



Altium Designer

Advanced Training with Altium 365

Accessing Altium 365

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

Accessing Altium 365	4
1 Purpose	4
2 Important notice	4
3 Shortcuts	4
4 Preparation	5
5 Sign In	6
5.1 Sign in to Altium User Account	6
5.2 Sign into the Training Workspace	7





Accessing Altium 365

1 Purpose

In this exercise, you will sign into Altium Designer and the training workspace.

2 Important notice



Caution: Please note that the training workspace will only be accessible during the training. Once the training is completed, you will no longer have access to the Altium training workspace. It is important to avoid uploading any personal files, projects, or any other materials to the Altium training workspace.

Caution: Some of the functions you see during training are configurable by Preference settings. We recommend using an Altium Designer installation without modified settings for this training. If possible, install a parallel installation of Altium Designer, to use only during the training.

A separate installation will prevent unintentional changes to your company preference settings in your central installation.

3 Shortcuts

Shortcuts used when working with Accessing Altium 365

	MS Windows – Open Windows Start Menu
	MS Windows – Start Altium Designer





4 Preparation

1. Using the Windows Start Menu, locate the Altium Designer application, as shown in Figure 1. You may also opt for any alternative method that suits your preferences. Ensure to open the latest version of Altium Designer or the version supported by your company.

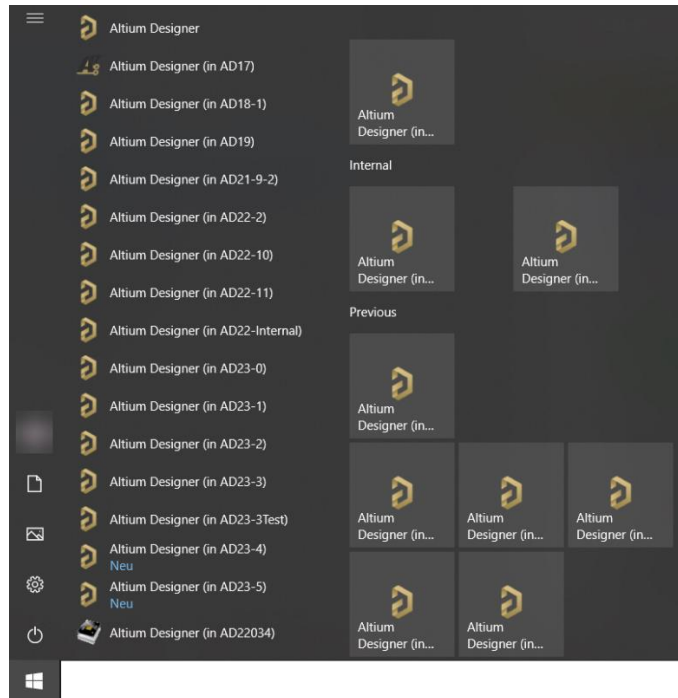


Figure 1. Start Altium Designer. Multiple, parallel installations are possible

2. After starting Altium Designer, close all existing projects and documents that are open, Menu **Window » Close All**.





5 Sign In

During the Altium Designer Advanced Training with Altium 365, we are using a Workspace provided by Altium, to which you are invited. The name of the Workspace is *Altium Designer Advanced Course*.

