

SUBJECT: FILE MANAGEMENT

KEYWORDS: DATA MANAGEMENT, FILE STRUCTURING

SOLIDWORKS FILE MANAGEMENT

The aim of this document is to provide the reader with an understanding of how the SolidWorks file structure and management works. It is loosely based upon the **SolidWorks File Management** training course.

SolidWorks File Structure

In order to properly manage SolidWorks file or use any PDM (Product Data Management) system in SolidWorks, it is important to know what is contained in a SolidWorks file and how they relate to one another.

A SolidWorks file is structured as a **single point database**. This means that when a part is used in an assembly, for example, the assembly purely references that part rather than copying the information to itself. It is this which keeps the file size of assemblies down. These references are known as **absolute** meaning they relate to a folder/ directory path on your computer or server, hence why it is vital NOT to move files from folder to folder without the use of a PDM system.

A SolidWorks file contains four sets of information:

Header: information about the file such as its format, file name, type, size and properties

Instructions Set: can be thought of as the FeatureManager Tree in a listed format, this is what is used to build the file

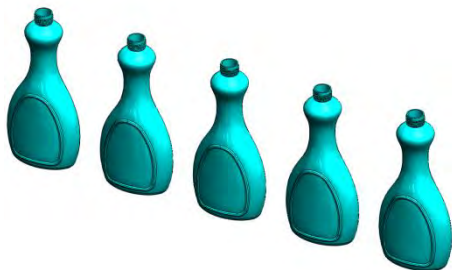
Database: The modelling kernel that build the files outputs a database that consists of the topology (shape) which we see in the form of the graphics of the model

Visualisation Data: This is the data used to preview models and the image seen on screen

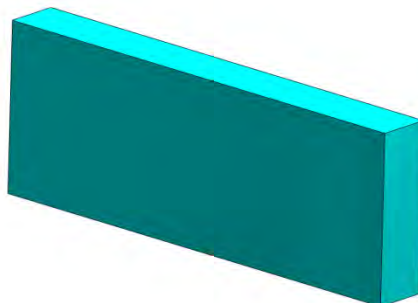
Knowing the structure of a SolidWorks file can help easily reduce the file size of a model using simple techniques.

The **Database** and **Visualisation** data contribute to file size and the more complex a model is (graphically) the bigger the Database and Visualisation data will be. An easy way to reduce this is to simplify the graphics.

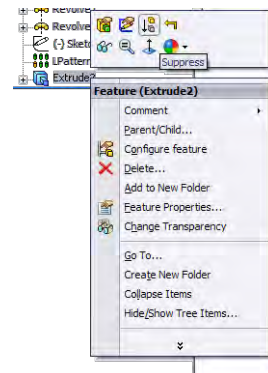
One way of doing this is using a technique called **Blocking a model**. All this does is creates a rectangular block that is extruded over the extremities of the model. This can be sent to a colleague at the reduced file size, opened and the block feature can be simply deleted leaving behind the original geometry:



