SolidWorks Installation and Administration Guide

3DS **Solid**Works

Contents

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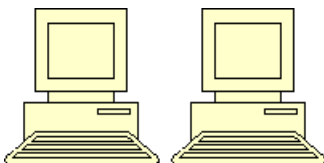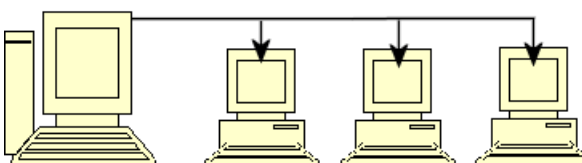This guide describes installation, licensing, upgrading and administration of SolidWorks products for both individual installations and multiple installations.

Individual Installations	Deploying Multiple Installations
Installation Installation on one or a few computers.	**Installation** Creation of an administrative image that is deployed to a number of client computers.
Licensing License activation or network licensing.	**Licensing** License activation or network licensing.
Upgrades Moving individual computers to a new release.	**Upgrades** Moving multiple computers to a new release by updating the administrative image used to install on those computers.
Administration Service packs, modifying, and so on.	**Administration** Network licensing administration, services, Workgroup PDM, vault administration, and so on.

SolidWorks Installation Manager enables you to configure settings for SolidWorks product components, including SolidWorks, SolidWorks Toolbox, PhotoView 360, Workgroup PDM, and SolidWorks SolidNetWork License Manager.

Workgroup PDM is project data management software that runs inside the SolidWorks environment or as a standalone application inside SolidWorks Explorer. Workgroup PDM controls projects with procedures for check out, check in, revision control, and other administration tasks.

SolidNetWork License Manager supports multiple license clients by distributing licenses to clients on the network. This allows the number of users to exceed the number of licenses by using floating licenses.

Individual Installation or Administrative Image

Should you install SolidWorks products on each computer individually, or should you create an administrative image and deploy the installation to all clients with one command?

If you are installing on only one computer, or if you manage dozens or hundreds of computers, the choice is obvious. For situations in between, consider the following comparisons.

Individual installation	Administrative image
Users can do their own SolidWorks administration.	The system administrator can customize multiple installations using the Option Editor, specifying installation options such as creating a new installation or updating an existing one, running the installation as an administrative user, running external commands before or after the installation, different product groupings, and so on.
The computers must be physically accessible to the person performing the installation.	Client computers can be distantly located as long as they are accessible from the same network.
You can install different versions, service packs, or products on different computers, with independent settings.	Using the Option Editor, administrative images can maintain consistency in versions, service packs, products, and settings for multiple installations.
The original source media (DVD) is required when applying service packs.	Media drives are not required when applying service packs.
The computer does not need to be on a network.	A local network connection is required to deploy the application; media drives are not required on client computers.
	Clients can be updated automatically. All clients deployed from the same administrative image are at the same service pack. To maintain multiple versions, you can create multiple administrative images.
	License activation can be automated.
	An administrative image requires additional disk space on the server; clients require the same space as individual installations.

Initiating SolidWorks Installation

You can initiate SolidWorks installation from loading a DVD on the local computer, accessing a shared installation directory, or downloading from SolidWorks.

To initiate SolidWorks installation, do one of the following:

- To install on a local computer from disk, insert the appropriate disk into the computer's disk drive.
- To install from a download directory, navigate to the directory and double-click `setup.exe`.
- To install by downloading from SolidWorks:

1. Browse to the SolidWorks Web site.
2. Login as a Subscription Service Customer, using your email address and password, or your SolidWorks serial number.
3. Under **Self Service**, click **Download Software and Updates**.
4. Under **Download Software**, select the SolidWorks version and service pack to download.

Individual Installations

Installation on Individual Computers

You can install SolidWorks on an individual computer using several installation methods.

The SolidWorks Installation Manager performs the installation, tailoring the installation to the products you have purchased. Before you start the installation:

- Have your SolidWorks serial number available.
- Verify that you have a working internet connection.

SolidWorks Installation Manager requires various Microsoft components, including Microsoft Windows Installer, Office Web Components, Microsoft Visual Studio Tools for Applications, and .NET Framework. If the correct versions of all Windows prerequisite components are not installed on the computer, SolidWorks Installation Manager installs them automatically prior to installing SolidWorks products.

Installation Method	Summary of the Process
Using the SolidWorks installation media	Insert the appropriate DVD disk into the computer disk drive.
	The SolidWorks installation media provides separate DVDs for 32-bit and 64-bit Windows installations.
Using a download directory on a server	In the download directory, double-click `setup.exe`.
	The download directory must be accessible from the computer on which you are installing SolidWorks products, and it must contain all of the SolidWorks installation files before installation can begin. Installation Manager can help you find missing files if you have internet access and write access to the download directory.
	This is not an administrative image. The only way to install a client computer from an administrative image is by clicking the link in the HTML page distributed by the SolidWorks administrator.
Updating an existing SolidWorks installation using Check for Updates	Do one of the following: • In SolidWorks, click **Help** > **Check for Updates**. • In Windows, click **Start** > **All Programs** > **SolidWorks Installation Manager** > **Check for Updates**. In either of the previous methods, you can schedule a regular check for updates by selecting **Check for Updates every _X_ days** and then specifying a value between 1 and 99.
	You must have a working internet connection. Workgroup PDM clients detect Vault updates automatically. You do not need to download a service pack to update clients. For details, see _Update Clients_ in the _Workgroup PDM online help_.

Installation Method	Summary of the Process
Downloading from the SolidWorks customer Web site.	1. Browse to the SolidWorks Web site. 2. Login as a Subscription Service Customer, using your email address and password, or your SolidWorks serial number. 3. Under **Self Service**, click **Download Software and Updates**. 4. Under **Download Software**, click the SolidWorks version you wish to install. You must have a working internet connection. You must be a Subscription Service Customer with login access to the Customer Portal site.

When the installation is complete, start the SolidWorks application and activate the license.

Administration of Individual Installations

Typically, administration of individual installations consists of modifying, repairing, removing, and upgrading installations.

Modifying an Installation

You can modify an installation to install products that have not been installed or to remove products.

If you installed SolidWorks and purchased an add-in at a later time, you can modify the installation to include the new add-in. You must install the add-in on each individual machine.

Workgroup PDM is not installed if you are modifying your original SolidWorks package to include SolidWorks Professional or SolidWorks Premium.

1. Make sure no SolidWorks sessions are active.
2. In Windows, open the Control Panel and run **Add or Remove Programs**.
3. Select the SolidWorks installation you want to modify and then click **Change**.
4. On the Welcome screen, ensure that **Modify the individual installation (on this computer)** is selected and click **Next**.
5. On the Serial Number screen, verify that the appropriate serial numbers are specified and click **Next**.
6. On the Product Selection screen, select the products to install or remove, and then click **Next**.

   When modifying an installation:

   - No products are installed or removed by default, even if you specify a new serial number. You must change the installation action of each product or feature.
   - To view the projected modification action for a particular component, click the component name (but not the check box). The intended modification action appears in the information field below the product listing.

- To see the available features for each product, click **+** next to that product and then click the icon next to the product or feature to change the installation actions.
- When you change a product selection from the initial specification, an asterisk (*****) appears next to that product component in the listing.

7. On the Summary screen, click **Modify Now**.
8. On the Installation is Complete screen, click **Finish**.

Repairing an Installation

If you encounter problems with an installed SolidWorks Product, you can use the SolidWorks Installation Manager to repair the installation.

> You can repair an individual installation. You cannot repair an administrative image; you must uninstall and then reinstall the administrative image.

1. Click **Start** > **Control Panel** > **Add or Remove Programs**.
2. Select the SolidWorks version you wish to repair and click **Change**.
3. On the Welcome to SolidWorks Installation Manager screen, click **Repair this installation.**
4. On the Products to Repair screen, select the product to repair.
5. Clear the rest of the products on the screen.
6. Click **Repair**.

Rolling Back an Individual Installation to a Previous Service Pack

To roll back an individual installation to a previous service pack release in the current major release family, uninstall the service pack version currently installed on the computer and then re-install the earlier version.

Removing an Installation

You can remove an installation on an individual computer or in a client/server environment.

1. Make sure no SolidWorks sessions are active.
2. If you wish to transfer the SolidWorks license to another computer, transfer the license to that computer before removing the SolidWorks installation on this computer.

> For information about transferring SolidWorks licenses, see Transferring a License on page 56.

3. In Windows, open the Control Panel and double-click **Add or Remove Programs**.
4. In Add or Remove Programs, select the product component to remove.

> In some cases, you might find multiple SolidWorks Service Pack components for a given major release version. To remove a SolidWorks major release, remove all Service Pack elements for that major release in Add or Remove Programs.

5. Click **Remove**.
6. In the SolidWorks Installation Manager, on the Products to Remove screen:

   a) Select the products to remove.

🖉 By default, all product components are selected.

b) Clear the products you do not want to remove.
c) Click **Remove Items**.

Upgrading to a New Release

There are several methods for upgrading an individual installation to a new SolidWorks release.

You can upgrade to a new release when you:

- Receive new installation media
- Are notified of upgrades when starting SolidWorks
- Check for updates manually
- Start a new instance of Installation Manager

When installing a new major release, you can upgrade a previously installed major release or install the new release in parallel. If you are installing a Service Pack release for a previously installed major release, that release is updated automatically.

Checking for Updates Manually

You can check for updates for a currently installed SolidWorks release using one of these methods.

- In SolidWorks, click **Help** > **Check for Updates**.
- In Microsoft Windows, click **Start** > **All Programs** > **SolidWorks Installation Manager** > **Check for Updates**.

You can also schedule a regular check for updates. Using either method, in the dialog box that displays, select **Check for Updates every *X* days**, and specify a value, where *X* can be between 1 and 99.

Upgrading Workgroup PDM Clients

Workgroup PDM clients detect when the Vault has been updated. You do not need to download a service pack to update clients.

For instructions, see *Update Clients* in the *Workgroup PDM online help*.

Upgrade Caveats

Products that can have multiple installations with different major versions do not always support major upgrades, or the ability to upgrade between major versions as a single operation.

In this case, the Installation Manager simulates the upgrade by first installing the new version, then removing the old one. At this time, the core SolidWorks product and DWGeditor support major upgrades, while the other products do not.

The consequences of this are:

- When upgrading the core SolidWorks product and DWGeditor between major versions:

  - Your existing installed locations do not change.
  - The upgrade can be rolled back if an error occurs or the you cancel during the upgrade.

- When other products are upgraded between major versions:

- Each product is first installed as a new product to a new location, then the original installation is removed.
- Once started, the upgrade cannot be rolled back if an error occurs or if you cancel during the upgrade.

Product Coexistence Rules

These product coexistence rules determine how new products are installed and existing products are upgraded.

Rule	Examples
Multiple major release versions of a core SolidWorks product are allowed, regardless of service pack or major release (with the exception of the three products mentioned below).	SolidWorks 2009 SP3.0, and SolidWorks 2010 SP0.0 can coexist on the same computer. However, if SolidWorks 2010 SP0.0 is installed on a computer, and you install SolidWorks 2010 SP1.0, Installation Manager replaces the SP0.0 version with the new SP1.0 version. SolidWorks 2010 SP0.0 and 2010 SP1.0 will not coexist on the same computer.
Only one version of the following products is allowed on the computer: • DWGeditor • Workgroup PDM Server • SolidNetWork License Manager	If SolidWorks 2010 SP0.0 is installed, DWGeditor is upgraded to 2010 SP0.0, even if an earlier major release of SolidWorks remains on the computer.
Only one version of an included product can be installed for any major version.	If SolidWorks 2010 SP0.0 is upgraded to 2010 SP1.0, SolidWorks Explorer 2010 SP0.0 will automatically upgrade to 2010 SP1.0. However, SolidWorks Explorer 2009 SP4.0 can also exist on the computer.

Multiple Installations

Deploying to Multiple Clients

You can install the SolidWorks product on multiple clients by creating an administrative image and then deploying the installation to clients.

The process for deploying to multiple clients is:

1. Create an administrative image.
   - Creating an Administrative Image from the Installation Manager on page 14
   - Creating an Administrative Image from the Command Line on page 19
2. Deploy the administrative image to clients.
   - Deploying an Administrative Image to Clients Through Email on page 19
   - Installing from the Administrative Image Using the Command Line on page 28
3. Implement licensing.
   - Administering Individual Licenses on page 55
   - Administering Licenses Using a SolidNetWork License Manager on page 56

Using Administrative Images

An administrative image allows configurable installations of SolidWorks products from a single network location.

You can install the administrative image from any computer on the network, including a client computer.

- The administrative image must be in a shared folder.
- If you include all products, the administrative image disk space requirements can be 7GB or more.
- If you update the share name when you upgrade the administrative image, the client upgrades automatically the next time the application is started. For details, see Upgrading Clients from Administrative Images on page 44.
- Clients must have administrative privileges on their computers, unless the **Run the installation as a different user** option is specified for the client installation in the Administrative Image Option Editor on page 15.

  This option is not supported on Windows Vista or later.

To manage groups of users with multiple products and serial numbers, see Administrative Image Option Editor on page 15.

Creating an Administrative Image from the Installation Manager

If you are installing SolidWorks on multiple computers, you can create an administrative image and then deploy the application to other computers.

1. Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).

2. On the Welcome screen, select the following:

   - **Administrative Image and Server products**
   - **Create or update an administrative image.**

3. Follow the on-screen instructions.

Administrative Image Option Editor

The Administrative Image Option Editor lets you customize installation parameter settings in an administrative image for individual machines and for groups of machines.

After creating an administrative image, the Installation Manager asks you to start the Option Editor. You can also start the Option Editor by double-clicking `sldim\sldadminoptioneditor.exe` in the administrative image folder.

To specify different installation configurations for groups of machines or individual machines, add groups and machines in the Option Editor and then specify the installation options for those groups and machines.

Adding and Deleting Groups and Machines

The Administrative Image Option Editor lets you specify different installation configurations for groups of machines and for individual machines.

For example, you might want to configure several machines to use a particular subset of the product components available in the image while another group might use a different subset of product components. Even though you are using one administrative image, the Option Editor group settings enable you to install different product subsets to different groups of machines.

To add a group:	1. Right-click **Global Settings** and then click **Add Group**. 2. Enter the group name.

To add machines:	1. Right-click an existing group or **Global Settings** and then click **Add Machine**.
	
	    ✏️ If you right-click an existing group, machines are added to that group. Otherwise, they are added at the top level.
	
	2. Do one of the following:
	
	    • Enter one or more machine names under **Machine Name**.
	
	      Multiple machines can be separated by a line break, space, tab, comma, period, colon, or semi-colon. You can copy and paste a list of machine names from an external file.
	
	      Machine names cannot include domain qualifiers. For example, pluto.solidworks.com is invalid.
	
	    • Click **Add Multiple Machines** to select machines from the network.

To move an existing machine to a group:	Drag the machine onto the group.
To move a machine out of a group:	Drag the machine onto **Global Settings** or onto a different group.
To delete a group or machine:	Right-click the group or machine and then click **Delete Group** or **Delete Machine**. ✏️ If you delete a group, all machines within that group are deleted.
To delete all groups and machines:	Right-click **Global Settings** and then click **Delete All**.

Specifying Option Settings for Groups and Machines
You can specify option settings for all machines, groups of machines, and individual machines.

Option settings are inherited as you move down the hierarchy. For example:

- Settings defined for **Global Settings** are inherited by all groups and machines.
- Settings defined for a group are inherited by all machines within that group.

You can change option settings for a machine or group to override default or inherited settings. When you change an option setting from the default or its inherited value, that option is highlighted.

To revert option settings to the original inherited values, right-click the group or machine and then click **Revert to Global Settings** (for a group or machine that is not a group member) or **Revert to Group Settings** (for a machine that is a group member).

To specify option settings:

1. Click the object to modify (for example, **Global Settings**, a group, or a machine).
2. On the tabs to the right, specify the option settings.

Option tab	Option settings
Serial Numbers	Enter serial numbers for the various products to install.
	Specify the port and server for a SolidWorks Network License (SNL) Server. For details, see Network License Overview.
Client Options	Create a new installation or to upgrade the most recent existing installation.
	Specify the installation directory on the client machine.
	Use the default SolidWorks user interface settings (keyboard shortcuts, menu customization, toolbar layout, and so on) or use settings saved from another installation using the Copy Settings Wizard.
	Choose whether to participate in the SolidWorks Customer Experience Program. For further information, see the Customer Experience Feedback Program Web site.
Image Options	Automatically activate client installations. For details, see License Activation.
	Create a diagnostic log during installation.
	Run the installation as a different user (for example, if administrative permission is required for installation).
	Installing the administrative image on the client by using a mapped drive (instead of a UNC Format on page 99) can fail if this option is specified, due to remote permissions problems.
	This feature is not supported on Windows Vista or later.
	Run external programs before or after the installation.
Software to Install	Specify the product components to install.
Standard Libraries	Use existing standard Toolbox library files or install new library files.

3. Close the Option Editor to apply changes in the option settings.

Deploying an Administrative Image to Clients Through Email

Once you have created a shared administrative image directory, you can notify clients by email.

Configure the administrative image directory to be shared on a machine that is accessible to other computers (for example, through a UNC Format on page 99).

You can send an email containing a link to
`admin_image_install_dir`\StartSWInstall.hta to all clients. Clients would then open the file and click **Install SolidWorks products now** to start the installation from the administrative image.

✏️ If you are upgrading an administrative image to a new SolidWorks version, you can configure your administrative image location and share properties so that any clients that installed previously using that image will upgrade automatically. For details, see Upgrading Clients from Administrative Images on page 44.

Advertising an Administrative Image to Clients

You can run a command to advertise a new administrative image installation to clients.

Email deployment is the method generally used to notify clients of a new SolidWorks installation.

However, as an alternative, you can execute a command to advertise a new installation to clients. For example, you might advertise to employees who do not normally use the SolidWorks application.

Advertising SolidWorks creates a shortcut to the SolidWorks application on the **Start** menu on the user's computer and creates a file association with the administrative image on the server. Initially, the executable files are not installed. Installation occurs only when the user first needs the application.

When the user clicks the shortcut or double-clicks a SolidWorks document, the installation begins and the SolidWorks application executes.

To advertise from an administrative image:

1. Create and configure the administrative image.
2. Advertise the product on each machine where the SolidWorks application is to be installed.

   For example:

   `msiexec /jm path_to_msi_file_of_admin_image`

   where *path_to_msi_file_of_admin_image* is a UNC Format on page 99 to the administrative image directory.

Command-Line Deployment

You can automatically deploy an administrative image using a command prompt or batch file (silent installation).

