



Altium Designer

Essentials Training with Altium 365

Module 7: Project Creation and Storage

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 7: Project Creation and Storage	3
1 Purpose	3
2 Shortcuts	3
3 Preparation	4
3.1 General	4
3.2 Create a User Training Folder	4
3.3 Schematic Template	6
4 Create A New Project	9
4.1 Create the Project	9
4.2 Adding New Documents	11
4.3 Save to Server	13





Module 7: Project Creation and Storage

1 Purpose

In this module, you will learn how to create a Project in Altium 365, how to save the project and how to update the A365 Workspace (Server).

In this exercise, you will create the project `WCTopping - [Add Your Name here].PrjPCB`. After you create the project you will add additional schematic pages, save the project, and upload the project to your training folder in our training Workspace.

2 Shortcuts

Shortcuts used when working with Module 7: Project Creation and Storage

Ctrl+S	Save Document
K » R	Open <i>Explorer</i> Panel





3 Preparation

3.1 General

1. Close all existing projects and documents.

3.2 Create a User Training Folder

Before creating the new project, we must create a user training folder in the Altium 365 Workspace. This task is a one-time requirement in the training process.

2. Open the *Explorer* panel with **K » R**, with the Panels tab **Panels**, or any other way you prefer.
3. Create a new folder following the steps you see in Figure 1:
 - a) Navigate to the section **Projects » For Attendees**.
 - b) **Right click** and select the command **Add Subfolder... » Project Catalog**.

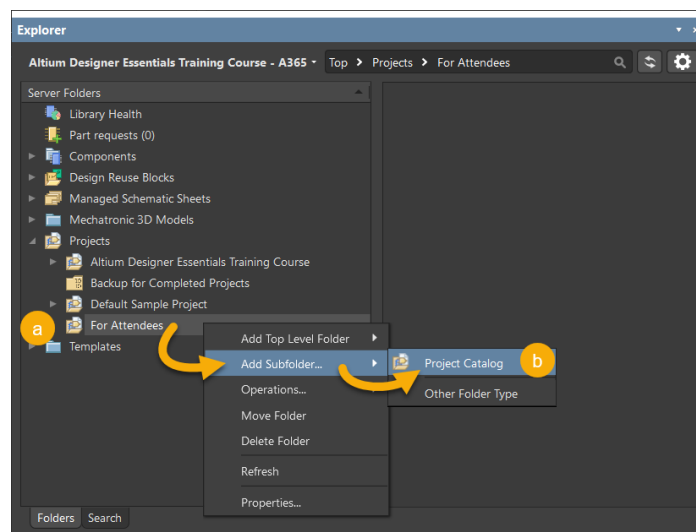


Figure 1. Navigate to User Area

4. In the *Add Folder* window, add **[Your Name]** as the folder name and select **ADD** to close the window, see Figure 2.

