Altium Designer

Essentials Training with Altium 365

Module 6: Navigating Schematics

Software, documentation and related materials:

Table of Contents

Module 6: Navigating Schematics

1 Purpose

This module will teach you some basic operations in the schematic editor such as preference settings, mouse actions, frequently used shortcut keys and search actions in the schematic environment.

2 Shortcuts

Shortcuts used when working with Module 6: Navigating Schematics

or T » P	Preferences
ALT key and left-click	Highlight Net
Shift+C	Clear Dimming
V » D	View » Fit Document
V » F	View » Fit All Objects
J » C	Jump to Component
J » L	Jump to Location
J » O	Jump to Origin
Hovering	Show Object information
Mouse Wheel	Page up - Page down
Mouse Wheel+Shift	Page left - Page right
Mouse Wheel+Ctrl	Zoom in - Zoom out
Page Up	Zoom in
Page Down	Zoom out

3 Preparation

1. If you have closed the project `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PrjPcb` from the last module, `Module 5 Schematic Preferences`, please reopen it. The project may also be available in the **File** menu, under **Recent Projects**.

4 Schematic Preferences Settings for Navigation

Schematic Preferences are system specific settings which help you control the behavior and display of the Schematic Editor to suit your working style and needs. Let's take a look at some basic preferences you may want to change for the Navigation.

4.1 Connectivity Insight

2. In the *Preferences* , within the *Design Insight* page, ensure that the checkbox for **Enable Connectivity Insight is** checked, as well as the *Document Tree* option for *Mouse Hover* as shown in Figure 1 below.

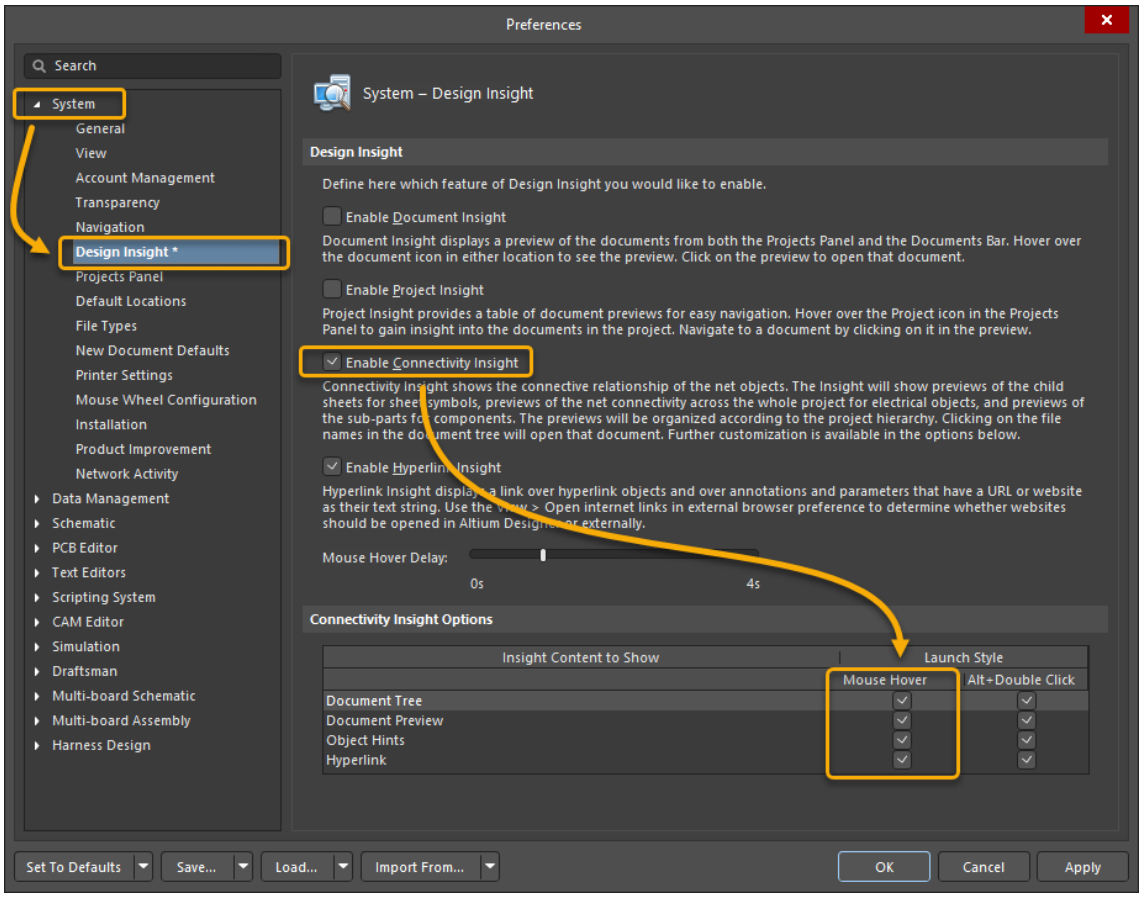

Figure 1. Preferences - Design Insight

3. Click **OK**.

4. In the *Projects* panel: **Right click** on the *Source Documents* and execute the command **Open all Schematic Documents,** Figure 2.

