

# How To Create A SOLIDWORKS 2023 Administrative Image

TITLE:	How To Create A SOLIDWORKS 2023 Administrative Image	
DATE:	September 2022	
SUBJECT:	Administrative images, multi-client Install	
ABSTRACT:	Guide to setting up, configuring and deploying administrative images (multi-client installs) for SOLIDWORKS 2023	



This guide is written for version 2023. If you are using an earlier version of SOLIDWORKS, please download our earlier installation documents.

Before loading any software, you should always log in with full administrative permissions and disable any anti-virus and anti-spyware programs. Be sure your operating system has been updated with Windows Updates recently to avoid any compatibility issues with the installer.

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. This allows for easier distribution of multiple installations and has a component to update all clients automatically when the image is upgraded to a newer service pack.

This is not a troubleshooting guide. If you have any technical issues with SOLIDWORKS, please contact [Hawk Ridge Systems technical support](https://www.hawkridgesys.com/technical-support).

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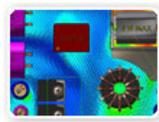
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## Before Installation

It is highly recommended you take a moment before installing SOLIDWORKS and check that you are ready to load your new software. Follow these steps to have better performance within SOLIDWORKS:

1. Check hardware compatibility.
  - Browse to [www.solidworks.com](http://www.solidworks.com) and click the support link at the top of the page and then click **System Requirements**. On this page, you will find links to [system requirements](#) as well as [graphics card compatibility](#).

### System Requirements



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

- You can also see hardware recommendations made by Hawk Ridge Systems by logging in to [support.hawkridgesys.com](http://support.hawkridgesys.com) and reviewing our hardware recommendations for specific computer models. We also have a [comprehensive hardware guide](#) that explains what to consider when hand-picking hardware components.
2. Check operating system compatibility.



- Here is a checklist of compatible operating systems for this version of SOLIDWORKS; you can find the latest version of this on the [system requirements page](#).

Operating Systems			
	SOLIDWORKS 2021 (EDU 2021-2022)	SOLIDWORKS 2022 (EDU 2022-2023)	SOLIDWORKS 2023 (EDU 2023-2024)
<b>Client Products</b> (e.g. SOLIDWORKS)			
Windows 11, 64-bit	✗	✓ (SW2022 SP2)	✓
Windows 10, 64-bit	✓	✓	✓
<b>Server Products</b> (e.g. SOLIDWORKS PDM, SNL Manager)			
Windows Server 2022	✗	✓ (SW2022 SP4)	✓
Windows Server 2019	✓	✓	✓
Windows Server 2016	✓	✓	✓ (End of Life: SW 2023)

- Be sure you are on the latest version of your operating system and your Windows is up to date before installing SOLIDWORKS.
3. Check hard drive for space and overhead.
    - In Windows Explorer go to Computer and check destination drive (by default the C drive), right-click this drive and choose Properties to confirm that you have 20GB of free storage.
    - Also be sure this drive has at least 15% free space. A full disk may cause out-of-memory crashes and cannot be defragmented to optimize performance.
  4. Confirm your community's working version.
    - SOLIDWORKS is not backwards compatible; meaning older versions cannot open files from newer versions. Be sure workmates, customers, or vendors are all on the same version (i.e. SOLIDWORKS 2023) you are loading.
  5. Back up your data.
    - Ask a system administrator or existing user to see if there are any shared libraries, such as Toolbox or a data management vault. Toolbox libraries and PDM vault information should all be saved to backup locations. Then you can migrate the original libraries and vault to the newer version.
    - If you have a shared location containing existing SOLIDWORKS files, it is recommended that you make backup copies of these files. Any templates or design libraries should also be saved to a backup folder.

# Creating an Administrative Image

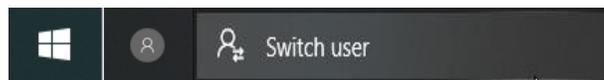
An administrative image allows configurable installations of SOLIDWORKS products from a single network location. You can install the administrative image from any computer on the network that can access the admin image location. There are a few important steps in preparing an administrative image:

1. Create a shared directory on a network computer that all clients can access.
2. Install an Administrative Image in this shared directory.
3. Once the image is created, use the Options Editor to customize it.

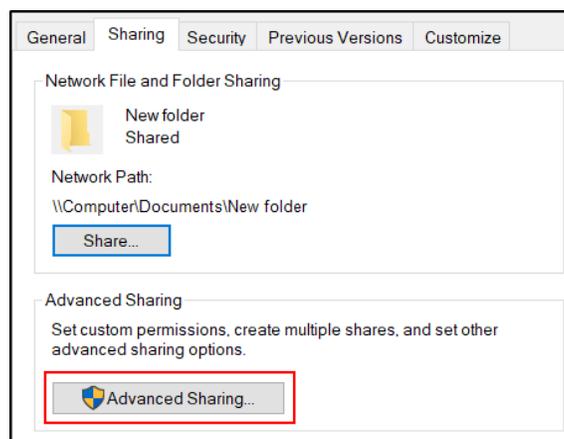
Other programs can be loaded before or after the SOLIDWORKS installation. The administrative image can also be used to administer certain SOLIDWORKS serial numbers or particular add-ins to be defined for individual users or groups throughout your company.

## Setting the Image Location

1. Make sure you are logged in as the administrator.

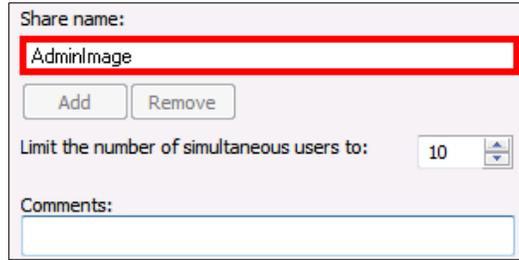


2. Temporarily disable all anti-virus and anti-spyware software on the computer on which you are creating the admin image.
3. In Windows Explorer, create a destination folder where you want the administrative image to reside. This should be a location that all client machines can access via the network.
4. Right-click this new folder and select **Properties** from the pop-up menu. Inside the properties dialog box switch to the **Sharing** tab and click **Advanced Sharing**.



