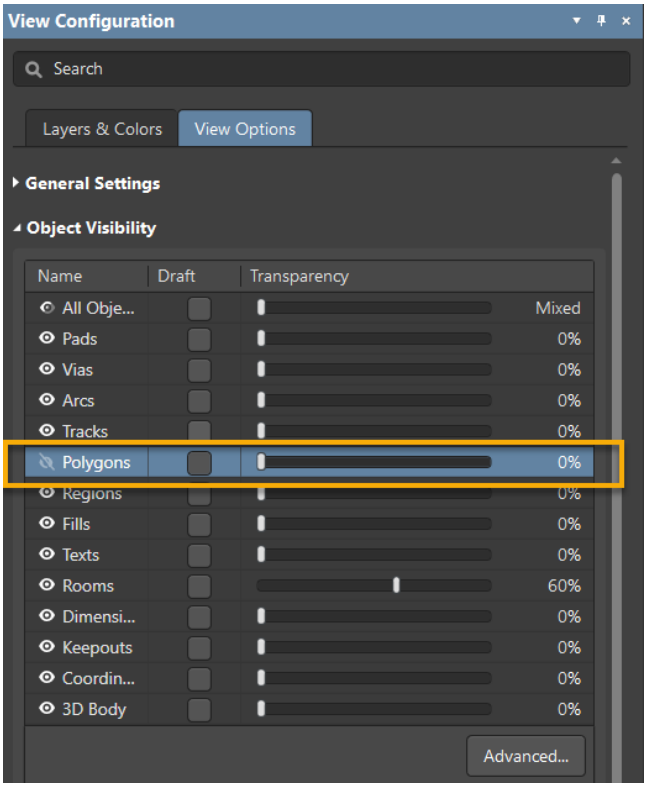


Figure 11. Object Visibility tab in the View Configuration panel

Caution: Although the polygons are not visible while hidden, they will still affect your design. They are still considered objects/obstacles for other features such as Interactive Routing. If you prefer to temporarily disregard them from your design, you can go to **Tools » Polygons Pours » Shelve Polygons.**
You will find more about Polygon Pours at Module 23**.**

    b) Under *Object Visibility,* click on the eye icon for polygons to show the polygons again.
    c) Move the slider for the polygon to change their transparency. You will also notice that you can manually change the % of transparency, beside the slider.
    d) View the results in the PCB. If you've changed the color of a layer in Step *33.c)* above, change it back to its original color.

35. Experiment with other options and observe changes in the PCB view.

36. You can revert to the default **Altium Standard 2D** configuration by selecting **Altium Standard 2D** from the *Configuration* drop-down list as shown in Figure 12 below.

Hint: Please note that changing the configuration back to Altium Standard 2D does not revert any changes that you've manually made to any layer colors.

