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SolidWorks New User Tips

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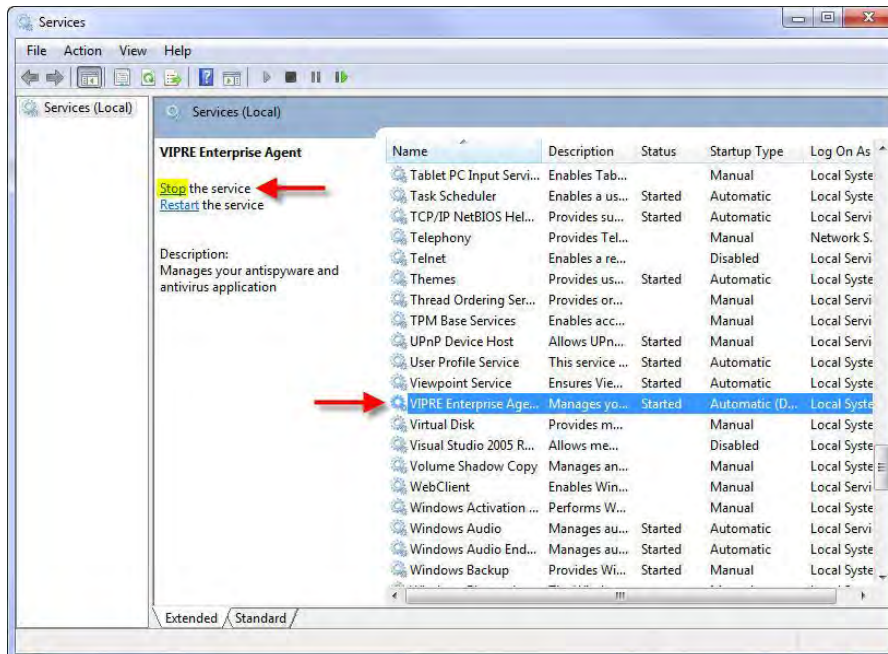
NOTICE: THIS DOCUMENT WAS WRITTEN BY A USER USING WINDOWS 7. THERE WILL BE SOME LOCATION AND APPEARANCE DIFFERENCES COMPARED TO WINDOWS XP AND VISTA.

Installing SolidWorks

Installing SolidWorks (Standalone)

When attempting to install SolidWorks, you should be logged in as the Administrator on that computer.

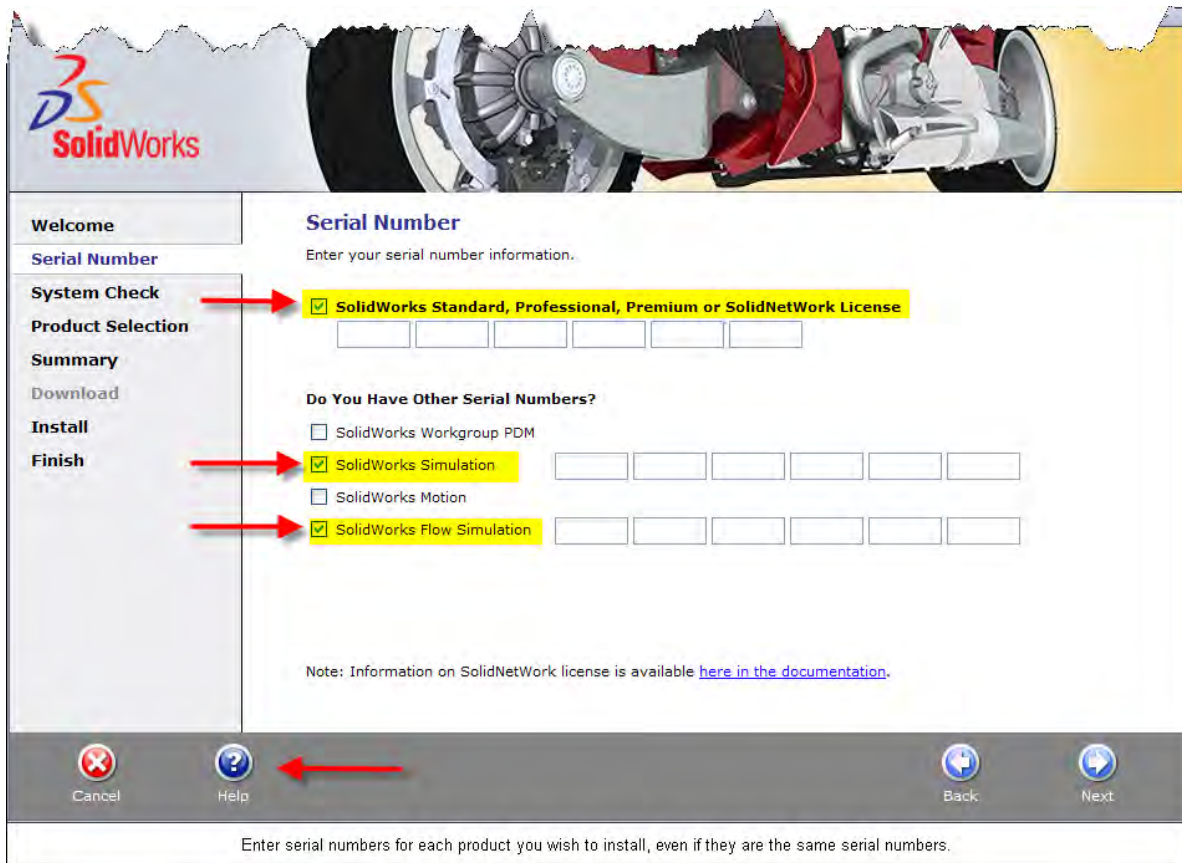
Before the installation begins, any anti-virus or anti-spyware should be completely disabled using the Services menu. This can be accessed by going to **Start-> Run**, type “**services.msc**” (without quotations). Once the Services window has opened select each instance of the anti-virus program and stop it.