Creating an Administrative Image from the Command Line

You can create an administrative image using the Installation Manager or from the MS-DOS command line. The command line allows you to customize the installation.

🖉 If you create an administrative image from the command line, you must create and deploy a

1. Open a Command Prompt window.

   a) In Windows, click **Start** > **Run**.

   b) Enter `cmd` and click **OK**.

2. Enter this command:

   ```
   msiexec /a Msi_path
   ```

   where *Msi_path* is the path and filename of the `.msi` file for your operating system. The `SolidWorks.msi` is located in the `swwi\data` subdirectory in the installation directory.

   For example:

   ```
   msiexec /a F:\swwi\data\SolidWorks.msi
   ```

3. In the SolidWorks Setup Wizard, follow the on-screen instructions.

Preparing Clients for Command-Line Installation from an Administrative Image

After creating the administrative image and before notifying clients, you must install Microsoft Windows components that cannot be installed by administrative images created using the command line or Microsoft Active Directory.

🖉 Installing these Windows components manually is required only if you are installing from an administrative image that was created manually using the command line or through Microsoft Active Directory. If you use the Installation Manager to create and install the administrative image, these Windows components are installed automatically.

Here is a summary of the files required by SolidWorks in the Windows components:

Windows Installer	Contains files needed to run the installation of SolidWorks Simulation Premium.
Office Web Components	Contains COM controls that are needed for DriveWorks.
Visual C++ 2005 and 2008 Redistributable Package	Contains runtime components for Microsoft Visual C++. 🖉 The 2005 and 2008 versions both must be installed.
Visual Studio Tools for Applications	Contains a tool set supporting Visual Studio applications.
Visual Studio Remote Debugger	Provides a debugging environment for 64-bit operating systems.
.NET Framework 3.0	Contains the common language runtime and .NET Framework components that are necessary to run .NET Framework applications.

| **Windows Desktop Search (optional)** | Supports fast searches of files and data across different data sources and locations, including the local computer and shared networks. |

Windows Desktop Search files are language specific.

💡 Visual C++ Redistributable Package and .NET Framework 3.0 are required for all SolidWorks products, not just the core SolidWorks product.

These components must be included in subdirectories of the administrative image directory.

This table identifies the specific files that are required for each supported operating system.

Operating System: Windows 2003 32-bit[1]

SolidWorks Media: 32-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer:	(Not applicable)
Office Web Components:	owc11.exe
Visual C++ 2005 Redistributable Package:	vcredist_x86.exe
Visual C++ 2008 Redistributable Package:	vcredist_x86.exe
Visual Studio Tools for Applications:	vsta_aide.msi
Visual Studio Remote Debugger:	(Not applicable)
.NET Framework:	dotnetfx3.exe

Windows Desktop Search (optional):

    WindowsDesktopSearch-KB917013-V301-Srv2K3-x86-*lang*.exe

Operating System: Windows 2003 64-bit[1]

SolidWorks Media: 64-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer:	(Not applicable)
Office Web Components:	owc11.exe
Visual C++ 2005 Redistributable Package:	vcredist_x86.exe
	vcredist_x64.exe

 Both must be installed.

Visual C++ 2008 Redistributable Package:	vcredist_x86.exe
	vcredist_x64.exe

 Both must be installed.

Visual Studio Tools for Applications:	vsta_aide.msi
Visual Studio Remote Debugger:	rdbgexp.msi
.NET Framework:	dotnetfx3.exe

Windows Desktop Search (optional):

    WindowsDesktopSearch-KB917013-V301-Srv2K3_XP-x64-enu.exe

Operating System: Windows 2008 32-bit[1]

SolidWorks Media: 32-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer:	(Not applicable)
Office Web Components:	owc11.exe
Visual C++ 2005 Redistributable Package:	vcredist_x86.exe

Visual C++ 2008 Redistributable Package:	vcredist_x86.exe
Visual Studio Tools for Applications:	vsta_aide.msi
Visual Studio Remote Debugger:	(Not applicable)
.NET Framework:	.NET Framework 3.0 and Windows Desktop Search are features of the operating system.
Windows Desktop Search (optional):	

Operating System: Windows 2008 64-bit[1]

SolidWorks Media: 64-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer:	(Not applicable)
Office Web Components:	owc11.exe
Visual C++ 2005 Redistributable Package:	vcredist_x86.exe
	vcredist_x64.exe

 Both must be installed.

Visual C++ 2008 Redistributable Package:	vcredist_x86.exe
	vcredist_x64.exe

 Both must be installed.

Visual Studio Tools for Applications:	vsta_aide.msi
Visual Studio Remote Debugger:	rdbgexp.msi
.NET Framework:	.NET Framework 3.0 and Windows Desktop Search are features of the operating system.
Windows Desktop Search (optional):	

Operating System: Windows XP 32-bit

SolidWorks Media: 32-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer:	WindowsInstaller-KB893803-v2-x86.exe
Office Web Components:	owc11.exe
Visual C++ 2005 Redistributable Package:	vcredist_x86.exe
Visual C++ 2008 Redistributable Package:	vcredist_x86.exe
Visual Studio Tools for Applications:	vsta_aide.msi
Visual Studio Remote Debugger:	(Not applicable)
.NET Framework:	dotnetfx3.exe
Windows Desktop Search (optional):	

WindowsDesktopSearch-KB917013-V301-XP-x86-*lang*.exe

Operating System: Windows XP 64-bit

SolidWorks Media: 32-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer:	(Not applicable)
Office Web Components:	owc11.exe
Visual C++ 2005 Redistributable Package:	vcredist_x86.exe
Visual C++ 2008 Redistributable Package:	vcredist_x86.exe
Visual Studio Tools for Applications:	vsta_aide.msi
Visual Studio Remote Debugger:	rdbgexp.msi
.NET Framework:	dotnetfx3_x64.exe

Windows Desktop Search (optional):

WindowsDesktopSearch-KB917013-V301-Srv2K3_XP-x64-enu.exe

Operating System: Windows XP 64-bit

SolidWorks Media: 64-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer: (Not applicable)

Office Web Components: owc11.exe

Visual C++ 2005 Redistributable Package: vcredist_x86.exe

vcredist_x64.exe

 Both must be installed.

Visual C++ 2008 Redistributable Package: vcredist_x86.exe

vcredist_x64.exe

 Both must be installed.

Visual Studio Tools for Applications: vsta_aide.msi

Visual Studio Remote Debugger: rdbgexp.msi

.NET Framework: dotnetfx3_x64.exe

Windows Desktop Search (optional):

WindowsDesktopSearch-KB917013-V301-Srv2K3_XP-x64-enu.exe

Operating System: Windows Vista 32-bit

SolidWorks Media: 32-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer: (Not applicable)

Office Web Components: owc11.exe

Visual C++ 2005 Redistributable Package: vcredist_x86.exe

Visual C++ 2008 Redistributable Package: vcredist_x86.exe

Visual Studio Tools for Applications: vsta_aide.msi

Visual Studio Remote Debugger: (Not applicable)

.NET Framework: .NET Framework 3.0 and Windows
                                             Desktop Search are features of the
Windows Desktop Search (optional):   operating system.

Operating System: Windows Vista 64-bit

SolidWorks Media: 64-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer: (Not applicable)

Office Web Components: owc11.exe

Visual C++ 2005 Redistributable Package: vcredist_x86.exe

                                             vcredist_x64.exe

                                              Both must be installed.

Visual C++ 2008 Redistributable Package: vcredist_x86.exe

                                             vcredist_x64.exe

                                              Both must be installed.

Visual Studio Tools for Applications: vsta_aide.msi

Visual Studio Remote Debugger: rdbgexp.msi

.NET Framework: .NET Framework 3.0 and Windows
                                             Desktop Search are features of the
Windows Desktop Search (optional):   operating system.

Operating System: Windows 7 32-bit

SolidWorks Media: 32-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer: (Not applicable)

Office Web Components: owc11.exe

Visual C++ 2005 Redistributable Package: vcredist_x86.exe

Visual C++ 2008 Redistributable Package: vcredist_x86.exe

Visual Studio Tools for Applications: vsta_aide.msi

Visual Studio Remote Debugger: (Not applicable)

.NET Framework: .NET Framework 3.0 and Windows
                                             Desktop Search are features of the
Windows Desktop Search (optional):   operating system.

Operating System: Windows 7 64-bit

SolidWorks Media: 64-bit version for Windows XP, Windows Vista, and Windows 7

Windows Installer:	(Not applicable)
Office Web Components:	owc11.exe
Visual C++ 2005 Redistributable Package:	vcredist_x86.exe
	vcredist_x64.exe

       ✏ Both must be installed.

Visual C++ 2008 Redistributable Package:	vcredist_x86.exe
	vcredist_x64.exe

       ✏ Both must be installed.

Visual Studio Tools for Applications:	vsta_aide.msi
Visual Studio Remote Debugger:	rdbgexp.msi
.NET Framework: Windows Desktop Search (optional):	.NET Framework 3.0 and Windows Desktop Search are features of the operating system.

[1]Installation on Windows 2003 or Windows 2008 is supported only for Workgroup PDM Server and SolidNetWork License (SNL) Server.

Installing Windows Prerequisites from the Command Line on 32-bit Operating Systems

Follow this process to install Windows prerequisites for administrative image installations using the command line on 32-bit Versions of Windows.

To install multiple components in one install operation, create a batch file with the commands provided in the following procedures and use it to push the components to the client computers.

1. Ensure that the target computer has at least 290 MB of available disk space and a valid TEMP directory.

2. Install Office Web Components by running this command:

    administrative_image_directory `\OfficeWeb_11\owc11.exe`

3. For Windows XP, install Windows Installer:

    administrative_image_directory
    `\WindowsInstaller\WindowsInstaller-KB893803-v2-x86.exe`

    ✏ It is not necessary to install Windows installer on any version of Windows other than Windows XP.

4. Install the Visual C++ 2005 Redistributable Package:

administrative_image_directory
`\Microsoft_C++_2005_Redistributable\vcredist_x86.exe /Q`

5. Install the Visual C++ 2008 Redistributable Package:

    administrative_image_directory
    `\Microsoft_C++_2008_Redistributable\vcredist_x86.exe /Q`

6. For Windows XP, install Microsoft .NET Framework 3.0 using Windows Server Update Services.

    🖊 It is not necessary to install the .NET Framework image on Windows Vista or later.

7. Install the Visual Studio Tools for Applications:

    administrative_image_directory `\Microsoft_VSTA\vsta_aide.msi`

8. For Windows XP or Windows 2003, optionally install Windows Desktop Search by running one of these commands:

    -   For Windows XP: *administrative_image_directory*
        `\SolidWorks\Temp\WindowsDesktopSearch-KB917013-V301-XP-x86-`*lang*`.exe /Q`
    -   For Windows 2003: *administrative_image_directory*
        `\SolidWorks\Temp\WindowsDesktopSearch-KB917013-V301-Srv2K3-x86-`*lang*`.exe /Q`

Installing Windows Prerequisites from the Command Line on 64-bit Operating Systems

Follow this process to install Windows prerequisites for administrative image installation using the command line on 64-bit Versions of Windows.

To install multiple components in one install operation, create a batch file with the commands provided in the following procedures and use it to push the components to the client computers.

1. Ensure that the target computer has at least 620 MB of available disk space and a valid TEMP directory.
2. Install Office Web Components by running this command:

    administrative_image_directory `\OfficeWeb_11\owc11.exe`

3. Install the Visual C++ 2005 Redistributable Package:

    administrative_image_directory
    `\Microsoft_C++_2005_Redistributable\vcredist_x64.exe /Q`

    administrative_image_directory
    `\Microsoft_C++_2005_Redistributable\vcredist_x86.exe /Q`

    🖊 Both the 32-bit and the 64-bit versions of this package must be installed.

4. Install the Visual C++ 2008 Redistributable Package:

    administrative_image_directory
    `\Microsoft_C++_2008_Redistributable\vcredist_x64.exe /Q`

administrative_image_directory
`\Microsoft_C++_2008_Redistributable\vcredist_x86.exe /Q`

🖉 Both the 32-bit and the 64-bit versions of this package must be installed.

5. For Windows XP, install Microsoft .NET Framework 3.0 using Windows Server Update Services.

🖉 It is not necessary to install the .NET Framework image on Windows Vista or later.

6. Install the Visual Studio Tools for Applications:

   administrative_image_directory `\Microsoft_VSTA\vsta_aide.msi`

7. Install the Visual Studio Remote Debugger:

   administrative_image_directory `\VSRemoteDebugger\install.exe /Q`

8. For Windows XP or Windows 2003, optionally install Windows Desktop Search:

   administrative_image_directory
   `\SolidWorks\Temp\WindowsDesktopSearch-KB917013-V301-XP-x86-`*lang*`.exe /Q`

SolidWorks Search and Microsoft Windows Desktop Search
SolidWorks Search is integrated with Microsoft® Windows® Desktop Search.

If the correct version of Windows Desktop Search is not already installed on your computer, you are asked to install or update it during SolidWorks installation.

If you are creating an administrative image manually on the command line, Microsoft Windows Desktop Search must be installed manually on client computers prior to installing from an administrative image.

Installing from the Administrative Image Using the Command Line

You can deploy installations from an administrative image using the command line.

To install silently:

1. Open a Command Prompt window (in Windows, click **Start** > **Run**.
2. Type the `msiexec.exe` command with the `/i` and `/qb` switches and set the properties for any options that are not the default values as shown below.

Properties are listed for these products:

- Command-Line Feature Properties on page 30
- Command-Line Global Properties on page 31
- DWGeditor Command-Line Properties on page 33
- SolidWorks eDrawings Command Line Properties on page 34
- SolidWorks Explorer/Workgroup PDM Command-Line Properties on page 36
- SolidWorks Flow Simulation Command Line Properties on page 38

Use the following examples as templates for silent installation.

SolidWorks Client

- Chinese-Simplified language
- PhotoWorks add-in
- Participation in the feedback program

32-bit version of SolidWorks on 32-bit operating system:

```
msiexec /i "Msi_path" INSTALLDIR="C:\Program Files\your_folder"
SOLIDWORKSSERIALNUMBER="xxxx xxxx xxxx xxxx xxxx xxxx" ENABLEPERFORMANCE=1
OFFICEOPTION=3
ADDLOCAL=SolidWorks,ChineseSimplified,PhotoWorksRender,Intel_Modules_AgeiaX86
/qb
```

32-bit version of SolidWorks on 64-bit operating system:

```
msiexec /i "Msi_path" INSTALLDIR="C:\Program Files\your_folder"
SOLIDWORKSSERIALNUMBER="xxxx xxxx xxxx xxxx xxxx xxxx" ENABLEPERFORMANCE=1
OFFICEOPTION=3
ADDLOCAL=SolidWorks,ChineseSimplified,PhotoWorksRender,Intel_Modules_AgeiaX64
/qb
```

64-bit version of SolidWorks on 64-bit operating system:

```
msiexec /i "Msi_path" INSTALLDIR="C:\Program Files\your_folder"
SOLIDWORKSSERIALNUMBER="xxxx xxxx xxxx xxxx xxxx xxxx" ENABLEPERFORMANCE=1
OFFICEOPTION=3 ADDLOCAL=SolidWorks,ChineseSimplified,PhotoWorksRender /qb
```

DWGeditor

- Standalone license

```
msiexec /i "Msi_path" INSTALLDIR="C:\Program Files\your_folder"
LICENSETYPE=1 /qb
```

eDrawings Client

- Log performance
- Italian language
- All features

```
msiexec /i "Msi_path" INSTALLDIR="C:\Program Files\your_folder"
LOGPERFORMANCE=1 SNLSERVER=eDrawings_SNL_server TRANSFORMS=1040.mst
ADDLOCAL=All /qb
```

SolidWorks Explorer and Workgroup PDM add-in

- SolidWorks Explorer with Workgroup PDM add-in

```
msiexec /i "Msi_path" INSTALLDIR="C:\Program Files\your_folder"
EXPLORERORPDM=1 ADDLOCAL=VaultAdmin,Client,Standalone,DocMgr,LicenseEXE
ACTIVSERIALNUMBER="xxxx xxxx xxxx xxxx xxxx xxxx" /qb
```

SolidWorks PhotoView 360

```
msiexec /i "Msi_path" /qb
```

This product is licensed through the PhotoView license.

This installation requires that a version of PhotoView 360 is already installed on your computer. There are no command options available when installing PhotoView 360.

Command-Line Properties

Command-line properties describe here include feature properties, global properties, product-specific properties and Windows location conventions.

Command-Line Feature Properties

This table lists the ADDLOCAL properties for command-line deployment.

✎ SolidWorks does not support product installation using the ADDSOURCE option.

In this table:

- Properties are case sensitive, cannot contain spaces or dashes, and must be separated by commas.
- The command must be in English regardless of the language installed.
- SolidWorks must be specified for the SolidWorks application to be installed correctly.

Property	Property
Languages:	Manuals
Czech	RealViewGraphicsdisplay
Chinese	SolidWorks
ChineseSimplified	FeatureWorks
English	InstantWebsite
French	PhotoWorksRender
German	ScanTo3D
Italian	SolidWorksAnimator
Japanese	SolidWorksDesignChecker
Korean	SolidWorksRoutedsystems
Polish	SolidWorksToolbox
PortugueseBrazilian	SolidWorksUtilities
Russian	TolAnalyst
Spanish	CircuitWorks
Turkish	DWGPartner
	Simulation
	Motion
	XchangeWorks
	CoreSolidWorksTaskScheduler
	ExampleFiles
	HelpFiles
	Intel_Modules_AgeiaX86
	(for 32-bit SolidWorks on 32-bit operating systems)
	Intel_Modules_AgeiaX64
	(for 32-bit SolidWorks on 64-bit operating systems)

Command-Line Global Properties
Each property is defined for administrative or individual installations, unless otherwise noted. Serial numbers can be passed to both an administrative installation and an individual installation using global properties.

