

Upgrading to SolidWorks 2012

More often than not, SolidWorks is deeply embedded into your IT system. This means that when it comes to making a change to the status quo, there is likely to be some wariness on the part of users. In line with the recent release of SolidWorks 2012, here are a few things to think about before you carry out an upgrade.

If any of this seems daunting at all, please contact our Technical Support team on 01223 200699 who will be happy to help. Good luck, and see you on the other side!

Templates and Sheet Formats

Often, your machine will be cleaned out before upgrading in order to keep improper data from being used. SolidWorks file templates and sheet formats reside in the default SolidWorks installation location that is usually deleted before installation. If not backed up, you might end up having to recreate the settings needed to make new file or recreate your title blocks in the sheet formats. This could easily be a day or two of work, so make sure you back up your templates and sheet formats in addition to your conventional SolidWorks files.

SolidWorks Settings & Toolbars

In the process of the system purge, some users remove the previous versions registry keys for SolidWorks. This will remove all the toolbars, file locations, and settings options you had been working with for the past year. To combat this, you should use the SolidWorks Copy Settings Wizard. This tool is installed with every seat of SolidWorks and is found in the SolidWorks Tools folder. This application will create a backup of your settings and paths so that you can quickly get back to your correct layout in your next installation. It may be wise at this point to take the time to create a default set of user settings that can be easily applied for new users or new PCs.



Toolbox

There are two ways to deal with upgrading to 2012 with respect to the toolbox:

1. Install a new "Solidworks Data" folder on your machine. If you are keeping a previous release installed, we would normally recommend you suffix this folder with the version number i.e. Solidworks Data 2012.
2. Upgrade the previous versions toolbox to the latest version. This is all done through the installation manager, and will run a utility to upgrade the database and toolbox parts to the version you are installing.

Please Note:

If you have any customisation on your toolbox at all (i.e. have added additional descriptions, part numbers) then we would recommend that you upgrade via the second method.

Due to problems reported with customised toolboxes in 2012 SP0.0, the release of hard copies was delayed until SP1.0 – which you have in front of you now. The downloadable version has been available for some time.

SolidWorks Files

Once the upgrade is complete, there are a few different approaches to migrating data from one release to another. It is not always necessary to convert all SolidWorks files to the latest version; however, performance can be increased by upgrading your files.

Listed below are some considerations:

- **Opening Files in Older Versions**

Files cannot be opened in versions older than the version they have been saved in. Be sure to make a backup of all data before starting the conversion process.

- **Conversion 'On the Fly'**

Files can be converted when the user opens them in SolidWorks. Note that the opening time for each file will increase initially as SolidWorks converts it, after which it will decrease. Files will not open lightweight unless they have been saved in the current version.

- **Batch Conversion**

You can convert data automatically using tools supplied with SolidWorks. One of which is the 'Convert Files' tool in the SolidWorks Task Scheduler. This will convert all files in a specified folder to the current version. This can be carried out using several PCs with the 'Network Monitor' functionality.

- **Activate Configurations**

In the Convert Files tool, there is an option to activate all configurations in the files during the conversion process. If this is selected, the 3D body information for each configuration will be written to the file but this can increase file size if there are configurations that have not been activated before. This can be the case for files containing Design Tables, Toolbox files and Design Library components. If this option is not selected, when the user manually changes configuration at a later time there will be a short delay while SolidWorks generates the 3D body information for that configuration.

- **PDM Conversion Tools**

PDM solutions often have a utility to extract data from the vault, convert it to the current version and check it back into the vault. These are designed to convert the files without increasing the revision number. Solution S-020320 in the SolidWorks Customer Portal describes the utility with SolidWorks Enterprise PDM.

- **Read-Only Parts**

If the read-only parts and assemblies have not been converted to the current version of SolidWorks, they will not open in lightweight mode. It is common that shared files (e.g. Design Library and Toolbox) will be 'read-only' so it would be advisable to convert these to the current version then reset the 'read-only' flag. To avoid potential problems it is recommended that you convert these files to the current version and test that they function as expected. A backup should be made before conversion.

- **Test Environment**

When upgrading from one release to another, it is recommended that you set up a test environment and do some testing with a copy of the data before moving all users in the production environment to the new release. This allows users to identify any potential issues in advance.