Original File ~3MB



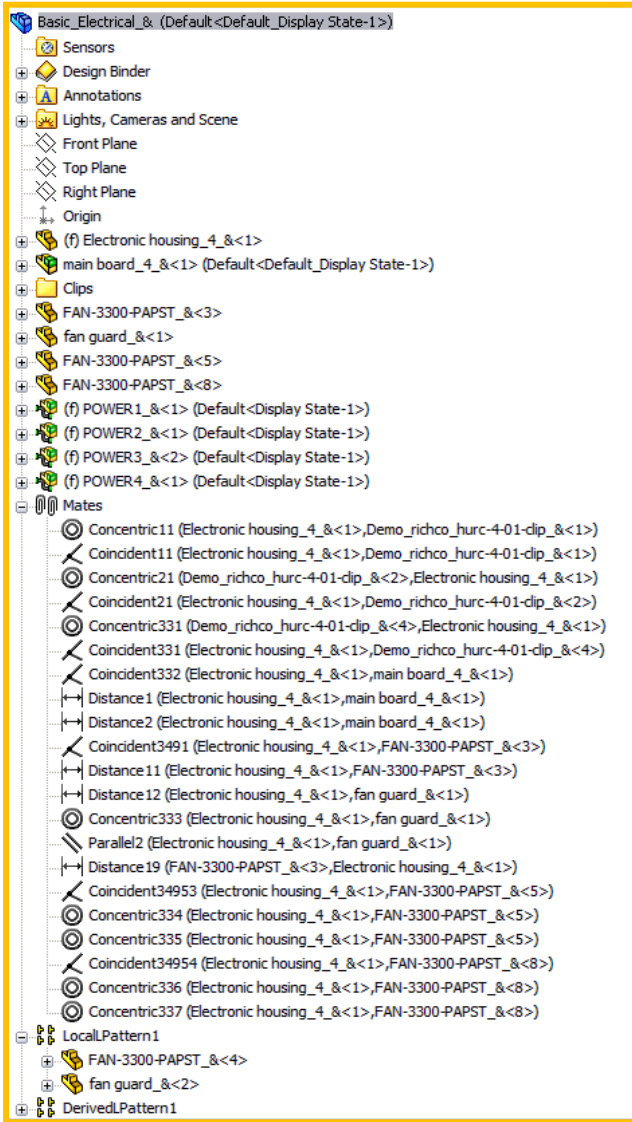
Blocked File ~ 1.5MB



Suppress/ Delete the feature on receipt

The other method of reducing file sizes is to suppress features that contribute the most to overall rebuild time.

Use **Tools> Feature Statistics** to diagnose the features in rebuild order (both % and seconds) and then suppress them. Not only will it speed up the general day to day working on the file, but reduce the file size. An even bigger file saver is to **suppress everything** in the tree, the file from above reduces to **350KB** when this is done!! This is because there is literally no **Database** and **Visualisation data** in the part but all the history is still kept in the **Instructions set**.



FeatureManager Design Tree

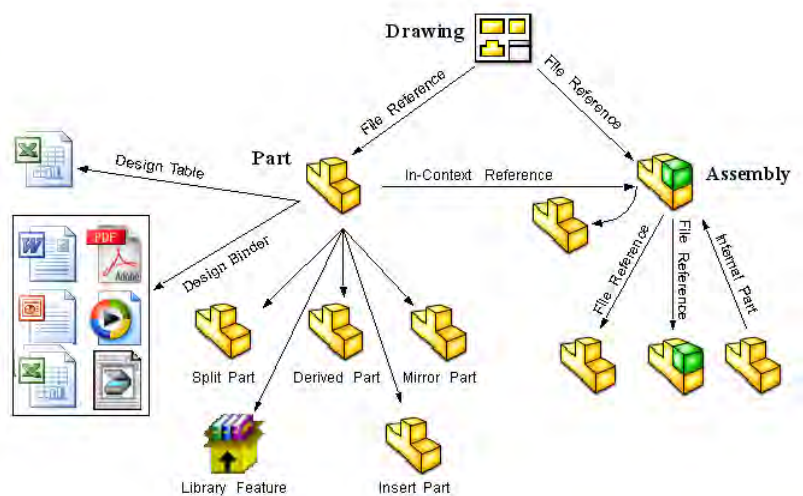
As well as showing you a visual representation of the order and chain of commands used to build a model, the Feature Tree is also ordered according to how the Instructions set build the model when opened.

1. Establish the reference geometry (Origin and Plane positions)
2. Load all the parts and sub assemblies into memory
3. Assemble the parts using the instructions in the mates list
4. Solve in context and assembly features such as patterns

An example of the SolidWorks file referencing structure can be seen to the right. Some of the references can be a link or an embedded file. On any assembly or drawing file you can view the references by using **File > Find References**. From here **Copy Files** can be selected which launches the **Pack and Go** function which enables you to copy all files and references to a new location.

