



ALTium **365**

Altium Designer

Essentials Course - Altium 365

Module 4: Design Environment

Software, documentation and related materials:

Copyright © 2024 Altium LLC

All rights reserved. You are permitted to use 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 are made. Unauthorized duplication, in the 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 LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

ACTIVEBOM®, ActiveRoute®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter®, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

Table of Contents

Module 4: Design Environment	3
1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	4
1.4 The Altium Designer Environment	4
1.5 Navigation within the Schematic Workspace	6
1.5.1 View Menu Hotkey Sequences	6
1.5.2 Basic Navigation using the Mouse.....	6
1.5.3 Basic Navigation using the Keyboard.	7
1.6 Documents & Panels	8
1.6.1 Document Bar	8
1.6.2 Panel Access	9
1.6.3 Panel Display Modes	10
1.6.3.1 Docked Mode	10
1.6.3.2 Pop-up Mode	11
1.6.3.3 Floating Mode	12
1.6.4 Grouping Panels	13
1.6.4.1 Tabbed Panel Grouping	13
1.6.5 Closing Panels	13
1.6.6 Maximizing/Restore Panels	13
1.6.7 Desktop Layouts	14
1.7 Toolbars	15
1.7.1 Active Bar	15
1.7.2 Classic Toolbars	16
1.8 Menu Operations	17
1.8.1 Right-Click Context Menu	17
1.8.2 Shortcut Keys	17
1.9 Status Bar and Command Status	18
1.10 Undo and Redo	19

Module 4: Design Environment

1.1 Purpose




In this exercise, you will learn to use some general GUI and navigation functions. You will have mastered the basic operations of the Altium Designer environment after the completion of this exercise. This will be the foundation for future studies

Altium Designer boasts a unified design environment. Upon your initial launch of Altium Designer, you will quickly notice that everything you require is seamlessly integrated within a single application. This encompasses schematic design, simulation, signal integrity analysis, PCB design tools, documentation outputs, and the capability to create libraries.

1.2 Shortcuts



Popular shortcuts when working with Module 4: Design Environment

Ctrl+Z or Alt+Backspace:	Undo.
Ctrl+Y or Ctrl+Backspace:	Redo
Ctrl+Tab or Ctrl+Shift+Tab:	Cycle between the open document tabs
 or T » P:	Preferences
V » D:	View » Fit Document
V » F:	View » Fit All Objects
Mouse Wheel:	Page up - Page down
Mouse Wheel+Shift:	Page left - Page right
Mouse Wheel+Ctrl:	Zoom in - Zoom Out
Page Up or V » I:	Zoom in
Page Down or V » O:	Zoom out

1.3 Preparation

1. If you have closed the project SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PrjPcb from the last module, reopen it. The project may also be available at the File menu, under Recent Projects.

1.4 The Altium Designer Environment

2. Locate the Projects panel on the left of the window, as shown in Figure 1 below, and double-click on the schematic file SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc. This will open the schematic document within the active workspace and provide the working view of the schematic.

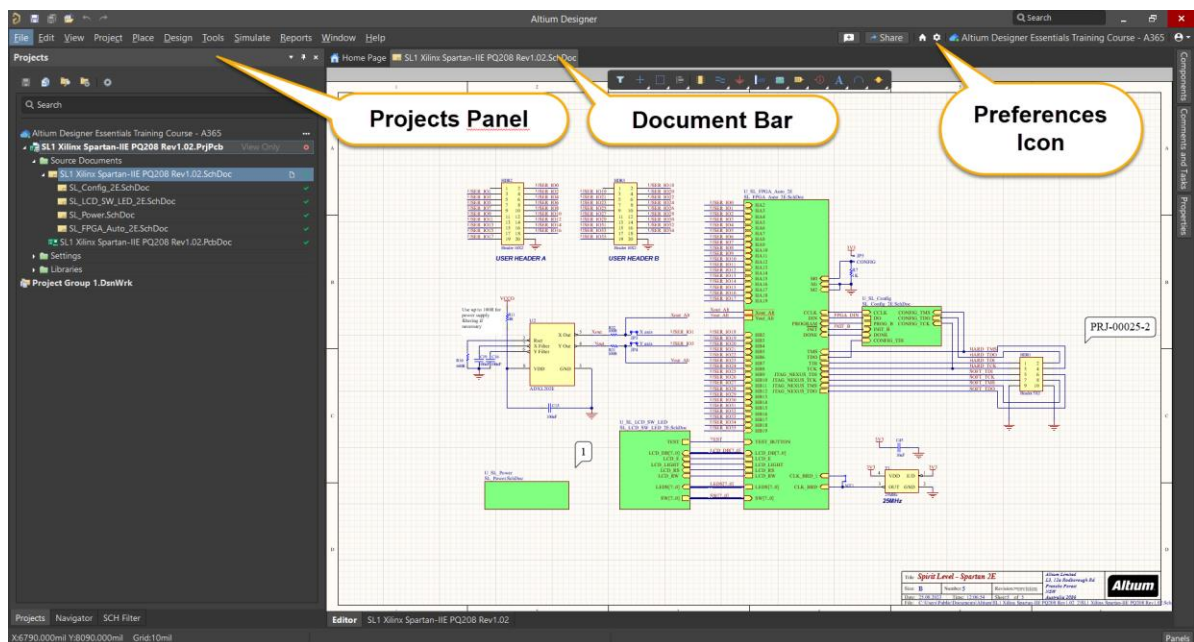


Figure 1. Altium Designer Environment

3. In the *Projects* panel, double-click on the PCB file SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PcbDoc, Figure 2. This will open the PCB document within the active workspace and provide the working view of the PCB.

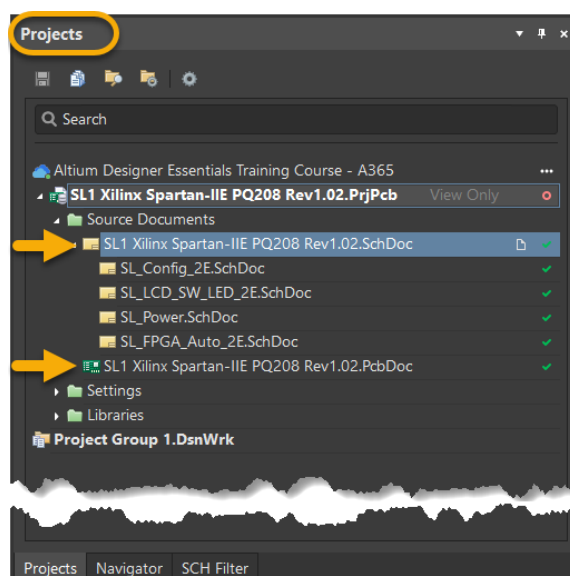


Figure 2. Project Panel, double click to open linked documents

4. Whether you are working in the schematic or the PCB, you will now notice several tabs in the Document Bar located at the top of the editor, Figure 3.
 - a) Click on each of these tabs and observe the workspace change to match the selected document.
 - b) You can also use the shortcut keys **Ctrl+Tab** or **Ctrl+Shift+Tab** to cycle between the opened documents.