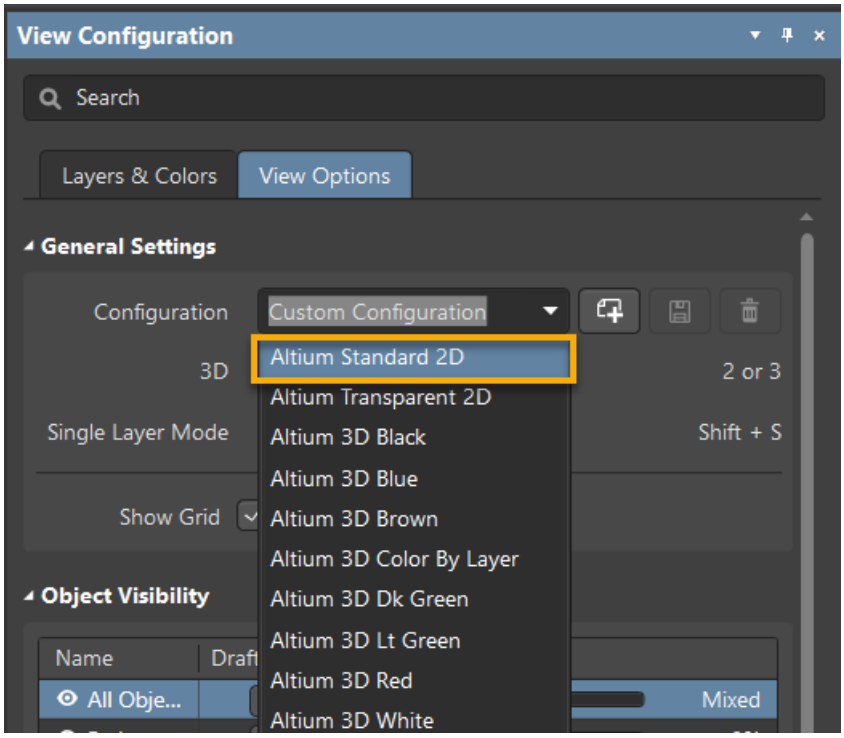


Figure 12. Different predefined Views

10 Explore the 3D Section View

37. Hit the **3** key to ensure you're in 3D mode.
38. Select the *View Options* tab as shown in Figure 13 below.
39. Activate the section view with **View »Toggle Section View** or select the **Edit** Section View | Edit | On | Off.
40. Use the colored arrow of the section view gizmo  to change the section view.

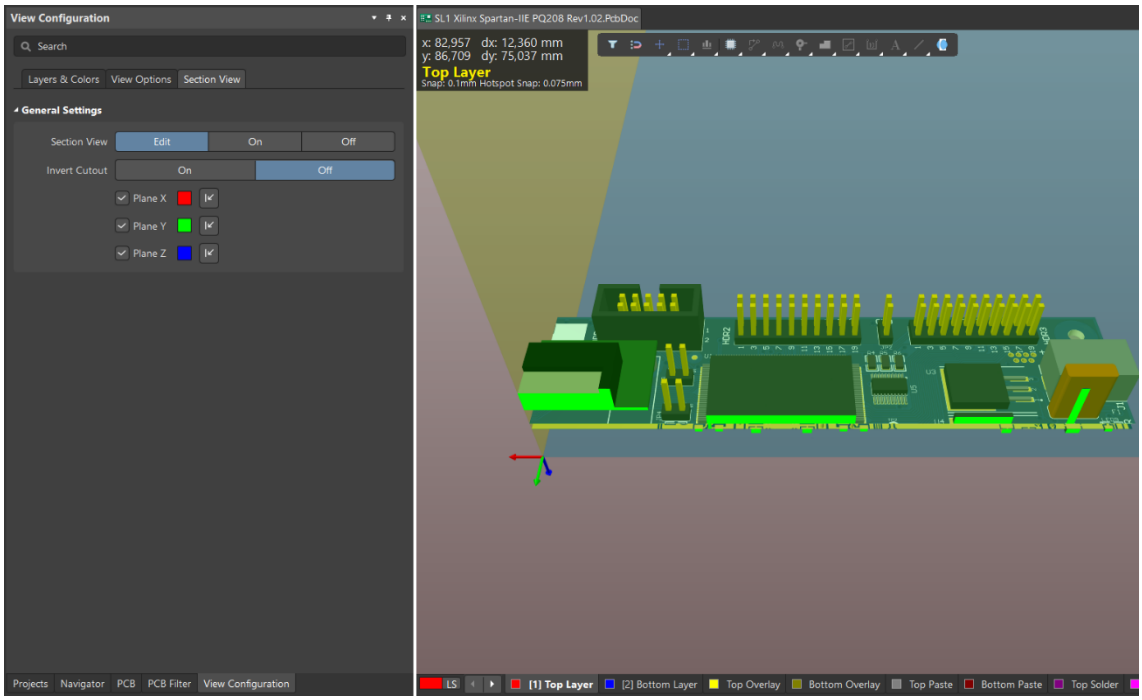


Figure 13. Section View

11 Board Insight Display - Single Layer Modes

41. Open the *Preferences*:
    a) From the *PCB Editor* section, select the *Board Insight Display* page.
    b) Enable all options under *Available Single Layer Modes* as shown in Figure 14, if not enabled already. Then, press **OK**.

