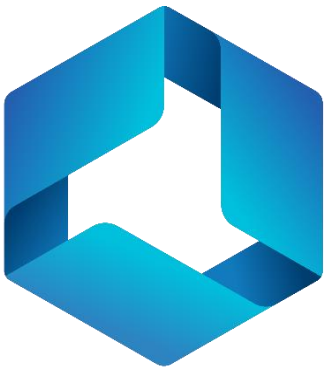


ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 10: Creating Hierarchy

Table of Contents

Module 10: Creating Hierarchy

1.1 Purpose

In this exercise, the basics of a hierarchical design will be explored. Creating a Hierarchical design allows you to control connectivity from sheet to sheet in a multi-sheet design.

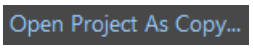
The top-level schematic will be completed by populating it with *Sheet Symbols* corresponding to each of the mid / lower-level schematics.

1.2 Shortcuts

Shortcuts when working with Module 10: Creating Hierarchy

P » S:	Place Sheet Symbol
P » E:	Place Sheet Entry
Ctrl+C:	Copy
Ctrl+V:	Paste
Ctrl+Shift+V:	Smart Paste
G:	Grid (3 Cycle Steps)
D » Y:	Create Sheet from Sheet Symbol
D » R:	Create Sheet Symbol from Sheet
Insert:	Hover over a Name Object - Copy Name to Mouse Object
P » N	Place Netlabel
P » B	Place Bus
P » W	Place Wire

1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project `Module 10 Creating Hierarchy`
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project `Module 10 Creating Hierarchy (Top\Projects\Altium Designer Essentials Training Course\...)`
5. Select **Open Project as Copy…** 

6. At the new dialog, *Create Project Copy*
   a) Add your name to the project: `Module 10 Creating Hierarchy - [Your Name].`
   b) Add a description: `Altium Essential Training - Module 10 - [Your Name]`
   c) Open the Advanced section.
   d) Select the Ellipsis Button  from the Folder configuration to open the *Choose Folder* dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your Name]`
      ii) Select **OK**
   e) Change the Local Storage path if needed.
   f) Select **OK** to create the copy.
7. Wait until Altium Designer creates the copy of the project and opens the project in the *Projects* panel; this can take up to 1 minute.

1.4 Create Top-Level Schematic

8. Right-Click on `Module 10 Creating Hierarchy - [Your Name]` in the *Projects* panel and select **Add New to Project » Schematic.**
9. Save the new schematic using **File » Save.** Name the file `Top_Level.SchDoc` when prompted.

1.5 Sheet Symbols

Additionally to the bottom-up method that we describe at this training, Altium Designer supports the top-down method.
For details see the Altium Documentation: Multi-sheet & Hierarchical Designs in Altium Designer.

1.5.1 Creating a Sheet Symbol from Existing Sheet – Processor Interface

10. With the `Top_Level.SchDoc` as active sheet.
11. Right-click anywhere on the schematic and select **Sheet Actions » Create Sheet Symbol from Sheet**.
12. Select the `Processor_Power.SchDoc` from the dialog and select **OK** as shown in Figure 1 below.