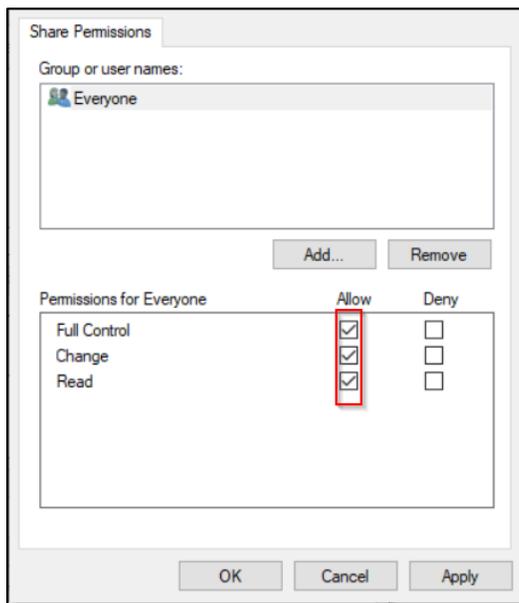
**NOTE:** If you don't see the sharing tab, you do not have administrative rights and sharing is disabled.

5. The first setting for a shared folder is determining the share name.



If you wish to update SOLIDWORKS service packs automatically the next time SOLIDWORKS is opened, it is important to note several things about the share name scheme (if unfamiliar with network protocols):

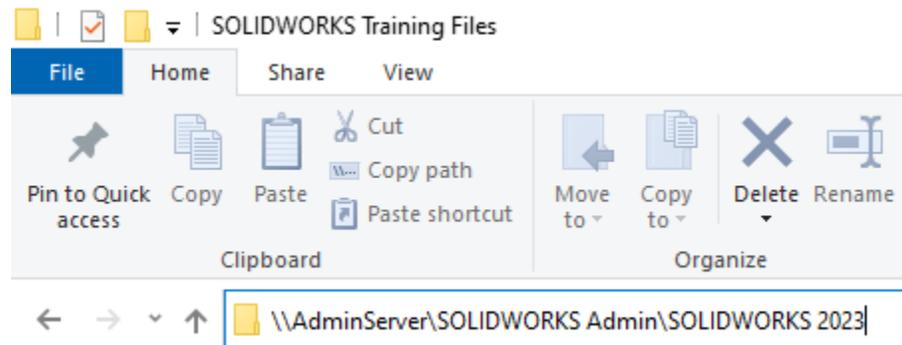
- a. The "share name" by default is the name of the folder you are trying to share.
    - i. This name can be changed so it differs from the original folder name.
    - ii. Users who view this folder on the network will see it by its "share name".
  - b. With regard to SOLIDWORKS:
    - i. All future service pack updates will be created in a specific folder but will have the "share name" changed to match the same "share name" as the original folder it is replacing.
    - ii. A good example of a naming scheme is to name the folder "SOLIDWORKS 2023 SPX.X" and have the share name just be "SOLIDWORKS2023".
6. Press the **Permissions** button and make sure **Everyone** has full access to this shared folder. Set all permissions to **Allow**.



Permissions for Everyone	Allow	Deny
Full Control	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Change	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Read	<input checked="" type="checkbox"/>	<input type="checkbox"/>

7. Press the **Apply** button, then press **OK** to finish sharing this folder.
8. If using a shared network location for your Toolbox folder, be sure to repeat creating a shared folder (steps # 3, 5, and 6 above) for this folder to make sure that all users can access the shared toolbox location.
9. Using Windows Explorer, locate the folder created in step 2 nested under "Network" and right-click on it and choose **Open**. A new Explorer window will open and in the address bar is the UNC path for this folder.

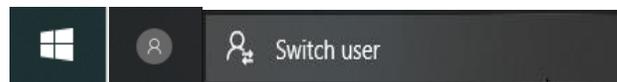
**IMPORTANT:** Write this address down as it will be the location clients will load & unload from later.



Now you have a location for the administrative image to reside, the next section illustrates the steps it takes to create the administrative image in this shared folder.

### Creating the Admin Image

1. Make sure you are logged in as the administrator.

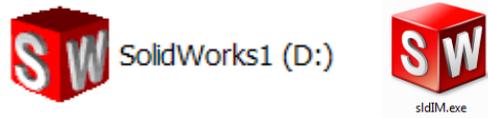


2. Temporarily disable all anti-virus and anti-spyware tools.



3. Run the SOLIDWORKS Installation Manager
  - a. If downloading from the [SOLIDWORKS downloads page](#), run the downloaded SolidWorksSetup.exe file and the Installation Manager should start automatically.

- b. If downloading from another source or the Installation manager does not automatically start, double-click on the setup.exe within the unzipped folder.

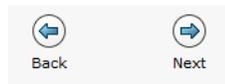


4. The Welcome page asks for the type of installation you would like to perform. You will want to choose **Administrative Image**.

Install on this computer  
 Create an administrative image to deploy to multiple computers  
      Create a new image using default settings.  
      Create a new image using settings and files from an existing image.  
         [No location chosen] Browse...  
 Install server components

- If you are loading an admin image for the first time, pick the option to **Create a new image using default settings**.
- If you have an existing admin image you would like to update, please go to the section **Upgrading an Image** in this document. This will give full instructions on this type of installation.

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



6. The first page will ask for your 16 or 24 digit serial number. If you have this information coming from a document, you can copy and paste this information into the first cell, and it will auto-fill correctly.

DS **SOLIDWORKS** 2023

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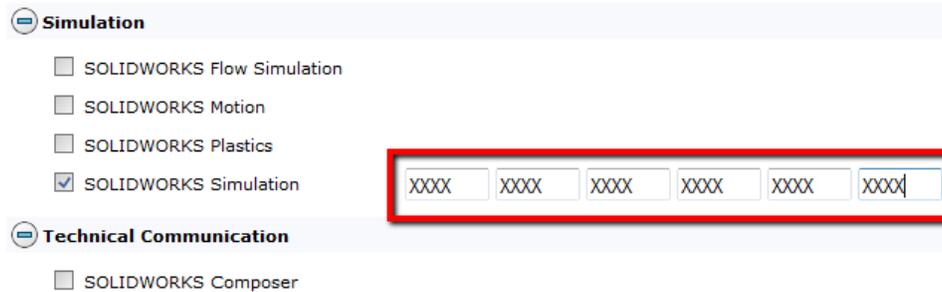
**Serial Number**  
Enter your serial number information

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3D Design  
 SOLIDWORKS

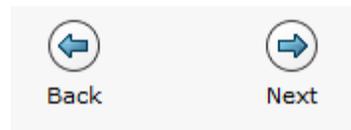
XXXX	XXXX	XXXX	XXXX	XXXX	XXXX
------	------	------	------	------	------

7. If you are loading other products with separate serial numbers, enter these numbers as well.

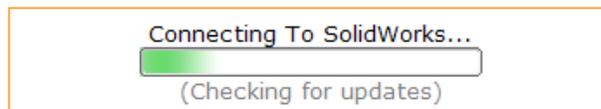


- NOTE: Enter **ALL** products you plan on installing. Any product with a distinct serial number should be entered here.
- SOLIDWORKS PDM Standard is included with SOLIDWORKS Professional and SOLIDWORKS Premium and only needs to be entered if this item was purchased separately with SOLIDWORKS Standard.
- Also, Basic Simulation and SOLIDWORKS Motion are included with SOLIDWORKS Premium; these items only need to be entered if you have purchased these separately.
- If you purchased Simulation Professional or Simulation Premium, you should enter your serial number for SOLIDWORKS Simulation to ensure the full version is loaded.

