

2017 Workgroup PDM Vault Installation

TITLE:	2017 Workgroup PDM Vault Installation
DATE:	09/15/2016
SUBJECT:	Installation of 2017 Workgroup PDM Vault
ABSTRACT:	Comprehensive installation guide of the 2017 SOLIDWORKS Workgroup PDM Vault.



This guide is written for SOLIDWORKS 2017. If you are using an earlier version of SOLIDWORKS, please download our earlier installation documents to correctly load and administer your version of SOLIDWORKS.

SOLIDWORKS Workgroup PDM is used to manage design documents. A "vault" is a windows directory setup on a server that will store design data generated by your company. The software used to control this data runs as a service on your **Windows Server 2008R2, Windows Server 2012, or Windows Server 2012 R2 (Windows Server 2003, Windows 8, Windows 7, and Windows XP/Vista are not supported)**.

Files in the vault directory should **NEVER** be altered manually. Administrators must use the SOLIDWORKS **Workgroup PDM VaultAdmin** program to control the permissions and projects. Users must use any one of the vault client programs to add files or manage properties.

This document covers loading the **Vault** directory and service as well as installing the **Workgroup PDM VaultAdmin** program. See our Workgroup PDM Client Installation document for information on how to load these types of programs, which are used to access the vault data.

This is not a troubleshooting guide. If you have any technical issues with SOLIDWORKS, please contact [Hawk Ridge Systems technical support](#).

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SOLIDWORKS Workgroup PDM



SOLIDWORKS Workgroup is a suite of software products that include:

Workgroup PDM Vault

	SOLIDWORKS Workgroup PDM VaultAdmin Program
	SOLIDWORKS Workgroup PDM Vault Windows Service

A vault must be located on a shared **Windows 2008R2, Windows 2012, or a Windows Server 2012 R2 64-bit server**. The server computer must be on and the users must have access to the vault data directory for successful document management.

Workgroup PDM Clients

	SOLIDWORKS Workgroup PDM Client Add-In
	SOLIDWORKS Explorer with Client or Contributor Add-In



SOLIDWORKS Workgroup PDM Viewer License

The Workgroup PDM client is how users access the vault. Each user, regardless of client style, must have access to the machine running the vault service in order to access any data.

SOLIDWORKS Workgroup PDM Client add-in for SOLIDWORKS allows users full access to input, manage, change and maintain SOLIDWORKS files in the vault. **This is for all SOLIDWORKS users.** *SOLIDWORKS Professional and SOLIDWORKS Premium both include the client add-in.*

SOLIDWORKS Workgroup PDM Contributor add-ins allows users to manage data, change properties, and attach non-SOLIDWORKS files. **This is for non-SOLIDWORKS users.**

The contributor is purchased separately and can be licensed standalone or run from a SolidNetWork License Manager.

SOLIDWORKS Workgroup PDM Viewer allows read-only web access to the vault. **This is for viewing and printing only. It has no access to make changes.**

The viewer portal is loaded on to the vault server and uses Microsoft Internet Information Services (IIS) as the host. The client is purchased in bundles of 5 and uses Internet Explorer to access the service.

This document covers loading the **Vault** directory and service, as well as installing the **Workgroup PDM VaultAdmin** program. See our Workgroup PDM Client installation document for information on how to load these types of programs.

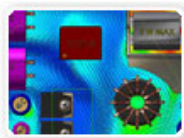
Before Installation

Hawk Ridge Systems highly recommends taking a moment before installing SOLIDWORKS to check that you are ready to load your new software. Follow these steps to have better performance using SOLIDWORKS:

1. Check hardware compatibility.

- Login to www.SOLIDWORKS.com, click the **Login** link at the top of the page and login to the customer portal. In the Self Service area, there are links to check [video card](#) compatibility as well as minimum [system requirements](#).

System Requirements



Research graphics cards, system requirements for SolidWorks products, and other hardware related topics.



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- You can also see Hawk Ridge Systems' recommended hardware guide by logging into <https://support.hawkridgesys.com/home> and reviewing our hardware recommendations. You can see brand name makes and models we suggest for best performance. We also have a more comprehensive hardware guide that discusses all types of performance considerations for your system.

2. Check operating system compatibility.

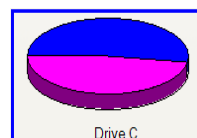
- You can find the latest version of the system requirements chart under the [system requirements](#) page of www.SOLIDWORKS.com.

