



Altium Designer

Essentials Training with Altium 365

Module 10: Creating Hierarchy

Altium
TRAINING





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.

TRADEMARKS

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 10: Creating Hierarchy	3
1 Purpose	3
2 Shortcuts	3
3 Preparation	4
4 Create Top-Level Schematic	5
5 Sheet Symbols	5
5.1 Creating a Sheet Symbol from Existing Sheet – Processor Interface	5
5.2 Creating a Sheet Symbol from Existing Sheet – IO Logic Block	7
6 Connecting the Sheet Symbols	8
6.1 Wiring the Sheet Symbols	8
6.2 Wiring the Relay Bus	8
6.3 Labeling the Bus and Bus Connections	9





Module 10: Creating Hierarchy

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.

2 Shortcuts


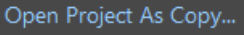

Shortcuts used when working with Module 10: Creating Hierarchy

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





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. Enable the folder view button .
5. Navigate to the predefined Training Project `Module 10 Creating Hierarchy`
(`Top\Projects\Altium Designer Essentials Training Course\...`)
6. Select **Open Project as Copy...** 
7. 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.
8. 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.





4 Create Top-Level Schematic

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

5 Sheet Symbols

Hint: 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.

5.1 Creating a Sheet Symbol from Existing Sheet – Processor Interface

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

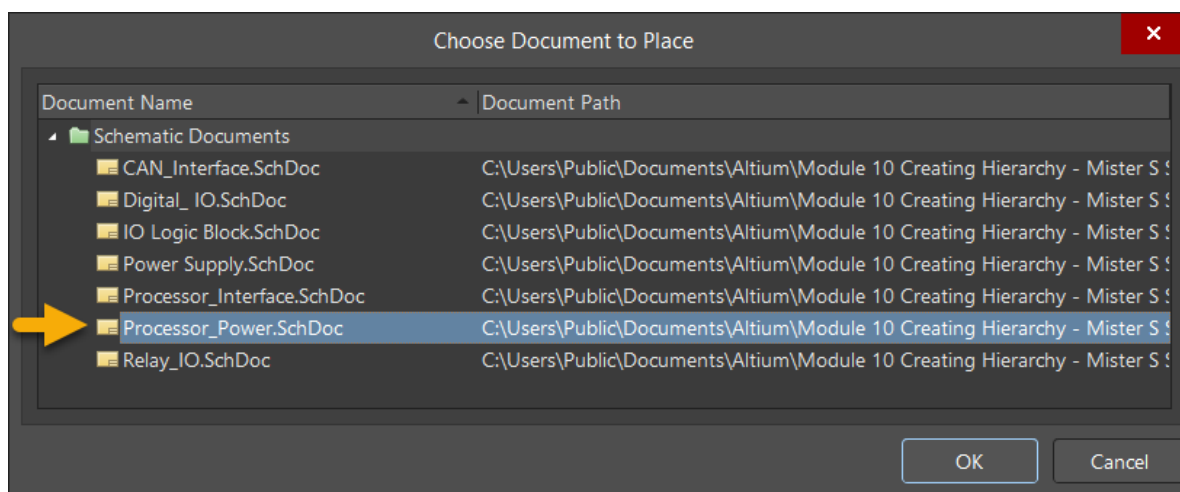


Figure 1. Choose Document to Place as a Sheet Symbol

14. 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**.
15. Place the Sheet Symbol left from the center of you schematic, for example, x:6000mil, y:8000mil.





16. 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**.