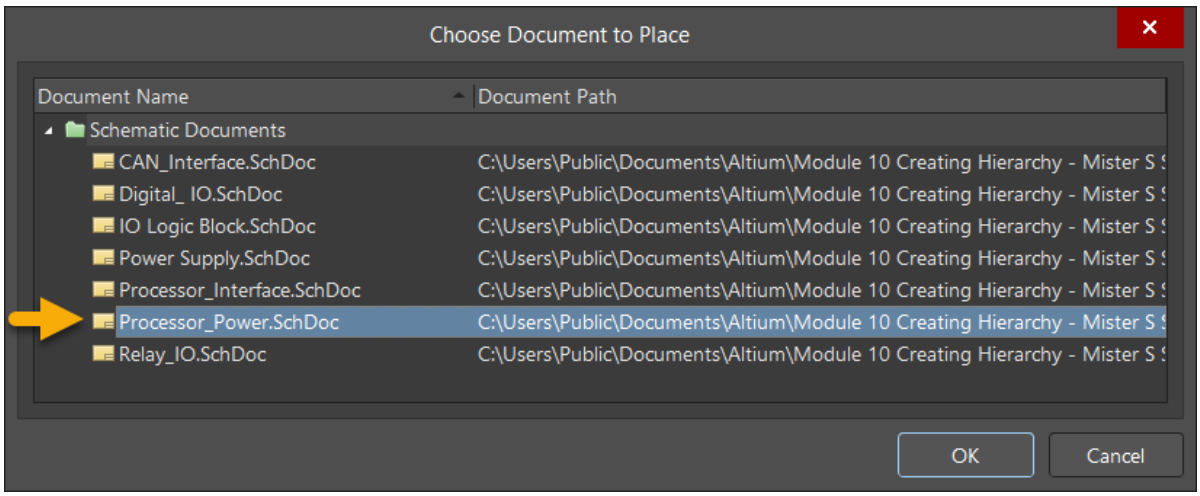


Figure 1. Choose Document to Place as a Sheet Symbol

13. With the Sheet Symbol still attached to your cursor, press the **TAB** key to bring up its properties.
    a) Enter `MID1` in the *Designator* properties and press **Enter**.
14. Place the Sheet Symbol left from the center of you schematic, e.g. x:6000mil, y:8000mil.

15. With the Sheet Symbol selected, resize it from the *Properties* panel.
    a) In the *Properties* panel, change the *Width* value to `1500mil` and the *Height* value to `3500mil` as shown in Figure 2 and hit **Enter**.

