SOLIDWORKS

3DEXPERIENCE®

INSTALLATION AND ADMINISTRATION

SOLIDWORKS 2026



DASSAULT SYSTEMES

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1

SOLIDWORKS Installation and Administration

This chapter includes the following topics:

- **Individual Installation or Administrative Image**
- **Check System and Software Requirements**
- **Download Installation Media**
- **Installing 3DEXPERIENCE Apps**

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 Machine activation or network licensing.	**Licensing** Machine 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 Modifying, repairing, and removing installations, upgrading to new releases.	**Administration** Upgrading administrative images and clients, and network licensing administration.

SOLIDWORKS Installation Manager enables you to configure settings for SOLIDWORKS product components, including SOLIDWORKS Design, SOLIDWORKS Toolbox, SOLIDWORKS Electrical, SOLIDWORKS Manage, SOLIDWORKS PDM, SOLIDWORKS Visualize, and SOLIDWORKS 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.

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 computers must have network or email access. A network or email connection is required to activate SOLIDWORKS Design and download files. Network access is also required when collecting licenses from a network license server.	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.

Check System and Software Requirements

Before installation, verify that your system meets hardware, operating system, and Microsoft products requirements. Add-ins, such as SOLIDWORKS Electrical, SOLIDWORKS Visualize, and eDrawings have additional requirements.

To ensure successful installation and optimum performance, see **System Requirements**.

Download Installation Media

All your installation media is available from the SOLIDWORKS Downloads Page.

If you cannot access the SOLIDWORKS Downloads Page, contact your Value Added Reseller to obtain DVDs.

To download the installation media:

1. Log in to the **SOLIDWORKS Downloads Page** as a Subscription Service Customer, using your email address and password, or your SOLIDWORKS Design serial number.
2. In the Downloads Page, for **Select Version**, select the *year*.
3. Select the SOLIDWORKS tab for **SOLIDWORKS products**.
4. In the Product area, select a service pack.
   For the latest major release, select SOLIDWORKS Products *version* 0.0.
5. Click **Accept Agreement & Continue** to accept the terms of the license agreement.
6. Click **Download**.
   Your web browser downloads `SolidworksSetup.exe` to the default location `C:\Users\`*`Username`*`\Downloads`.
7. Unzip `SolidworksSetup.exe` to the `Downloads` folder.
   You can also unzip `SolidworksSetup.exe` to your preferred location.
8. Run `setup.exe` with administrative permissions.
9. In Installation Manager, click **Download and share all files**.
   Create individual installs or administrative images on multiple machines with a single download and then follow the prompts to complete the process.
   The Installation Manager downloads the installation media for you.
   This option is useful for:
   - Installing the software on a small number of machines
   - Downloading using a portable drive or a network drive
   - Installing service packs

   To download the installation files selectively, follow the prompts to select the manual download method.

Installing 3DEXPERIENCE Apps

If you manage data using the **3D**EXPERIENCE platform, you can purchase the **Collaborative Designer for SOLIDWORKS** role to connect your Desktop environment to the platform.

This role provides access to **3D**EXPERIENCE apps that enable some SOLIDWORKS products to save, search, and manage data stored on the platform. You can also use SOLIDWORKS Design installed from the **3D**EXPERIENCE platform.. For installation details, see **Installation**.

You install and license these apps from the **3D**EXPERIENCE platform, not the SOLIDWORKS Installation Manager. For installation instructions, in the **SOLIDWORKS Design Help**, see **Working with the 3DEXPERIENCE Platform and 3DEXPERIENCE Apps** > **Installing the 3DEXPERIENCE Apps**.

2

Individual Installations

This chapter includes the following topics:

- **Installation on Individual Computers**
- **Administration of Individual Installations**

Installation on Individual Computers

You can install SOLIDWORKS Design 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:

- If you are installing **a standalone seat of SOLIDWORKS Design**, have your SOLIDWORKS Design serial numbers available.
- Verify that you have a working internet connection. The connection is required for checking entitlement.
- Create a **3D**EXPERIENCE ID. For details on **3D**EXPERIENCE ID creation, see **https://www.3ds.com/support**.
- Verify that you are the designated admin for your account or that your admin has listed you as a contact.

SOLIDWORKS Installation Manager installs various Microsoft components, including Microsoft Visual Studio Tools for Applications, Visual Basic, and .NET Framework. If the correct versions of all Windows prerequisite components are not installed on the computer, SOLIDWORKS Installation Manager installs them automatically before installing SOLIDWORKS products.

The SOLIDWORKS Chromium Embedded Framework (CEF) is another component installed by the SOLIDWORKS Installation Manager. Its purpose is to show content for the SOLIDWORKS Apps and the **3D**EXPERIENCE platform.

The SOLIDWORKS Login Manager is also installed by the SOLIDWORKS Installation Manager. It allows login to **3D**EXPERIENCE Marketplace and **3D**EXPERIENCE Apps.

Installation Method	Summary of the Process
Downloading from the SOLIDWORKS Downloads Page	All your installation media is available from the SOLIDWORKS Downloads Page. 1. Browse to the **SOLIDWORKS Downloads Page**. 2. Log in with your **3D**EXPERIENCE ID. 3. In the Downloads page, for **Select Version**, select the *year*. 4. Click the SOLIDWORKS tab for **SOLIDWORKS products**, and under **Product**, select a service pack. 5. Click **Accept Agreement & Continue** to accept the terms of the license agreement. 6. Click **Download**. You must have a working internet connection with login access to the Downloads Page. If you cannot access the Downloads Page, contact your Value Added Reseller to obtain installation media.
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 the SOLIDWORKS installation files before installation can begin. SOLIDWORKS Installation Manager can help you find missing files if you have internet access and write access to the download directory.
Updating an existing SOLIDWORKS Design installation using **Check for Updates**	Do one of the following: • In SOLIDWORKS Design, click ⓘ > **Check for Updates**. • In Windows, click **Start** > **SOLIDWORKS Installation Manager** > **Check for Updates**. You can also click **Start** > **SOLIDWORKS Installation Manager** > **SOLIDWORKS Background Downloader**. In the Windows notification area, right-click **SOLIDWORKS Background Downloader** and click **Check for Updates**. You must have a working internet connection and administrator privileges on the machine.

In the SOLIDWORKS Installation Manager, you can also choose the option to **Download and share all files**, including the prerequisite software. This option is useful if you are installing the software on a few machines and you want to copy the download using a

portable drive or a network drive. It is also helpful for installing service packs because the full version and service pack are contained in one install.

When the installation is complete, start SOLIDWORKS Design and **activate the license**.

To display a list of products that are licensed on your machine, from the SOLIDWORKS Design main menu:

- Click ⓘ > **My Products**
- Click **Home** in the task pane, **SOLIDWORKS Resources** > **My Products**
- Click **Start** > **SOLIDWORKS Tools *Version*** > **My Products**.

The check marks in the dialog box indicate products that you or your organization own based on your activation and network licenses. You can also learn more about SOLIDWORKS products by clicking the link at the top of each product category.

If you have a SOLIDWORKS Design license, to see what licenses are in use in Windows, click **Start** > **SOLIDWORKS Tools <version>** > **SolidNetWork License Manager Client** > **License Usage** tab.

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, provided that the previous installation was completed with a full set of installation files. This option is not available if you upgraded the previous installation with a service pack.

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

1. Make sure that no SOLIDWORKS Design sessions are active.
2. In Windows, select **Settings** > **Apps** > **Installed apps**.
3. Select the SOLIDWORKS Design installation you want to modify and then click **Change**.
4. On the Welcome to the SOLIDWORKS Installation Manager screen, click **Modify your installation**.
5. On the Serial Number screen, verify that the appropriate serial numbers are specified.

6. On the Product Selection screen, select the products to install or remove.

   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 component.
   - To view the projected action for a particular component, click the component name (but not the check box). The intended installation action appears in the information field below the product listing.
   - You can click the check box next to the product component to change the installation action for that component. (For example, if you are installing or removing components, selecting the check box specifies that the component is to be installed or removed and clearing the check box specifies that the component is not to be installed or removed.)
   - When you change a component installation action from the initial specification, an asterisk (*****) appears next to that component in the listing.
   - If a component has optional installation subcomponents in it, a  icon appears next to the component. To see the available subcomponents for a component, click the  icon.
   - If a component has been expanded to show optional subcomponents, a  icon appears next to the component. To collapse the subcomponent listing, click the  icon.

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 installation of SOLIDWORKS Design, you can use the SOLIDWORKS Installation Manager to repair the installation.

To repair an installation, your computer must have all the source files from the previous installation and any applied service packs. These must be in the location where you originally installed them.

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

1. Open the Windows **Control Panel** and click **Programs** > **Programs and Features**.
2. Right-click the SOLIDWORKS Design version that you want to repair and click **Change**.
3. On the Welcome to SOLIDWORKS Installation Manager screen, click **Repair your installation** and click **Next**.
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 that no SOLIDWORKS Design sessions are active.
2. If you want to move the SOLIDWORKS Design license to another computer, deactivate the license before removing the SOLIDWORKS Design installation on this computer.

   For more information, see **Moving a License** on page 79.

3. Open the Windows **Control Panel** and click **Programs and Features**.
4. Select the product component to remove and click **Uninstall**.

   In some cases, you might find multiple SOLIDWORKS Design Service Pack components for a given major release version. To remove a SOLIDWORKS Design major release, remove all Service Pack elements for that major release.

5. On the Uninstall screen:
   a) Select the products to remove.

      By default, all product components are selected.

   b) Clear the products that you do not want to remove.
   c) Click **Change** in **Advanced Options**.

      The Advanced Options screen appears and the Program Files and Folders option is selected.

   d) Select one or more options to perform a custom uninstall, or select all options to perform a complete uninstall.

      By default, the SOLIDWORKS Installation Manager performs a complete uninstall and removes all the files and folders that were added when the installation took place.

   e) Click **Back to Summary**.
   f) On the Summary screen, expand **Advanced Options** to display the uninstall method and entries.
   g) Click **Remove Items**.

Upgrading to a New Release

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

You can upgrade to a new release when you:

- Receive new installation media
- Are notified of upgrades when starting SOLIDWORKS Design
- Check for updates manually

- Start a new instance of SOLIDWORKS 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

If you have administrator privileges on the machine, you can check for updates for an installed SOLIDWORKS Design release using one of these methods.

- In SOLIDWORKS Design, click ⓘ > **Check for Updates**.
- In Windows, click **Start** > **SOLIDWORKS Installation Manager** > **Check for Updates**.

> You can also click **Start** > **SOLIDWORKS Installation Manager** > **SOLIDWORKS Background Downloader**. In the Windows notification area, right-click **SOLIDWORKS Background Downloader**, and click **Check for Updates**.

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, SOLIDWORKS Installation Manager simulates the upgrade by first installing the new version, then removing the old one. At this time, SOLIDWORKS Design supports major upgrades, while the other SOLIDWORKS products do not.

The consequences of this are:

- When upgrading SOLIDWORKS Design between major versions:
  - Your existing installed location does not change.
  - The upgrade can be rolled back if an error occurs or 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 SOLIDWORKS Design are allowed, except for the products mentioned below.	SOLIDWORKS 2025 SP3.0 and SOLIDWORKS Design 2026 SP0.0 can coexist on the same computer.

Rule	Examples
Multiple service packs from the same major release version are not allowed.	SOLIDWORKS Design 2026 SP0.0 and 2026 SP1.0 cannot coexist on the same computer. If you install SOLIDWORKS Design 2026 SP1.0 on a computer where SOLIDWORKS Design 2026 SP0.0 is already installed, SOLIDWORKS Installation Manager replaces SP0.0 with SP1.0.
Only one version of the following products is allowed on the computer: • SolidNetWork License Manager	If SolidNetWork License Manager 2025 SP3.0 is installed, SolidNetWork License Manager upgrades to 2026 SP0.0, even if an earlier major release of SOLIDWORKS Design remains on the computer.
Only one version of the following product is allowed on the computer: • SOLIDWORKS Electrical	If you create a new installation of SOLIDWORKS Electrical 2026 SP1.0, the SOLIDWORKS Installation Manager automatically removes earlier versions of SOLIDWORKS Electrical from the machine. Back up your SOLIDWORKS Electrical environment before upgrading.
Only one version of the following products is allowed on the computer: • SOLIDWORKS PDM Client • SOLIDWORKS PDM Server	If you install SOLIDWORKS Design 2026 SP0.0 or SP1.0, you should also upgrade to SOLIDWORKS 2026 PDM Server and PDM Client. If you install SOLIDWORKS 2026 PDM Server, you should use the PDM Server installation process to install the SOLIDWORKS PDM Client. Do not install the SOLIDWORKS PDM Client separately with the client installation process. SOLIDWORKS PDM Client must be the same release version and service pack as SOLIDWORKS PDM Server on the same machine. See **SOLIDWORKS PDM and SOLIDWORKS Manage** on page 59 for more information.
Only one version of an included product can be installed for any major release version of SOLIDWORKS Design. However, versions of included products from different major release versions of SOLIDWORKS Design can coexist.	If SOLIDWORKS Design 2026 SP0.0 is upgraded to 2026 SP1.0, SOLIDWORKS Composer 20265 SP0.0 automatically upgrades to 2026 SP1.0. However, SOLIDWORKS Composer 2025 SP4.0 can coexist with SOLIDWORKS Composer 2025 SP0.0 or with SOLIDWORKS Composer 2026 SP1.0.

3

Multiple Installations

This chapter includes the following topics:

- **Deploying to Multiple Clients**
- **Using Administrative Images**
- **Command-Line Deployment**
- **Administration of Multiple Installations**

Deploying to Multiple Clients

You can install SOLIDWORKS Design 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 SOLIDWORKS Installation Manager** on page 19
2. Deploy the administrative image to clients.
   - **Deploying an Administrative Image to Clients Through (Email) Manual/ Pull Method**
   - Deploying an Administrative Image to Clients Through Automatic/ Push Method
   - **Installing from the Administrative Image Using the Command Line** on page 40
3. Implement **licensing**.
   - **Administering Individual Licenses** on page 77
   - **Administering Licenses Using a SolidNetWork License Manager** on page 79

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 17GB 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 an Administrative Image** on page 55.
- When deploying an administrative image, the path to the image must be fewer than 58 characters.
- Clients must have administrative privileges on their computers. If a client has insufficient privileges, a client can run an installation as a different user. This option is useful when an individual user does not have local administrative permissions on the machine, which is required to install SOLIDWORKS products.

  See **Deploying Installations from the Option Editor** on page 28. It also describes an alternate method for installing images, when a client has insufficient privileges.

- If Windows logging is enabled or if you use the status functionality in the Administrative Image Option Editor, you must have read/write permissions to this folder: `{Administrative image directory}\x64\Logs`.

Once you have created an administrative image, the Administrative Image Option Editor can manage installation option settings for users and groups installing from that image. For details, see **Administrative Image Option Editor** on page 20.

Types of Administrative Images

You can use the Administrative Image Option Editor to create several types of images for remote users and for those on your network.

Administrative Image Type	Description
Standard	This image is the default for all new administrative images. It is a standard Windows Installer Administrative image ideal for users who have reliable access to the company network. It stores all the files required to install the selected SOLIDWORKS products in a shared image folder. When users upgrade to a new service pack, they can download and apply patch files and do not need to download the entire image.

Administrative Image Type	Description
Remote Client	This image is designed for remote users who do not have reliable access to the company network. When installing this type of image, remote users inherit all the installation options you specify, such as the serial number and the products to install. Users download the required files over the internet from the **SOLIDWORKS Downloads Page**. Accessing and installing these files from the internet is more reliable than downloading them from the company network and may reduce the load on the networks. Remote users install this image using the same techniques supported by the standard administrative image. You can still script installations using the `sldim.exe` and `startwinstall.exe` files. Installation files download to a path specified in the Administrative Image Option Editor before they install. Users must access the administrative image folder to start the installation and run any batch or settings files hosted there. Remote users must have enough available disk space to store all the files in the image, although users can delete the image after the installation completes.

Administrative Image Type	Description
Compressed	The compressed administrative image is for organizations that need to copy administrative images to multiple sites or client machines and want to minimize the impact on the network. This image is 60% smaller than a standard administrative image, but it installs in the same amount of time. It supports all install operations, such as modify and repair. Compressed administrative images also have short file paths. This avoids problems when images are stored in deep folder structures. When users upgrade to a new service pack, they must download the entire image, not only patches.

Creating an Administrative Image from SOLIDWORKS Installation Manager

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

1. Start SOLIDWORKS Installation Manager.

   See **Download Installation Media** on page 7.

2. On the Welcome screen, select **Create an administrative image to deploy to multiple computers** and select one of the following options:

Option	Description
Create a new image using default settings.	Create the image using the default configuration settings (that is, not based on the settings established for an existing administrative image at your site.)
Create a new image using settings and files from an existing image.	Create the image using configuration settings specified for an existing administrative image at your site. Click **Browse** to specify the installation directory for the existing administrative image.

3. Enter SOLIDWORKS Design **Serial Numbers**.
4. Click **Change** to modify products or default options that include:
   - **Administrative Image Location**
   - **Administrative Image Type**
5. Accept the license agreement and click **Create Now**.

6. Click **Customize Image** to open and display settings in the Administrative Image Option Editor.
7. Optional: Click **Create desktop shortcut** to access the image on your desktop.

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, SOLIDWORKS Installation Manager asks you to start the Option Editor. You can also start the Option Editor by double-clicking `sldAdminOptionEditor.exe` in the administrative image folder.

To specify different installation configurations for groups of machines or individual machines, in the Option Editor, select machines. Then specify the installation options for those machines.

> All dialog boxes that prompt for a file or folder path include an option to map a selected folder to an environment variable. For example, if you select `C:\Program Files\SOLIDWORKS Corp`, the file path is mapped to *%ProgramFiles%\SOLIDWORKS Corp*. The most common variables are *%ProgramFiles%* and *%SystemDrive%*.
>
> You cannot map all paths to environment variables. For example, you cannot map `Z:\SOLIDWORKS Data` to an environment variable.

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 can configure several machines to use a subset of the product components available in the image while another group might use a different subset of 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. You can move settings between groups and edit groups of machines.

You can also divide groups of machines into subgroups. The subgroups inherit settings from the parent group, but you can change individual settings for each subgroup.

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: • Select **By Machine Name** and enter a machine name. A machine name cannot include a domain qualifier. For example, pluto.solidworks.com is invalid. • Select **By IP Address** and enter the IP address of a machine. • Select **By IP Address Range** and enter the starting IP address and the ending IP address of the range. • Click **Add Multiple Machines** to enter the machine names. You can separate machine names by a line break, space, tab, comma, period, colon, or semicolon. You can also copy and paste a list of machine names from a delimited file, such as an `*.xls` or `*.cvs` file into the dialog box. If you add machines by Name, IP Address, and IP Address Range, the installing machine will look for a match in this order: **Machine Name**, **IP Address**, then **IP Address Range**. Installation options will be applied to the installing machine based on the first match found.
To move an existing machine to a group or subgroup:	Drag the machine onto the group or subgroup.
To move a machine out of a group or subgroup:	Drag the machine onto **Global Settings** or onto a different group or subgroup.
To delete a group, subgroup, or machine:	Right-click the group, subgroup, 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**.

Viewing and Editing Option Settings

SOLIDWORKS Administrative Image Option Editor enables you to view and change option settings for groups and machines specified for the image.

- When you first start the Option Editor, option settings are displayed in read-only mode. The Change icon is not shaded.

  Clicking on **Global Settings**, a group, or a machine displays a summary of option settings for that item:

Scope	Option Settings Displayed
Global Settings	All option settings specified for the administrative image
Group	Changes in the group settings relative to the Global Settings
Machine	Changes in the machine settings relative to the group

- To make changes to option settings, click **Change**. The icon becomes shaded and a more detailed option settings listing appears.

  Clicking on **Global Settings**, a group, or a machine lets you change option settings for that item. When you make a change in an option setting that differs from the inherited Global Settings or group settings, that option setting highlights.

  If you click **Change** again, the display reverts to read-only mode.

  If you make changes, the Option Editor prompts you to save or discard them when you exit.

Specifying Option Settings for Groups and Machines

You can specify option settings for all machines (Global Settings), groups of machines, and individual machines.

1. In the Option Editor, click the object to modify (**Global Settings**, a group, or a machine).
2. On the right, click **Change**.

   The icon becomes shaded and a more detailed option settings listing appears.
3. Make option setting changes by clicking option settings on the right for the item selected on the left.

   Changes in option settings that override **Global Settings** or group settings are highlighted.
4. Click **OK** or **Close** to save your changes.

