

Multiple versions of Solidworks and "Version Management"

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Introduction:

Multiple versions of Solidworks on the same system can be helpful. However, there is a restriction that once part/assembly/drawings files are opened and saved in newer versions, they can no longer be opened with older versions of Solidworks.

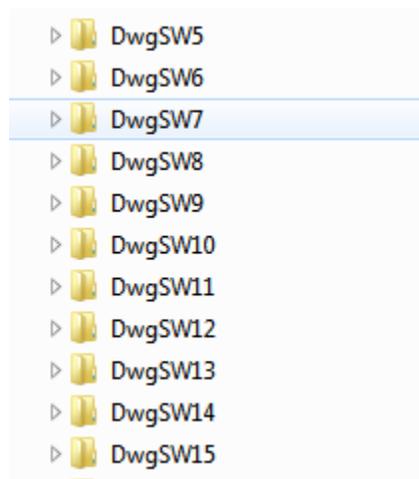
In addition to issues with backwards compatibility of SW files, there is a significant problem with library parts. If Library Part A is used in SW 2012 and then for a different project in SW 2013, Part A will no longer "open" in SW 2012.

Another common conflict is when a project is "moved" from an older version to a newer version. If suppliers are restricted to the older version of SW, and if the files are "moved up" to a new version, the supplier will not be able to open or use the files.

The solution for each of these situations is to incorporate methods of isolating every file for every required version. "Version management" can be accomplished through a number of methods. This paper presents techniques that have been successfully implemented over a number of years.

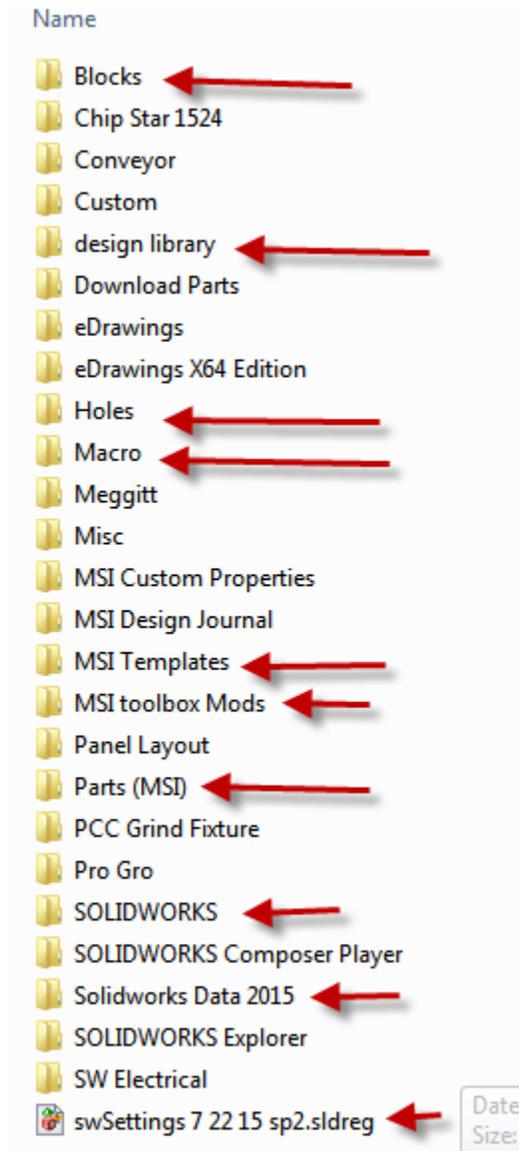
Version management file structure:

MSI has used a simple file structure for version management. The foundation of this approach is to create a specific directory for each version. (The names below are not required. What is important is to be able to identify which directory is identified with the specific version of SW).



When the new directory for the latest version is created, a number of sub-directories should be copied from the previous version. Not all of these sub-directories are necessary but they have been copied to

maintain consistency. Because there will be multiple copies of the same file, some additional HD space will be required. However, the space is not significant because the size of most of these directories is relatively small. The extra space is a minor sacrifice to maintain consistency.



Blocks, design library, Holes, Macro, MSI Templates, are copied as a general practice (as shown from the top down). The MSI toolbox Mods directory probably doesn't need to be copied. This directory is used to copy any original toolbox files before they are modified.

The "Parts (MSI)" directory is used to store all MSI custom library parts. This is one of the more important directories that need to be copied so that library parts that are used in the current version of SW will continue to be available in the previous versions of SW.

Note: If the file name or structure of any library parts are changed, the link to any parts used in previous projects will be broken. Ie. SW will not be able to locate the file and they will need to

be manually located when first opened. Therefore, if any changes in the file structure or file name is desired to make the library more user friendly, one of the best opportunities to make these changes is when a new version is installed. The exception is that the links for any components that are used with projects that are copied from previous versions will need to be manually re-established.