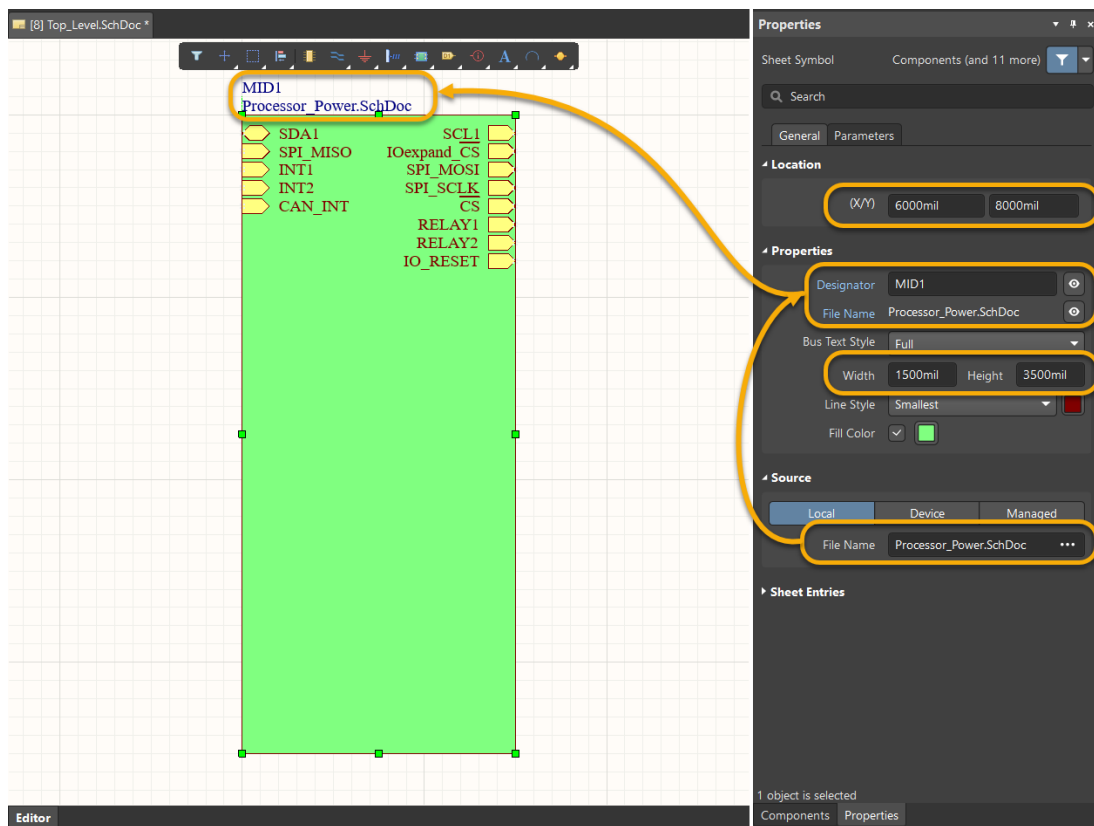


Figure 2. Resize Sheet symbol

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

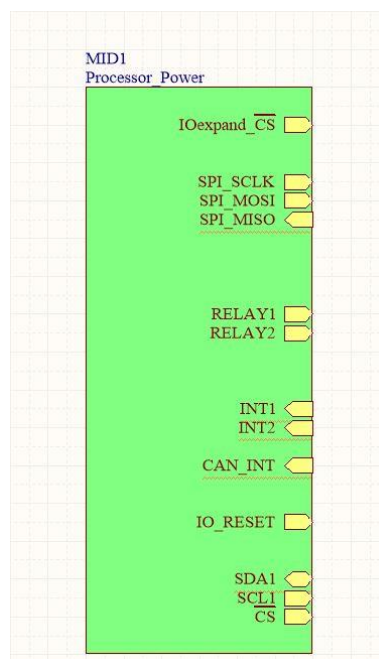


Figure 3. Sheet Symbol Alignment





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

18. Right-click anywhere on the schematic and select **Sheet Actions » Create Sheet Symbol from Sheet**.

19. Select the IO Logic Block.SchDoc from the dialog and select **OK**, Figure 4 below.

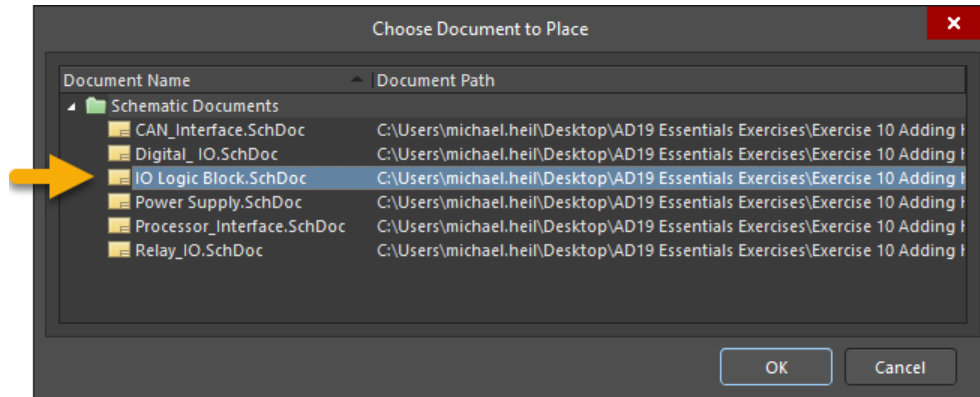


Figure 4. Choose Document to Place as a Sheet Symbol

20. With the Sheet Symbol still attached to your cursor:

- Press the **TAB** key to bring up its properties
- Enter **MID2** in the *Designator* properties and press **Enter**.

21. Place the Sheet Symbol to the right of the MID1 Sheet Symbol.

22. Resize the sheet symbol to the same size as the sheet symbol MID1 (1500x3500).

23. 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.