Operating Systems	2015	2016	2017
Windows Server 2012 R2	✓	✓	✓
Windows Server 2012	✓	✓	✓
Windows Server 2008 R2, SP1	✓	✓	✓
Hardware			
RAM	2 GB or more recommended		
Disk Space	50 GB or more		
Processor	Intel® or AMD® processors		
Other			
Web Server Requirements	Microsoft Internet Information Services (IIS) 7.0 or later.		
API Trigger Requirements	Microsoft Message Queuing (MSMQ), Microsoft Message Queuing Triggers		
Virtual environments	Not Supported		

- Be sure you are on the latest version of your operating system. You will want to check for, and install any Windows updates before installing SOLIDWORKS.
- All clients will need network access to the vault server.
- Virtual servers are not supported.**

3. Check hard drive for space and overhead.

- In Windows Explorer, go to My Computer and check destination drive (by default the **C drive**). Right-click this drive and choose **Properties** to see if it has the "Estimated installation size" (4.5 GB in this case) free required for the full program (not including Simulation or PDM add-ins).
- Also, be sure this drive has at least **15%** free space. A full disk may cause out-of-memory crashes and cannot be defragged to optimize performance.



4. Confirm community's working version.

- SOLIDWORKS is not backwards compatible, meaning older versions cannot open files from newer versions. Be sure workmates, customers or vendors are all on the same version (SOLIDWORKS 2016 or SOLIDWORKS 2017) you are loading.

5. Backup your data.

- Ask a system administrator or existing user to see if there are any shared libraries, such as Toolbox or a data management vault. Toolbox libraries and PDM vault information should all be saved to backup locations. Then you can migrate the original libraries and vault to the newer version.
- If you have a shared location containing existing SOLIDWORKS files, it is recommended that you make backup copies of these files. Any templates or design libraries should also be saved to a backup folder.



6. Uninstall previous version of SOLIDWORKS.

- It is recommended that after you have saved out all shared data, you should fully uninstall any previous versions of SOLIDWORKS from your machine. After fully uninstalling SOLIDWORKS and its add-ins, you should install the new software.



Vault Installation

The vault installation will load **two** items onto your server:

C:\VaultData - A Windows directory that is stored on a network location.

Files in the vault directory should **NEVER** be altered manually. Administrators must use the SOLIDWORKS **Workgroup PDM VaultAdmin** program to control the permissions and projects. Users must use any one of the vault client programs to add files or manage properties.

This directory can be located on any network drive but it is **highly recommended** that it be run on the same physical machine as the service.

SOLIDWORKS Workgroup PDM Server -The vault server is a service that runs on a **Windows Server 2008R2, Windows Server 2012 or a Windows Server 2012 R2 server**. The server computer must be on and the users must have access to the vault data directory for successful document management.

Users access the vault through the Workgroup PDM client program. Each user, regardless of client style must have access to the machine running the vault service in order to access any data.

New Users:

If you do not have an existing vault, go to the section on [creating a vault](#) section of this document.

Existing Users:

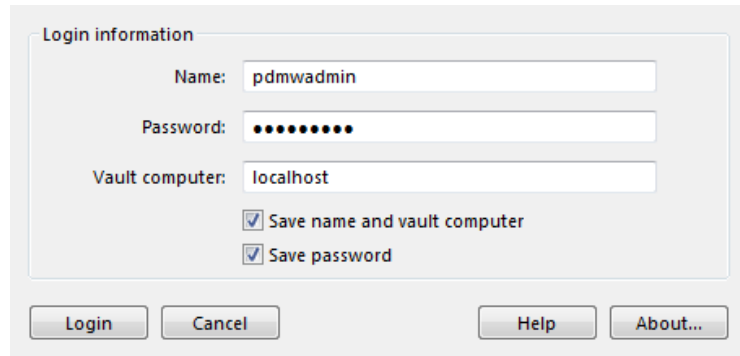
If you are upgrading an older vault to 2017, you will want to continue on with the instructions on managing an existing vault.

Managing an Existing Vault

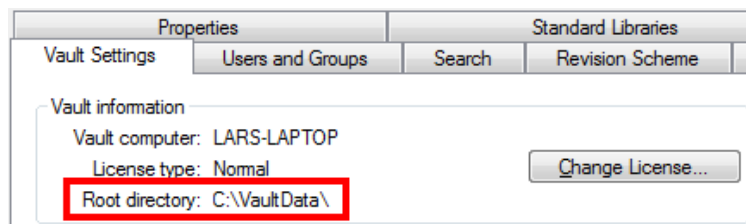
If you are upgrading an older vault to 2017, you will want to first have all users close out of the vault and stop the service so you can create a backup copy of the vault to ensure you have no loss of data.