8. Press the **Next** button to continue. You may be informed that there is a newer version of SOLIDWORKS available for download. We recommend you choose to continue installing with your original service pack and upgrade later. Press the **Next** button to continue.



9. The installer will check to see that it is up to date, as well as confirming what products you are entitled to load.

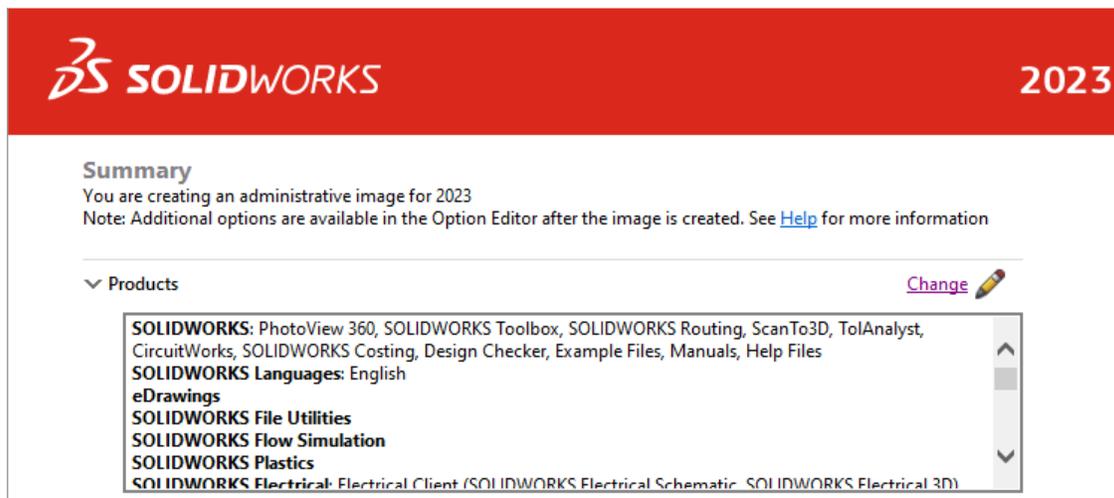




10. If you are not online, a warning will appear informing that you cannot automatically select products to install. Pressing **Cancel** will continue with the load, but you must manually select all your add-ins during the installation instead of being given a list based on your serial number.



11. With the serial number, you are provided the version of SOLIDWORKS and its add-ins in a list. If you would like to edit what products will be loaded, you can press the **CHANGE** button. However, we recommend that **all** products be included, as this makes installation easier in the future if additional products are purchased at a future date.



12. To change any of the options of how the administrative image will behave, you can make changes on this page.

> Download Options	<a href="#">Change</a> 
v Administrative Image Location New image location: D:\SOLIDWORKS Admin\SOLIDWORKS 2023	<a href="#">Change</a> 
v Administrative Image Type Type: Standard Administrative Image	<a href="#">Change</a> 

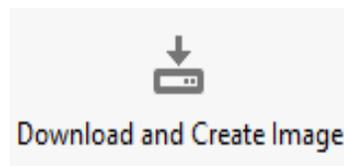
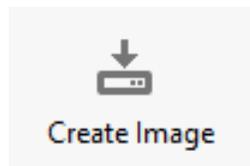
- Administrative Image Location
  - We recommend that you press the **CHANGE** button here and browse from the default location to inside the shared folder that was created earlier in previous section of this document.
- Background Downloader
  - We recommend that users **disable** this feature as we find that most anti-virus software interferes with the background downloader, preventing it from collecting all upgrade files.
- Administrative Image Type (There are three ways to save an administrative image now in 2023)

#### Administrative Image Type

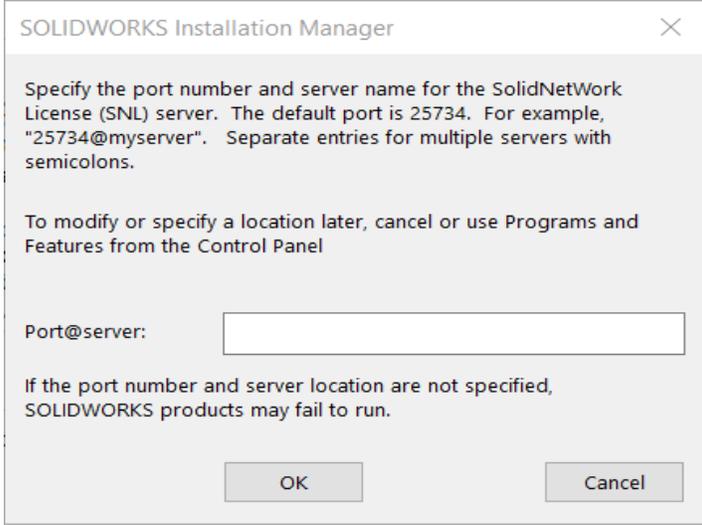
- Standard Administrative Image**  
This is a standard Windows Installer administrative image. It stores all the files required to install the selected SOLIDWORKS products in a shared folder. It is for users who have reliable access to the company network.
- Remote Client Administrative Image**  
This administrative image enables remote users to access and install specified SOLIDWORKS files over the internet from the Dassault Systèmes SOLIDWORKS Downloads site.
- Compressed Administrative Image**  
This is a standard administrative image with files compressed, so it can be copied easier. For upgrades, users must download the entire image rather than just patches.

13. You will also be shown the required file size, available disk space and the size of any downloads. You may need to load the selected programs and settings specified in the summary at the bottom of this page.

14. Click the **Create Image** (or **Download and Create Image**) button to continue.



If you are using a network serial number, you will be asked for the location of the network server. Use the default port number "25734@[server name]". If you have not loaded the network server, answer **Yes** to the following warning and you can load the server later.