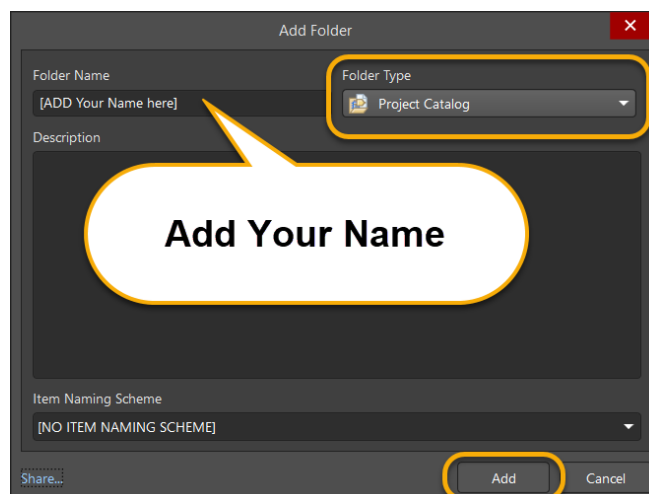


Figure 2. Add Training User Folder Name





5. Back in the *Explorer* panel, you can now see the folder that you just created.

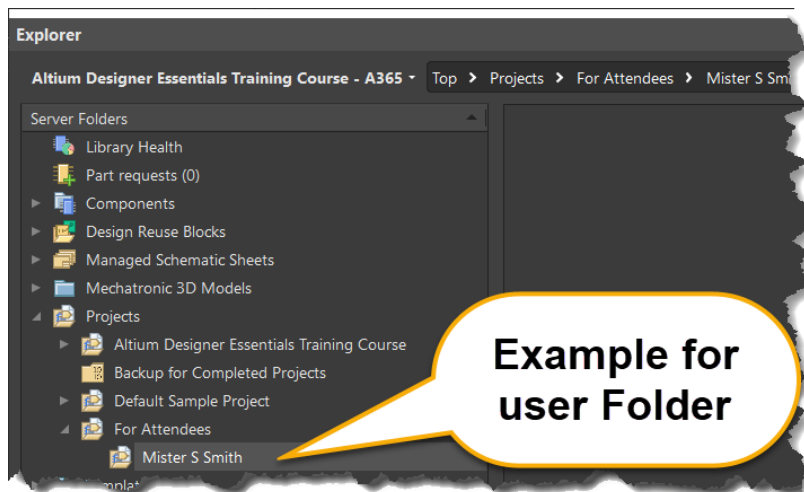


Figure 3. Example of a User Folder






3.3 Schematic Template

Before we create the new project, let's have a look at how we could set a default template for the schematic sheets.

Caution: If a Template is already configured in the Preferences, and you select a new Template with the following steps, the new Template will replace the existing one. The Template we use is configured for imperial units, MILs. This is important for the following modules and the symbols we use during the Training.

6. Open *Preferences* using the gear icon  in the top right corner.
7. Open the *Data Management* branch and look at the *Templates* page. Change the configuration as shown in Figure 4, Figure 5 and Figure 6.
8. To add the Altium Training Template:
 - a) Select **Defaults** (Figure 4).
 - b) Select **Add...** (Figure 4).
 - c) Select **From Server » Schematic** (Figure 4).

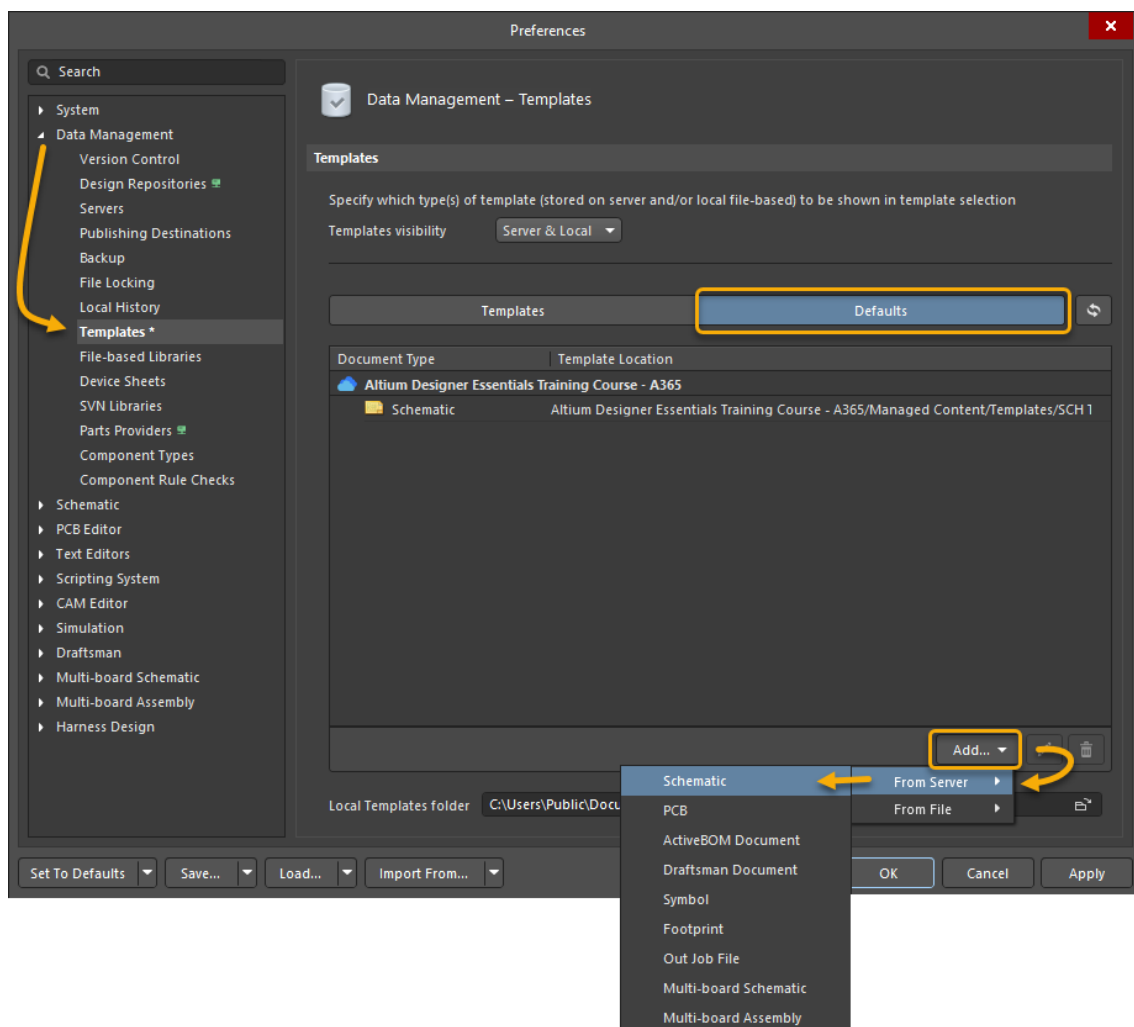


Figure 4. Add a Schematic Template



- d) Navigate to the section *Managed Content » Templates » SCH Templates* (Figure 5).
- e) Select the Template *ANSI B Landscape – Altium Training* (Figure 5).
- f) Close the *Choose Item revision for template* dialog with **OK**.

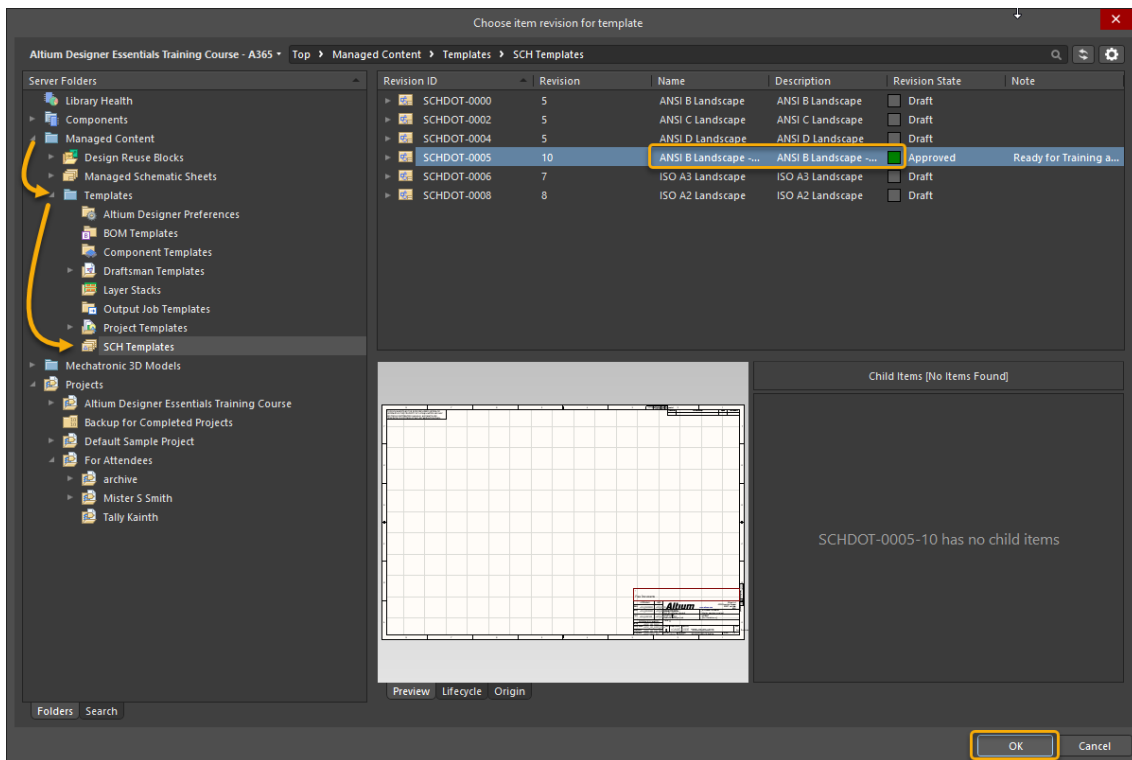


Figure 5. Select ANSI B Landscape – Altium Training Template from the Server

- g) Your *Preferences* should now look like Figure 6 .
- h) Close the *Preferences* with **OK**.

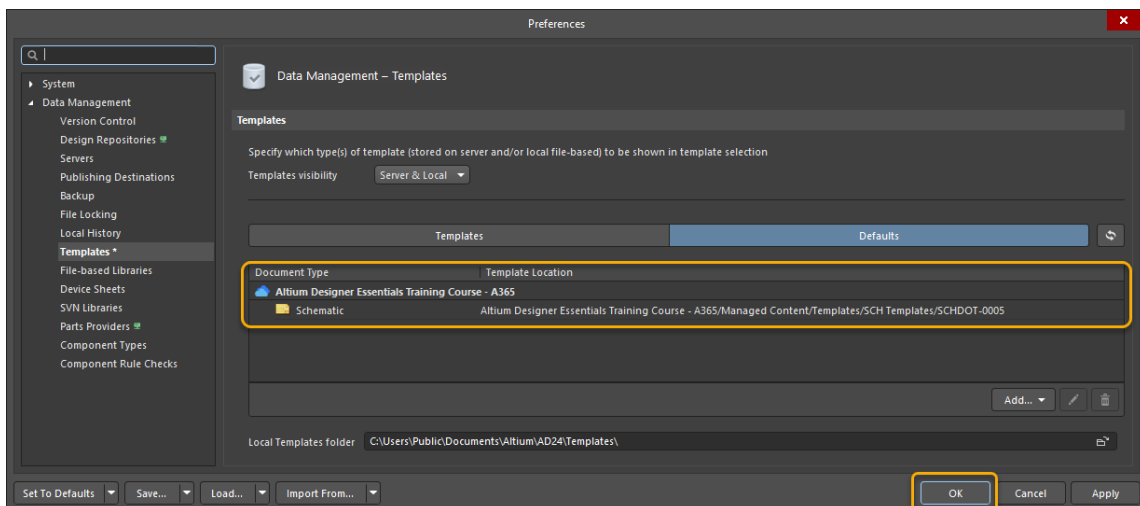


Figure 6. SCH Template Added to Preferences



Hint: Next time you create a new schematic, you will not see the Altium default page with the default Title Block, instead you will see a page layout based on the Template we added to the Preferences, Figure 7.

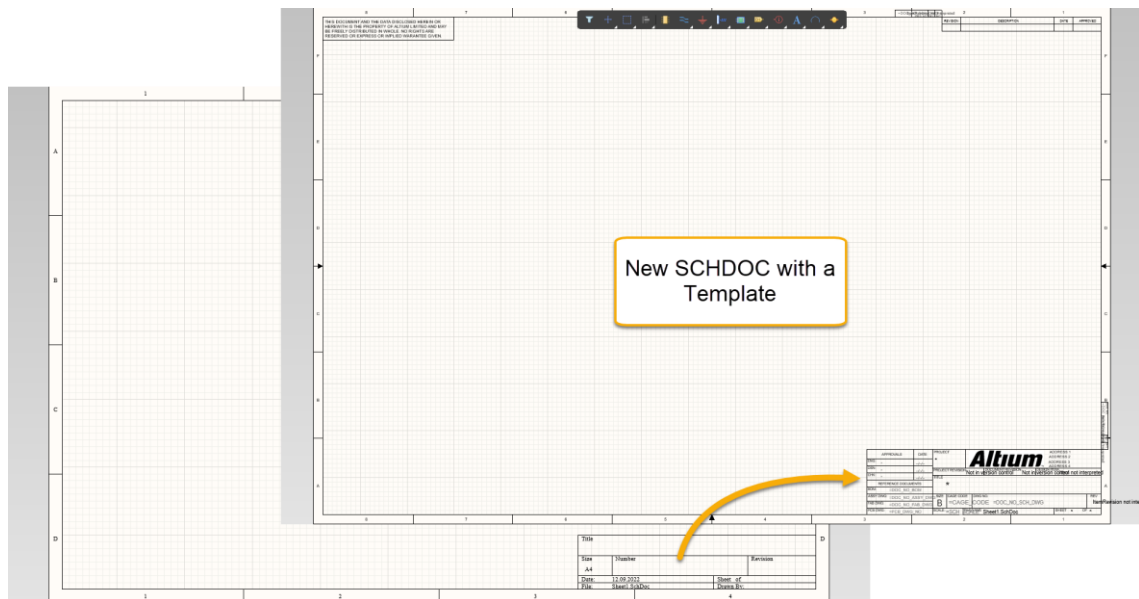


Figure 7. SCHDOC based on a Template



4 Create A New Project

4.1 Create the Project

9. Go to **File » New » Project...** to launch the *Create Project* dialog. The dialog shown in Figure 8 will appear.

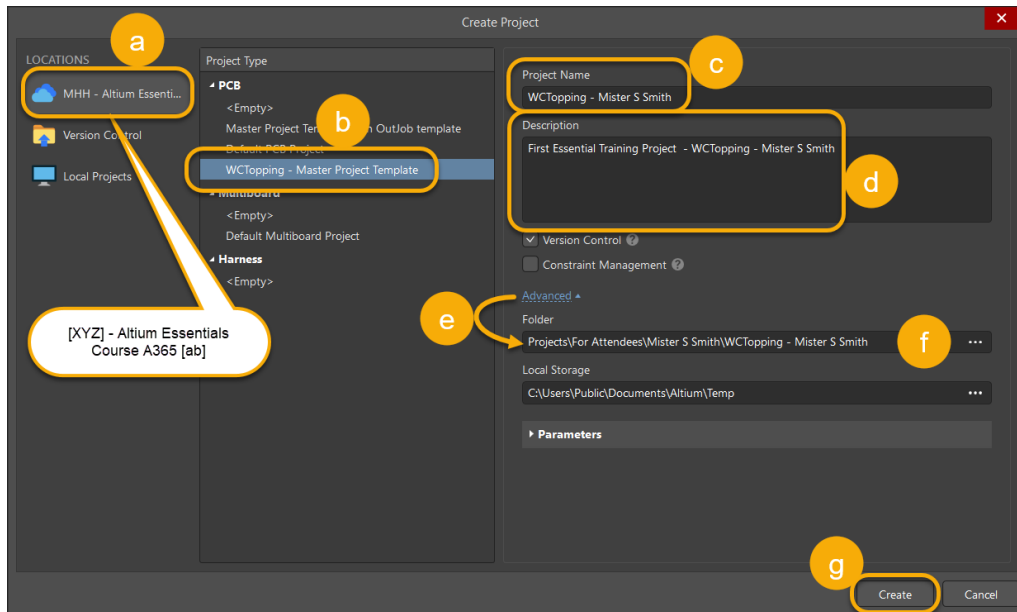


Figure 8. Create a New Project

10. Use Figure 8 above as a reference for the following steps. You may notice the **Constraint Management** option, we will not be using this specifically for this project, but we will cover it in a later module.
 - a) Select the workspace [XYZ] – Altium Essentials Course A365 [ab] in the *Locations* area.
 - b) Select **WCTopping – Master Project Template** in the *Project Type* pane, so that we can create a new project with some preconfigured / predefined settings.
 - c) Enter the project name: WCTopping – [Add Your Name here].
 - d) Add the following Description: First Essential Training Project – WCTopping – from [Add Your Name here].
 - e) Check that the Version Control Checkbox is enabled.
 - f) Open the **Advanced Settings**. This is the configuration for where your project will be saved.
 - i) *Folder*: Select the ellipsis [...] and chose the *Folder* Projects » For Attendees » [Your Name Here] that you created at the beginning of this Module, 3.2 Create a User Training Folder.
 - ii) *Local Storage*: Check if the predefined *Local Storage* Path is okay for you. If not, feel free to change the path.
 - g) Select the **Create** button to create the new project. It may take a few seconds for Altium Designer to create the new project.



11. Return to Altium Designer and notice that in the *Projects* panel there is a project called WCTopping - [Your Name].PrjPCB with a schematic document, a PCB document and additional documents that we will explain later, as shown in Figure 9 below.

- a) Select Blank WCTopping PCB.PcbDoc.
- b) **Right-click** and select **Rename...** change Blank WCTopping PCB.PcbDoc to WCTopping.PcbDoc.

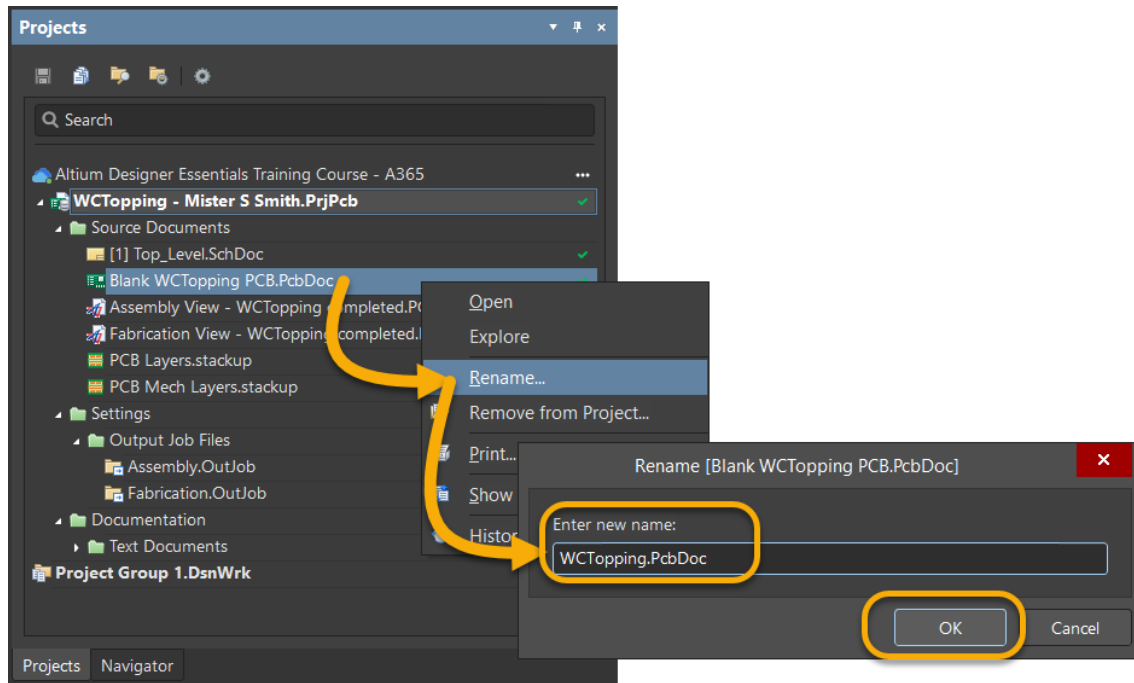


Figure 9. Projects panel with the new WCTopping project





4.2 Adding New Documents

12. Add new schematic pages using the *Projects* panel:

- Select the Project *WCTopping - [Your Name]*.
- Right-click** select **Add New to Project » Schematic** from the pop-up menu, as shown in Figure 10 below. This will add a new schematic to the project with the default name of *[2] Sheet1.SchDoc*.
- Repeat this operation and add **two additional** schematic files.

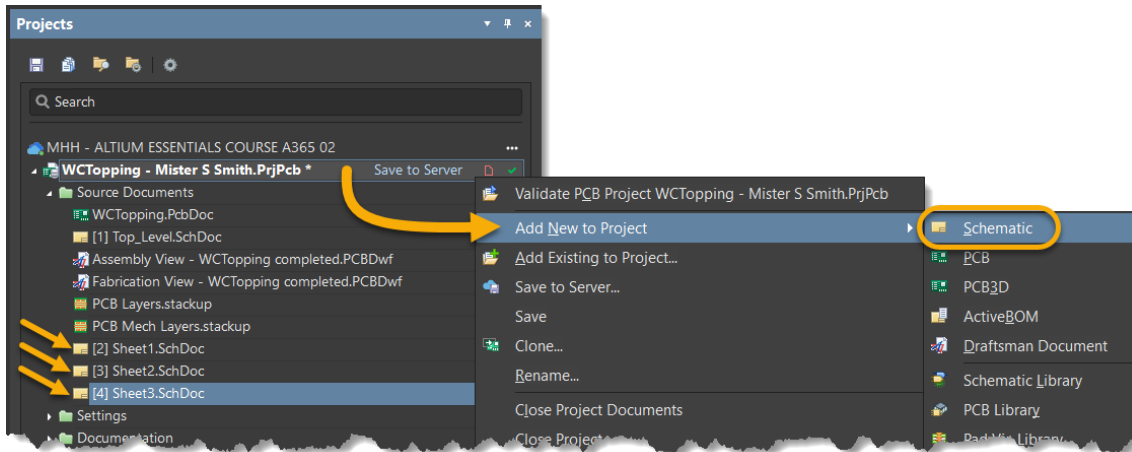


Figure 10. Add New Schematics to the Project





13. Next, we will save and update the file names for the schematic pages.

- a) Select [1]Top_Level.SchDoc.
- b) **Right click** and select **Rename...** change Top_Level.SchDoc to CAN_Interface.SchDoc.
- c) Go to **File » Save All** to save the rest of the files all at once.
- d) A dialog box will pop-up prompting you to save the new schematics you added. This will save the files as local copies on your hard drive. Name the schematics in your project using the file names below:
 - 2nd schematic as Digital_IO.SchDoc
 - 3rd schematic as Relay_IO.SchDoc
 - 4th schematic as Processor_Interface.SchDoc

14. Drag-and-drop to change the positions of the schematic pages, see Figure 11.

15. Now that we have modified the project locally, the project is no longer in sync with the server, indicated by the red and white icons next to the project name and the renamed schematic pages, see Figure 11.

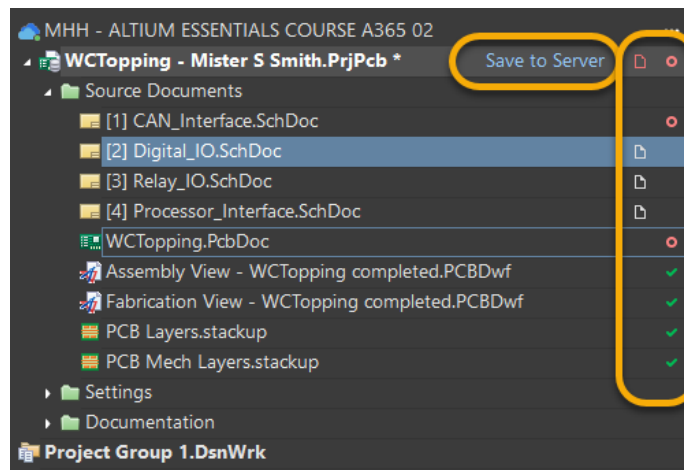


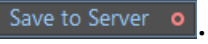
Figure 11. Project with Additional Schematics





4.3 Save to Server

Now that we have set up the project and the schematic files, and a local copy on a hard disk, we can now update the Altium 365 workspace with our new project.

16. In the *Projects* panel, next to the Project name, select **Save to Server** .

17. Update the *Save [Project Name]* configuration as seen in Figure 12.

- Add the comment `Initial Commit` in the comment section.
- Click **OK**.

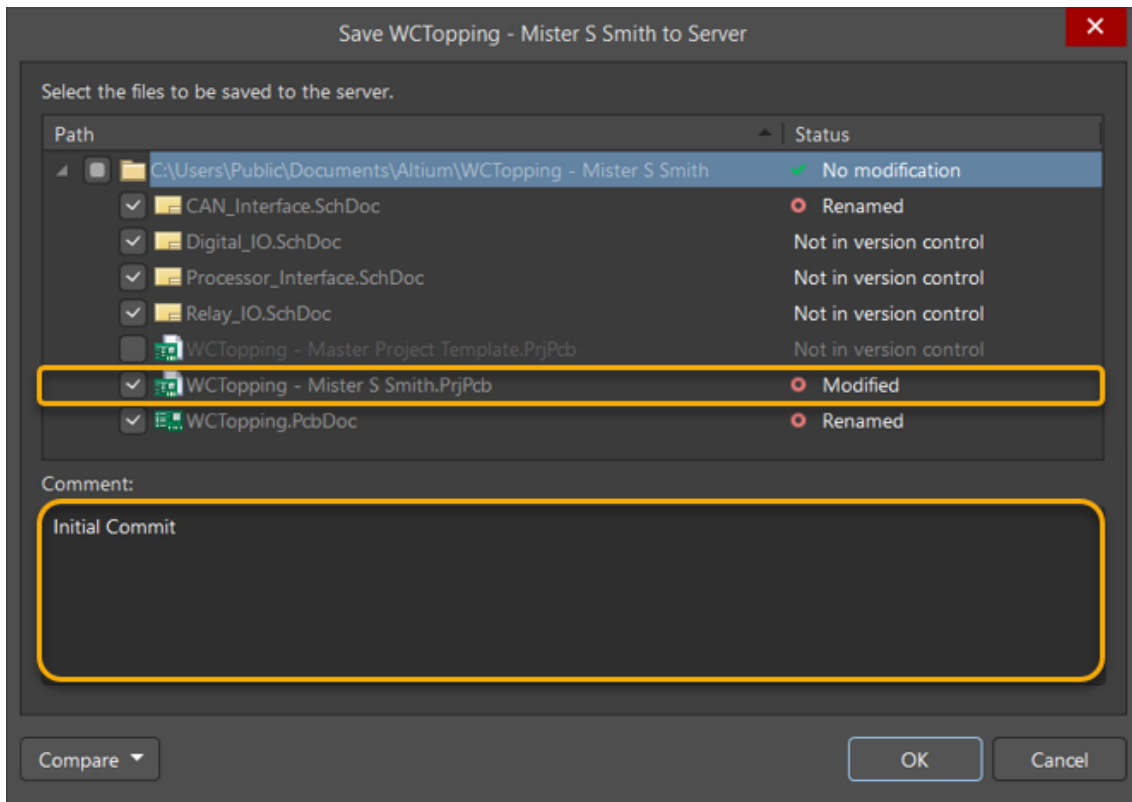


Figure 12. Save to Server



18. The *Projects* panel should now look like the one in Figure 13 below. Next to all the file names we see a green checkmark, indicating the local files are in-sync with the server.

Hint: To find the location of the folder where the project and the project files are stored locally, **right-click** on the project name and select the **Explore** command. The File Explorer will open showing the project files and the local GIT repository folder.

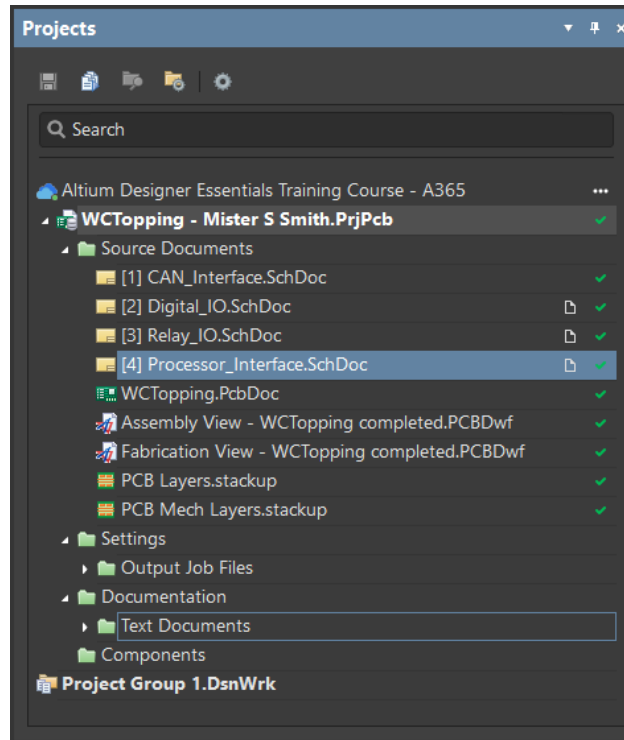


Figure 13. Renamed Project Files

19. Open the *Explorer* Panel and check the Folder *Projects » For Attendees* for the new folder and project you create (Figure 14).

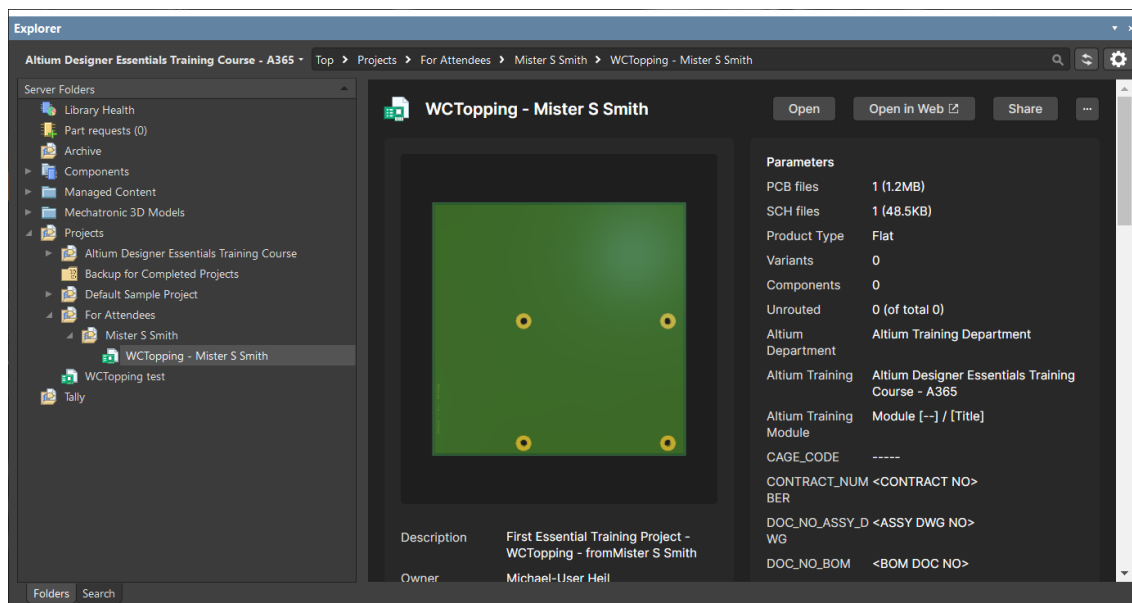


Figure 14. Workspace folder structure with new subfolder for the training attendee

20. When finished, close the project and all the schematic document with the command **Window » Close All**.



Congratulations on completing the Module!

Module 7: Project Creation and Storage

from

**Altium Designer Essentials Training
with Altium 365**

Thank you for choosing **Altium Designer**