1. Start the Workgroup PDM VaultAdmin on the server. Go to **Start, All Programs, SOLIDWORKS20XX, SOLIDWORKS Workgroup PDM Server** or **PDMWorks Workgroup Vault Admin** if 2008 or older.

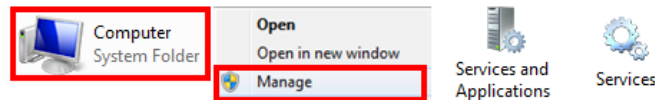
2. Enter an **Administrator** level name and password. Then press **Login**.



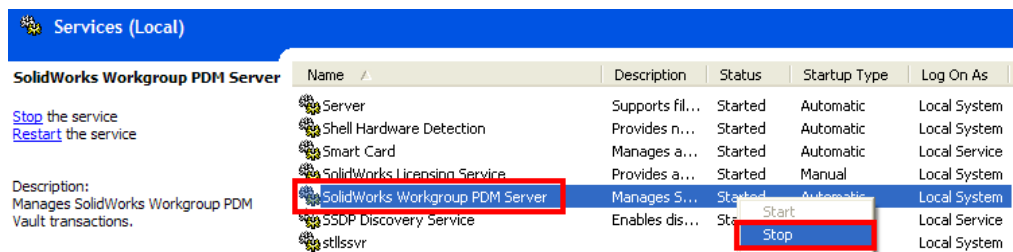
3. On the Vault Settings tab, verify vault location listed as Root directory; press **OK** to close the tool.



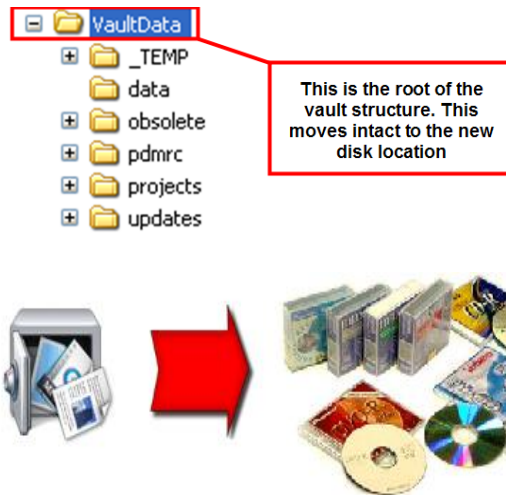
4. In Windows Explorer, right-click **Computer**, choose **Manage** from the popup menu, go to Services and Applications > Services.



5. Find the **SOLIDWORKS Workgroup PDM Server** (or **PDMWorks Workgroup Server** if 2008 or older) service by right-clicking on this item in the list (they are alphabetically listed) and selecting **Stop** from the popup menu.



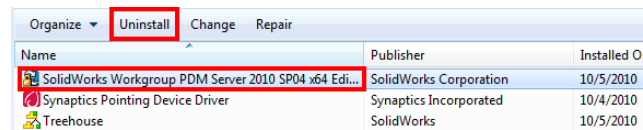
6. Now you can backup your files. It is recommended that you make a copy of your entire root directory folder (**C:\VaultData** default) at this time. It is also a good idea to make a hard copy backup of your vault location on disk or other medium at this time.



7. (Skip this step if you are just adding a service pack to 2017.) If upgrading from an older version, uninstall the existing vault server and client software from the server. From the **Control Panel** choose **Uninstall a program**.



8. From the list of programs, find SOLIDWORKS Workgroup PDM Server. Press the **Uninstall** button. If you have older Workgroup Clients or SOLIDWORKS Explorer on the server **REMOVE** this also. **This will not delete the vault data.**



9. You can now load the new 2016 vault service to run with your root directory (C:\VaultData by default) following the instructions in the [Creating a Vault](#) on the following page. Use the same root directory path from step 3 (**C:\VaultData** by default) to upgrade your vault.
10. You will want to upgrade the client software for the users using our document on Workgroup PDM Client Install.
11. You will also want to then update the software on the client machines and use one of the updated clients and update their vault data as instructed in the Workgroup PDM Vault Maintenance document.

Creating a Vault

All SOLIDWORKS installations should be handled through the SOLIDWORKS Installation manager. This can be started by inserting a SOLIDWORKS DVD, or using the **sldim.exe** from a web download found in the **SOLIDWORKS/Sldim** folder.

1. On the welcome page you will be asked for the type of installation you would like to perform. Choose **Server products**; this will expand a list of these types of programs. Select **Install SOLIDWORKS Workgroup PDM Server (vault) on this computer**.