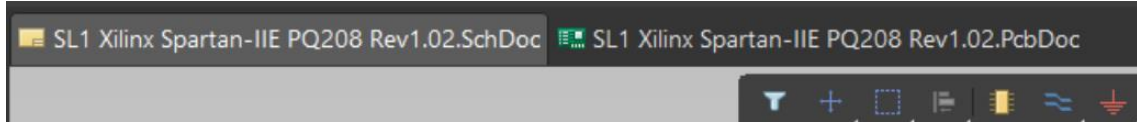



Figure 3. Document bar with one SCHDOC and one PCBDOC

5. If we open multiple documents to the point where the tabs fill up the entire bar, we can configure its grouping method. Click the *Preferences* icon  in the upper right corner of Altium Designer, Figure 4 Figure 1.

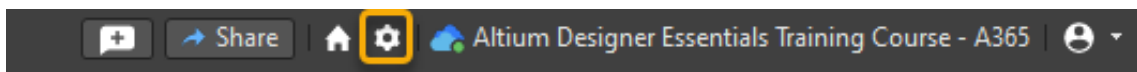


Figure 4. System Preferences

6. Expand the *System* branch and open the *View* page. Within the *Documents Bar* settings, you can enable the **Group documents if need** checkbox and set it to **By document kind** as shown in Figure 5.

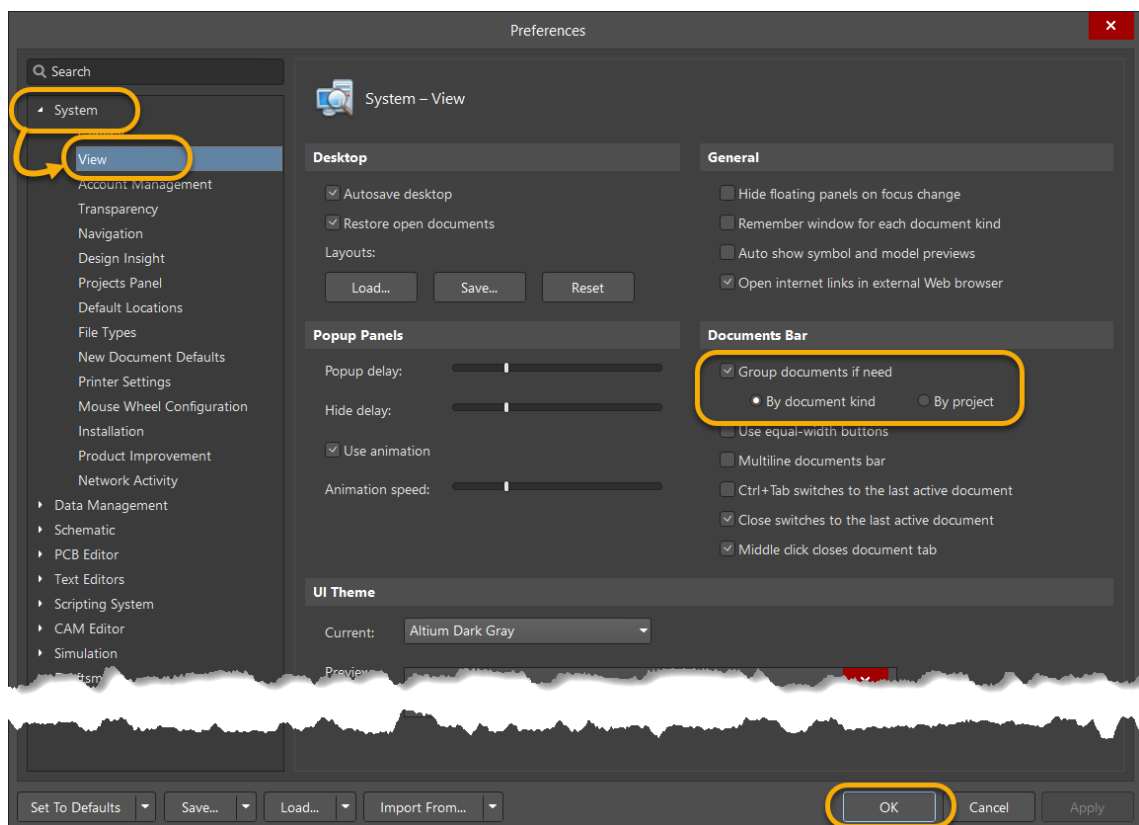


Figure 5. Preferences for document grouping

7. Click **OK** to close the *Preferences* window.



If enabled, the documents will be grouped as needed. If there is adequate space within your documents bar, the documents will not be grouped.

1.5 Navigation within the Schematic Workspace

1.5.1 View Menu Hotkey Sequences


8. Make the schematic file `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc` the focused document if not already.



Follow the underlined letters within menus and submenus to learn hotkey sequences.

- a) To **View » Fit Document** press the **V » D** keys in succession.
- b) To **View » Fit All Objects** press the **V » F** keys in succession.
- c) To **View » Area** or press the **V » A** keys in succession, define the viewing area by drawing the diagonal corners of a rectangular area.
- d) To **View » Around Point** or press the **V » P** keys in succession, by defining the view similar to the View Area command.
- e) To **View » Selected Objects** press the **V » E** keys in succession by selecting the desired objects in advance.

1.5.2 Basic Navigation using the Mouse

9. Maintain focus on the Schematic.
 - a) From the **View** menu, select **Fit All Objects** or use the key combination **V » F**.
 - b) In the schematic, right-click and hold until the panning hand cursor appears , Figure 6
 - c) Drag the schematic around while holding the right mouse button down.
 - d) Release the right mouse button to no longer pan.