Option	Description
OK	The Option Editor saves your changes and exits.
Close	The Option Editor asks if you want to save your changes and then exits.

Administrative Image Option Settings

The Option Editor enables you to specify option settings for individual clients or groups, including serial numbers, client installation options, which software components to install, and toolbox location.

Serial Numbers

Enter the serial numbers to install products on client machines.

> Because there are multiple ways to purchase SOLIDWORKS products, some serial numbers can cover two or more products. For example, a SOLIDWORKS Design 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 Value Added Reseller (VAR).

If you are managing licensing using SolidNetWork Licensing:

- In the first serial number field, enter the SolidNetWork serial number.
- In the **SNL Server port@server** field, enter the SolidNetWork License Manager server name and port in the form *nnnnn@servername*. Separate multiple entries using semicolons.

For details about SolidNetWork licensing, see **Installing SolidNetWork Licensing Overview** on page 81.

Client Installation Options

How do you want to perform client installations of major versions?	You can specify how client installations handle an existing prior major version installation: • **Upgrade an existing major version (if one exists).** Specify this option to replace the prior major version installation with the new version installation. • **Create a new major version.** Specify this option to keep the prior major version and create a new installation for the new major version. To specify a location for the new version installation, click **Browse**.
Do you want to uninstall the most recent previous version before installing a new major version?	You can specify whether to uninstall the existing prior major version installation: • **Do not uninstall.** Specify this option to retain the prior major version installation. • **Uninstall the most recent version.** • **Uninstall all versions.**

How do you want to apply SOLIDWORKS Design settings?	You can specify a settings file when starting the software. Client installations reference a settings file. Store the settings (`*.sldsettings`) file on a shared UNC path that is visible to all installations. Specify a `*.sldsettings` path before installing SOLIDWORKS Design. Client installations detect the `*.sldsettings` file path during the installation. Secure the `*.sldsettings` file by specifying Windows file permissions to allow write access to designated users only. To create or edit a `*.sldsettings`, click **Launch Settings Administrator**. The SOLIDWORKS Settings Administrator tool lets you define which SOLIDWORKS Design system options to apply and optionally lock for end users. You can also define how and when to apply the settings to individual installations, and you can modify the `*.sldsettings` file at any time by running the Settings Administrator tool and overwriting the existing file. You can use different `*.sldsettings` files for different machines or groups. The path to `*.sldsettings` sets at install time. Once deployed, do not rename or move the `*.sldsettings` file. You must have an existing SOLIDWORKS Design [current major version] installation to run the Settings Administrator tool.
How do you want to activate client installations?	You can specify whether to complete product activation automatically after running the installation, or require that the user perform product activation manually after installation. • **Activate automatically when each client installation completes.** Selecting this option setting specifies that product activation runs automatically. Enter the email address that will be supplied when activation runs on the client computer. • **Do not activate automatically.** Selecting this option setting specifies that no automated product activation occurs. The client user performs product activation after installation.

Administrative Options

Create a diagnostic log during each installation.	A client installation can create a diagnostic log to troubleshoot installation problems. Creating this log can make the client installation run slower. The log file is stored in `Administrative image directory}\x64\Logs\<machine_name>`.

Run the installation as a different user.	You can specify a different user with administrative permissions to run the installation instead of the logged in user. This option is useful when individual users do not have local administrative permissions on their machines, which are required to install SOLIDWORKS products. An alternate method of deployment and installation is available when clients do not have administrative privileges. See **Deploying Installations from the Option Editor** on page 28.
Run a program before or after installation.	You can specify that a custom program runs before or after the client installation.
Do you want clients to participate in the SOLIDWORKS Customer Experience Program?	You can specify that client machines participate in the SOLIDWORKS Customer Experience Program. Performance information is sent to SOLIDWORKS Corporation to help improve the product. The process is transparent and confidential. For details, see **Customer Experience Improvement Program Website**.

Limitations on the Run a Program Before and After Installation Options

These options let you run other installation programs or scripts on the client machines before or after deploying SOLIDWORKS Design, for example, a script that installs Microsoft Office.

There is a limitation on using these options. A failure occurs if you use either option to deploy an installation program or script that displays a user interface, for example, a script that displays dialog boxes. When the automatic deployment of SOLIDWORKS Design runs, it fails to install SOLIDWORKS Design on the target machines.

To use the **Run a program before installation** and **Run a program after installation** options, select only programs that can be installed in silent mode.

Software to Install

You can specify what SOLIDWORKS product components to install.

- Selecting the check box next to the product component specifies that the component installs during the client installation.
- Clearing the check box specifies that the component does not install.
- If a component has optional installation subcomponents in it, a **+** icon appears next to the component. To see the available subcomponents for a component, click the **+** icon.
- If a component has been expanded to show optional subcomponents, a **-** icon appears next to the component. To collapse the subcomponent listing, click the **-** icon.

The **Languages** specification limits the languages that are installed for the SOLIDWORKS product only. Other product installations are not affected by this specification. For example, eDrawings® installs all languages.

Toolbox Options

You can specify which standards to support and the default Toolbox location for client installations.

It is common to have multiple users accessing a shared Toolbox, typically through a network location (for example, a UNC path, a shared folder, or a common network drive specification).

If the location contains an older major version of Toolbox, the client installation upgrades that Toolbox to the new major version. The files are no longer compatible with previous major versions of SOLIDWORKS Design.

If some clients continue to use an older version of SOLIDWORKS Design, you must specify a different Toolbox network location for clients that are upgrading to the newer version.

For details about SOLIDWORKS Toolbox administration, see the *Toolbox* information in *SOLIDWORKS Design Help*.

Installing Toolbox Software Without Installing Toolbox Data

If your environment has a PDM or shared Toolbox/Hole Wizard location, you can specify client installations to not install or upgrade Toolbox data in that location.

Do this if:

- You did not yet specify the SOLIDWORKS PDM local view before installing SOLIDWORKS Design.
- The files in the location are in use and could block SOLIDWORKS Design installations from completing.
- The location is unavailable and could block SOLIDWORKS Design installations from completing.

When you specify client installations to not install Toolbox data, the SOLIDWORKS Design software and settings point to the Toolbox/Hole Wizard location, but you must install the data in a central location later.

To install SOLIDWORKS Design without including Toolbox/Hole Wizard data:

- When asked if you want this client to install/upgrade the Toolbox files, select **No, install the Toolbox software without including the data files**.

At least one person, typically the CAD Administrator, must install or upgrade the Toolbox/Hole Wizard data in the central location or the software does not function properly. Anyone in the organization can handle this task with a single manual installation of SOLIDWORKS Design. If you manage Toolbox inside of SOLIDWORKS PDM, follow the steps outlined in the Toolbox section of the *SOLIDWORKS PDM Administrator Guide* to install or upgrade the Toolbox data.

SOLIDWORKS PDM Client Options

You can select the following options for deploying SOLIDWORKS PDM clients and add-ins:

- **PDM product type**
- **PDM client type**
- **Features**
- **Vault View**

> To open the View Setup Wizard, you must run the Administrative Image Option Editor on a machine that has the SOLIDWORKS PDM client installed.
>
> To create the vault view on the client machine, you need a Conisio View Setup (`.cvs`) file that defines the parameters for configuring the vault view on the client machine. You can reference an existing `.cvs` file or create it from the Administrative Image Option Editor.

See the section on *SOLIDWORKS PDM* in *SOLIDWORKS Design Help* for more information about these clients and add-ins.

Electrical Options

You can specify the following parameters for the administrative components of SOLIDWORKS Electrical:

- **Data path**
- **Collaborative Server machine name and port number**
- **SQL Server machine name and password**

See **SOLIDWORKS Electrical** on page 66 for more information about these components.

Inheritance of Global and Group Option Settings

Option settings for **Global Settings** or groups are inherited by items underneath them.

For example:

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

You can change option settings for a group, subgroup, or machine 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, subgroup, or machine and then click one of the following:
>
> - **Revert to Global Settings** (for a group, subgroup, or machine that is not a group member)
> - **Revert to Group Settings** (for a machine that is a group member)
> - **Revert All Members to Group Settings** (for all the members of that group at once)

Customize Settings for Each Machine

You can also modify settings that you have applied to specific machines by right-clicking the machine name.

From the shortcut menu, you can do the following tasks for each machine:

- **Rename**
- **Delete machine**
- **Revert To Global Settings**
- **Edit Notes**
- **Edit Options**

Edit options let you modify all the options for the machine, such as the client installation options and the administrative options.

Deploying Installations from the Option Editor

You can deploy installations directly from the Option Editor by using either a manual or a push method.

- Use the Deploy Manually page of the Option Editor to email instructions to users on how to start the installation. Use this method when users have administrative privileges for their client machines.

  > Under **Administrative Options**, you can assign administrative permissions to users by selecting **Run the installation as a different user**.

- Use the Deploy Automatically page of the Option Editor to push installations automatically to target machines. Use this method when users do not have administrative privileges to install software on their systems. You can also perform custom installs with this method.

To deploy manually or automatically, you must store the administrative image installation directory to a Windows shared directory, so it is available through a network location, for example `\\machine\shared_directory`. You can also make the image available by copying it to a local directory on each machine, such as `C:\SOLIDWORKS Admin\SOLIDWORKS 2026 SP0`.

Using a UNC Path to Deploy an Administrative Image

Instead of mapping network drive locations, use a UNC path `(\\machine_name\share_name)` when specifying a network location for the administrative image installation directory.

Mapped network drives are not compatible with the push and automatic install method, as well as with the Run As Different User in the pull and manual method.

Using StartSWInstall.exe to Deploy an Administrative Image

You can install or uninstall an administrative image on clients by running the command *admin_image_install_dir*`\StartSWInstall.exe`. This method installs SOLIDWORKS Design using the options, machines and groups defined in the Administrative Image Option Editor.

You can append the following switches to the `StartSWInstall.exe` command:

Switch	Description
`/install`	Installs the administrative image on client machines.
`/uninstall`	Uninstalls the software from client machines, with two optional switches: **`/removedata`** Available for uninstall only. Removes SOLIDWORKS data files and folders during uninstall. **`/removeregistry`** Available for uninstall only. Removes SOLIDWORKS registry entries during uninstall.
`/showui`	Displays a progress window for the SOLIDWORKS Installation Manager. Otherwise the SOLIDWORKS Installation Manager is hidden from the user.
`/now`	Starts the install or uninstall immediately. The 5 minute warning dialog box does not appear.

`StartSWInstall.exe` is preferred to using `StartSWInstall.hta` in cases where you want the installation to proceed without any user interaction, but do not want to use automatic deployment from the Administrative Image Option Editor.

Deploying Manually

You can use this method if users have the privileges required to install on their systems.

To deploy installations manually using email:

1. In the Option Editor, click **Deploy**.

   Installation availability and status for each machine appears.

2. Click **Send Email**.

3. Specify the network path to the administrative image installation directory.

   Typically, the directory path to the administrative image installation directory is shown initially as a Windows file path (for example, `C:\SOLIDWORKS Admin\SOLIDWORKS 2026 SP0`). The dialog box indicates **This path is not a network path**.

   a) Click **Browse** to specify the network location for this administrative image installation directory.

   b) Navigate to the computer where the administrative image is installed, and select the Windows shared directory specified for the administrative image installation directory.

   Do not navigate inside the administrative image shared directory (for example, into the `64bit` subdirectories).

   c) Click **OK**.

   The dialog box indicates **This path is a network path**.

   An email is created to instruct the machine owner how to install from the administrative image using the network location for the `StartSWInstall.hta` file in the administrative image.

4. Send the email to the users responsible for installing SOLIDWORKS Design.

Users can double-click the link to the network location for `StartSWInstall.hta` in the email to install SOLIDWORKS Design from the administrative image.

In the Option Editor, **Installation Status** shows:

Pending	The user has not yet run the installation on this machine.
Successful	The user successfully completed the installation on this machine.
Failed	The user ran the installation on this machine, but it failed.

Other status messages that might display are: Installed Successfully, Uninstalled Successfully, Installing, Uninstalling, Requires Reboot, Install Failed, and Uninstall Failed.

At the top of the **Status** column, click the green arrow to refresh and show updated information.

Prerequisites for Deploying Automatically

Automatic deployment is effective for users without administrative privileges, as it ensures software installation can proceed seamlessly..

To use automatic deployment, both the client and server must belong to the same Active Directory domain. This configuration allows installation tasks to be pushed to target machines, where they are carried out by the Microsoft Task Scheduler without requiring any user actions.

With a push installation, the Task Scheduler runs a task that starts the SOLIDWORKS Installation Manager on a client machine using credentials specified by an administrator in the Options Editor. The Task Scheduler runs under the computer's LocalSystem account. This account is represented on the network as a computer security principle. The computer security principle must have the following access:

- Read access to the Admin Images folder.
- Read and write access to the Admin Images folder\x64\logs folder.

In Active Directory, when you add a computer to a group of a *Shared Folder*, select **Object Types** and **Computers**.

In addition, the computer serving the administrative image and the target client machines must be members of the same Active Directory domain.

Deploying Automatically

To deploy automatically by pushing installations to user systems:

1. In the Option Editor, click the **Deploy Automatically** tab.

   Installation availability and status for each machine appears.

2. Under the **Deploy** column, select the target client machines, and groups in the **Deploy** column.

   > Select one or more machines or groups under the **Deploy** column before starting the deployment process.

3. Under **Install Time**, select **Now** to deploy immediately or **Later** to schedule a specific date and time for the deployment.
4. Under **Client Options**, select the following:
   - **Install or Upgrade SOLIDWORKS Design 2026**
   - **Uninstall SOLIDWORKS Design 2026**

     > If you select to uninstall, you can specify whether to uninstall registry entries and data files and folders along with the program files and folders.

5. Select **Reboot computer automatically** if you want one or more target machines to reboot if required during or after the deployment.
6. Enter the **Client Credentials** of one or more target machines.
7. Click **Deploy** to start the deployment process.

In the Option Editor, **Installation Status** shows:

Pending	The installation has not yet run on this machine.
Successful	The installation successfully completed on this machine.
Failed	The installation on this machine failed.

Other status messages that might display are: Installed Successfully, Uninstalled Successfully, Scheduling, Connecting, Installing, Uninstalling, Requires Reboot, Install Scheduled, Uninstall Scheduled, Install Failed, Uninstall Failed, Connection Failed, and Scheduling Failed.

Testing a Setup for a Push Deployment

The Deploy Automatically tab in the Administrative Image Option Editor contains a test feature.

You can run a mock installation that exercises the systems used for the deployment, such as:

- Connecting to the target computer.
- Creating an install task on that computer.
- Completing the install task on that computer.
- Updating the status and logs on the administrative image.
- Detecting a program to run before or after the installation.

The test verifies file paths, validates permissions, and reports that the installation ended successfully. It does not install any software or modify the registry on the target system. If a system fails on any of the target computers, the test indicates which system failed.

Tooltips provide more information about failures. You can export the results to an `.xml` spreadsheet or `.csv`, which is helpful for reviewing multiple failures or sharing results with other support personnel.

To test a setup for a push deployment:

1. Using the Installation Manager, create an **Administrative image**.
2. Start the Administrative Image Option Editor.

   You can start the option after the Installation Manager completes creating the administrative image or double-click `sldAdminOptionEditor.exe` in the administrative image folder.

3. On the Setup tab, add one or more computers.
4. Click **Deploy Automatically**:
   a) Supply a domain administrator's `username` and `password`.
   b) Select one or more computers to test.
   c) Click **Test**.
5. When the Setup Test appears, click **Run Test**.
6. Allow several minutes for the test to complete or click **Cancel Test** at any time.
7. Review and correct any setup problems.

Exiting the Option Editor

You can click OK or Close to apply changes and exit the Option Editor.

- Click **OK** to apply option setting changes and exit the Option Editor.
- Click the Windows application **Close** button to exit the Option Editor. If the software detects that there are unsaved changes, it prompts you to save.

Creating a Desktop Shortcut to the Option Editor

You can create a Windows desktop shortcut to the Option Editor for a particular Administrative Image from within the application.

1. Start the Option Editor.

   For example, in File Explorer, navigate to the administrative image installation directory and double-click `sldAdminOptionEditor.exe`.

2. Click **Create Desktop Shortcut to this Option Editor**.

   A shortcut to the Option Editor for the administrative image appears on the Windows desktop.

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 path).

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

If you are upgrading an administrative image to a new SOLIDWORKS Design 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**.

Settings Administrator Tool

If you are connected to the **3D**EXPERIENCE platform, you can use the Settings Administrator Tool to specify uniform system options, ensuring a consistent user experience in SOLIDWORKS Design. These settings automatically take effect at user login, overwriting local settings and preventing unintended user changes.

If you are not connected to the **3D**EXPERIENCE platform, you can use the Settings Administrator Tool to specify system options that are applied when deploying or upgrading SOLIDWORKS Design, and to prevent users from changing those options. This tool replaces the SOLIDWORKS Settings Wizard for deploying settings, although that wizard is still available for copying settings.

For users connected to the **3D**EXPERIENCE platform, there are two steps to using the Settings Administrator Tool:

1. Use the Settings Administrator Tool to generate a `*.sldsettings` file, which contains your selected SOLIDWORKS Design options.
2. Upload and deploy the `*.sldsettings` file from the **3D**Experience platform using the **Platform Manager** app.

For users not connected to the **3D**EXPERIENCE platform, there are three steps to using the Settings Administrator Tool with the Administrator Image Option Editor:

1. Create an administrative image to access the tool.
2. Use the Settings Administrator Tool to generate a `*.sldsettings` file, which contains your selected SOLIDWORKS options.
3. Include the `*.sldsettings` file in SOLIDWORKS Design installations.

If you are using the **3D**EXPERIENCE add-in, you can also deploy a `*.sldsettings` file outside of an administrative image. An advantage of using this method is that the settings automatically deploy at login.

1. To access the Settings Administrator Tool, follow the steps given in **Accessing the Settings Administrator Tool** on page 34 .
2. Follow the instructions to apply and lock settings.
3. From the **3D**EXPERIENCE **Compass**, expand **My Roles** and click the **Platform Manager** app. Go to **Content** > **Collaboration Spaces Configuration Center** > **CAD Collaboration** > **SOLIDWORKS** > **Settings** > **Resources** > **CAD Settings**.
4. In **CAD Settings**, browse, upload, and download the `*.sldsettings` file.

Accessing the Settings Administrator Tool

To access the Settings Administrator Tool, you must create an administrative image first.

You can start the Settings Administrator Tool from within an administrative image.

To access the Settings Administrator Tool:

1. Run `sldAdminOptionEditor.exe` in the administrative image folder.
2. In the Administration Image Option Editor, click **Change** .
3. In the **Client Installation Options** section, do the following:
   a) Click **Use a settings file exported from the SOLIDWORKS Settings Administrator**.
   b) Click **Launch Settings Administrator**.

> To access **Launch Settings Administrator**, SOLIDWORKS Design <*version*> must be installed on the computer that runs the `sldAdminOptionEditor.exe`.

Using the Settings Administrator Tool

To use the Settings Administrator Tool:

1. On the **Welcome** tab, select the following options:
   - **Default set of options** to start for this release of SOLIDWORKS.
   - **Import SOLIDWORKS Design 2026 options from this computer**.
   - **Browse to an existing sldreg file**.
   - **Browse to an existing sldsettings file**.

   > The `*.sldreg` file is generated by the SOLIDWORKS Settings Wizard, and the `*.sldsettings` file is generated by the Settings Administrator Tool.

2. Click **Next**.

Applying and Locking Options

In the **System Options** tab, two columns of check boxes let you select which of these options to apply, and which of the applied options to lock so users cannot change them.

When users access the System Options dialog box in SOLIDWORKS Design, the locked options appear unavailable with a lock icon . However, you can create a password that

allows users to unlock options. When users hover over a locked option, a tooltip displays your name and email, so they can contact you and request that password.

To apply and lock options:

1. In the **System Options** tab, do the following:
   a) To apply a system option, select the option and check the corresponding box in the **Apply** column.
   b) To lock a system option, select the option and check the corresponding boxes in *both* the **Apply** and **Lock** columns.

      When users access the **System Options** tab, these options display with a lock .

2. Depending on the option, you might have to specify additional parameters for the option.
3. You can also block users from dismissing messages by applying controls for **Messages/Errors/Warnings**.
   a) Click **Messages/Errors/Warnings**.
   b) Select **Apply** and optionally select **Lock** for the messages you want to manage.
   c) Set the message to **Show** or **Don't Show**.