*** IMPORTANT** - If you are running Windows Vista or Windows 7, it is recommended to disable your User Account Controls as this can cause conflicts with your SolidWorks installation. For steps to disable this option, browse to <http://www.javelin-tech.com/blog/2011/01/uac/>

Once the anti-virus and User Account Controls have been completely disabled, and you are logged in as the system administrator, you can begin the SolidWorks installation. When prompted to enter your SolidWorks serial number, enter the full, 24 digit serial number.

Note: If you have purchased multiple SolidWorks products (such as Simulation, Motion, etc), then you must click the blue link “**Click here to enter additional serial numbers**” to enter the serial number for each product that you are trying to install, *even if* the serial numbers are the same for each product.



Continue with the installation. For further information, click the “**Help**” button within the SolidWorks Installation Manager. For information on installing multiple releases of SolidWorks in unique directories, see the section of this document entitled [Installing to a Unique Directory](#).

Installing SolidNetwork License Manager (SNL)

Beginning with the SW2010 release, the SNL SolidNetwork License Manager uses activation based licensing. This document will detail installing the licensing server for SW2010 and later. If you require assistance with this for SW2009 (or earlier), please contact Javelin Technical Support for instructions specific to this scenario.

When installing SolidWorks with a network license, there are several steps you must complete. For best results, these steps should be completed in this order:

1. Uninstall the existing SolidNetwork License Server (if applicable)
2. Install the SolidNetwork License Server
3. Install SolidWorks client installations

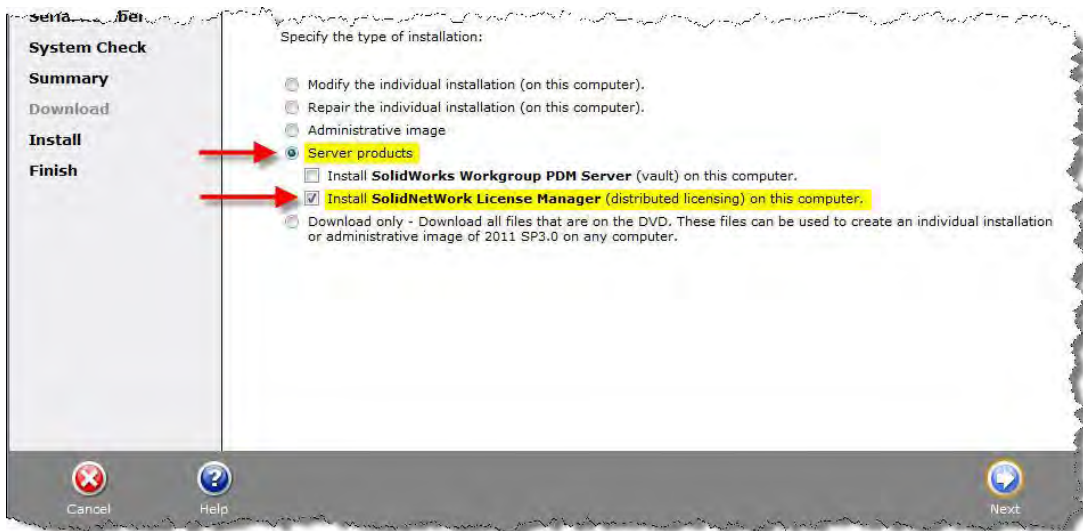
If the server has an existing SolidNetwork License Server installed and running, then you must launch this tool and press “**Stop**” to stop the licensing service. You can then uninstall this via Control Panel > **Programs and Features** (Add/Remove Programs in WinXP).

The SolidNetwork License Server software can be installed from the SolidWorks DVD by launching the regular SolidWorks Installation Manager, or by downloading just the license manager software from the SolidWorks Customer Portal. For more information on using the Customer Portal, see the section of this document entitled [Using the Customer Portal](#).

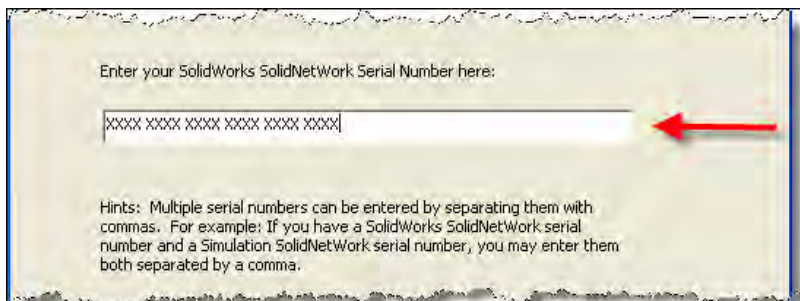
The SolidNetwork License Server is typically installed on a server, but it could alternatively be installed on a SolidWorks client machine if desired. If installing on a server, it is only necessary to install the license server software, not the entire SolidWorks application.

After launching the standard SolidWorks Installation Manager, enter your SolidWorks Network license serial number. When prompted for the installation type, choose **“Server Products”**, followed by selecting only **“Install SolidNetwork License Server...”**.

Note that if you have specifically downloaded *just* the SolidNetwork License Manager then you may not need to choose this.



Continue with the installation. When prompted, enter your network serial number. If you have multiple network serial numbers, enter those at the *next* screen, with a comma separating each serial.