SOLIDWORKS Installation Manager

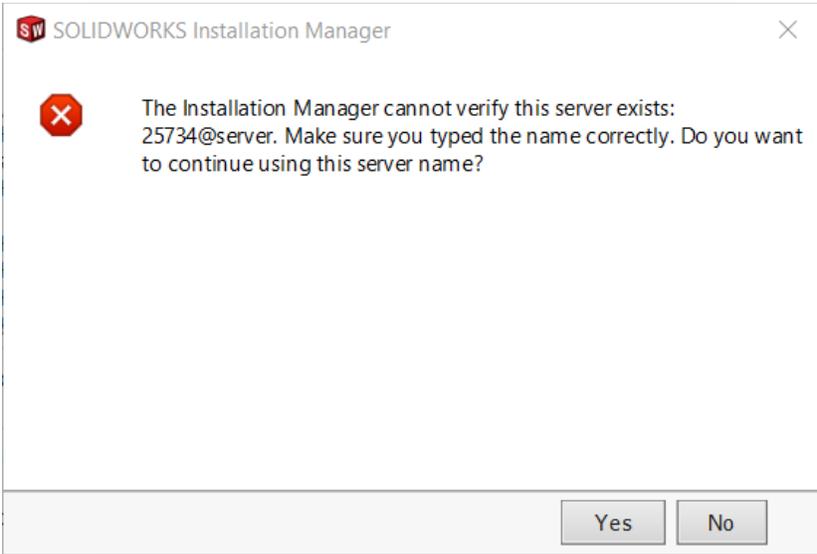
Specify the port number and server name for the SolidNetWork License (SNL) server. The default port is 25734. For example, "25734@myserver". Separate entries for multiple servers with semicolons.

To modify or specify a location later, cancel or use Programs and Features from the Control Panel

Port@server:

If the port number and server location are not specified, SOLIDWORKS products may fail to run.

OK Cancel



SOLIDWORKS Installation Manager

 The Installation Manager cannot verify this server exists: 25734@server. Make sure you typed the name correctly. Do you want to continue using this server name?

Yes No

At this point, your administrative image will be written to the shared location you setup in the previous section of this document.

## Creating Administrative Image



Creating image: SOLIDWORKS Installation Manager

1% complete



**HAWK RIDGE SYSTEMS**

All the files needed to load SOLIDWORKS have been copied over to your administrative folder. You now have the option of opening a help file to guide you through the client installation. You can also launch the Image Option Editor. Both of these are explained in the following section.

If you selected to activate the product, you will need to enter an e-mail address as contact information. Press **OK** to continue.

SolidWorks Product Activation

Enter the following activation contact information.

E-mail:

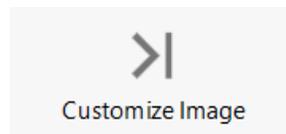
OK

At this point, the administrative image has been loaded on this server. You can choose to have instructions on how to install a client, though we have a more complete guide in the following section of this document. Press the **Customize Image** button to continue. This will start the Options Editor.

### Administrative Image Creation Is Complete

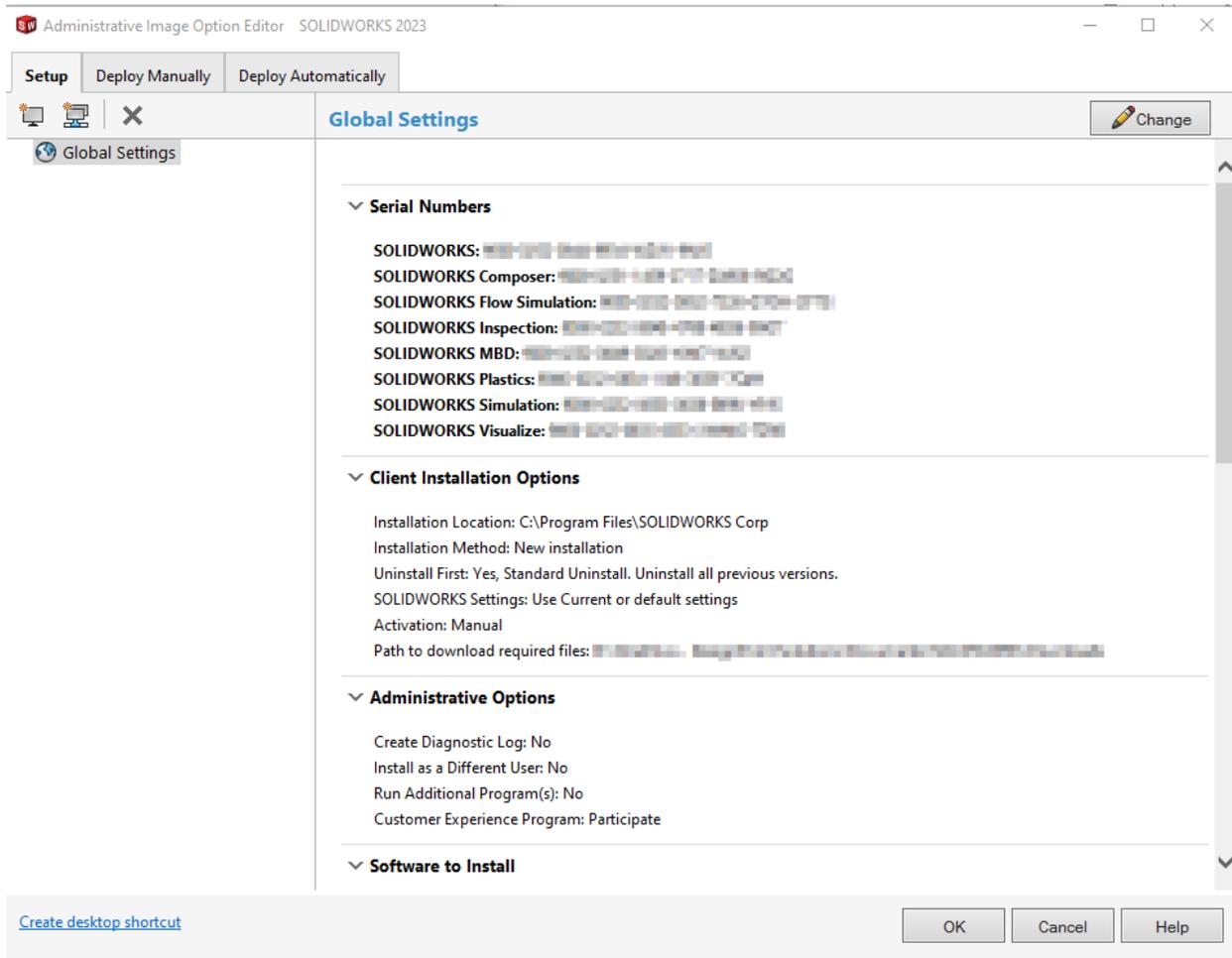
Show me how to install this image on a client.