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

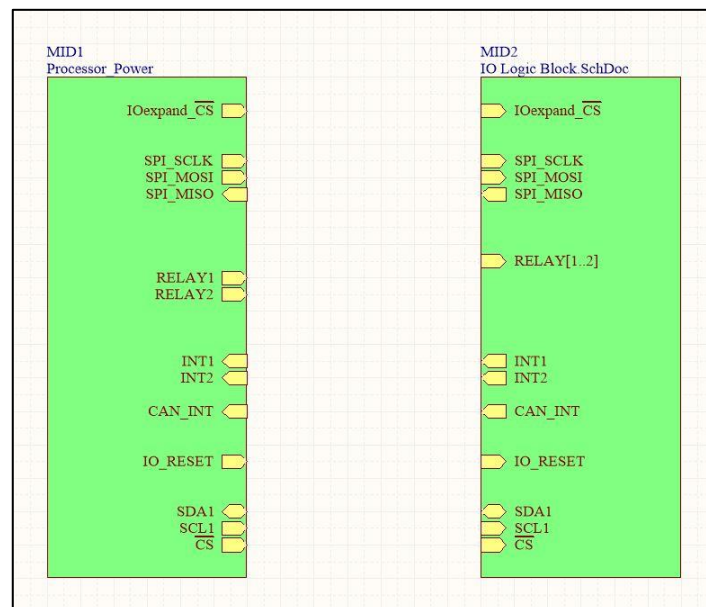


Figure 5. Sheet Symbol Alignment





6 Connecting the Sheet Symbols

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

6.1 Wiring the Sheet Symbols

24. 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.
25. 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).

6.2 Wiring the Relay Bus

26. 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.
27. 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.

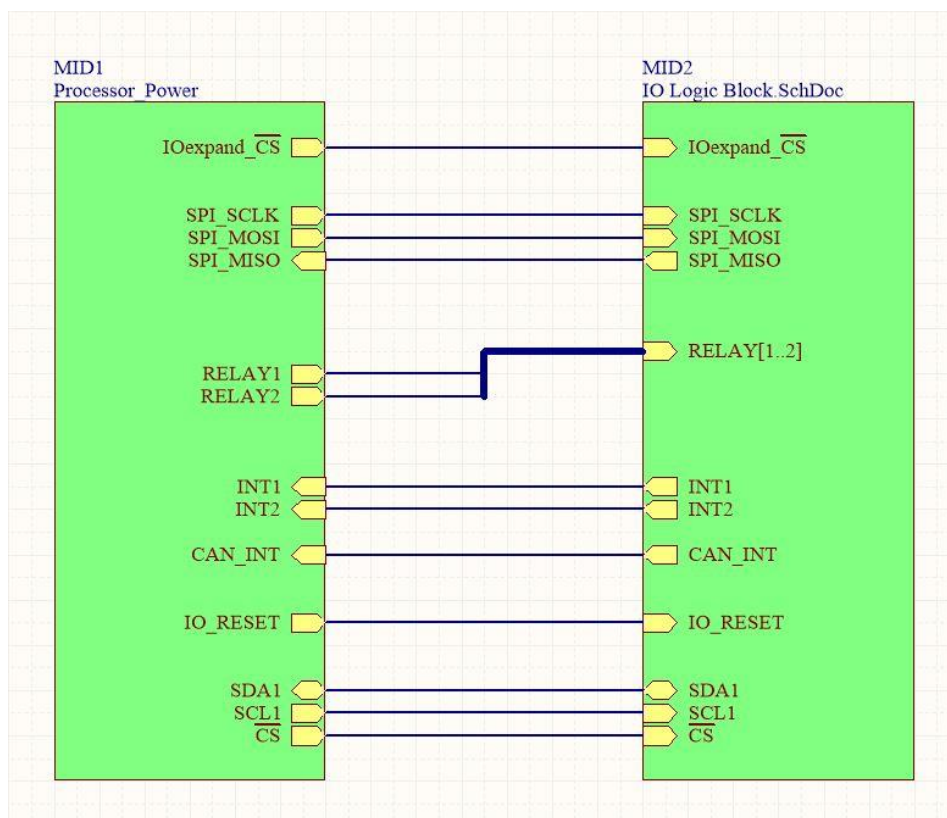


Figure 6. Wired Sheet Symbols





6.3 Labeling the Bus and Bus Connections

28. 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.