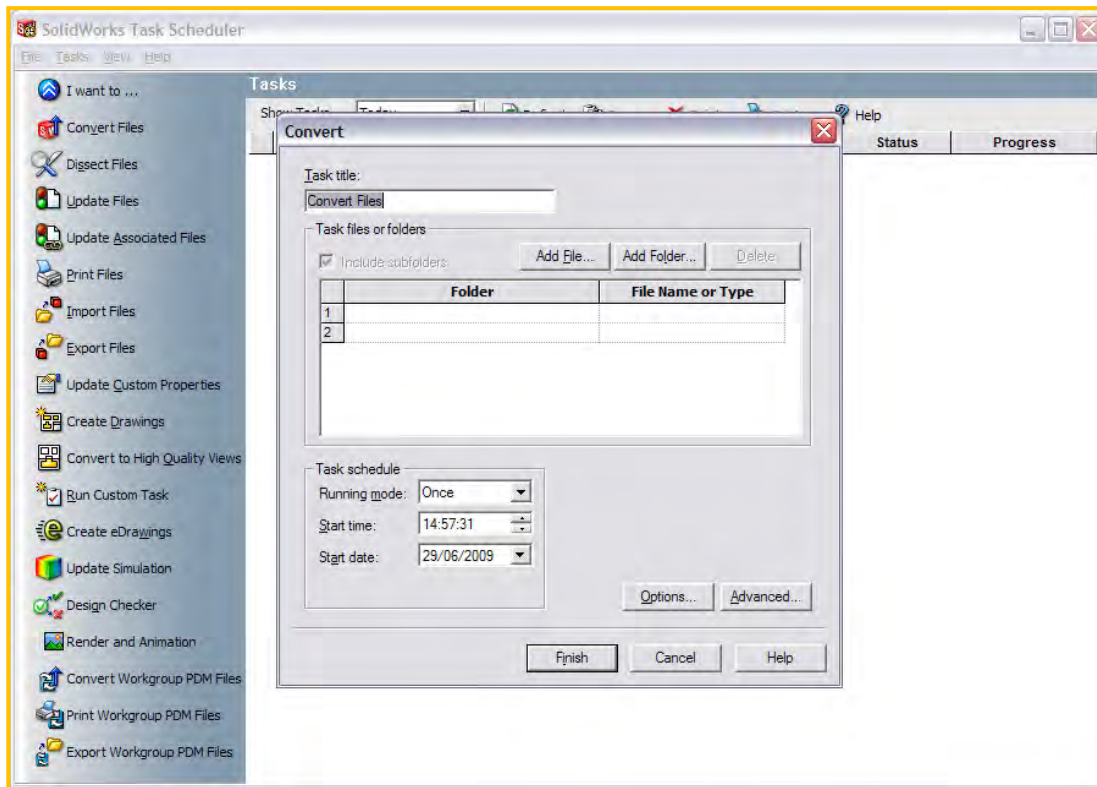
To find any **External References (-> symbol)** right mouse click the part name and choose **List External Refs.**

Tip: unique file names are the key to managing files in SolidWorks. Only one document of a given name can be open at any one time.



File Conversion

The SolidWorks file structure changes with each major release, which is the main reason why SolidWorks files are not back compatible. The file structure is only changed to the new release when the file is **saved** in the new version. This file conversion means, initially opening the file may take a little longer, to automate the conversion use the **SolidWorks Task Scheduler**, an add-in of **SolidWorks Professional (START> All Programs> SolidWorks 2009> SolidWorks Tools)**. Here there are a variety of tasks that can be ran when you are out of the office by specifying a folder where you want the action to take place.



RAM Resident opening of files

When a parent document is opened, all of the referenced documents are also loaded into RAM. You may have seen that when you make changes to a part it will update the loaded assembly. If you choose to close that part and NOT save, the assembly will still show the newer part, as it is loaded into RAM. The way to get this back to normal (without closing a reopening everything) is to use **Reload**. By selected the part that needs reloading and selecting **File> Reload** the file saved on disk will be copied into RAM overwriting the existing version.