**Note:** Click **Customize Image** to customize your Administrative Image options using the Option Editor. (See [Help](#) for more information). You can return to the Option Editor at any time.



# Administrative Image Option Editor

The Administrative Option Editor can be started from the installation wizard or it can be accessed later. It can be found in the following location within the administrative image: **<installation directory>\sldadminoptioneditor.exe**.

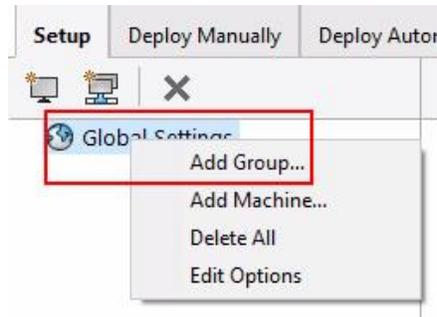


The Option Editor lets you assign serial numbers and options to specific users and groups which will be distributed when the admin image is used to install on a user's computer.

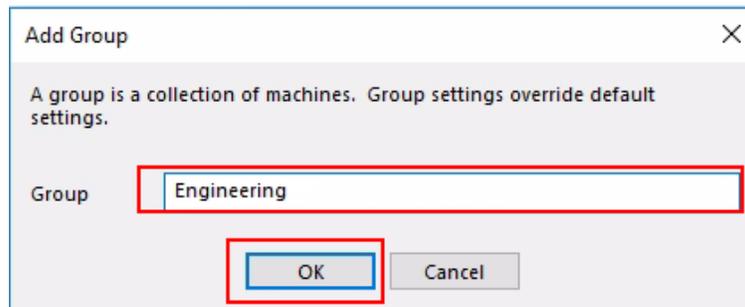
## Adding Groups and Machines

You do not need to create groups if you want all clients to use the global settings, but if you would like to assign serial numbers or set specific options and software to load on particular machines, you can set the editor to push these settings automatically.

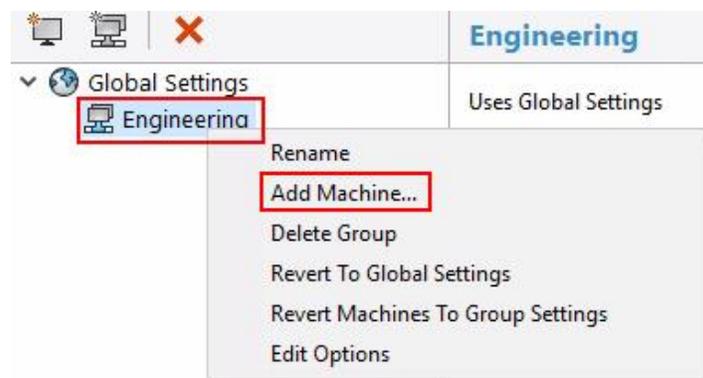
1. Start with right-clicking the **Global Settings** in the left column of the editor and choose to **Add Group**.



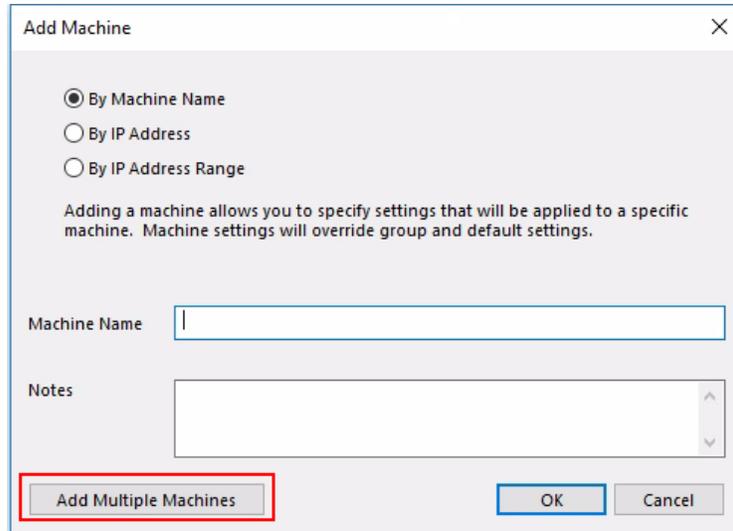
2. In the resulting dialog box, assign a "Group Name" and press **OK**. You can add more groups by repeating these last two steps or continue on to add users.



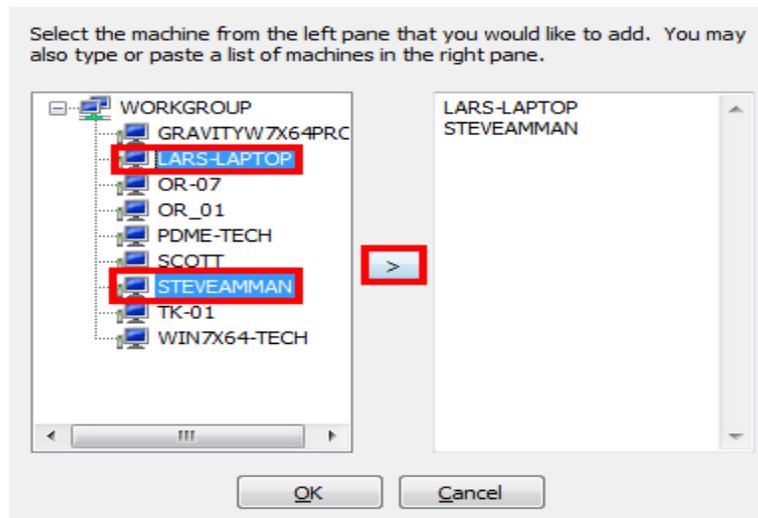
3. Right-click on a group and choose **Add Machine** to add users' machines to the editor.



The resulting dialog box lets you type in a machine name or an IP address or a range of IP addresses, but it may be easier to use the multiple-machines interface. Press the **Add Multiple Machines** button in the lower left corner of the box.

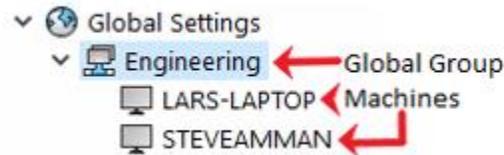


Select the machines from the network list that you want to load SOLIDWORKS onto. Pressing the **arrow** in the center column will populate a list of machines you want to add to the group. Press the **OK** button to finish.



## Setting Options

Settings can be assigned globally or to specific groups or machines to change different aspects of the installation. By default, the "Global Settings" will be displayed, and these will be the default settings for all users. If you have setup groups or machines as in the previous section, you can select on one of these items from the left-hand column to edit a specific option for the item selected.



