



# Altium Designer

## Essentials Training with Altium 365

### Module 3: Open 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

<b>Module 3: Open a Project</b>	<b>3</b>
<b>1 Purpose</b>	<b>3</b>
<b>2 Important notice</b>	<b>3</b>
<b>3 Shortcuts</b>	<b>3</b>
<b>4 Opening Files</b>	<b>4</b>
4.1 Option One	4
4.2 Option Two	6
4.3 Altium Designer with Loaded Project	6
<b>5 Project Panel</b>	<b>7</b>





# Module 3: Open a Project

## 1 Purpose

In this exercise, you will learn how to open a Project, stored in the Altium cloud system Altium 365.

## 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 Essentials 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 Module 3: Open 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

- From the **File** menu, select **Open Project....** follow the steps as seen at Figure 1 below.
  - Select Workspace [XYZ] - Altium Essentials Course A365 [ab] from the column on the left.
  - Enable the folder view button** .
  - 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 Essential Training Course » Module 03  
Open a Project

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

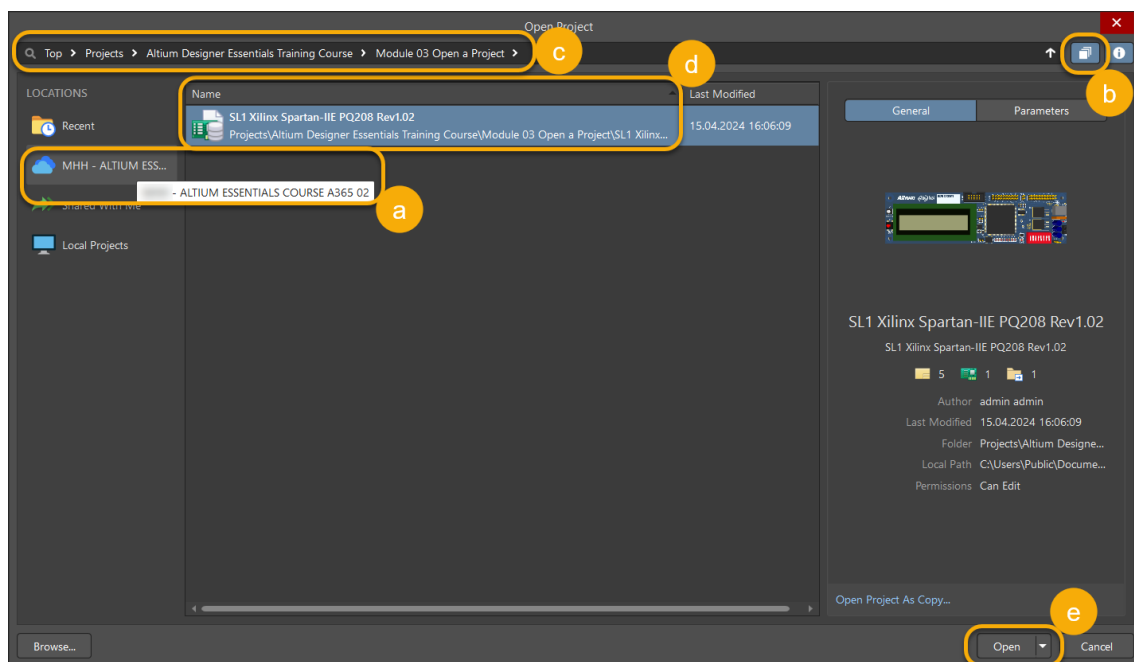
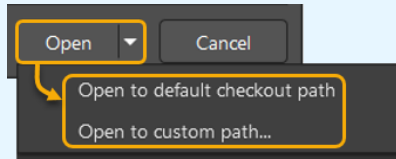