      If you select **Show**, the message is removed from a user's dismissed message list, and displays to the user at the appropriate time. If you select **Don't Show**, the message is automatically added to a user's dismissed messages list, and does not display when the user is working on a model.

      This table describes the behavior in the software:

Apply	Lock	Show	Don't Show	Behavior
X		X		Force the message to show.
X			X	Dismisses the message and adds the message to the dismissed message list.
X	X	X		Forces the message to show and disables the option to **Don't show again** in the message box.
X	X		X	Dismisses the message and bars user from enabling it in **Tools** > **Options** > **Messages/Errors/Warnings**.

4. In the **System Options** tab, click **Next**.

Finish Setting Options

To finish setting options:

1. On the **Finish** tab, select when system options apply:
   - **the first time a user starts the software**.
   - **every time the software starts**.
   - and whether **customizations** apply at the same time.
2. You can choose to **apply locked options when users are off the network**, and **allow the API to override locked options**.
3. For locked system options, you must provide your **Name** and **Email** address. SOLIDWORKS Design displays this information in a tooltip when users hover over a locked option.
4. You can also **allow users to override locked options**. You must then specify a **Password** that you can supply to users.
5. Click **Save** to generate an `*.sldsettings` file for deployment.

Including the sldsettings File in SOLIDWORKS Design Installations

To include an `*.sldsettings` file in your installation:

1. Double click **sldAdminOptionEditor.exe** in the administrative image folder.
2. In the Administration Image Option Editor, click **Change**.
3. In the **Client Installation Options** section, do the following:
   a) Click **Use a settings file exported from the SOLIDWORKS Settings Administrator**.
   b) Click **Edit Path** and browse to the `*.sldsettings` file.

   You can create and apply different `*.sldsettings` file for different groups or machines.
4. Deploy and install SOLIDWORKS Design.

After deployment, you can repeat step 2 to modify an existing `*.sldsettings` at any time. SOLIDWORKS Design reads the modified `*.sldsettings` the next time it starts.

Command-Line Deployment

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

To deploy an administrative image using the command line for SOLIDWORKS PDM clients, see the *SOLIDWORKS PDM Installation Guide*.

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 using the command line or through Microsoft Active Directory. If you use the SOLIDWORKS 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 Design in the Windows components:

Windows Performance Toolkit	Enables extended logging in SOLIDWORKS Rx.
Visual C++ 2022 Redistributable Package	Contains runtime components for Microsoft Visual C++. The 2022 version must be installed.
Visual Basic for Applications 7.1	Provides support for Visual Basic macros.
Visual Studio Tools for Application 2022	Optional. Contains a toolset supporting Visual Studio 2022 applications (VSTA17).
.NET Framework 4.8.1	Contains the common language runtime and .NET Framework components that are required to run .NET Framework 4.8.1 applications.
SWCEF	Chromium Embedded Framework (CEF) is used to show **3D**EXPERIENCE platform content.
Microsoft Edge Web View2	Required to show web content.
SOLIDWORKS Login Manager	Allows login to the **3D**EXPERIENCE Marketplace and other **3D**EXPERIENCE Apps.

Visual C++ Redistributable Package and .NET Framework 4.8.1 are required for all SOLIDWORKS products, not only the core SOLIDWORKS product.

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

The following tables identify the specific files that are required for each supported operating system.

Operating System: Windows Server 2022 64-bit	
Windows Performance Toolkit:	WPTx64-x86_en-us.msi
Visual C++ 2022 Redistributable Package:	vc_redist.x86.exe vc_redist.x64.exe Both must be installed.
Visual Basic 7.1:	vba71.msi and vba71_1033.msi Plus an optional language pack.
.NET Framework 4.8.1:	ndp481-x86-x64-allos-enu.exe
SWCEF	CEF for SOLIDWORKS Applications.msi
Microsoft Edge Web View2	MicrosoftEdgeWebView2RuntimeInstallerX64.exe

Operating System: Windows 11 64-bit	
Windows Performance Toolkit:	WPTx64-x86_en-us.msi
Visual C++ 2022 Redistributable Package:	vc_redist.x86.exe vc_redist.x64.exe

Operating System: Windows 11 64-bit	
	Both must be installed.
Visual Basic 7.1:	vba71.msi and vba71_1033.msi Plus an optional language pack.
.NET Framework 4.8.1:	.NET 4.8.1 is part of the operating system, which fulfills this requirement.
SWCEF	CEF for SOLIDWORKS Applications.msi
Microsoft Edge Web View2	MicrosoftEdgeWebView2RuntimeInstallerX64.exe

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

You can follow this process to install the 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 the Visual C++ *RELEASE* Redistributable Packages. You must install the 32-bit and the 64-bit versions of each package.

   Install the package for *RELEASE* = 2022.

   administrative_image_directory\Microsoft_C++_*RELEASE*_Redistributable_(x64)\vc_redist.x64.exe /Q

   administrative_image_directory\Microsoft_C++_*RELEASE*_Redistributable\vc_redist.x86.exe /Q

3. Install Microsoft .NET Framework 4.8.1:

   administrative_image_directory\.Net_Framework_4.8.1\ndp481-x86-x64-allos-enu.exe

4. Install the Visual Studio Tools for Applications 2022:

   administrative_image_directory\Microsoft_VSTA17\vsta_setup.exe

5. Install Visual Basic 7.1:
   administrative_image_directory\Microsoft_VBA\vba71.msi and *administrative_image_directory*\Microsoft_VBA\vba71_1033.msi are required for all languages.

   Only the 64-bit version of this package must be installed.

   In addition, install the following package for the desired language:

vba71_1028.msi	Chinese
vba71_1031.msi	German
vba71_1036.msi	French
vba71_1040.msi	Italian
vba71_1041.msi	Japanese
vba71_1042.msi	Korean
vba71_1046.msi	Portuguese-Brazilian
vba71_2052.msi	Chinese-Simplified
vba71_3082.msi	Spanish

   English, Czech, Polish, Russian, and Turkish do not require additional language packages.

6. Install SOLIDWORKS Chromium Embedded Framework (SWCEF):
   administrative_image_directory\swcef\CEF for SOLIDWORKS Applications.msi.
7. Install SOLIDWORKS Login Manager:
   administrative_image_directory\swloginmgr\SOLIDWORKS Login Manager.msi.

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 specify the properties for any options that are not the default values as shown below.

```
msiexec /i
"administrative_image_directory\64bit\SWFileUtilities\SOLIDWORKS File
Utilities.msi" INSTALLDIR="C:\Program Files\your_folder /qb
```

Individual installation component command-line properties include:

- **Command-Line Feature Properties** on page 42
- **Command-Line Global Properties** on page 43
- **eDrawings Command Line Properties** on page 47
- **SOLIDWORKS Flow Simulation Command Line Properties** on page 49

Use the following examples as templates for silent installation.

SOLIDWORKS Client

- SOLIDWORKS Design (English)
- Toolbox add-in
- Participation in the feedback program
- French language

SOLIDWORKS Design:

```
msiexec /i
"administrative_image_directory\64bit\SOLIDWORKS\SOLIDWORKS.Msi"
INSTALLDIR="C:\Program Files\your_folder" SOLIDWORKSSERIALNUMBER="xxxx
xxxx xxxx xxxx xxxx xxxx" ENABLEPERFORMANCE=1 OFFICEOPTION=4
ADDLOCAL=SolidWorks, SolidWorksToolbox TOOLBOXFOLDER="toolbox path for
example C:\SolidWorks Data\" /qb
```

The SOLIDWORKS Design French language installation component must be installed separately:

```
msiexec /i "administrative_image_directory\64bit\SOLIDWORKS
French\french.msi" /qb
```

When specifying SOLIDWORKS Design language component installation commands, do not specify command-line arguments.

eDrawings Client

- All features

```
msiexec /i "administrative_image_directory\64bit\eDrawings\eDrawings.msi"
INSTALLDIR="C:\Program Files\your_folder" SNLSERVER=eDrawings_SNL_server
TRANSFORMS=1040.mst ADDLOCAL=All /qb
```

SOLIDWORKS Electrical

Due to the technical complexity of installing SOLIDWORKS Electrical with Microsoft SQL Server, command line examples for SOLIDWORKS Electrical are excluded from the documentation.

SOLIDWORKS Visualize and Visualize Boost

For SOLIDWORKS Visualize:

```
msiexec. /i "administrative_image_directory\64bit\SOLIDWORKS
Visualize.msi" ACTIVSERIALNUMBER="1234123412341234ABCDEFGH"
INSTALLDIR="C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS Visualize" /qb
```

For SOLIDWORKS Visualize Boost:

```
msiexec /i "administrative_image_directory\64bit\SOLIDWORKS Visualize
Boost.msi" ACTVISERIALNUMBER="1234123412341234ABCDEFGH"
INSTALLDIR="C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS Visualize Boost"
/qb
```

Command-Line Properties

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

Command-Line Feature Properties

SOLIDWORKS Design installation supports these ADDLOCAL properties for command-line deployment.

- CircuitWorks
- CoreSolidWorksTaskScheduler
- ExampleFiles
- FeatureWorks
- HelpFiles
- Manuals
- Motion
- ScanTo3D
- Simulation
- SolidWorksCosting
- SolidWorksDesignChecker
- SolidWorksRoutedsystems
- SolidWorksToolbox
- SolidWorksUtilities
- TolAnalyst

When specifying properties for command-line installation:

- You cannot use the `ADDSOURCE` option.
- Properties are case sensitive, cannot contain spaces or dashes, and must be separated by commas.

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** Do not send performance data.
INSTALLDIR	Installation folder for SOLIDWORKS Design.
MBDSERIALNUMBER	Serial number for the SOLIDWORKS Model Based Definition product.
MOTIONSERIALNUMBER	Serial number for the SOLIDWORKS Motion product.
OFFICEOPTION	Specify product package to install: **0** SOLIDWORKS Design Standard **1** SOLIDWORKS Design Professional **2** SOLIDWORKS Design Premium **3** SOLIDWORKS Design Ultimate
REINSTALL	Specify `ALL` to update all features currently installed. Specify this property only when updating to a Service Pack release within an existing major release (for example, updating an existing SOLIDWORKS Design 2026 SP01 installation to SOLIDWORKS Design 2026 SP02).

Property	Description
REINSTALLMODE	Microsoft Installer re-installation options. For example, when upgrading an installation from one Service Pack release to another within the same major release, you should specify `REINSTALLMODE=vomus`. For details about REINSTALLMODE settings, see the Windows Installer documentation at **www.microsoft.com**. Specify this property only when updating to a Service Pack release within an existing major release (for example, updating an existing SOLIDWORKS Design 2026 SP01 installation to SOLIDWORKS Design 2026 SP02). Do not specify this property the first time you are installing a new major release of SOLIDWORKS Design.
SERVERLIST	SolidNetWork server list. The SolidNetWork server list is a delimited list of strings, each of which refers to a port number on a server with the format: • Triad (comma delimited) =*xxxxX@server_name1,yyyyY@server_name2,zzzzZ@server_name3* • Non-Triad (semi-colon delimited) = *xxxxX@server_name1;yyyyY@server_name2;zzzzZ@server_name3* *X* and *Y* are single digits. After the last server in the list, do not put a delimiter.
SIMULATIONSERIALNUMBER	Serial number for the SOLIDWORKS Simulation product.
SOLIDWORKSSERIALNUMBER	Serial number for SOLIDWORKS Design, which includes all add-ins.

Property	Description
SWMIGRATE	SOLIDWORKS Design version to upgrade as it would be shown in the list in the SOLIDWORKS Install Choice dialog box. For example: `SWMIGRATE="SOLIDWORKS 2026 x64 Edition SP01"` Specify this property only when installing a new major release while there is a prior major release currently installed. Do not specify this property when upgrading from one Service Pack release to another within the same major release.
TOOLBOXFOLDER	Installation folder for SOLIDWORKS Toolbox files. For example: `TOOLBOXFOLDER = "C:\SOLIDWORKS Data"` If your environment has a PDM or shared Toolbox/Hole Wizard location, you can specify a client installation to not install or upgrade Toolbox data in that location using the NOTTOOLBOXSETUP property. Specify: **0** Install Toolbox files. **1** Not install Toolbox files.
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)

Property	Description
UPGRADESWINSTALL	Specify whether to upgrade an existing version of SOLIDWORKS Design to the current version. **0** New installation, do not update. **1** Update Specify this property only when installing a new major release while there is a prior major release currently installed. Do not specify this property when upgrading from one service pack release to another within the same major release.

eDrawings Command Line Properties

These command-line properties are specific to eDrawings®.

<table>
<tr><th>Property</th><th>Description</th></tr>
<tr><td>ADDLOCAL</td><td>Specifies which portion of the eDrawings application to install.
All is the recommended option.
<table>
<tr><th>Command Line Syntax</th><th>Feature</th></tr>
<tr><td>All</td><td>Installs all the features. The eDrawings application searches for installed SOLIDWORKS Design and AutoCAD publisher applications to validate the install.</td></tr>
<tr><td>CALS</td><td>View CAL files in the eDrawings Viewer</td></tr>
<tr><td>Inventor</td><td>Create eDrawings files from the Autodesk Inventor application.</td></tr>
<tr><td>eDrawingsViewer</td><td>View eDrawings and other files.</td></tr>
<tr><td>ProE</td><td>View ProE files using the eDrawings Viewer</td></tr>
<tr><td>DXFDWGTranslator</td><td>View DXF/DWG files in the eDrawings Viewer.</td></tr>
<tr><td>Shortcuts</td><td>Install desktop shortcut.</td></tr>
<tr><td>XML3D</td><td>View 3DXML files in the eDrawings Viewer.</td></tr>
</table></td></tr>
<tr><td>INSTALLDIR</td><td>Specifies the installation folder for eDrawings</td></tr>
</table>

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 and a SERVERLIST is only required for the SOLIDWORKS SNL server configuration. The SolidNetWork server list is a delimited list of strings, each of which refers to a port number on a server with the format: • Triad (comma delimited) *= xxxx@server_name1,yyyy@server_name2, zzzz@server_name3* • Non-Triad (semi-colon delimited) *= xxxx@server_name1,yyyy@server_name2, zzzz@server_name3* *X* and *Y* are single digits. After the last server in the list, do not put a semi-colon.

SOLIDWORKS Composer/Composer Player Command-Line Properties

These command-line properties are specific to SOLIDWORKS Composer and SOLIDWORKS Composer Player.

For SOLIDWORKS Composer:

Property	Description
ACTIVSERIALNUMBER	SOLIDWORKS Composer serial number.
ACTIVSERIALNUMBERPLAYERPRO	Optional. SOLIDWORKS Composer Player Pro serial number.

Property	Description
ADDLOCAL	Optional. If you install the complete application, you do not need to specify the ADDLOCAL property. For a custom installation, use ADDLOCAL with the following required items: • Licensing • CLOSA • InterOps • LicenseSupportFiles • PlugIns • SWComposerCommon • SWPlayer You can also use the optional feature Sync3Dvia.
INSTALLDIR	Specifies the installation folder for the SOLIDWORKS Composer.

For SOLIDWORKS Composer Player:

Property	Description
ACTIVSERIALNUMBERPLAYERPRO	Optional. SOLIDWORKS Composer Player Pro serial number.
INSTALLDIR	Specifies the installation folder for the SOLIDWORKS Composer Player.

On machines without SOLIDWORKS Design installed, you must use SOLIDWORKS Translator to import SOLIDWORKS files into SOLIDWORKS Composer. To deploy SOLIDWORKS Translator, you must use the SOLIDWORKS Installation Manager; you cannot use the command line.

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)

Property	Description
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 SOLIDWORKS Design is installed. (Individual installations and client installations from an administrative image)

SOLIDWORKS Inspection Command Line Properties

These command-line properties are specific to SOLIDWORKS Inspection.

Property	Description
ACTIVSERIALNUMBER	SOLIDWORKS Inspection serial number. (Individual installations and client installations from an administrative image)
INSTALLDIR	Specifies the installation for the SOLIDWORKS Inspection application software. (Individual installations only)

Property	Description
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 Visualize Command Line Properties

These command-line properties are specific to SOLIDWORKS Visualize.

Property	Description
ACTIVSERIALNUMBER	SOLIDWORKS Visualize serial number. (Individual installations and client installations from an administrative image)
INSTALLDIR	Specifies the installation for the SOLIDWORKS Visualize 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.

MSI File Locations for Administrative Images

To create administrative images from the command line, the product component `.msi` files are stored in these locations on the SOLIDWORKS media.

SOLIDWORKS Design (English)

`swwi\data\SOLIDWORKS.msi`

SOLIDWORKS Design (non-English language components)

Traditional Chinese	`swwi\lang\chinese\chinese.msi`
Simplified Chinese	`swwi\lang\chinese-simplified\chinese-simplified.msi`

Czech	swwi\lang\czech\czech.msi
French	swwi\lang\french\french.msi
German	swwi\lang\german\german.msi
Italian	swwi\lang\italian\italian.msi
Japanese	swwi\lang\japanese\japanese.msi
Korean	swwi\lang\korean\korean.msi
Polish	swwi\lang\polish\polish.msi
Portuguese (Brazilian)	swwi\lang\portuguese-brazilian\portuguese-brazilian.msi
Russian	swwi\lang\russian\russian.msi
Spanish	swwi\lang\spanish\spanish.msi
Turkish	swwi\lang\turkish\turkish.msi

DocManager APIs

swdocmgr\SOLIDWORKS Document Manager API.msi

eDrawings

eDrawings\eDrawings.msi

eDrawingsAPI SDK

eDrwAPISDK\eDrawings API SDK.msi

Flow Simulation

Flow Simulation\SOLIDWORKS Flow Simulation.msi

3DExperience Marketplace for SOLIDWORKS

marketplace\3dexperience marketplace.msi

SolidNetWork License Manager

swlicmgr\SOLIDWORKS SolidNetWork License Manager.msi

SOLIDWORKS API SDK

`apisdk\SOLIDWORKS API SDK.msi`

SOLIDWORKS CAM

`cam\SOLIDWORKS CAM.msi`

SOLIDWORKS Composer

`swComposer\SOLIDWORKS Composer.msi`

SOLIDWORKS Composer Player

`swComposerPlayerSOLIDWORKS Composer Player.msi`

SOLIDWORKS Electrical

`swelectric\SOLIDWORKS Electrical.msi`

SOLIDWORKS File Utilities

`SWFileUtilities\SOLIDWORKS File Utilities.msi`

SOLIDWORKS Inspection

`Inspection\SOLIDWORKS Inspection.msi`

SOLIDWORKS Manage Client

`SWManageClient\SOLIDWORKS Manage Client.msi`

SOLIDWORKS PDM

SOLIDWORKS PDM Professional provides an API, but it is available only for the Client and is not available in an `.msi` file. For information, see the *API guide* in the SOLIDWORKS PDM Client *`install`*`\API` directory.

SOLIDWORKS Plastics

`plastics\SOLIDWORKS Plastics.msi`

SOLIDWORKS Visualize

`visualize\SOLIDWORKS Visualize.msi`

SOLIDWORKS Visualize Boost

`visualizeboost\SOLIDWORKS Visualize Boost.msi`

Deploying an Administrative Image Using Microsoft Active Directory

Microsoft Active Directory Deployment is supported for deployment of SOLIDWORKS Design to clients.

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

For information on installing SOLIDWORKS Design through Microsoft Active Directory, contact your reseller.

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 SOLIDWORKS Installation Manager from the **SOLIDWORKS Downloads Page** so that the correct files are referenced.

Once you update 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 shared directory, you can redefine that shared directory using the location of the updated 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 SOLIDWORKS Installation Manager.
2. On the Welcome screen, select **Administrative image**.
3. On the Serial Number screen, enter the SOLIDWORKS Design serial number.

   SOLIDWORKS 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, download and install a newer version:** *version*.

   SOLIDWORKS Installation Manager starts a new session to install the newer version.
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. For details, see **Downloading Updated Installation Files for Administrative Images**.

1. Start SOLIDWORKS Installation Manager.
2. On the Welcome screen, select **Administrative image**.

   Administrative image creation options appear.

Option	Description
Create a new image using default settings.	Create the image using the default configuration settings (that is, not based on the settings established for an existing administrative image at your site.)
Create a new image using settings and files from an existing image.	Create the image using configuration settings specified for an existing administrative image at your site. Click **Browse** to specify the installation directory for the existing 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. For details, see **Upgrading Clients from Administrative Images**.

Upgrading Clients from Administrative Images

SOLIDWORKS Installation Manager can automatically update client computers installed from an administrative image.

Typically, administrative images are organized in folders on a server. To support the automatic upgrade of clients, the 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 Design starts on a client machine that installed from the original administrative image, it sees the new version and upgrades the client.