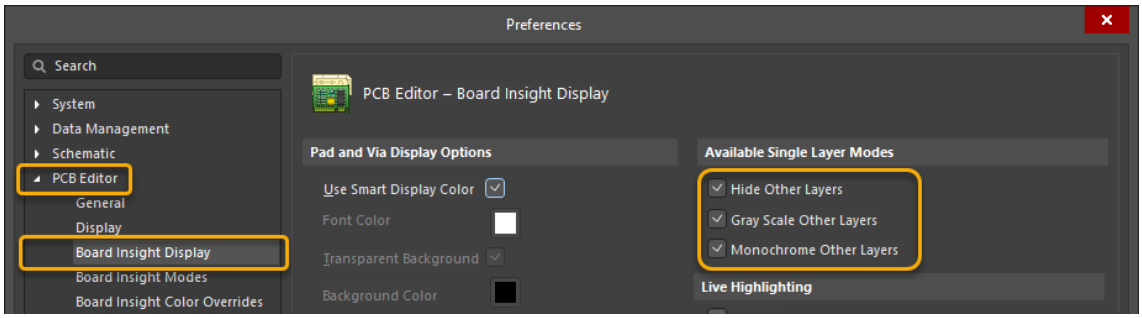


Figure 14. Available Single Layer Modes

42. Select the `Top Overlay` layer tab to make it the active layer.

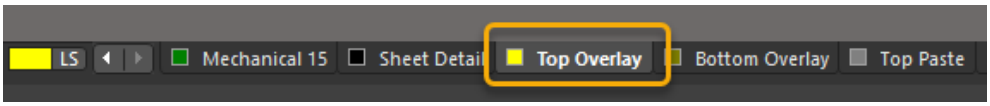


Figure 15. Top Overlay is the Active Layer

43. In the PCB document, toggle through the single layer modes by pressing **Shift + S**. Notice the differences between the modes as indicated in Figure 16, Figure 17, Figure 18, and Figure 19 on the following pages.

Hint: Notice in the HUD. It will show (Single) when you are in Single-Layer Mode. Single-Layer mode is a very useful viewing mode when you simply want to work on a specific layer and would like to hide the other layers that you're currently not working on.