Property	Description
ENABLEPERFORMANCE	Whether the application should automatically send performance data to SolidWorks Corporation.
	1 to send performance data, 0 otherwise.
INSTALLDIR	Installation folder for SolidWorks.
	Individual installations only.
INSTALLWDS	Install the SolidWorks extensions that use Microsoft Windows Desktop Search Engine. For details, see Preparing Clients for Command-Line Installation from an Administrative Image on page 20.
	1 to install the extensions, 0 otherwise.
MOTIONSERIALNUMBER	Serial number for the SolidWorks Motion product
OFFICEOPTION	Specify product package to install:
	0 SolidWorks Client
	1 SolidWorks Office
	2 SolidWorks Professional
	3 SolidWorks Premium
SERVERLIST	SolidNetWork server list.
	list_of_port@server_strings is a list of strings delimited by semi-colons, each of which refers to a port number on a server with the format:
	$xxxxx@server\_name1;yyyyy@server\_name2;\&$
	where *x* and *y* are single digits. After the last server in the list, do not put a semi-colon.
	Individual installations only.
SIMULATIONSERIALNUMBER	Serial number for the SolidWorks Simulation product
SOLIDWORKSSERIALNUMBER	Serial number for the SolidWorks product, which includes all add-ins.
SWMIGRATE	SolidWorks version to upgrade as it would be shown in the list in the SolidWorks Install Choice dialog box. For example:
	`SWMIGRATE="SolidWorks 2010 SP01"`

Property	Description
TARGETDIR	Installation folder for SolidWorks.
	Administrative installations only.
TOOLBOXFOLDER	Installation folder for SolidWorks Toolbox files.
	Individual installations only.
TRANSFORMS	Contains language-specific items of the form:
	`TRANSFORMS` *number*`.mst`
	- Chinese (1028) - Chinese-Simplified (2052) - Czech (1029) - English (1033) - French (1036) - German (1031) - Italian (1040) - Japanese (1041) - Korean (1042) - Polish (1045) - Portuguese-Brazilian (1046) - Russian (1049) - Spanish (1034) - Turkish (1055)
UPGRADESWINSTALL	Whether to upgrade an existing version of SolidWorks to the current version.
	0 New installation, do not update
	1 Update
	Individual installations only.

DWGeditor Command-Line Properties
These command-line properties are specific to DWGeditor.

Property	Description
DWGASSOC	**1** Associates `.dwg` files with DWGeditor **2** Does not associate `.dwg` files with DWGeditor
INSTALLDIR	Specifies the installation folder for DWGeditor.
LICENSETYPE	**1** Standalone **2** Network License 🖉 If you specify `2`, you must also specify `SERVERLIST`.
SERVERLIST	List of strings delimited by semi-colons, each of which refers to a port number on a server with the format: *xxxxx@server_name1;* *yyyyy@server_name2;&* where *x* and *y* are single digits. After the last server in the list, do not put a semi-colon.

SolidWorks eDrawings Command Line Properties
These command-line properties are specific to eDrawings.

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Property	Description	
ADDLOCAL	Specifies which portion of the SolidWorks eDrawings application to install. `All` is the recommended option.	
	Command Line Syntax:	Feature:
	`All`	Installs all the features. The eDrawings application searches for installed SolidWorks and AutoCAD publisher applications to validate the install.
	`eDrawingsViewer`	View eDrawings and other files.
	`PublisherForAutoCADR14`	Create eDrawings files from the AutoCAD R14 application.
	`PublisherForAutoCAD2000`	Create eDrawings files from the AutoCAD 2000 application.
	`PublisherForAutoCAD2004`	Create eDrawings files from the AutoCAD 2004 application.
	`PublisherForAutoCAD2005`	Create eDrawings files from the AutoCAD 2005 application.
	`PublisherForAutoCAD2006`	Create eDrawings files from the AutoCAD 2006 application.
	`DXFDWGTranslator`	View DXF/DWG files in the eDrawings Viewer.
	`Shortcuts`	Install desktop shortcut.
	`XML3D`	View 3DXML files in the eDrawings Viewer.
INSTALLDIR	Specifies the installation folder for SolidWorks eDrawings.	
LOGPERFORMANCE	Specifies whether the application should automatically email performance data to SolidWorks Corporation. **1** Send performance data **0** Otherwise	

Property	Description
SNLSERVER	Enter the *port@servername* for your SolidNetWork License server. Use semi-colons to separate multiple *port@servername* combinations. A port server name for SNLSERVER is only required for the SolidWorks SNL server configuration. `list_of_port@server_strings` is a list of strings delimited by semi-colons, each of which refers to a port number on a server with the format: *xxxxx@server_name1; yyyyy@server_name2;* where *x* and *y* are single digits. After the last server in the list, do not put a semi-colon.

SolidWorks Explorer/Workgroup PDM Command-Line Properties
These command-line properties are specific to SolidWorks Explorer and SolidWorks Workgroup PDM.

For SolidWorks Explorer/Workgroup PDM client:

Client Property	Description
EXPLORERORPDM	**0** SolidWorks Explorer only **1** Workgroup PDM features can be installed
ADDLOCAL	• Standalone • Client • Client64 • VaultAdmin • AutoCADR*xxxx* • DocMgr • DocMgr64 • LicenseEXE Standalone is the Contributor and Client is the SolidWorks Add-in for SolidWorks Explorer silent installations. If you specify AutoCAD, AutoCAD must be installed on your system. Workgroup PDM locates the appropriate version of AutoCAD. You can specify more than one version of AutoCAD: • AutoCADR14Addin • AutoCADR2000Addin • AutoCADR2004Addin • AutoCADR2005Addin • AutoCADR2006Addin Client64 and DocMgr64 work with only 64-bit SolidWorks Explorer installations.
INSTALLDIR	Specifies the installation folder for Workgroup PDM.
ACTIVSERIALNUMBER	Specifies the Solidworks serial number.

For Workgroup PDM Server:

Server Property	Description
ADDLOCAL	• Vault • WebPortal • API If you specify WebPortal (the Viewer), you must also specify VAULTNAME.
INSTALLDIR	Specifies the installation folder for Workgroup PDM.
LICENSE_SERVER	Specifies the name of the computer where the license server resides.
VAULTDATA	Specifies the path and folder of the vault.
VAULTNAME	Specifies the name of the computer where the vault resides.

SolidWorks Flow Simulation Command Line Properties
These command-line properties are specific to SolidWorks Flow Simulation.

Property	Description
ACTIVSERIALNUMBER	SolidWorks Flow Simulation standalone serial number. (Individual installations and client installations from an administrative image)
CWUPGRADEINSTALL	Upgrades an existing version of a particular SolidWorks Flow Simulation application software. **1** Update **2** New installation; install in a new directory (Individual installations and client installations from an administrative image)
INSTALLDIR	Specifies the installation for a particular SolidWorks Flow Simulation application software. (Individual installations only)
SERVERLIST	List of strings delimited by semi-colons, each of which refers to a port number on a server with the format: $xxxxx@server\_name1;$ $yyyyy@server\_name2;\&$ where x and y are single digits. After the last server in the list, do not put a semi-colon.
SOLIDWORKSPATH	Specifies the folder where the SolidWorks software is installed. (Individual installations and client installations from an administrative image)
TARGETDIR	Specifies the installation for a particular SolidWorks Flow Simulation application software. (Administrative image installations only)

MSI File Locations for Administrative Images
For administrative images created from the command line, the `product .msi` files are stored in these locations on the SolidWorks media.

Product	Media Location
SolidWorks (core product)	`swwi\data\SolidWorks.msi`
SolidWorks API SDK	`apisdk\SolidWorks API SDK.msi`
SolidWorks Simulation products	`cwwi\`
	`COSMOSFloWorks\SolidWorks Flow Simulation.msi`
	`COSMOSM\COSMOSM_lang.msi`
DWGeditor	`DWGeditor\DWGeditor.msi`
eDrawings	`eDrawings\eDrawings.msi`
eDrawingsAPI SDK	`eDrwAPISDK\eDrawings API SDK.msi`
Workgroup PDM Server	`pdmwserver\SolidWorks Workgroup PDM Server.msi`
PhotoView 360	`PhotoView\PhotoView 360.msi`
DocManager APIs	`swdocmgr\SolidWorks Document Manager API.msi`
SolidWorks Explorer	`swexplorer\SolidWorks Explorer.msi`
SolidNetWork License Manager	`swlicmgr\SolidWorks SolidNetWork License Manager.msi`

Windows Desktop Search Language Support

Use this table to identify the Windows Desktop Search file to install on client computers.

_chs	Simplified Chinese
_cht	Chinese
_csy	Czech
_deu	German
_enu	English
_esn	Spanish
_fra	French
_ita	Italian
_jpn	Japanese
_kor	Korean
_plk	Polish
_ptb	Brazilian Portuguese
_rus	Russian

Deploying an Administrative Image Using Microsoft Active Directory

Microsoft Active Directory Deployment is supported for deployment of the core SolidWorks product to clients.

Microsoft Active Directory is a directory service for managing clients and servers in network environments. It simplifies management, migration, and deployment.

1. Create an administrative image using SolidWorks Installation Manager or the command line.
2. If you have created the administrative image using the command line, install the Windows prerequisites for installation from an administrative image Installing Windows Prerequisites Using Microsoft Active Directory on page 42.

   🖉 Perform this step only if you created the image manually using the command line. If you created the image using SolidWorks Installation Manager, all Windows prerequisites were installed automatically.

3. Create a transform file to customize the SolidWorks software:

   a) Open the Command Prompt window.
   b) In the folder that contains the administrative image, drag SWCreateMST.exe into the Command Prompt window.

      The command should be in the format:

      administrative_image_directory\SWCreateMST.exe
      path_to_MSI_file_of_an_administrative_image list_of_properties

      The properties can be feature properties or global properties. Use the following as templates:

      - If you are installing SolidWorks (32-bit version) on a 32-bit operating system:

        ```
        C:\admin\SWCreateMST.exe "c:\admin\SolidWorks.msi"
        INSTALLDIR="C:\Program Files\your_folder"
        SOLIDWORKSSERIALNUMBER="xxxx xxxx xxxx xxxx"
        ADDLOCAL=SolidWorks,English,Manuals,Intel_Modules_AgeiaX86
        OFFICEOPTION=0 SERVERLIST="25734@your_server"
        ```

      - If you are installing SolidWorks (32-bit version) on a 64-bit operating system:

        ```
        C:\admin\SWCreateMST.exe "c:\admin\SolidWorks.msi"
        INSTALLDIR="C:\Program Files\your_folder"
        SOLIDWORKSSERIALNUMBER="xxxx xxxx xxxx xxxx"
        ADDLOCAL=SolidWorks,English,Manuals,Intel_Modules_AgeiaX64
        OFFICEOPTION=0 SERVERLIST="25734@your_server"
        ```

      - If you are installing SolidWorks (64-bit version) on a 64-bit operating system:

        ```
        C:\admin\SWCreateMST.exe "c:\admin\SolidWorks.msi"
        INSTALLDIR="C:\Program Files\your_folder"
        SOLIDWORKSSERIALNUMBER="xxxx xxxx xxxx xxxx"
        ADDLOCAL=SolidWorks,English,Manuals OFFICEOPTION=0
        SERVERLIST="25734@your_server"
        ```

      The transform file, SolidWorks.mst, is created in the folder that contains the administrative image.

4. In the Active Directory, edit the Published Apps policy for the group or groups to deploy the SolidWorks software.
5. In the Group Policy dialog box, expand Computer Configuration and Software Settings.
6. Right-click Software installation and select **New** > **Package**.
7. Browse to the administrative image and click **Open**.
8. In the Deploy Software dialog box, select **Advanced published or assigned**, then click **OK**.
9. In the SolidWorks Properties dialog box, on the Modifications tab:
   a) Click **Add**.
   b) Browse to the `.mst` file.
   c) Click **Open**.
10. Click **OK**.

Installing Windows Prerequisites Using Microsoft Active Directory
Follow this process to prepare an existing administrative image that will install SolidWorks onto client computers using Microsoft Windows Active Directory.

In these processes, use the command that is appropriate to the version of SolidWorks you are deploying.

1. Extract the 2005 Visual C++ Redistributable Package installer files and install them.
   a) Run one of these commands.

      For 32-bit versions of Windows:

      administrative_image_directory\Microsoft_C++_2005_Redistributable\vcredist_x86.exe /C /T:{*C++_target_directory*}

      For 64-bit versions of Windows:

      administrative_image_directory\Microsoft_C++_2008_Redistributable(64)\vcredist_x64.exe /C /T:{*C++_target_directory*}

   b) Create a policy to install the Microsoft C++ 2005 redistributable module:

      {*2005_C++_target_directory*}\vcredist.msi –

          No transform file is needed for this installation.

2. Extract the 2008 Visual C++ Redistributable Package installer files and install them.
   a) Run one of these commands.

      For 32-bit versions of Windows:

      administrative_image_directory\Microsoft_C++_2008_Redistributable\vcredist_x86.exe /x:{*C++_target_directory*}

      For 64-bit versions of Windows:

      administrative_image_directory\Microsoft_C++_2008_Redistributable(64)\vcredist_x64.exe /x:{*C++_target_directory*}

   b) Create a policy to install the Microsoft C++ 2008 redistributable module:

      {*2008_C++_target_directory*}\vcredist.msi –

✏️ No transform file is needed for this installation.

3. Using Windows Server Update Services, install Microsoft .NET Framework 3.0.

   ✏️ It is not necessary to install the .NET Framework image on Windows Vista or later.

4. Extract the Office Web Components files and install them.
   a) Run this command:
   ```
   administrative_image_directory\OfficeWeb_11\owc11.exe
   /extract:{Office_Web_components_directory}
   ```
   b) Create a policy to install the Office Web Components:
   ```
   {Office_Web_components_directory}\owc11.MSI
   ```

   ✏️ No transform file is needed for this installation.

5. Install the Visual Studio Tools for Applications (VSTA), by creating this policy:
   ```
   {Microsoft_VSTA_directory}\vsta_aide.MSI
   ```

   ✏️ No transform file is needed for this installation.

6. Install the Visual Studio Remote Debugger (64-bit versions only), by creating this policy:
   ```
   {Microsoft_remote_debugger_directory}\rdbgexp.MSI
   ```

   ✏️ No transform file is needed for this installation.

Administration of Multiple Installations

For multiple installations, administrative tasks include upgrading administrative images to new releases, applying service pack updates, and removing installations.

Upgrading Administrative Images to a New Release

To update clients who installed from an administrative image, first update the administrative image and then configure the clients to reference the new copy of the image.

Run the Installation Manager from the Internet so that the correct files are referenced. The Installation Manager will redirect itself to do this, or you can run the update directly from the SolidWorks Downloads Web site (http://www.solidworks.com/sw/support/downloads.htm).

Once you have updated the administrative image (which creates a new copy of the image), you can update clients by having them reference the new copy of the administrative image.

✏️ If clients referenced the previous version of the administrative image using a mapped drive or a shared directory, you could redefine that mapped drive or shared directory using the location of the updated copy of the administrative image.

Downloading Updated Installation Files for Administrative Images
SolidWorks Installation Manager can detect when you need updated installation files when upgrading an administrative image.

1. Start the SolidWorks Installation Manager by inserting the DVD.
2. On the Welcome screen, select:
   - **Administrative image and server products**
   - **Create or update an Administrative Image.**
3. On the Serial Number screen, enter the SolidWorks serial number.

   The Installation Manager checks the SolidWorks Web site for your licensed products and available service packs.
4. If the Newer Version Available screen appears, select **Yes, I would like to get the latest version: _version_**.
   Installation Manager starts a new Installation Manager session.
5. If there is a problem with the download folder, Download Options might appear, in which case, accept the default directory for the new installation files or enter a new directory name.

   🖉 If you see a message that the directory does not exist, click **Yes** to create it.

6. On the Download Complete screen, click **Close** to close the current installer and start the installer for the new version.

Upgrading an Administrative Image
To upgrade an administrative image, SolidWorks Installation Manager creates a new image directory. You then direct clients to use the new administrative image directory to upgrade to that product version.

This process assumes you have downloaded any updated installation files for the administrative image.

1. Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).
2. In the Welcome screen, select the following:
   - **Administrative image and server products**
   - **Create or update an Administrative Image.**
3. On the Serial Number screen, check the serial number.
4. On the Summary (**Ready to Create Image**) screen, check that the administrative image options are correct.

   For details about individual administrative image and installation options, click **CHANGE** to go to the option page and then click **Help**.
5. To create the new version of the administrative image, click **Create Now**.

You can now upgrade clients to this new version of the administrative image.

Upgrading Clients from Administrative Images
The Installation Manager can automatically update client computers installed from an administrative image.

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Typically, administrative images are organized in folders on a server. To support the automatic upgrade of clients, the SolidWorks administrative image must be located in a shared folder.

After upgrading the administrative image, the original installation will be in one folder and the upgraded installation will be in another folder. To use the upgraded installation, remove the sharing option from the original administrative image folder and then add the sharing option to the upgraded administrative image folder.

When SolidWorks starts on a client machine that installed from the original administrative image, it sees the new version and upgrades the client.

Removing the Sharing Option from the Original Administrative Image Folder
When upgrading an administrative image that is accessed from a shared folder, you must remove the sharing option from the original folder.

1. Browse to the folder where the original administrative image files are located.
2. Right-click the folder and select **Properties**.
3. On the Sharing tab, select **Do not share this folder**, and then click **OK**.

Now add the sharing option to the upgraded administrative image folder as described in Adding the Sharing Option to an Upgraded Administrative Image Folder on page 45.

Adding the Sharing Option to an Upgraded Administrative Image Folder
Once you have created an updated administrative image folder and disabled the sharing for the original administrative image folder, you must add the sharing option to the new folder to re-enable the administrative image.

1. Browse to the folder where the upgraded administrative image files are located.
2. Right-click the folder and select **Properties**.
3. On the Sharing tab, select **Share this folder** and then click **New Share**.
   In the New Share dialog box, change the share name to the share name used for the original administrative image folder and then click **OK**.

Computers that installed from the original administrative image share location will upgrade to the new version automatically when starting SolidWorks.

Upgrading Clients of Administrative Images Manually Using the Command Line
Once you have upgraded an administrative image, you can upgrade clients manually using the command line.

1. Make sure all clients are at the same service pack of the previous release. (Clients that do not already have the application installed are not upgraded.)
2. Optionally, on one of the client systems, run the Copy Settings Wizard to capture the current SolidWorks options.
3. Create a new administrative image using one of these methods:

   - SolidWorks Installation Manager
   - Command Line

   ⚠ Do not upgrade or overwrite the previous image.

4. Upgrade all clients to the new version using the `msiexec` command, with these specifications:

   - Include the `/I` and `qb msiexec` command-line options.
   - Specify any required feature and global command-line properties.

For example, to upgrade an existing version of SolidWorks:

```
msiexec /I Msi_path" INSTALLDIR="C:\Program Files\your_folder"
SOLIDWORKSSERIALNUMBER="xxxx xxxx xxxx xxxx xxxx xxxx" UPGRADESWINSTALL=1
SWMIGRATE="SolidWorks version servicepack" /qb
```

Service Pack Product Update Administration

Service Pack product updates can contain updates for system reliability, program compatibility, security, and so on.