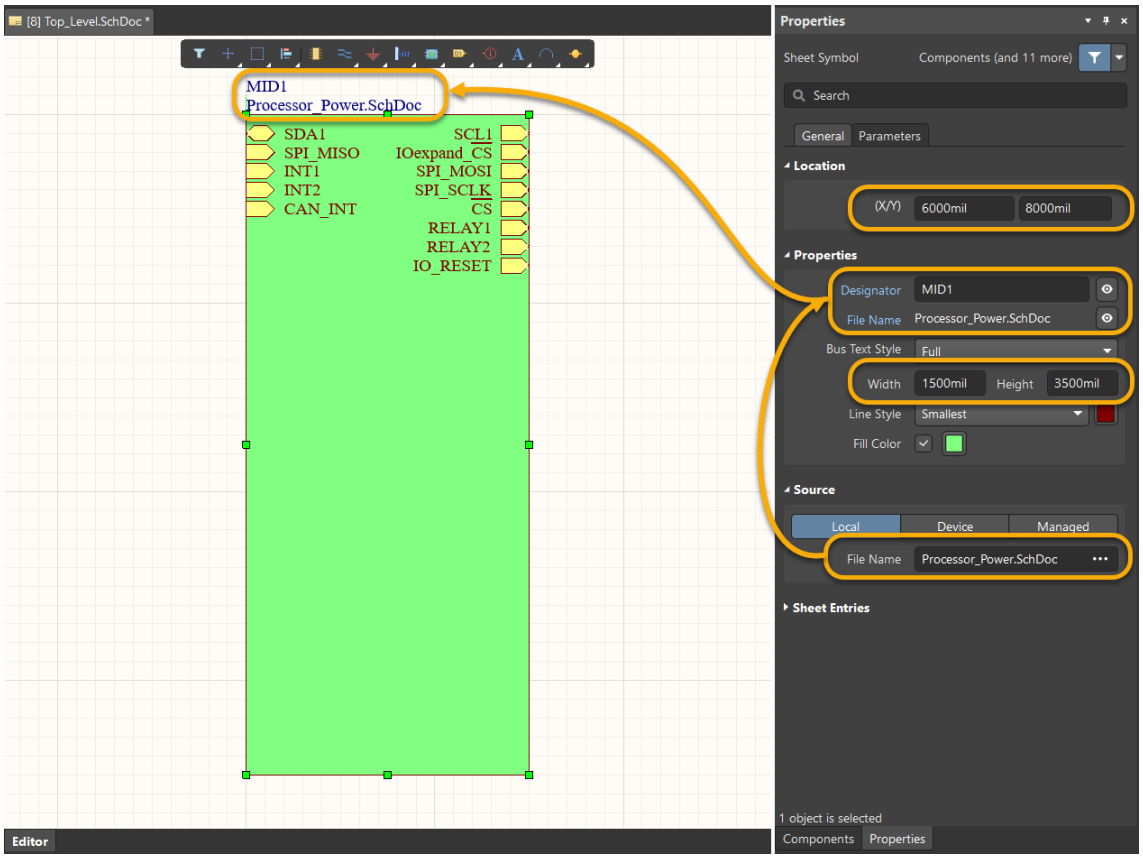


Figure 2. Resize Sheet symbol

16. Select and reposition the sheet entries using Figure 3 as reference.

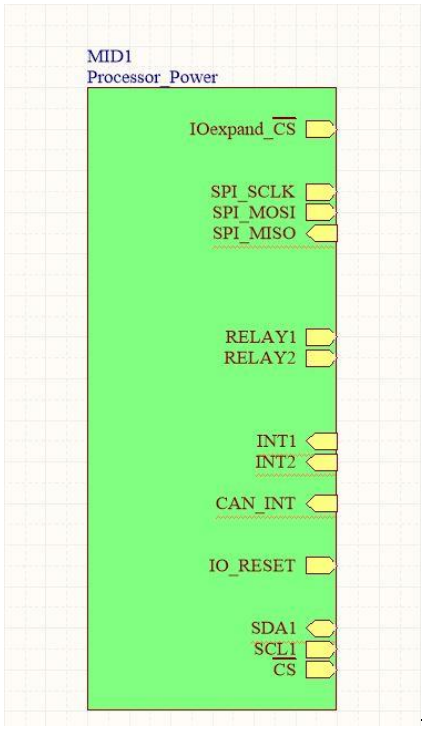


Figure 3. Sheet Symbol Alignment

1.5.2 Creating a Sheet Symbol from Existing Sheet – IO Logic Block

17. Right-click anywhere on the schematic and select **Sheet Actions » Create Sheet Symbol from Sheet**.
18. Select the `IO Logic Block.SchDoc` from the dialog and select **OK,** Figure 4 below.

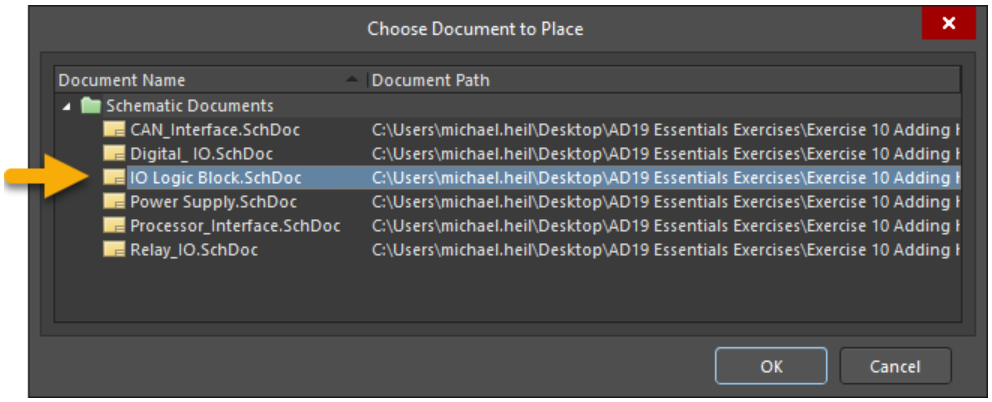


Figure 4. Choose Document to Place as a Sheet Symbol

19. With the Sheet Symbol still attached to your cursor:
    a) Press the **TAB** key to bring up its properties
    b) Enter `MID2` in the *Designator* properties and press **Enter**.
20. Place the Sheet Symbol to the right of the `MID1` Sheet Symbol.
21. Resize the sheet symbol to the same size as the sheet symbol `MID1(1500x3500)`.
22. Align the Sheet Entries from `MID2` to the sheet entries in Sheet Symbol `MID1`, Figure 5. Ensure to offset the `Relay[1..2]` sheet entry on `MID2` as shown in Figure 5 as we will create a Bus connection for this shortly.