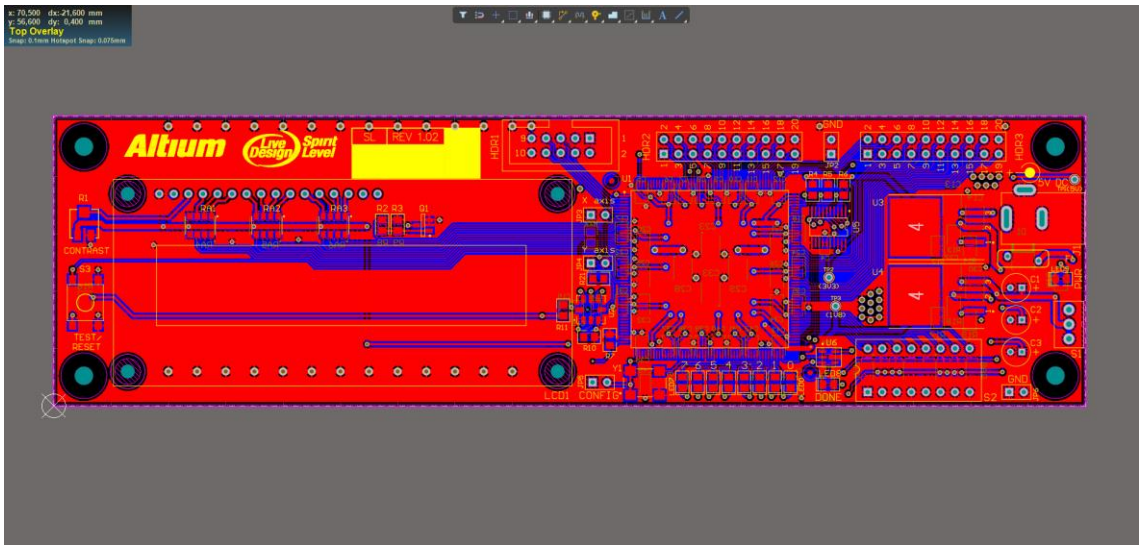


Figure 16. Regular 2D view

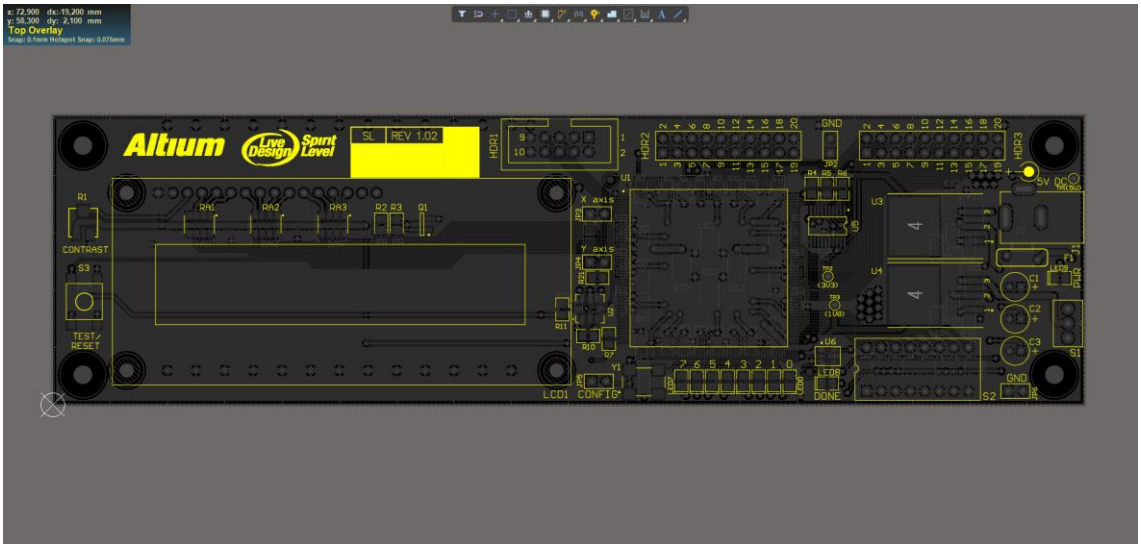


Figure 17. Single Layer Mode - Grayscale other layers

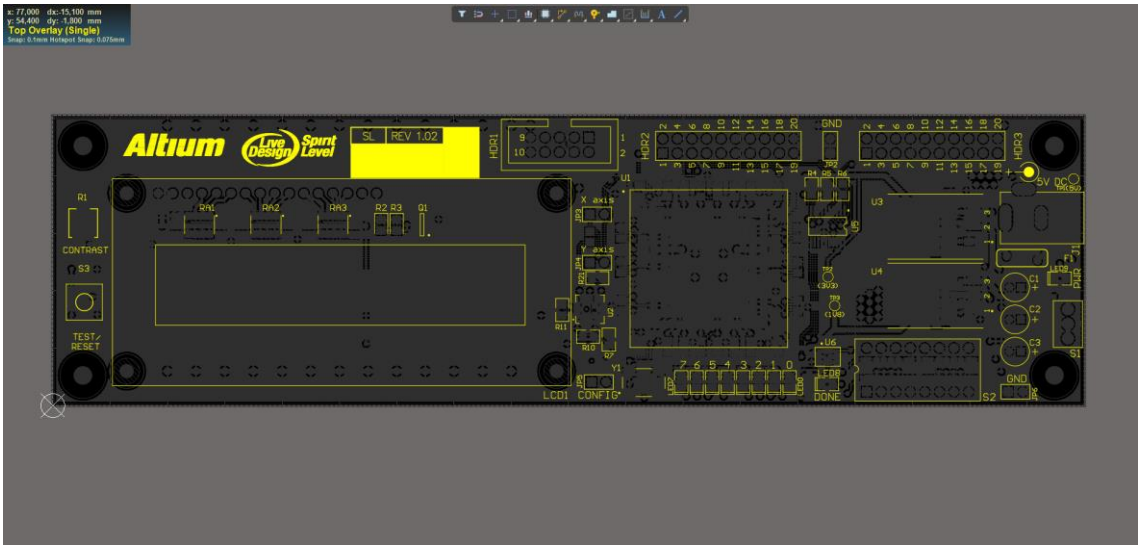


Figure 18. Single Layer Mode - Monochrome other layers

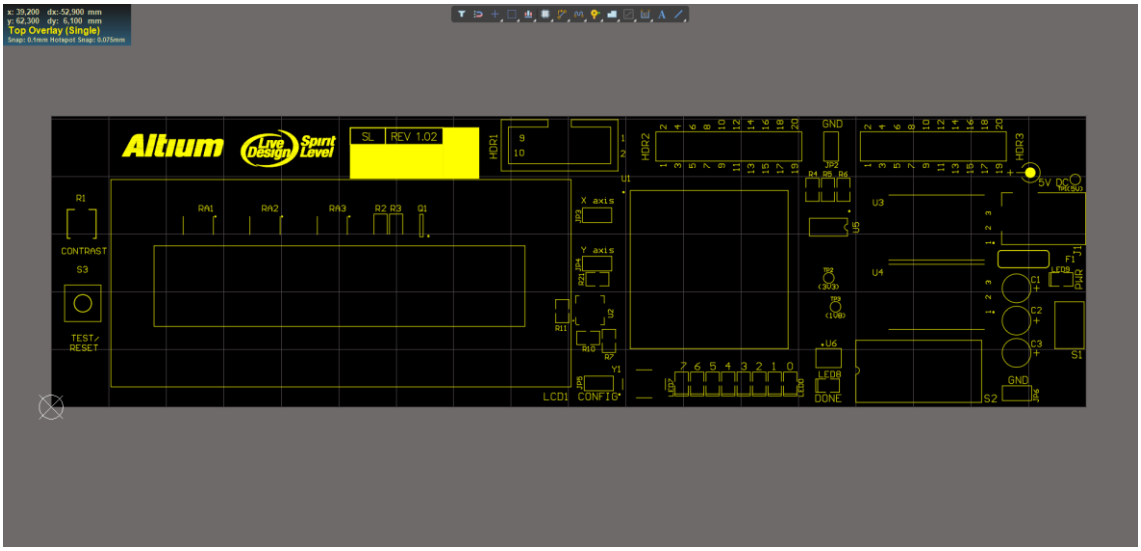


Figure 19. Single Layer Mode - Hide Other Layers

44. Return to the non-single layer mode using **Shift+S** until you see all of the layers again.

12 Display

45. Open the *Preferences* and head to the *Display* page from the *PCB editor branch*.
46. Under the *Highlighting Options* section:
    a) Enable all of the options, Figure 20.
    b) Press **OK** to exit the *Preferences* dialog.