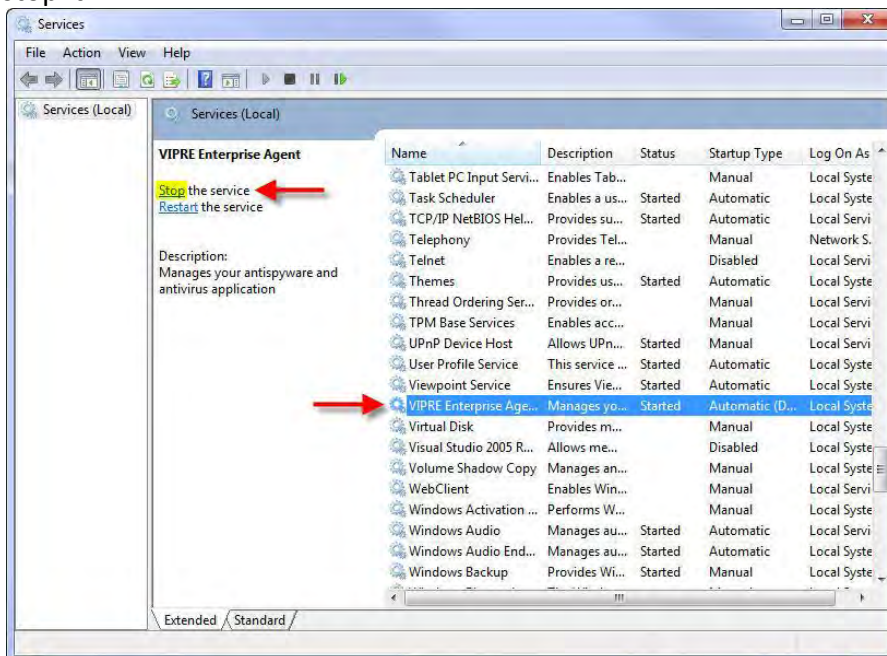
Enter multiple network serial numbers here, separated by a comma

Once the installation completes, you can launch the SolidNetwork License Manager from the SolidWorks program group via the **Start** menu. You will need to activate the license manager before it will become fully functional; for instructions on this, please see the section of this document entitled [Activating the SolidNetwork License Server](#).

Installing SolidWorks on Clients (SNL)

* **IMPORTANT** - When attempting to install SolidWorks, you should be logged in as the Administrator on that computer.

Before the installation begins, any anti-virus or anti-spyware should be completely disabled using the Services menu. This can be accessed by going to **Start-> Run**, type “**services.msc**” (without quotations). Once the Services window has opened select each instance of the anti-virus program and stop it.

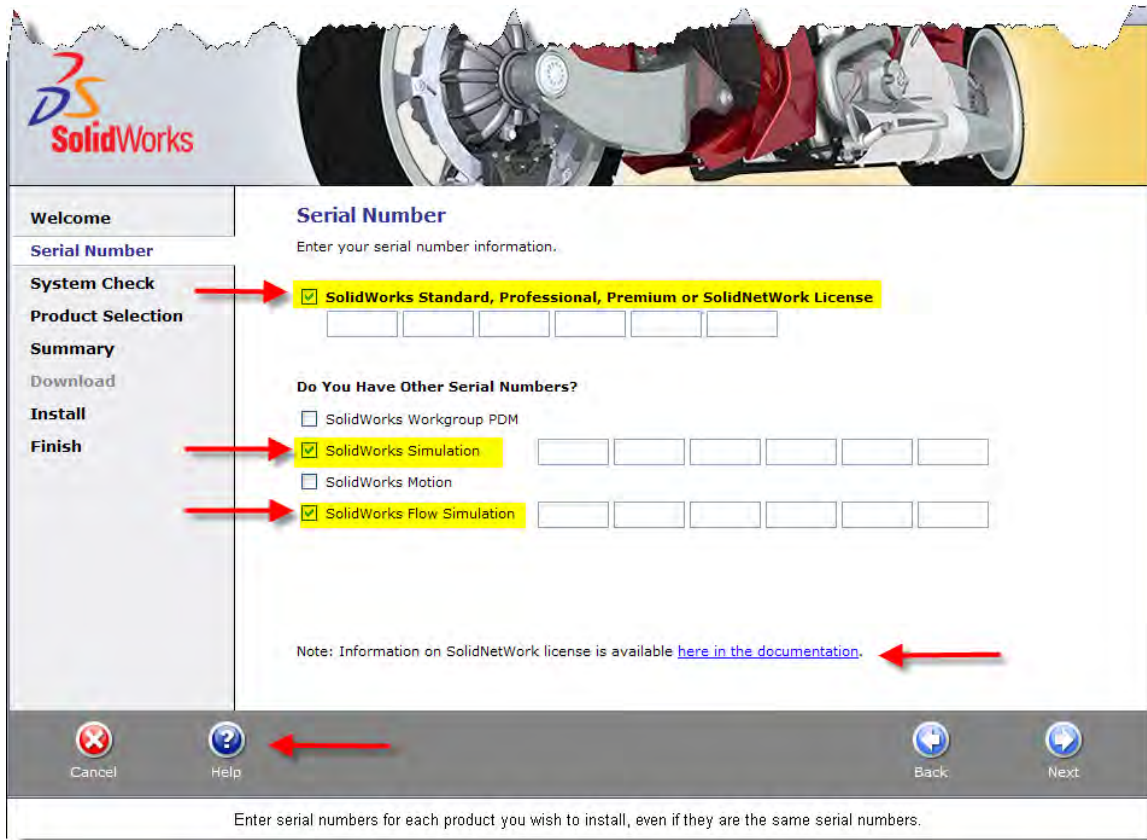


* **IMPORTANT** - If you are running Windows Vista or Windows 7, it is recommended to disable your User Account Controls as this can cause conflicts with your SolidWorks installation. For steps to disable this option, browse to <http://www.javelin-tech.com/blog/2011/01/uac/>

Once the anti-virus and User Account Controls have been completely disabled, and you are logged in as the system administrator, you can begin the SolidWorks installation. When prompted to enter your SolidWorks serial number, enter the **16 or 24 digit** serial number. If you have a 16 digit serial, leave the last 2 boxes empty when entering the serial number (there is room for 24 digits).