Activate the large 90° cursor (Preferences, section Schematic – Graphical Editing). With an active Large Curser 90, it is easier to position the entries.

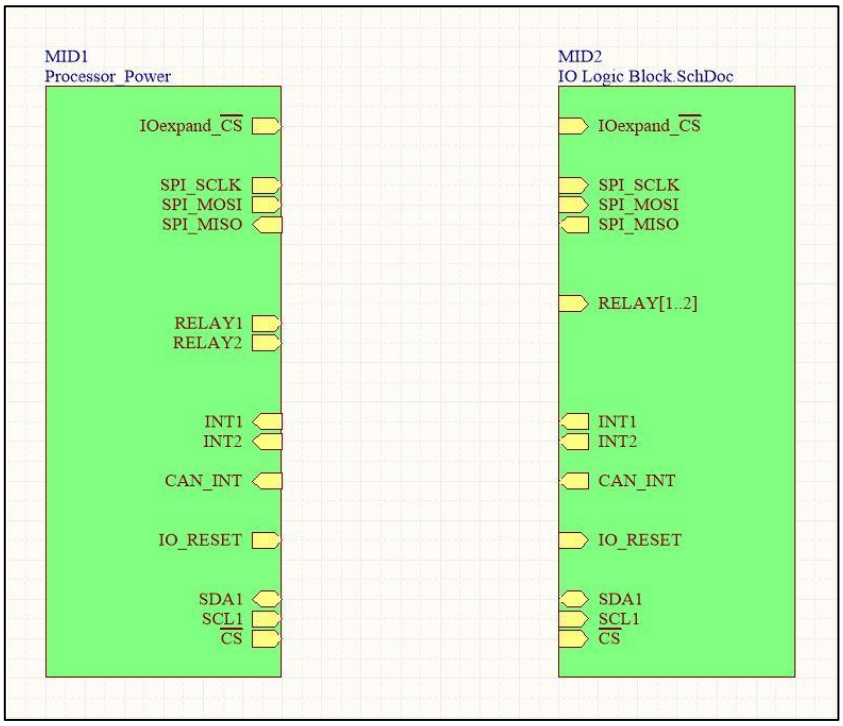


Figure 5. Sheet Symbol Alignment

1.6 Connecting the Sheet Symbols

Now that the sheet symbols have been placed, you will wire the connections between the two sheet symbols.

1.6.1 Wiring the Sheet Symbols

23. Move the `Processor_Power` sheet symbol directly in contact with the `IO Logic Block` sheet symbol so that the sheet entries line up beside each other. You will notice a small green checkmark that will appear.
24. Drag the `Processor_Power` sheet symbol away from the other sheet symbol, back to its original position. Wires should appear connecting the two blocks (this is dependent on your preferences; you may need to try again while holding down the **Ctrl** key).

1.6.2 Wiring the Relay Bus

25. From the **Place** menu, select **Place » Bus**.
    a) Start the Bus at the `RELAY[1..2]` sheet entry on the `MID2` sheet symbol and extend half way to the `MID1` sheet symbol.
    b) Proceed downwards so that there is sufficient amount of bus wire available for wires to extend to the `RELAY1` and `RELAY2` sheet entries of the `MID1` sheet symbol, as shown in Figure 6, and left-click.
    c) Right-click twice to exit the Bus command.
26. Wire the `RELAY1` and `RELAY2` sheet entries of the `MID1` sheet symbol to the bus using **Place » Wire**, as shown in Figure 6 below. Right-click to end the command when you are finished.