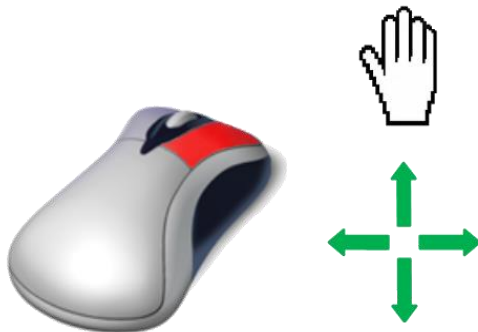


Figure 6 Pan with Mouse

10. To change the zoom, click & hold the mouse wheel, then move the mouse forward/back, Figure 7.

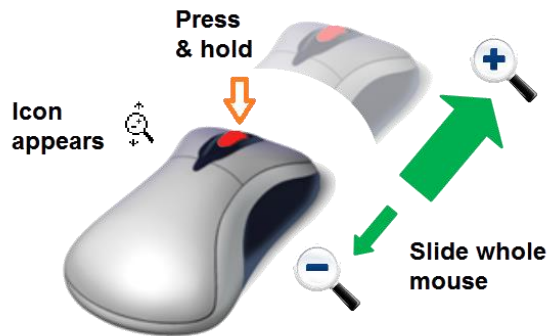


Figure 7. Zoom - Option with Mouse wheel

11. To Zoom in or out, hold the **Ctrl** key and scroll the mouse wheel.
12. To Scroll Up/Down, simply roll the mouse wheel (scrolling).
13. To move Left/Right: click and hold the **Shift** key and scroll the mouse wheel.



You can customise the Mouse Wheel - Key Combinations by accessing the Preferences in the 'System' section under 'Mouse Wheel Configuration'.

1.5.3 Basic Navigation using the Keyboard.

14. Maintain the focus on the Schematic and observe the behavior of the following Shortcut Keys:
- To Zoom out, press **PAGE DOWN**.
 - To Zoom in, press **PAGE UP**.
 - To Fit All Objects, press **CTRL+PAGE DOWN**.
 - To re-center your workspace based on your cursor position, press the **HOME KEY**.
 - To move in a direction by one Grid Step, press the **ARROW KEYS**.
 - To move in a direction by 10 Grid Steps press **SHIFT+ARROW KEYS**.

1.6 Documents & Panels

1.6.1 Document Bar

15. Right-click on a document in the *Document Bar* (as shown in Figure 8 below) and select **Tile All** in the pop-up menu. All of the currently open documents will be split into individual windows.



To resize the window, simply move the cursor to the divider of the two split windows. When a bi-directional arrow appears, left-click and drag to resize the split windows.

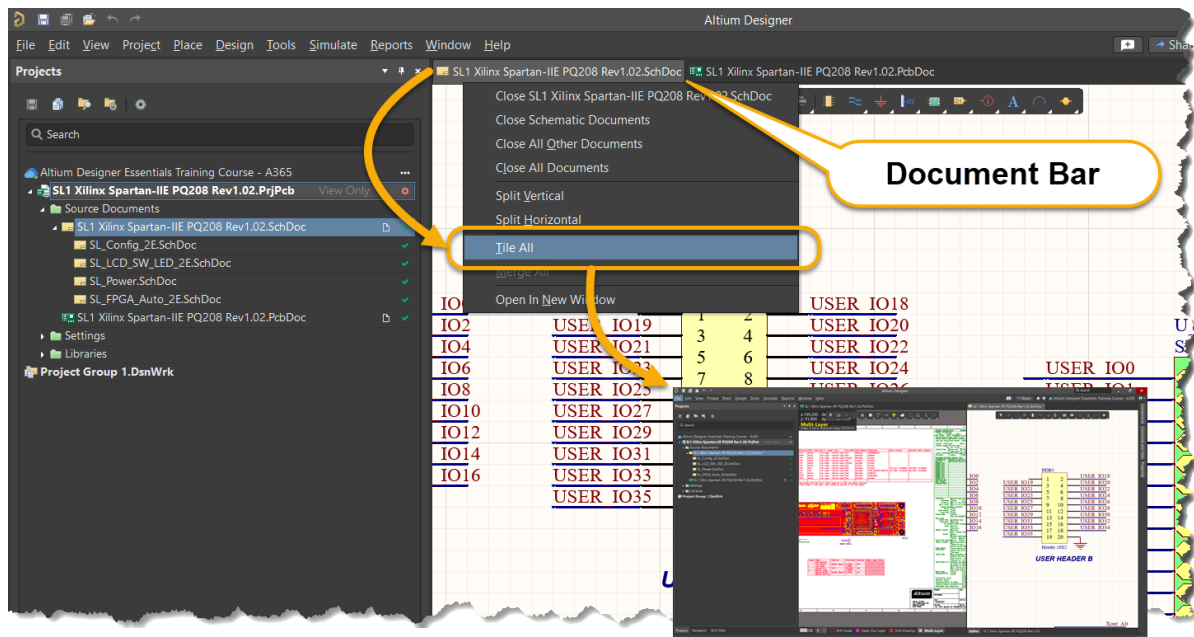



Figure 8. Document Bar

16. While documents are tiled or split within the editing workspace, right-click on any document in the *Document Bar* and select **Merge All** from the pop-up menu. The workspace will be restored to a single document display.
17. Feel free to experiment with the other options in the right-click menu like **Split Vertically** and **Split Horizontally**. For multi-monitor display, click on **Open in New Window** in the right-click menu to open a new instance or simply drag a document to the secondary display.

1.6.2 Panel Access

18. Locate the **Panels** button  in the lower right corner as shown in Figure 9 below.
19. Select **Properties** from the popup menu to open the *Properties* panel. The panels are also listed in the **View** menu, under **Panels**.

