Installation of Altium Designer is both intuitive and fast, courtesy of the Altium Designer Installer. This wizard-based installer streamlines the initial installation process, while letting you choose the initial 'flavor' of the installation, in terms of installed functionality.

This article takes a look at the procedure for installing Altium Designer on a computer.

Each major version of Altium Designer is released as a separate full installation with unique default installation paths. This allows for side-by-side installation with other major released versions of the software. While it is possible to update to the next point release within a version 'stream', it is not possible to update from one version stream to the next, later version stream.

### Pre-Install Preparation

To access the Altium Designer Installer, you need to first download and run a small (approx. 9MB) executable - AltiumDesignerSetup<Version>.exe. This executable is downloaded from the Downloads page of the Altium website, and can be saved anywhere on your hard disk.

Each version of the software has its own dedicated installer.

### Running the Installer

The installer itself is wizard-based, providing a set of progressive (and intuitive) pages that collectively gather the information to effectively install the version of the software, and its initial feature set, as you require.

Click Next to continue. The following sections provide a summary of the various pages you will encounter.

**License Agreement**

This page of the Installer presents you with the Altium End-User License Agreement (EULA).
Read and accept Altium's End-User License Agreement.

Before you can continue with the installation of the software, you must read through, and accept, the terms of this agreement. The agreement can be viewed in different languages, including English (the default), German, Chinese and Japanese.

Clicking the Advanced button will access an Advanced Settings pop-up window. From here, you are able to specify proxy-settings, if applicable. Click OK to effect any changes and return to the License Agreement page.
Specify the use of basic authentication using proxy settings, as part of advanced options for the install.

With the EULA read, continue with the install by checking the **I accept the agreement** box and clicking the **Next** button.

**Installation Mode**

If you already have Altium Designer installed, and are running the installer for a subsequent point release within the same version stream (e.g. installing Altium Designer 15.1.x, and Altium Designer 15.0.x is already installed), an additional page of the *Altium Designer Installer* wizard will be available - **Installation Mode**. This page gives you the choice of either updating an existing version instance of Altium Designer, or installing as a separate new version instance. Choose the **New installation** option to maintain separate version instances of the software on the same computer. Alternatively, choose the **Update existing version** option, then use the drop-down field to select the specific instance of the software you wish to update.
If you already have a previous installation of Altium Designer within the same version stream, you can choose to update that version. Or, simply install as a separate unique instance.

With your choice of installation mode made, click **Next** to proceed.

**Account Log-In**

You will now be presented with an *Account Log In* pop-up window. This simply means logging in to your Altium Account – to which your valid license is associated – using your AltiumLive Credentials. The reason for this is two-fold:

- Firstly, it verifies your identity and that installation of the software is valid, as determined by your account and licensing.
- Secondly, it provides secure access to the Altium Vault in which the installation files reside – the files that are required to be downloaded before the software can be ultimately installed on the PC.
Log in to your account using your AltiumLive Credentials.

**Select Design Functionality**

This next page of the Installer allows you to specify which features are installed initially. The PCB Design option provides the base software and core technologies required to perform PCB design. It is always installed and cannot be deselected. Use the remaining options to tailor the install to suit your needs. This will also have an impact on the initial installation time too!

![Click to expand an area for greater control over sub-functionality available.](image)
What initial functionality would you like in your installation of Altium Designer? - The choice is yours!

Platform Extensions offer extended functionality to the core platform. Each platform extension is a dedicated package of core platform functionality that has been 'brought out' of the platform and delivered in the guise of an extension. By delivering this functionality as an extension, Altium can deliver updates to that core functionality without having to release an update to the entire platform itself.

Simply click to enable the functionality you require. As you select and deselect functionality, text at the bottom of the page dynamically updates to reflect the size of the download involved and how this translates to required disk space, once those downloaded files are installed.

Click to enable functionality at the parent level, to quickly enable all of its child functionality too.