Server products

Install **SOLIDWORKS PDM Server** components on this computer.

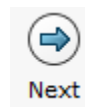
Install **SOLIDWORKS Workgroup PDM Server (vault)** on this computer.

Important information regarding SOLIDWORKS Workgroup PDM



SOLIDWORKS Workgroup PDM support will end after SOLIDWORKS 2017.
Contact your Value Added Reseller for more information and planning vault migration to the new SOLIDWORKS PDM Standard product or to the SOLIDWORKS PDM Professional product (formerly SOLIDWORKS Enterprise PDM).

Install **SolidNetWork License Manager** (distributed licensing) on this computer.

2. Press the **Next** button to continue.



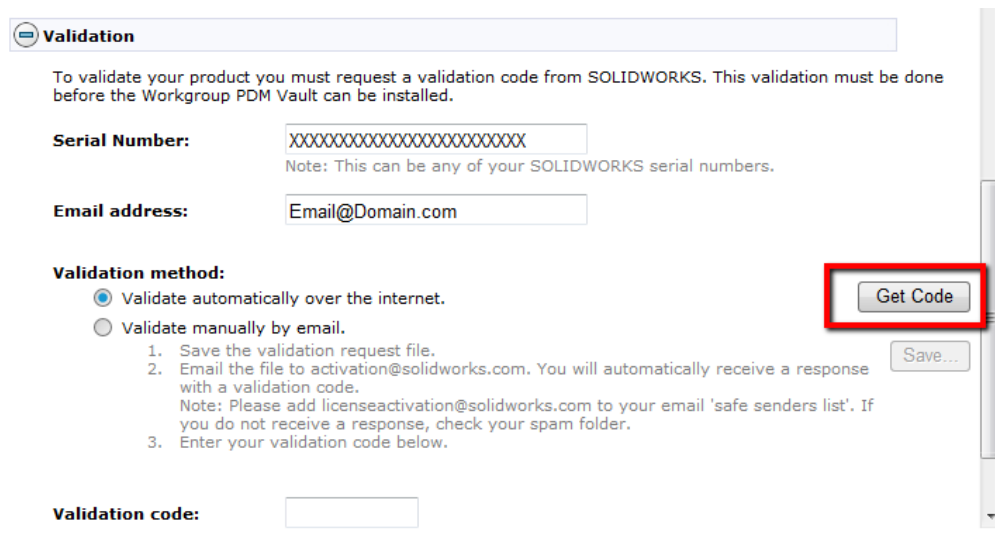
3. The Summary page will show you the installation location and the products to be installed under the SOLIDWORKS Workgroup PDM Server heading. Download Options allow you to change the location of the source files. Click the **Change** link on either of these headings to modify these settings.

SolidWorks Workgroup PDM Server - Validation Code required for installation		CHANGE 
Installation location:	C:\Program...\SolidWorks Corp\SolidWorks Workgroup PDM	
Products to install:	Workgroup PDM Vault and API	
Vault Data location:	C:\VaultData	
Viewer license key:	None	
Validation code:	Required for installation. Click Change to enter validation code.	
Download Options		CHANGE 
Operation:	Install only (do not download)	
Background downloader:	Do not use	

The vault data location is where your actual data (files & metadata) is going to be stored.

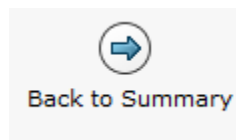
- If you are upgrading an existing vault, be sure this is the same location you were using originally.

- If you are loading for the first time, the vault data can reside at any networked drive. Though it is **HIGHLY RECOMMENDED** that this location reside on the same physical server as the PDM service is being loaded under **Installation location**.
4. You will need to obtain a **Validation Code** to install your vault. Click the **Change** link to the right of **SOLIDWORKS Workgroup PDM Server** and you will be brought to a screen where you can obtain that code. Enter your email address and choose a validation method. Your serial number should automatically be populated.
- If you wish to **Validate automatically over the internet**, click **Get Code** to automatically retrieve your validation code.
 - If you wish to **Validate manually by email**, please follow the on-screen instructions to obtain your validation code.