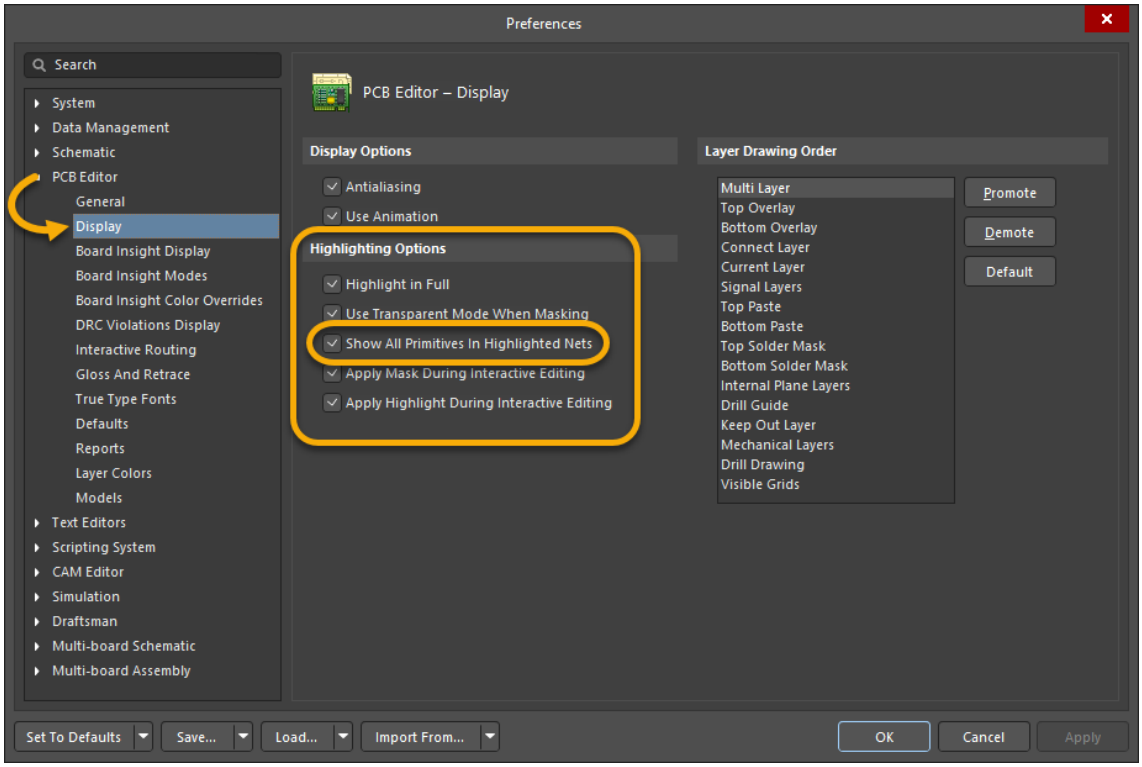


Figure 20. Highlighting options for PCB

47. Jump to component `R4` using the shortcut **J»C** and entering `R4` into *Component Designator* dialog. Then, zoom into the component.
48. If not already the active layer, make the Top Layer the active layer by selecting it from the layer bar at the bottom of the workspace. As alternative hit **'*'** from the keypad to change to the routing layers.
49. Enter Single Layer Mode again by pressing **Shift+S. All layers are now gray** except the active layer (Top Layer).

50. Select Pad 2 of `R4`, using the command **Ctrl + left-click**. Notice that tracks connected to that pad are now highlighted as well as the pads, as shown in Figure 21 below. **Do Not** Clear the highlight.

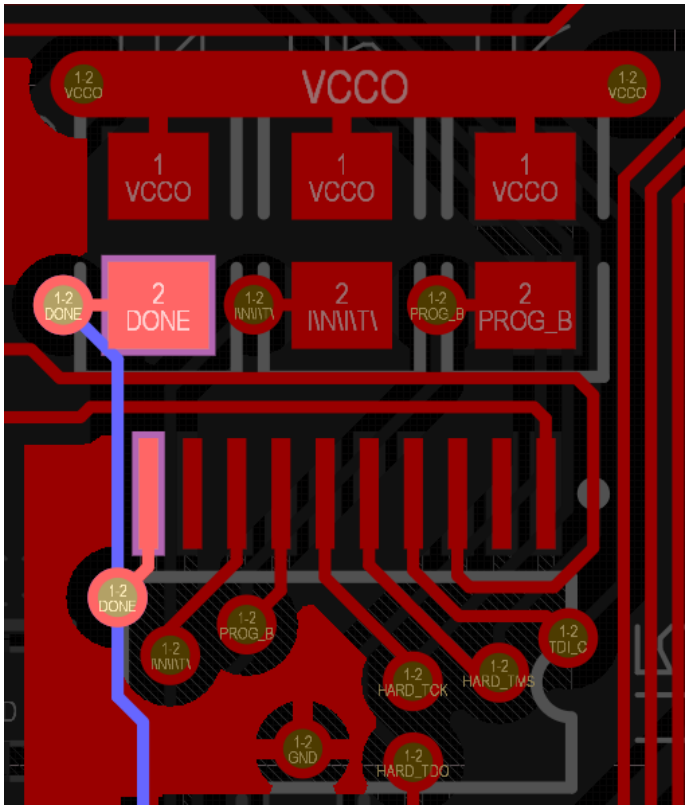


Figure 21. Highlight all primitives

51. Press **Shift+S** again**,** and now all other layers are hidden instead of the Top Layer and the highlighted tracks on the Bottom Layer.
52. Open the *Preferences again, and head to* the Display page under PCB Editor.
    a) Disable **Show All Primitives in Highlighted Nets**.
    b) Click **OK** to confirm. Notice that tracks on the Bottom Layer are no longer highlighted.
53. Clear any selections or highlights by pressing **Shift + C**.
54. Exit single layer by pressing **Shift + S** until all layers are showing.

13 Board Insight Display - Live Highlighting

55. Open the *Preferences* and head to the *Board Insight Display* page from the *PCB editor branch*.
56. Deactivate the Option **Live Highlighting only when Shift Key Down,** Figure 22**.**