Note: If you have purchased multiple SolidWorks products (such as Simulation, Motion, etc), then you must click the blue link “**Click here to enter additional serial numbers**” to enter the serial number for

each separate product that you have purchased and wish to install, *even if* the serial numbers are the same for each product.



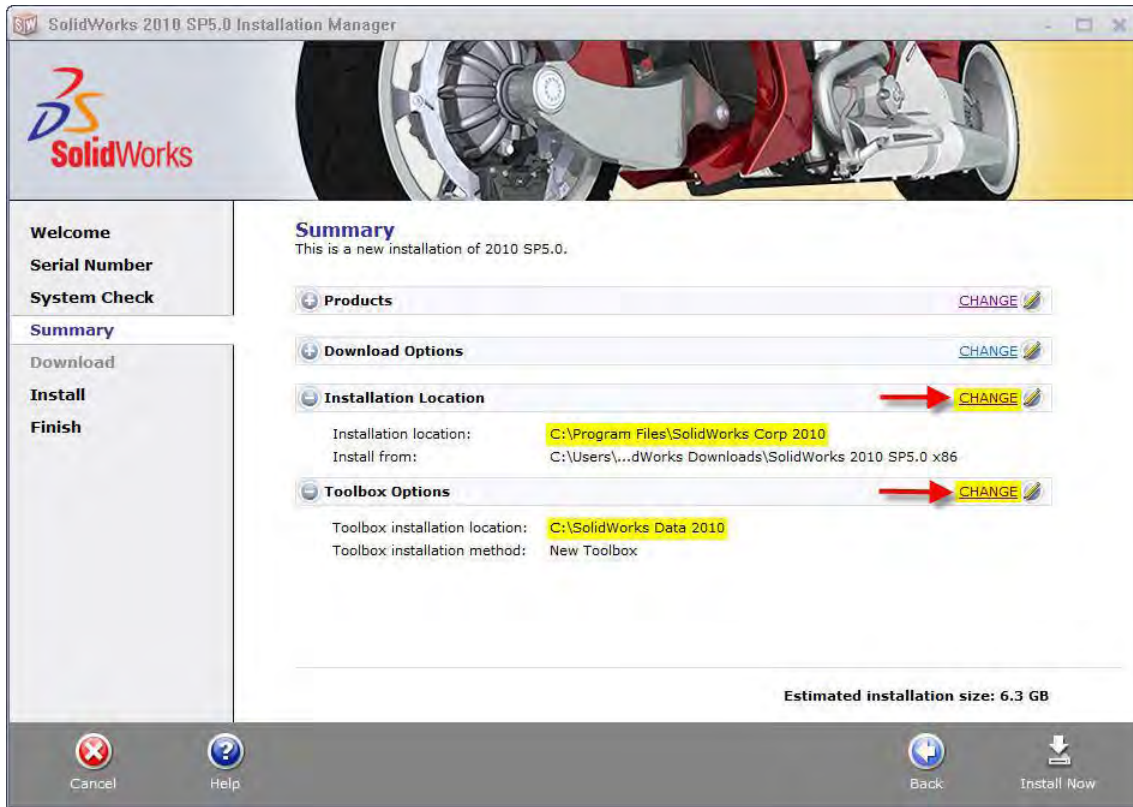
Continue with the installation. For further information, click the blue link **“here in the documentation”** or click the **“Help”** button within the SolidWorks Installation Manager. For information on installing multiple releases of SolidWorks in unique directories, see the section of this document entitled [Installing to a Unique Directory](#).

Installing SolidWorks with Unique Install Directories

It is possible to install multiple releases of SolidWorks (i.e. SW2011, SW2010, etc.) on the same computer. If you wish to install more than one version of SolidWorks (or may in the future), it is essential to have the different versions of SolidWorks, as well as the Toolbox databases, installed in unique locations. This will ensure that having multiple installs is successful and all functionality is working as intended because each version of SolidWorks has a completely unique set of files to use.

When installing SolidWorks, you will be prompted for an installation directory; the default is typically **“C:\Program Files\SolidWorks Corp”**. Simply edit this location by adding the release year to the folder name to make it **“C:\Program Files\SolidWorks Corp 2010”** (for SW2010).

Similarly, when choosing a location for the Toolbox, you can add the version to the install folder. This can be done regardless of whether you are creating a local database (for a single user) or a network based Toolbox (for multiple users).



Making these small changes when first installing SolidWorks can prevent many issues later on if multiple versions of the software are installed at the same time.

