

SolidWorks Integration for Vault Professional 2014 - Readme

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Release-Notes - Readme-File
Autodesk, Inc.

This file contains information that became available prior to releasing SolidWorks Integration for Vault Professional 2014 or that you will find useful while using SolidWorks Integration for Vault Professional 2014. If you are experiencing problems not covered in this Readme, please visit the Autodesk Product Support Web site <http://support.autodesk.com>, or contact your dealer/reseller.

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System Recommendations

Operating Systems	
<i>32-bit Operating Systems</i>	<i>64-bit Operating Systems</i>
32-bit is not supported	Microsoft Windows 7 Business
	Microsoft Windows 7 Professional
	Microsoft Windows 7 Ultimate
	Microsoft Windows 7 Enterprise
Software	
SolidWorks 2012 64-Bit (SP5.0 or newer)	
SolidWorks 2013 64-Bit (SP4.0 or newer)	
Autodesk Data Management Server 2014 (18.1.8.0 Update 1 or newer)	
Autodesk Vault Professional 2014 Client (18.1.8.0 Update 1 or newer)	
Hardware	
Minimum Requirements: Pentium 4, Xeon, Athlon 64/Opteron, 1,6 GHz or higher 1 GB memory or higher 120 GB disk space	Recommended Requirements: Pentium 4, Xeon, Athlon 64/Opteron, 2 GHz or higher 2 GB memory or higher 120 GB disk space
Preferred Requirements: Pentium 4, Xeon, Athlon 64/Opteron, 2GHz 4 GB memory or higher 500 GB disk space	

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Installation Information

General Information

- SolidWorks Integration for Vault Professional 2014 requires Autodesk Data Management Server 2014 and is not compatible with prior versions of Autodesk Vault.
- Before installing SolidWorks Integration for Vault Professional 2014, it is recommended that you first close all running software
- The installation path must not contain XML reserved characters such as & (ampersand).

Installation Information

- SolidWorks Integration for Vault Professional 2014 could only be installed on machines with a required SolidWorks installation. The named versions of SolidWorks in System recommendation will be supported.
- SolidWorks Integration for Vault Professional 2014 consist of a Server- and a Client-Setup
 - Use "Autodesk VaultPro 2014 SolidWorks Add-In (Server).exe" for your Autodesk Data Management Server. The Server-Setup support systems with 64-bit.
 - Use "Autodesk VaultPro 2014 SolidWorks Add-In (Client 64-Bit).exe" on Vault Professional client 64-bit machines.

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Vault Professional 2014 SolidWorks Add-In

- SolidWorks Integration for Vault Professional 2014 honors the working folder setting (as defined in Vault Explorer) for the root Vault folder ('\$') only. Working folder settings for other vault folders are ignored. If you do not set the working folder for the root vault folder, the default working folder will be set when you first run SolidWorks after installation.
- Changes to working folder setting will not take effect in SolidWorks until SolidWorks Integration for Vault Professional 2014 is re-logged in (log out and then log in).
- Adding files to the Vault from outside the working folder is not supported in this release. All files (including SolidWorks library files) need to be inside the working folder in order to be added to the Vault.
- SolidWorks Integration for Vault Professional 2014 generates thumbnail images for preview. Therefore it is essential to add the required mapping in VaultPro.

You must have administrative access to perform these tasks:

1. Click **Tools > Administration > Vault Settings**.
2. On the Vault Settings dialog box Behaviors tab, click **Properties**.