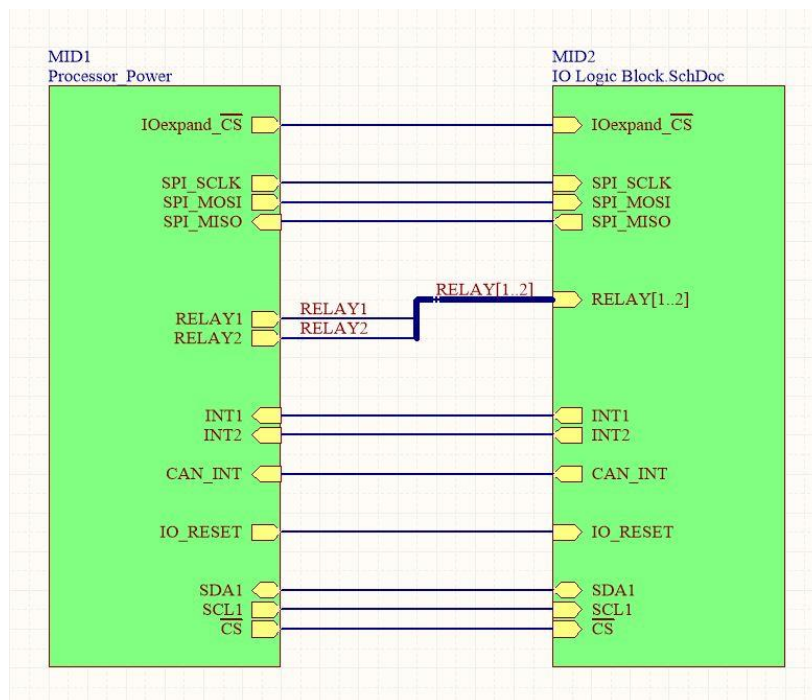


Figure 8. Net Labeled Bus and Bus Connections





29. After completion the project files will show the hierarchical connectivity in the project panel.
30. 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.

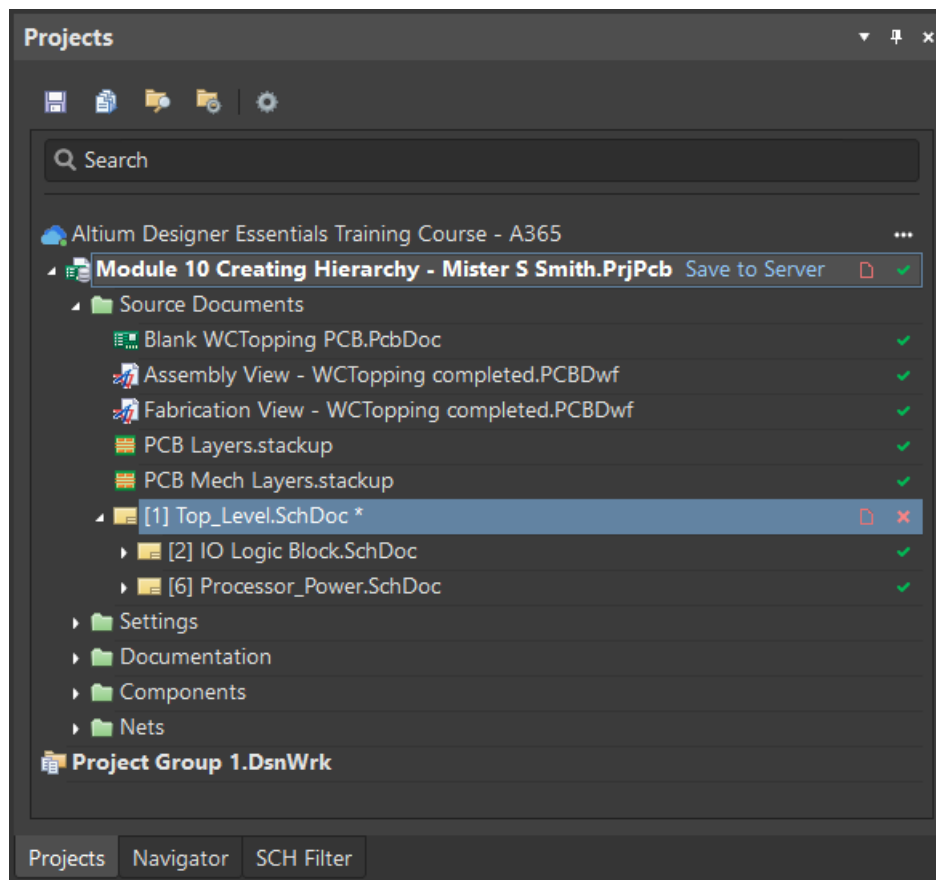


Figure 9. Hierarchical Design Project Structure

31. Select **File » Save All** to save all modifications.
32. Save the modifications to the server:
 - a) At the *Project* panel, next to the Project name you find the command **Save to Server** Save to Server.
 - b) Select **Save to Server**.
 - c) At the dialog *Save [Project Name]*,
 - i) Add the comment `Module 10: Creating Hierarchy - [Add Your Name] - Finished`
 - ii) Select **OK**.
33. When ready, close the project and any open documents, **Window » Close All**.





Congratulations on completing the Module!

Module 10: Creating Hierarchy

from

**Altium Designer Essentials Training
with Altium 365**

Thank you for choosing **Altium Designer**