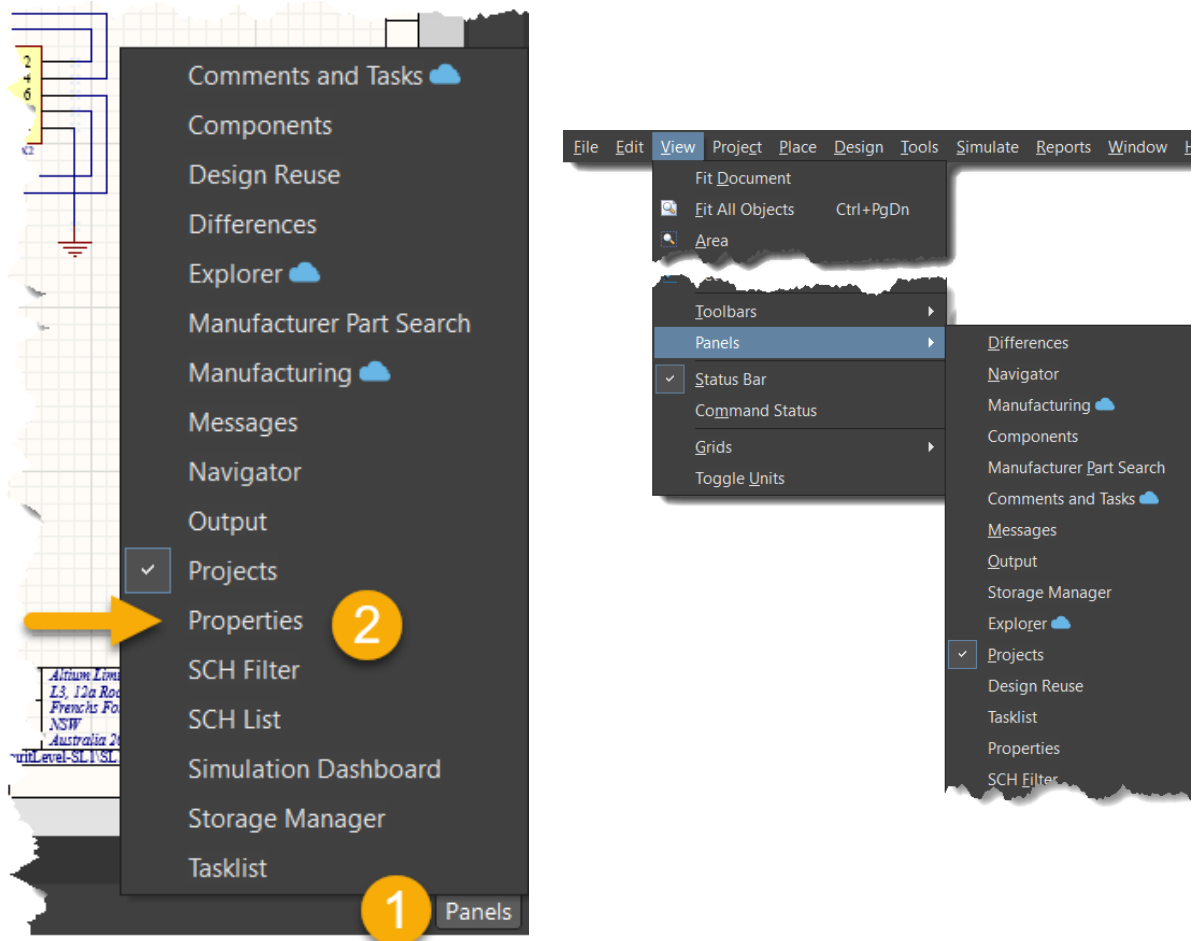


Figure 9. Panel access

1.6.3 Panel Display Modes

Three different display modes are supported for panels: Docked, Pop-up and Floating.

1.6.3.1 Docked Mode

20. Right-click on the title bar of the *Projects* panel. Click on **Allow Dock** in the pop-up menu as shown in Figure 10 below and ensure that it is enabled for both **Horizontally** and **Vertically**.

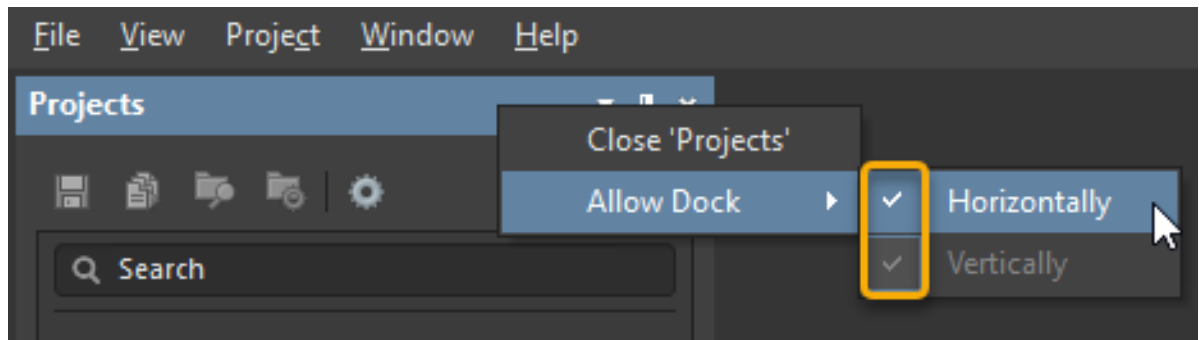


Figure 10. Allowed docking mode settings

21. Left click on the title bar of the *Projects* panel to move it.

- While holding the left-click, drag it to the center of the window and notice that while the panel is attached to the cursor, we have both horizontal and vertical docking options as shown in Figure 11 below.
- Hover the cursor above the left docking option and release the left-click hold.

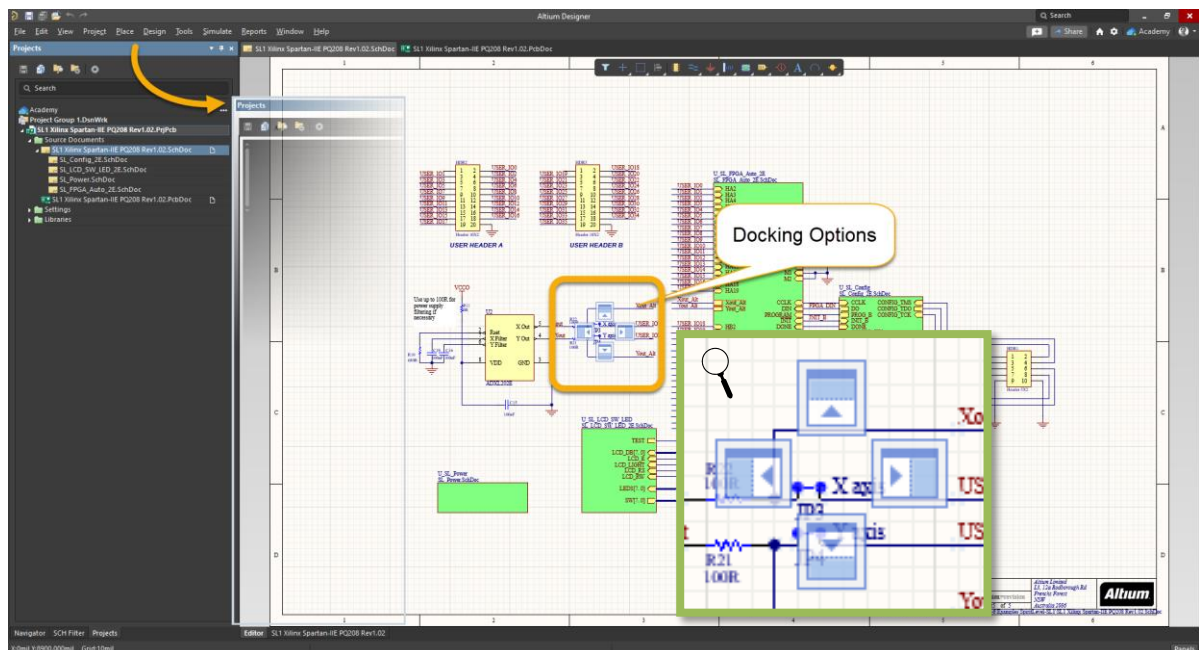



Figure 11. Docking Options

1.6.3.2 Pop-up Mode

22. Locate the *Components* panel tab on the right of the window, Figure 12. You may need to open the *Components* panel manually, by clicking on the **Panels** button  in the lower right. It will open in Pop-up mode along the right side of the workspace, by default. Hover above its tab to open the panel. This mode is what we refer to as Pop-up mode and is often used to maximize the screen's real estate, especially when working on a small monitor.