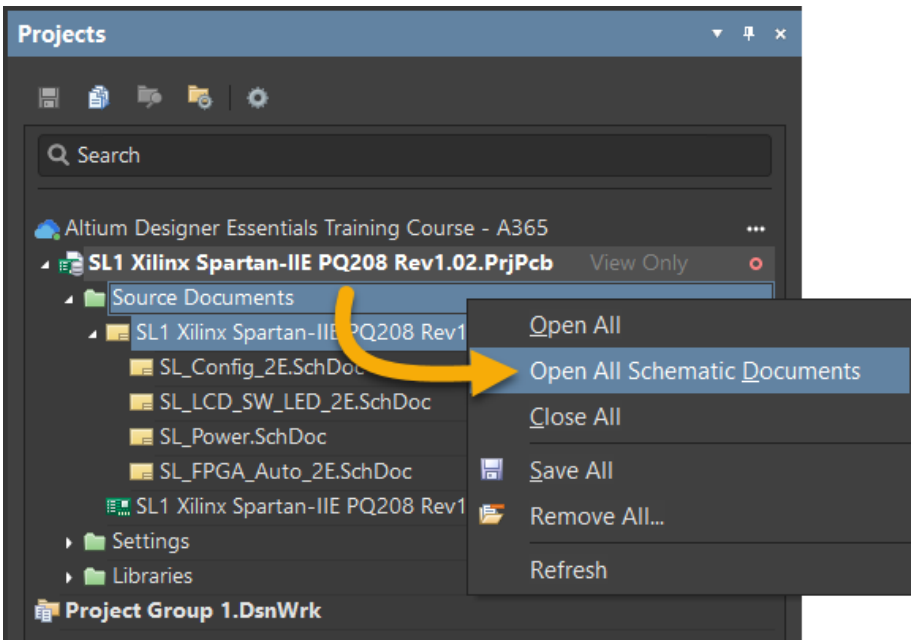

Figure 2. Open all Schematics

5. Make the `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc` the focused document.
6. Hover the mouse over the `U_SL_LCD_SW_LED` Sheet Symbol block as shown in Figure 3. A large thumbnail of the `SL_LCD_SW_LED_2E.SchDoc` will appear next to the cursor. If the thumbnail is not shown the first time, move the cursor outside of the Sheet Symbol and start hovering again.