Saving Files

- Save** copies the file in RAM to disk, leaving the RAM version open.
- Save As** copies the file in RAM to disk with a new name. The old version (that was in RAM) is closed without saving and any open documents that reference this file are updated with the new reference
- Save As Copy** copies the file in RAM to disk with a new name, leaving the existing RAM version (old name) open. No additional references are updated

Editing references on Open

References can be changed when a file is opened, i.e. change the part a drawing references. Choose **File> Open** and select the assembly or drawing document by left clicking once. Select **References** in the bottom right corner of the dialogue and you will be shown the current references. Double click the reference to change and browse for its replacement. The file name turns **green** indicating a change to the reference.

Back Ups and Recovery

SolidWorks provides methods of regaining lost information through backups and Autorecover files. That said it is always advised to **Save regularly and often** to avoid losing work, this may be even more apparent if saving across a network. Set the options through **Tools> Options> System Options> Backup/ Recover**

Backup Backups are additional copies of SolidWorks files optionally saved to an alternative location. The backup is the version before the last save. You can save up to 10 backups of each document and set to clear them out after a certain number of saves.

AutoRecover Automatically saves information about the active file which can be redeemed after an unexpected termination or crash. AutoRecover is different to AutoSave as the Recover tool can be set after a certain number of **rebuids** or a certain amount of **time**. AutoRecover information is only available after an unexpected crash where it will be available on the AutoRecover tab on the RHS of the screen.

Save Notification Provides an on screen prompt after a set amount of time of changes reminding you to save.

It is advised to set the locations for Backups and AutoRecover to a local directory instead of a network, as a network drop out could let to file corruption or "Fail to Save" issues.

File Properties (Metadata)

File properties are details about a file that helps identify it e.g. a descriptive title, author etc. They also provide additional search criteria to which you can find files. As well as Windows™ based properties, SolidWorks provides its own more suited to engineering such as revision, Checked By etc.

Classes of properties

Automatic- maintained by the application included date created, last modified date, and file size

Preset- properties that exist but need to be filled in by the user

Custom- defined by the user and applied to entire document (part, assemblies and drawings)

Configuration Specific- defined by the user but applied to one configuration only

SolidWorks Custom Properties- These are automatically updated by SolidWorks including the parts mass, material etc.

There are a variety of ways of adding properties to SolidWorks files which include:

Windows Explorer (RMB Properties on the file)

Custom Properties Tab in SolidWorks (**File> Properties**)

Design Tables within SolidWorks using the syntax **\$PRP@property**

SolidWorks Workgroup PDM during the check in process

SolidWorks Enterprise PDM defined by variables on data/ template cards.

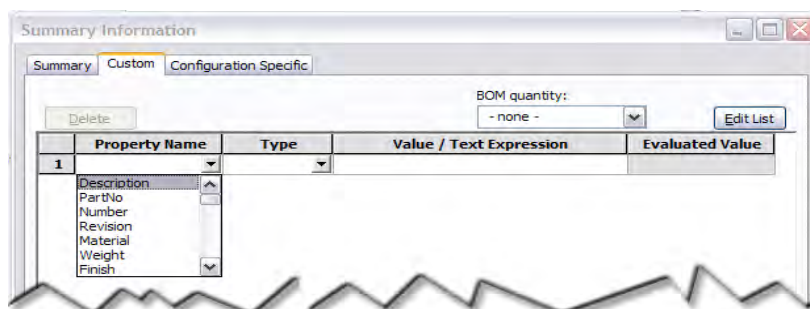
Properties can be used in a variety of ways:

Create parametric notes and annotations in parts, assemblies and notes. The notes will update as the property does

Advanced Selection and Show/Hide in Assemblies- can select parts based on the custom property value

Populate drawing borders and title blocks, as well as BOMs, and revision tables.