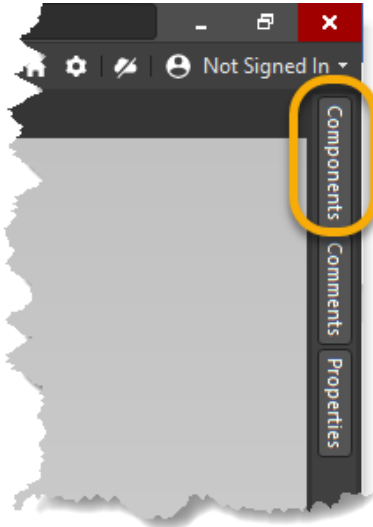





Figure 12. Pop-Up Panels on Right side of the Screen

23. To change the *Components* panel to Docked mode, click on the pin symbol  while the panel is open. The pin symbol will then change to  indicating that it is in Docked mode.
24. Selecting the pin symbol again will change the panel back to Pop-Up mode. While in this mode, moving the cursor away would hide the panel after a brief amount of time.
25. Note that the pop-up and hide delay is controlled in the  Preferences under the *System* branch, in the *View* page as shown in Figure 13.

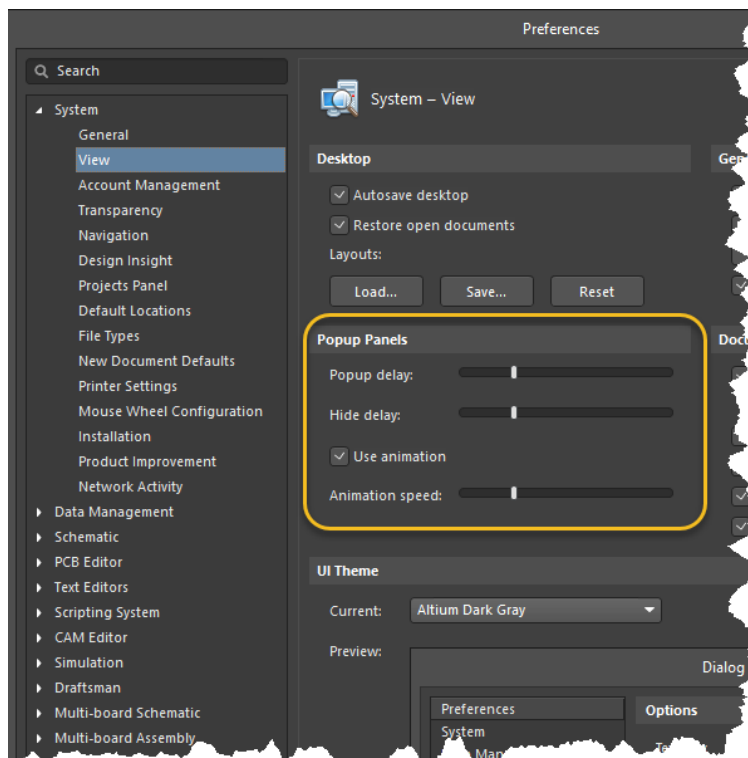


Figure 13. Pop-up panel delay settings

1.6.3.3 Floating Mode

26. In the bottom right corner of Altium Designer, click on **Panels** and select **Storage Manager** to open the *Storage Manager* panel shown in Figure 14 below. By default, this is a floating panel and handy to explain Floating Mode.

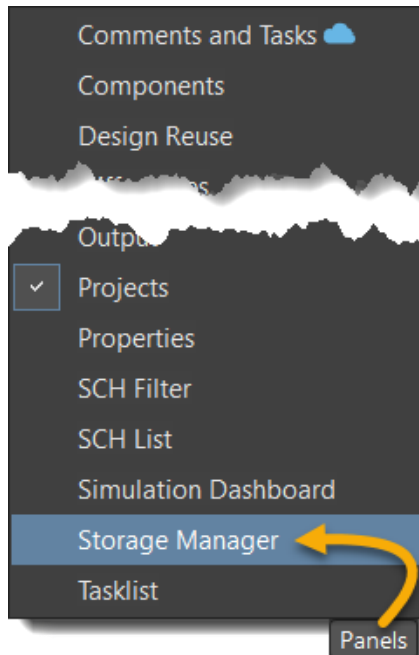


Figure 14. Opening the Storage Manager panel

27. A panel in this mode can be placed anywhere within or outside the Altium Designer environment. For example, the *Components* panel is commonly made to be floating and placed on another monitor. Move the *Storage Manager* panel to the middle of the window.

1.6.4 Grouping Panels

Panels are usually grouped together, this can be more commonly either on the left or the right of the editing area, it is also possible to group together at the bottom, for such as the *Storage Manager* and the *Messages* panel.

1.6.4.1 Tabbed Panel Grouping

28. By default, the *Projects*, *Navigator* and *SCH Filter* panels are grouped together into one panel. To open any one of them, click on the tabs at the bottom of the panel shown in Figure 15. This is what we refer to as Tabbed Panel Grouping.

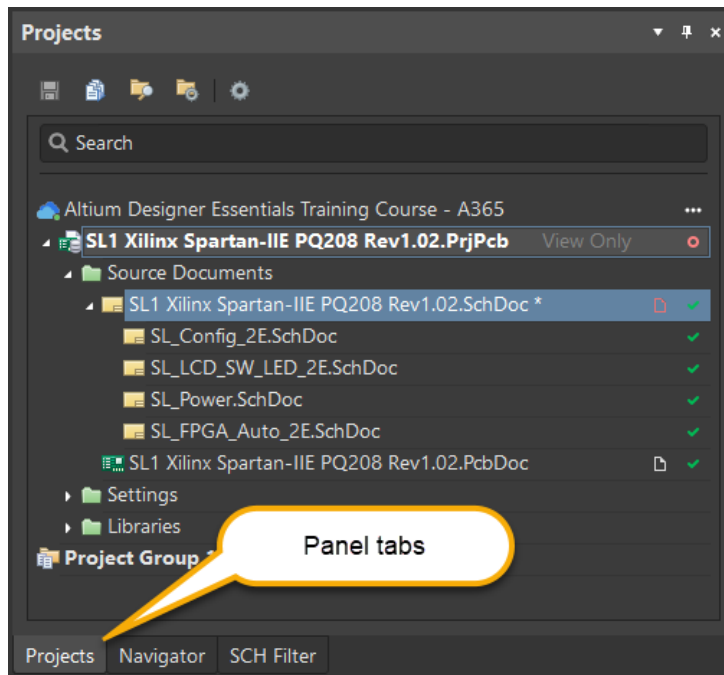



Figure 15. Tabbed Panel Grouping



To prevent a moving panel from automatically docking, grouping or snapping, hold the **CTRL** key


1.6.5 Closing Panels

29. Close any panel by right-clicking in its title bar or on its tab, then choose **Close** from the pop-up menu. Alternatively if you want to close a group of tabbed panels, you can click the **close icon**  at the top right of a panel's caption bar.

