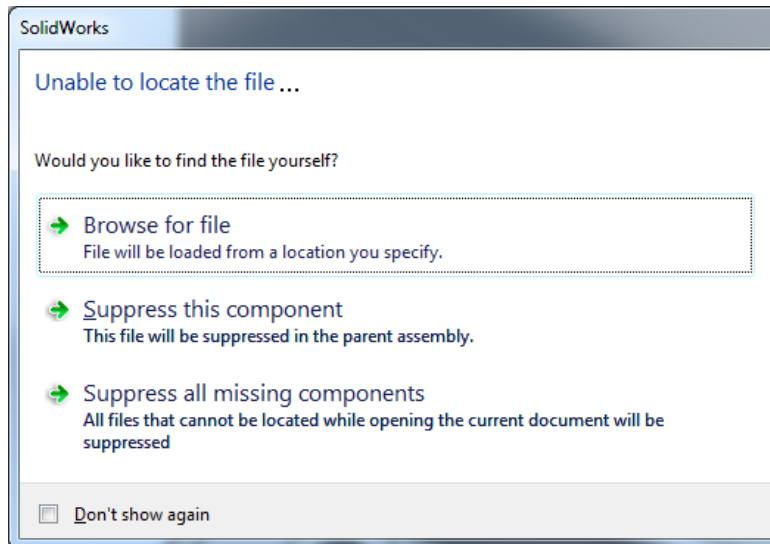


File Management Using Solidworks Explorer

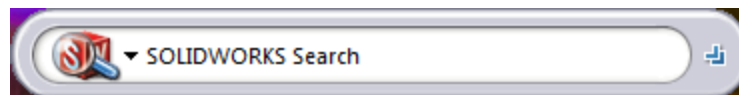
If you have ever experienced the headache of finding an error similar to the one below when opening an assembly or file only to find that something is broken, then you may have accidentally altered something using a method that would not be considered “best practice”. However, fortunately, these kinds of mistakes can be fixed, and a method can be implemented such that it never happens again.



Changing File Names

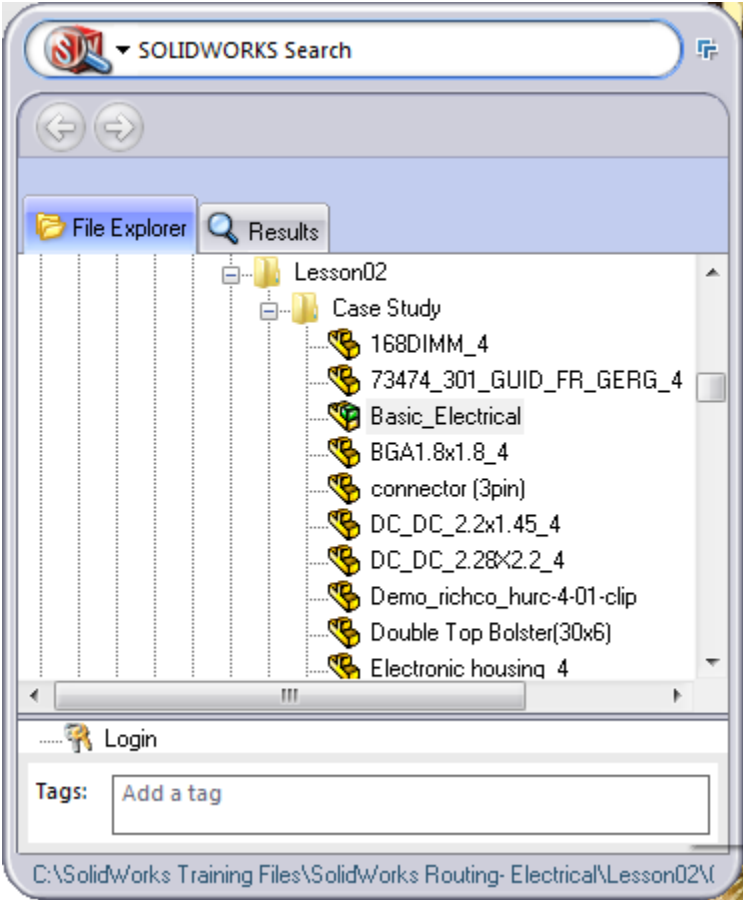
Generally this sort of problem occurs when files are renamed without the use of Solidworks tools such as Solidworks, Workgroup PDM, Enterprise PDM, or Solidworks Explorer. The references stored within a Solidworks file that tells the Solidworks application where to look may remain the same, but if file names are changed or files are moved, the link to a referenced file may break.