Also, when SOLIDWORKS Design triggers an update, other products installed from an administrative image automatically update.

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*.

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, specify the sharing properties appropriate for your organization or group and change the share name to the name used for the original administrative image folder.

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

Limitation: When you create an administrative image and deploy it automatically through sharing, SOLIDWORKS Design does not upgrade to a new service pack if you replace the image. SOLIDWORKS Design only upgrades automatically if you installed the original image manually using the Installation Manager.

For example, if you create an administrative image for SOLIDWORKS Design <version> SP1 and deploy it automatically using the sharing options, the client installs and runs as expected. Later, you create a new image for SP5 in the same folder, disable sharing on the SP1 image, and enable sharing on the SP5 image using the same share name.

When you open SOLIDWORKS Design on the client, it does not detect the new image or trigger an upgrade. It opens SP1. Even matching the folder name to the original image does not resolve this issue.

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:

- By using SOLIDWORKS Installation Manager to check for, download, and install updates.

  You can initiate SOLIDWORKS Installation Manager manually, or schedule it to check periodically for updates.

- By downloading service packs manually from the SOLIDWORKS support website.

  Service packs are applied as patches to an existing software version. For example, SOLIDWORKS Design 2026 SP1.0 is a service pack for SOLIDWORKS Design 2026 SP0.0. You can install these service packs using SOLIDWORKS Design 2026 Installation Manager; you cannot install them using SOLIDWORKS Installation Manager in SOLIDWORKS Design 2025.

  For help on downloading and installing service packs, see the installation documentation for the SOLIDWORKS Design 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 SOLIDWORKS Design 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 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 SOLIDWORKS Design.

Removing an Installation

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

1. Make sure that no SOLIDWORKS Design sessions are active.
2. If you want to move the SOLIDWORKS Design license to another computer, deactivate the license before removing the SOLIDWORKS Design installation on this computer.

   For more information, see **Moving a License** on page 79.

3. Open the Windows **Control Panel** and click **Programs and Features**.
4. Select the product component to remove and click **Uninstall**.

   In some cases, you might find multiple SOLIDWORKS Design Service Pack components for a given major release version. To remove a SOLIDWORKS Design major release, remove all Service Pack elements for that major release.

5. On the Uninstall screen:
   a) Select the products to remove.

      By default, all product components are selected.

   b) Clear the products that you do not want to remove.
   c) Click **Change** in **Advanced Options**.

      The Advanced Options screen appears and the Program Files and Folders option is selected.

   d) Select one or more options to perform a custom uninstall, or select all options to perform a complete uninstall.

      By default, the SOLIDWORKS Installation Manager performs a complete uninstall and removes all the files and folders that were added when the installation took place.

   e) Click **Back to Summary**.
   f) On the Summary screen, expand **Advanced Options** to display the uninstall method and entries.
   g) Click **Remove Items**.

4

SOLIDWORKS PDM and SOLIDWORKS Manage

This chapter includes the following topics:

- **Listing of PDM Server Components**
- **Understanding Differences between PDM Clients**
- **Before Installing PDM Server**
- **Installing a PDM Server with the SOLIDWORKS Installation Manager**
- **Installing a PDM Client with SOLIDWORKS Installation Manager**

SOLIDWORKS PDM is a document management product based on the client/server architecture known formerly as SOLIDWORKS Enterprise PDM. SOLIDWORKS PDM is available in two types:

- SOLIDWORKS PDM Standard
- SOLIDWORKS PDM Professional

SOLIDWORKS PDM uses a Microsoft SQL Server database to manage file vaults.

This table lists the database systems, server components, and client options available for each product type.

	PDM Standard	PDM Professional
Designed for	Small to medium office networks, generally with less than 20 concurrent users.	Large office networks, large numbers of concurrent users
Limits	Lesser of 1 socket or 4 cores, 1 GB of RAM, and 10 GB database size	SQL Standard limits: Lesser of 4 sockets or 16 cores, 128 GB of RAM, and 524 PB database size
Database Management System	Microsoft SQL Server Express 2019 or later versions up to 2022.	Microsoft SQL Server 2019 or later versions up to 2022.
Requires SolidNetWork License Manager?	Yes	Yes
Includes Archive Server and Database Server?	Yes	Yes

	PDM Standard	PDM Professional
Includes SOLIDWORKS PDM Web2 Server and WebAPI Server?	No	Yes
Client types	Native	Native, browser-based
Client add-ins for CAD Editors	SOLIDWORKS Design, DraftSight	SOLIDWORKS Design, DraftSight, Microsoft Office Integration
Client add-ins for Contributors	DraftSight	DraftSight, Microsoft Office Integration
Client add-ins for Viewers		Microsoft Office Integration

SOLIDWORKS Manage Professional provides advanced data management tools to SOLIDWORKS PDM Professional. It offers capabilities for advanced version control, and for project, process, and item management. Interactive dashboards and reports help you track items, activities, and progress toward completing projects.

SOLIDWORKS Manage Professional has a similar architecture as SOLIDWORKS PDM Professional. It uses a client-server architecture, with a Microsoft SQL Server database to manage item and project data. There are three services for database, file server, and web server functions, which support both thick clients and web browser clients. The three services can be hosted on the same server or separate servers. Users with SOLIDWORKS Manage Professional Editor licenses can use a SOLIDWORKS Manage Add-in to access Manage data directly from within SOLIDWORKS.

You can use the SOLIDWORKS Installation Manager to install the SOLIDWORKS Manage Professional server and client.

For instructions on installing SOLIDWORKS Manage Professional, see the *SOLIDWORKS PDM/SOLIDWORKS Manage Installation Guide*.

Listing of PDM Server Components

This table summarizes the server components of the SOLIDWORKS PDM Server.

Server	Function	Installation
Microsoft SQL Server Express for PDM Standard	Manages data in SOLIDWORKS PDM data vaults.	Can install with PDM Standard, or use a version previously licensed and installed.
Microsoft SQL Server for PDM Professional	Manages data in SOLIDWORKS PDM data vaults.	Required. You can also use a version previously licensed and installed.

Server	Function	Installation
SolidNetWork License Manager Client	Manages SOLIDWORKS PDM licenses.	Must be installed before new vaults can be created. Can be installed with SOLIDWORKS PDM server components when using the SOLIDWORKS Installation Manager.
Archive Server	Manages transfers of files between clients and the file vault archive, and manages users and their credentials.	Required. Install with PDM Standard or Professional.
Database Server	Polls PDM databases for updates, manages data import and export rules, manages the message system, and schedules replication and cold storage.	Required. Install with PDM Standard or Professional.
Web2 Server	Provides access to file vaults over the internet from browser-based clients.	Optional. Install with PDM Professional only.
Web API Server	EXALEAD OnePart users can use a Web API server to index and retrieve SOLIDWORKS PDM data.	Optional. Install with PDM Professional only.

Archive servers run as services on the system account of the computers where they are installed. One archive server can host multiple file vaults on the same computer.

In a stand-alone installation, the archive server, database server, license server, and web API server, as well as the SQL Express or SQL Server database, can all be installed on the same machine.

In a distributed environment, the server components can reside on different machines. SOLIDWORKS PDM Professional also supports replication of file vault archives and the SQL server database to improve performance at remote locations. For more details, see the *SOLIDWORKS PDM Professional Replication Guide*.

Understanding Differences between PDM Clients

Before installing the client software, you must determine what clients are needed depending on your PDM environment.

This table summarizes the clients for SOLIDWORKS PDM Standard:

Standard Client type	Description
CAD Editor	Supports working with SOLIDWORKS Design, DWG/DXF format, Microsoft Word, and Microsoft Excel files. All file formats, that Office plug-in handles, behave in the same way in SOLIDWORKS PDM Standard except Preview.
Contributor	Supports working with SOLIDWORKS Design, DWG/DXF format, Microsoft Word, and Microsoft Excel files.
Viewer	Allows read-only access to file vaults. User can transition files through a workflow, but user cannot add or modify (check out, check in, update values) files.

This table summarizes the clients for SOLIDWORKS PDM Professional:

Professional Client type	Description
CAD Editor & Web	Supports working with all file types, including enhanced management and previewing of CAD formats such as SOLIDWORKS. CAD add-ins allow users to access SOLIDWORKS PDM Professional from within the CAD application. Remote users can search vaults and perform basic functions over the Internet using Web2 clients.
Contributor & Web	Supports working with all file types, including CAD files. However, the CAD add-ins are not supported on this client type. Remote users can search vaults and perform basic functions over the Internet using Web2 clients.
Viewer	Allows read-only access to file vaults. User can transition files through a workflow, user cannot add or modify (check out, check in, update values) files or use CAD add-ins.

Before Installing PDM Server

Keep these requirements in mind before installing PDM server components:

- Obtain full administrative rights to the machines on which you are going to install servers and clients.

- Ensure that the database server has access to the archive server over TCP port 3030 and to the SQL Server over TCP port 1433.
- If you are going to use an existing SQL Server Express or SQL Server database, obtain the name of the SQL Server database and the account ID and password of a SQL Server administrator or database owner with read/write access to the database. You cannot use the `tew` user.
- If a PDM client is already installed on the system, uninstall it. If you want the PDM Server and PDM Client on the same system, you can install them both from the Server products page in the SOLIDWORKS Installation Manager.

Installing a PDM Server with the SOLIDWORKS Installation Manager

When you install the PDM Server for PDM Standard, you can install the archive server, the database server, and the PDM CAD Editor client on the same machine. You do not need to run the client installation separately.

When installing the PDM Server for PDM Professional, you can include these components, and also the Web2 server and Web API server.

To install a PDM Server and its components:

1. Start the SOLIDWORKS Installation Manager.
2. On the Welcome page, select **Install server components**, and click the following:
   - **Install SOLIDWORKS PDM Server components on this computer**.
   - **Install SolidNetWork License Manager on this computer**.

     If the SolidNetWork License Manager is installed on this or another computer, this option does not display.
3. If the SolidNetwork License Manager Options page displays, enter a serial number and specify a location for the directory, and click **Next**.
4. Follow instructions for system warnings and click **Next**.
5. On the Summary page, for **SOLIDWORKS PDM Server**, click **Change**.

6. In the SOLIDWORKS PDM Server Options page:
   a) For **Select Server Product**, click **SOLIDWORKS PDM Standard** or **Professional**.
   b) For **Installation Location**, click **Browse** to specify a file path.
   c) In **Select Features**:
      1. Click **Archive Server**, **Database Server**, and **Client** to install these components on the same machine. If you are in a distributed environment, you can install the archive server and database server by running the Installation Manager on another machine.

         - If a PDM client is already installed on the machine, you must cancel the installation and uninstall the client. You can then run the SOLIDWORKS Installation Manager to install the PDM server with the client together.
         - The SOLIDWORKS PDM client component included with the SOLIDWORKS PDM server install includes Item Explorer, which cannot be excluded.

      2. If you are installing PDM Professional, you can also click **Web2 server** and **Web API Server**.

         If you are installing the Web API server, you can specify a value for **HTTP port number** that is between `0` and `65535`. The default value is `65453`.

   d) In **SQL Server**, you can install a new instance of the database tools that come with PDM Server (SQL Server Express for PDM Standard), or you can specify the database name, username, and password of an existing instance.

      A full version of SQL Server must be installed for SOLIDWORKS PDM Professional.

   e) Click **Back to Summary**.
7. On the Summary page, accept the terms of the SOLIDWORKS License Agreement and click **Install Now**.

   When the SOLIDWORKS Installation Manager completes installing, the Archive Server Configuration Tool starts. Follow the prompts in the wizard to configure the archive server.

To set up, configure, and troubleshoot the archive server, database server, Web2 server, and SQL Server databases, see the relevant topics in the *SOLIDWORKS PDM Installation Guide*.

Modifying a SOLIDWORKS PDM Server Installation

To modify an installation of SOLIDWORKS PDM Server and its components, use Programs and Features from the **Windows Control Panel**. You cannot modify the installation with SOLIDWORKS Installation Manager (SLDIM), even if that is what you used to install SOLIDWORKS PDM Server.

Installing a PDM Client with SOLIDWORKS Installation Manager

The major version of the SOLIDWORKS PDM Client must match the major version of the SOLIDWORKS PDM Server. When you install or upgrade the SOLIDWORKS PDM Client, make sure the versions match.

To install a PDM Client:

1. Start the SOLIDWORKS Installation Manager.
2. On the Welcome page, select **Install on this computer**.
3. On the Serial Number page, click **Next**.

   SOLIDWORKS PDM clients do not require a separate serial number.

4. Follow instructions for system warnings and click **Next**.
5. On the Summary page, under Products, click **Change**.
6. On the Product Selection page, click **Select different packages or products**.
7. In the Select the package you want to show dialog box, select a SOLIDWORKS Design package that includes a PDM license. Then click **OK**.
8. On the Product Selection page, select **SOLIDWORKS PDM Client**. Then select **eDrawings** to enable the PDM Preview functionality. If you are using SOLIDWORKS PDM Professional, you can also select **Item Explorer**.
9. On the Summary page, expand **SOLIDWORKS PDM Options** and click **Change**.
10. On the SOLIDWORKS PDM Client Options page, select the following and click **Back to Summary**:
    - **PDM product type**
    - **PDM client type**
    - **Add-ins**
11. On the Summary page, accept the terms of the SOLIDWORKS License Agreement and click **Install Now**.

Administrators can also use the Administrative Image Option Editor to create and deploy images of PDM clients. For details, see **Using Administrative Images** on page 16.

5

SOLIDWORKS Electrical

This chapter includes the following topics:

- **Requirements for Installing SOLIDWORKS Electrical**
- **Standalone or Client/Server Installation**
- **Configuring the Collaborative Server**
- **Configuring Microsoft SQL Server**
- **Specifying the Data Location**

The SOLIDWORKS Installation Manager allows you to install all the components of SOLIDWORKS Electrical, in a standalone or client/server mode.

SOLIDWORKS Electrical uses several components. During the installation, you must select these components according to the type of installation you want.

- **SOLIDWORKS Electrical Schematic** - 2D application to create and manage electrical schemes.
- **SOLIDWORKS Electrical 3D** - SOLIDWORKS add-in to integrate 2D schematics with 3D models. The SOLIDWORKS Electrical 3D allows you to manage the layout of electrical devices and to route the wires, the cables, and the harnesses.
- **Data** - The Data groups all the data required by the software. This data is graphical (such as files representing the symbols and title blocks) or technical (such as Manufacturer catalogs and data associated with the symbols). You can share the data to be accessible for all users.
- **Collaborative Server** - A utility to manage the opening and updating of electrical drawings.
- **SQL Server** - SOLIDWORKS Electrical requires a Microsoft SQL Server database. You can choose to create a new database or use an existing one during the installation.

Type of Installation

You can install all SOLIDWORKS Electrical components on the same machine. This is a standalone install. All program data, such as project drawings and reports, is stored on the workstation.

You can also install the SOLIDWORKS Electrical components as a client/server application. Use this type of installation if you want to store the data on a network server. The client/server installation lets several users work on the same project. Client systems must have network access to the server and read/write privileges for creating projects, libraries, and symbols.

Information

Whatever the type of installation (standalone or client/server), you must start your Windows session in full rights mode to install SOLIDWORKS Electrical.

You can install only one version of SOLIDWORKS Electrical on a machine. If you are installing SOLIDWORKS Design, and select SOLIDWORKS Electrical as one of the products to install, the installation program removes the old version of SOLIDWORKS Electrical.

When you start the SOLIDWORKS Installation Manager, it automatically detects the existing versions of SOLIDWORKS Electrical installed on the workstation or on the server.

Always back up an existing SOLIDWORKS Electrical environment before you install a new version. From within SOLIDWORKS Electrical Schematic, select **File** > **Archive Environment**, or from within SOLIDWORKS Electrical 3D, select **SOLIDWORKS Electrical** > **Tools** > **Archive Environment**.

Requirements for Installing SOLIDWORKS Electrical

To access current system requirements, see **System Requirements**.

Standalone or Client/Server Installation

You can install SOLIDWORKS Electrical in two modes. In standalone mode, you install the software on the workstation. In client/server mode, users share the data.

SOLIDWORKS Electrical must always connect to an instance of SQL Server.

Stand-Alone Installation

For a first installation, you MUST install all the components on the workstation.

- SOLIDWORKS Electrical Schematic
- SOLIDWORKS Electrical 3D
- Collaborative Server
- Application Data

1. In the **Summary** tab, click **Change** of the **Product Selection** section and select all the SOLIDWORKS Electrical components.
2. If you want to modify the installation options, in the **Summary** tab, click **Change** of the **Electrical Options** section. It is possible to create a new instance of SQL Server or to use an existing one. See specific chapters about the Data and Microsoft SQL Server in this documentation for more details. If not, keep the default options.
3. Click **Install now** to continue.

If a SOLIDWORKS Electrical is already installed on the workstation, the **SOLIDWORKS Installation Manager** automatically finds it and prompts you to update it. Install all the components without changing the **Electrical Options**.

Client/Server Installation

A client/server installation allows you to share the data between several users.

On the server, you must install the following components:

- Collaborative Server
- Application Data
- SQL Server

On the clients, you must install the following components:

- SOLIDWORKS Electrical Schematic
- SOLIDWORKS Electrical 3D

1. On the server, install the server components.
2. In the **Welcome SOLIDWORKS Installation Manager** tab, click **Install server component**.
3. Select the **SOLIDWORKS Electrical** option.
4. In the **Summary** tab, you can modify the installation options. See specific chapters about the Data and Microsoft SQL Server in this documentation for more details. If not, keep the default options.
5. Click **Install now** to continue.
6. On the clients, install the applications.
7. In the **Welcome SOLIDWORKS Installation Manager** tab, click **Install on this computer**.
8. In the **Summary** tab, click **Change** of the **Product Selection** section and clear the **Electrical Server** option.
9. In the **Summary** tab, click **Change** of the **Electrical Options** section and modify the settings to connect to the server.
10. Click **Install now** to continue.

If a SOLIDWORKS Electrical is already installed on the workstation or on the server, the **SOLIDWORKS Installation Manager** automatically finds it and prompts you to update it. Use the same process than the first installation to update the software.

Configuring the Collaborative Server

The Collaborative Server is a Windows Service that must be running at all times, and have constant, uninterrupted, bidirectional communication with clients through firewalls and virus scanners.

To configure the Collaborative Server in individual or server mode, specify the following parameters:

- The machine name (for example `LOCALHOST` or `MyMachine)`, or its IP address (for example `127.0.0.1`, or `192.168.255.255`).
- The port number (by default `26666)`.

You can configure the Collaborative Server during the installation or later in the **Tools** > **Application Settings** command.

For client/server installations, to provide uninterrupted bidirectional communication, make sure that port number 26666 is available and open on the machine where the Collaborative Server is installed. If not, ask the system administrator to find another port for SOLIDWORKS Electrical. You need to ensure that incoming connections are allowed on the server machine for SOLIDWORKS Electrical port (26666).

Configuring Microsoft SQL Server

Ensure that an instance of the SQL Server database is available for any type of SOLIDWORKS Electrical installation. You can use an existing SQL Server database if it is already installed on the system, or install a new instance as part of the SOLIDWORKS Electrical installation.

When you install a new instance of SQL Server, SOLIDWORKS Design installation provides a free version of Microsoft SQL Express 2022. This version is limited to 1 CPU, 1 GB of memory, and 10 GB of data. The memory and CPU limitations are sufficient to support 10 concurrent users.

If you install SQL Express 2022, ensure that the server software and hardware meet the requirements listed in **System Requirements**.

As a best practice, if there are more than 10 concurrent users on the same server, you should purchase and install a full version of SQL Server. Use the workflow described for installing SOLIDWORKS Electrical with an existing instance of SQL Server.

For information about the SQL Express 2022, consult the Microsoft SQL Server documentation.

Managing Data in SOLIDWORKS Electrical

Many programs use a single database which contains all the program data. This is not the case with SOLIDWORKS Electrical. Each project in SOLIDWORKS Electrical is assigned a unique ID number, and every project has a related database that can be identified based on its unique ID number. For example, in the SOLIDWORKS Electrical Project Manager, if you find a project with ID#112, then the associated database is named "tew_project_data_112."

The reason for having a unique database for each project is that it significantly reduces process time. Also, in the case of a critical database failure, only one project is affected, retaining the integrity of the majority of the program information.

However, some companies have policies that restrict the creation of new databases. To run SOLIDWORKS Electrical, there can be no restrictions on creating new databases by users. Check with the system or database administrator to ensure that no configuration settings prevent users from creating new databases.