You can acquire product updates in two ways:

For current versions of SolidWorks:	By using the Installation Manager to check for, download, and install updates.
	You can initiate the Installation Manager manually, or schedule it to check periodically for updates.
For versions of SolidWorks prior to 2008:	By downloading service packs manually from the SolidWorks support Web site.
	Service packs are applied as patches to an existing software version. For example, SolidWorks 2007 SP1.0 is a service pack for SolidWorks 2007 SP0.0. You can install these service packs using the SolidWorks 2007 Installation Manager; you cannot install them using the Installation Manager in SolidWorks 2008 or later.
	For instructions on downloading and installing service packs, see the installation documentation for the SolidWorks version you are updating.

Rolling Back Service Packs in an Administrative Image

You can roll back an administrative image to an earlier installed service pack. If you installed the SolidWorks software with disks, you need the original disks to restore a service pack. You must have saved a copy of the earlier administrative image.

🖉 Administrative images that have been upgraded cannot be rolled back to a previous service pack. Administrative images created with any method other than the SolidWorks Installation Manager cannot be rolled back to a previous version. Clients installed from these administrative images must be uninstalled and then reinstalled using an administrative image created from the previous version.

1. Delete the folder that contains the current administrative image. To do this, open the Command Prompt window and enter:

   ```
   rmdir /s/q path_to_current_administrative_image
   ```

2. Move the backup copy of the administrative image to the location where you deleted the current administrative image. To do this, open the Command Prompt window and enter:

   ```
   move path_to_backup_administrative_image
   path_to_current_administrative_image
   ```

   After you roll back an administrative image, client computers roll back automatically the next time they run the SolidWorks software.

Applying Service Packs to Clients Deployed with Microsoft Active Directory
If you have installed the SolidWorks software using the Microsoft Active Directory, you can apply service packs to clients with it.

1. Copy the administrative image to a new folder using the following command line:

   `xcopy path_to_current_administrative_image path_to_backup_administrative_image /s/i/v`

2. Apply the service pack to the copy of the administrative image.

3. Create a new transform file to specify an upgrade installation for the SolidWorks software:

   a) Open the Command Prompt window.

   b) In the folder that contains the administrative image, drag SWCreateMST.exe into the Command Prompt window.

   The command format is:

   `path_to_administrative_image\SWCreateMST.exe path_to_MSI_file_of_an_administrative_image list_of_properties`

   🖉 The command line must include all of the parameters used to create the original transform file, plus the global properties `UPGRADESWINSTALL` and `SWMIGRATE`.

   For example, if the original command was:

   ```
   C:\admin\SWCreateMST.exe "c:\admin\SolidWorks.msi"
   INSTALLDIR="C:\Program Files\your_folder" SOLIDWORKSSERIALNUMBER="xxxx
   xxxx xxxx xxxx"
   ADDLOCAL=SolidWorks,English,Manuals,Intel_Modules_AgeiaX86
   OFFICEOPTION=0 SERVERLIST=25734@your_server
   ```

   the update command must be:

   ```
   C:\admin\SWCreateMST.exe "c:\admin\SolidWorks.msi"
   INSTALLDIR="C:\Program Files\your_folder" SOLIDWORKSSERIALNUMBER="xxxx
   xxxx xxxx xxxx"
   ADDLOCAL=SolidWorks,English,Manuals,Intel_Modules_AgeiaX86
   OFFICEOPTION=0 SERVERLIST=25734@your_server UPGRADESWINSTALL=1
   SWMIGRATE="SolidWorks version service_pack"
   ```

   The transform file, `SolidWorks.mst`, is created in the folder that contains the administrative image.

   🖉 The original and update commands in this example are for installing the 32-bit version of SolidWorks on a 32-bit version operating system. If you are installing the 32-bit version of SolidWorks on a 64-bit operating system, you would use "`Intel_Modules_AgeiaX64`" in place of "`Intel_Modules_AgeiaX86`". If you are installing the 64-bit version of SolidWorks on a 64-bit operating system, you would not use either "`Intel_Modules_AgeiaX86`" or "`Intel_Modules_AgeiaX64`".

4. In the Active Directory, edit the `Published Apps` policy for the group or groups to deploy the SolidWorks software.

5. In the Group Policy dialog box, expand **Computer Configuration and Software Settings**.
6. Right-click **Software installation** and select **New** > **Package**.
7. Browse to the administrative image and click **Open**.
8. In the Deploy Software dialog box, select **Advanced published or assigned**, then click **OK**.
9. In the SolidWorks Properties dialog box, on the Modifications tab:
   a) Click **Add**.
   b) Browse to the `.mst` file.
   c) Click **Open**.
10. Click **OK**.

Removing an Installation

You can remove an installation on an individual computer or in a client/server environment.

1. Make sure no SolidWorks sessions are active.
2. If you wish to transfer the SolidWorks license to another computer, transfer the license to that computer before removing the SolidWorks installation on this computer.

   For information about transferring SolidWorks licenses, see

3. In Windows, open the Control Panel and double-click **Add or Remove Programs**.
4. In Add or Remove Programs, select the product component to remove.

   In some cases, you might find multiple SolidWorks Service Pack components for a given major release version. To remove a SolidWorks major release, remove all Service Pack elements for that major release in Add or Remove Programs.

5. Click **Remove**.
6. In the SolidWorks Installation Manager, on the Products to Remove screen:
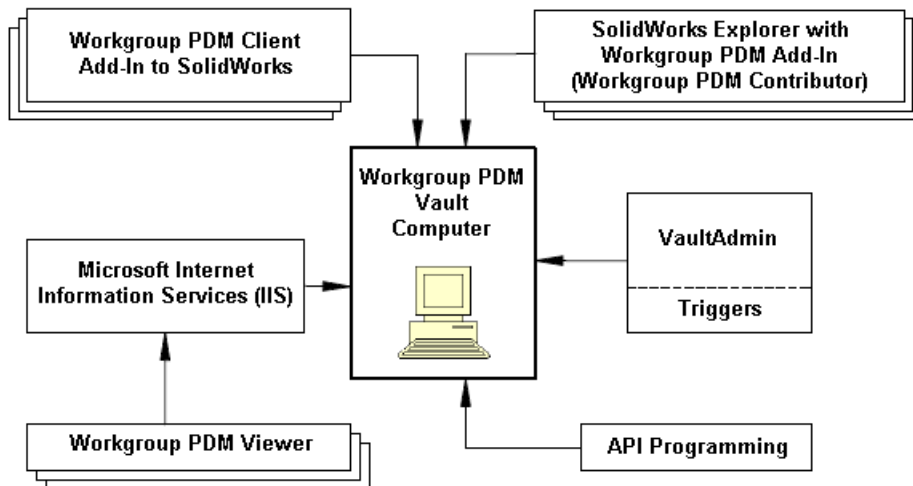   a) Select the products to remove.

      By default, all product components are selected.

   b) Clear the products you do not want to remove.
   c) Click **Remove Items**.

Workgroup PDM

The Workgroup PDM application is product data management software that runs inside the SolidWorks environment or as a standalone application inside SolidWorks Explorer. Workgroup PDM controls projects with procedures for check out, check in, revision control, and other administration tasks.

A Workgroup PDM system is illustrated as follows:

🖉
- A computer can contain only one Vault.
- The Vault directory cannot be a hidden or read-only directory.
- As a client/server application, Workgroup PDM performance depends on a robust network.
- If your users maintain a large number of documents, performance is improved if you create more projects or subprojects with fewer documents per project instead of keeping the documents in a smaller number of projects with more documents per project.

Installation

- Clients and the VaultAdmin, API, and Viewer can be installed on the same computer as the Vault, or on different computers.
- It is not necessary to install the Workgroup PDM Server to install the client API. The client API is installed during the Workgroup PDM Client installation.
- To configure the vault folder on a remote location, type the full Universal Naming Convention (UNC) path when prompted for the server destination.
- Installing or updating the vault requires a computer reboot.
- If the Workgroup PDM Server Service does not start automatically after rebooting, you must start it manually. In Windows Explorer, run `pdmwvault.exe` in the `install_dir\SolidWorks Workgroup PDM\Vault` folder.
- When you install a Workgroup PDM client, you must install `solidworksexplorer.exe` on each computer. This executable can be on a shared drive. The Workgroup PDM Vault, Viewer, API Client, and VaultAdmin are not installed on each client computer.

Licenses

- A license for Workgroup PDM API is included with the licenses of SolidWorks clients and Explorer clients. Check-in APIs require client licenses.
- A Workgroup PDM Viewer license must be installed with the vault to enable the Viewer.

Permissions

- The vault administrator should have write permission for installation and for the vault folder.
- The vault administrator should manage the vault folder from the VaultAdmin, not with Windows Explorer.
- Users should never have write permission in Windows Explorer for the vault folder.

Security, Ports, and Duplex Settings

- Workgroup PDM must be able to install as a Windows service, so other applications cannot be allowed to block or filter the service. For example, if anti-spyware tools such as Microsoft AntiSpyware display an alert requiring approval during the installation process, click **Allow** so the Workgroup PDM Server is installed as a Windows Service.
- If you install Workgroup PDM on a Windows XP system with Service Pack 2, a Windows 2003 server with Service Pack 1, or a computer with an active firewall, you must enable Workgroup PDM ports as exceptions in the firewall setup.

  Typically, the ports are:

  - 20000 (VaultAdmin)
  - 30000 (Data)
  - 40000 (Request)
  - 80 (Viewer)

  For information on changing the port settings, see *Troubleshooting for Vault Administrators* in the *Workgroup PDM* and *SolidWorks Explorer* help and the Microsoft Windows help for *Security Center*. To install the Workgroup PDM Vault and the Viewer concurrently, use the **Custom** option in the Setup Type dialog box.
- The duplex setting for servers and their corresponding ports should be the same; for example, both could be **100MB Full** or both could be **Auto**.

SolidWorks Toolbox Documents

SolidWorks Toolbox documents are common hardware components that usually do not require revision control and are not checked in to the Workgroup PDM Vault. See the *Workgroup PDM* online help for details about configuring SolidWorks Toolbox and other standard libraries.

Workgroup PDM Viewer

The Workgroup PDM software automatically sets up a virtual directory during installation using Microsoft Internet Information Services (IIS).

> You must install Microsoft Internet Information Services (IIS) before installing the Viewer. See Microsoft documentation for installation instructions.

The Workgroup PDM Viewer installation is subject to these restrictions:

- The **Web Site Content Directory** is the Viewer installation folder.
- You cannot use underscore characters in the name of the machine that hosts the Viewer.

- If you install the Viewer on Windows XP Service Pack 2, you must enable a port in the Windows Firewall for HTTP (port 80).
- The Viewer can point to only a single vault at any time.

Users access the Workgroup PDM Viewer by typing this URL into the address field of Internet Explorer:

```
http://ViewerComputerName/pdmweb/
```

Installing a Workgroup PDM Server

Installing the Workgroup PDM Server creates a Workgroup PDM vault, which is the database that stores the managed SolidWorks projects.

1. Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).
2. In the Welcome screen, select the following:

    - **Administrative Image and Server products**
    - **Install SolidWorks Workgroup PDM Server (vault) on this computer**

3. On the Serial Number screen, click **SolidWorks Workgroup PDM**, and enter the SolidWorks Workgroup PDM serial number.
4. Follow the SolidWorks Workgroup PDM Server installation instructions, selecting **Vault** as the **Setup Type**.
5. On the final screen, do one of the following:

    - If you have already installed SolidWorks Explorer, clear **Launch SolidWorks Explorer Install** and then click **Finish**.
    - If you have not installed SolidWorks Explorer, ensure that **Launch SolidWorks Explorer Install** is checked and then click **Finish**.

Configuring Workgroup PDM for Clients

After you have installed the Workgroup PDM server vault, you can then configure the vault for clients.

1. Log in to the VaultAdmin (installed with SolidWorks Explorer) with the default user name and password.
2. In the VaultAdmin, set up user and group accounts, project folders, permissions, and so on.

    For instructions, see *Workgroup PDM Help*.

3. Email client users with the server computer name and their user names and passwords, which are usually not the same as their Microsoft Windows user names and passwords.

Configuring the Workgroup PDM Viewer to Connect to a Different Vault Location

You can configure Workgroup PDM Viewer to connect to a different Vault location using the Windows Registry.

1. Update the registry path by changing the **Vault Computer** value:

HKEY_LOCAL_MACHINE\SOFTWARE\SolidWorks\SolidWorks 2009\PDMWorks Workgroup\Vault Computer

2. Reinstall the Workgroup PDM Viewer. In the Vault Computer Name dialog box, type the new computer name.

Configuring Mime Types for Document Downloading

You can configure mime types for document downloading using Windows Registry settings.

By default, the Viewer sends Open Document requests with the generic mime type "application/octet-stream." Microsoft Internet Explorer then opens the requested document with the supplied file extension. You can map additional mime types or turn off mime type mappings by modifying the Mime Types registry sub-key in Windows Registry.

To map additional mime types, create additional string values under this registry sub-key:

HKEY_LOCAL_MACHINE\SOFTWARE\SolidWorks\SolidWorks 2009\PDMWorks Workgroup\WebPortal\Mime Types

in the form:

"ext" = "mime type"

For example:

`"dxf" = "image/vnd.dxf"`

✏️ You must manually create this registry sub-key before you add the values.

To turn off mime type mappings, change the **Use Custom MIME Types** registry value to 0 (zero).

HKEY_LOCAL_MACHINE\SOFTWARE\SolidWorks\SolidWorks 2009\PDMWorks Workgroup\WebPortal\Use Custom MIME Types

Restarting the web server is not required to activate this setting.

Workgroup PDM Client Access

Workgroup PDM clients use Microsoft Windows login access control, have Web access through the Workgroup PDM Viewer, and are updated automatically whenever the Workgroup PDM Vault has been updated.

To log in as a Workgroup PDM client:	Client user names and passwords are not the same in Workgroup PDM as for Microsoft Windows login. Contact your system administrator for login information.
To connect to the Workgroup PDM Viewer:	Contact your system administrator for the URL to connect to the Viewer.
To update the Workgroup PDM clients:	The Workgroup PDM clients (SolidWorks add-in and Contributor) detect when the Vault has been updated. You do not need to download a service pack to update clients.

Workgroup PDM VaultAdmin Administration

After Workgroup PDM is installed, vault administrators should perform these tasks.

- Log in to the VaultAdmin.
- Change the vault administrator password.
- Create user accounts.
- Notify users of their account names and passwords.

For details, see *SolidWorks Workgroup PDM* in the main SolidWorks help.

Initial Login to Workgroup PDM VaultAdmin

When you first login to Workgroup PDM VaultAdmin, specify the Workgroup PDM administrator name and password, and the name of the computer hosting the Workgroup PDM Vault.

1. In Windows, click **Start** > **All Programs** > **SolidWorks *version*** > **Workgroup PDM VaultAdmin**.
2. In the dialog box, enter this information:

Name	pdmwadmin
Password	pdmwadmin
Vault computer	The host name of the computer where the vault is installed. If you run the VaultAdmin on the computer where the vault is installed, type localhost.

3. Click **Login**.

For security reasons, we recommend that you change the password for the administrator as soon as possible. Select the Users and Groups tab in the VaultAdmin.

Upgrading the Workgroup PDM Vault Server

You can upgrade the Workgroup PDM Vault Server using the SolidWorks Installation Manager **Check for Updates** functionality, a SolidWorks installation DVD, or a Workgroup PDM upgrade file manually downloaded to a local folder.

🖉 You can install only one version of Workgroup PDM Vault Server

1. Ask all users to log out of the Workgroup PDM vault and then lock it.
2. Backup the vault data.
3. Use Microsoft Windows Control Panel **Add/Remove Programs** to remove the current version of Workgroup PDM Server.
4. Install the new version of Workgroup PDM Server.

   - To use SolidWorks Installation Manager Check for Updates:

     1. In Windows, click **Start** > **All Programs** > **SolidWorks Installation Manager** > **Check for Updates**.
     2. In the Newer Versions Available dialog box, select **Download and install a new version**, select the version to install, and then click **Next**.

        The new version of the Installation Manager downloads and runs.

- To use a SolidWorks installation DVD, insert the DVD into the drive and follow the Installation Manager instructions.
- To use the manually downloaded Workgroup PDM installation file (`pdmwserver.exe`), double-click the file and then follow the instructions to upgrade the Workgroup PDM Server.

5. Use SolidWorks Task Scheduler to update the Workgroup PDM files in the vault.
6. Unlock the vault and then notify users that they can return to using it.

Upgrading Workgroup PDM Clients

Clients are prompted to update the first time they log in after the vault is updated. No service pack is required for client updates.

Client Status Relative to Vault	Action
Same version as vault	None
Same major release, newer than vault	None
Newer major release, newer than vault	Login is blocked
Same major release, older than vault	Optional update
Older major release, older than vault	Mandatory update

For details about updating Workgroup PDM clients, see *Update Clients* in the *Workgroup PDM Help*.

License Administration

After installation, you can activate SolidWorks licenses either by activating individual computers (license activation) or by installing and activating floating network licenses on a server (SolidNetWork Licensing).

Typically, installations on individual computers use license activation; however, you can install the software individually and then control the licenses with SolidNetWork License Manager. Conversely, although larger sites typically use floating network licenses, you can deploy the installations from an administrative image and then activate the serial numbers from the individual computers.

Administering Individual Licenses

Typically, administration of individual licenses consists of activating and transferring licenses directly from SolidWorks.

Activation is a product licensing and registration process for SolidWorks standalone (non-SolidNetWork License) licenses. Typically, if your product license has not yet been activated, SolidWorks tries to activate it when you start the product.

You can also transfer individual licenses, by clicking **Help** > **Transfer Licenses** in SolidWorks.

Activating a License

Before you can run SolidWorks products on an individual computer, you must activate the license for that computer.

Activation requires that you have Internet or email access. After activation, you do not need to be online to use the products. If you do not have email access on the computer where you use this product, you can save the activation file and send it from another computer.

This process:

- Activates all software licenses across the SolidWorks product line.
- Allows you to transfer licensing rights from one computer to another.
- Enables you to install and activate the SolidWorks products on one work computer and one home computer.
- Can be performed immediately (recommended) or within 30 days of installation.
- Eliminates the need for dongles. However, if you run previous versions of SolidWorks products that do not use activation, you need to retain the dongle.

For system administrators who have multiple serial numbers to manage, you can use the Administrative Image Option Editor.

To activate a license:

1. Start the SolidWorks application.
2. Start SolidWorks Product Activation using one of these methods:

   - If you have not yet activated the license for this computer, the SolidWorks Product Activation window appears automatically when you start SolidWorks.