Figure 1. Open Project Dialogue

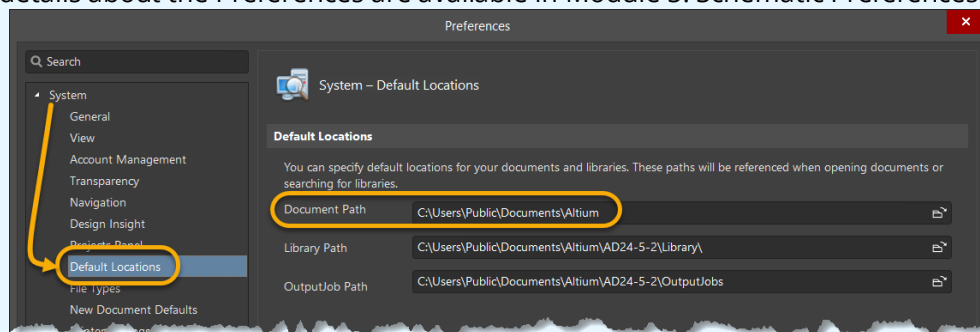
Hint: If this is your first time opening Altium Designer, some panels may appear in the middle of your screen. They can be closed by clicking the **"X"** in the top-right corner of the panel, or by right-clicking in the title bar of the panel and selecting **Close**. Panels can be re-opened, from the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



Hint: Using **File » Open Project** allows you to use a pull down, next to the Open button, to define a custom Path file location.



Hint: You can configure the default location for a new project inside the Preferences. Click  at the upper right to open the Preferences. More details about the Preferences are available in Module 5: Schematic Preferences.



## 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**.
- Please follow the steps as illustrated in Figure 2:
  - Navigate to the section **Projects » Altium Designer Essential Training Course » Module 03**.
  - Select **Open** to open the project **SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PrjPcb**.

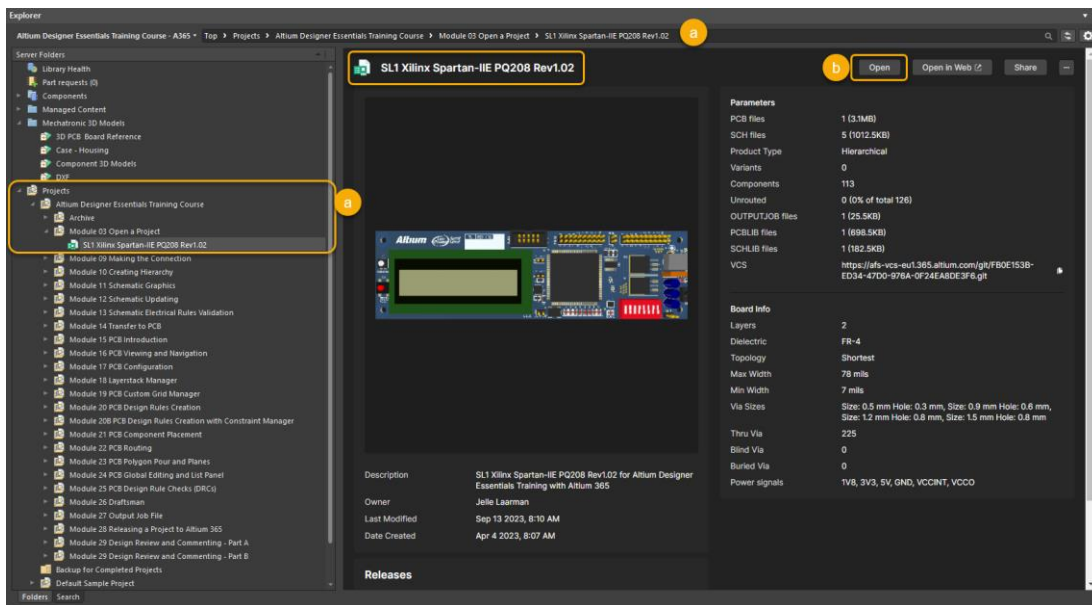


Figure 2. Open a Project - Explorer panel

## 4.3 Altium Designer with Loaded Project

- With the project now open in Altium Designer, the interface should resemble what is shown in Figure 3.