Once you have selected Global Settings, a Group, or a particular Machine, you will see a button with a pencil icon next to the button. The level of assignment and the operating system will be listed. Press this pencil button to edit the installation options.

All settings will default to the higher-level settings, meaning Global Settings will be the basic options. These can be overridden by group settings and group settings can be overridden by machine settings. If the default settings are overridden, they will be highlighted in yellow to let the administrator easily see what items have been modified.

Let's go through sets of options in order now.

1. **Serial Numbers** can be assigned in the first group. Also, the Network Server can be mapped if using Network Licenses.

**Serial Numbers**

**SOLIDWORKS Standard, Professional, Premium or SolidNetWork License**

SOLIDWORKS Composer

SOLIDWORKS Composer Player Pro

SOLIDWORKS Electrical Schematic

SOLIDWORKS Flow Simulation

SOLIDWORKS Inspection

SOLIDWORKS MBD

SOLIDWORKS Motion

SOLIDWORKS Plastics

**SOLIDWORKS Simulation**

SOLIDWORKS Workgroup PDM

**SNL Server port@server**   
(ex. 25734@myserver) Separate multiple entries with semicolons.



2. The next area is **Client Installation Options**; these allow you to determine if the installation creates a new version or updates an older installation. We recommend upgrading the existing image if available. However, the Admin Image now has the ability to uninstall the previous major version of SOLIDWORKS. There are options for how you would like to do this once you select the option to uninstall. You can also install with SOLIDWORKS default options or use a .sldreg file created with the Copy Settings Wizard. You can also determine if the client will activate their SOLIDWORKS license automatically.

▼ **Client Installation Options**

How do you want to perform client installations of major versions?

**Note:** Both options will update the installation when applying a service pack.

Upgrade an existing major version (if one exists).

Create a new major version.

Location for new installation (if needed):

C:\Program Files\SOLIDWORKS Corp Browse...

How do you want to apply SOLIDWORKS Settings?

**Note:** These settings can be obtained by running the SOLIDWORKS Settings Administrator.

Use current settings (if they exist) or use SOLIDWORKS default settings

Use a settings file exported from the SOLIDWORKS Settings Administrator.

How do you want to activate client installations?

Activate automatically when each client installation completes.

Do not activate automatically.



3. The third area is **Administrative Options**, where you can choose to create a log file during each installation. (We recommend selecting this option, as it can be a useful troubleshooting tool.)

You can select the option to load a program before and after installation. If these are checked on, you can browse to a network location where these programs reside.

Finally, you can select if the clients will participate in the Customer Experience Program which will periodically send e-mails to SOLIDWORKS regarding performance and feature usage.

▼ **Administrative Options**

Create a diagnostic log during each installation (slower.)

Run the installation as a different user.

**Note:** Use this for client computers that do not have administrative privileges and are not using Deploy Automatically.

Run a program before installation:

Run a program after installation:

Do you want clients to participate in the SOLIDWORKS Customer Experience Program?

Yes, I want to join

No, thank you

Remind me later

4. The next area, **Software to Install**, allows you to determine which components of SOLIDWORKS will be loaded onto the client machine.

▼ **Software to Install**

- ▼  SOLIDWORKS
- ▼  SOLIDWORKS Languages
  - eDrawings
  - SOLIDWORKS File Utilities
  - SOLIDWORKS Flow Simulation
  - SOLIDWORKS Plastics
- ▼  SOLIDWORKS Electrical
- ▼  SOLIDWORKS Composer
  - SOLIDWORKS Inspection
  - SOLIDWORKS Visualize
  - SOLIDWORKS Visualize Boost
  - SOLIDWORKS CAM
  - SOLIDWORKS PCB
- ▼  SOLIDWORKS Manage Client
- ▼  SOLIDWORKS PDM Client
  - SOLIDWORKS Simulation
  - SOLIDWORKS Motion



5. The final tab is **Standard Libraries**. If you are using a central toolbox library, which is recommended for a large number of users, you can set the location of this shared library to be accessed by the clients.

▼ **Toolbox/Hole Wizard Options**

SOLIDWORKS Toolbox includes a library of standard hardware. If you have multiple users, you might want to choose a central location for these files.

Specify the Toolbox location for the client computer.

C:\SOLIDWORKS Data Browse...

If you have a central or PDM Toolbox location, you can choose not to install the Toolbox data with each installation of SOLIDWORKS. At least one person in your organization must install/upgrade the data.

Do you want this client to install/upgrade the Toolbox files?

Yes, include the data files during installation

No, install the Toolbox software without including the data files.

Note: If the location contains an older major version of Toolbox, that Toolbox will be upgraded, and the files will no longer be compatible with previous major versions of SOLIDWORKS.

6. To save the settings, simply close the editor, press the **Close** button or the "X" in the upper right corner of the option editor window and then click the **Yes** button when asked to save changes.

## Deploying Installation from Option Editor

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

- Use the Deploy Manually page of the Option Editor to email instructions to users on how to start the installation. This method should also be used when users have administrative privileges for their client machines
- Use the Deploy Automatically page of the option Editor to push installations automatically to target machines. Customs uninstalls can also be performed. This method should also be used when users do not have administrative privileges to install software on their systems.

To use either method, you must change the administrative image installation directory to a Windows shared directory, so it is available through a network location (for example, \\machine\shared\_directory or copy the whole image to a local directory on each machine, such as C:\SOLIDWORKS Admin\SOLIDWORKS 2023

# Administrative Client Installation

1. Make sure you are logged onto the client machine as an administrator.



2. Disable all anti-virus and anti-spyware software on the computer you are loading the client. If loading from a remote location, be sure to disable the anti-virus and anti-spyware while loading the administrative image to the desired drive.



3. Clients should browse to the Administrative Image that is installed on the server, find the location **<install\_dir>\StartSWInstall.hta** and double-click on this file.
4. A webpage will open in their web browser. They will need to click the **Install SOLIDWORKS products now** button.

*Click **Install SolidWorks products now** to start the installation of SolidWorks on the local machine:*



- SOLIDWORKS will automatically begin to load according to the options set in the last section.
- Once the installation is complete, the client will then be able to run SOLIDWORKS.
- When SOLIDWORKS is loaded using this "one touch" deployment, service pack upgrades will happen automatically when the administrative image is updated as outlined in the following section. When the user starts SOLIDWORKS and a newer administrative image is detected, they will be prompted to update. When they choose to update, the process will happen automatically. They can choose to continue without updating but will be prompted until they have upgraded to the same level with the administrative image.