3. In the Property Definitions dialog box, select the property **Thumbnail** for editing and click **Edit**.
 4. Select the Mapping tab to add a property mapping.
 5. Click in the * **(Click here to add a new mapping)** field beneath the Entity column.
 6. Select File in the entity type for this property mapping.
 7. Click in the field under Provider and select **SolidWorksPropertyProvider (*.sldprt;*.sldasm;*.slddrw)**.
 8. Click in the field under File Property and then click Import Properties
 9. Select whether you want to import properties from the vault or from a source file.
 10. On the Select File dialog, locate the file from which you want to import. Click Open.
The Import Properties list is automatically updated with the property **Thumbnail**.
 11. Click OK to create the mapping.
 12. On the Mappings tab, click **OK** to commit all changes and return to the Edit Property dialog box.
- For the detection of the SolidWorks document type you have to add another mapping. Do it exactly as thumbnail mapping only select the property **Type** in Step 3 and select **Component Type** in Step 10.
 - The following commands within the Autodesk VaultPro 2014 client are not supported for SolidWorks files:
 - Copy Design. SolidWorks offers the own functionality 'Save As Copy'
 - Update preview
 - **If SolidWorks is configured as the default viewer in Microsoft Windows for SolidWorks files, SolidWorks will be launched when using the View feature of VaultPro. Because the file being viewed is stored in a temporary folder and is not the original file, SolidWorks will display the file as "File is not in the workingfolder". It is highly recommended that files being viewed are not added to the Vault during this workflow.**
 - Drag & Drop of SolidWorks files from VaultPro to SolidWorks or a folder in Windows Explorer is not supported.
 - If you change the exclusion status of a part which is a child of a sub assembly (using the Exclude From BOM command in SolidWorks), the sub assembly must be checked out manually. Otherwise, the exclusion status change will not be reflected in the VaultPro BOM when the file is checked in.
 - Fileproperties will be committed from VaultPro to SolidWorks during the check-out process within SolidWorks.
 - If a component inside of an assembly is edited without being checked out, edit operations performed on the component may fail. To avoid this, check out the component before editing it.

- SolidWorks weld bead components will not be assigned to an item when using the assign item command in VaultPro. To avoid this, update the version in the Vault using the following steps:
 - save and re-open the assembly in SolidWorks
 - then check out and check in the assembly
- Weld bead components need to be reopened in their own window in order for AutoPublish to work during Add Files or Check In.
- If you get latest Revision of an assembly that is open and checked out, a warning dialog may pop up. This only happens if you have deleted a part or subassembly in the checked out file. Click [Yes] and the operation should complete successfully.
- A file that is checked out automatically when a design table is edited may encounter an issue when checking the file back into VaultPro. We recommend checking out the file containing the design table before editing it.
- SolidWorks may issue Out of Memory warnings when certain large assemblies are checked in with the Create DWF Attachment option turned on. The workaround is to check in the assembly by using a Jobserver. To set up a Jobserver, run the same client setup and activate the additional component **Jobserver** in the Setup Program.
- With the modul Jobserver all dialogs are automatically suppressed. This may NOT be installed on a standard Workstation.
- The DWF creation of SolidWorks drawing files will be done on the Printer "Autodesk DWF Writer for 2D". If this Printer is not installed, the DWF will be created on the VaultPro SolidWorks Add-in.
- If files have been renamed in the vault, it may happen that a message 'Wrong file references loaded' appears when a parent drawing is loaded in SolidWorks. The affected files get the file status 'New File'. In this case it is necessary to synchronized the local working directory with the current Vault by using the Vault function *Workspace Sync*.

What's New

- Suppressed and Lightweight loaded components are now supported on check-in, check-out, undo check-out and update properties.
- This version of the SolidWorks Integration for Vault Professional 2014 supports the auto ownership transfer feature.

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Autoloader

Install Information

- The Autoloader will be installed automatically on each client.

Autoloader Information

- Only users with the administrator role will be able to load files into the vault using Autoloader.
- Autoloader supports SolidWorks 2012 and newer data.

- Autodesk strongly recommends backing up all existing databases and necessary files PRIOR to running Autoloader.
- Autoloader requires SolidWorks 2012 or SolidWorks 2013 as well as Vault 2014 based products to be installed on the system if you wish to use Autoloader to upload SolidWorks files.
- It is recommended that you close all other programs when running Autoloader to avoid memory problems while uploading data.
- SolidWorks files may be uploaded to any folder in the vault.
- Autoloader copies all files to be uploaded to the vault to a temporary folder on the system that is running Autoloader. Therefore, enough free space must be available to accommodate the duplication of such files. If publishing dwf files enough space must also be available on the Jobserver to accommodate the dwf files as well.
- Autoloader cannot detect Derived or Virtual components when scanning, hence these will not be added to the Vault using Autoloader.

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Known Issues

Inside Vault Professional 2014 Client with SolidWorks files

- The Drag & Drop Function of SolidWorks files from VaultPro to SolidWorks or a folder in Windows Explorer is not recommended.

Inside SolidWorks

- Parents and maybe GrandParents are getting dirty if a Child of them has been renamed in via Vault Explorer. To resolve this you have to check out the dirty Files, save them and Check them in again.
- Parent relationships are not supported in the transactions check-in, check-out and undo check-out.
- If DWF files from SolidWorks drawings are created using the Printer "Autodesk DWF Writer for 2D" a message appears when you print. This message must be confirmed with [OK] to continue.

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