Updating Existing SolidWorks Installations - Major Release

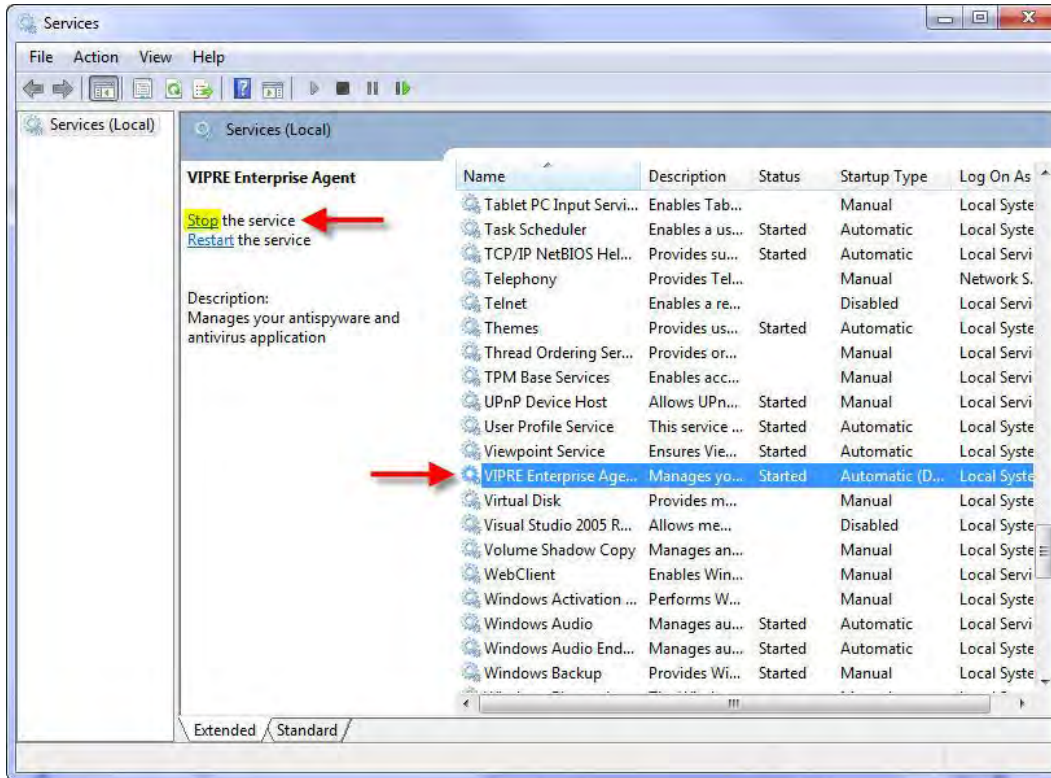
When updating your SolidWorks installation from one major release to another (ie. From 2010 to 2011), if you don't intend to have multiple releases of SolidWorks installed on the same machine then it is typically recommended to uninstall your previous version of SolidWorks before installing the new version. Alternatively, you could upgrade the existing installation to the new version.

Updating Existing SolidWorks Installations - Minor Release (Service Packs)

To update the service pack of the SolidWorks installation, you can download the installation files from the SolidWorks Customer Portal.

When attempting to update the SolidWorks service pack, you should be logged in as the Administrator on that computer.

Before the modification of the SolidWorks installation begins, any anti-virus or anti-spyware should be completely disabled using the Services menu. This can be accessed by going to Start-> Run, type "services.msc" (without quotations). Once the Services window has opened select each instance of the anti-virus program and stop it.



After downloading the necessary files from the SolidWorks Customer Portal, you can launch the Installation Manager, continuing through the steps to modify the individual installation. Once the service pack update has completed, restart the computer and proceed.

Updating Home Use License (HUL) Installations

Companies that own a network seat of SolidWorks can apply for a Home Use License (HUL). HUL seats for SolidWorks are typically installed and activated in the same way as a standalone seat of SolidWorks. A HUL serial number does typically not allow direct downloading of files from the SolidWorks Customer Portal (for installation/upgrade purposes). To get around this then, you can use the "Download Only" option to download without being prompted for the serial number. If you *are* prompted for a serial number during the *download* phase, then you should enter your *network* serial number. Once you have downloaded the files and wish to start the install/update, you can again use the HUL serial number.

Upgrading SolidWorks package from one package to another

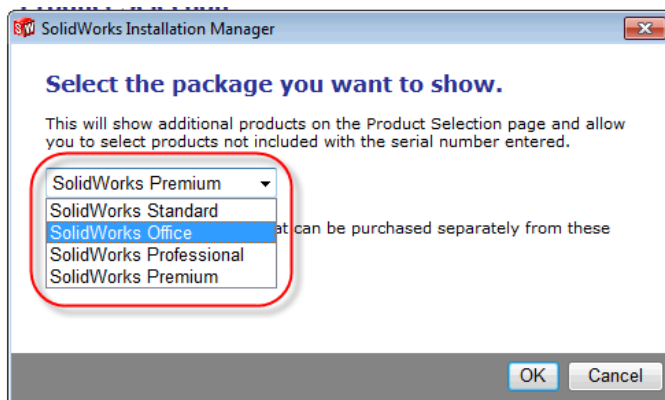
If you have just purchased an upgrade, i.e. SolidWorks to SolidWorks Premium, to apply this to existing installations is very easy and straight forward by following these steps.

1. Start by ensuring SW is not running.
2. Ensure you disable your anti-virus service; see page 2 for instructions how this is done.
3. Go to **START > Control Panel > Programs > Programs & Features** (Add/Remove Programs in XP) > locate the SolidWorks item, right-click on it, select **Change**.
4. The SolidWorks Installation Manager will appear.
5. Pick the **Modify** option and select Next:



6. The screen showing your serial number(s) will appear, leave this as-is and select Next.
7. SolidWorks will then connect online to the activation server and will “see” your new product package entitlement and will automatically select it.
8. Click Next to acknowledge the new product and have the additional components installed. Once completed, reboot your machine.
9. You are now ready to begin using your new SolidWorks package upgrade.

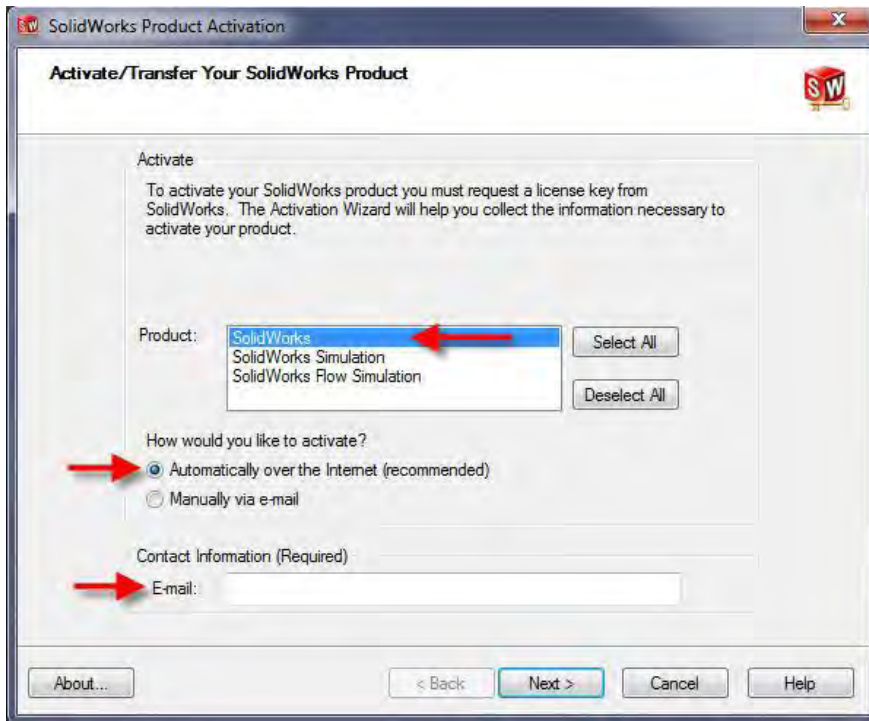
NOTE: *if the machine does not have Internet access, then it will NOT automatically “see” the upgrade and you’ll have to manually select it, and then proceed with step 8:*