New Instances of SQL Server

If you install a new instance of SQL Server, you need full administrative rights to the server, and also to any client systems. You must also ensure that clients have read/write privileges for the database, so they can create and modify projects, libraries, symbols and other project data.

When you install a new instance of SQL Server, select a name other than the default.

Existing Instances of SQL Server

You need a user account that has full administrative rights to the SQL Server database to install new SOLIDWORKS Electrical databases or to connect to a SOLIDWORKS Electrical installation on an existing instance of SQL Server.

For SOLIDWORKS Electrical to function properly, you must enable SQL Server for mixed mode authentication. Mixed mode authentication allows the use of either Windows or SQL Server credentials.

To connect to an existing SQL Server database, you can use Windows or SQL Server authentication. Enter a user name and password for a Windows or SQL Server login that has full SQL Server administrative rights.

You cannot use the 'tew' user to connect to SQL Server for installation. Contact your SQL Server administrator or system administrator for help to obtain a login with the proper credentials for SQL Server.

SOLIDWORKS Electrical Databases

After you install SOLIDWORKS Electrical, the installation program creates the following databases in the SQL Server instance.

- TEW_APP_DATA
- TEW_APP_MACRO
- TEW_APP_PROJECT
- TEW_CATALOG
- TEW_CLASSIFICATION

Specifying the Data Location

For the data location, select a directory to store program data such as parts, templates, and 2D and 3D symbols. This location must be a dedicated directory. It cannot be a root directory or the same directory used to store SOLIDWORKS data.

The default directory is `C:\ProgramData\SOLIDWORKS Electrical`.

In the case of a client/server installation, you must share the Data directory.

The installation program creates a new folder called `Update` in the default directory to store all of the new data.

When you start SOLIDWORKS Electrical for the first time, the program automatically merges the data stored in the `Update` folder with the existing data. In the Update Data wizard, you can select the data to be added into the application and the SQL Server database.

6

SOLIDWORKS Visualize and SOLIDWORKS Visualize Boost

This chapter includes the following topics:

- **System Requirements for SOLIDWORKS Visualize**
- **Installing SOLIDWORKS Visualize with the SOLIDWORKS Installation Manager**
- **Installing Visualize Boost with the SOLIDWORKS Installation Manager**

SOLIDWORKS® Visualize provides a suite of stand-alone tools that combine advanced rendering capabilities with visual design-oriented features and workflows, so you can create highly realistic images of 3D CAD models.

SOLIDWORKS Visualize products are available in two packages:

- Standard
- Professional

A complimentary seat of SOLIDWORKS Visualize Standard is included with each SOLIDWORKS Design Professional and Premium license on active subscription. For stand-alone activation SOLIDWORKS customers, you need an additional serial number. Check with your SOLIDWORKS administrator or reseller if you do not know your SOLIDWORKS Visualize serial number.

SOLIDWORKS Visualize also supports SolidNetWork Licensing (SNL) license types such as floating, as well as offline activation.

Using the SOLIDWORKS Installation Manager (SLDIM), you install SOLIDWORKS Visualize as part of the individual (**Install on this computer**) installation. You can install it as a stand-alone application, which is an advantage if you want to use separate computers for modeling and rendering. It does not consume a SOLIDWORKS Design license.

SOLIDWORKS Visualize Boost is a service that allows you to distribute rendering from a computer running SOLIDWORKS Visualize Professional to one or more computers on a network. You can perform other tasks on your computer while the rendered image generates, and the final rendered images are available sooner. You can also install Visualize Boost as part of the individual (**Install on this computer**) installation in the SOLIDWORKS Installation Manager.

A complimentary seat of SOLIDWORKS Visualize Boost is included with each seat of SOLIDWORKS Visualize Professional on active subscription.

> Visualize Boost is a product add-on for Visualize Professional only. Visualize Boost does not work with Visualize Standard.

System Requirements for SOLIDWORKS Visualize

For optimal performance and setup of SOLIDWORKS Visualize and SOLIDWORKS Visualize Boost, follow these guidelines prior to installation:

- Ensure you have a separate serial number for each product before installing SOLIDWORKS Visualize and SOLIDWORKS Visualize Boost.
- Verify that every computer running SOLIDWORKS Visualize or SOLIDWORKS Visualize Boost meets the requirements listed in **System Requirements**.
- Install the SOLIDWORKS Visualize Boost application on dedicated rendering computers within the same network. These computers should be different from those running SOLIDWORKS Visualize Professional to offload rendering tasks.
- If an applicable GPU is not found, SOLIDWORKS Visualize as well as SOLIDWORKS Visualize Boost default to the central processing unit (CPU) instead. See **SOLIDWORKS Hardware Certification** for more information on specific graphics cards and drivers.

For GPU Support Using 3DS Stellar Physically Correct

Stellar **Accurate Render Mode**:

- NVIDIA® graphics card based on at least the Maxwell™ microarchitecture
- Recommended driver version of 538.78 with support for CUDA® 12.2
- A minimum of 4 GB of video memory

Stellar **Fast Render Mode**, you need one of the following:

- NVIDIA graphics card based on at least the Turing® microarchitecture (with RT Cores)
- Or AMD® graphics card based on RDNA® 2 or newer microarchitecture

For GPU Support Using AMD ProRender

AMD ProRender™ **Accurate Render Mode**:

- A GPU and driver supporting the Vulkan® standard
- A minimum of 4 GB of video memory

Installing SOLIDWORKS Visualize with the SOLIDWORKS Installation Manager

To install SOLIDWORKS Visualize:

1. Start the SOLIDWORKS Installation Manager.
2. On the Welcome page, select **Install on this computer**.
3. Do the following:
   a) On the Serial Number page, expand **Visualization**.
   b) Select **SOLIDWORKS Visualize** and enter the serial number.
   c) Click **Next**.
4. Follow instructions for system warnings and click **Next**.
5. On the Summary page, accept the terms of the SOLIDWORKS License Agreement and click **Install Now**.

Follow the instructions in the Product Activation wizard to activate your product.

After activation, double-click the SOLIDWORKS Visualize shortcut icon on the desktop to start the application or from the Windows **Start** menu, click **SOLIDWORKS *version*** > **SOLIDWORKS Visualize *version***.

Post Installation Tasks

After installing, there are a few tasks you perform to configure SOLIDWORKS Visualize.

For more information, see *Getting Started with SOLIDWORKS Visualize* in the SOLIDWORKS Visualize **Help**.

Creating the Content Folder

When you first run the software, SOLIDWORKS Visualize prompts you for a location to create a Content folder.

The Content folder contains the following subfolders where you can store assets for your projects:

Subfolders	Stores
Cameras	Default cameras and cameras that you have saved
Environments	High dynamic range (HDR) images
Palettes	Backplate images
Images	Renderings and snapshots
Appearances	Materials in your Appearance library The contents update when you add an appearance to the library.
Models	Model files By default, SOLIDWORKS Visualize searches this folder first when you use **File** > **Import** a model.
Plates	Standard 2D images for use in backplates
Projects	Native project files in `.SVPJ` format
Textures	2D textures for use in appearances

Accessing Help, Tutorials, System Info, and Licensing

The Help menu provides documentation and options to update the software and troubleshoot your system.

Visualize Help	Opens the SOLIDWORKS Visualize online help.

Visualize Tutorials	Opens MySolidWorks and lets you access the SOLIDWORKS Visualize tutorials.
Release Notes	Displays online Release Notes for the current version.
What's New	Displays new functionality for SOLIDWORKS products.
System Info	Performs a system check of the SOLIDWORKS Visualize software with your PC hardware, and suggests upgrades when needed.
Check for Updates	Checks the server for the latest SOLIDWORKS Visualize update.
Activate License	(Node-locked licenses) Uses SOLIDWORKS Product Activation wizard to activate your product.
Deactivate License	(Node-locked licenses) Uses SOLIDWORKS Product Activation wizard to deactivate your product.
SolidNetWork License Manager	Uses SOLIDWORKS Product Activation wizard to deactivate your product.
About Visualize	Displays the version number and licensing information of your copy of SOLIDWORKS Visualize.

Installing Visualize Boost with the SOLIDWORKS Installation Manager

You install Visualize Boost using the SOLIDWORKS Installation Manager. It requires a separate serial number.

You should install Visualize Boost on machines different from the ones running SOLIDWORKS Visualize Professional. You can maximize rendering performance by installing Visualize Boost on computers with multiple graphics processing units (GPUs).

To install SOLIDWORKS Visualize Boost:

1. Start the SOLIDWORKS Installation Manager.
2. On the Welcome page, select **Install on this computer**.
3. Do the following:
   a) On the Serial Number page, expand **Visualization**.
   b) Select **SOLIDWORKS Visualize Boost** and enter the serial number.
   c) Click **Next**.
4. Follow instructions for system warnings and click **Next**.

5. On the Summary page, accept the terms of the SOLIDWORKS License Agreement and click **Install Now**.

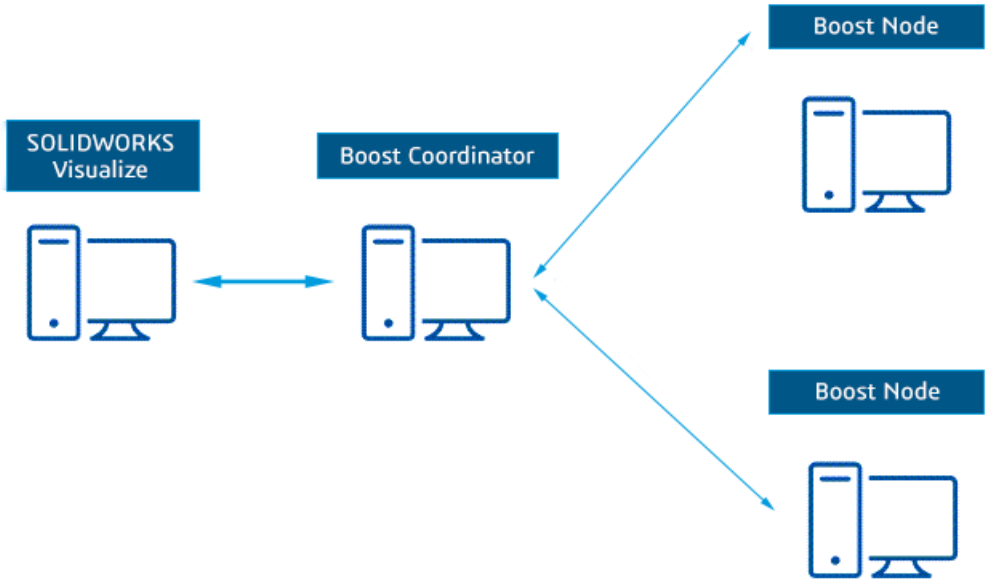
Follow the instructions in the Product Activation wizard to activate your product.

After activating, open the app by double-clicking the SOLIDWORKS Visualize Boost desktop shortcut .

Configuring Visualize Boost

You can configure the primary Visualize Boost machine as the Coordinator and additional machines in the render cluster as Nodes.

Visualize Boost allows you to render images on a single machine or across a cluster of machines. The setup is simple and intuitive. A primary computer, the **Coordinator**, manages the cluster and distributes the rendering jobs from SOLIDWORKS Visualize. Additional machines called **Nodes**, connect to the Coordinator to assist with these tasks on a network. In a render farm, each machine in the cluster runs an instance of Visualize Boost continuously.



To configure Visualize Boost:

1. Open Visualize Boost:

   - Double-click the Visualize Boost desktop shortcut .
   - From the Windows **Start** menu, click **SOLIDWORKS <version>** > **SOLIDWORKS Visualize Boost <version>**.

   After activation, the Visualize Boost window identifies and shows the name and IP address of the licensed machine.

2. In Visualize Boost, click **File** > **Settings**.

3. In the Settings dialog box:
   a) Select one machine as the **Coordinator**. Leave the **Coordinator** clear when configuring all other **Nodes**.
   b) Accept the default values of `8001` for the **Boost Port** and `8002` for the **DStellar Port**.

      You can update these ports to meet network requirements. Make sure to add firewall exceptions for node communications. For questions, consult your network administrator.

4. Optional: Additional parameters include:

Option	Description
Language	The machine automatically uses the language of the operating system.
Theme	Choose between a **Light** or **Dark** background for the **Theme**. This setting affects SOLIDWORKS Visualize if it is installed on the same machine as Visualize Boost.
Notifications Enabled	Shows messages from DStellar.Boost in the Windows notification area.
Export	Saves the Visualize Boost configuration settings to a `.json` file.
Import	Loads existing settings to an instance of Visualize Boost in a `.json` file.

5. Click **Apply**.

To finalize the setup, see the instructions for **Working with Visualize Boost**.

7

License Administration

This chapter includes the following topics:

- **Administering Individual Licenses**
- **Administering Licenses Using a SolidNetWork License Manager**

After installation, you can activate SOLIDWORKS licenses either by activating individual computers (machine 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

You can specify an individual license to use machine activation.

Typically, administration of individual licenses consists of activating and deactivating licenses directly from SOLIDWORKS Design.

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

Starting with SOLIDWORKS 2022 and later, if you purchased a perpetual license of SOLIDWORKS Design, you must also reactivate the license once a year. A perpetual license never expires but needs to be reactivated. SOLIDWORKS Design prompts you to reactivate the perpetual license before the required reactivation date.

You can deactivate individual licenses by clicking ⓘ > **Licenses** > **Deactivate** in SOLIDWORKS Design.

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 move licensing rights from one computer to another.
- 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. For details, see **Administrative Image Option Editor** on page 20.

To activate a license:

1. Start SOLIDWORKS Design.
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 Design.
   - If the SOLIDWORKS Product Activation window does not appear automatically, start activation by clicking ? > **Licenses** > **Activate**.
3. Select 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**.
   - Internet: Activation occurs automatically.
   - Email: Click **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 required, you can exit and rerun the activation procedure to open the response file.
7. Click **Finish**.

Reactivating a License

If you have perpetual commercial or educational licenses of SOLIDWORKS 2022 and later products, you must reactivate those licenses once a year. SOLIDWORKS Product Activation prompts you to do this before the required reactivation date.

To reactivate a license:

1. Start SOLIDWORKS Design.

   SOLIDWORKS Product Activation prompts you to reactivate. For each SOLIDWORKS product license, it displays the following:
   - The date the current **Activation Expires**
   - The **Action Required** to continue using the product, such as reactivating a perpetual license or renewing and reactivating a term or evaluation license.

2. Click **Reactivate Now** to start the Product Activation wizard.

   If you do not want to reactivate the license immediately, click **Remind Me Later**.

   For example, in SOLIDWORKS Design, click ⓘ > **Licenses** > **Activate** to start the Product Activation Wizard. See **Activating a License** on page 77.

Moving a License

To move a SolidNetWork license to a new or upgraded computer, first deactivate it from the license key server at Dassault Systèmes SOLIDWORKS Corporation. Then reactivate it on the other computer.

> If you cannot deactivate the license from Dassault Systèmes SOLIDWORKS Corporation (because of hard disk drive failure, for example), reactivation is still successful in most cases. If not, contact your Value Added Reseller for help.

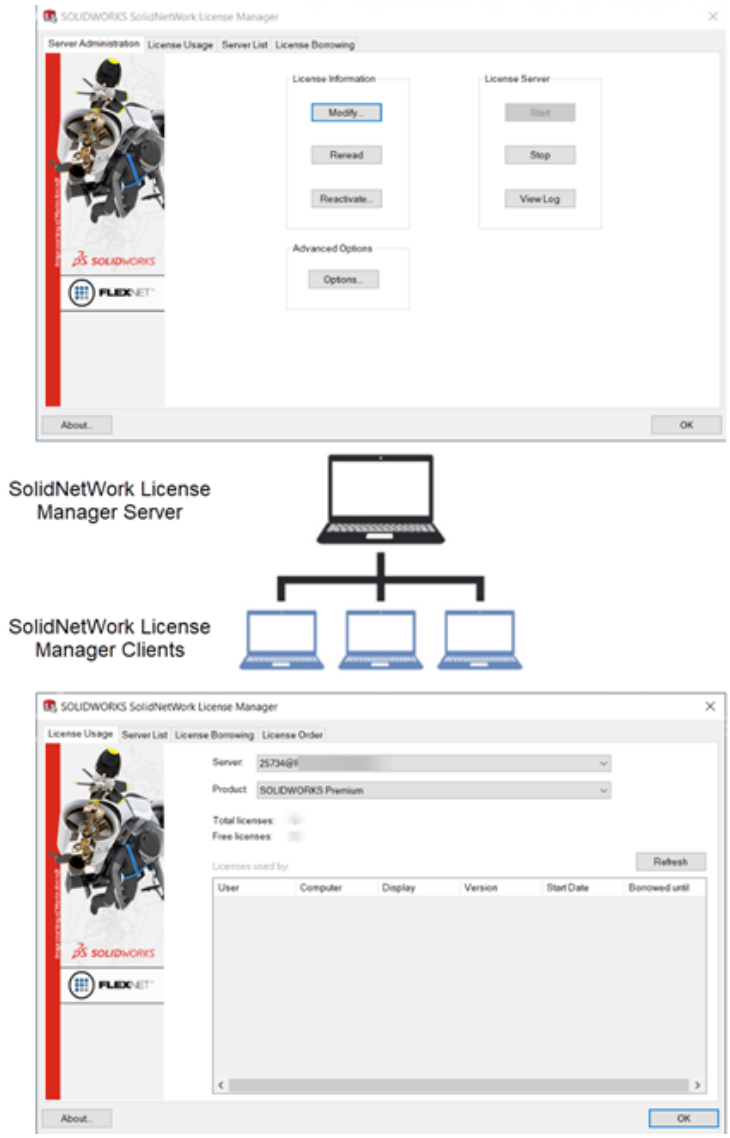
To move a license:

1. In SOLIDWORKS, click ⓘ > **Licenses** > **Deactivate**, and follow the instructions.
2. To reactivate, go to the new or upgraded computer and activate the license as you did during the initial activation. For details, see **Activating a License**.

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:



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.

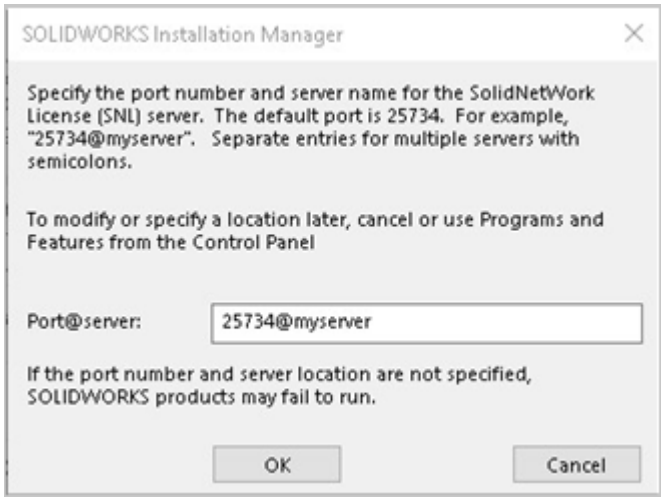
  If you specify a SolidNetWork license serial number when installing, SOLIDWORKS Installation Manager identifies that computer as a SolidNetWork license client. SOLIDWORKS Installation Manager prompts you for the SolidNetWork License Manager computer name and port number to connect the client to the license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

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

Installing SolidNetWork Licensing Overview

You use the SOLIDWORKS Installation Manager to install the SolidNetWork License Manager Server on at least one computer before you install SOLIDWORKS client software on any computers. You can then use the Product Activation Wizard to add and activate serial numbers for perpetual and term licenses.

When you install SOLIDWORKS client software on a computer, specify a SolidNetWork license serial number. SOLIDWORKS Installation Manager identifies that computer as a SolidNetWork license client and prompts you for the computer name and port number of the SolidNetWork License Manager Server. It displays an error if the SolidNetWork License Manager Server is not installed or if the name or port number is incorrect.



Activation Process

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

Activation requires that you have an active internet connection 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.
- Lets you move SolidNetWork licensing rights from one SolidNetWork License Manager computer to another.
- 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 Design, SolidNetWork licenses for previous versions of SOLIDWORKS Design are activated. You no longer need to retain the dongle.

Installing the SolidNetWork License Manager Server

The SolidNetWork License Manager Server supports multiple license clients. It distributes licenses to clients on the network.

If you are upgrading from a previous version of SolidNetWork License Manager Server that used hardware dongles for license activation, remove the hardware dongle before installing the new version of SolidNetWork License Manager Server.