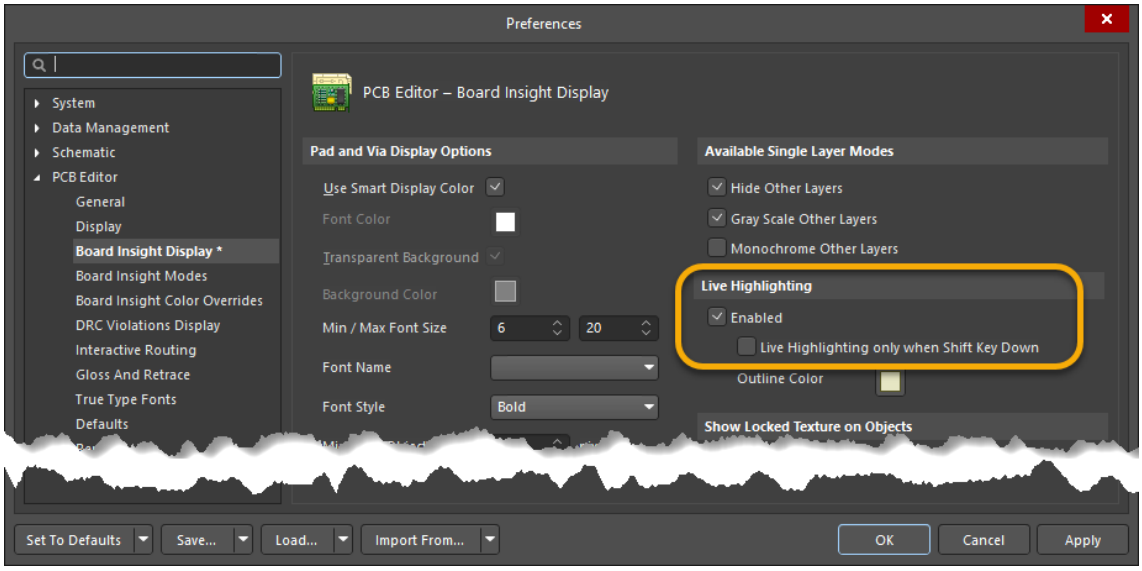


Figure 22. Live Highlighting only when Shift Key Down Option

57. Close the *Preferences*.
58. Hover with your mouse over `R4 – Pad2`, a Live Highlight of the net `Done` is shown, Figure 23.

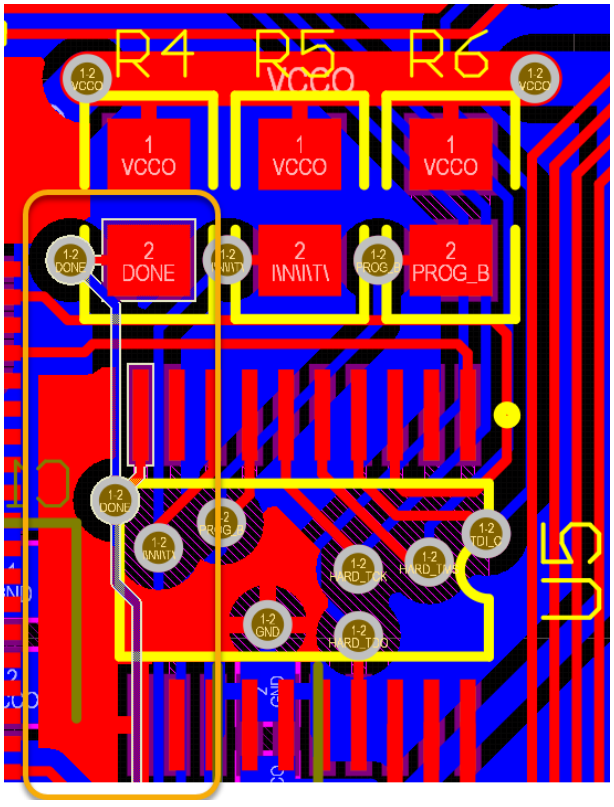
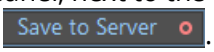


Figure 23. Live Highlight for net DONE

59. Feel free to change the Preference setting back to **Live Highlighting only when Shift Key Down.**
60. Feel free to save the modifications you have done, using **File » Save All**.
61. Save the modifications to the server:
    a) At the *Project* panel, next to the Project name you find the command **Save to Server** 

    .
    b) Select **Save to Server.**
    c) At the dialog *Save [Project Name],*
        i) Add the comment `Module 16: PCB Viewing and Navigation - [Add Your Name] - Finished`.
        ii) Select **OK.**
62. When ready, close the project and any opened documents, **Window » Close All**.

Congratulations on completing the Module!

Module 16: PCB Viewing and Navigation

from

Altium Designer Essentials Training with Altium 365

Thank you for choosing **Altium Designer**