SolidWorks Activation

Activating a SolidWorks Standalone License

Each standalone SolidWorks product (doesn't use network licensing) must be activated to make use of it. Each seat will typically have a maximum of 2 activations. These 2 activations are typically meant for a machine at the office and at home, or a desktop and a laptop; to make use of both activations at the same time would violate the license agreement. Using the Activation Wizard, you can contact the SolidWorks activation server directly or via email to activate or transfer an existing activation. When launching SolidWorks for the first time, the Activation Wizard will typically launch automatically. If it does not launch, go the **Help-> Activate Licenses**. Simply choose which product to activate, and the activation method – if the computer has an active internet connect then you would typically choose **“Automatically over the internet”**.



You can then follow the steps to complete the activation.

Activating a SolidNetwork License Server

Beginning with SW2010, the SolidNetwork License Server is activation based. After installing the license server software you must activate the product to identify that specific computer as the licensing server. There can only be a single licensing server activated for any SolidNetwork serial number.

*** IMPORTANT** – The activation process should always be done on the physical server. Using a Remote Desktop application can cause the computer ID to change which will cause the activation to fail.

To activate the server, launch the SolidNetwork License Manager. If you are not immediately prompted to activate, then, from the Server Administration tab choose **Modify-> Activate/Reactivate a software license**. Follow the steps to complete the activation. If the licensing server has internet access then you would typically choose to activate “**Automatically over the internet**”. If you have entered multiple network serial numbers, use “Select All” to transfer all of the product licenses.

Transferring a SolidWorks Standalone License

To transfer a license from one computer to another (i.e. if you are uninstalling SolidWorks) you must first transfer the license from the existing computer (**Help-> Transfer Licenses**), followed by activating SolidWorks on the new computer (**Help-> Activate Licenses**). Simply follow the steps to complete both of these steps and the SolidWorks license will be transferred to the new machine.

Transferring a SolidNetwork License Server

For each network serial number, there can only be a single computer activated as the licensing server. If you wish to change this licensing server, you must first transfer the activation *away* from the existing licensing server.

To transfer the server, launch the SolidNetwork License Manager. From the Server Administration tab choose **Modify-> Transfer a software license**. Follow the steps to complete the transfer. If the licensing server has internet access then you would typically choose to transfer “**Automatically over the internet**”.

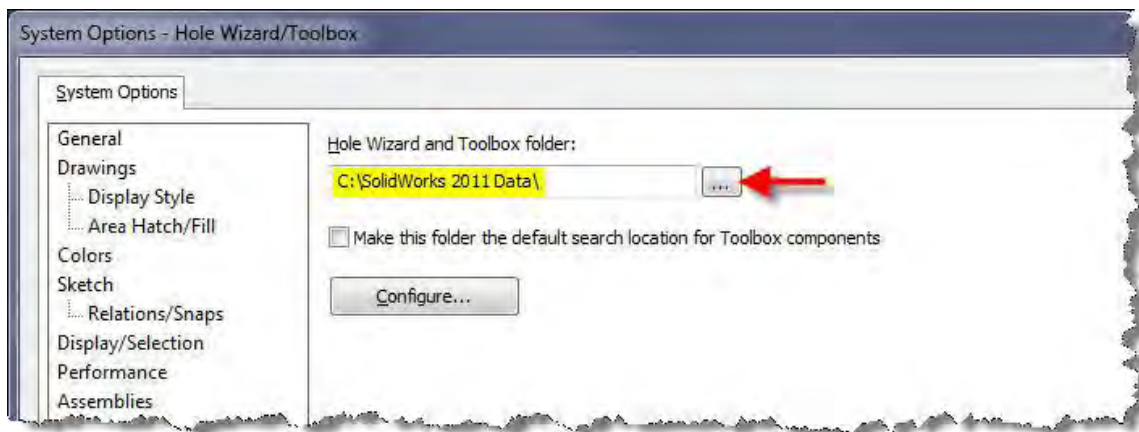
If you have entered multiple network serial numbers, use “Select All” to transfer all of the product licenses.

The Toolbox Database

Custom Database Location

The SolidWorks Toolbox can be located on a local machine for access by a single SolidWorks client, or it can be located on a network location for access by multiple SolidWorks clients. The default location for the Toolbox is “C:\SolidWorks Data”.

If you wish to create a network Toolbox, you can simply copy this entire SolidWorks Data folder to a network location. After having moved the Toolbox, it is important to update SolidWorks to reference the correct Toolbox location. You can do this by going to **Tools-> Options-> Hole Wizard/Toolbox**, then clicking the button to point to the correct Toolbox location.



Toolbox Permissions and File Attributes

To ensure proper functionality of the Toolbox, it is important to ensure that the proper permissions and file attributes are assigned, especially for network based Toolboxes where there may be multiple users.

For permissions, each user that requires access to the Toolbox should have **Full Control** to the entire SolidWorks Data folder (and all sub-folders and files).