1. Start SOLIDWORKS Installation Manager as described in **Download Installation Media**.
2. For the type of installation, select **Install server components** and click **Install SolidNetWork License Manager (distributed licensing) on this computer**.
3. On the Summary page, review the installation location and background downloader, agree to the license terms, and click **Install Now**.
4. In the SOLIDWORKS Product Activation Wizard from the SolidNetWork License Manager Server, add a serial number and activate at least one license. See **Adding Serial Numbers and Activating SolidNetWork Licenses**.

Activating SolidNetWork Licenses

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

To activate a SolidNetWork license:

1. Start the SolidNetWork License Manager by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server*version***.
2. Optional: If you have not yet activated the SolidNetWork license, the software asks you to activate it. Click **Yes**.

   This starts the SOLIDWORKS Product Activation wizard.
3. Optional: If a firewall is in place, select **A firewall is in use on this server**.
4. Enter the port numbers in the SolidNetWork License Server Information page and then click **Next**.
5. In **Serial #:** select specific serial numbers or click **Select All** to highlight and activate all your SOLIDWORKS Design serial numbers.
   a) Select the option to activate **Automatically over the internet (recommended)**.
   b) Under **Contact Information**, enter an email address.

   The Activation/Reactivation Succeeded dialog box displays the products that have been activated.
6. Click **Finish**.

Manual Activation with a SolidNetWork License Manager Server

When you install or upgrade the SolidNetWork License Manager Server, you need to activate or reactivate the license. You can activate or reactivate the license over the internet or choose to do it manually.

When you activate manually:

- For a new SolidNetWork License Manager Server, you need to activate twice to configure the server correctly. You are initializing the license system on the server the first time you activate. The second time you repeat the procedure, you are activating the license.

- For an upgraded SolidNetWork License Manager Server, you need to reactivate the existing license. If the server does not have activated licenses, you run activation by using **Modify** on the Server Administration tab instead.

Initializing a New SolidNetWork License Server

To initialize a new SolidNetWork License Server:

1. After installing and opening the SolidNetWork License Manager Server, click **Yes** when prompted to activate.
2. In the SolidNetWork License Server Information dialog box, verify the **Firewall** and **Ports**, and click **Next**.
3. In the Product Activation Wizard, select the **Serial Numbers** to activate, select **Manually via E-mail** as the activation method, and enter the **Email Contact Information**.

   The manual method requires that you create a request file and email it to activation@solidworks.com. When you receive the email containing the license response file, extract the file, and click **Open** to load it to the server.

   You can close and rerun the Product Activation Wizard to open and load the response file.

4. When a message box notifies you that the initialization succeeded, click **OK** and in the Product Activation Wizard, click **Finish**.

Manually Activating a New SolidNetWork License Manager Server

After initializing the SolidNetWork License Manager Server, open the SolidNetWork License Manager Server and create a second request to complete activation.

To manually activate a new SolidNetWork License Manager Server:

1. Do one of the following:
   - If a prompt asks you to activate, click **Yes**.
   - If the prompt does not appear, click **Modify** in the Server Administration tab.

   By default, Advanced Options are set to Use Activation mode.

2. Verify the **Firewall** and **Ports** in the SolidNetWork License Server Information dialog box, and click **Next**.
3. In the Product Activation Wizard, select the **Serial Numbers** to activate, enter **Email Contact Information**, and select **Manually via E-mail** as the activation method.
4. Follow the steps to create a request file and send it to activation@solidworks.com. When you receive the license response file, click **Open** to load the file.
5. Click **Finish** when activation completes.

Manually Activate an Upgraded SolidNetWork License Manager Server

After upgrading the SolidNetWork License Manager Server with the SOLIDWORKS Installation Manager, open the SolidNetWork License Manager Server.

To manually activate an upgraded SolidNetWork License Manager Server:

1. In the Server Administration tab, under **License Information**, click **Reactivate**.

   If you deactivated the licenses before uninstalling and upgrading the SolidNetWork License Manager Server, click **Modify** instead in the Server Administration tab. **Reactivate** is only available if there are activated licenses on the server.

   Continue with step 3.

2. In the Reactivate dialog box, select the installed serial numbers to view their associated licenses and click **Reactivate**.
3. Verify the **Firewall** and **Ports** in the SolidNetWork License Server Information dialog box and click **Next**.
4. In the Product Activation Wizard, select the **Serial Numbers** to activate, enter **Email Contact Information**, and select **Manually via E-mail** as the activation method.

   Manual activation is a simple process. The instructions in the Product Activation wizard will guide you through the procedure.

5. Click **Finish** when activation completes.

Installing SolidNetWork License Clients

When you install SOLIDWORKS Design on a computer, you specify a SolidNetWork License serial number. SOLIDWORKS Installation Manager identifies that computer as a SolidNetWork License Client and prompts you for the SolidNetWork License Manager computer name and number. It displays an error if the SolidNetWork License Manager Server is not installed, or if the name or port number is incorrect.

To install SOLIDWORKS Design with a SolidNetWork license:

1. Start SOLIDWORKS Installation Manager as described in **Download Installation Media**.
2. On the Welcome page, click **Install on this computer** if this is the first SOLIDWORKS Design 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** .

3. In the Add Server dialog box, enter 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`.

   SOLIDWORKS Installation Manager attempts 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.

4. Complete the SOLIDWORKS Design installation by following the SOLIDWORKS Installation Manager instructions.

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 a supported Windows operating system, 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

1. Click **Start** and open **Windows Security**.
2. Select **Firewall & network protection** > **Advanced Settings** and click **Yes** to confirm the changes.
3. In the **Windows Defender Firewall with Advanced Security** dialog box, select **Inbound Rules** > **New Rule**.
4. In **New Inbound Rule Wizard**, select the rule type **Port** and then click **Next**.
5. Type the port number in **Specific local ports:** (for example, *80, 25734, 443*) and then click **Next**.
6. Select **Name**. Type a **Name** (for example, *SNL-in-port*).
7. Click **Finish**.

Modifying SolidNetWork License Manager Settings for Windows Firewall

1. Start SolidNetWork License Manager by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server *version***.
2. On the Server Administration tab, under **License Server**, click **Stop**. Click **Yes** to confirm and **OK** to stop 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 the procedure above).
   c. Select **Options File** and click **Browse** to locate the file. Click **Edit** to change any of its parameters.

The options file allows the license administrator to control the various operating parameters of FlexNet Licensing.

Recommendation: Locate the options file in the SolidNetWork License Manager installation directory (*SolidNetWork_License_Manager_install_dir*\Licenses\sw_d.opt). This file must be in the same directory as the sw_d.lic file.

For more information about the options file, see the FlexNet Publisher *License Administration Guide* in the SolidNetWork License Manager installation directory:

```
C:\Program Files\SOLIDWORKS SolidNetWork License
Manager\Docs\flexuser\fnp_LicAdmin.pdf
```

Reactivating a SolidNetWork License

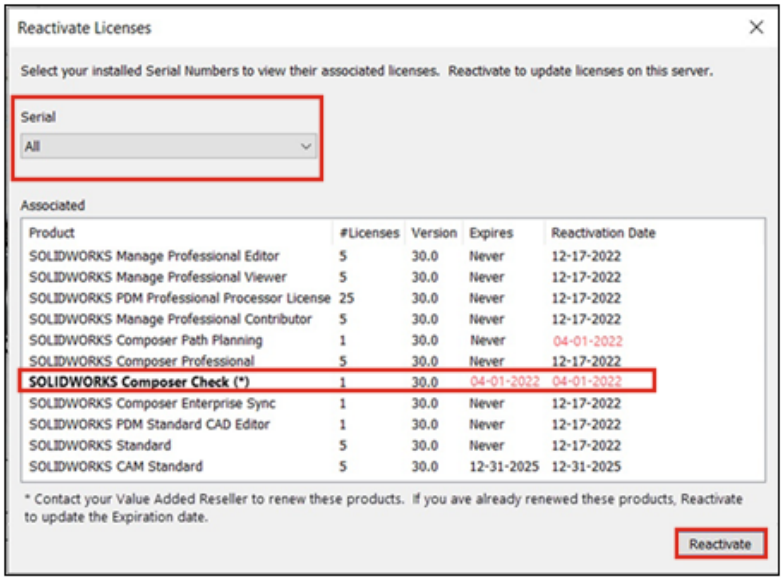
Reactivation applies to SolidNetWork License products after an organization upgrades the SolidNetwork License Manager Server to version 2022 or later. Even if the installed products are from an earlier release, licenses starting from version 2022 on the server require reactivation once a year.

Within 30 days of the reactivation date, SOLIDWORKS products on the network notify users that licenses will expire. Administrators must be aware of the expiration to reactivate licenses in time.

To reactivate a SolidNetWork license:

1. Start the SolidNetWork License Manager Server by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server** *version*.
2. In the Server Administration tab, under **License Information**, click **Reactivate**.
3. In **Serial**, select **All** for all licenses or enter a license number to search for a specific license.

   The Reactivate Licences dialog box highlight in red all license expiration and reactivation dates within 30 days. Asterisks mark licenses that require renewal.



4. Click **Reactivate**.

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:

- 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

This procedure shows how to install the SolidNetWork License Manager on the first computer in the triad.

1. On the first triad license manager computer, install the SolidNetWork License Manager using the SOLIDWORKS Installation Manager:
   a) Start SOLIDWORKS Installation Manager as described in **Download Installation Media**.
   b) For type of installation, select **Install server components** and click **Install SolidNetWork License Manager (distributed licensing) on this computer**.
   c) On the Summary page, review the SolidNetWork License Manager **Installation Location**, **Download Options**, and click **Install Now** to complete the SOLIDWORKS Installation Manager process.

2. Using the SolidNetWork License Manager, configure the initial triad license manager computer:
   a) Start the SolidNetWork License Manager. Click **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server *version***.

      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) In the Server Administration tab, under **Advanced Options**, click **Options**.
   d) Under SolidNetWork Server License Mode, click **Use a License File** and click **OK**.
   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, and port number for the initial triad license manager computer are correct, and click **Next**.

      Check that the Network Interface Card (NIC) address matches the information in the license file received from SOLIDWORKS, and that the port number for the triad license manager computer is correct. The NIC address is also referred to as the MAC (Media Access Control Address) address. Then click **Next**.

      To find the NIC address, open a Command Prompt window and type:

      ```
      ipconfig /all
      ```

      Also, if your computer uses a firewall, you might have to add specifications on this screen. For more information, see **Using SolidNetWork Licenses with Firewalls**.

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

      This ensures that if one triad 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. For this computer, you use a license file that you received from a Value Added Reseller or Dassault Systèmes' SOLIDWORKS Customer Support.

   j) Click **Finish**.
   k) Click **OK** to exit and **Finish** to close the SolidNetWork License Manager.

Installing the Triad License Manager on Subsequent Computers

Follow these instructions for installing the SolidNetWork License Manager software on the second and third computers in the triad. This procedure is almost the same as installing the software on the initial computer, except for steps 2(f) and 2(i).

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 the SOLIDWORKS Installation Manager:
   a) Start SOLIDWORKS Installation Manager as described in **Download Installation Media**.
   b) For the type of installation, select **Install server components** and click **Install SolidNetWork License Manager (distributed licensing) on this computer**.
   c) On the Summary page, verify the SolidNetWork License Manager **Installation Location**, **Download Options**, and click **Install Now** to complete the SOLIDWORKS Installation Manager process.

2. Using the SolidNetWork License Manager, configure the triad license manager computer:
   a) Start the SolidNetWork License Manager. Click **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server *version***.

      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 Triad Server Information screen, enter the computer name and host ID values. Then click **Next**.

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

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

      The licenses appear in the SolidNetWork License File wizard.

      The licenses file is generated by the first triad license manager computer that you configured. Typically, the licenses file is located in `\licenses\sw_d.lic`. For example: `C:\Program Files\SOLIDWORKS SolidNetWork License Manager\licenses\sw_d.lic`

   a) Click **Finish**.

      A SolidNetWork License File Wizard warning appears.

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

Repeat this process for the third triad license manager computer.

Starting the SolidNetWork License Manager

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

In Windows, click **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server *version***.

You can use the same path from the Windows **Start** menu to open the **SolidNetWork License Manager Client *version***.

Upgrading the SolidNetWork License Manager

For major releases of SOLIDWORKS Design, upgrading the license manager consists of uninstalling the old license manager, installing a new one, and reactivating 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 SolidNetWork License Manager by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server *version***.
2. On the License Usage tab, verify that no client computers on the network are running SOLIDWORKS Design.
3. Uninstall the old license manager. For details, see **Uninstalling the SolidNetWork License Manager** on page 99.
4. Install the new license manager and reactivate the SolidNetWork license on the license manager computer. For details, see **Installing the SolidNetWork License Manager Server** on page 82.

If you are upgrading the SolidNetWork License Manager on a computer that also runs SOLIDWORKS Design, upgrade SolidNetWork License Managers and clients before installing any new SOLIDWORKS Design 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. For details, see **Using the Same Computer for the License Manager and Administrative Image Installations** on page 99.

Moving a SolidNetWork License

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

> If you were unable to deactivate the license to Dassault Systèmes SOLIDWORKS Corporation (because of hard disk drive failure, for example), reactivation is still successful in most cases. If not, contact your Value Added Reseller for help.

To move a SolidNetWork license:

1. Start the SolidNetWork License Manager Server by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server** *version*.
2. In the Server Administration tab, under **License Information**, click **Modify**. The SOLIDWORKS Product Activation wizard starts.
3. Under **What would you like to do?**, select **Move your product license(s) to a new/upgraded computer** and then click **Next**.
4. Under **Serial #:** list, select the serial numbers or click **Select All** to activate all your SOLIDWORKS Design serial numbers.
   a) Select the **Automatically over the internet (recommended)** option.
   b) Under **Contact Information (Required):**, specify an email address.

5. The Deactivation Succeeded message box appears.

   The deactivation of the product license completes.

6. Click **Finish**.

Deleting SolidNetWork License Files

You can delete expired SolidNetWork license files before upgrading to new licenses.

1. Start the SolidNetWork License Manager by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server**.
2. On the Server Administration tab, under **License Information**, click **Modify**, and then click **Next**.
3. On the Activate/Deactivate Your SOLIDWORKS Product page, select the expired SolidNetWork license files, and click **Remove**.
4. Click **Finish** and **OK** to exit.

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 current user and machine on which they were borrowed; they cannot be moved.

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.

Borrowing and returning a license does not work if the SOLIDWORKS Design license is in use.

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

Always start the SolidNetWork License Manager from the **Start** menu. You must connect to the SolidNetWork License Manager to borrow the license.

These steps might vary depending on your system configuration. Consult Microsoft Windows documentation for additional instructions and options.

1. Start SolidNetWork License Manager on the client machine by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Client *version***.

2. On the License Borrowing tab:
   a) Select a date in **Borrow until**.
   b) Under **Available product licenses**, select the products to borrow and click **Borrow**.
   c) In the dialog box prompt, **Select Product**, select the product from the list, and click **OK** to confirm.
   d) At the bottom of the dialog box, click **Refresh**.

      The **Borrow until** column indicates when the licenses are due back.

3. Click **OK**.

In the License Usage tab, you can also view the **Total licenses**, **Free licenses**, and **Licenses used by** the user, and **Borrowed until** dates.

Select **Only use borrowed licenses (faster startup)** if you are working remotely and SOLIDWORKS Design is slow to open.

This option blocks queries to your license servers, which can be slow in some environments. The applications can start faster by omitting the queries, but only use the licenses you borrowed.

Restart your applications for this option to take effect.

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 on the client machine by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Client *version***.
2. On the License Borrowing tab:
   a) Under Borrowed product licenses, select the products to return, and click **Return**.
   b) Click **Refresh**. The **Borrowed Until** column becomes blank, indicating that the licenses are returned.
3. Click **OK**.

In the License Usage tab, you can also view the number of **Total licenses**.

> You do not need 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.

Managing Access to SOLIDWORKS Design Versions

You can restrict users from using specific versions of SOLIDWORKS Design in a SolidNetWork environment. This capability allows you to manage the versions of SOLIDWORKS Design used by different individuals, projects, and business units.

To manage access using a FlexNet options file:

1. In the SolidNetWork License Manager installation directory, create a text file:
   SolidNetWork_License_Manager_install_dir\Licenses\sw_d.opt

2. In the license file, add SOLIDWORKS Design version control using the following syntax next to SOLIDWORKS:

:SWVERSION=*xx*	where *xx* is the version of the SOLIDWORKS Design release you want to manage

3. To manage multiple versions of SOLIDWORKS Design, add a line for each version you want to manage.

   Not all FlexNet options are supported. The supported options for version filters are:

`INCLUDE`	Allows only the specifed user/group to start a specific version.
`INCLUDE_BORROW`	Allows only the specifed user/group to start a specific version.
`EXCLUDE`	Blocks the specified user/group from starting a specific version (overrides `INCLUDE`).
`EXCLUDE_BORROW`	Blocks the specified user/group from borrowing a specific version (overrides `INCLUDE_BORROW`).

   Examples:

1	`INCLUDE solidworks:SWVERSION=`*`xx`* `USER jane`
	`INCLUDE_BORROW solidworks:SWVERSION=`*`xx`* `USER jane`
	Only user `jane` can run or borrow the specified version of SOLIDWORKS Design, Other users can access other versions as permitted.
2	`GROUP chicago USER john robert eva`
	`EXCLUDE solidworks:SWVERSION=`*`xx`* `GROUP chicago`
	`EXCLUDE solidworks USER don`
	The group `chicago` is restricted from using a specific version. User `don` is restricted from using any version of SOLIDWORKS Design.
3	`EXCLUDE swinspection_std:SWVERSION=`*`xx`* `HOST machine1`
	Prevents users on `machine1` from launching the specified version of SOLIDWORKS Inspection.

   For information about FlexNet, see the FlexNet Publisher *License Administration Guide* in the SolidNetWork License Manager installation directory (`\Docs\flexuser\fnp_LicAdmin.pdf`).

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.

Using a Temporary License

You can use a temporary "any" license when the SolidNetWork license is unavailable.

By default, the SolidNetWork License Manager can only activate one server at a time. To work in a test environment, you can acquire a temporary license file through Technical Support and load an "any" license into the SolidNetWork License Manager on another server. If working with an offline server, you can use the same instructions to load an activation-exempt license.

> By using an "any" license file, you agree to implement a license on receipt of a SolidNetWork license from Dassault Systèmes SOLIDWORKS Corporation. Compliance with license restrictions still is required.

To use an "any" license:

1. Start the SolidNetWork License Manager by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server *version***.
2. If the software prompts you to activate, click **No**.
3. In the Server Administration tab, click **Advanced Options** > **Options**.
4. Under SolidNetWork Server License Mode, click **Use a License File** and **OK**.
5. In the Server Administration tab, under **License Information**, click **Modify**.

   The SolidNetWork License File Wizard appears.
6. Under **Server Type**, select **Single** and click **Next**.
7. Select **Dongle ID** and enter **Any** or the **NIC address** for server information. Then click **Next**.

   Click **Cancel** when you see this message: `Could not read the dongle ID number.`

   > If your computer uses a firewall, you might have to make additional specifications on this screen. For more information, see **Using SolidNetWork Licenses with Firewalls** on page 85.
8. Under Licenses, click **Read** to read the license file.

9. In the Read dialog box, browse to the `SNL_ANY.txt` or `SNL_(NIC Card Number).txt` file provided by Technical Support and click **Open**. Then click **OK**.

   The SolidNetWork File License Wizard lists the licenses from the text file.

   > Click **Remove** to delete temporary license entries that have expired in the list.

10. Click **Finish** to close the SolidNetWork License File Wizard.
11. Activate an "any" license:
    a) In the Server Administration tab, click **Advanced Options** > **Options**.
    b) Under SolidNetWork Server License Mode, select **Use a License File** and click **OK**.
    c) In the Server Administration tab, click **Modify** > **Activate/Deactivate your product licenses**.
    d) Select the **Serial Number**, enter **Contact Information**, and select the activation method. Follow the instructions if you choose manual activation.
    e) Click **Finish** to confirm activation.

Using SolidNetWork Licenses with Add-in Products

You can use the SolidNetWork License Manager to distribute licenses for SOLIDWORKS add-in products.