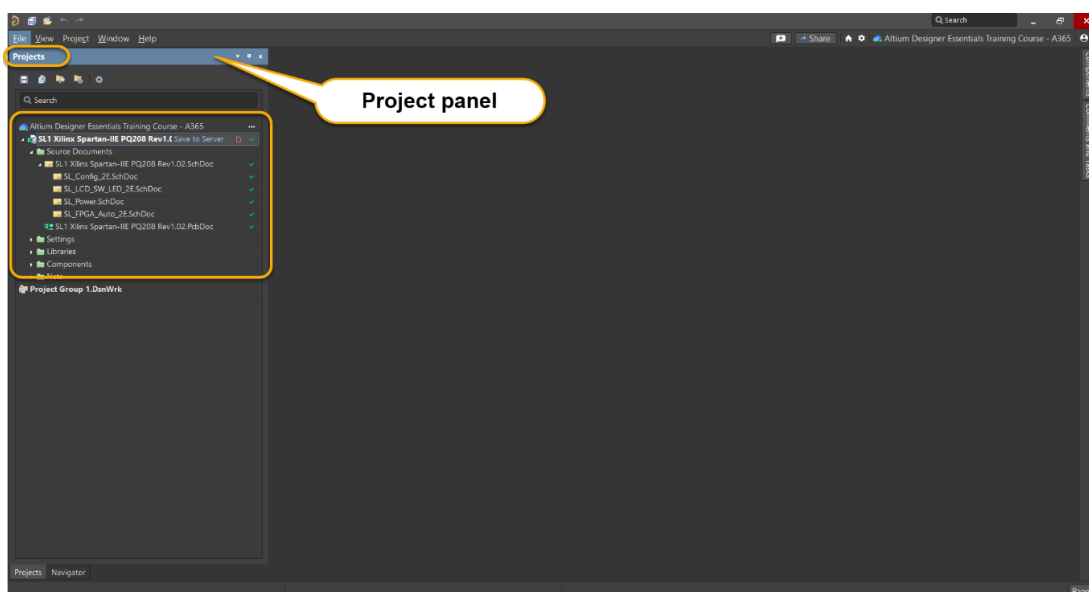


Figure 3. First Project loaded



## 5 Project Panel

6. Locate the Projects panel on the left side of the window, as shown in Figure 4 below. If the panel is not currently active or you accidentally closed it, you can reopen it by clicking 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 'View 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 a green check mark; this signifies the project is in the GIT Version Control System (VCS). The green check mark also indicates that the files are up to date, meaning that the local information matches the workspace.

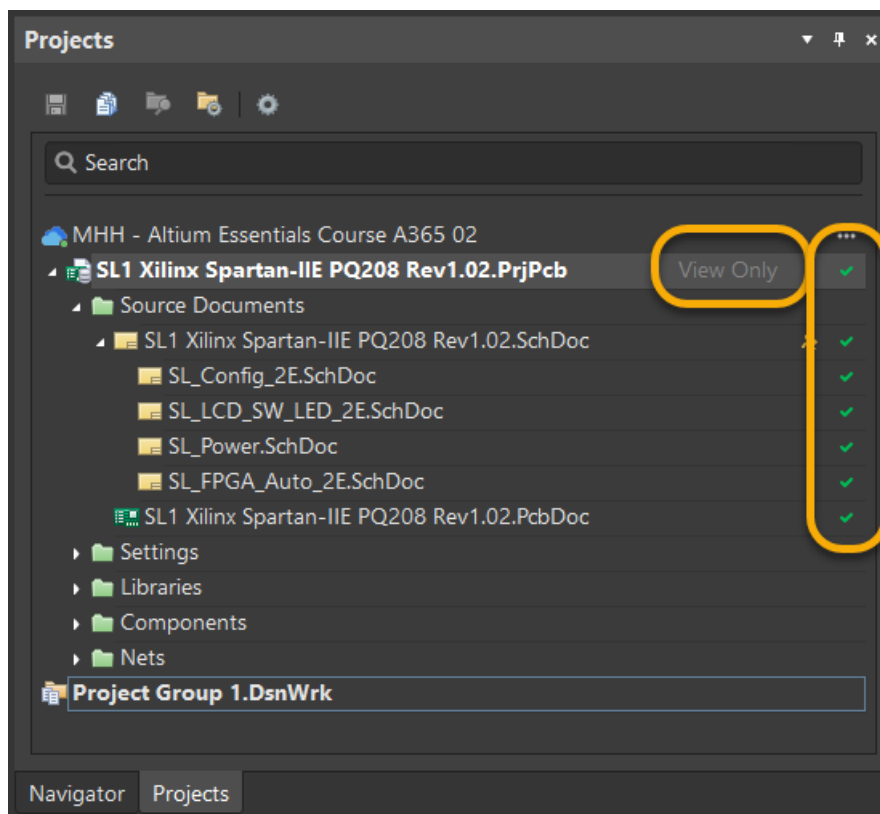


Figure 4. Project Panel

7. Do not close the project, we will also use this project in the next module.







**Congratulations on completing the Module!**

Module 3: Open a Project

from

**Altium Designer Essentials Training  
with Altium 365**

Thank you for choosing **Altium Designer**