The screenshot shows the 'Validation' screen in SolidWorks. It includes a title bar with a back arrow and the word 'Validation'. Below the title bar is a paragraph: 'To validate your product you must request a validation code from SOLIDWORKS. This validation must be done before the Workgroup PDM Vault can be installed.' There are two input fields: 'Serial Number' with a placeholder of 'XXXXXXXXXXXXXXXXXXXXXXXXXX' and a note 'Note: This can be any of your SOLIDWORKS serial numbers.'; and 'Email address' with a placeholder of 'Email@Domain.com'. Under 'Validation method:', there are two radio buttons: 'Validate automatically over the internet.' (selected) and 'Validate manually by email.' (unselected). Below the second radio button are three numbered steps: 1. Save the validation request file. 2. Email the file to activation@solidworks.com. You will automatically receive a response with a validation code. Note: Please add licenseactivation@solidworks.com to your email 'safe senders list'. If you do not receive a response, check your spam folder. 3. Enter your validation code below. At the bottom left is a 'Validation code:' label and an empty input field. On the right side, there is a 'Get Code' button highlighted with a red rectangle, and a 'Save...' button below it.

5. Once the validation code has been entered, press the **Back to Summary** button to continue.



6. Press the **Install** (or **Download and Install**) button to continue.

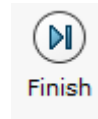


7. The vault will now load onto your system.



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8. Now that the vault server has been loaded, you have the options to open the "What's New" PDF, or login to the website and comment on the installation interface.
9. You can choose to join the Customer Experience Program if you wish. This will periodically send fault and usage information via e-mail to SOLIDWORKS to help improve the product. This will not affect the performance of SOLIDWORKS on your computer. The logging takes place whether you choose this or do not. The only difference is whether SOLIDWORKS sends the log files to development to process the information.
10. Press the **Finish** button to complete the installation.



New and existing users will want to continue to the next section at this point to load the Vault Admin Tool to be able to manage the vault settings.

New Users will want to begin installing the client software for the users to access the vault as instructed in our [Workgroup PDM Client Install](#) document.

Existing users will now want to update the software on the client machines and use one of the updated clients to update their vault data as instructed in the [Convert Files After Upgrade](#) section of this document.

Loading Vault Administrator

The SOLIDWORKS Workgroup PDM VaultAdmin program is a tool that allows administrators to control vault settings and user access. It is also the tool for setting up the revision scheme, lifecycles and other vault options. It should be loaded on the server as well as on one or more client stations for successful vault management.

1. Insert your DVD or start the **sldim.exe** from a download. On the welcome page, for the installation type, choose **Individual** (choose **Modify** if you already have SOLIDWORKS loaded on this machine).

Welcome to the SOLIDWORKS Installation Manager

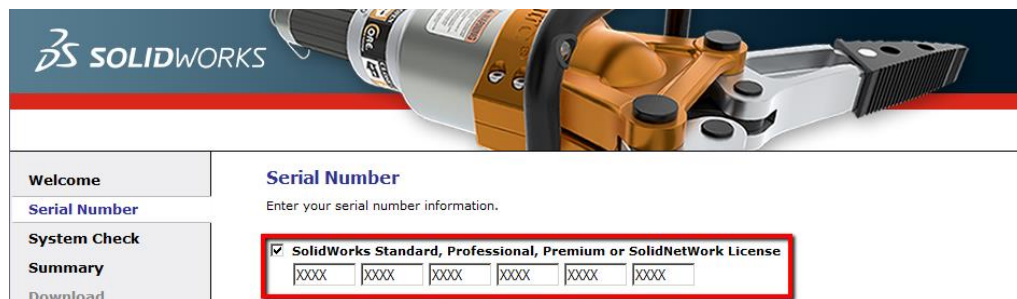
Specify the type of installation:

- Individual (on this computer)
- Administrative image
- Server products
- Download and Share all files. Create individual installs or administrative images on multiple machines with a single download.

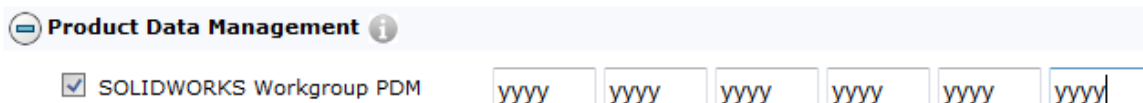
2. Press the **Next** button to continue.



3. Enter your SOLIDWORKS Professional or SOLIDWORKS Premium serial number on the following page.



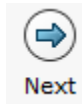
If you purchased Workgroup PDM without either one of these versions of SOLIDWORKS, check the option **"if you have other serial numbers from SOLIDWORKS Corporation enter them here"**. Then enter your SOLIDWORKS Workgroup PDM serial number.



These may appear already, being read out of the registry of your current vault if upgrading.

Your serial number and installation will be verified at this time.

4. Press the **Next** button to continue.



5. If this is a new version of 2017, you will be taken to the Summary page showing the products you will install. Click the **Change** link to the upper right of the product list to be taken to the product selection page.

If this is a modification of an existing installation, you will be automatically taken to a product selection page.