1.6.6 Maximizing/Restore Panels

- 30. When in floating mode, a panel can be maximized by right-clicking in its title bar (or on its tab) and choosing the **Maximize** entry from the pop-up menu.
- 31. To restore a maximized panel back to its original size, simply right-click in its title bar or tab and choose the **Restore** command from the pop-up menu. Alternatively, you can double-click the title bar to toggle between maximized and restored states.

1.6.7 Desktop Layouts

32. The panel arrangements are what we refer to as Desktop Layouts. It can be saved, loaded and reset to default in the  Preferences from the upper right corner of Altium Designer.
33. Open the *System* branch and open the *View* page as shown in Figure 16 below. Within the *Desktop* section, desktop layouts can be saved, loaded or reset to default.
34. Click on **Reset**. Close any panels that open automatically. Then, click **OK** to exit the Preferences.

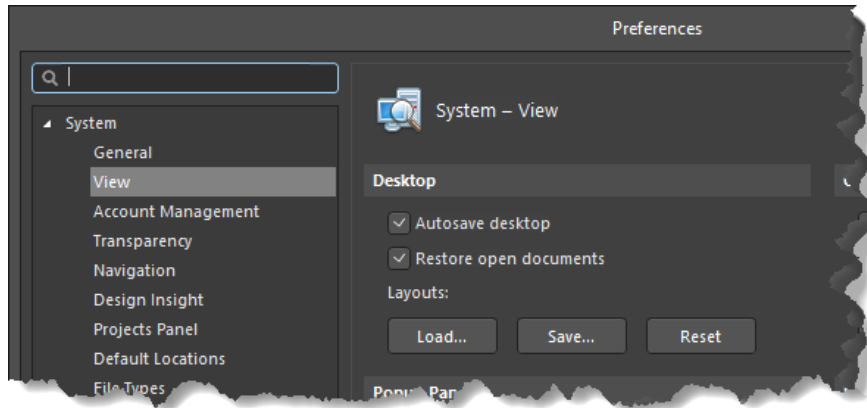



Figure 16. Desktop Layouts

1.7 Toolbars

Like menus and panels, both the *Active Bar* and *Classic Toolbars* are context sensitive, so that only commands that are relevant to the document of focus are displayed.

1.7.1 Active Bar

35. Make the schematic the current document of focus. At the top of the document, the *Active Bar* appears, as shown in Figure 17 below.
36. Move the cursor to any of the icons in the *Active Bar*. Wait a second for a hint to appear describing the function of the tool.
37. Right click on a toolbar icon with a triangle at the lower right side e.g.,  to open the option for that icon.

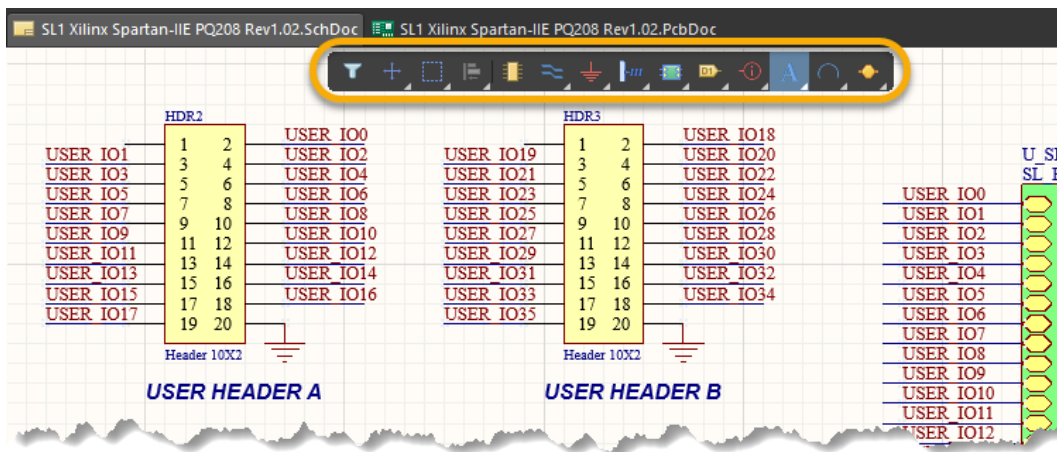


Figure 17. Active Bar in the Schematic Editor

38. Open the PCB document, if in 3D view switch to the 2D view, by hitting the **2** key.
39. Observe the icons as shown in the *Active Bar* and note the changes shown in Figure 18.