It is also important to ensure that the file “**SWBrowser.mdb**” (located within “**\SolidWorks Data\lang\English**”) does **not** have a Read-Only Windows attribute. Typically, the entire “**\SolidWorks Data\Browser**” folder is set to have a Read-Only Windows attribute; this is acceptable.

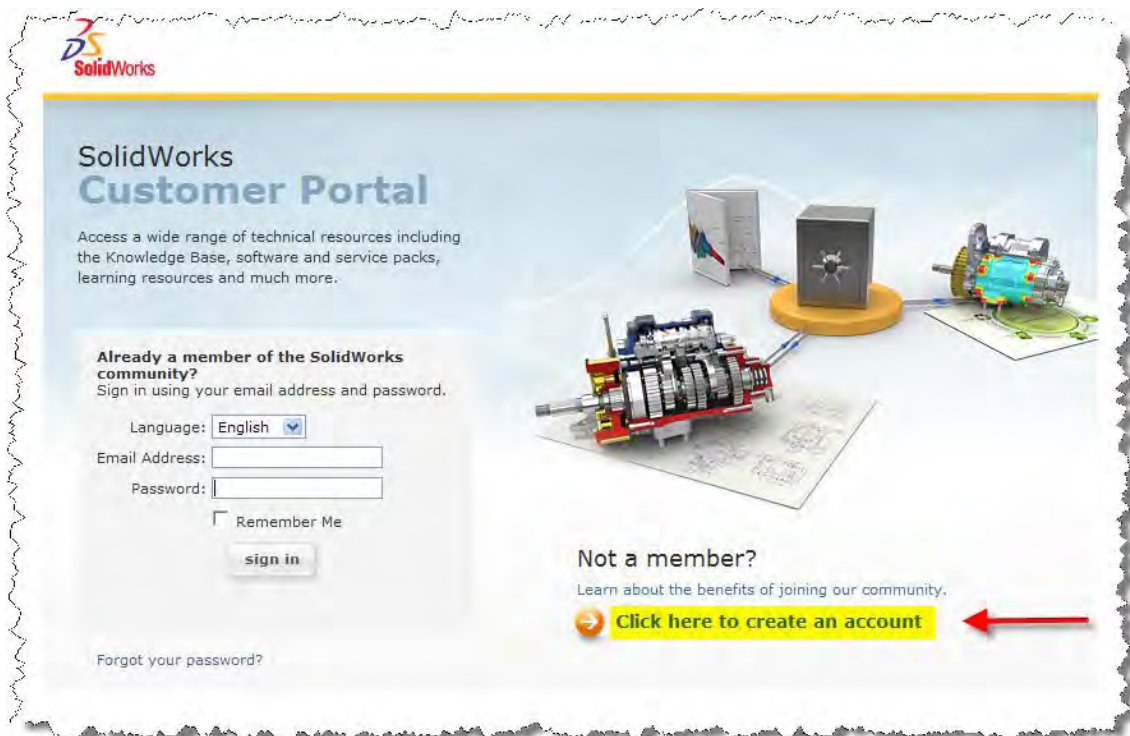
Using the SolidWorks Customer Portal

If you are enrolled in the SolidWorks Subscription Service program, then you will be able to use the SolidWorks Customer Portal with full access. The Customer Portal provides access to software service pack updates, a Knowledge Base for troubleshooting, user forums, etc.

The SolidWorks Customer Portal can be accessed via <http://customerportal.solidworks.com>. At this page, you can login using your account email and password.

Creating an Account

If you haven't yet created a Customer Portal account, you can create a new one using the link “**Click here to create an account**” on the right side of the page. Note that you will need your SolidWorks serial number to create an account. Simply follow the steps, including the creation of a login (your email) and password.



Unlocking the Customer Portal

After you have created your account and logged in, there may be a “lock” symbol next to some of the items on the page. This indicates that your Customer Portal account has not been completely registered. To register the account, click **Register My Products** from the Customer Portal homepage.

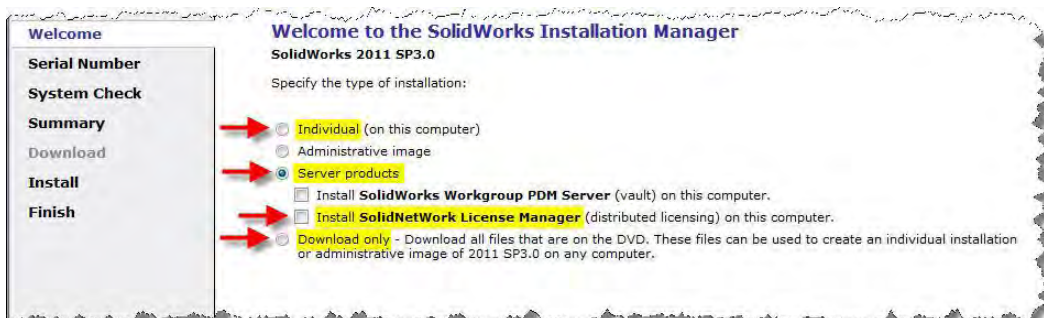


You can then follow the steps to register the account, ensuring to **select the latest version of SolidWorks** (i.e. 2011) as the product that you are registering. After you have completed the registration procedure, the account should be fully functional after logging back in.

Downloading from the Customer Portal

The Customer Portal can be used to download service pack updates, individual smaller installs (such as the SNL License Manager), *and* the entire SolidWorks installation fileset which can be used in place of a DVD or to burn a new DVD.

There are several options when downloading using the Installation Manager from the Customer Portal. Typically, the best option would be to choose the **“Download Only”** option as these files that are downloaded are the *complete* SolidWorks files that can be used to a) burn an install DVD, b) install SolidWorks, or c) upgrade the SolidWorks service pack. This option is particularly useful when multiple machines need to upgrade or be installed, due to the fact that after being downloaded this full fileset can be transferred to any computer that requires it. Alternatively, you can choose **“Individual Installation”** to download and install for this specific computer only. Finally, if you wish to download just server products (such as the SolidNetwork License Manager) you can choose the option for **“Server Products”**.



