

Altium Designer Initial Configuration

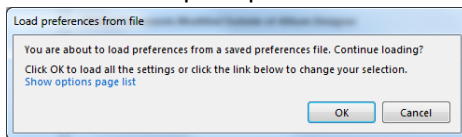
PURPOSE

The following steps should be completed the first time you use Altium Designer. These steps simplify the initial configuration of Altium and enable you to use the WWU-EE Templates. This process only needs to be completed once.

Prerequisite: The R: drive must be mapped for the follow process to work.

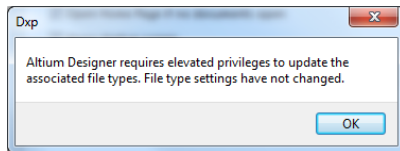
CONFIGURATION

1. Open Altium Designer.
2. In the top left menu select **DXP > Preferences**.
3. Click **Load...**
4. Load **AltiumWWUEEDXPPref.DXPPref** from **R:\Altium\Templates\Preferences and Options**
5. Click **OK** when prompted to continue loading preferences.



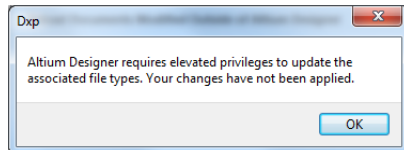
Click **NO** when prompted to allow Altium to make changes to the computer. (Admins can click YES)

Click **OK** to acknowledge Dxp prompt.

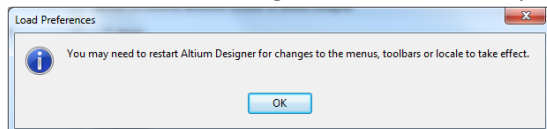


Click **NO** when prompted to allow Altium to make changes to the computer. (Admins can click YES)

Click **OK** to acknowledge Dxp prompt.



Click **OK** to acknowledge Load Preferences prompt.



6. Restart Altium Designer.
7. Verify new preference settings. Browse to **DXP > Preferences > Data Management > Vaults**. You should see the *Altium Content Vault* listed with a green check mark under status.

Name	Description	Address	Status
Altium Content Vault	Altium Content Vault	http://vault.live.altium.com	✓

8. Create a new PCB Project to verify templates work. Click **Project > Add New Project > PCB Project**. A new project should appear in the file browser on the left. You should see the following three files appear inside the structure of a new project.