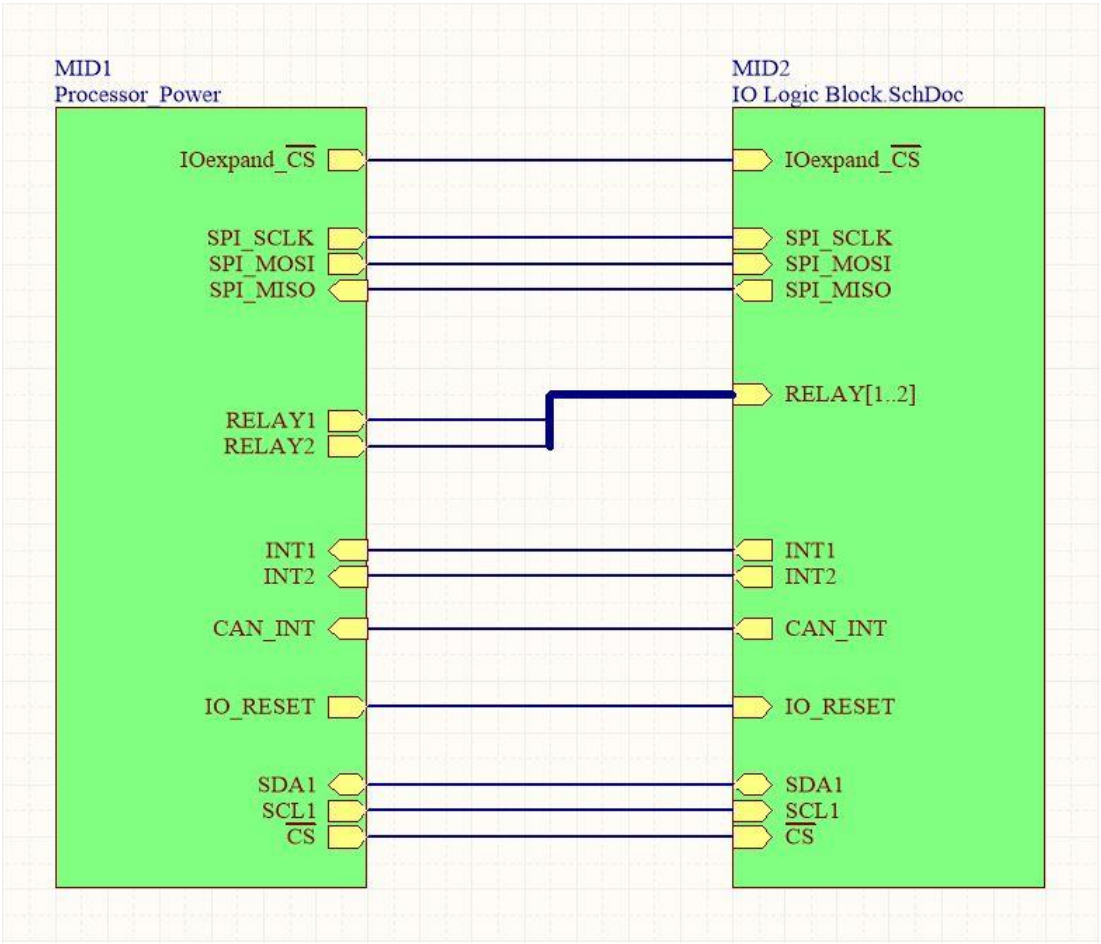


Figure 6. Wired Sheet Symbols

1.6.3 **Labeling the Bus and Bus Connections**

27. Label the Bus and Bus connections using the **Place » Net Label** command.
    a) With the net label attached to your cursor, hover over the `RELAY[1..2]` sheet entry and press **Insert** key. The new net label inherits the `RELAY[1..2]` name as shown in Figure 7. Alternatively, you can press **Tab** and manually change the value.