- If the SolidWorks Product Activation window does not appear automatically, start activation by clicking **Help** > **Activate Licenses**.

3. Click the **Product** listing to specify the product whose license you want to activate.
4. Specify automatic activation over the internet or manual activation using email.
5. Supply email contact information.
6. Click **Next** to proceed.

- Internet: Activation occurs automatically.
- Email: Press **Save** to create a request file, then email the file to activation@solidworks.com. When you receive the email containing the response file, extract the file from email and then click **Open** to load it.

  🖍 If necessary, you can exit and rerun the activation procedure to open the response file.

7. Click **Finish**.

Transferring a License

To transfer a license to a new or upgraded computer, first transfer it to the license key server at Dassault Systèmes SolidWorks Corporation, then reactivate it on the other computer.

🖍 If you were unable to transfer the license to Dassault Systèmes SolidWorks Corporation (because of hard drive failure, for example), reactivation is still successful in most cases. If not, contact your VAR for help.
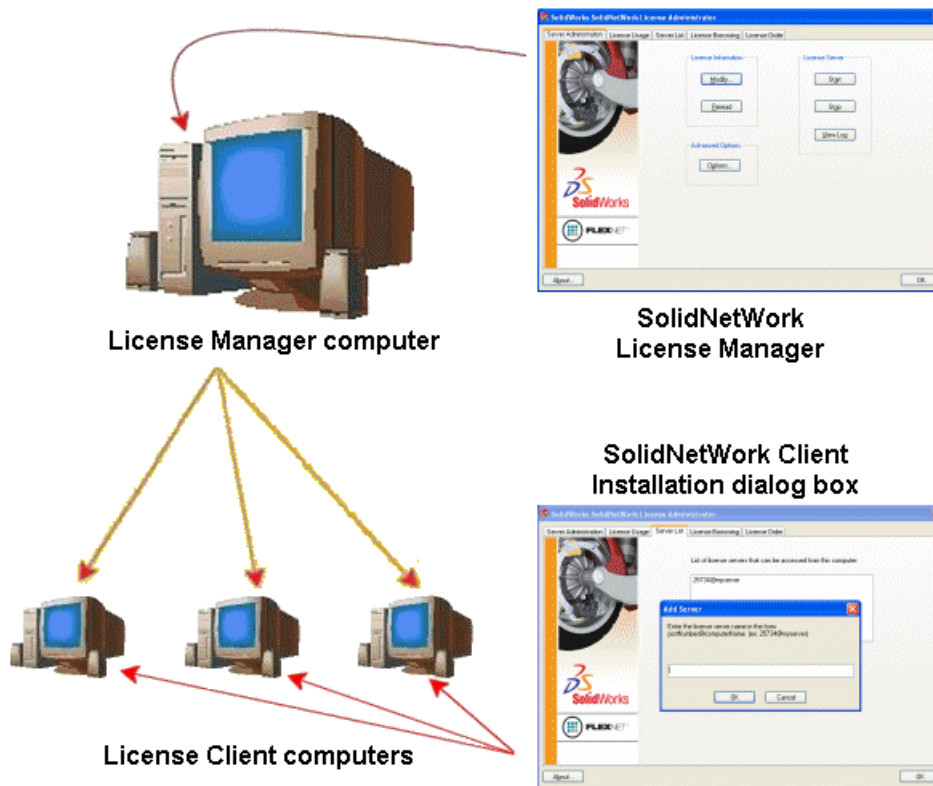
To transfer a license back to the server:

1. In SolidWorks, click **Help** > **Transfer Licenses**, and follow the instructions.
2. To reactivate, go to the new or upgraded computer and activate the license like the initial activation.

Administering Licenses Using a SolidNetWork License Manager

SolidNetWork License Manager supports multiple license clients by distributing licenses to clients on the network. This allows the number of users to exceed the number of licenses by using floating licenses.

This image depicts a typical SolidNetWork licensing installation:

License Manager computer

SolidNetWork
License Manager

SolidNetWork Client
Installation dialog box

License Client computers

This image illustrates:

- SolidNetWork License Manager software is installed and activated on the license manager computer only.
- License client computers do not have any additional SolidNetWork-specific software installed.

  If you specify a SolidNetWork License serial number when installing, SolidWorks Installation Manager identifies that computer as a SolidNetWork license client. Installation Manager prompts you for the SolidNetWork license manager computer name and port number to connect the client to the license manager.

- The license manager and all license clients must have TCP/IP (Transmission Control Protocol/Internet Protocol) connectivity.

Activating a SolidNetWork License

Before users can run SolidWorks products on their individual computers, you must activate a SolidNetWork license on the SolidNetWork License Manager computer.

Activation requires that you have Internet or email access. After activation, you do not need to be online to use the products. If you do not have email access on the computer where you use this product, you can save the activation file and send it from another computer.

This process:

- Activates the SolidWorks SolidNetWork License Manager with SolidNetWork licenses across SolidWorks products.
- Allows you to transfer SolidNetWork licensing rights from one SolidNetWork License Manager computer to another.

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- Enables you to install and activate the SolidNetWork licensing rights on one SolidNetWork License Manager computer.
- Must be performed before you can use SolidNetWork licenses.
- Eliminates the need for dongles. If you activate a SolidNetWork license for this version of SolidWorks, SolidNetWork licenses for previous versions of SolidWorks are activated. (You no longer need to retain the dongle.)

To activate a SolidNetWork license:

1. Start the SolidNetWork License Manager (**Start** > **Programs** > **SolidWorks** *version* > **SolidNetWork License Manager** > **SolidNetWork License Manager**).
2. If you have not yet activated the SolidNetWork license, you are asked whether you want to activate it now. Click **Yes** to activate.
3. Under **How would you like to activate?**, select **Automatically over the internet (recommended)**.
4. Under **Contact Information**, specify an email contact address.

The Activation/Reactivation Succeeded dialog box displays the products that have been activated.

Transferring a SolidNetWork License

To transfer a SolidNetWork license to a new or upgraded computer, first transfer the license to the license key server at Dassault Systèmes SolidWorks Corporation, then reactivate it on the other computer.

> If you were unable to transfer the license to Dassault Systèmes SolidWorks Corporation (because of hard drive failure, for example), reactivation is still successful in most cases. If not, contact your VAR for help.

To transfer a SolidNetWork license back to the server:

1. Start the SolidNetWork License Manager (**Start** > **Programs** > **SolidWorks** *version* > **SolidNetWork License Manager** > **SolidNetWork License Manager**).
2. Under **License Information**, click **Modify**. The SolidWorks Product Activation wizard starts.
3. Under **What would you like to do?**, select **Transfer a software license** and then click **Next**.
4. Under **How would you like to transfer?**, select **Automatically over the internet (recommended)**.
5. Under **Contact Information**, specify an email contact address.
6. Click **Next**. The product license transfer completes.

Installing and Configuring SolidNetWork Licensing

Configuring SolidNetWork Licensing consists of installing the SolidNetWork License Manager on the license manager computer and then installing SolidWorks on client computers using the SolidNetWork license serial number.

Installing and Activating a License Manager
The SolidNetWork License Manager supports multiple license clients. It distributes licenses to clients on the network.

1. Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).

   a) For type of installation, select the following:

      - **Administrative Image and Server products**
      - **Install SolidNetWork License Manager (distributed licensing) on this computer**

   b) For serial number, provide the SolidNetWork License serial number.

   c) Follow the online instructions to complete the installation in SolidWorks Installation Manager.

2. To activate SolidNetWork licensing, start the SolidNetWork License Manager.

   a) When prompted to activate, click **Yes**.

   b) On the SolidNetWork License Server Information screen:

      - If your company uses a firewall, select **A firewall is in use on this server**.

        For more information, see Using SolidNetWork Licenses with Firewalls on page 63.

      - Use the default value for **Port Number** (and for **Vendor Daemon Port Number** if you have a firewall) or specify the port numbers your company uses.

        🖊 The port number is a TCP/IP port number in the 1024-60000 range, used exclusively by this process. Normally, the default (25734) is suitable and is unlikely to be the same as any other FLEXnet server process currently on the license manager computer.

   c) On the Activate Your SolidWorks Product screen:

      - Select the Internet or email procedure.
      - Supply email contact information.
      - Click **Next** to proceed.

        - Internet: Activation occurs automatically.
        - Email: Press **Save** to create a request file, then send the file to activation@solidworks.com. When you receive the email containing the response file, extract the file from email and then click **Open** to load it.

          🖊 If necessary, you can exit and rerun the activation procedure to open the response file.

3. Click **Finish**.

Installing a License Client
When installing SolidWorks on SolidNetWork License Client computers, the serial number identifies that computer as a SolidNetWork License Client.

1. Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).

2. On the Welcome screen click **Individual (on this computer)** if this is the first SolidWorks installation on this computer for this major release.

✏️ If you are upgrading from an existing release, and are changing from individual licensing to SolidNetWork licensing, click **Modify the individual installation (on this computer)**. For details, see Upgrading from Individual Licensing to SolidNetWork Licensing on page 68.

3. When prompted for a serial number, enter the SolidNetWork License serial number.
4. In the Add Server dialog box, type the SolidNetWork License Manager's address in the form:

   port@ComputerName

   The port number must be the same one used for the license manager computer. The default is `25734` (as described in Installing and Activating a License Manager on page 58).

   ✏️ Installation Manager will attempt to validate that this SolidNetWork License Manager is accessible. If you have not yet installed a license manager, this validation will fail; however, you still can proceed with the license client installation.

5. Complete the SolidWorks installation by following the Installation Manager instructions.

Install SolidNetWork License Managers Before Installing License Clients
You should install and configure SolidNetWork License Manager on at least one computer prior to installing any SolidNetWork license client computers.

When installing with a SolidNetWork License serial number, SolidWorks Installation Manager prompts you to enter the SolidNetWork License Manager computer name and port number, and then attempts to validate that information.

If you have not yet installed SolidNetWork License Manager on the computer you specified, this validation cannot succeed. However, in this case validation failure does not indicate whether you have supplied correct license manager information. If the license manager is installed, validation failures indicate that you have specified incorrect license manager information.

Triad License Manager Installation
Triad license managers use three computers for extra stability. If the computer managing the license file is not available, one of the other two computers takes over.

Requirements for using triad license managers:

• Three hardware dongles are required.
• Install triad license managers only on stable computers not scheduled for frequent shutdowns.
• The three license manager computers should be on the same subnet. Never configure triad license managers with slow communications or dial-up links.
• The system administrator should detect when a license manager computer goes down and should restart it as soon as possible.

   ✏️ If only two license manager computers are running, the situation is worse than a single license manager computer, because there are two points of failure.

Installing the Initial Triad License Manager
Triad license managers use three computers for extra stability. This process shows how to install SolidNetWork License Manager on the first computer in the triad.

1. On the first triad license manager computer, install the SolidNetWork License Manager using SolidWorks Installation Manager:

    a) Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).

    b) For type of installation, select the following:

        -   **Administrative Image and Server products**
        -   **Install SolidNetWork License Manager (distributed licensing) on this computer**

    c) For serial number, provide the SolidNetWork License serial number.

    d) Follow the SolidWorks SolidNetWork License Manager Setup instructions to complete the installation.

        The serial number in the license manager installation wizard is pre-set to the SolidWorks SolidNetWork serial number you provided when you started the Installation Manager.

2. Using the SolidNetWork License Manager, configure the initial triad license manager computer:

    a) Start the SolidNetWork License Manager (**Start** > **Programs** > **SolidWorks version** > **SolidNetWork License Manager** > **SolidNetWork License Manager**). Since this is the first time you are running the SolidNetWork License Manager on this computer, you are asked whether you would like to activate now.

    b) Click **No**.
        The Server Administration tab of the SolidWorks SolidNetwork License Manager appears.

    c) Under **Advanced Options**, click **Options**.
        The Advanced Options dialog box appears.

    d) Under SolidNetWork Server License Mode, click **Use a License File** and click **OK**.
        The Advanced Options dialog box closes.

    e) Under **License Information**, click **Modify**.
        The SolidNetWork License File Wizard appears.

    f) Under **Server Type**, select **Triad** and click **Next**.

    g) On the Server Information pane, verify that the computer name, dongle ID, and port number for the initial triad license manager computer are correct, and click **Next**.

        If your computer uses a firewall, you might have to make some additional specifications on this screen. For more information, see Using SolidNetWork Licenses with Firewalls on page 63.

    h) On the Triad Partner Server Information screen, enter the computer name and dongle ID values for the other two triad license manager computers.

        This ensures that if one license manager is unavailable, client computers can get a license from the other two servers.

i) On the Licenses screen, click **Read** to browse to and open the SolidNetWork license file.
The licenses appear in the SolidNetWork License File wizard.
j) Click **Finish**.
k) Click **OK** to exit the SolidNetWork License File wizard and then **Finish** to exit the SolidNetWork License Manager.

Installing the Triad License Manager on Subsequent Computers
Triad license managers use three computers for extra stability. This process shows how to install SolidNetWork License Manager on the second and third computers in the triad.

It is assumed that you already have installed at least one triad license manager on another computer.

1. On the triad license manager computer, install the SolidNetWork License Manager using SolidWorks Installation Manager:
   a) Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).
   b) For type of installation, select the following:
      - **Administrative Image and Server products**
      - **Install SolidNetWork License Manager (distributed licensing) on this computer**
   c) For serial number, provide the SolidNetWork License serial number.
   d) Follow the SolidWorks SolidNetWork License Manager Setup instructions to complete the installation.

   The serial number in the license manager installation wizard is pre-set to the SolidWorks SolidNetWork serial number you provided when you started the Installation Manager.

2. Using the SolidNetWork License Manager, configure the triad license manager computer:
   a) Start the SolidNetWork License Manager (**Start** > **Programs** > **SolidWorks version** > **SolidNetWork License Manager** > **SolidNetWork License Manager**). Since this is the first time you are running the SolidNetWork License Manager on this computer, you are asked whether you would like to activate now.
   b) Click **No**.
   The Server Administration tab of the SolidWorks SolidNetwork License Manager appears.
   c) Under **Advanced Options**, click **Options**.
   The Advanced Options dialog box appears.
   d) Under SolidNetWork Server License Mode, click **Use a License File** and click **OK**.
   The Advanced Options dialog box closes.
   e) Under **License Information**, click **Modify**.
   The SolidNetWork License File Wizard appears.
   f) Under **Server Type**, select **Triad** and **Synchronize server/license information** and then click **Next**.
   g) On the Server Information pane, verify that the computer name, dongle ID, and port number for this triad license manager computer are correct, and click **Next**.

If your computer uses a firewall, you might have to make some additional specifications on this screen. For more information, see Using SolidNetWork Licenses with Firewalls on page 63.

h) On the Triad Partner Server Information screen, enter the computer name and dongle ID values for the other two triad license manager computers.

This ensures that if one license manager is unavailable, client computers can get a license from the other two servers.

i) On the Licenses screen, click **Read** to browse to and open the SolidNetWork license file.

Typically, the licenses file is located on `\licenses\sw_d.lic` on the first triad license manager computer you configured.

The licenses appear in the SolidNetWork License File wizard.

j) Click **Finish**.

A SolidNetWork License File Wizard warning appears.

k) Click **OK** to exit the SolidNetWork License File wizard and then **Finish** to exit the SolidNetWork License Manager.

If you are configuring the second triad license manager computer, repeat this process for the third triad license manager computer.

Using SolidNetWork Licenses with Firewalls

If your computer uses a firewall, you must configure ports to support SolidNetWork Licensing.

In the firewall software, you must add two SolidNetWork License ports (for example, `SNL-in-port` and `SNL-out-port`) to the firewall exceptions list. For details about configuring these ports, consult with your system administrator.

If you are using Windows Firewall on Windows XP SP2 and Windows Server 2003 SP1, you must modify ports on the SolidNetWork license manager computer and modify the SolidNetWork license file to distribute licenses to the license client computers.

Modifying License Manager Computer Ports for Windows Firewall

If you are using Windows Firewall, you must modify firewall port settings on the SolidNetWork license manager computer.

1. In Windows, open the Control Panel and then double-click **Security Center**.
2. Click **Windows Firewall**.
3. On the Exceptions tab, click **Add Port**.
4. Type a **Name** (for example, `SNL-in-port`) and a **Port Number** (for example, `25734`) and then click **OK**.
5. For the second port, click **Add Port** on the Exceptions tab again, specify the other port's name and number (for example, `SNL-out-port` and `25735`), and then click **OK**. You will reference this port number when modifying the SolidNetWork license file.

If you use Workgroup PDM, you must modify Workgroup PDM port numbers in Windows Firewall. Typically, these ports are 20000 (VaultAdmin), 30000 (Data), 40000 (Request), and 80 (Viewer).

6. Close Windows Security Center and Control Panel.

Modifying SolidNetWork License Manager Settings for Windows Firewall
If you are using Windows Firewall, you must modify the SolidNetWork license file to use the firewall ports to distribute licenses to the license client computers.

1. Start the SolidNetWork License Manager.
2. On the Server Administration tab, under **License Server**, click **Stop**. (Click **Yes** when prompted for confirmation and **OK** to complete stopping the license server.)
3. Under **License Information**, click **Modify**.
4. Under **Server Type**, select the server arrangement for your network (**Single** or **Triad**) and then click **Next**.
5. Under **Server Information**:
    a) Select **A firewall is in use on this server.**
    b) Port Number and Vendor Daemon Port Number should contain the port numbers specified for the Windows Firewall (for details, see Modifying License Manager Computer Ports for Windows Firewall on page 63).
    c) Click **Next** and then click **Finish**.
6. On the Server Administration tab, under **License Server**, click **Start**.
7. Click **OK** to close the **SolidNetWork License Manager**.

SolidNetWork License Administration

Tasks include starting, upgrading, and uninstalling the license manager, and retrieving, borrowing and returning licenses.

Starting the SolidNetWork License Manager
You can start the SolidNetWork License Manager from the Windows **Start** menu.

In Windows, click **Start** > **All Programs** > **SolidWorks *version*** > **SolidNetWork License Manager** > **SolidNetWork License Manager**.

Upgrading the SolidNetWork License Manager
For major releases of SolidWorks, upgrading the license manager consists of uninstalling the old license manager, installing a new one, and re-activating the SolidNetWork license.

The license manager for a newer release can distribute licenses for client computers remaining at the previous release as well.

1. Start the SolidNetWork License Manager to verify that no client computers on the network are running SolidWorks.
    The License Usage tab shows clients running SolidWorks.
2. Uninstall the old license manager.
3. Install the new license manager and re-activate the SolidNetWork license on the license manager computer.