# Updating an Image

To update an Administrative Image, you load a new image from the latest version of SOLIDWORKS and switch the shared directory location to this new image. The clients will automatically update the next time they start SOLIDWORKS and the new image is detected. Alternatively, you can control when the updates go out manually, by creating a new image on the network and simply running the new StartSWInstall.hta file on your clients.

## To Update the Administrative Image

1. Login to the computer with your administrative image with full administrative rights.



2. Disable all anti-virus and anti-spyware tools.



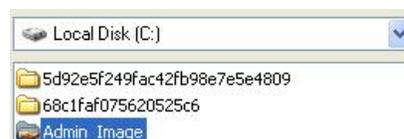
3. Start the SOLIDWORKS Installation Manager by inserting the DVD. Or you can download the latest SOLIDWORKS Installation manager by going to <https://www.SOLIDWORKS.com/sw/support/downloads.htm> and selecting the service pack you require. The SOLIDWORKS installer for this version can be downloaded from here.
4. Start the SLDIM.exe. On the **Welcome** screen, you will be asked for the type of installation. Pick **Administrative image and server products** and **Create or update an administrative image**. Then select **Use installation settings from an existing image**.

- Install on this computer
- Create an administrative image to deploy to multiple computers
  - Create a new image using default settings.
  - Create a new image using settings and files from an existing image.

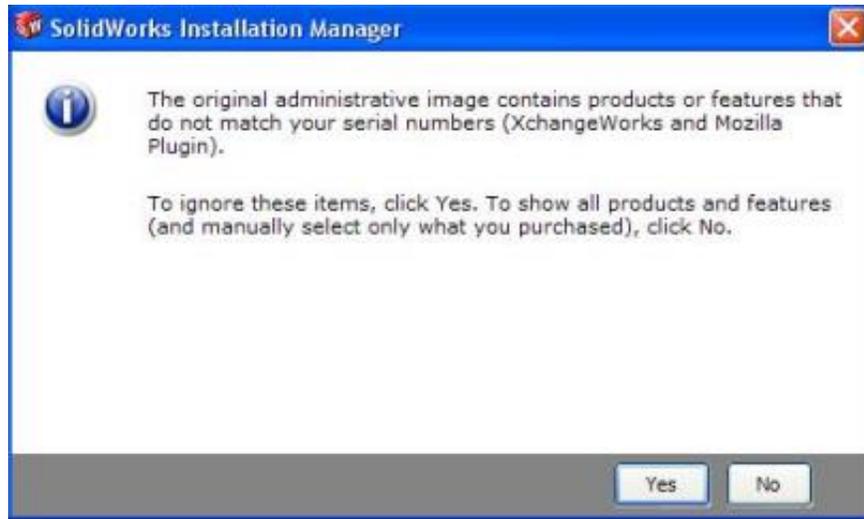
[Click **Browse** to choose a location]

Browse...

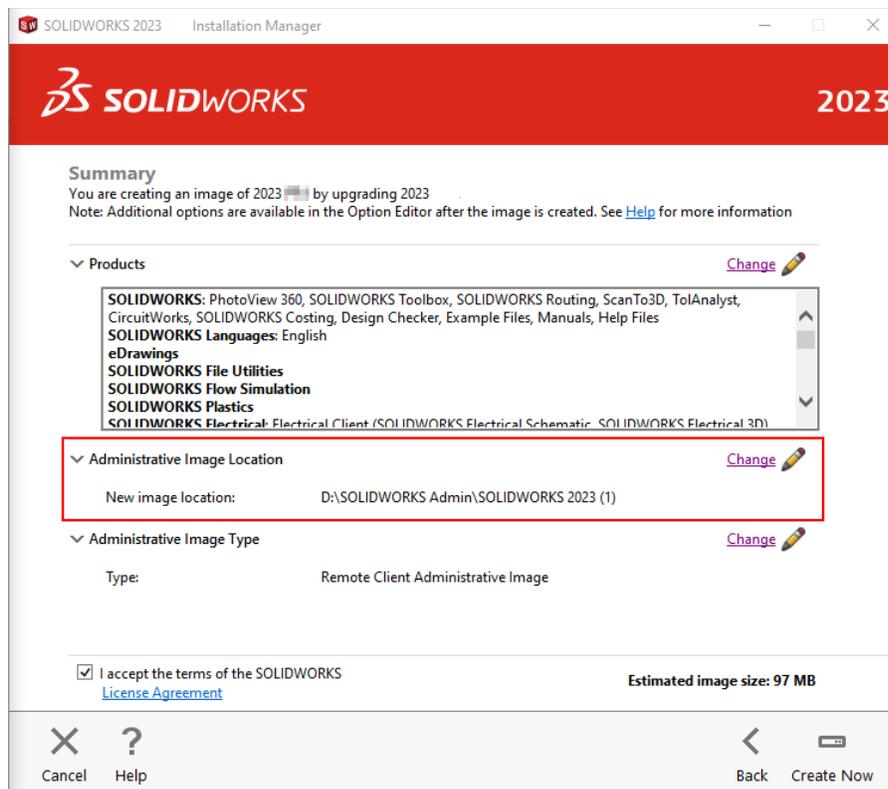
Press the **Browse** button and locate your existing admin image to use the same settings from this location.



5. Press the **Next** button to continue.
6. If you are warned about any products with different serial number, press **Yes** to ignore and use the current serial numbers.



7. The Summary page will show all the products of which you are going to create an image, and how and where any downloads will come across.

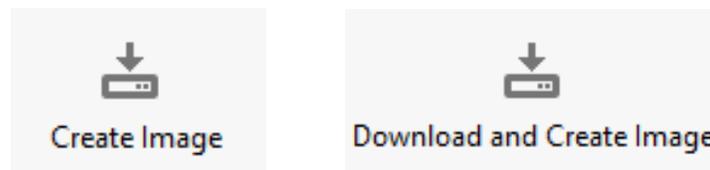




The most important setting on this page is the **Administrative Image Location**, this is where your new administrative image will be loaded. The new admin image must reside in an entirely new folder separate from the old admin image. You will want to take note of this. Once the new image is created, you will share this location with the same share name you used for your original image. If you are not sure about this, please see the **Creating an Admin Image** section about how to properly set up a network share name for an admin image.

Client options can also be set here but can later be changed in the Option Editor as in the first section of this guide.

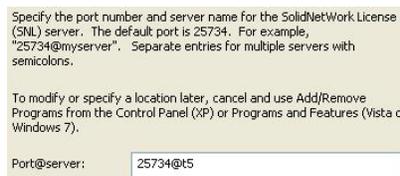
8. Click **Create Now** or **Download and Create Image**. If you see a message that the install directory does not exist, click **Yes** to create it.



