

製品マニュアル

Modified by Jason Howie on Apr 11, 2017



Altium Designer provides a central location from where you can set up various preferences across different functional areas of the software. These are global system settings that apply across projects and relevant documents.

Configuration of preferences is performed from within the [Preferences dialog](#) (DXP » **Preferences**). Use the controls and options available on the loaded page to configure your preferences for that area of the software as required. This could be a mixture of satisfying company policy, and your preferred working environment.

The *Preferences* dialog provides a number of useful tools to ensure your set of preferences is just as you require, including:

- Ability to import preferences defined in a previous instance, or version of the software.
- Ability to store your preferences in, and retrieve them from, the Cloud.
- Ability to save preferences to, and load preferences from, a DXP Preferences file (*.DXPPrf).
- Ability to set the options and controls on the active child preferences page, or all pages, back to their defaults.

And if you have an appropriately licensed Altium Vault, you can formally release your Altium Designer Preferences into a target Item (and revision thereof) in that Vault. Once the preferences set has been released, and its lifecycle state set to a level that the organization views as ready for use at the design level, the preferences can be reused across installations of the software.

Preferences are divided over the following categories:

- [System Preferences](#)

- [Data Management Preferences](#)
- [Schematic Preferences](#)
- [PCB Editor Preferences](#)
- [Text Editors Preferences](#)
- [Simulation Preferences](#)

Source URL: <https://www.altium.com/documentation/jp/display/ADES/Altium+Designer+Preferences>