To download these files, login to the Customer Portal and click “**Downloads and Updates**” followed by choosing the version of SolidWorks that you wish to download. The first file that will be downloaded is the Installation Manager; upon running this file the Installation Manager will launch and you will be able to choose your download options.



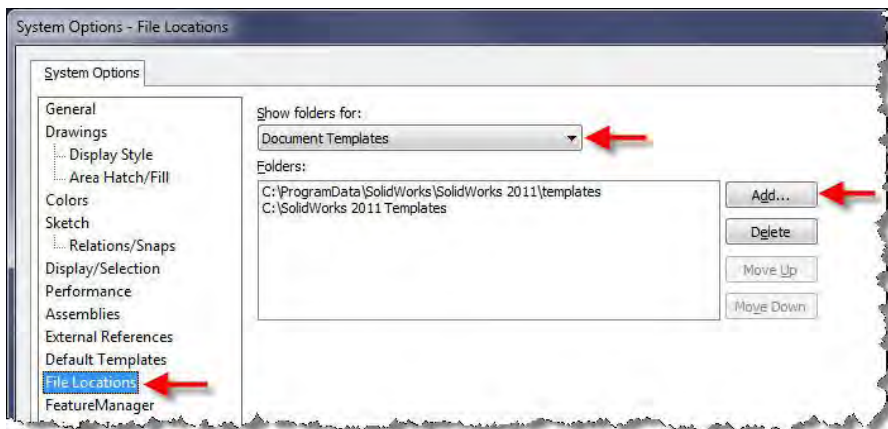
After the files have been downloaded (the default location is ~My Documents\SolidWorks Downloads\) if you need to launch the SolidWorks Installation Manager, you can do this by launching the “**setup.exe**” file from within the root download directory.

Saving Document Templates to Custom Locations

It is often necessary to create your own customized document templates and drawing sheet formats. The default SolidWorks save locations for these files is within a directory that should be deleted if SolidWorks is fully uninstalled properly. If you have not created a backup of your templates, they will also be deleted. It is because of this that it is typically advisable to create a unique folder location (outside of the SolidWorks install directory) for these files to be saved in to prevent future data loss.

When saving a document template (go to **File-> Save As**), upon choosing the template file type from the drop down list, the save location automatically changes to the SW default template location. This is typically "**C:\Documents and Settings\All Users\Application Data\SolidWorks\SolidWorks 2011\templates**". You can then change the save location to a custom location, such as "**C:\SolidWorks 2011 Templates**".

After saving the template, you must modify the SolidWorks options to actually reference the new template location. Go to **Tools -> Options-> File Locations**, choosing Document Templates from the drop down list. You can then click Add and browse to your new template location.



After adding the new document template file location, click OK to accept this change. If prompted to add this location as a search location, you can typically click **Yes**.



To use this new template, go to **File-> New**, and the new template location will be listed as a separate tab within the New SolidWorks Document dialogue.

A similar procedure can also be followed to save a drawing sheet format and then update the Sheet Format file locations within SolidWorks.

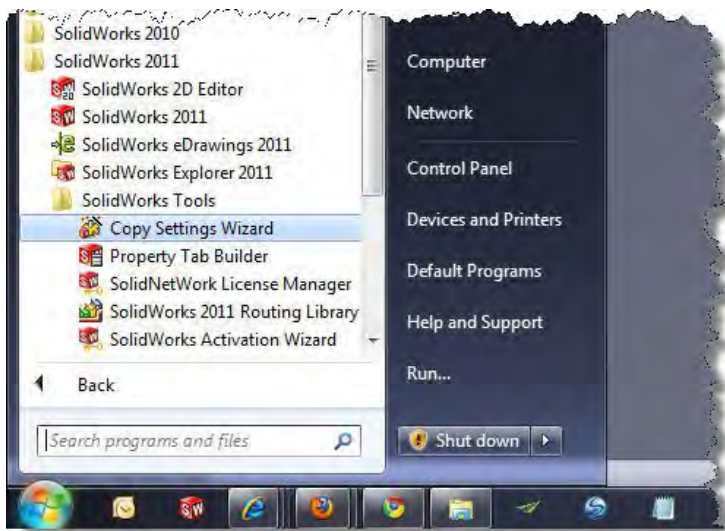
It's also possible to save document templates and drawing sheet formats to network locations. This can then allow multiple computers to access this network location, thereby ensuring that all SolidWorks users are utilizing the exact same templates.

Using the Copy Settings Wizard

SolidWorks makes extensive use of the Windows registry, including saving individual user customizations such as file locations, toolbar settings, hotkeys, etc. Unfortunately, in some cases files within the registry can become corrupted, which requires the deletion of entire registry folders. This will result in all of a user's customizations being lost, which can be time consuming and frustrating to recreate. It is because of this that is highly recommended to backup your SolidWorks settings using the Copy Settings Wizard as soon as you have created your customized settings.

Saving your SolidWorks Settings

To use the Copy Settings Wizard, with SolidWorks closed go to **Start-> All Programs-> SolidWorks-> SolidWorks 2011-> SolidWorks Tools-> Copy Settings Wizard**.



After launching the program, choose the option to **Save** your settings. After selecting which options to save, click Finish. This will create an .sldreg file which contains all the settings information.

Restoring your SolidWorks Settings

If you need to reload your SolidWorks settings from the file, simply double-clicking the .sldreg file will launch the Copy Settings Wizard. Follow the steps to **Restore** your settings – typically restoring only for the current user.

Note that the Copy Settings Wizard is also commonly used to propagate identical SolidWorks settings to multiple machines. The .sldreg settings file can be run on other machines to load the SolidWorks settings.