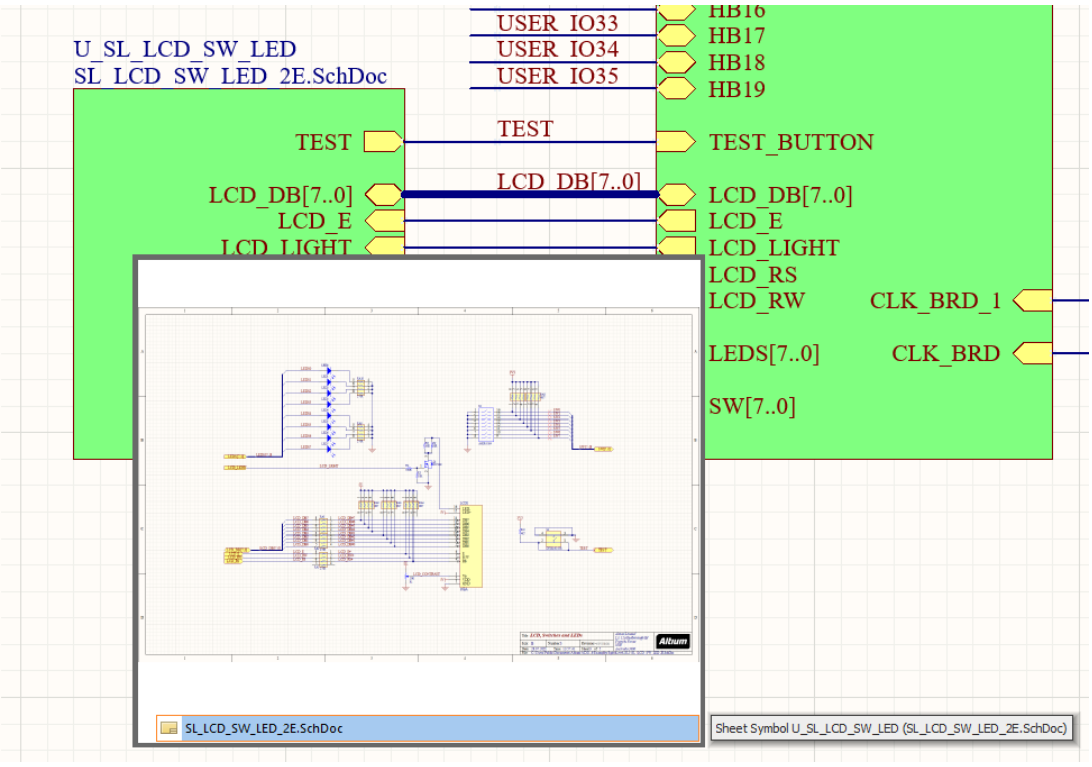

Figure 3. Hover over the Sheet Symbol to view the Schematic thumbnail

7. Next, hover the cursor over the bus `LEDS[7..0]` as shown in Figure 4 below.

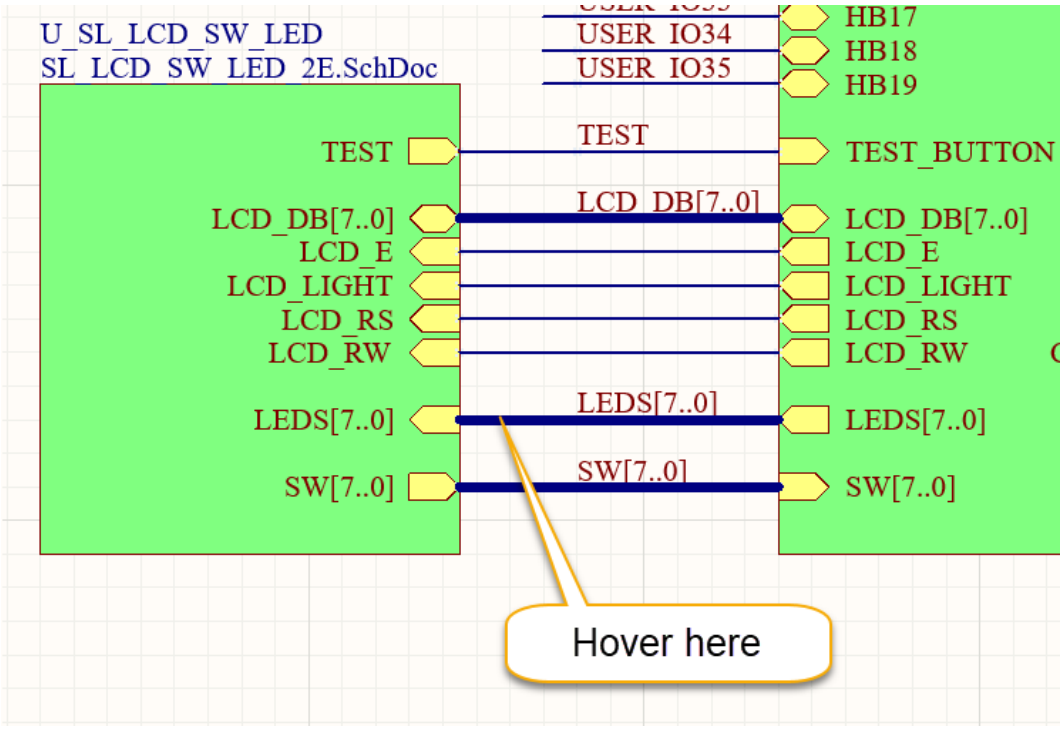

Figure 4. Hover over the Bus to view connectivity

8. A large thumbnail will appear next to the cursor, as shown in Figure 5 below, which shows the connections in the design.

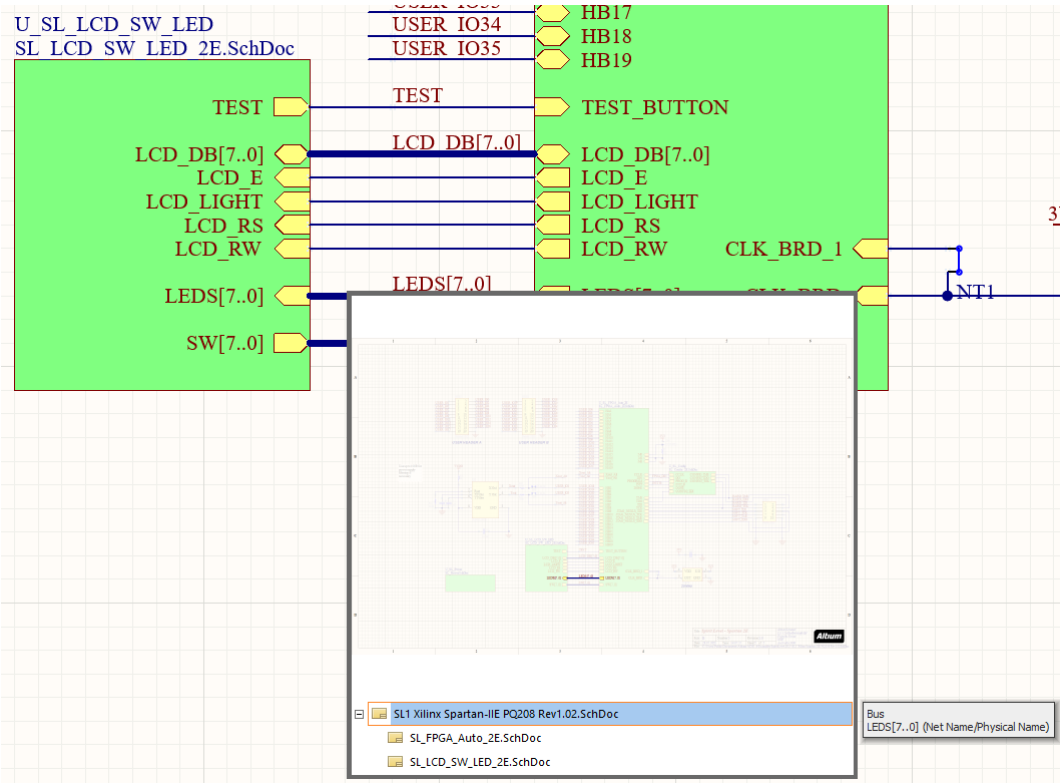

Figure 5. Connectivity Insight after hovering over a Bus

9. Slowly move your cursor down to `SL_LCD_SW_LED_2E.SchDoc` at the bottom of the thumbnail as shown in Figure 6 below. This will change the thumbnail view to the lower-level schematic.
10. Left-click `SL_LCD_SW_LED_2E.SchDoc` from the pop-up insight to open the schematic, Figure 6.