If you are upgrading the SolidNetWork License Manager on a computer that also runs the SolidWorks application, upgrade SolidNetWork license managers and clients before installing any new SolidWorks application service packs on that computer.

If you are upgrading the license manager on a computer that hosts administrative images, upgrade the license manager before upgrading the administrative images.

Uninstalling the SolidNetWork License Manager
When upgrading the SolidNetWork License Manager, you must uninstall the old license manager before installing the new one.

1. Start the SolidNetWork License Manager.
2. Verify that no client computers on the network are running SolidWorks.

   The License Usage tab shows clients running SolidWorks.

3. If you wish to transfer the SolidNetWork license to another computer, transfer the license to that computer before removing the SolidNetWork License Manager from this computer.

   🖉 For information about transferring SolidNetWork licenses, see Transferring a SolidNetWork License on page 58.

4. On the Server Administration tab, under **License Server**, click **Stop**.
5. Click **OK** to exit the license manager.
6. In Windows, open **Control Panel** > **Add or Remove Programs**.
7. Select **SolidWorks SolidNetWork License Manager**, click **Remove**, and confirm the deletion.
8. In Windows Explorer, delete the `SolidWorks SolidNetWork License Manager` folder.

   (Typically, this folder is located under `Program Files` in the SolidWorks installation directory.)

You now are ready to install a new version of the license manager.

Deleting SolidNetWork License Files
Delete old and expired SolidNetWork license files before upgrading to new ones.

1. Start the SolidNetWork License Manager.
2. On the Server Administration tab, under **License File**, click **Modify**, and then click **Next** until you reach the Licenses screen.
3. On the Licenses screen, select the old or expired SolidNetWork license file, then click **Remove**.
4. Click **Finish** and then click **OK**.

Retrieving Client Licenses
To terminate a client SolidNetWork connection, a system administrator can stop the license and return it to the free license pool.

- The client must have been connected to the license manager for at least two minutes.
- You can configure the SolidNetWork timeout feature to be the 15-minute minimum. After 15 minutes of idle time, that license is returned to the free license pool.

For details, see
SolidNetWork_License_Manager_install_dir\Docs\flexuser\licensingenduserguide.pdf.

1. Click **Start** > **Run**, type cmd, and then click **OK** to open a Windows command window.
2. In the command window, enter:

```
cd SolidNetWork_License_Manager_install_dir\utils
lmutil lmremove feature user computer display
```

where feature, user, computer, and display are the terms shown on the License Usage tab of the SolidNetWork License Manager.

For example:

```
lmutil lmremove solidworks carlos designcmp1 corpserver
```

🖉 You must terminate the license for each add-in product (PhotoView 360, FeatureWorks, etc.) separately.

Borrowing and Returning SolidNetWork Licenses
You can borrow a SolidNetWork license to use the software from a remote location. Since the license is removed from the pool of available licenses, a borrowed license must be returned for it to become available again.

🖉 Borrowed licenses are valid only for the system on which they were borrowed; they cannot be transferred.

The license is lent for a specified period of time, up to 30 days (or as defined by the system administrator), and the license is removed from the pool of available licenses.

System administrators can customize borrowing to:

- Define users or groups that are allowed to borrow licenses
- Set the number of licenses that cannot be borrowed

For details, see
SolidNetWork_License_Manager_install_dir\Docs\flexuser\licensingenduserguide.pdf.
Borrowing SolidNetWork Licenses
You can borrow a SolidNetWork license to use the software from a remote location.

You must be connected to the SolidNetWork license manager to borrow the license, but not necessarily to use it.

1. Start the SolidNetWork License Manager.
2. On the License Borrowing tab:
   a) Select a date in **Borrow until**.
   b) Select the products to borrow, then click **Borrow**, and click **Yes** to confirm.
   c) Click **Refresh**. The **Expire Date** column indicates when licenses are due back.
3. Click **OK**.

🖉 To run SolidWorks, you must borrow a SolidWorks license. SolidWorks Professional and SolidWorks Premium enable only the add-ins associated with the package, but not SolidWorks.

Returning a Borrowed SolidNetWork License
Since a borrowed SolidNetWork license is removed from the pool of available licenses, that borrowed license must be returned for it to become available again.

1. Start the SolidNetWork License Manager.
2. On the License Borrowing tab:
   a) Select the products to return, click **Return**, and then click **Yes** to confirm.
   b) Click **Refresh**. The **Expire Date** column becomes blank, indicating that the licenses have been returned.
3. Click **OK**.

It is not necessary to return licenses after the expiration date. The license is returned to the pool on the SolidNetWork license manager at midnight on the expiration date and the applications on the remote system no longer run.

Monitoring License Idle Time

SolidWorks can register idle time with the SolidNetWork License Manager so that licenses that have not been used for a certain amount of time are returned to the free license pool.

1. In the SolidNetWork License Manager installation directory, create a text file:

   SolidNetWork_License_Manager_install_dir\Licenses\sw_d.opt

2. Add the following line to the file:

   TIMEOUTALL *seconds*

   where *seconds* is a number greater than or equal to 900 (15 minutes), which is the minimum time allowed.

Installing Temporary Licenses

The temporary "ANY" license can be used temporarily when the original SolidNetWork license is not available.

By installing the "ANY" license file, you agree to implement the traditional license upon receipt of a SolidNetWork license from Dassault Systèmes SolidWorks Corporation. Compliance with license restrictions still is required.

To install an "ANY" license:

1. If you are an existing SolidNetWork user, uninstall the SolidNetWork License Manager and delete the installation folder.
2. Start SolidWorks Installation Manager (as described under Initiating SolidWorks Installation on page 6).
   a) For type of installation, select the following:
      - **Administrative Image and Server products**
      - **Install SolidNetWork License Manager (distributed licensing) on this computer**
   b) For serial number, provide the SolidNetWork License serial number.
   c) Follow the online instructions to complete the installation in SolidWorks Installation Manager.
3. Start the SolidNetWork License Manager (**Start** > **Programs** > **SolidWorks *version*** > **SolidNetWork License Manager** > **SolidNetWork License Manager**).
4. If you are prompted to activate, click **No**.

The Server Administration tab of the SolidWorks SolidNetwork License Manager appears.

5. Under **Advanced Options**, click **Options**.
   The Advanced Options dialog box appears.

6. Under SolidNetWork Server License Mode, click **Use a License File** and click **OK**.
   The Advanced Options dialog box closes.

7. Under **License Information**, click **Modify**.
   The SolidNetWork License File Wizard appears.

8. Under **Server Type**, select **Single** and click **Next**.

9. When you see this message:

   ```
   Could not read the dongle ID number.
   ```

   click **Cancel**.

   The ID "**ANY**" appears automatically.

   🖉 If your computer uses a firewall, you might have to make some additional specifications on this screen. For more information, see Using SolidNetWork Licenses with Firewalls on page 63.

10. Continue with the installation.

Upgrading from Individual Licensing to SolidNetWork Licensing
You can upgrade a computer from individual SolidWorks licensing to SolidNetWork licensing by specifying the SolidNetWork license serial number in SolidWorks Installation Manager.

If a computer does not have SolidWorks installed, you can specify SolidNetWork licensing by entering the SolidNetWork serial number when installing with SolidWorks Installation Manager.

If a computer already has SolidWorks installed and you are switching from individual licensing to SolidNetWork Licensing, you can Modifying an Installation on page 10 for the client computer and specify the SolidNetWork License serial number in place of the individual license serial number when running SolidWorks Installation Manager.

Using SolidNetWork Licenses with Add-in Products
You can use the SolidNetWork License Manager to distribute licenses for SolidWorks core add-in products (such as FeatureWorks, Workgroup PDM Server, and so on).

- An equal number of SolidWorks software licenses and SolidWorks add-in licenses is not required.
- To check out a license for an add-in product, you first must check out a license for SolidWorks.
- To borrow an add-in product license, you must borrow both the SolidWorks software license and the SolidWorks product license that includes the add-in. For example, for PhotoWorks you would borrow either a SolidWorks Professional or SolidWorks Premium product license.
- To release an add-in product license:

  - In SolidWorks, click **Tools** > **Add-ins** and clear the add-in.
  - Exit SolidWorks.

Using the Same Computer for the License Manager and Administrative Image Installations

You can configure one computer on a network to act both as a license manager distributing SolidNetWork licensing and as an administrative image location from which users can install SolidWorks software.

Client computers can obtain licenses from the license manager and then install SolidWorks from the same computer.

This configuration requires that the license manager and administrative image be installed on the same computer. Install the license manager first and then create and deploy the administrative image.

FLEXnet Publisher License Management

SolidNetWork Licensing is based on FLEXnet Publisher® license management.

For more information about FLEXnet Publisher license management, see the FLEXnet Publisher *License Administration Guide* in the SolidNetwork License Manager installation directory (

SolidNetwork_License_Manager_install_dir\Docs\flexuser\licensingenduserguide.pdf).

For the latest information about FLEXnet Publisher license management, contact Acresso® Software.

Troubleshooting

Consult these topics to assist you in troubleshooting installation problems.

Installation Manager Log Files

The Installation Manager log files are available to assist you in troubleshooting installation problems.

The Installation Manager log files are located with the other SolidWorks application log files.

- For Windows Vista or later:

  `C:\Users\`*`username`*`\AppData\Roaming\SolidWorks\Installation Logs\`*`installed_version`*

- For Windows operating system versions prior to Vista:

  `C:\Documents and Settings\`*`username`*`\Application Data\SolidWorks\Installation Logs\`*`installed_version`*`\`

🖉 To see the log file directories, you might have to configure the Windows Folder options to **Show hidden files and folders**.

If the application data is located on a drive other than `C:`, adjust the directory path accordingly.

This directory contains the `SummaryIMLog_`*`nnnnn_nnnnn_nnnn_nnnnn`*`.txt` file, a general log of the Installation Manager progress.

The `Other Logs` subdirectory contains more detailed installation log information.

SolidWorks Installation Manager Download Folder

The SolidWorks Installation Manager can download the installation files it needs to a folder that you specify. This topic provides answers to frequently asked questions about the download folder.

1. How do I install the files in this folder? on page 71
2. Can I install MSI files manually from this folder using msiexec or some other application? on page 71
3. Can I select the same download folder for each version I install? on page 71
4. I downloaded some products manually from the Dassault Systemes SolidWorks Corporation Web site. Can the Installation Manager install these? on page 71
5. Can the files I download be shared by other users? on page 71
6. How do I know when a new version of SolidWorks products becomes available? on page 72
7. The Installation Manager shows only SW2006 or SW2007 installs, but not the current version. on page 72
8. The Installation Manager does not let me select a different service pack of a product to install. on page 72
9. The Installation Manager is prevented from downloading files. on page 72

How do I install the files in this folder?

Run `setup.exe`. You may still be required to download some files if you select to install items that were not downloaded previously.

Do not install using `msiexec` (see below).

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Can I install MSI files manually from this folder using msiexec or some other application?

No. The Installation Manager does not download some files that would be required when running `msiexec`.

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Can I select the same download folder for each version I install?

Download folders are version-specific. Starting in SolidWorks 2010, SolidWorks Installation Manager automatically creates a version-specific sub-folder within the specified folder.

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I downloaded some products manually from the Dassault Systemes SolidWorks Corporation Web site. Can the Installation Manager install these?

Starting in SW2009, files downloaded manually can be used by the Installation Manager. However, it is better to run the Installation Manager, selecting the option to download files. The Installation Manager shows which files you need based on what you are installing, providing links to download each file. This option can be used when the built-in download page does not work (most likely because of downloading restrictions enforced by a proxy server).

    If Microsoft Internet Explorer asks whether you want to save or run files during a download or installation, always specify save.

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Can the files I download be shared by other users?

Yes. To do so, download to a network share. Each client can then run the Installation Manager from the share to install.

No further downloads will be required unless a user selects some items to install that were not selected during the initial download or if a computer that uses this download to install has a different set of prerequisites than the computer to which you downloaded the files.

If you are downloading on behalf of others but are not installing, select the **Download Only** option when installing. This option will allow you to select products regardless of what may be installed on your system. However, this option cannot be used to download patches or to create administrative images.

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How do I know when a new version of SolidWorks products becomes available?

You will be informed when a new version is available in the following cases:

- If you run Check for Updates (**Start** > **All Programs** > **SolidWorks Installation Manager** > **Check For Updates**)
- If you previously scheduled a Check for Updates and run SolidWorks
- If you select an installed version of SolidWorks in Windows **Add/Remove Programs** and click **Change**
- If you run an older version of the Installation Manager (`sldIM.exe`) from a download folder

In these cases, you may select to download the newer version. The Installation Manager will download and run the newer version of itself, and not any installation files. After the newer version of the Installation Manager runs, you may select a set of installation actions. The Installation Manager will then determine which installation files are needed (and not downloaded previously) and will offer to download those files.

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The Installation Manager shows only SW2006 or SW2007 installs, but not the current version.

You are running an older version of the Installation Manager that is no longer supported for the current release. It can only download service packs for SW2006 and SW2007 versions.

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The Installation Manager does not let me select a different service pack of a product to install.

That is by design. Each Installation Manager is built only to install a specific version of SolidWorks products. If you want to install a different service pack, you need to run that version's Installation Manager. To find the latest version available, see How do I know when a new version of SolidWorks products becomes available? on page 72

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The Installation Manager is prevented from downloading files.

If proxy server restrictions prevent applications from downloading files, the Installation Manager cannot download files automatically.

You can download files manually and then install them using the Installation Manager. Select the option in the Installation Manager to download individual files (for example, **Conduct manual download** on the Download Options page). Based on your product selections and system requirements, a web page appears in a separate window containing links to download all required files manually. These links are identical to those provided by the SolidWorks Customer Portal, but you do not need to search for the files.

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Not all of my SolidWorks products are updated to the latest service pack, but Check For Updates tells me I have the latest version?

Check For Updates only tells you whether you have installed (or are running) the latest version of the Installation Manager.

If you want to update all products to the latest version that is installed, in **Add/Remove Programs**, select the latest installed version of SolidWorks and click **Change**.

Can I copy a DVD to a download folder?

Yes, but installations may fail unless you copy the entire contents of each DVD released for that platform (x86 or x64). The Installation Manager looks for the files `swdata*.id` to determine the version. If these files are not present, it will fail to recognize any of the installation files that you copied and will require that they be downloaded.

Troubleshooting SolidNetWork Licenses

If the SolidNetWork License Manager installation fails, check for these possible explanations.

Accessing the SolidNetWork License log files

When a license error occurs, the answer often appears in the *SNL_Manager_install_dir*\lmgrd.log file.

Could not obtain a license for SolidWorks. Cannot connect to license server.

Typically, this problem is caused by port settings.

- Port settings between the SolidNetWork license manager and the license client might not match.
- Additional port settings for multiple versions of SolidNetWork license managers might not have been specified.

For details about port settings for SolidNetWork license managers and clients, see Using SolidNetWork Licenses with Firewalls on page 63.

Troubleshooting Upgrades

If you encounter problems while upgrading to a new SolidWorks release, check for these possible explanations.

Upgrading from a Source That Has Moved

When SolidWorks is installed, the Installation Manager remembers the location of the installation sources (disk media, disk image on a network, etc.). It expects the sources to be available at the same location when updating.

If the source moves or is deleted, or if the source media changes, the Installation Manager produces an error message. However, you can continue the upgrade by browsing to another location as long as it contains all the same file and folder structure.

SolidWorks Installation Manager Application Help

The SolidWorks Installation Manager is a standalone application that installs and updates your SolidWorks products.

You can run the Installation Manager:

- From a SolidWorks DVD or a shared source or download folder (when installing a new version)
- From the Control Panel Add or Remove Programs page (when modifying or removing an existing version)
- Through a scheduled or manual update check (when detecting, downloading, and installing new release versions)

Each version of SolidWorks has an Installation Manager specific to that version.

For versions prior to SolidWorks 2008, use the SolidWorks 2007 Installation Manager or download product versions manually from the SolidWorks Support Web site.

Check for Updates Results

Check for Updates looks for later versions of SolidWorks.

You can run Check for Updates manually:

- In Windows, click **Start** > **Programs** > **SolidWorks Installation Manager** > **Check for Updates**
- In SolidWorks, click **Help** > **Check for Updates**.

Check for Updates is not available on clients installed from administrative images.

Check for Updates can run automatically as a result of configuring a SolidWorks installation option.

Newer Version Available

Installation Manager has detected a newer version of SolidWorks products. You can update to a newer version by selecting the Installation Manager for that version.

This newer version can download installation files based on the products you specify in the installation. You can share the downloaded files with other users who install using the Installation Manager.

These downloads are only for use by the Installation Manager.

These options are available:

- If there are multiple versions available, you can select a version from the list.
- For details about this version, click **Check to see what this upgrade includes**.

Clicking **Next** obtains and runs the selected version of the Installation Manager.

No New Downloads Available/Unable to Check for New Versions

Installation Manager has not found a newer version of SolidWorks products, either because you are installing the latest version available or because it could not connect to the SolidWorks site to check.

Scheduling Automatic Update Checks

You can schedule future update checks by selecting **Check for updates every ____ days** and specifying the frequency of the update check.

The automated update check runs when you start SolidWorks.

Client Administrative Options

Specify administrative options for the client installations running from an administrative image.

Creating a Diagnostic Log

Creating a diagnostic log during client installation helps troubleshoot installation problems. However, the client installations are slower.

Running the Installation as a Different User

Enter a user name and password to run the installation as a user other than the one currently logged in.

For example, if your users do not have administrative privileges, you can create an administrative account with full privileges on each client computer and specify it here.

This feature is not supported on Windows Vista or later.

Running a Program Before or After Installation

Browse to the program to run before or after installation.

Use this option to instruct the administrative image installer to run the script before beginning the installations or after the installation completes.

SolidWorks Customer Experience Program

Specify whether you want the client to participate in the SolidWorks Customer Experience Program.

Performance information is sent to SolidWorks Corporation to help improve the product. The process is transparent.

SolidWorks uses this information to determine trends concerning product usage and quality. Our goal is to deliver the highest quality product and the performance logs are a useful tool in this process.

All information contained within the log files is treated confidentially. It is not used for any other purpose and no one will contact you.

For further information, see http://www.solidworks.com/pages/services/CustomerExperienceFeedbackProgram.html.

Client Installation Options

When creating an administrative image, you can specify how the client installations are to be run.

Client Installations of Major Versions

When installing a new SolidWorks major version, the client installation can do one of the following:

- Upgrade the latest existing major version, if one exists
- Create a new major version (click **Browse** to change the new installation directory)

If you specify that the client upgrades the latest existing major version, the older version will no longer be available after the upgrade.