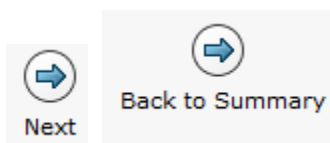
If you are loading on the server and only need to load the SOLIDWORKS Workgroup PDM VaultAdmin component, you can de-select all other SOLIDWORKS programs.

- SolidWorks Explorer/Workgroup PDM (159 MB)
- SolidWorks Explorer (252 MB)
 - SolidWorks Workgroup PDM Client Add-in (198 MB)
 - SolidWorks Workgroup PDM VaultAdmin (32 MB)
 - SolidWorks Workgroup PDM x64 Client Add-in (89 MB)

If you are loading a client, you can load the Explorer and the PDM Client Add-in (or x64 Client depending on your operating system) with SOLIDWORKS or you can modify the existing SOLIDWORKS installation to load with this add-in.

- SolidWorks Explorer/Workgroup PDM
 - SolidWorks Explorer
 - SolidWorks Workgroup PDM Add-in
 - *SolidWorks Workgroup PDM Client Add-in
 - *SolidWorks Workgroup PDM VaultAdmin (38 MB)
 - SolidWorks Workgroup PDM x64 Client Add-in

6. Press the **Next** button or **Back to Summary** to go to the Summary page showing products and installation locations.





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You will be shown a list of products you are going to install and their locations. If you wish to modify any of these items, you can press the **Change** link for any line item.

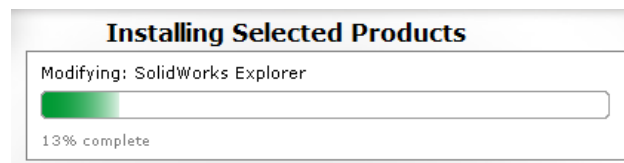
+ Download Options	CHANGE
+ Installation Location	CHANGE

7. Press the **Install Now** or **Modify Now** to continue.



If you are using a network SOLIDWORKS serial number, you will be asked to specify the location of the license server at this time.

8. The selected software will now be installed on this machine.



9. The selected software is now installed on this machine. You have the options to automatically check for service pack updates periodically, open the "What's New" PDF, or login to the website and comment on the installation interface.
10. You can choose to join the Customer Experience Program if you wish. This will periodically send fault and usage information via e-mail to SOLIDWORKS to help improve the product. This will not affect the performance of SOLIDWORKS on your computer. The logging takes place whether you choose this or do not. The only difference is whether SOLIDWORKS sends the log files to development to process the information.
11. Press the **Finish** button to continue.

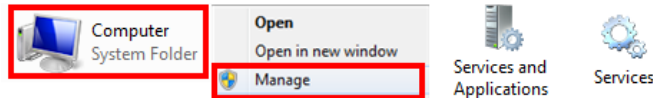
New Users will want to begin installing the client software for the users to access the vault as instructed in our Workgroup PDM Client Install document.

Existing users will also want to update the software on the client machines and use one of the updated clients and update their vault data as instructed in the Workgroup PDM Vault Maintenance document.

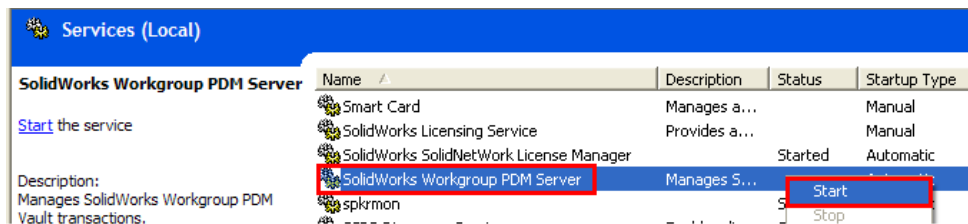
Check the Vault

Verify the service is started.

1. From Windows Explorer right-click **Computer**, choose **Manage** from the popup menu and go to Services and Applications > Services.



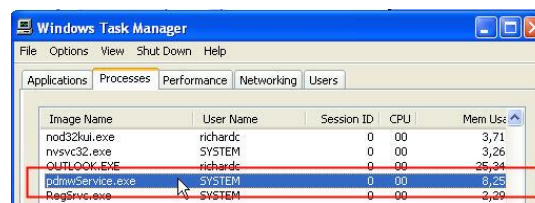
2. Search for SOLIDWORKS Workgroup PDM Server. Right-click on **SOLIDWORKS Workgroup PDM Server** and select **Start**, if available.