Hint: The schematic preview will be blank and only show a name and icon if the corresponding schematic is not currently opened.

11. Hit **Shift+C** or left click anywhere in the design to clear any selection or highlights or dimming.

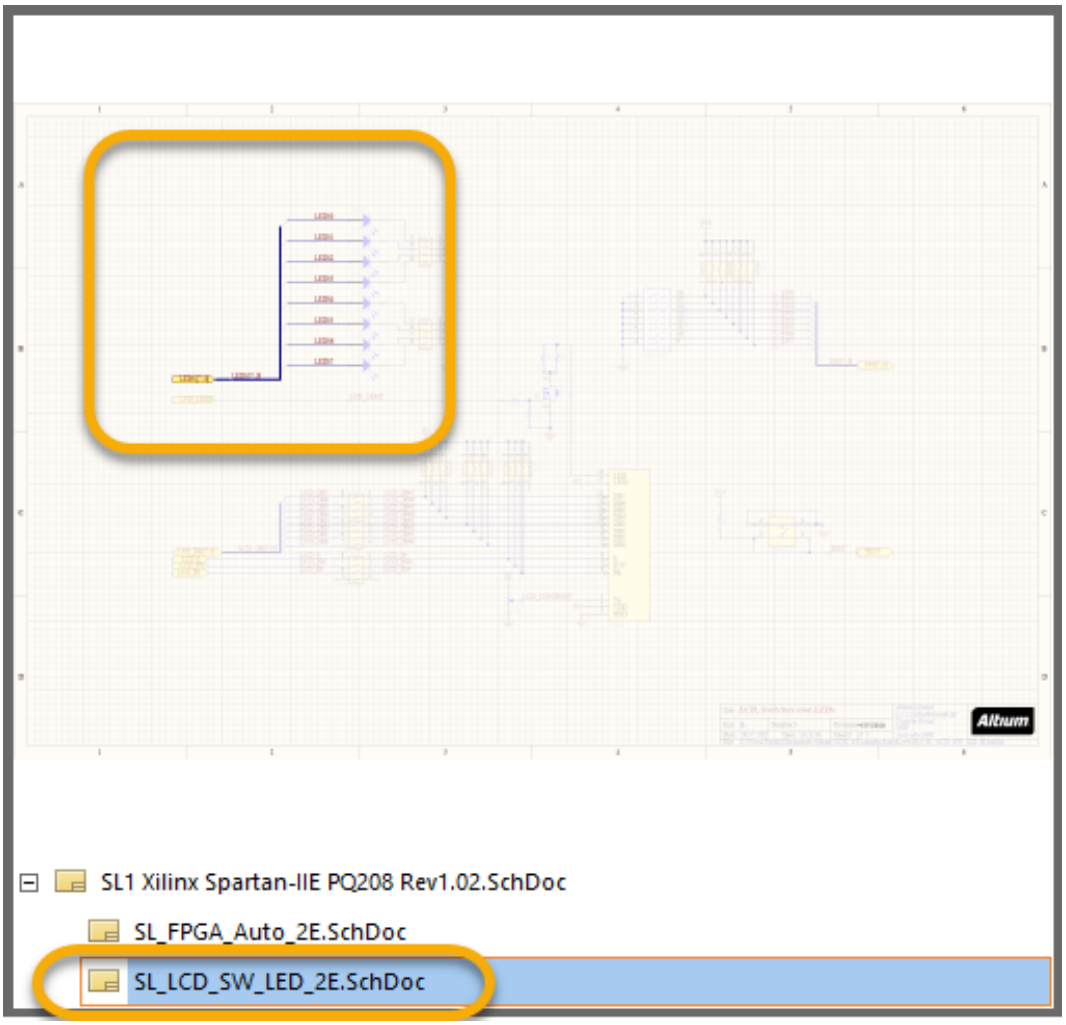

Figure 6. Connectivity Insight with SL_LCD_SW_LED_2E.SchDoc

12. Open the Preferences.
    a) Select the page *System - Design Insight*
    b) Click **Set to Defaults » Default (Page)** in the lower left corner of the *Design Insight* page as shown in Figure 7 below. This will reset all settings to their original default values.
    c) Click **OK** to exit the dialogue.