Remember, no matter what decision you take at initial install, you can add or remove features and functionality at any stage after installation. This puts you in the driving seat as it were, to fully customize your installation in-line with your, possibly ever-changing, design needs.

After defining what functionality you'd like, click **Next** to proceed.

**Destination Folders**

This next page of the Installer allows you to specify the destination folders for the install, in terms of both the *Program Files* for the software and the *Shared Documents* that are accessed and used by the software.
Specify where the software and associated documents are to be installed.

These locations become especially important when installing multiple instances of Altium Designer. If one or both of these destinations is not currently an empty folder (i.e. devoid of content) you will be prevented from proceeding with the installation until you specify alternate (and empty) destination(s).

The default destinations (Windows 7 and above) are:

**Program Files**

C:\Program Files (x86)\Altium\AD15

**Shared Documents**

C:\Users\Public\Documents\Altium\AD15

To specify alternate locations, simply type the location directly into the applicable field, or click the folder icon at the right of a field and browse for the required destination folder.

Once the install locations are specified, click **Next** to continue.

**Ready To Install**

The Installer now has all the information it requires to proceed with the installation. If you need to change anything, click the **Back** button. If you want to cancel out of the install, click **Cancel**.
To go ahead and proceed with the installation, click **Next**.

**Installing Altium Designer**

The page will change from **Ready to Install** to **Installing Altium Designer** and the installation will proceed, with the required files first being downloaded (from the secure cloud-based Altium Vault). Progress of this download is displayed within the Installer itself.
Installation commences by downloading the required set of install files.

Once the download of files is complete, they will be installed, again with progress displayed in the Installer.
After the download is complete, the software is then installed.

**Installation Complete**

When the installation itself completes, you have reached the end of the Installer wizard.
An option is provided to launch Altium Designer when you exit the Installer, enabled by default. Simply click **Finish** to exit the wizard.

That's it. You have installed Altium Designer on your computer, and initially tailored its functionality to suit your design needs. Now it's time to do what you do best and go create that next innovative (and hopefully lucrative) product!

Within Altium Designer, the **F1** shortcut is definitely worth getting acquainted with. Hover the mouse over anything inside Altium Designer – a button, command, dialog, panel, or design object – and press **F1** to access documentation for it. **F1** also works for specific pages in the Preferences dialog, and specific rule constraint pages in the PCB Rules and Constraints Editor dialog. And if you're running a command, say you're placing a component and you're not sure how to rotate it, press **Shift+F1** for a list of command-dependent shortcuts.

**Importing Preferences from a Previous Version or Major Release**

After installing and launching your new version of Altium Designer, the Import Settings dialog will appear, from where you are given the opportunity to import preferences from the most recent previous installation at startup.
Import settings from your previous version's installation.

Not only will the preferences be imported, but the user settings will be copied across too - ensuring the new version opens with the same look and feel created in the previous version, even down to the last opened workspace and panel positioning!

If the option to import preferences on initial startup is not taken, don't worry - preferences can be quickly imported, at any time, from within the Preferences dialog. Simply click the Import From... button, at the bottom of the dialog. The associated menu will list all previous versions/releases currently installed on the computer, and that can be imported from.

Preferences can be imported from any individually installed version instance of Altium Designer (installed builds of AD10 or later), or any of the following pre-AD10 major releases of the software: Summer 09, Winter 09, Summer 08, and Altium Designer 6.

The benefit of importing at initial startup is getting the working environment looking exactly the way it was set up in the previous installation. Importing preferences at a later stage will only 'restore' a subset of this environment.

Post-Installation Management

For information on how to manage your instance of Altium Designer after initial installation, use the following links:

Licensing Altium Designer
Extending the Capabilities of Altium Designer

Installing Multiple Instances of the Software

Changing Installed Core Functionality after Installation

Updating to a Later Version

Uninstalling

Source URL: https://www.altium.com/documentation/ru/display/ADES/Installing+Altium+Designer