If you specify that the client create a new major version, the existing major version is preserved on the client computer after the installation. You can specify a default location for the new major version installation.

> If this is a Service Pack release of an existing major release, that major release installation is always updated to the Service Pack. (That is, Installation Manager will not create a new installation for a Service Pack if the major release for that Service Pack is already installed.)

Installed products other than SolidWorks are not affected by this selection.

For some Service Pack updates, Installation Manager installs new versions of those products if they were not installed previously.

Applying SolidWorks Settings

You can specify one of the following:

- Allow client installations to use their current SolidWorks settings.

  (If none exist, the installation will use the default SolidWorks settings for that release.)
- You can use settings exported from the Copy Settings Wizard. Click **Browse** to find an existing Copy Settings Wizard settings file.

If SolidWorks is installed on this computer, you can run the Copy Settings Wizard by clicking **Run Copy Settings Wizard Now**.

> You can run the Copy Settings Wizard on other computers that have SolidWorks installed by clicking **Start** > **Programs** > **SolidWorks version** > **SolidWorks Tools** > **Copy Settings Wizard**.

Activating Client Installations

You can specify that client licenses are activated automatically when the client installation completes, or you can leave it to the users to do manually after installation.

> This option is available only if both of the following are true:
> - You entered one or more non-SolidNetWork License serial numbers.
> - One or more products requiring activation were selected for installation.

Download Options

Specify details for completing the download operation, such as the location of the downloaded files, how to download the files, and whether to install the downloaded files.

SolidWorks Installation Manager supports electronic software distribution (ESD). If you are not installing from a SolidWorks DVD, you are prompted to specify the source for the SolidWorks products you are installing.

For answers to questions about the download folder, see SolidWorks Installation Manager Download Folder on page 70.

What do you want to do?

Choose one of the following:

Download and install/Download and create image	After the download completes, Installation Manager runs an installation using the downloaded files.
Download only. You can download once and then share downloaded files with multiple computers.	Installation Manager does not run an installation after the download completes.
	Typically, you would specify this option if you are downloading the files to configure an installation file set to share among multiple computers.
	If you specify this option, you are then asked to choose between two sets of files to download:

- **Files for THIS computer**: The download will contain only those files that are missing from the installation file set currently on this computer.

  This should be a smaller download because it contains:

  - Only files for products selected on the Product Selection page
  - Only the prerequisite files that are required for your computer but not yet installed

  However, if you try to use this downloaded file set on another computer to perform an installation, some files could be missing (because the installation file set on the other computer might be different from this one). You can use these download files to install on another computer, but when the installation runs you may have to download additional files to complete the installation file set on that computer.

- **Files to share with ANY computer**: The download will contain the complete installation file set that can be used to install this version on any computer.

  This will be the larger download because it contains the complete file set that you can use to install this version of any product on any computer.

  If you specified a limited set of products to download, this option ignores that specification and downloads all products contained in the product release.

The download will contain all files required to create an administrative image, not just those files required to perform an individual installation.

Install only/Create image only. I already have all the installation files needed.	Installation Manager does not download any files, but uses the files in the selected directory. Click **Browse** to navigate to a different location. For example, you can browse to an installation DVD or a download folder, provided that location contains all the required files. If you receive an error message that the folder is incomplete, and you were expecting all installation files to be present at that location, possible explanations include: • You selected a component for installation that had not been downloaded previously or it was selected by default. • The files were downloaded on another computer with a different set of system requirements. • You previously performed a client installation and now are creating an administrative image, which requires a larger file set. You can either browse to a folder containing all the files (for example, a DVD) or select one of the download options.

Where do you want to download the required files?

This is the download folder location.

In some cases, this option might be visible but disabled. For example, if **Install only** is specified, the Installation Manager will install any downloaded files from the directory specified for that option, rather than using a separate download folder.

Whether you are downloading these files or accessing them from your own location, all files must be downloaded to the same download folder.

Estimated download size

The estimated download disk space requirements are based on the complete component listing chosen, regardless of whether this is a new download or you are resuming a download from a previous session.

For example, if you are resuming a previous download the Installation Manager processes only those files that have not already been downloaded. The download time could be less than the estimate.

If a file completed downloading in the previous download, its size is not included in this estimate. Therefore, the estimate might be higher than the actual download size.

How do you want to download?

There are two options:

Conduct manual download	In some cases, a proxy server might prohibit automatic downloads from Installation Manager. By selecting this option, you can download the files manually through a Web page and then proceed within Installation Manager

to complete the installation operations. This Web page is customized by Installation Manager to download only the files required to complete the installation.

Speed up downloads by using more network bandwidth	The downloader uses an algorithm to increase the speed of the download by increasing the network bandwidth used during the download.

In some cases, this can disturb network performance. If you have trouble downloading, this option might not be supported by the proxy server. Try clearing the option and attempting the download again. For details, check with your network administrator.

This option applies only to automatic downloads. It has no impact on manual downloads. |

Download Options for Installation Manager Files

You have chosen to download a later version of the SolidWorks Installation Manager. Use this screen to specify where and how it is downloaded.

For answers to questions about the download folder, see SolidWorks Installation Manager Download Folder on page 70.

This screen appears only if there is a problem with the default download folder (for example, if write access or network access is not available). Typically, you can just specify an alternate download directory to proceed.

Download Directory

Browse to the destination for the Installation Manager files.

The size of the download and the amount of space needed is shown. The amount of space needed includes extra space required to unzip certain downloads and is larger than the actual download size.

Use Download Acceleration

If you choose this option, the downloader uses an algorithm to increase the speed of the download.

Download acceleration increases bandwidth and can disturb network performance. Check with your network administrator before enabling download acceleration.

If download acceleration is not supported by the proxy server, this option is automatically disabled.

Download Product Installation Files

Specify where and how to download missing files for the specified product.

The Installation Manager downloads, verifies, and extracts all files in parallel during the download process.

Cancelling out of this screen cancels the rest of the installation sequence.

For the consequences of cancellation and instructions on completing interrupted installations, see Installation Errors.

Download Directory

Browse to the destination for the downloaded files.

The download size and the required space is shown. The required space includes any extra space needed to unzip certain downloads and is larger than the actual download size.

For answers to questions about the download folder, see SolidWorks Installation Manager Download FolderThe SolidWorks Installation Manager can download the installation files it needs to a folder that you specify. This topic provides answers to frequently asked questions about the download folder. .

Download Options

Speed up downloads by using more network bandwidth	The downloader uses an algorithm to increase the speed of the download by increasing the network bandwidth used during the download.

In some cases, this can disturb network performance. If you have trouble downloading, this option might not be supported by the proxy server. Try clearing the option and attempting the download again. For details, check with your network administrator. |

Download acceleration increases bandwidth and can disturb network performance. Check with your network administrator before enabling download acceleration.

If download acceleration is not supported by the proxy server, this option might be automatically disabled.

Download Progress

The Installation Manager downloads, verifies, and extracts all files in parallel during this process.

To cancel the download after it has begun, click **Cancel**.

If you cancel the download or the download fails, a Download Canceled or Download Problem screen indicates which downloads were not completed and lets you resume the download or close the application.

For answers to questions about the download folder, see SolidWorks Installation Manager Download Folder on page 70.

Download Results

Indicates whether the download completed successfully, failed during operation, or was cancelled.

Download Problem/Download Cancelled

If you see Download Problem, Installation Manager encountered problems that prevented the download from completing.

If you see Download Cancelled, you cancelled the download before it could complete.

To proceed:

Retry/Resume automatic downloads	To resume the automatic download from when you cancelled, select this option and click **Retry Download** or **Resume Download**.
Download files individually in a Web browser	In some cases, a proxy server might prohibit automatic downloads from Installation Manager.
	By selecting this option, you can download the files manually through a Web page and then proceed within Installation Manager to complete the installation operations. This Web page is customized by Installation Manager to download only the files required to complete the installation.

Download Complete

The Download Complete screen appears when the download completes if you specified the **Download Only** option.

Click **Finish** to exit Installation Manager.

⚠ Do not attempt to install any of these products manually. Always use Installation Manager to install downloaded files.

Installation Errors/Installation Cancelled

The installation has encountered an error or was cancelled.

If an error or cancellation occurs:

- Products that installed before the error or cancellation remain on the computer.
- The product that was being installed at the time of the cancellation is rolled back, if possible, so that there are no partial installations.
- Remaining products are not installed or uninstalled. A list is provided under **Not Yet Attempted**.

To resume installing or uninstalling these products, click the link provided.

🖉 The installation manager first installs any prerequisites that are not installed on the computer, such as .NET framework. It then installs the selected products in this order:

1. SolidWorks
2. eDrawings
3. SolidWorks Search
4. DWGeditor
5. SolidWorks Explorer/Workgroup PDM
6. PhotoView 360
7. SolidWorks Flow Simulation

8. COSMOSM

Recovering from a Failed or Cancelled Installation

If an error or cancellation occurs during installation, you can recover from that point by re-running the installation.

1. Record the error message and product code for the installation that failed, as shown on the Installation Errors or Installation Cancelled screen.

   If you are prompted for a source file and you are not modifying an existing SolidWorks installation, the installation or computer may be corrupt. Contact your SolidWorks reseller.

2. Reboot the computer.
3. To ensure that no SolidWorks products are running, in Windows, press Ctr+Alt+Del to open the Task Manager.
4. On the Processes tab, if any of the following processes appear under **Image Name**, select them and click **End Process**.
   - SLDWORKS.exe
   - EModelViewer.exe
   - DWGEditor.exe
   - PDMworks.exe
   - pdmwVault.exe
   - SWViewer.exe
   - solidworksexplorer.exe
   - swlmwiz.exe
   - photoview360.exe
   - SWBoengine.exe
   - SWVBAServer.exe
   - Sldshellextserver.exe

5. If the application is already installed and you were trying to modify or remove it, try to repair the installation.
6. Click **Start** > **Control Panel** > **Add or Remove Programs**.
7. Select the version of SolidWorks that failed.

   If there is no entry for the SolidWorks version you are trying to install, repeat the installation process.

8. Do one of the following:
   - To retry, click **Change** and select all products that failed to install or were skipped during the installation.
   - To skip the failed installation, click **Change** and select all products that were skipped during the installation. Clear the product that failed to install.
   - To abandon the installation, click **Remove** and select any products that were installed.

   This will not restore the previous version of any products you are upgrading.

Repairing the Installation

If an error or cancellation occurs during installation, you can use Windows Control Panel to repair the installation.

You can repair an individual installation. You cannot repair an administrative image installation; you must uninstall and then reinstall the administrative image.

To repair the installation:

1. In Windows, click **Start** > **Control Panel** > **Add or Remove Programs**.
2. Select the version of SolidWorks that failed and click **Change**.
   SolidWorks Installation Manager starts.
3. In the welcome screen, click **Repair the individual installation (on this computer).** and click **Next**.
4. Select the products that failed to install or remove and click **Repair**.

Contacting the SolidWorks Reseller

If modifying or repairing the installation fails, contact your local SolidWorks reseller with the information gathered from this screen. You might also be asked to provide a zip file of the Installation Manager log files.

1. Locate the log files:

   - For Windows operating system versions prior to Vista:

     ```
     C:\Documents and Settings\username\Application
     Data\SolidWorks\Installation Logs\installed_version\
     ```

   - For Windows Vista or later:

     ```
     C:\Users\username\AppData\Roaming\SolidWorks\Installation
     Logs\installed_version\
     ```

     If you cannot find the folder, it might be hidden. In Windows Explorer, click **Tools** > **Folder Options**. On the View tab, under **Advanced Settings**, select **Show hidden files and folders** for **Hidden files and folders**.

2. Zip the entire contents of this folder, including sub-folders.

Installation Is Complete

The Installation Manager has completed its processing.

One or more of these options may be displayed.

Updates

The option to check for updates is available for individual installations. To specify frequency of checks, select **Check for Updates every ____ days**, and provide a value.

You must be connected to the Internet when the check is performed.

If an update is found at the scheduled time, you are notified the next time you run SolidWorks.

For administrative image installations, upgrades are managed by administrators.

Service packs are available for subscription customers.

What's New

For individual installations, when you finish the installation process, a document describing new functionality in SolidWorks and its add-in products appears.

For administrative image installations, the document appears on client computers when they complete their installations.

Survey

Give us your opinions about the installation process by answering a short survey.

Customer Experience Program

Performance information is sent to SolidWorks Corporation to help improve the product. The process is transparent.

SolidWorks uses this information to determine trends concerning product usage and quality. Our goal is to deliver the highest quality product and the performance logs are a useful tool in this process.

All information contained within the log files is treated confidentially. It is not used for any other purpose and no one will contact you.

For administrative images, you specify this in the Client Administrative Options screen in the Summary page.

For further information, see the Customer Experience Feedback Program Web site.

Administrative Image Creation Is Complete

When you finish creating an administrative image, the only options that appear are:

Show me how to install this image on a client.	Display instructions on using administrative images to install SolidWorks on multiple client computers.
Start the Option Editor now.	Start the Administrative Image Option Editor for the administrative image that was just created.
	For details about configuring an administrative image using the Option Editor, see Administrative Image Option Editor on page 15.

Checking for Updates to SolidWorks

You can use SolidWorks Installation Manager to check for updates manually or to configure automatic periodic checking for updates.

1. Start the Installation Manager by doing one of the following:

   • In the SolidWorks application, click **Help** > **Check for Updates**.
   • In Windows, click **Start** > **Programs** > **SolidWorks Installation Manager** > **Check for Updates**.

   Installation Manager tells you whether any updates are currently available.

2. If updates are available, choose whether to download and install them.

3. To enable automatic periodic checks for updates, when Installation Manager completes its processing select **Check for Updates every ___ days** and specify the frequency of checks.

Installation Location

Specify SolidWorks installation locations.

For installations except administrative image creation, these options apply:

Install selected products to	This is the directory location to which you are installing SolidWorks.
	✏️ When upgrading or modifying an installation, all files must be installed to the existing location for that installation. In this case, you cannot change the location of the existing SolidWorks installation.
Estimated space required for installation	This is the additional space required for the products selected for installation or modification.
	Verify that the disk to which you are installing has sufficient free space to support the installation requirements.
Install from	This is the directory location from which you are installing SolidWorks.
	✏️ This option appears only if the **Install Only** option is specified. For details, see Download Options on page 77.

⚠️ Consult your system administrator before changing either SolidWorks installation location.

Administrative Image Locations

When creating an administrative image, these options apply:

New image location	This is the directory location in which Installation Manager will create the new image.
	✏️ This location cannot contain an existing administrative image.
Estimated space required for image creation	This is the additional space required for the administrative image.
	Verify that the disk to which you are installing has sufficient free space to support the image requirements.

Create image from	This is the directory location from which you are installing SolidWorks products.
	🖊 This option appears only if the **Install Only** option is specified. For details, see Download Options on page 77.
Existing image location	When upgrading an existing administrative image, Installation Manager creates a new image based on the settings for that existing image. This existing image is not modified. This shows the location of the original image you specified on the Welcome screen.

Avoid Using Version-Specific Directory Paths

If you specify a version number in an installation directory location (for example, `C:\Program Files\SolidWorks Corp\SolidWorks 2009\`), and then upgrade that installation to SolidWorks 2010, you will have a SolidWorks 2010 installation directory that is labeled `SolidWorks 2009`.

Installation Options

One or more older major versions of SolidWorks are currently installed on this computer. Choose whether to upgrade the latest existing version to the new version or to create a new installation for the new version.

🖊 This choice appears only if no previous version of this major release is installed on this computer. If a previous version of this major release is installed, SolidWorks Installation Manager always updates that installation.

(Starting in SolidWorks 2010, parallel installations of multiple Service Pack versions of the same major release are no longer supported.)

When upgrading:

- All specified products are upgraded.
- All installed products are selected by default.
- If you deselect a product, that product is not upgraded.

🖊 If you deselect a product component, you are not specifying that Installation Manager remove that component. To remove a product component, use **Add/Remove Programs**, as described under Removing an Installation on page 48.

Select the appropriate option and click **Next** to continue.

Create a new installation of SolidWorks *version*.

Selecting this option preserves the older major version installation in addition to creating a new installation for this major version.

💡 This can be useful if you want to access the older version of SolidWorks even after the newer version is installed. For example, if you are sharing files with other users

who are still running the older version, you can avoid version incompatibility with files saved using the newer version.

Upgrade SolidWorks *previous_major_version* to SolidWorks *current_major_version*.

Selecting this option upgrades the older version installation to the newer version.

🖉 Once the installation completes, the older SolidWorks version will no longer be available on this computer.

Installation Progress

This screen shows the progress of the installation.

To cancel the installation after it has begun, click **Cancel**.

For the consequences of cancellation and instructions on completing interrupted installations, see Installation Errors/Installation Cancelled on page 81.

Installation Type

Specify the installation operation you would like to run, such as creating a new installation, modifying or repairing an existing installation, creating an administrative image, installing administrative server products, or downloading files without installing.

Individual

Install SolidWorks products on this computer.

🖉 This option appears only if this version is not currently installed on this computer.

Modify the individual installation

Modify the existing SolidWorks installation on this computer.

🖉 This option appears only if this version is currently installed on this computer.

Modifications include:

- Adding or removing SolidWorks product components
- Changing to a different SolidWorks package (for example, SolidWorks Professional, SolidWorks Premium, and so on)
- Changing the SolidWorks serial number setting

  🖉 If you already specified a SolidWorks serial number during a prior installation, the Installation Manager might not prompt for it again. To change the serial number that is assumed by the Installation Manager, you must select this option.

Repair the individual installation

Repair the existing SolidWorks installation on this computer.

🖊 This option appears only if this version is currently installed on this computer.

Repair verifies that all the files are present and registered properly and fixes any problems that are found. Data you have created is not affected.

- To repair a product, ensure that it is selected.
- Clear the check boxes for products you do not want to repair.

🖊 You can repair an individual installation. You cannot repair an administrative image installation; you must reinstall using the administrative image.

Administrative image and server products

Create or update an administrative image or install administrative server products on this computer.

You can use an administrative image to install on multiple computers. For details about installing SolidWorks on multiple clients using administrative images, see Using Administrative Images on page 14.

When creating or updating administrative images:

- You can use installation settings from an existing image to configure the new or updated image, or you can use the default SolidWorks installations settings.
- If you are updating an existing image:
  - Installation Manager creates a new copy of the image, even if you are updating. The original image is not modified.
  - You can create the updated image using files from the existing image you are updating. This results in a smaller download.