5.1 Sign in to Altium User Account

3. If not already signed in to Altium User account, sign in with your Altium Credentials, as shown in Figure 2.

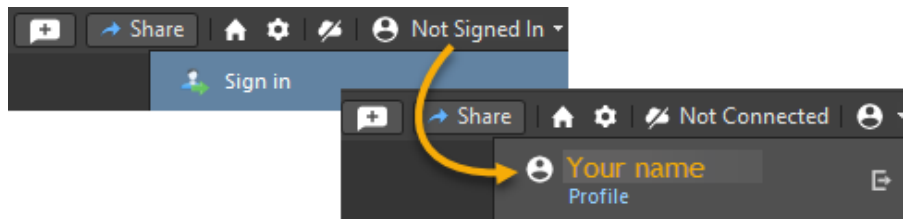


Figure 2. Sign in into Altium User account

4. Based on you Altium Designer version you may see the Information to Sign In with your Browser, Figure 3.

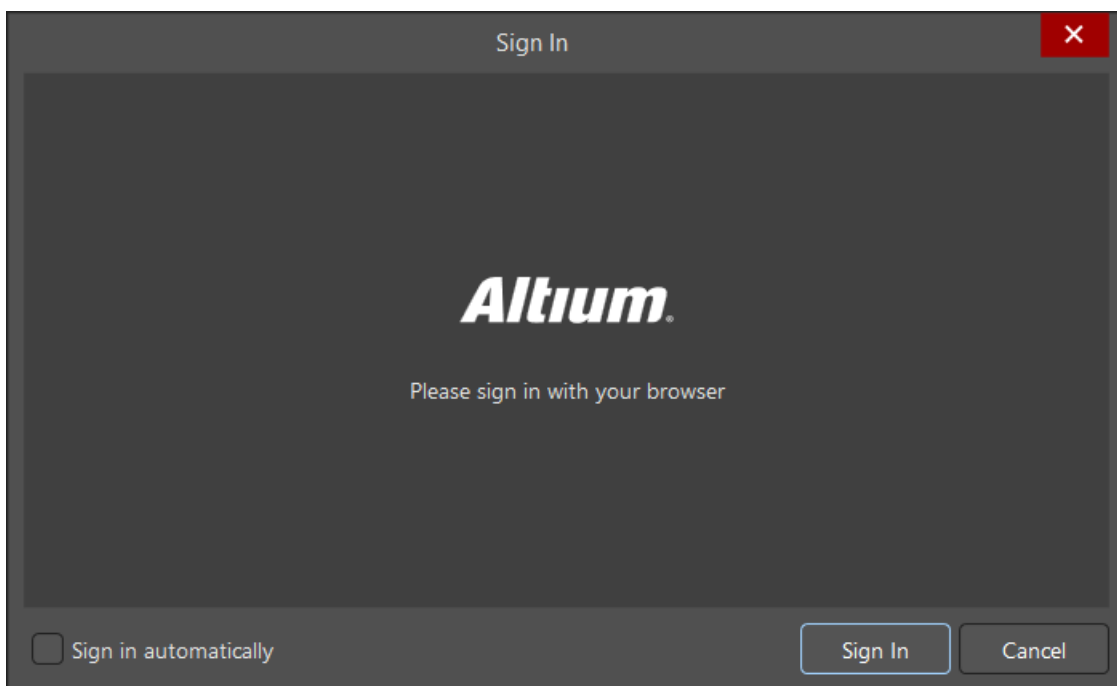


Figure 3. Browser Sign In





5.2 Sign into the Training Workspace

Next, we will connect to the Training Workspace.

5. Open the drop-down **Not Connected** list to see the workspaces that are available.
6. Select the shared Workspace: *Altium Designer Advanced Course*, Figure 4.

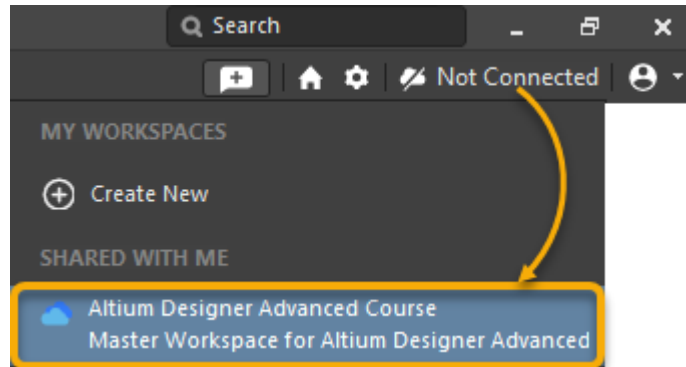


Figure 4. Select the Training Workspace - Altium Designer Advanced Course

7. Depending on your connection or Internet speed, you may see the following message: *Connecting - Please Wait*, Figure 5.

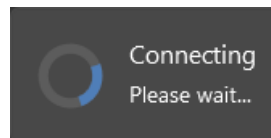


Figure 5. Connecting - Please Wait

8. If the connection is established, you will see the name of the Workspace, that is *Altium Designer Advanced Course*, as shown in Figure 6.

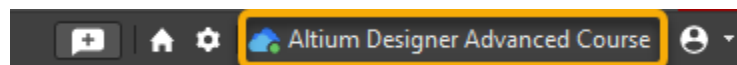


Figure 6. Connected with the training workspace





Congratulations on completing the Module!

Accessing Altium 365

from

**Altium Designer Advanced Training
with Altium 365**

Thank you for choosing **Altium Designer**