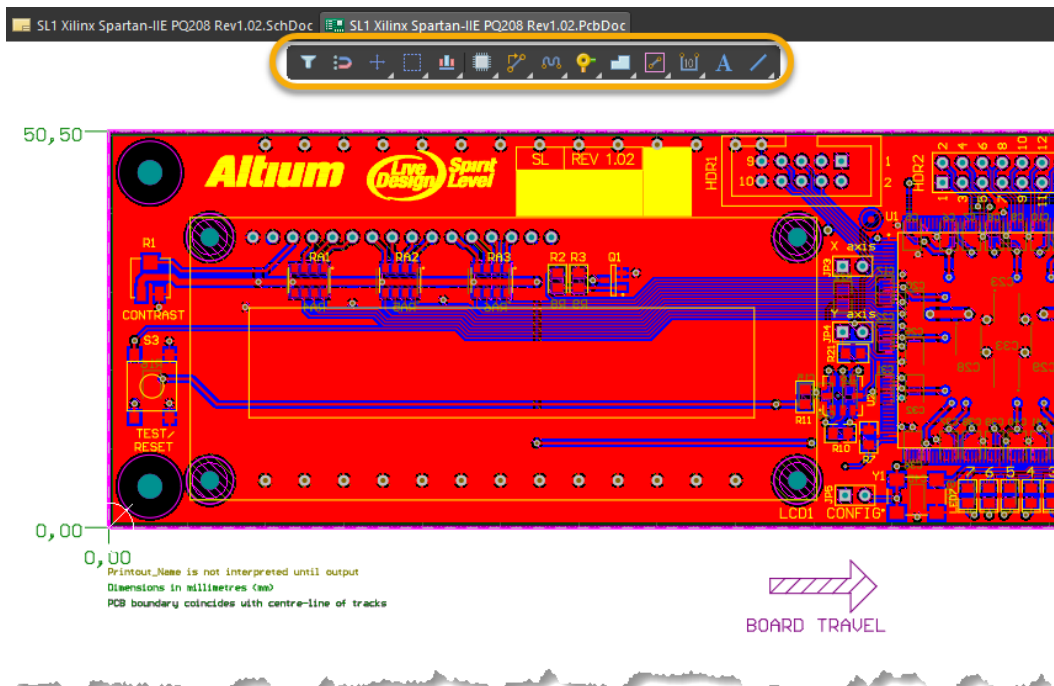


Figure 18. Active Bar in the PCB Editor

40. Return to the schematic document.

1.7.2 Classic Toolbars

In addition to the Active Bar, classic toolbars are also available.

41. To open a classic toolbar, go to **View » Toolbars** and select **Schematic Standard** to enable it.
42. Notice that a new toolbar will be added underneath the menu list as shown in Figure 19 below. To disable it, simply return to **View » Toolbars** and uncheck on **Schematic Standard**.

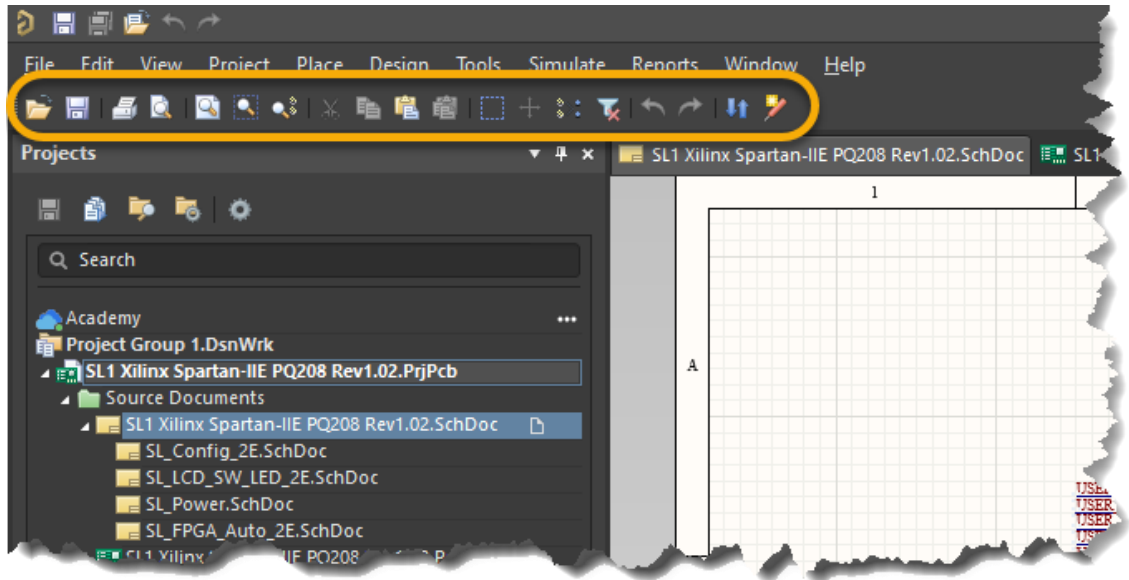


Figure 19. Classic Toolbar

1.8 Menu Operations

1.8.1 Right-Click Context Menu

43. Right-click in any free space in the schematic editor and observe the changes to the menu as shown on the left side of Figure 20 below.
44. In the SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc, left click to select any component, then right-click to observe the content in the right-click menu as shown on the right side of Figure 20.

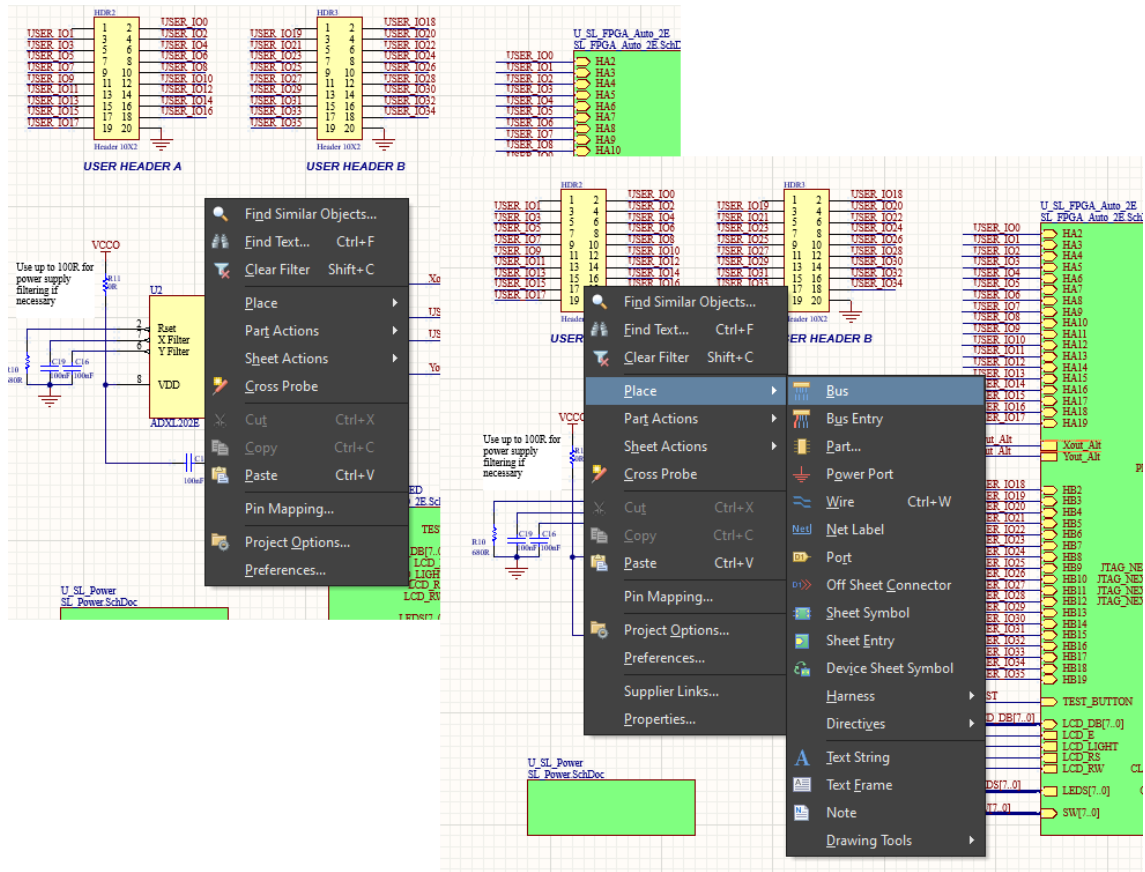


Figure 20. Right-click context menu

45. Feel free to explore the menus and options available while in the right-click menu.

1.8.2 Shortcut Keys

46. As mentioned earlier, some menus and menu entries have underlined letters, these can be used as keyboard shortcuts for quick access. For example, with the schematic in focus, access the **F**ile Menu directly by pressing the **F** key. The File menu opens at the location of your cursor.
47. Left click in a free space in the schematic to close this menu.