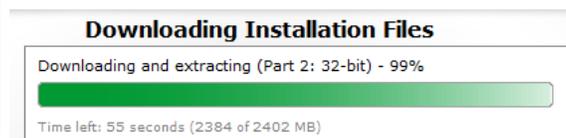
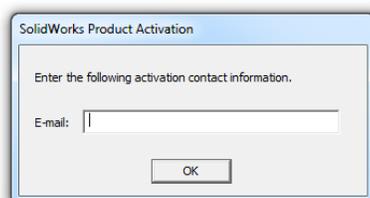
9. If any downloads are required, based on the programs you selected on the summary page, these will now be retrieved from the internet.



10. If you are using a network license serial number, you will be prompted for the port and machine location of your SolidNetWork License server. Press **OK** to continue.



11. If you selected to activate the product, you will need to enter an e-mail address as contact information. Press **OK** to continue.



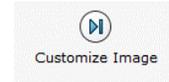


- At this point the administrative image has been loaded on this server. You can choose to have instructions on how to install a client, though we have a more complete guide in the following section of this document. Press the **Customize Image** button to continue; this will start the Options Editor.

### Administrative Image Creation Is Complete

Show me how to install this image on a client.

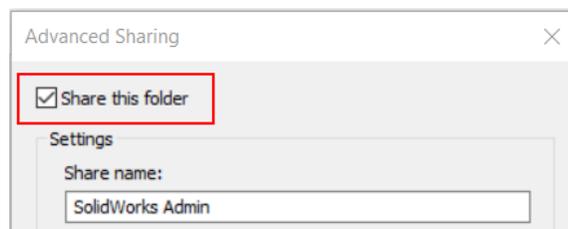
**Note:** Click **Customize Image** to customize your Administrative Image options using the Option Editor. (See [Help](#) for more information). You can return to the Option Editor at any time.



- Your Options Editor will have the same information that was in the options from your previous administrative image. You can close the options editor to complete the update.

### Updating Your Image Location

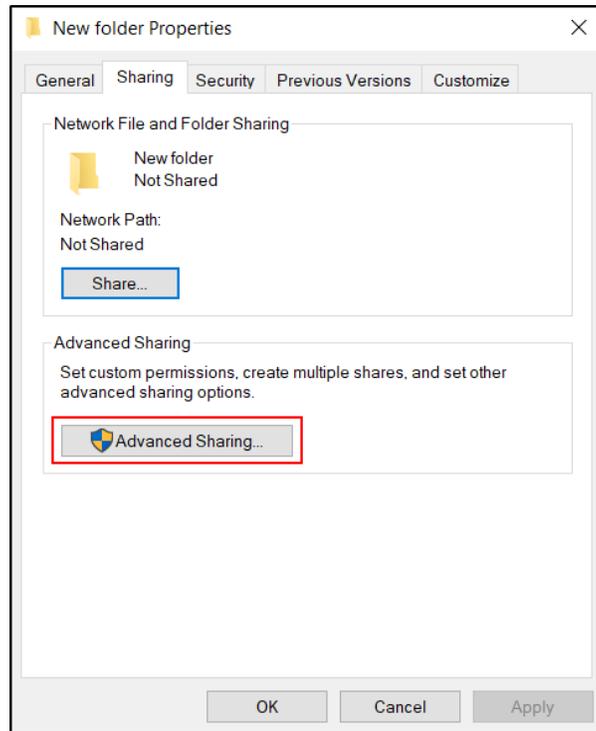
- Find your original administrative image with Windows explorer, right-click and choose **Properties**. Go to the **Sharing** tab, note the share name as you will need to use this for the new image. Pick the **Advanced Sharing** button and on the resulting dialog box deselect the **Share this Folder** check box.



- Move the new administrative image to the same, top level area on the server as the old administrative image with Windows Explorer.

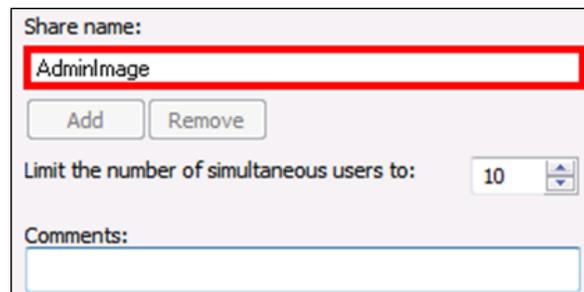


3. Right-click this new folder and select **Properties** from the pop-up menu. Inside the properties dialog box, switch to the **Sharing** tab and select **Advanced Sharing**.



**NOTE:** If you don't see the sharing tab you don't really have administrative rights or are an administrator with limited rights.

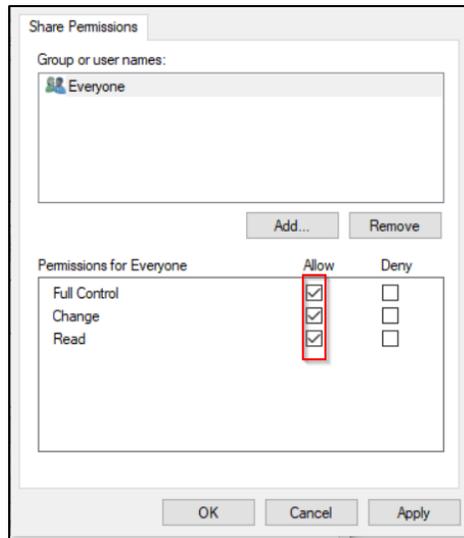
The first setting for a shared folder is determining the share name.





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Press the **Permissions** button to set Share Permissions and make sure **Everyone** has full access to this shared folder. Set all permissions to **Allow**.



Press the **Apply** button, then press **OK** to finish sharing this folder.

The Option Editor will retain all of the settings from the original administrative image. It is now a good time to make any adjustments to the Option Editor and save the changes before deploying.

- You cannot be using the Option Editor while users are updating. The Option Editor writes XML directly to the administrative image and may corrupt or prevent SOLIDWORKS from successfully updating if this action takes place while users are accessing the administrative image.
- When SOLIDWORKS starts on a client machine that installed from the original administrative image, it sees the new version and prompts the user to upgrade the client.
- When the user accepts the upgrade, the client machine is automatically upgraded to the same version as the administrative image with the options setup within the admin image.
- For additional documentation, please refer to the [SOLIDWORKS Authored Administration Guide](#) which also includes information on silent and active directory installations.
- Official System Requirements page <https://www.solidworks.com/support/system-requirements>

For further assistance, please contact our support team at [support@hawkridgesys.com](mailto:support@hawkridgesys.com), or 877-266-4469 (US) or 866-587-6803 (Canada).