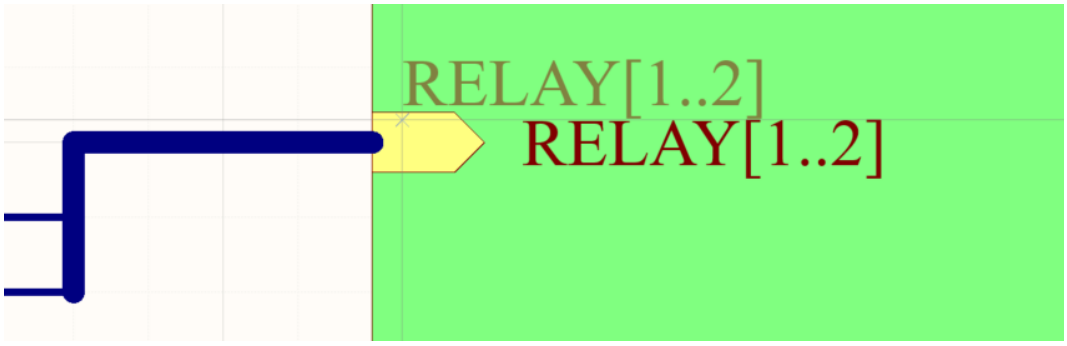


Figure 7. Net change using the Insert key

    b) Left-click to place the `RELAY[1..2]` net label on the Bus as shown in Figure 8 below.
    c) With a new Net Label still attached your cursor, hover your cursor over the `RELAY1` Sheet Entry in the `MID1` Sheet Symbol.
    d) Press **Insert** so that the Net Label picks up the `RELAY1` Sheet Entry name.
    e) Left-click to place the Net Label on the extended wire from the `RELAY1` sheet entry. (After placing the net label, the net label on the cursor automatically increments to `RELAY2`).
    f) Left-click to place the `RELAY2` Net Label on the wire extending from the `RELAY2` Sheet Entry. Use Figure 8 as a reference.
    g) Right-click to exit the command.