The best alternative to a PDM tool for managing your files manually is Solidworks Explorer where you can not only view References but also see where files are being used (Where Used, essentially what other files reference the file your are viewing). The Solidworks Explorer icon can be

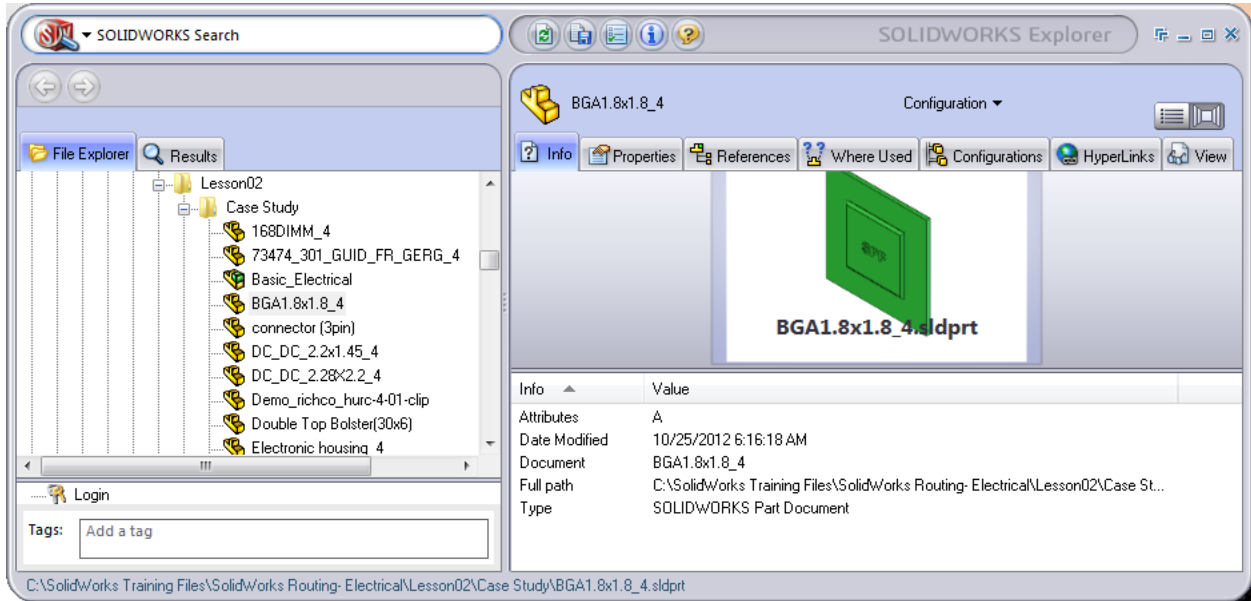


found on your Desktop (if you have not deleted it) or it can be found in Windows 7 under Start > All Programs > Solidworks > Solidworks Explorer. The Solidworks window upon startup should look something like this:

Click on the double arrow button to expand the window and expose the File Explorer tree. The File Explorer tab displays the folders and documents on your computer. If Solidworks Workgroup PDM Contributor is added in, the vault view appears at the bottom of the pane.






Navigate to a Solidworks file. Click on that file and the right hand side of the Solidworks Explorer window should pop out, showing you information about the file as well as other options and commands.



The Solidworks Explorer window should look something similar to the one shown here. Simply clicking on a Solidworks file in the tree should show a toolbar that helps you perform some of the main tasks which Solidworks Explorer is purposed to do, and it should look like this:



Here is a summary of some of the task functions that can be done through the pop out toolbar:

	Pack and Go	Gathers all related files for a model design (parts, assemblies, drawings, references, design tables, Design Binder content, custom and default decals, appearances, and scenes, and SOLIDWORKS Simulation results) into a folder or zip file.
	Rename	Renames one or more selected documents and updates all the references.
	Replace	Replaces a selected part or assembly document and updates its references.

Now that you have been introduced to Solidworks Explorer, make sure to always use this tool when changing names or references to ensure that no “Out of Context” or broken reference errors arise in the future.