  🖊 If you do not specify an image for which a Service Pack release is available to update the previous image, you cannot take advantage of the smaller download.

SolidWorks administrative server products include SolidWorks Workgroup PDM Server (Vault) and SolidNetWork License Manager. For details about these products, see Workgroup PDM on page 49 and Administering Licenses Using a SolidNetWork License Manager on page 56.

Download only

Download the installation files for selected SolidWorks products without installing the products.

🖊 This option does not appear if you are installing from a DVD.

The content of the downloaded files is the same as the files that are available on the SolidWorks installation DVD. Selecting this option downloads the full set of installation files, regardless of what files might already be installed on your computer.

You can use this method to set up a server from which your users can perform installations. This is different from creating an administrative image.

Installing and Managing Multiple Computers

The SolidWorks Installation Manager provides the following tools to install and manage multiple SolidWorks computers.

SolidWorks Administrative Images

If you are installing SolidWorks on multiple computers, you can create an administrative image and then deploy the application to other computers.

Workgroup PDM Server

The Workgroup PDM application is project data management software that runs inside the SolidWorks environment or as a standalone application inside SolidWorks Explorer. Workgroup PDM controls projects with procedures for check out, check in, revision control, and other administration tasks. Installing the Workgroup PDM Server creates a Workgroup PDM vault, which is the database that stores the managed SolidWorks projects.

SolidNetWork License Manager

The SolidNetWork License Manager allows the number of users to exceed the number of licenses by floating the use of licenses. When a client license is not in use, it is available for another SolidWorks user.

You can also use the SolidNetWork License Manager to distribute licenses for SolidWorks core add-in products (for example, FeatureWorks).

Manual Download

You have chosen to download files manually (or you might have chosen to download manually the last time you ran SolidWorks Installation Manager).

Click the link to open an automatically generated Web page that contains a complete list of the files you need to complete your download.

⚠ You must download to the folder specified on the Summary page. Installation cannot begin until all the files are downloaded into that folder.

Once you have completed the manual downloads, click **Next** to continue.

If you want to download automatically, you can change the option by going back to the Summary Page and changing Download Options.

Newer Version Available

The Installation Manager has detected a newer version of SolidWorks products. You can choose to install the version contained in the original installation DVD or download, or you can choose to download and install the newer version.

If You Are Installing from a DVD

Under **Would you like to use the newer version?**, choose one of the following:

- **No, continue installing *version* from this DVD. You can update afterwards with a smaller download.**

  Select this option to continue with the installation using the current Installation Manager.

After the installation completes, you can check for and install updates using the Installation Manager. To check for updates, run **Start** > **SolidWorks Installation Manager** > **Check for Updates**.

The advantage is that this results in a smaller download. The disadvantages are that this is a two-step process and the files that you download are version specific (and cannot be shared by other users using different versions).

- **Yes, perform installation using this DVD, then download and update to *version***.

  Select this option to download and run a later version of the Installation Manager to install SolidWorks products for that version.

  If a small Service Pack release is available to update the DVD version, Installation Manager first will install the original version from the DVD and then apply the Service Pack update. This results in a smaller download.

  You can achieve the same result by selecting the option to continue installing from the DVD and then applying the update at a later date.

  The advantage of this option is that it is a one-step process.

💡 In most cases, installing the full version from the DVD and then using the Installation Manager to check for and install updates is preferred because it requires smaller downloads.

If You Are Installing from a Download

Choose one of the following:

- **Continue installing *version*.**

  Select this option to continue installing the version contained in the initial download.

- **Download and install a new version: *version***

  Select this option to download the new version and install that. If there are multiple new versions available, you can select the version to download and install.

No New Versions Available

There are no newer versions available for download. Proceed with installing the specified version.

To proceed with the installation, click **Next**.

Processes Detected

Installation Manager has detected processes on this computer that might prevent a successful installation.

- To continue the installation do one of the following:
  - Close these processes (for example, using Microsoft Windows Task Manager) and then press **Retry**.
  - Click **Ignore** to proceed without closing the processes.
- To stop the installation, press **Cancel**.

Product Selection

Specify the product components to install, modify, download, or remove.

In the product listing:

- The product listing is the complete listing of products that are available in the package specified above the product listing.
- Products that contain additional components are annotated with **+**. To view all components, click **+** to expand the listing for that component.
- To specify that a product component is to be processed, select the checkbox to the left of the component.
- If you click on a product component, the status bar underneath the product listing lists the action currently specified for that component.
- If you are performing a Modify installation, an asterisk (*****) appears to the left of components when you make any change in the product selection.

If you are applying a Service Pack, features of SolidWorks such as PhotoWorks and FeatureWorks appearing under SolidWorks cannot be selected. This is because you cannot add or remove a feature when performing this type of upgrade.

The actions performed by the Installation Manager are controlled by a set of product coexistence rules.

    Installation Manager selects these product components by default:

- If you are performing an upgrade or a Service Pack update, Installation Manager selects the products currently installed.
- Otherwise, Installation Manager selects all products to which you are entitled.

To reduce your installation or download time and disk space requirements, expand the product components and clear the check box for those components you do not need.

If the Installation Manager was unable to verify the products to which you are entitled, you can click **Select different packages or products** to change the product package (such as SolidWorks Professional or SolidWorks Premium) listing to install or modify. The Installation Manager lists all SolidWorks products on your installation media that are compatible with the package you select. You can select any products you purchased or are entitled to evaluate.

SolidWorks Search

SolidWorks Search provides a powerful full text search of SolidWorks documents, the Workgroup PDM vault, and 3D ContentCentral. Search displays a preview and the file location and name.

After the installation, SolidWorks Search indexes SolidWorks documents for faster searches. Only SolidWorks files are indexed by default.

    Installing SolidWorks Search automatically installs Windows Desktop Search if it is not already installed on your computer.

Language Support

Expand the SolidWorks product and the **Languages** component to see the languages you can install.

Space Needed

The Installation Manager reports either total installation size or total download size at the bottom of the page.

- If you selected the **Download Only** option, **download size** appears. Download size is the total amount of disk space required for all of the downloaded files (not including extra space required to unzip the files after the download completes). This figure is an estimate.

  All files needed for installation are included in the total download size. If some files already were downloaded into the current download folder, they will not be downloaded again. As a result, your actual download size might be smaller than the total download size shown, or you might not need to download any files.

- Otherwise, **installation size** appears. Installation size is the change in disk space usage that will occur when you complete this installation. Due to file compression, these figures are estimates.

  If you are upgrading or removing an existing installation, the installation size could be zero.

  The figure shown does not include any space needed to download or extract files if required for the installation.

  If downloads are required and you are downloading to the same drive, the actual disk space requirements could be significantly greater than the installation size that appears.

  To reduce download or installation time for future modifications, do not delete the downloaded files after installation. Subsequent updates and patches typically require access to the last full installation file set.

Products to Download

Select SolidWorks products to download to a specified location.

The products available for download are determined by the serial number you supplied:

- The action to be performed for each component appears to the right of each item.
- Products that contain additional components are marked with a **+**. To view all components, expand those products.
- Click the icon to the left of each component to select how it is handled by the Installation Manager.

The Installation Manager downloads all product components to which you are entitled, unless you specify otherwise. To reduce your download time and disk space requirements, expand the product components and specify **Do not download** for those components you do not need.

If the Installation Manager was unable to verify the products to which you are entitled, you can select a SolidWorks package to download. The Installation Manager lists all SolidWorks products that are compatible with the package you select.

If the Installation Manager was unable to verify the products to which you are entitled, you can click **Select different packages or products** to change the product package

(such as SolidWorks Professional or SolidWorks Premium) listing to install or modify. The Installation Manager lists all SolidWorks products on your installation media that are compatible with the package you select. You can select any products you purchased or are entitled to evaluate.

Language Support

Expand the SolidWorks product and the **Languages** component to see the languages you can install.

Space Needed

Based on the products you select, the space needed to perform the download is shown. Download size is the total amount of disk space required for all of the downloaded files (not including extra space required to unzip the files after the download completes). This figure is an estimate.

To reduce download or installation time for future updates and patches, do not delete the downloaded files after installation. Subsequent updates and patches typically require access to the last full installation file set.

Products to Repair

When you specify **Repair this installation**, this screen displays the SolidWorks products, with version numbers, that can be repaired by this Installation Manager.

Repair verifies that all the files are present and registered properly and fixes any problems that are found. Data you have created is not affected.

- To repair a product, ensure that it is selected.
- Clear the check boxes for products you do not want to repair.

Click **Repair** to begin repairing the selected products.

You can repair an individual installation. You cannot repair an administrative image installation; you must reinstall the administrative image.

Serial Number

Serial numbers, found on the distribution box, associate your computer with the SolidWorks products you have purchased.

If SolidWorks products are already installed on your computer, their serial numbers are displayed by default.

- Enter your SolidWorks serial number if it is not already displayed.
- If you have serial numbers for products that are not included with your SolidWorks license, click the product entry underneath **Do You Have Other Serial Numbers?** and then enter the serial number for that product.

If you have an internet connection, the serial numbers you provide will determine the products to which you are entitled as you continue through the installation process.

Because there are multiple ways to purchase SolidWorks products, some serial numbers can cover two or more products. For example, a SolidWorks serial number

might enable you to install SolidWorks Simulation without entering a separate SolidWorks Simulation serial number. In addition, some products do not require a serial number. If you have any questions about your serial numbers, contact your VAR.

Server Installation

The server products you are attempting to install are listed, with the results of the installations.

Server Installation Status

Server product installations are performed by an external installation application. If you are not currently in that application, you must switch to it to complete the installation.

Click **Cancel** to stop the server product installations.

Server Installation Complete

The external installation application has completed successfully.

Click **Finish** or **Next**.

- If you are installing other administrative products (for example, a SolidWorks administrative image), that installation continues.
- If the server product installations were the only installation operation, Installation Manager closes.

Server Installation Cancelled or Failed

If a server product installation failed or was cancelled after it was begun, servers that were not installed are listed.

Click **Close** or **Next**.

- If you specified other administrative product installations, Installation Manager will resume.
- If the server product installations were the only installation operation, Installation Manager closes.

Summary

The SolidWorks Installation Manager has all of the information needed to complete the installation operation.

To continue:	Click the *action* **Now** button to initiate the installation operation (for example, **Install Now**, **Create Now**, **Download Now**, **Modify Now**).
To change installation settings:	Click **CHANGE** 🖊 for the section you want to change (for example, **Products**, **Installation Location**, **Toolbox Options**).
To cancel:	Click **Cancel** ❌ to exit the Installation Manager without changing installation components.
To revert to a previous page:	Click **Back** 🔵 to return to a previous page in the Installation Manager.

💡 For information about a particular set of installation options (for example, **Products**, **Installation Location**, **Toolbox Options**, and so on):

1. Click **CHANGE** 🖊 next to the section for which you want information.

2. In the detail page that appears, click **Help** ❓ to get information about those installation option settings.

System Check Warnings

Installation manager system check reported issues that could impact how you continue with the installation. In most cases, you should be able to proceed with the Installation Manager.

Toolbox Options

If SolidWorks Toolbox is installed, SolidWorks includes a standard library of hole types and hardware.

For details about configuring, extending, and using SolidWorks Toolbox, click **Help** > **Toolbox Help** in SolidWorks.

🖊 To view this help, you might have to start the Toolbox add-in, using **Tools** > **Add-Ins**.

Before using a SolidWorks Toolbox, make sure it is properly configured according to your company or group policy. In many cases, a group will configure and maintain a central Toolbox. Check with the SolidWorks administrator for guidance on how to specify a Toolbox during SolidWorks installation.

You can change the selected SolidWorks Toolbox after installation. In SolidWorks, click **Tools** > **Options**. On the System Options tab, click **Hole Wizard/Toolbox**.

The following sections describe all options you can encounter on this screen for all configurations. The actual Toolbox options you see depends on the pre-existing SolidWorks and Toolbox configurations on this computer and the Installation Manager choices you have specified to this point.

Create a new *new-version* Toolbox

Create a new version of SolidWorks Toolbox on this computer.

Any existing Toolboxes on this computer are not modified.

This option appears in all installation scenarios except creating an administrative image.

Reference or upgrade an existing Toolbox

Use an existing Toolbox. Click **Browse** to specify the location of the Toolbox you want to use with this installation.

- If the Toolbox is configured for a previous SolidWorks major release version, it is upgraded to this version. Any customizations made to that Toolbox are preserved in the upgraded Toolbox.

  ⚠️ Upgrading a Toolbox to a new SolidWorks major release version makes that Toolbox incompatible with computers running prior major release SolidWorks versions.

- If the Toolbox is configured for this SolidWorks version (as might be the case if you specify a shared network Toolbox that has already been upgraded), no upgrade to that Toolbox occurs.
- If you are upgrading a Toolbox in an Enterprise PDM environment, additional procedures are required. For details, see Upgrading Toolbox in a SolidWorks Enterprise PDM Environment on page 98.

This option appears in all installation scenarios except creating an administrative image.

Upgrade a *previous-version* Toolbox

Copy a Toolbox configured for a previous major release version to a new location and upgrade that copy to this version.

This option appears if a previous SolidWorks major release version is installed on this computer, but you have chosen not to upgrade that installation to this release.

  🖉 The purpose of this option is to upgrade a Toolbox for a previous major release SolidWorks version that is installed without impacting users that are using the older version of Toolbox.

If you are upgrading a Toolbox in an Enterprise PDM environment, additional procedures are required. For details, see Upgrading Toolbox in a SolidWorks Enterprise PDM Environment on page 98.

Upgrade this installed *previous-version* Toolbox

Upgrade this Toolbox to the newer version.

⚠️ Upgrading a Toolbox to a new SolidWorks major release version makes that Toolbox incompatible with computers running prior major release SolidWorks versions.

This option appears if you are upgrading a previous SolidWorks major release version to this version and a Toolbox configured for that release is found on this computer.

If you are upgrading a Toolbox in an Enterprise PDM environment, additional procedures are required. For details, see Upgrading Toolbox in a SolidWorks Enterprise PDM Environment on page 98.

Use the installed *current-version* Toolbox

Installation Manager has detected on this computer a Toolbox configured for this SolidWorks major release version.

If you select this option, SolidWorks will use this Toolbox. No upgrade occurs.

This option appears if a previous SolidWorks major release version is installed on this computer, but a Toolbox is configured for this version. This might occur if a shared Toolbox is created on a computer that is still running an older SolidWorks version.

Specify the Toolbox location for the client computer

Specify a network or directory location as the default Toolbox location when clients install from an administrative image.

> If the location contains a Toolbox configured for a previous SolidWorks major release version at the time the client installation occurs, Installation Manager upgrades that Toolbox.

This option appears when you are creating a new administrative image.

Upgrade or use the Toolbox referenced in the existing image for the client computer

Use the Toolbox location specified in an existing administrative image you are updating.

> If the location contains a Toolbox configured for a previous SolidWorks major release version at the time the client installation occurs, Installation Manager upgrades that Toolbox.

Select a different Toolbox location for the client computer

Specify a different Toolbox location from that specified in an existing administrative image you are updating.

> If the location contains a Toolbox configured for a previous SolidWorks major release version at the time the client installation occurs, Installation Manager upgrades that Toolbox.

Notes About Configuring SolidWorks Toolbox

SolidWorks Toolbox is a powerful and complex feature. This topic summarizes some concepts to consider before configuring a Toolbox using SolidWorks Installation Manager.

Finding More Information About SolidWorks Toolbox

For details about configuring, extending, and using SolidWorks Toolbox, click **Help** > **Toolbox Help** in SolidWorks.

> To view this help, you might have to start the Toolbox add-in, using **Tools** > **Add-Ins**.

Using a Shared Toolbox

When you install Toolbox, you can have Toolbox data on the local computer or in a shared network location. Storing Toolbox data in a network location is recommended. By using a common location, all SolidWorks users share a consistent set of fastener information.

If this is the first time installing, browse to the new shared location where you would like to install the Toolbox.

Upgrading an Existing Toolbox

Upgrading a Toolbox to a new SolidWorks major release version makes that Toolbox incompatible with computers running prior major release SolidWorks versions.

If other computers remain at that prior major release and continue to require a Toolbox configured for that release, you should make a copy of that Toolbox and upgrade that copy. The new SolidWorks installation will upgrade the copy of the Toolbox and the previous SolidWorks installations will continue using the old Toolbox.

Configuring Toolbox Access

After installation, when you access Toolbox components SolidWorks checks how you have configured Toolbox and makes suggestions to improve access and performance.

For accessing the shared network Toolbox (recommended), use a UNC path to the server on which the shared Toolbox was installed.

An administrator can create a password for Toolbox and set permissions and preferences for the workgroup. Restricting access to configuration and setting common preferences ensures consistent Toolbox data.

Upgrading Toolbox in a SolidWorks Enterprise PDM Environment

Before you upgrade the first SolidWorks installation that accesses the Toolbox database in an Enterprise PDM vault:

1. In Windows Explorer, navigate to the Toolbox folder in the Enterprise PDM vault.
2. Right-click the Toolbox folder and click **Get Latest Version**.

   This enables the Installation Manager to verify whether parts need to be updated.

3. Check out the `SWBrowser.mdb` Toolbox database file from:

   `Vault\Toolbox_folder\lang\Language`

   The database file must be checked out so SolidWorks Installation Manager can update the Toolbox.

4. After upgrading, check in the Toolbox folder to check in the database file and any new Toolbox parts.

The Toolbox library in the vault has now been upgraded.

For subsequent SolidWorks upgrades, before initiating the upgrade, Enterprise PDM clients must get the latest version of the vault Toolbox folder (using **Get Latest Version**). This enables Installation Manager to verify that the files are up to date.

For details about Toolbox configuration in SolidWorks Enterprise PDM, see *Configuring Toolbox* in the *SolidWorks Enterprise PDM Administration Tool* help.

UNC Format

Uniform (or Universal) Naming Convention format specifies a file name syntax format to describe a shared file location.

\\server\volume\path

For example:

`\\Disk_Server\SolidWorks\AdminImage`

Uninstall Products

When you select **Remove** from the Add or Remove Programs list, this screen displays the SolidWorks products (with version numbers) that can be uninstalled by this Installation Manager.

Select the SolidWorks products to uninstall. By default, all SolidWorks products for the selected release are specified for uninstall.

- To uninstall a product, ensure that it is selected.

  If you uninstall all products, the Installation Manager is also uninstalled.

- To retain a product, clear its check box.

  If you retain any product, the Installation Manager is not removed from the Control Panel Add or Remove Programs list.

To remove the selected products, click **Remove Items** .

⚠ When you click **Remove Items**, Installation Manager starts removing products immediately.

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