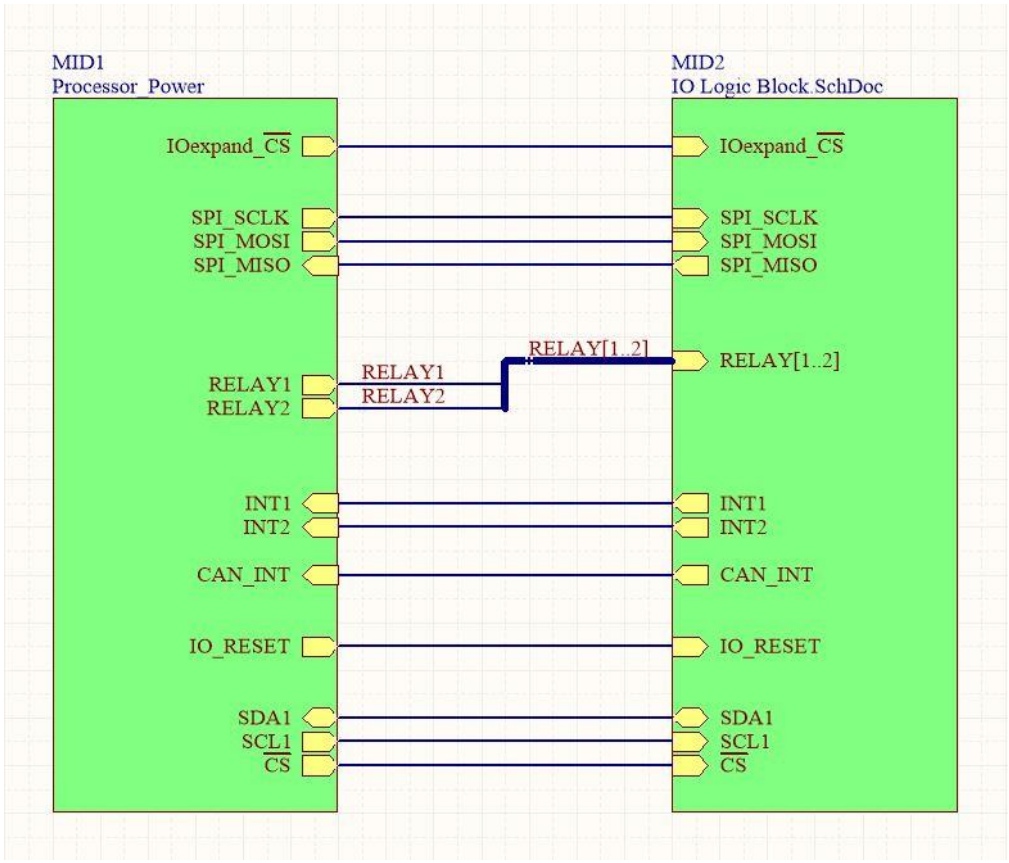


Figure 8. Net Labeled Bus and Bus Connections

28. After completion the project files will show the hierarchical connectivity in the project panel.
29. If the Schematics are not the first documents shown at the *Projects* panel, section Source documents, select and move the `Top_Level.SCHDoc` to the first position, See *Figure* 9 for reference.

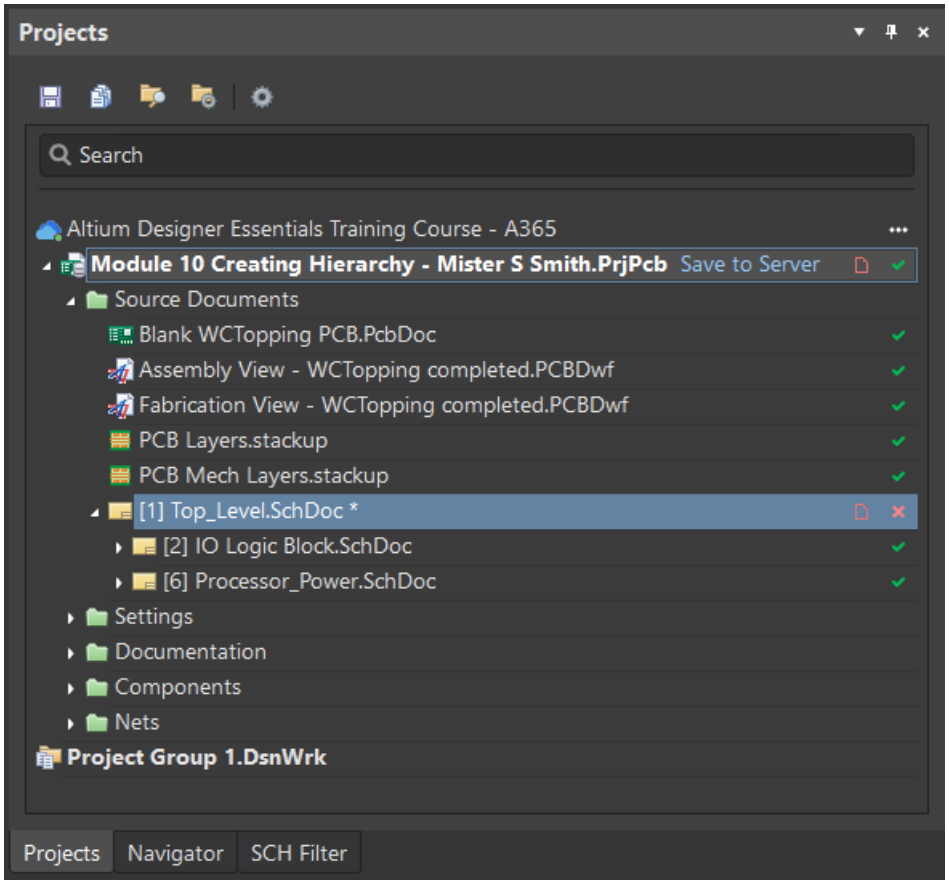


Figure 9. Hierarchical Design Project Structure

30. Select **File » Save All** to save all modifications.
31. 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) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Module 10: Creating Hierarchy - [Add Your Name] - Finished`
        iii) Select **OK.**
32. **When ready, close the project and any open documents, Window » Close All**

Congratulations on completing the Module!

Module 10: Creating Hierarchy

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer