



Advanced Training with Altium 365

Opening a Project

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

| | |
|---|-----------|
| Opening a Project | 3 |
| 1 Purpose | 3 |
| 2 Important notice | 3 |
| 3 Shortcuts | 3 |
| 4 Opening Files | 4 |
| 4.1 Option One | 4 |
| 4.2 Option two | 5 |
| 4.3 Altium Designer with Loaded Project | 6 |
| 5 Projects Panel | 7 |
| 5.1 Creating a User Training Folder | 8 |
| 6 Copying an Existing Project | 10 |
| 6.1 Option one | 10 |
| 6.2 Option Two | 12 |
| 6.3 New Project after Copy Process | 14 |





Opening a Project

1 Purpose

In this exercise, you will learn how to open a project from the training workspace.

2 Important notice

Note: The training workspace comes preconfigured with a designated folder structure, including subfolders, and specific user and folder permissions.

The training projects are protected. You can download, open, and make local changes, but saving modifications back to the server is restricted to prevent overwriting the original files.

Later, when working with your workspace, most of the projects will not be protected. This will enable you to freely load, modify, and save the projects. You will experience this workflow later in the training.

Throughout the Altium Designer Advanced Training with Altium 365, you will learn a distinct method for accessing protected projects. This method lets you update the training projects and save your modifications to a designated user folder within the workspace.

3 Shortcuts

Shortcuts used when working with Opening a Project

| | |
|-------------------|--------------------------------|
| K » R | Open the <i>Explorer</i> panel |
| K » P » P » ENTER | Open <i>Projects</i> panel |




4 Opening Files

Next, we will describe two methods for opening a training project, please select one of the options to proceed.

4.1 Option One

1. From the **File** menu, select **Open Project....** follow the steps as seen in Figure 1 below.
 - a) Select workspace `Altium Designer Advanced Course` on the left side.
 - b) Browse to the `SL1 Xilinx Spartan-IIE PQ208 Rev1.02` example project, which is available in the workspace.

Hint: You can select the Button  to change from a list orientated view to a project folder hierarchy view. If you choose the folder hierarchy, observe the project path:
 Top » Projects » Altium Designer Advanced Training Course » Module 01 Getting started » SL1 Xilinx Spartan-IIE PQ208 Rev1.02

- c) Select `SL1 Xilinx Spartan-IIE PQ208 Rev1.02`.
 - d) Select **Open** to open the `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PrjPcb`.
2. If you choose option one, continue to section 4.3 Altium Designer with Loaded Project.

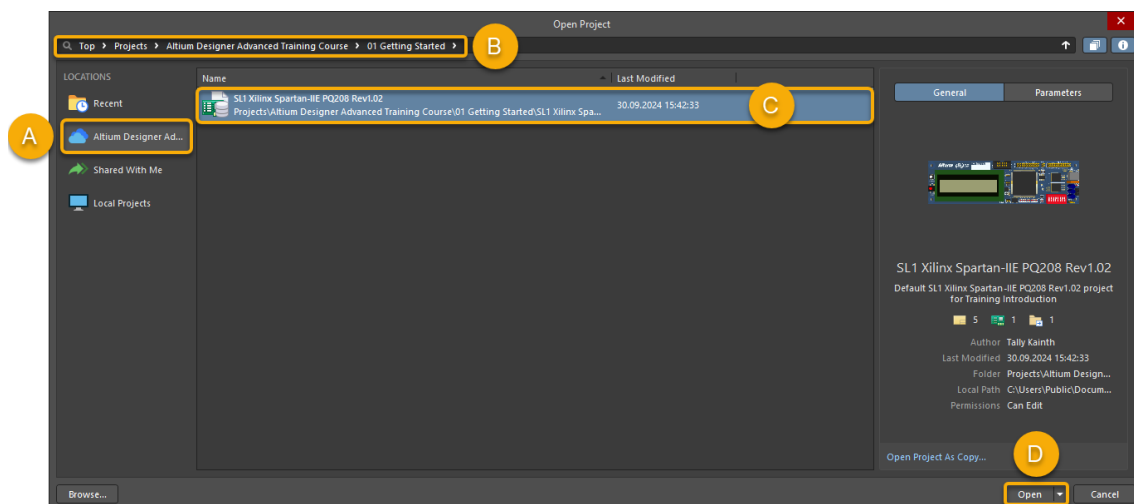


Figure 1. Open Project Dialog

Hint: If this is your first time opening Altium Designer, some panels may appear in the middle of your screen. You can close them by clicking the "x" in the top-right corner of the panel, or by right-clicking in the title bar of the panel and selecting **Close**. Panels can be re-opened, from the **Panels** button at the bottom right corner of the Altium Designer interface.

4.2 Option two

- Open the Explorer panel by either selecting **K » R**, clicking the **Panels** button, or accessing it from the main menu. **View » Panels » Explorer**.

Note: If the design space only has the Home Page active, the shortcut keys **K » R** will not work. In this case, to open the *Explorer*, use the *Panels* button at the bottom right, or **View » Panels** menu. Close the Home Page if it's open by default.

- Please follow the steps as illustrated in Figure 2:
 - Navigate to the section Projects » Altium Designer Advanced Training Course » Module 01 Getting started.
 - Select **Open** to open the project SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PrjPcb.

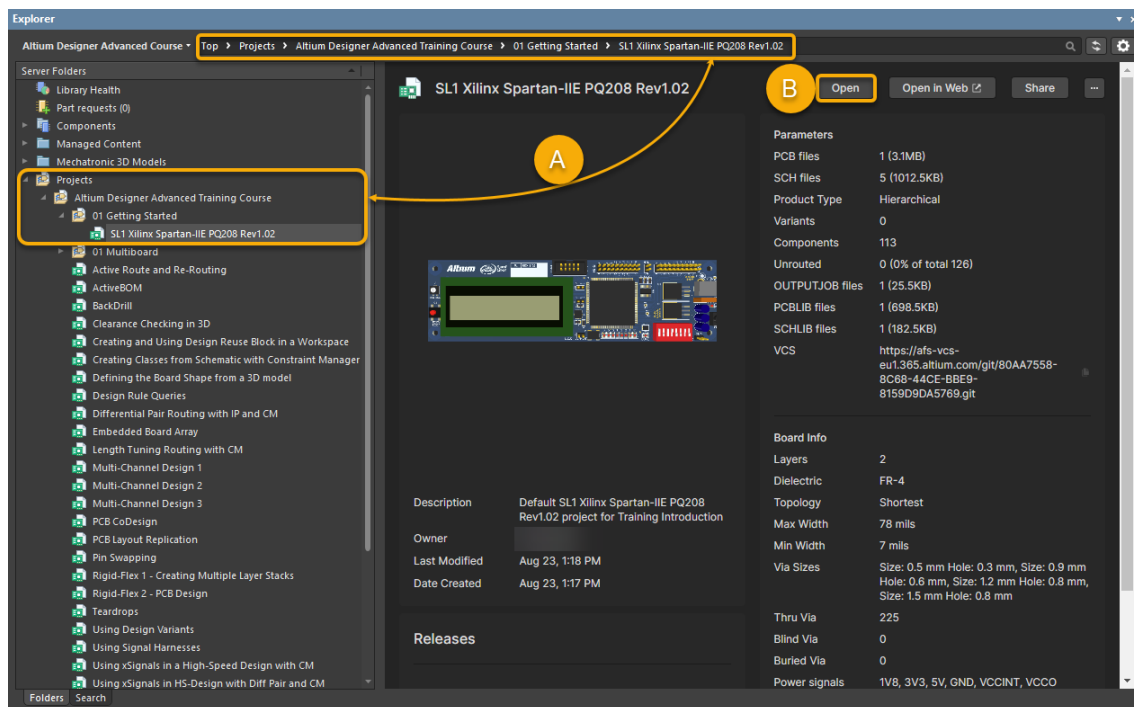


Figure 2. Open a Project - Explorer panel



4.3 Altium Designer with Loaded Project

5. With the project open, the interface should resemble what is shown in Figure 3.

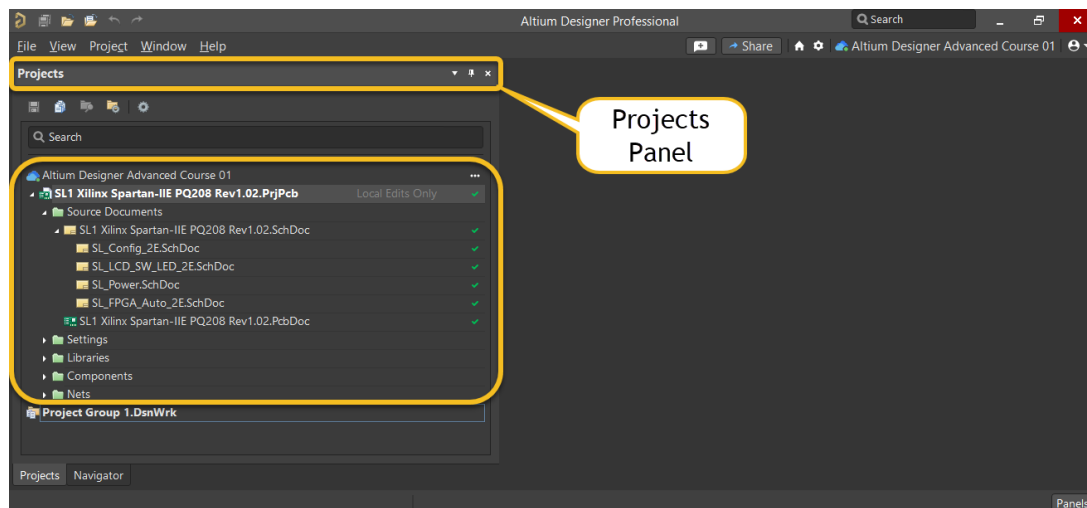


Figure 3. First Project loaded





5 Projects Panel

6. Find the Projects panel on the left side of the window, as demonstrated in Figure 4 below. If the panel is not currently active or you accidentally closed it, you can reopen it by selecting the 'Panels' button located in the lower right corner of your Altium Designer window or using your keyboard **K » P » P » Enter**.
 - a) Note that the Project name includes a `Local Edits Only` indication, signifying that the project is protected and can only be viewed.
 - b) Next to the project name, schematics, and the PCB, you will notice green check marks. They signify that the project is in the GIT Version Control System (VCS). The green check marks also indicate that the files are up to date, meaning the local information matches the workspace.

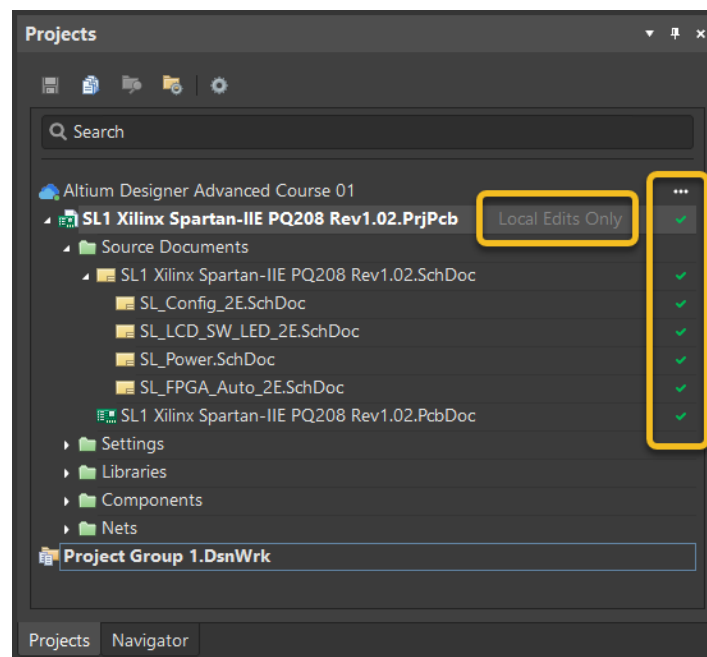


Figure 4. Project Panel





5.1 Creating a User Training Folder

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

7. Open the *Explorer* panel with **K » R**, with the Panels tab **Panels**, or any other way you prefer.
8. Create a new folder following the steps you see in Figure 5:
 - a) Navigate to the section **Projects » For Attendees**.
 - b) **Right click** and select the command **Add Subfolder... » Project Catalog**.

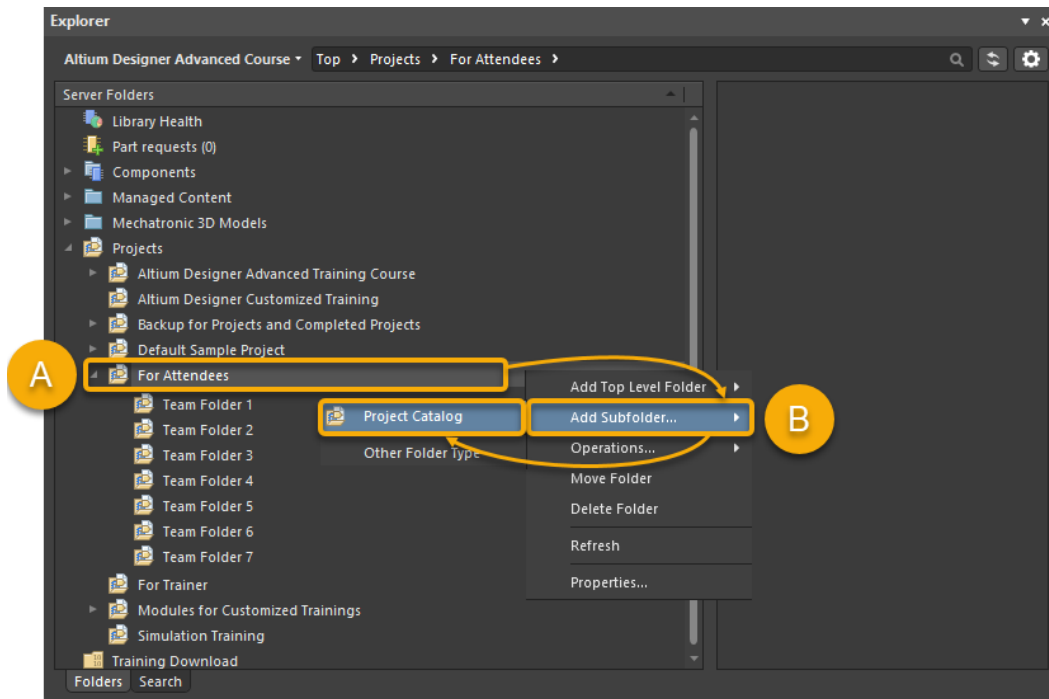


Figure 5. Navigate to User Area

9. In the *Add Folder* window, add [**Your Name**] as the folder name and select **ADD** to close the window, see Figure 6.

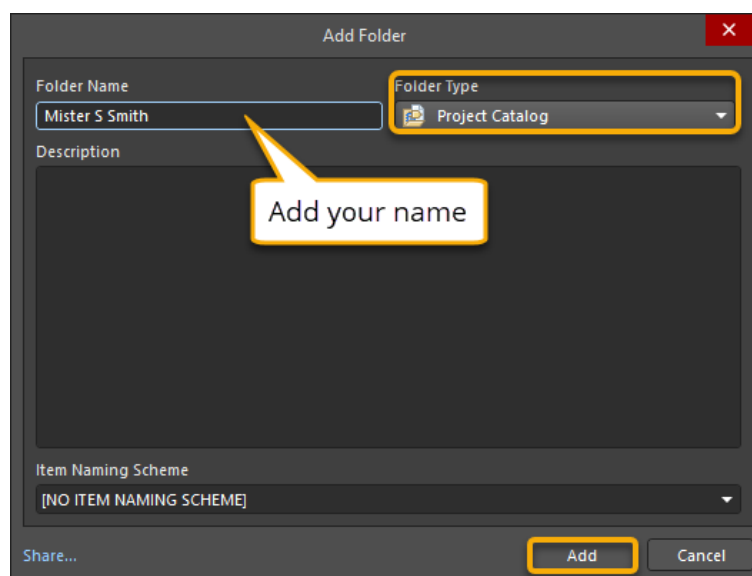


Figure 6. Add Training User Folder Name





10. Back in the *Explorer* panel, you can now see the folder that you just created, Figure 7.

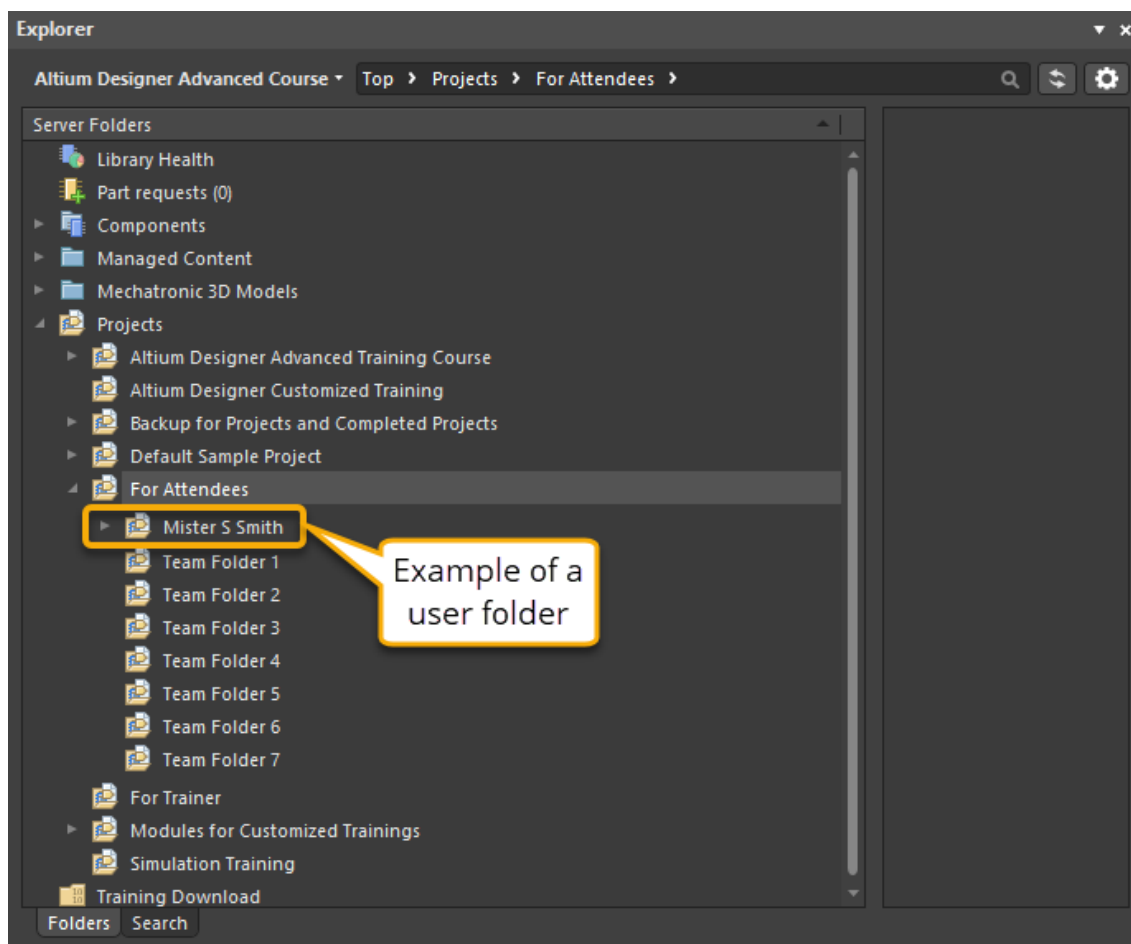


Figure 7. Example of a user folder



6 Copying an Existing Project

In this section, you will create a copy of an existing project. You can use a copy of the project for practical tutorial. By using the "Open Project as Copy" feature in Altium Designer, you gain the flexibility to work safely and efficiently without the risk of affecting your original project.

For training purposes, we have implemented a specific workflow that enables you to copy the predefined training Project. You can create a new local copy that will be saved in your personal training folder within the workspace.

11. Close all existing projects and documents.

6.1 Option one

12. Select **File » Open Project...** to open the *Open Project* dialog, Figure 8.

13. Enable the folder view button .

14. Select Altium Designer Advanced Course workspace.

15. Navigate to the predefined training project SL1 Xilinx Spartan-IIE PQ208 Rev1.02 (Top\Projects\Altium Designer Advanced Training Course\01 Getting started).

16. Select **Open Project as Copy...** .

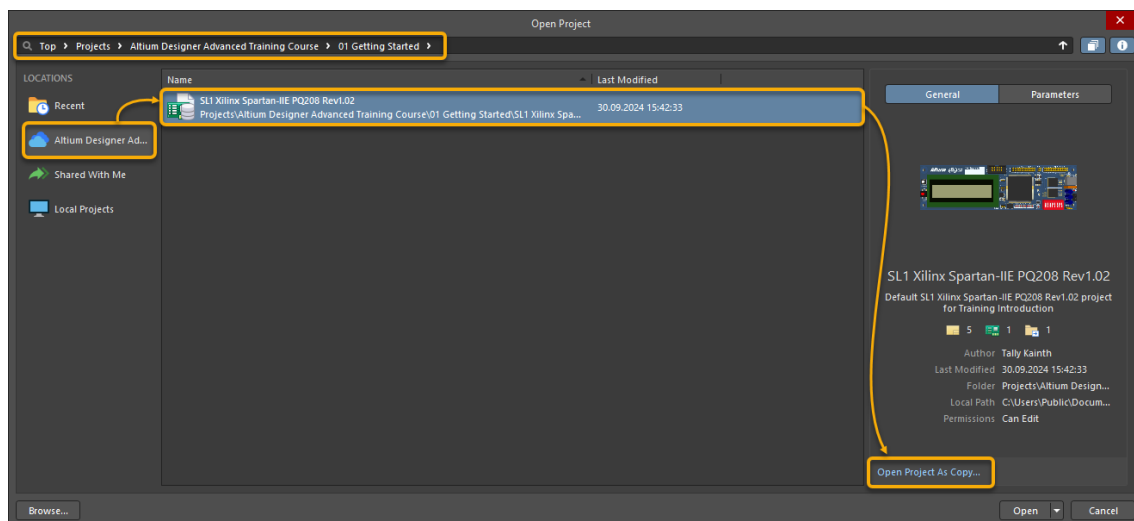


Figure 8. Copy an existing training project

Hint: With hovering over the Project name, you can start the copy command by using Right Click » Make a copy.



17. In the new dialog *Create Project Copy*, as seen in Figure 9.

- a) Add your name to the project: SL1 Xilinx Spartan-IIE PQ208 Rev1.02 - [Your Name]
- b) Add a description: Altium Advanced Training - [Your Name]
- c) Open the *Advanced* section.

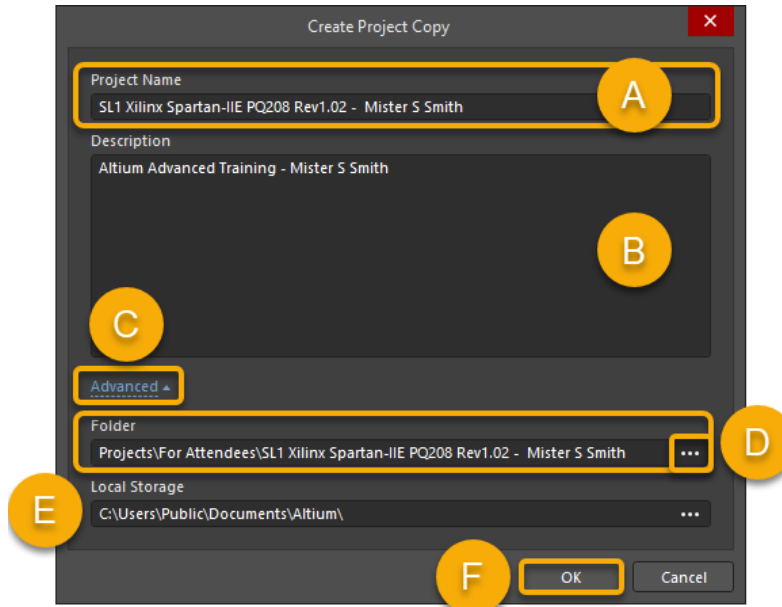


Figure 9. Create Project Copy

- d) Select the Ellipsis Button **...**, to open the *Choose Folder* dialog, as shown in Figure 10.
 - i) Select the folder with your name: Project\For Attendees\[Your Name]
 - ii) Select **OK**

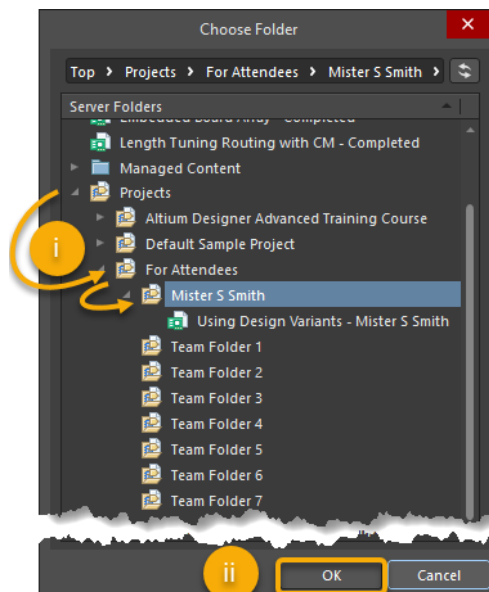


Figure 10. Step d - Choose Folder dialog

- e) Back in the *Create Project Copy* dialog, change the *Local Storage* path if needed.
 - f) Select **OK** to create the copy.
18. 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.



6.2 Option Two

19. Open the *Explorer* panel, **K » R**.
20. Navigate to the predefined training project SL1 Xilinx Spartan-IIE PQ208 Rev1.02 (Top\Projects\Altium Advanced Course A365\01 Getting started).
21. Select the Ellipsis Button **...** and the command **Make a Copy**, as seen in Figure 11.

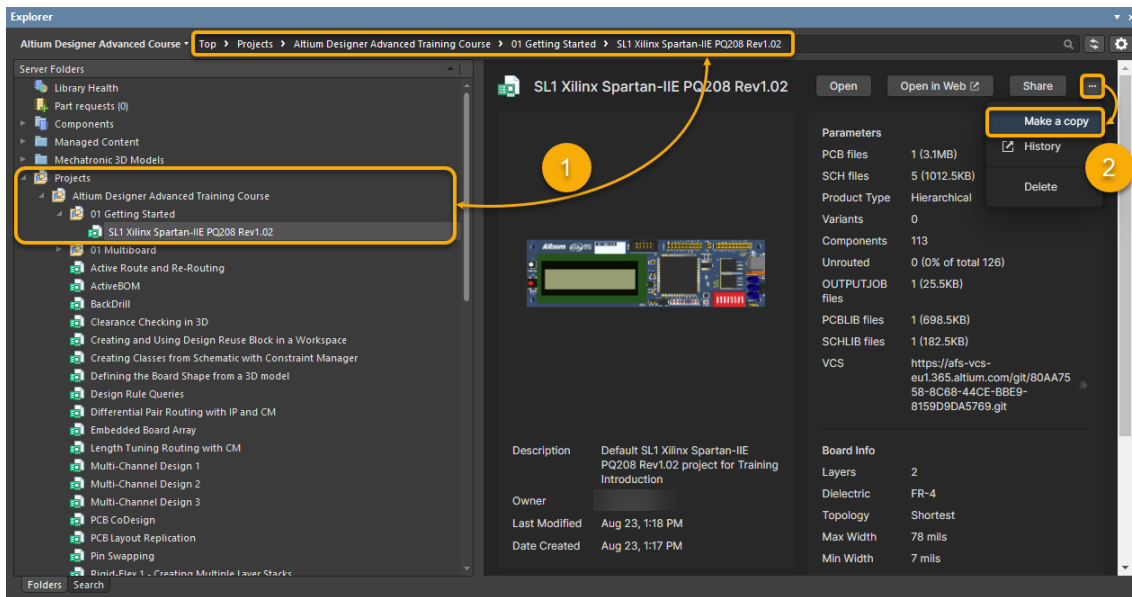


Figure 11. Copy Training Project



22. In the new dialog *Create Project Copy*, as seen in Figure 12.

- a) Change the project name to SL1 Xilinx Spartan-IIE PQ208 Rev1.02 - [Your Name] .
 - b) Add a description: Altium Advanced Training - [Your Name] .
 - c) Open the Advanced section if needed.
 - d) Select the Ellipsis Button **...** from the **Folder** configuration to open the *Choose Folder* Dialog, Figure 13.
 - i) Change the path to Projects\For Attendees\[Your Name]
 - ii) Select **OK**.
 - e) Change the *Local Storage* path if needed.
 - f) Select **OK** to close the Dialog.
23. Please wait until Altium Designer creates a copy of the project and adds it to the Projects panel for you. This process may take up to 1 minute.

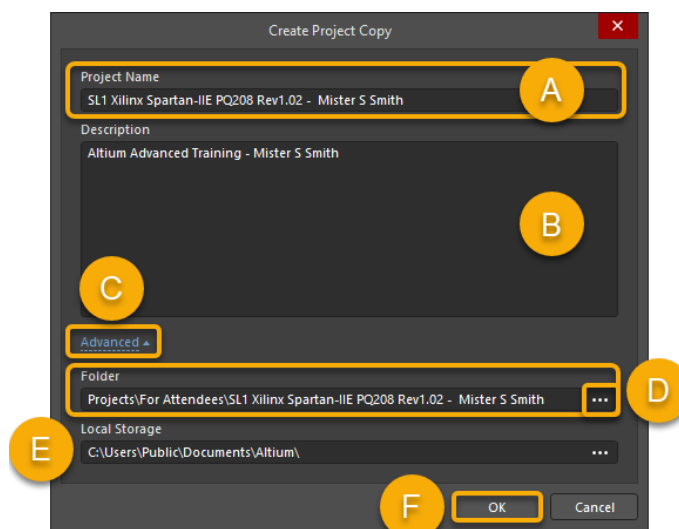


Figure 12. Create Project Copy

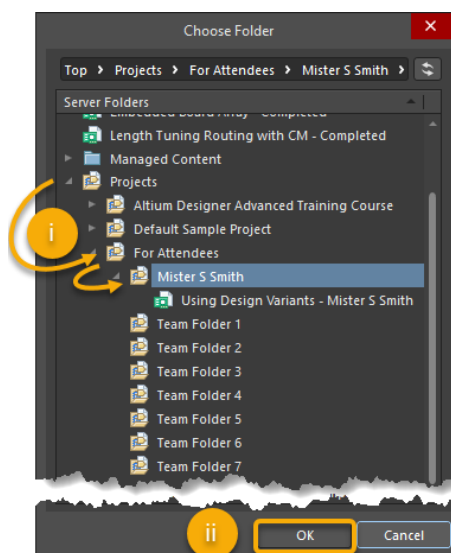


Figure 13. Choose Folder Dialog





6.3 New Project after Copy Process

24. As a result of the Copy process, you will see a new project in your Project panel, similar to Figure 14.

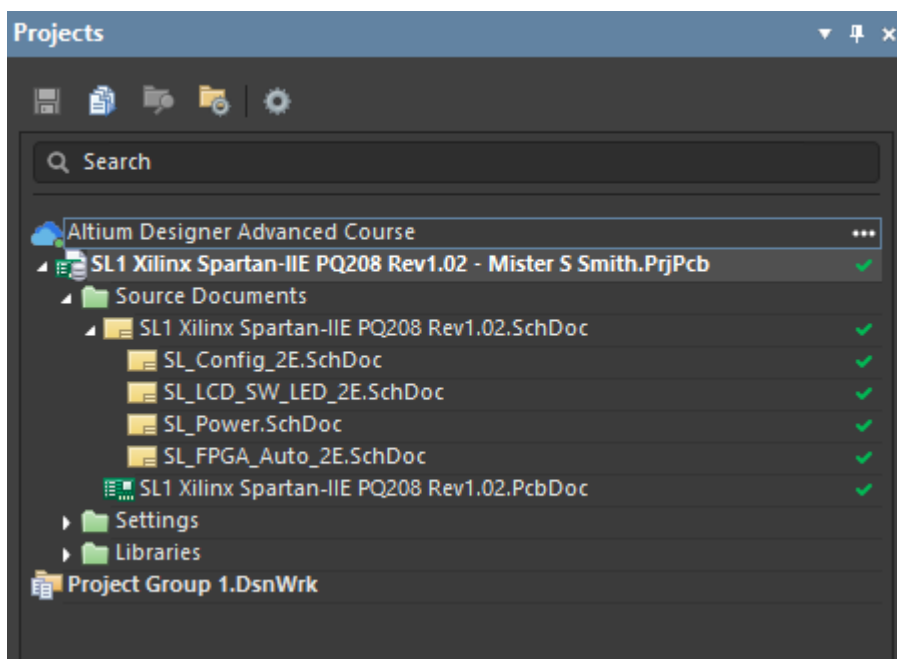


Figure 14. New Project based on the Predefined Training Project





Congratulations on completing the Module!

Opening a Project

from

**Altium Designer Advanced Training
with Altium 365**

Thank you for choosing **Altium Designer**