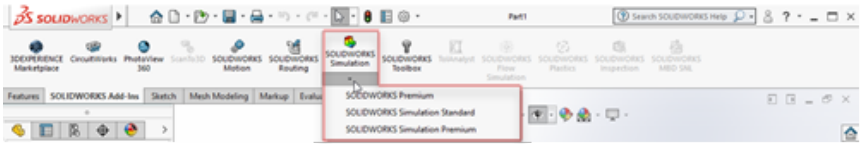
- An equal number of SOLIDWORKS Design licenses and SOLIDWORKS add-in licenses are not required.
- To check out a license for an add-in product, you first must check out a license for SOLIDWORKS Design.
- To borrow an add-in product license, you must borrow both the SOLIDWORKS Design license and the SOLIDWORKS Design license that includes the add-in. A SOLIDWORKS Design Standard license will be borrowed by default, but to enable an add-in, you will have to borrow a SOLIDWORKS Design Professional, SOLIDWORKS Design Premium, or SOLIDWORKS Design Ultimate product license.
- To release an add-in product license:
  - In SOLIDWORKS Design, click **Tools** > **Add-ins** and clear the add-in.
  - Exit SOLIDWORKS Design.

Selecting a SOLIDWORKS Simulation License with a SolidNetWork License

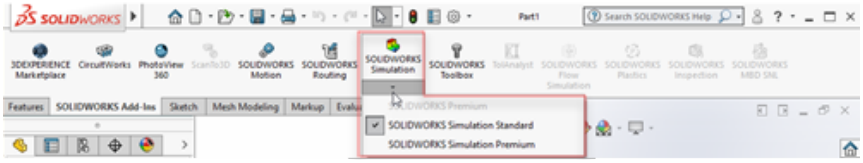
SolidNetWork license users can control which of the SOLIDWORKS® Simulation licenses in their license pool to use when they add in SOLIDWORKS Simulation. They can also

upgrade the SOLIDWORKS Simulation license without unloading the add-in in case they need a higher license type for more advanced studies.

1. To use a specific SOLIDWORKS Simulation license:
   a) In SOLIDWORKS Design, click **Tools** > **Add-Ins** and in the Add-in dialog box, select **SOLIDWORKS Simulation**.
   b) On **SOLIDWORKS Simulation** (CommandManager), click the **Down** arrow to display a list of licenses on your server that can support Simulation.



   c) Select the appropriate SOLIDWORKS Simulation license.
2. To check or change your current SOLIDWORKS Simulation license, click the **Down** arrow on **SOLIDWORKS Simulation** (CommandManager).



   The license that is in-use displays a check mark. If more advanced license types are available, you can upgrade.

   > You cannot downgrade a SOLIDWORKS Simulation license. Disable the SOLIDWORKS Simulation add-in and add the license back in at a lower level.

3. Optional: To disable the add-in, click SOLIDWORKS Simulation (CommandManager) or clear it from **Tools** > **Add-Ins**.

The following cases still use license order:

- When SOLIDWORKS Simulation starts running.
- If SOLIDWORKS Simulation is added in **Tools** > **Add-Ins**.
- If the you click SOLIDWORKS Simulation without specifying a license from the pull-down list.

Loading Flow Simulation Modules

As a SolidNetWork license customer, you can control how to load the HVAC and Electronics Cooling add-in modules for Flow Simulation using two methods.

The first method lets you specify the default behavior for the add-ins when you load Flow Simulation. The option **Automatically load Flow Simulation add-in modules if available** is in the **SolidNetWork License Manager Client** > **License Order** tab. Clear this option if you do not have a need for the Flow Simulation add-ins. The HVAC and Electronics Cooling add-in modules do not load when Flow Simulation is added in with existing methods.

The second method lets you override default behavior and include the HVAC and Electronics Cooling add-in modules when you load Flow Simulation. Click the Down arrow on

SOLIDWORKS Flow Simulation (SOLIDWORKS® Add-ins toolbar) or **SOLIDWORKS Flow Simulation** with one or both of the add-modules to load.

Without these methods, the licenses for the add-in modules are consumed whether or not you need them and may be denied to other users on your network.

Inactivity Timeout for SOLIDWORKS Simulation, SOLIDWORKS Flow Simulation, and SOLIDWORKS Plastics

When you run SOLIDWORKS Simulation, Plastics, or Flow Simulation studies, the network licenses remain active and do not time out. SOLIDWORKS Design holds the licenses during the calculation process, which is considered an activity.

Inactivity periods, defined by a TIMEOUT option, only take effect after the studies finish calculating.

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 Design.

Client computers can obtain licenses from the license manager and then install SOLIDWORKS Design 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.

For details, see:

- **Installing the SolidNetWork License Manager Server** on page 82
- **Deploying to Multiple Clients** on page 16

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 SolidNetwork License Manager by clicking **Start** > **SOLIDWORKS Tools *version*** > **SolidNetWork License Manager Server** *version*.
2. Verify that no client computers on the network are running SOLIDWORKS Design.

   The License Usage tab shows clients running SOLIDWORKS Design.
3. If you want to move the SolidNetWork license to another computer, deactivate the license to that computer before removing the SolidNetWork License Manager.

   For information about moving SolidNetWork licenses, see **Moving a SolidNetWork License**.
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** > **Programs and Features**.

7. Select **SOLIDWORKS SolidNetWork License Manager**, click **Uninstall**, and confirm the uninstall.
8. In File Explorer, delete the `SOLIDWORKS SolidNetWork License Manager` folder.

   (Typically, this folder is located under `Program Files (x86)` in the SOLIDWORKS installation directory.)

You now are ready to install a new version of the license manager. For details, see **Installing the SolidNetWork License Manager Server**.

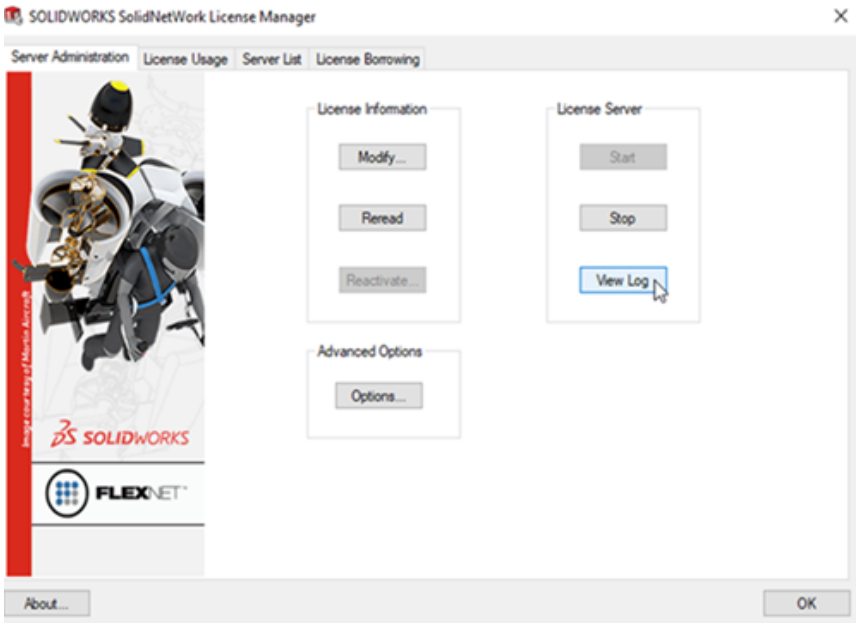
> If the SolidNetWork License Manager computer has a hardware dongle attached, remove the hardware dongle before installing the new version of SolidNetWork License Manager.

FlexNet Publisher License Management

SolidNetWork Licensing is based on FlexNet Publisher® license management.

For information about FlexNet Publisher license management, see the FlexNet Publisher *License Administration Guide* in the SolidNetWork License Manager installation directory: `C:\Program Files\SOLIDWORKS SolidNetWork License Manager\Docs\flexuser\fnp_LicAdmin.pdf`

For the latest information about FlexNet Publisher license management, contact Flexera® Software.



You can check the FlexNet version by reviewing the log on the License Server. Open the SolidNetwork License Manager Server and click **Server Administration** > **View Log** . The log lists the FlexNet Licensing version at the start in parenthesis.

8

Troubleshooting

This chapter includes the following topics:

- **Network License Usage**
- **SOLIDWORKS Installation Manager Log Files**
- **SOLIDWORKS Installation Manager Download Folder**
- **Troubleshooting SolidNetWork Licenses**
- **Troubleshooting Upgrades**
- **Validating Graphics Cards When Installing or Upgrading**

Consult these topics to assist you in troubleshooting installation problems.

Network License Usage

This dialog box opens when the server is out of licenses for a product.

Click the arrow in the **Product** box to display all the product choices. The choices include SOLIDWORKS Design and all purchasable add-ins.

Total licenses is the total number of licenses that you have purchased.

Free licenses is the number of licenses that are not in use.

The **Licenses used by** box gives you information about each person using a license.

SOLIDWORKS Installation Manager Log Files

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

These log files are located with the other SOLIDWORKS Design log files.

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

If you run the Installation Manager from a system account, the logs are located in this folder:

- `C:\Windows\SysWOW64\config\systemprofile\AppData\Roaming\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 SOLIDWORKS Installation Manager progress.

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

SOLIDWORKS Installation Manager Download Folder

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.

Question	Answer
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.)
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`.
Can I select the same download folder for each version I install?	Download folders are version-specific. SOLIDWORKS Installation Manager automatically creates a version-specific subfolder within the specified folder.
I downloaded some products manually from the Dassault Systèmes SOLIDWORKS Corporation Web site. Can the Installation Manager install these?	Files downloaded manually can be used by SOLIDWORKS Installation Manager. However, it is better to run SOLIDWORKS Installation Manager, selecting the option to download files. SOLIDWORKS Installation Manager shows which files you need based on what you are installing, providing links to download each file. You can use this option when the download page does not work (most likely because of downloading restrictions enforced by a proxy server). If a dialog box asks whether you want to save or run files during a download or installation, always specify **Save**.

Question	Answer
Can the files I download be shared by other users?	Yes. To do so, download to a network share. Each client can then run SOLIDWORKS Installation Manager from the share to install. No further downloads are 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 lets you select products regardless of what may be installed on your system. However, you cannot use this option to download patches or to create administrative images.
How do I know when a new version of SOLIDWORKS products becomes available?	You are informed when a new version is available in the following cases: • If you run Check for Updates (**Start** > **SOLIDWORKS Installation Manager** > **Check For Updates**) • If you have enabled SOLIDWORKS Background Downloader • If you select an installed version of SOLIDWORKS Design 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. SOLIDWORKS Installation Manager will download and run the newer version of itself, and not any installation files. After the newer version of SOLIDWORKS Installation Manager runs, you may select a set of installation actions. SOLIDWORKS Installation Manager determines which installation files are needed (and not downloaded previously) and offers to download those files.
Why does SOLIDWORKS Installation Manager not let me select a different service pack of a product to install?	Each version of SOLIDWORKS Installation Manager is designed to install only a specific version of SOLIDWORKS products. If you want to install a different service pack, you need to run that version of SOLIDWORKS Installation Manager. To find the latest version available, see the preceding item.

Question	Answer
Why is SOLIDWORKS Installation Manager prevented from downloading files?	If proxy server restrictions prevent applications from downloading files, SOLIDWORKS Installation Manager cannot download files automatically. You can download files manually and then install them using SOLIDWORKS Installation Manager. Select the option in SOLIDWORKS Installation Manager to download individual files (for example, **manual download method** 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.
Not all of my SOLIDWORKS products are updated to the latest service pack. Why does Check For Updates tell me I have the latest version?	**Check For Updates** only tells you whether you have installed (or are running) the latest version of SOLIDWORKS Installation Manager. If you want to update all products to the latest version that is installed in **Control Panel** > **Programs** > **Programs and Features**, select the latest installed version of SOLIDWORKS Design 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. SOLIDWORKS Installation Manager looks for the files `swdata*.id` to determine the version. If these files are not present, it fails to recognize any of the installation files that you copied and requires 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.

- Check that the client machine can accurately resolve and route to the server, given the server name.

For details about port settings for SolidNetWork license managers and clients, see **Using SolidNetWork Licenses with Firewalls** on page 85.

Troubleshooting Upgrades

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

Modifying or Repairing Source That Has Moved

When SOLIDWORKS Design is installed, SOLIDWORKS 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 modifying or repairing.

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

Validating Graphics Cards When Installing or Upgrading

When you finish installing or upgrading SOLIDWORKS Design on your machine, run SOLIDWORKS Rx to ensure that your machine is running a supported graphics card and graphics card driver.

This graphics card validation process is not available for users who install SOLIDWORKS Design from the **3D**EXPERIENCE platform.

To validate graphics cards when installing or upgrading:

1. When the Installation Manager completes installing and displays the **Finish** page, under **Check Your System**, click **Run System Diagnostics**.

   SOLIDWORKS Rx starts and runs diagnostics.

2. SOLIDWORKS Rx confirms if the graphics card and graphics card driver are supported.

   If the graphics card is supported but the driver is not, a status message may provide an indication for a driver update.

Every tenth time you start SOLIDWORKS Design, a Check Your System message appears and recommends that you run system diagnostics again. To suppress these notifications, right-click **SOLIDWORKS Resource Monitor** in the Windows notification area and click **Dismiss Graphics Notifications**.

You can validate a graphics card and driver at any time by right-clicking the SOLIDWORKS Resource Monitor in the Windows notification area. The menu also lets you:

- Access **Help** for the SOLIDWORKS Resource Monitor
- **Run System Diagnostics**
- **Dismiss Graphics Notifications**

- **Dismiss Subscription Notifications**
- **View Certified Graphics Drivers**

9

SOLIDWORKS Installation Manager Application Help

This chapter includes the following topics:

- **SOLIDWORKS Background Downloader**
- **Specifying Options for the Installation Manager**
- **Check for Updates Results**
- **Administrative Image Location**
- **Download Options**
- **Download Options for SOLIDWORKS Installation Manager Files**
- **Download Product Installation Files**
- **Download Progress**
- **Download Results**
- **Installation Errors/Installation Cancelled**
- **Installation Is Complete**
- **Installation Location**
- **Upgrade Options**
- **Installation Progress**
- **Installation Type**
- **Installing and Managing Multiple Computers**
- **Manual Download**
- **Newer Version Available**
- **No New Versions Available**
- **Processes Detected**
- **Product Selection**
- **Products to Download**
- **Products to Repair**
- **Serial Numbers**
- **Server Installation**
- **Summary**
- **System Check Warnings**
- **Hole Wizard/Toolbox Options**
- **Uninstall Products**

SOLIDWORKS Installation Manager is a stand-alone application that installs and updates your SOLIDWORKS products.

SOLIDWORKS Installation Manager is the primary tool for downloading, installing, modifying, updating, repairing, and removing SOLIDWORKS Design.

You can run SOLIDWORKS 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 Design has a version of SOLIDWORKS Installation Manager specific to that SOLIDWORKS Design version.

SOLIDWORKS Background Downloader

SOLIDWORKS Background Downloader performs automated file downloads (for example, when a new Service Pack release is found) using background processes, enabling you to continue other work while downloading. When the download completes, you can decide whether you want to install now, postpone the download, or remove the download.

Background Downloader is enabled by default when you install SOLIDWORKS Design.

> When Installation Manager is running, whether for an installation or download operation, Background Downloader is paused. For example, it will pause any running downloads, not allow any new downloads to start, and will not allow you to change the download folder specification.

You can disable the Background Downloader during installation. On the Summary page, under Download Options, clear **Use the Background Downloader for future service packs** option.

When enabled, this icon appears in the Windows notification area in the toolbar:



Clicking this icon enables these operations:

- **Help with Background Downloader**
- **Pause Download**
- **Resume Download**
- **Cancel Download**
- **Installation Options**
- **Change Download Settings**
- **Check for Updates**
- **Automatically Check for Updates**
- **Automatically Download Updates**
- **Disable Background Downloader**

Help with Background Downloader

Open this help topic for Background Downloader in the *SOLIDWORKS Installation Guide*.

Pause/Resume Download

If a download is running in the background, click **Pause Download** to pause that download, with the ability to resume later by clicking **Resume Download**.

Cancel Download

If a download is running, stop that download and delete any files Background Downloader has already downloaded.

> You cannot resume a canceled download.

Installation Options

When a download completes, you can choose from the following options:

- **Install now**: Install the downloaded files.
- **Remind me later**: Do not install the downloaded files, but set up a message to remind you later.
- **Delete this download**: Do not install the downloaded files, and delete them. (This is useful if you have decided you no longer want to install the downloaded files.)

Change Download Settings

Change the folder location where the Background Downloader downloads files.

> This option is not available if a download is either running or paused.

> You can also specify the **Number of days between update checks**.

Check for Updates

Run the **Check for Updates** feature of SOLIDWORKS Installation Manager.

Automatically Check for Updates

Periodically run **Check for Updates**, notifying you when updates are available for download.

Automatically Download Updates

Periodically run **Check for Updates**, downloading updates as they become available.

Disabling and Re-Enabling SOLIDWORKS Background Downloader

You can disable Background Downloader by clicking the icon in the Windows system tray and select **Disable Background Downloader**.

To re-enable Background Downloader, click **Start** > **SOLIDWORKS Installation Manager** > **SOLIDWORKS Background Downloader**.

Specifying Options for the Installation Manager

These options include settings for logging, internet access, and the language used in the Installation Manager.

To specify Installation Manager options before installing the software:

1. In the Installation Manager, right-click the SOLIDWORKS icon on the title bar and select:

Option	Description
Installation Logging Levels	Specifies the logging level as **High**, **Medium**, or **Low**. Higher levels collect more data for troubleshooting, but can slow down the installation. **Low** (Fastest) is the default, and provides the best performance and stability. **Create Windows Installer Logs (Slow)** stores logs in the same folder as other installation logs.
Administrative Image Options	Select **Use short file names (8.3 format)** or **Use full folder and file names**.
Gather Installation Logs	Specifies where the installation logs are stored. The Installation Manager stores these files in `IM_Logs.zip`.
Disable Internet Access	
Language	Specifies the language used in the Installation Manager. The default is English.

2. Click **Close**.

Check for Updates Results

Check for Updates looks for more recent versions of SOLIDWORKS Design.

If you have administrator privileges on the machine, you can run **Check for Updates** manually:

- In Windows, click **Start** > **SOLIDWORKS Installation Manager** > **Check for Updates**
- In SOLIDWORKS Design, click ⓘ > **Check for Updates**

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

New Versions Available

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

> These downloads are only for use by SOLIDWORKS 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 SOLIDWORKS Installation Manager.

No New Downloads Available/Unable To Check for New Versions

SOLIDWORKS 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.

Administrative Image Location

Specify the installation location of the administrative image.

When creating an administrative image, these options apply:

New image location

This is the directory location in which SOLIDWORKS Installation Manager will create the new image.

> This location cannot contain an existing administrative image. The administrative image must be in a folder that can be shared for client deployment.

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 111.

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 102.

What Do You Want To Do?

Choose one of the following:

Download and install/Download and create image

After the download completes, SOLIDWORKS Installation Manager runs an installation using the downloaded files.

Download only. You can download once and then share downloaded files with multiple computers.

SOLIDWORKS 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.

SOLIDWORKS 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, SOLIDWORKS 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 SOLIDWORKS 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?

Conduct manual download

In some cases, a proxy server might prohibit automatic downloads from SOLIDWORKS Installation Manager.

By selecting this option, you can download the files manually through a Web page and then proceed within SOLIDWORKS Installation Manager to complete the installation operations. This Web page is

customized by SOLIDWORKS Installation Manager to download only the files required to complete the installation.

Using SOLIDWORKS Background Downloader

You can use SOLIDWORKS Background Downloader to complete this download operation or to download future SOLIDWORKS Service Pack releases as they become available.

Solidworks Background Downloader can automatically detect and download new Service Pack updates in the background, notifying you when downloading is complete and the files are ready for installation. For details, see **SOLIDWORKS Background Downloader** on page 108.

Download Options for SOLIDWORKS Installation Manager Files

You have chosen to download a later version of 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 102.

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 SOLIDWORKS 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.

Download Product Installation Files

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

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

Canceling 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/Installation Cancelled** on page 116.

Download Directory

Browse to the destination for the downloaded files.

The download size and the required space are 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 Folder** on page 102.

Download Progress

SOLIDWORKS 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 102.

Download Results

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

Download Problem/Download Cancelled

If you see Download Problem, SOLIDWORKS 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 SOLIDWORKS Installation Manager. By selecting this option, you can download the files manually through a Web page and then proceed within SOLIDWORKS Installation Manager to complete the installation operations. This Web page is customized by SOLIDWORKS 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 SOLIDWORKS Installation Manager.

Do not attempt to install any of these products manually. Always use SOLIDWORKS 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.

SOLIDWORKS Installation Manager first installs any prerequisites that are not installed on the computer, such as .NET framework.