1.9 Status Bar and Command Status

48. From the **View** menu, locate the **Status Bar** and **Command Status** entries as shown in Figure 21 below.

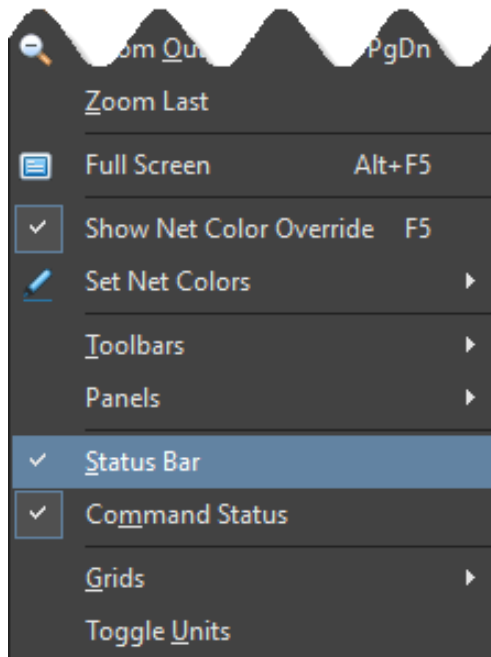


Figure 21. Activation status for the Status Bar and Command Status

49. Ensure that both entries are enabled by clicking on them until the check mark appears.
50. Move the cursor in the schematic editor area and you'll notice the X and Y values changing in the Status Bar, as shown in Figure 22 below.
51. Go to **Place » Wire**. The active command is displayed in the command status area shown in Figure 22 below. Hit **Escape** or right-click to exit from the command.

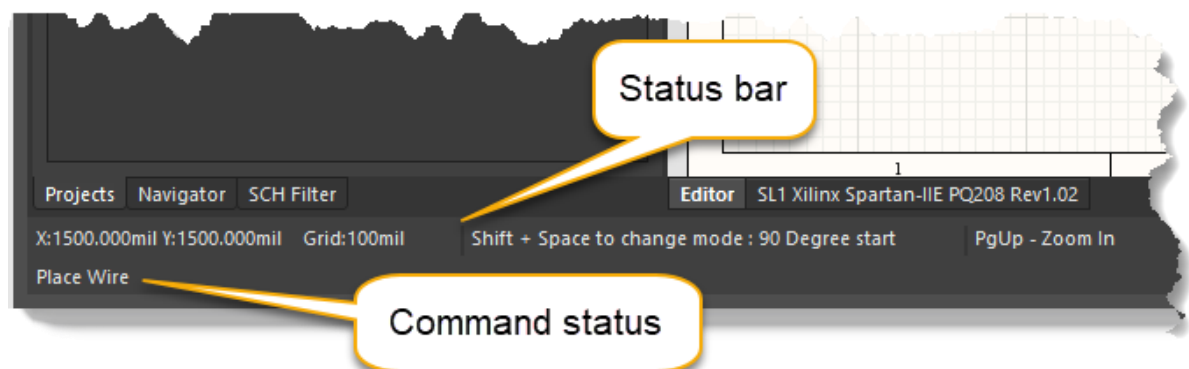


Figure 22. Status Bar and Command Bar information within the **Place » Wire** command

1.10 Undo and Redo

52. Select a component and hit **Delete** to delete a component in the schematic.
53. Click the Undo and Redo icons in the top left corner as shown in Figure 23 below.

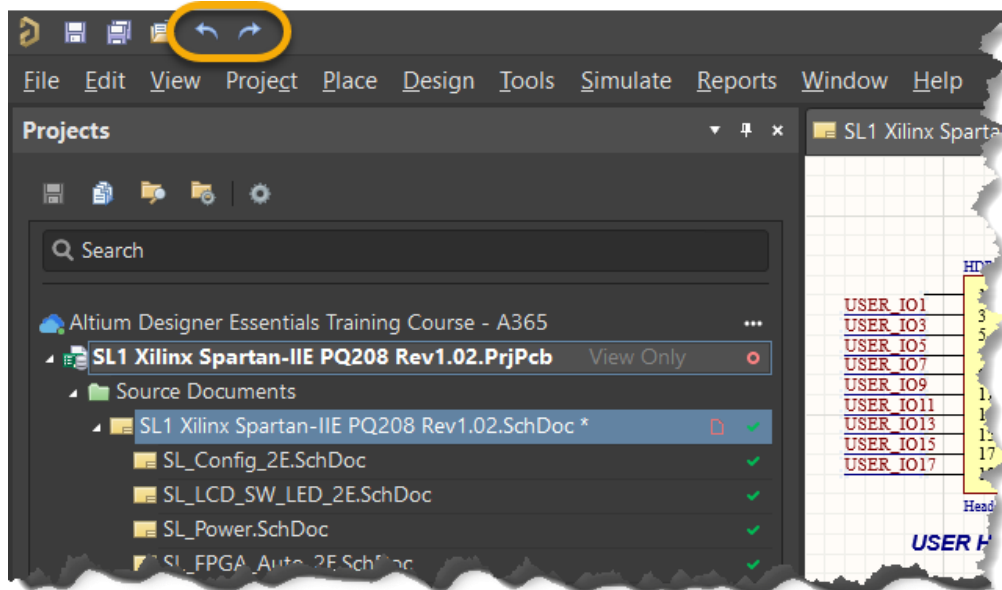


Figure 23. Undo and Redo icons

54. As an alternative, use the shortcut key **Ctrl+Z** or **Alt+Backspace** for Undo. For redo, the shortcut key is **Ctrl+Y** or **Ctrl+Backspace**.
55. Do not close the project, we will use this project in the next module.

Congratulations on completing the Module!

Module 4: Design Environment

from the

**Altium Designer Essential Course
with Altium 365**

Thank you for choosing Altium Designer