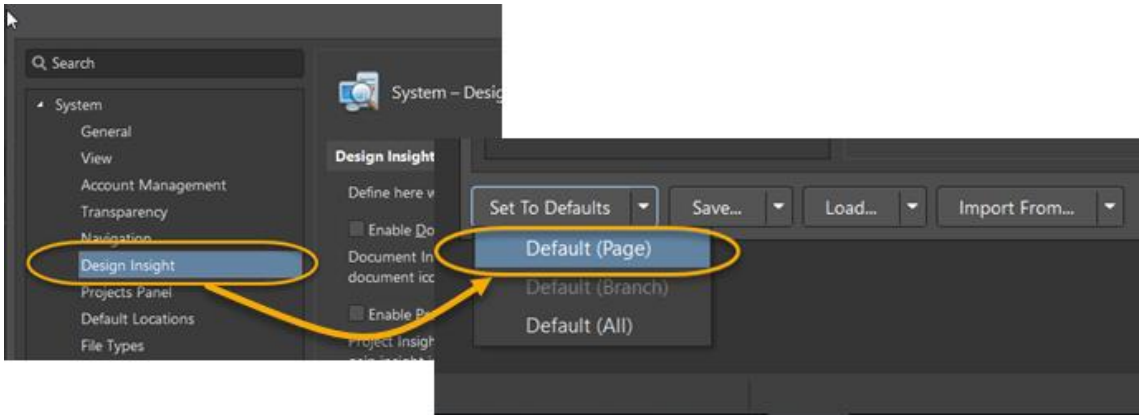

Figure 7. Set Design Insight Preferences back to Default

Hint: In the *Preferences*, within the **Navigation** page, in the section *Highlight Methods*, you will find a slider that allows you to control the dim level for the objects. The dim level is used for navigation, cross-probing and exploring differences. To clear any dimming, hit **Shift+C** to clear any selections or dimming, Figure 8.

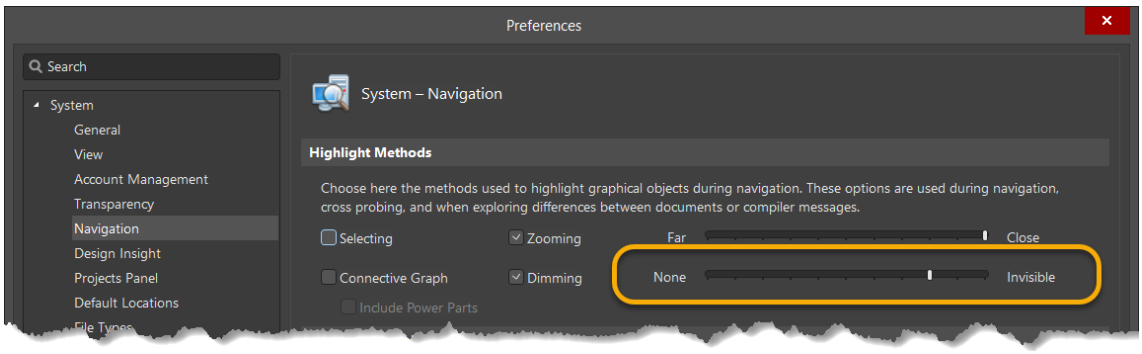

Figure 8. Dim Control

5 Schematic Navigator

We can use the *Navigator* panel to browse for nets, components, or pins across the entire design.

13. If not already opened, open the *Navigator* panel by clicking the **Panels** button Panels and select **Navigator** as shown in Figure 9 below.

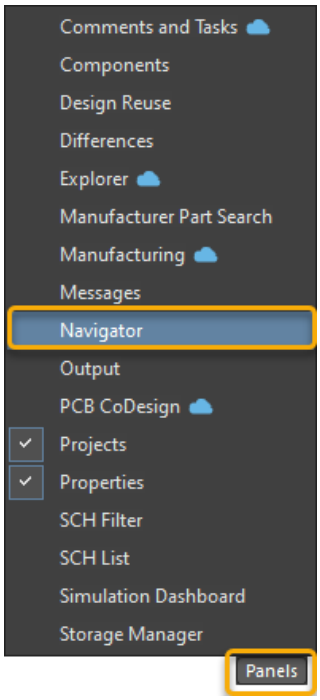

Figure 9. Opening the Navigator panel

14. The *Navigator* panel will open and be docked to the left by default. Use Figure 10 as a reference for the following steps.
15. Click on any document name in the *Navigator* panel to see nets and components just for that page. You can expand the arrow beside the document name to see the other sheets in the project.

Hint: If no components or nets are displayed inside the *Navigator* panel, validate the project by selecting **Project » Validate SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PrjPcb**.

16. Click on the **Flattened Hierarchy** to see the nets and components for all schematics in the project.
17. With the **Flattened Hierarchy** tab selected, search for component `R9` in the list of components to jump to it, as shown in Figure 10.