Recovering from a Failed or Canceled Installation

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

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

   If the software prompts 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
   - PDMworks.exe
   - pdmwVault.exe
   - SWViewer.exe
   - swlmwiz.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. For details, see **Repairing an Installation**.
6. In Windows, click **Start** > **Control Panel** > **Programs and Features**.
7. Select the version of SOLIDWORKS Design that failed.

   If there is no entry for the SOLIDWORKS Design 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 does not restore the previous version of any products you are upgrading.

Repairing an Installation

If you encounter problems with an installation of SOLIDWORKS Design, you can use the SOLIDWORKS Installation Manager to repair the installation.

To repair an installation, your computer must have all the source files from the previous installation and any applied service packs. These must be in the location where you originally installed them.

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

1. Open the Windows **Control Panel** and click **Programs** > **Programs and Features**.
2. Right-click the SOLIDWORKS Design version that you want to repair and click **Change**.
3. On the Welcome to SOLIDWORKS Installation Manager screen, click **Repair your installation** and click **Next**.
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**.

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 need to provide a zip file of SOLIDWORKS Installation Manager log files.

1. Locate the log files. For Windows:
   `C:\Users\`*`username`*`\AppData\Roaming\SOLIDWORKS\Installation Logs\`*`installed_version`*`\`.

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

2. Zip the entire contents of this folder, including subfolders.

Installation Is Complete

SOLIDWORKS Installation Manager has completed its processing.

One or more of these options may be displayed.

What's New

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

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

Join the SOLIDWORKS Customer Experience Improvement Program

Performance information is sent to SOLIDWORKS Corporation to help improve the product. The process is transparent and confidential. Select one of the following:

- **Yes, I want to join**
- **No, thank you**
- **Remind me later**

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

Administrative Image Creation Is Complete

When you finish creating the administrative image, you can click **Customize Image** to run the Administrative Image Option Editor to customize the image.

Click **Show me how to install this image on a client.** to show instructions on using the Option Editor.

For details about configuring an administrative image using the Option Editor, see **Administrative Image Option Editor** on page 20.

Click **Customize Image** to start the Administrative Image Option Editor to customize the option settings for this administrative image.

> You can start the Option Editor later by double-clicking `sldAdminOptionEditor.exe` in the `SOLIDWORKS Admin` folder.

Checking for Updates to SOLIDWORKS

If you have administrator privileges on the machine, you can use SOLIDWORKS Installation Manager to check for updates manually or to configure automatic periodic checking for updates.

1. Start SOLIDWORKS Installation Manager by doing one of the following:
   - In SOLIDWORKS Design, click **Help** > **Check for Updates**.
   - In Windows, click **Start** > **Programs** > **SOLIDWORKS Installation Manager** > **Check for Updates**.

   You can also click **Start** > **SOLIDWORKS Installation Manager** > **SOLIDWORKS Background Downloader**. In the Windows notification area, right-click **SOLIDWORKS Background Downloader** and click **Check for Updates**.

   SOLIDWORKS Installation Manager tells you whether any updates are available.
2. If updates are available, choose whether to download and install them.
3. To enable automatic periodic checks for updates, when SOLIDWORKS Installation Manager completes its processing select **Check for Updates every ___ days** and specify the frequency of checks.

SOLIDWORKS Customer Experience Improvement Program

When installing SOLIDWORKS Design, you can specify that performance information is sent to SOLIDWORKS Corporation to help improve the product.

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.

The process is transparent. 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 the **Customer Experience Improvement Program website**.

Installation Location

For installations except administrative image creation, these options apply:

Install selected products to	This is the directory location to which you are installing SOLIDWORKS Design. 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 Design 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 Design.

> This option appears only if the **Install Only** option is specified. For details, see **Download Options**.

Consult your system administrator before changing either SOLIDWORKS Design installation location.

Administrative Image Locations

When creating an administrative image, these options apply:

New image location

This is the directory location in which SOLIDWORKS 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**.

Existing image location

When upgrading an existing administrative image, SOLIDWORKS 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 2025\`), and then upgrade that installation to SOLIDWORKS Design 2026, you will have a SOLIDWORKS Design 2026 installation directory that is labeled `SOLIDWORKS 2025`.

Upgrade Options

One or more older major versions of SOLIDWORKS products are installed on this computer. You can create a new installation, which will coexist with the older version, or you can upgrade, which removes the older version.

Create a New Installation of SOLIDWORKS *version*.

If you choose to create a new installation, you can select which SOLIDWORKS products to install.

For each of the selected products, a new installation of this major version is created. The older major versions remain on the machine, except for SOLIDWORKS Electrical and SOLIDWORKS PDM Client.

If you clear a product, the new version does not install.

Create a new installation is useful if you want to access the older version of the SOLIDWORKS product 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*.

If you choose to upgrade, you can select which SOLIDWORKS products to upgrade. For each of the selected products, a new installation is created, and older versions are removed.

If you clear a product, the new version does not install, and the older version remains on the machine.

Considerations for SOLIDWORKS Electrical and SOLIDWORKS PDM Client

You can only install one version of SOLIDWORKS Electrical on a machine. If you are creating a new installation of a SOLIDWORKS product, and select SOLIDWORKS Electrical as one of the products to install, the old version of SOLIDWORKS Electrical is removed.

Back up the existing SOLIDWORKS Electrical environment before you install a new version. From within SOLIDWORKS Electrical Schematic, select **File Archive Environment**, or from within SOLIDWORKS Electrical 3D, select **SOLIDWORKS Electrical** > **Tools** > **Archive Environment**.

The major version of the SOLIDWORKS PDM Client must match the major version of the SOLIDWORKS PDM Server. If you do not plan to upgrade the SOLIDWORKS PDM Server, do not select SOLIDWORKS PDM Client for the upgrade.

To remove a product in Windows, use **Add/Remove Programs**. For details, see **Removing an Installation** on page 13.

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 Type

You can specify the installation operation you would like to run. For example, creating a new installation, modifying or repairing an existing installation, creating an administrative image, installing administrative server products, or downloading files so you can share and install them on other computers.

If the most recent release of SOLIDWORKS Design is already installed on your computer, you are given options to modify or repair it.

Individual

Install SOLIDWORKS products on this computer.

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

Modify the Individual Installation

Modify the existing SOLIDWORKS Design installation on this computer.

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

Modifications include:

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

  > If you already specified a SOLIDWORKS Design serial number during a prior installation, SOLIDWORKS Installation Manager might not prompt for it again. To change the serial number, you must select this option.

Repair the Individual Installation

Repair the existing SOLIDWORKS Design installation on this computer.

> This option appears only if this version is 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

Create or update an administrative image to deploy to multiple computers.

For details about installing SOLIDWORKS Design on multiple clients using administrative images, see **Using Administrative Images**.

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:
  - SOLIDWORKS 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.

Server Products

SOLIDWORKS server products include SOLIDWORKS Electrical, SOLIDWORKS PDM, and SolidNetWork License Manager.

For SOLIDWORKS Electrical and SOLIDWORKS PDM, you can choose to install the client and server tools on the same computer or on different computers.

For details about installing these servers, see corresponding topics in the Help.

Download and Share All Files

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

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

The content of the downloaded files is the same as the files that are available on the SOLIDWORKS installation DVDs. Selecting this option downloads the full set of installation files, regardless of what files might already be installed on your computer. You can then copy the files to a portable device or network directory, and use them to install SOLIDWORKS Design on other computers. You can also use this method to set up a server from which your users can perform installations.

This method is different from creating an administrative image.

Installing and Managing Multiple Computers

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

SOLIDWORKS Administrative Images

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

SOLIDWORKS PDM

SOLIDWORKS PDM is a document management product based on the same client/server architecture as SOLIDWORKS Enterprise PDM. It is available in two versions: Standard and Professional. SOLIDWORKS PDM uses a Microsoft SQL Server database to manage file vaults.

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

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**.

  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**.

  > **Check for Updates** is only available if you have administrative privileges on the machine.

  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 installs the original version from the DVD and then applies 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.

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

SOLIDWORKS 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

You can 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.
- To view the projected action for a particular component, click the component name (but not the check box). The intended installation action appears in the information field below the product listing.
- You can click the check box next to the product component to change the installation action for that component. (For example, if you are installing or removing components, selecting the check box specifies that the component is to be installed or removed and clearing the check box specifies that the component is not to be installed or removed.)
- 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 a component has optional installation subcomponents in it, a **+** icon appears next to the component. To see the available subcomponents for a component, click the **+** icon.
- If a component has been expanded to show optional subcomponents, a **-** icon appears next to the component. To collapse the subcomponent listing, click the **-** icon.

If you are applying a Service Pack, you cannot select features of SOLIDWORKS Design appearing under SOLIDWORKS Design (for example, FeatureWorks). This is because you cannot add or remove a feature when performing this type of upgrade.

The actions performed by SOLIDWORKS Installation Manager are controlled by a set of product coexistence rules. For details, see **Product Coexistence Rules** on page 14.

SOLIDWORKS Installation Manager selects these product components by default:

- If you are performing an upgrade or a Service Pack update, SOLIDWORKS Installation Manager selects the products currently installed.
- Otherwise, SOLIDWORKS 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 SOLIDWORKS 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 Design Professional, SOLIDWORKS Design Premium, or SOLIDWORKS Design Ultimate) listing to install or modify. SOLIDWORKS 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. 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.

Language Support

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

The **Languages** specification limits the languages that are installed for the SOLIDWORKS product only. Other product installations are not affected by this specification. For example, eDrawings® and SOLIDWORKS PDM install all languages.

Space Needed

SOLIDWORKS 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 the downloaded files (not including extra space required to unzip the files after the download completes). This figure is an estimate.

  All files required for installation are included in the total download size. If some files already were downloaded into the current download folder, they are not 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.
- If a component has optional installation sub-components in it, a **+** icon appears next to the component. To see the available sub-components for a component, click the **+** icon.
- If a component has been expanded to show optional sub-components, a **-** icon appears next to the component. To collapse the sub-component listing, click the **-** icon.
- Click the check box to the left of each component to select how it is handled by SOLIDWORKS Installation Manager.

SOLIDWORKS 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 select those components you do not need.

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

If SOLIDWORKS 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 Design Professional, SOLIDWORKS Design Premium, or SOLIDWORKS Design Ultimate) listing to install or modify. SOLIDWORKS 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

The Languages specification limits the languages that are installed for the SOLIDWORKS product only. Other product installations are not affected by this specification. For example, for users not connected to the **3D**EXPERIENCE platform, eDrawings® installs all languages.

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 version of SOLIDWORKS 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 Numbers

Serial numbers associate your computer with the SOLIDWORKS products that you have purchased.

If SOLIDWORKS products are already installed on your computer, their serial numbers display by default:

- Enter your SOLIDWORKS Design serial number if it does not display.
- If you have serial numbers for products not included with your SOLIDWORKS Design license, enter them in the appropriate fields.

If you have an internet connection, the serial numbers you provide 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 Design 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 that 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 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, SOLIDWORKS Installation Manager closes.

Server Installation Canceled or Failed

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

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

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

For More Information

For details about installing, configuring, and using the SolidNetWork License Manager, see **Administering Licenses Using a SolidNetWork License Manager** on page 79.

Summary

SOLIDWORKS Installation Manager has all the information needed to complete the installation operation.

To continue:	Click the *action* **Now** 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 SOLIDWORKS Installation Manager without changing installation components.
To revert to a previous page:	Click **Back** to return to a previous page in SOLIDWORKS 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

System Check reported issues that could impact how you continue with the installation. In most cases, you should be able to proceed with SOLIDWORKS Installation Manager.

The warnings you can encounter at this stage are listed below with suggestions about how to proceed.

Serial Number Validation Problems

`The Installation Manager was not able to check for new versions because it could not validate your serial number. If you are connected and entered a valid serial number, this could indicate a temporary problem on our server. Try again later.`

`Serial Number Validation: The Installation Manager could not validate your serial number to determine which products you are entitled to install. You will still be able to choose the products you want to install, but you will not be able to download any installation files. You will need to provide a complete set of download source files or a DVD from which to install.`

Resolution:

- Check the internet connection. The serial number check requires a valid internet connection.
- Check that the serial number you entered is valid for the products you are trying to install.
- Some installations require that you have a valid subscription in place at the time the version was released. For details, consult your SOLIDWORKS reseller.
- In most cases, you can proceed with the installation, provided you are installing from a DVD or a previously downloaded file set.

Newer Version Available

`Newer Version Available`

`New Versions Available`

`A newer version of this product is available for download.`

Resolution: When performing a Check for Updates operation, the installation might prompt you to install a later version that is available by downloading from SOLIDWORKS. You can continue this operation using the version you initially specified or you can download this newer version and continue the operation using that version instead.

Latest Version Available

`This is the latest version available. Click Next to continue.`

`No New Downloads Available`

Resolution: This message can appear when you are checking for updates during an installation or modification. It indicates that you are currently installing (or have installed) the latest version available.

Internet Connection Problems

`Internet Connection: The Installation Manager could not connect to the SOLIDWORKS server to determine which products you are entitled to install. You will still be able to choose the products you want to install, but you will not be able to download any installation files. You need to provide source files on DVD or a previously downloaded file set.`

`Unable to Check for New Versions`

`The Installation Manager was not able to check for new versions because it could not connect to the server.`

Resolution:

- Check the internet connection.
- You can proceed with the installation, provided you are installing from a DVD or a previously downloaded file set.

Hole Wizard/Toolbox Options

If you install SOLIDWORKS Toolbox, SOLIDWORKS Design includes a standard library of hole types and hardware.

For details about configuring, extending, and using SOLIDWORKS Toolbox, see *SOLIDWORKS Design Help*: *Toolbox*.

Before using SOLIDWORKS Toolbox, make sure to properly configure it 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 Design installation.

You can change the selected SOLIDWORKS Toolbox after installation. In SOLIDWORKS Design, click **Tools** > **Options**. On the System Options tab, click **Hole Wizard/Toolbox**.

The following sections describe all options that you can encounter on this screen for all configurations. The actual Toolbox options that you see depends on the pre-existing SOLIDWORKS Design and Toolbox configurations on this computer and SOLIDWORKS 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.

When you create a new version, in the Product Selection screen, expand **SOLIDWORKS** > **SOLIDWORKS Toolbox** and select which standards to support for the Toolbox. All standards are selected by default.

Any existing Toolboxes on this computer are not modified.

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 Design 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 Design major release version makes that Toolbox incompatible with computers running prior major release SOLIDWORKS Design versions.

- If the Toolbox is configured for this SOLIDWORKS Design 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 a SOLIDWORKS PDM environment, additional procedures are required. For details, see **Notes About Configuring SOLIDWORKS Toolbox** on page 135.

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 Design 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 Design version that is installed without impacting users that are using the older version of Toolbox.

If you are upgrading a Toolbox in a SOLIDWORKS PDM environment, additional procedures are required. For details, see **Notes About Configuring SOLIDWORKS Toolbox** on page 135.

Upgrade This Installed *previous-version* Toolbox

Upgrade this Toolbox to the newer version.

Upgrading a Toolbox to a new SOLIDWORKS Design major release version makes that Toolbox incompatible with computers running prior major release SOLIDWORKS Design versions.

This option appears if you are upgrading a previous SOLIDWORKS Design 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 a SOLIDWORKS PDM environment, additional procedures are required. For details, see **Notes About Configuring SOLIDWORKS Toolbox** on page 135.

Use the Installed *current-version* Toolbox

SOLIDWORKS Installation Manager has detected on this computer a Toolbox configured for this SOLIDWORKS major release version.

If you select this option, SOLIDWORKS Design will use this Toolbox. No upgrade occurs.

This option appears if a previous SOLIDWORKS Design 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 Design 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 Design major release version at the time the client installation occurs, SOLIDWORKS 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 that you are updating.

If the location contains a Toolbox configured for a previous SOLIDWORKS Design major release version at the time the client installation occurs, SOLIDWORKS 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 Design major release version at the time the client installation occurs, SOLIDWORKS 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 the SOLIDWORKS Installation Manager.

Finding More Information About SOLIDWORKS Toolbox

For details about configuring, extending, and using SOLIDWORKS Toolbox, click **Help** in SOLIDWORKS Design, and refer to the Toolbox section.

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 Design major release version makes that Toolbox incompatible with computers running prior major release SOLIDWORKS Design versions.

If other computers remain at that prior major release and continue to require a Toolbox configured for that release, make a copy of that Toolbox and upgrade that copy. The new SOLIDWORKS Design installation upgrades the copy of the Toolbox and previous SOLIDWORKS Design installations continue to use the old Toolbox.

Configuring Toolbox Access

After installation, when you access Toolbox components SOLIDWORKS Design checks the Toolbox configuration 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 is installed. For details, see **UNC Format** on page 136.

An administrator can create a password for Toolbox and set permissions and preferences for the workgroup. Restricting access to the configuration and setting common preferences ensures consistent Toolbox data.

Upgrading Toolbox in a SOLIDWORKS PDM Environment

When you upgrade SOLIDWORKS Design, Toolbox is upgraded if parts have been added. If the Toolbox root folder is in a SOLIDWORKS PDM vault, before you start the SOLIDWORKS Design upgrade, you must prepare the Toolbox folder.

On the first system to perform the upgrade, you must use **Get Latest Version** to download the Toolbox files to the local cache and check out the Toolbox database file. On other systems that are upgraded, the only requirement is to download the Toolbox files.

Before you upgrade:

- Ensure that there is a full recent backup of the vault database and archive.
- Make sure that the Toolbox parts are not checked out or in use by any checked out assemblies.

To upgrade Toolbox:

1. In File Explorer, log in to the vault as a user with full permissions (check out, check in, add, delete).
2. Right-click the Toolbox folder and click **Get Latest Version** to copy all Toolbox files and the toolbox database to the local cache.

   If your archive server is remote, this may take several minutes.

3. Check out the Toolbox database file (`SWBrowser.mdb`) or (`swbrowser.sldedb`) from `\`*`Toolbox_folder_name`*`\lang\`*`your_lang`*.
4. Run the SOLIDWORKS Installation Manager to perform the upgrade.
5. On the Summary screen, ensure that the **Toolbox Options** installation location is the location in the vault.

   If it is not, click **Change**, select **Reference or upgrade an existing Toolbox**, and browse to the Toolbox location in the vault.

6. When the upgrade completes, in File Explorer, check in the Toolbox folder to add any new or updated files to the vault so that other users can work with Toolbox.
7. For upgrades, navigate to your Toolbox folder in the vault and ensure that the following are present:

   - `\`*`Toolbox_folder_name`*`\Updates`
   - `\`*`Toolbox_folder_name`*`\ToolboxStandards.xml`
   - `\`*`Toolbox_folder_name`*`\Browser\ToolboxFiles.index`

   If they are not, contact your Value Added Reseller.

8. Upgrade the rest of the computers that use SOLIDWORKS PDM integration with Toolbox:

   a. In File Explorer, log in to the vault as a user with at least read permission to the Toolbox folder.
   b. Right-click the Toolbox folder and click **Get Latest Version** to copy the Toolbox files to your local cache.
   c. Run the SOLIDWORKS Installation Manager to upgrade.

      The installer detects that Toolbox is already upgraded.

For details about Toolbox configuration in SOLIDWORKS PDM, see *Configuring Toolbox* in the *SOLIDWORKS 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 to **Remove** or **Uninstall** programs from the Control Panel, this screen displays the SOLIDWORKS products (with version numbers) that can be uninstalled by this version of SOLIDWORKS Installation Manager. You can choose to perform a standard, custom or complete uninstall.

The standard uninstall removes Program Files and Folders of a SOLIDWORKS product. By default, all SOLIDWORKS products for the selected release are specified for a standard uninstall.

A custom uninstall removes one or more of the following items: Program Files and Folders, registry keys and data folders, such as the SOLIDWORKS Toolbox, and files and folders from the original download location.

A complete uninstall removes the install directories, registry keys and data folders.

To uninstall a product:

1. On the Uninstall screen, ensure that the product is selected.

   If you uninstall all products, the Solidworks Installation Manager is also uninstalled.

2. Clear the products you do not want to remove.

   If you retain any product, SOLIDWORKS Installation Manager is not removed.

3. Choose whether to perform a custom or complete uninstall.
   a. Click **Change** in **Advanced Options**.

      The Advanced Options screen appears. The Program Files and Folders option is checked.

   b. Select one or more options to perform a custom uninstall, or select all options to perform a complete uninstall.
   c. Click **Back to Summary**.
   d. On the Summary screen, expand **Advanced Options** to display the uninstall method and entries.

4. Click **Remove Items**.

When you click Remove Items, SOLIDWORKS Installation Manager starts removing products immediately.



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