The list of properties available when selecting from the **Property Name** field from **File> Properties** is governed by a text file residing on your system. Check the location using **Tools> Options> File Locations> Custom Property Files**. If you want to reduce the list to only the properties that interest you, edit the text file and delete/ add rows as required (take a backup of original beforehand)



File References

Knowing the SolidWorks reference order is useful for determine how SolidWorks reads in files. It takes the reference purely based on file names using the following hierarchy order:

1. **RAM-** if a file of the correct name is already open, it will use this file, reiterating the need to use unique file names
2. **Directories set in Tools> Options> File Locations> Referenced Documents-**setting these additional directories can speed up finding files if you specify the location you normally save files to
3. **The last path you opened a file from-** When a parent document is opened, SW will search the same location for the children

4. **The last path the system used to open a file-** this applies if the system automatically opened a reference document last
5. **The path where the reference document was located when the parent document was last saved-** in case the parent document was moved. This excludes the drive (C:, D:) location
6. **The path where the reference document was located when the parent was last saved with the original drive destination-** as above but with the absolute path
7. **If still not found, a message appears asking you to browse for it yourself.**

Once re-linked to a missing document (moved or renamed using Windows Explorer) using step 7, the next save operation will update the new references.

Changing References

There are many occasions where you need to exchange a part or sub-assembly for a replacement

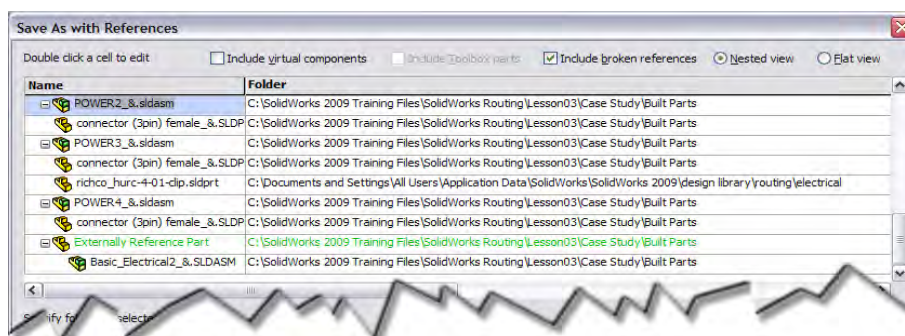
- File> Save As** changes the references of the open assembly to the newly resaved part/ sub-assembly with the new name
- File> Replace** Allows substitution of a selection or all instances of one component with an alternative
- File> Open>References** Changes the reference before the document is opened (page 3)
- File> Save As> References** When a parent document is saved, the children can be saved to new names and so new documents
- SolidWorks Explorer** Has the ability to rename files and tell any reference documents about the name change to ensure the reference is updated without error. The files do not need to be open to perform this function

External References

There are more considerations needed when resaving assemblies that contain external references. When we perform a **Save As** on an assembly that contains external references we receive a message as follows:

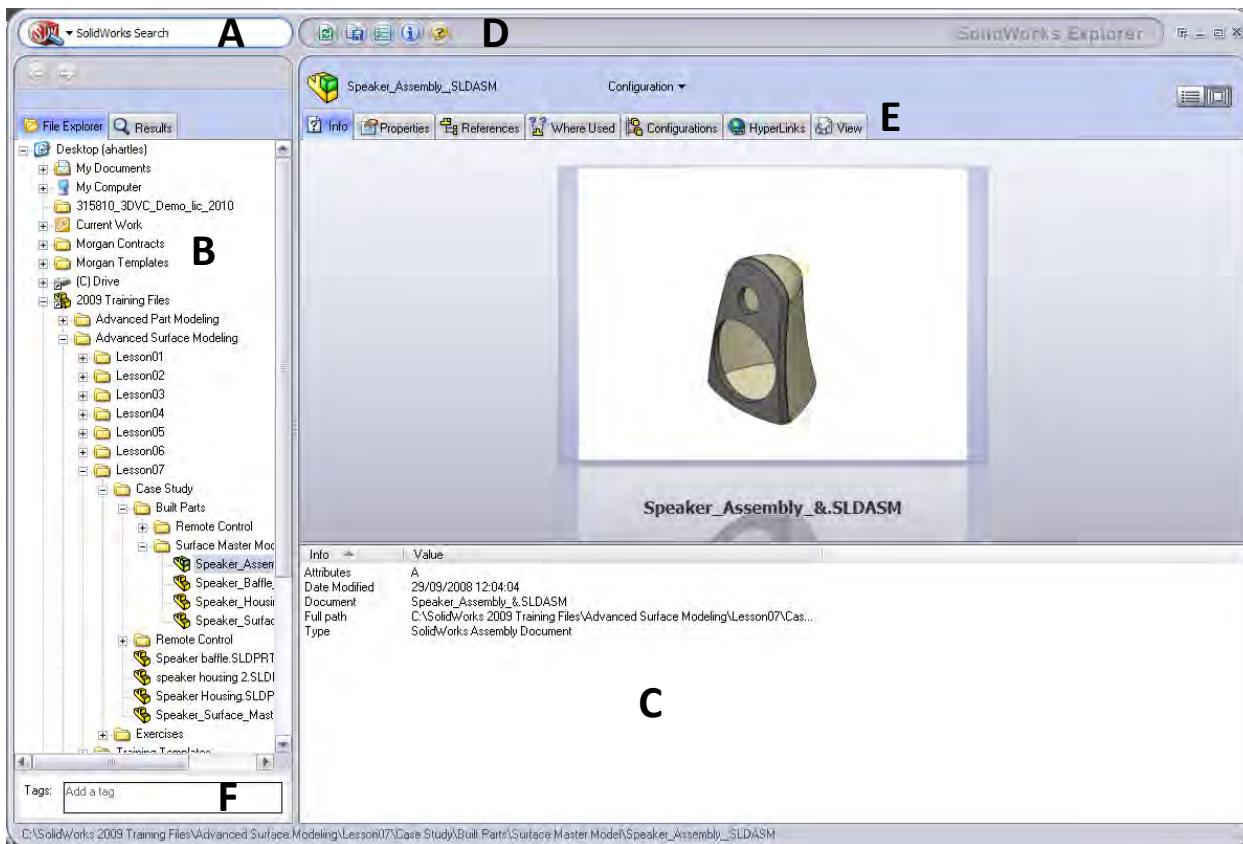
There are parts that have features defined in the context of <assembly path and name>. If you select OK to continue this operation these features will not update in the context of the newly created assembly. It is highly recommended you consider saving new versions of these parts now which will update in the context of this newly saved assembly. **To do this, select Cancel, select References from the Save As dialog box, then input new names for those parts that you want to update in the new assembly.**

The highlighted section informs you of the way to get around this problem. The reason why resaving the assembly does not take through external references is because it is unwise to have two assemblies externally referenced to the same part, this can create **Circular References**.



SolidWorks Explorer

SolidWorks Explorer is a program included with SolidWorks used to modify unopened parts, assemblies and drawings. It can be started as a standalone application or can be launch from within SolidWorks – **Tools> SolidWorks Explorer***



- A- SolidWorks Search Assistant- easy method to search files locally and over the 3D Content Central web directory (uses Windows Desktop Search)
- B- File Explorer Pane- shows the content of your computer is a Windows Explorer type fashion
- C- Document Information Pane- information about the active document
- D- Standard toolbar for setting options and help topics
- E- Property/ Operation tabs- for finding references, where a document is used, its configurations, hyperlinks, and an e-drawings 3D representation
- F- Tags- add a tag (searchable property) to the selected document

File Management Options

When a SolidWorks document is selected from the File Explorer Pane, four icons will appear, allowing you to **Open the document, Pack and Go the Assembly, Rename the document (as well as updating the references) and Replace the Component within an assembly.**



*Be on the lookout for SolidWorks Explorer within Windows Explorer. When you RMB on a SolidWorks file in Windows Explorer you should find a SolidWorks entry in the list, this allows you to rename without the need to open the SolidWorks Explorer software. *Further information on File Management can be found under the SolidWorks help topics and training courses specifically targeted towards Product Data Management detailing the Workgroup PDM and Enterprise PDM suite of products*