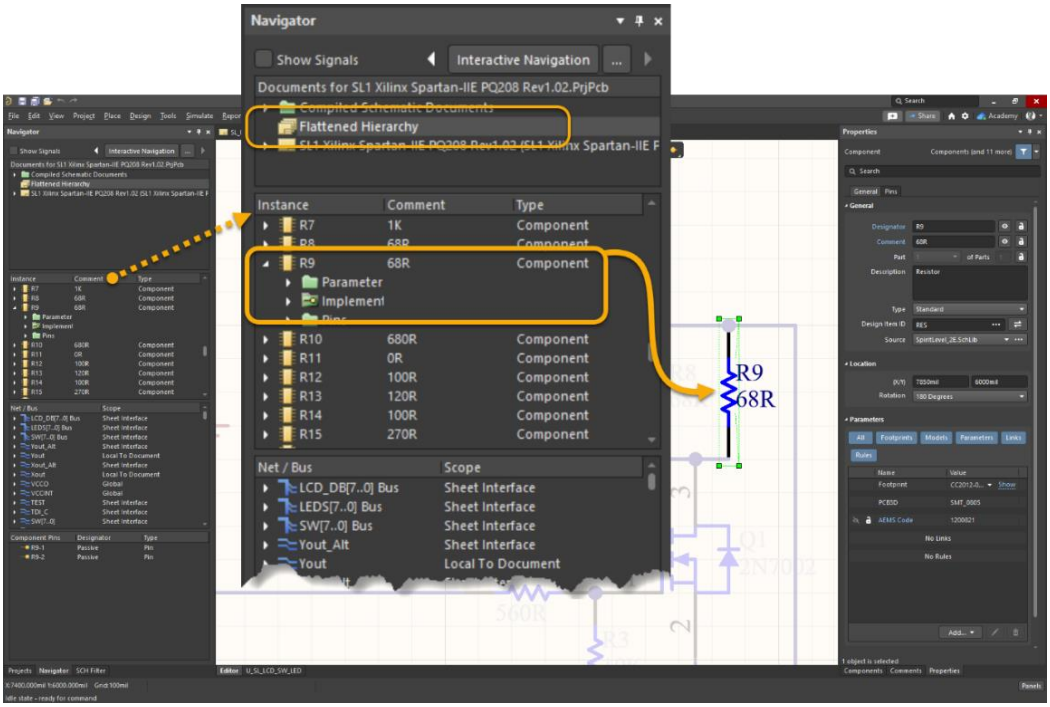

Figure 10. The Navigator Panel docked to the left

18. We can also highlight nets globally, across the entire design. Zoom out from `R9` and left-click anywhere in the schematic to clear the highlight.
19. While holding the **ALT** key, left-click the GND port below of component `R9`.
20. You will see that every GND port will be highlighted as shown in Figure 11 below. They will also be highlighted across every schematic sheet.

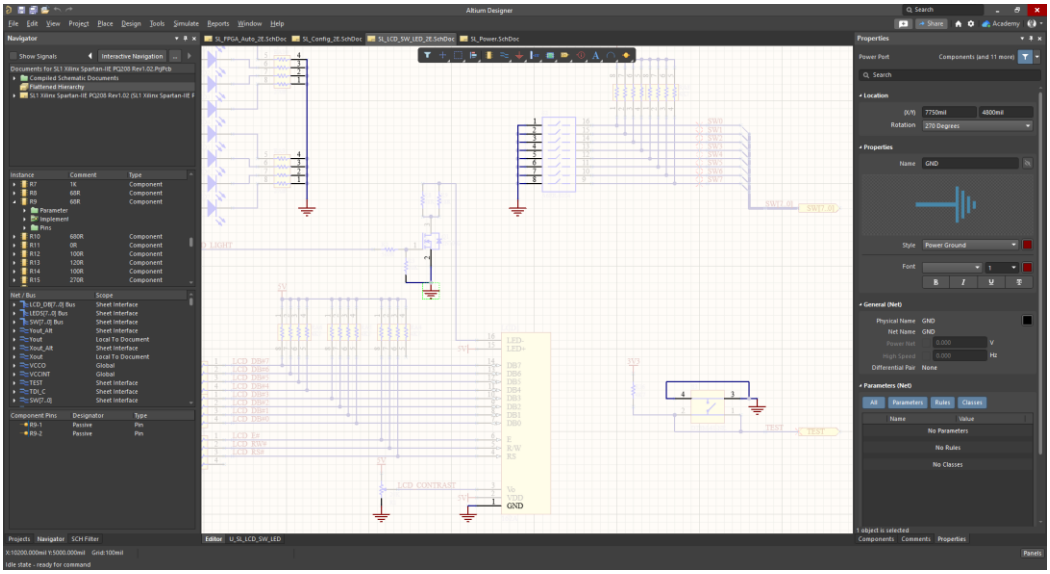

Figure 11. Global Net Highlighting

21. Hit **Shift+C** or left click anywhere in the design to clear any selection or highlights or dimming.

6 Port Cross Reference

Now we will add Port Cross Reference information. This adds a note at the end of each port, telling you the sheet and zone the port is connected to. Notice the Power Supply.SchDoc has no port cross-references because it has no ports – only power ports.

22. Open all schematics in the project from the *Projects* panel.
23. Open the Project options, **Project » Project Options…** , Figure 12.
    a) In the Tab *Options* activate **Automatic Cross References**.
    b) Use the default configuration *Sheet Style*: **Name** and *Location Style*: **Zone**
    c) Set *Display Cross Reference for* to **Sheet Entry & Ports**
    d) Select **OK** to close the *Project Options*.