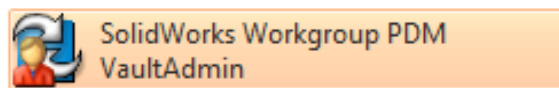
The vault can take 15+ minutes to start if it contains a lot of data. You will not be able to login during this time. You can tell when the service is complete as pdmwService.exe finish logging into the VaultAdmin tool.

Check that the Vault is Operating.

1. Check for the **pdmwService.exe** in the Windows Task Manager under the Processes tab.



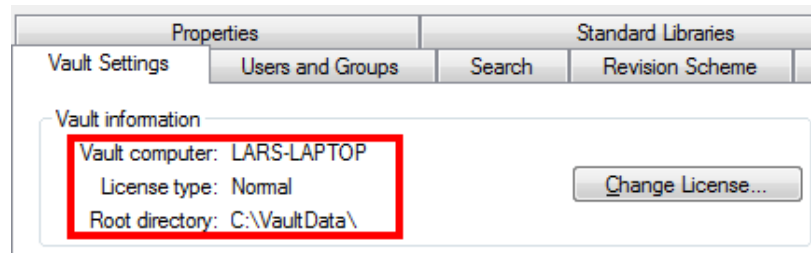
2. Launch the Vault Admin tool to verify connectivity.





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3. The vault will need a password to allow access. Login to VaultAdmin. Default username is "**pdmwadmin**". Default password is "**pdmwadmin**". Default vault computer is **localhost** if you are on the server, or the name of the server.
4. Verify the vault info: the vault computer name (clients will need this); license type (normal or SOLIDWORKS Workgroup PDM Viewer); root directory on the server (should not be shared to users through Windows Explorer).



Convert Files After Upgrade

After you have upgraded a vault from a previous version (i.e. moving from 2011 to 2012), it is recommended you upgrade the files in your vault to bring them up to the new version without changing the revisions of the files themselves.

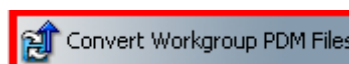
This operation uses the Task Scheduler tool. You must run this from a user's client machine that has full administrative access to the vault; **do not run this from the vault server**. The files will be converted one at a time on this machine, so be sure the machine has memory and hardware requirements for SOLIDWORKS. All converted documents are opened in resolved (not lightweight) state. **The computer must have enough memory to handle the resolved components.**

The update process may take hours or days depending on the amount of information you have in the vault. This is best done afterhours or on the weekend when there is no vault activity. **The vault is locked automatically during the conversion process.**

Hawk Ridge Systems recommends upgrading only the most recent version of files in the vault. (See step 7 regarding setting the task options.)

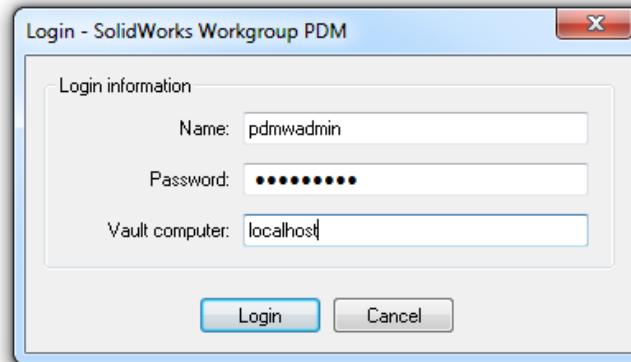
For large vaults, it is recommended to update project by project rather than attempting to upgrade the entire vault in one task. You can also setup several workstations to upgrade different projects to reduce the overall upgrade time. When using several workstations, you should login to all workstations using the same user information.

1. Go to **Start > All Programs > SOLIDWORKS20XX > SOLIDWORKS 20XX SP X.X > SOLIDWORKS Tools > SOLIDWORKS Task Scheduler**.
2. In the side bar, along the left side of the program, select the **Convert Workgroup PDM Files** from the list of tasks.





3. You will be asked for a login to SOLIDWORKS Workgroup PDM. **Only administrators are allowed to convert files.** Login with an **administrator** level name and password.