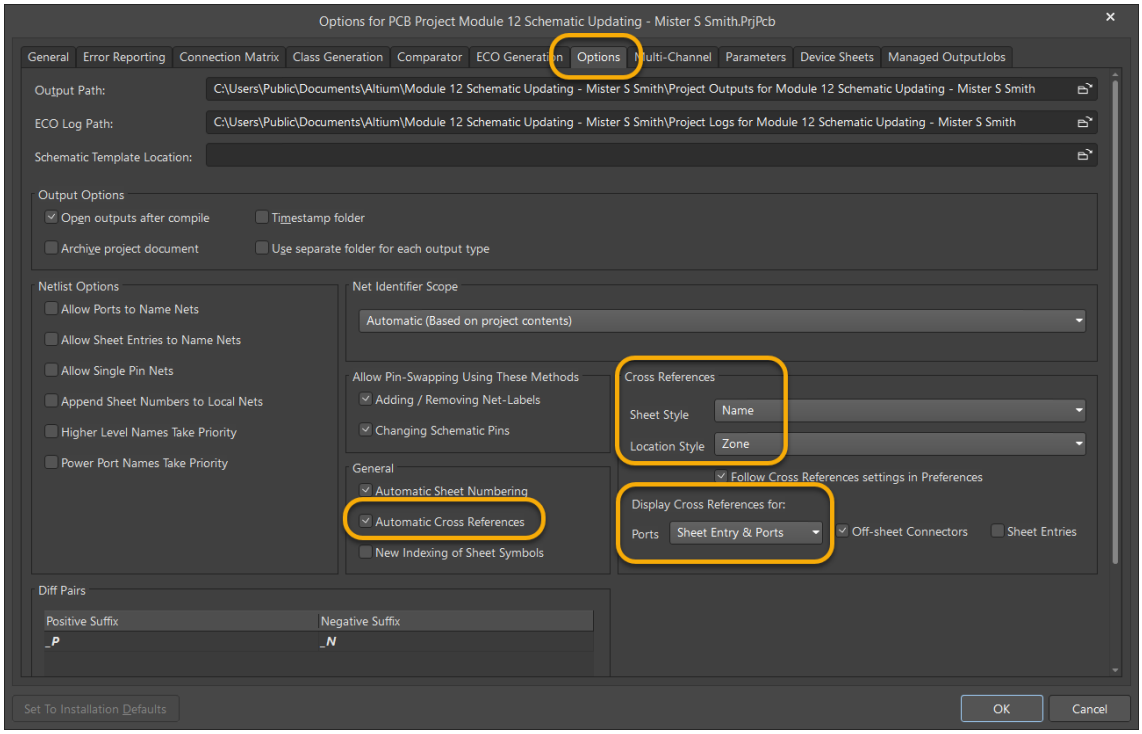

Figure 12. Port Cross Reference settings

24. View each schematic sheet to see where the Port Cross References have been added.

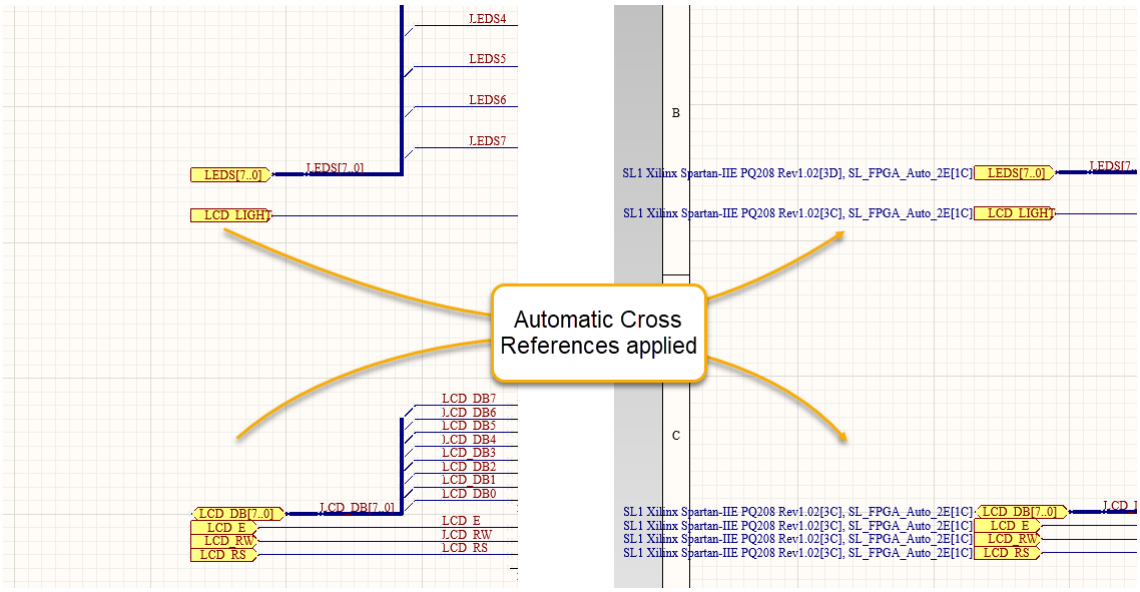

Figure 13. Example Schematic with Ports and Port Cross Reference information

25. Feel free to change the setting for *Display Cross reference for: Ports* to something different than **Sheet Entry and Ports**.