4. In the Convert Workgroup PDM, select either of the following:
 - **Update entire Vault.** All files in the vault will be updated.
 - **Update by project.** Specify projects and file types as described in the following section.
5. If you are **Updating entire vault** continue on to step 6. If you have chosen **Update by project**, set the following options:
 - Press the **Add Project** button to begin a list of projects to update. Select a project from the list in the **Workgroup PDM Projects** dialog box and press **OK**. The name of the project appears in the **Project** column. Repeat to add more projects.
 - Only projects to which you have read/write access are displayed. Ask your administrator to enable read/write access to the projects you need to update.
 - Check the **Include subprojects** box to include documents in subprojects in the update operation.
 - To remove a project from the **Project** column, select the project and click the **Delete** button.
 - **File Name or Type**- To filter the files in a project based on type, select a type from the list for each project. The choices are parts (*.prt,*.sldprt), assemblies (*.asm,*.sldasm), drawings (*.drw,*.slddrw), all SOLIDWORKS files (*.sldprt,*.sldasm,*.slddrw) or all files (*.*)

If a selected project uses standard library files from a shared location, these files must be accessible to the computer on which you are running the update.

6. Now you need to determine when the task will run. Set Task Schedule.
 - Under **Running Mode**, set the frequency **Once, Daily, Weekly** or **Monthly**.

- Set the **Start Time**.
 - Set the **Start Date**.
7. Press the **Options** button to determine the following update parameters. Press **Apply** and **OK** after setting these parameters to set how your update will behave.
- If **Latest revision only** is selected, then only the most recent revisions in the selected projects or vault are updated. (This cuts down update time considerably and is recommended).
 - If **Latest revision only** is not selected, then all revisions in the projects or vault are updated.
 - Additional file checking- If **check if files can be reopened after update** will double-check the converted files, but will also greatly increase update time.
8. In the dialog box, select from the following options and then press **Apply** and **OK**:
- **Task working folder**- Type the name of the folder where you want to store task working files, or click **Browse** and browse to the folder. Task working files can include backup files, reports and so on.
 - **Time out (seconds)**- Type the number of seconds for the time-out duration. SOLIDWORKS Task Scheduler ends the task after the task runs for this amount of time, whether or not the task completes.
 - **Backup task files**- Select to save backup files to the **Task working folder** before the task is performed.
- If the system is saving a document when the time-out duration is reached, the save operation completes before the task terminates.**
9. Press **Finish** to complete setting the conversion task. A dialog will show how many files are scheduled for update and an estimated time for completion. If you wish to continue with this, press **Yes** and the task will be set. Once a task is scheduled it will be added to the task list.

		Title	Date	Time	Status	Progress
1		Convert Files	4/22/2009	12:00:00 PM	Completed - Subtasks Generated	7 of 7
2		Dissect Files	4/22/2009	4:00:00 PM	Scheduled	

When you create a task, you click **Advanced** in the task properties dialog box to change the task options

At the scheduled start time, the task is initiated. If the task acts on more than one file, a sub-task is generated for each file. If you selected **Backup task files** in **Task Options**, a backup copy of each file is saved to a **.zip** file. Then each file is updated. When the task completes, the status changes to **Completed**.



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You can expand a task to show subtasks by pressing the "+" symbol.

	Convert Files	4/22/2009	12:00:00 PM	Completed - Subtasks Generated
+	Subtask 1 of 7 - convert file [Column Lamp.SLDPR1]	4/22/2009	12:00:00 PM	Completed
	Subtask 2 of 7 - convert file [Shade Frame.SLDPR1]	4/22/2009	12:00:00 PM	Completed
	Subtask 3 of 7 - convert file [Socket.SLDPR1]	4/22/2009	12:00:00 PM	Completed
	Subtask 4 of 7 - convert file [SW3dPS-bulb-bayonet]	4/22/2009	12:00:00 PM	Completed
	Subtask 5 of 7 - convert file [Shade Cloth.sldprt]	4/22/2009	12:00:00 PM	Completed with warning
	Subtask 6 of 7 - convert file [Lamp Shade.SLDASM]	4/22/2009	12:00:00 PM	Completed
	Subtask 7 of 7 - convert file [Column Lamp.SLDASM]	4/22/2009	12:00:00 PM	Completed

A report of the operation can be viewed by pressing the **status description**. If the "**Completed with warning**" status is pressed from the previous example, a web page will give a report like this.



SolidWorks Task Scheduler Report

General Information

Task Title: Subtask 5 of 7 - convert file [Shade Cloth.sldprt]
Task Type: Convert Files
Task File Path: F:\Examples\Lamp\Shade Cloth.sldprt
Application Path: C:\Program Files\SolidWorks Corp\SolidWorks\sldworks.exe
Scheduled Time: 4/22/2009 12:00:00 PM
Actual Start Time: 4/22/2009 12:00:44 PM
Finish Time: 4/22/2009 12:00:46 PM
Task Status: **Completed with warning**

Comments: **WARNING: Out of Context external reference (c:\documents and settings\lars\desktop\lamp\lamp shade.sldasm).**
WARNING: Out of Context external reference ().

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