7 Jump Menu

26. Maintain the focus on the Schematic and observe the behavior of the following jump commands. The **Edit » Jump** menu can be easily accessed using the **J** shortcut key.
27. To jump to origin, use **Edit » Jump » Origin** or press the **J » O** keys in succession.
28. To jump to location, use **Edit » Jump » New Location** or press the **J » L** keys in succession. Then enter the X, Y coordinates in the *Jump to Location* dialogue as shown in Figure 14 below.
29. Click **OK** to move the cursor to the defined coordinate location. The view will then zoom and pan to the target location.

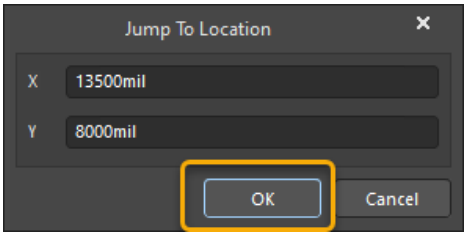

Figure 14. Jump to Location (Shortcut Keys J » L)

30. Maintain the focus on the schematic `SL_LCD_SW_LED_2E.SchDoc`.
    a) To jump to a component, use **Edit » Jump » Jump Component** or press the **J » C** keys in succession.
    b) Enter `U1` into the *Component Designator* dialogue.
    c) Press **OK** and the cursor will jump to the component `U1` on page `SL_FPGA_Auto_2E.SchDoc`. In this example `U1` is a component with three parts. You have the option to jump to the next parts of `U1` when the *Find Text – Jump* dialogue opens.
    d) Click **Close** for the *Find Text – Jump* dialogue, then close the *Messages* panel.

8 Search Field

31. Make the `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc` schematic the focused document by left-clicking on its document tab or opening it from the *Projects* panel.
32. Locate the Search area at the top right corner, as shown in Figure 15 below. Next, let's search for component R10. Type `r10` (not case sensitive) in the search area, select it from the list or hit **Enter** to jump to the component, this search can also be applied to searching for components in the PCB.

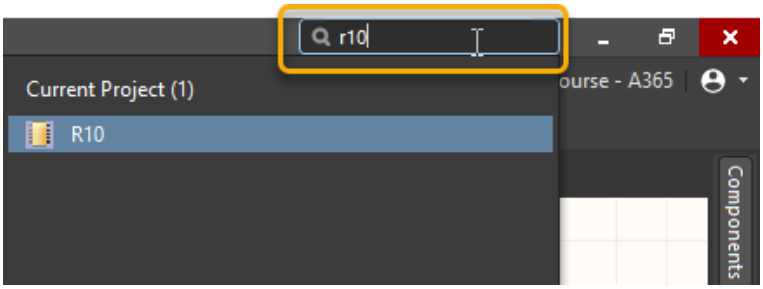

Figure 15. Component Search

Please note that the Search area shown in Figure 15 can also be used for searching features, functionality, component values, and many more things. Explore what else you can search for, e.g., resistance value 4K7.

Another very useful area for searching for components or net connections is the Project Panel, see Figure 16.

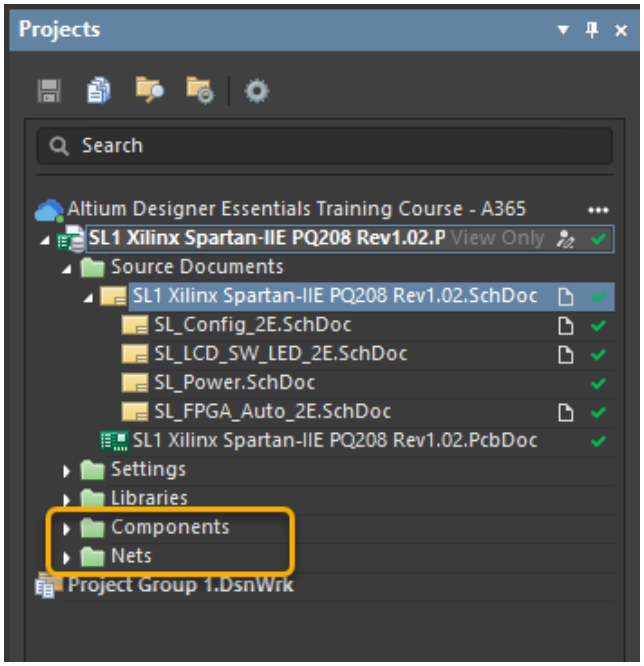

Figure 16. Search using Project Panel

In most cases you will notice some temporary component and nets folders in the Projects panel, if you expand these, you can lump and locate components and nets in the project. Try exploring these folders by expanding them and double clicking on a component or net.

33. Close the project and all files with **Window » Close**.
34. **If you save the modifications, they are just locally saved but not uploaded to the workspace. An upload isn't possible as the user rights are set to viewing only**.

Congratulations on completing the Module!

Module 6: Navigating Schematics

from

Altium Designer Essentials Training with Altium 365

Thank you for choosing **Altium Designer**