The next two directories are the "Solidworks" and the "Solidworks Data" directories. These will be automatically copied into the current directory if properly designated during the Solidworks installation process. If the location of the Solidworks installation is not re-directed, the Solidworks files will be saved under the default directory on the C drive. There has been no problems using the default directory locations, but for consistency and easier backup purposes these two sub-directories have been defined under the SW version "working" directory for the last few years without any problems.

Transferring custom settings to the latest version:

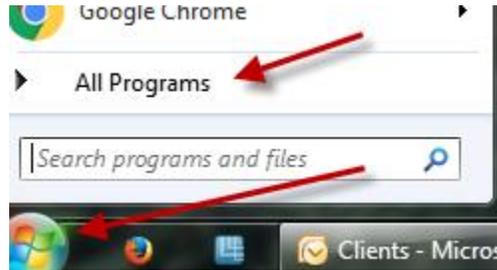
Once installed, properly registered, and successfully started, the first step is to copy the custom properties over to the new version.

There may be a better way to accomplish this and while it has not been officially recommended, MSI has successfully used the "Copy Settings Wizard" to move custom settings from a previous version over to the new version.

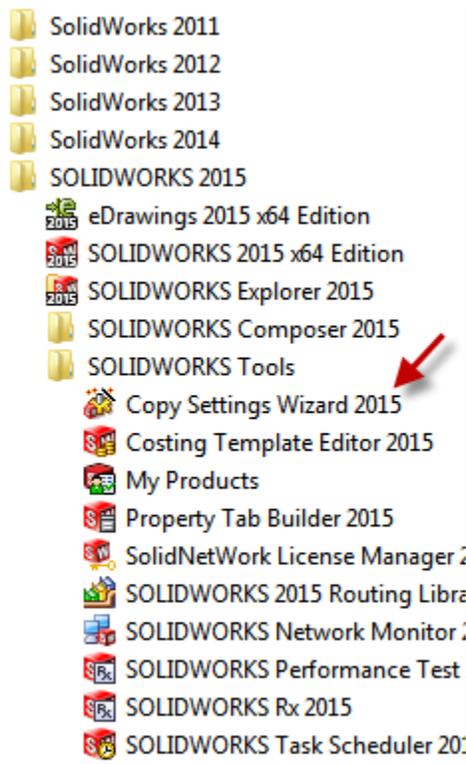
MSI has also found advantages with periodically using the "Copy Settings Wizard" and recommends saving an icon next to related version of Solidworks . Once copied to the desktop, MSI has renamed the icon as is shown below:



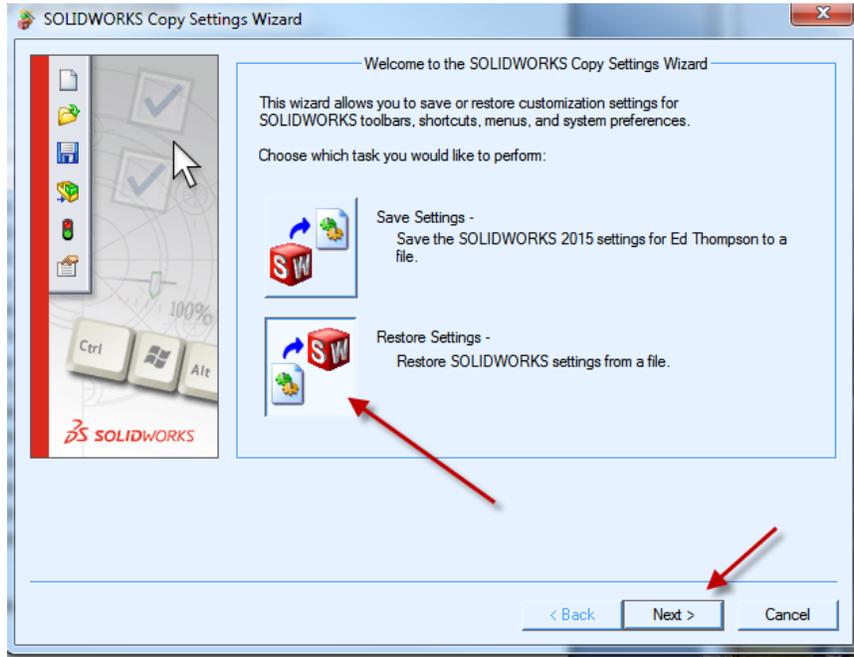
It is relatively easy to add the "Copy Settings Wizard" icon to the Desktop. The Copy Setting Wizard can be found by first selecting the "Start" button and the by selecting "All Programs".



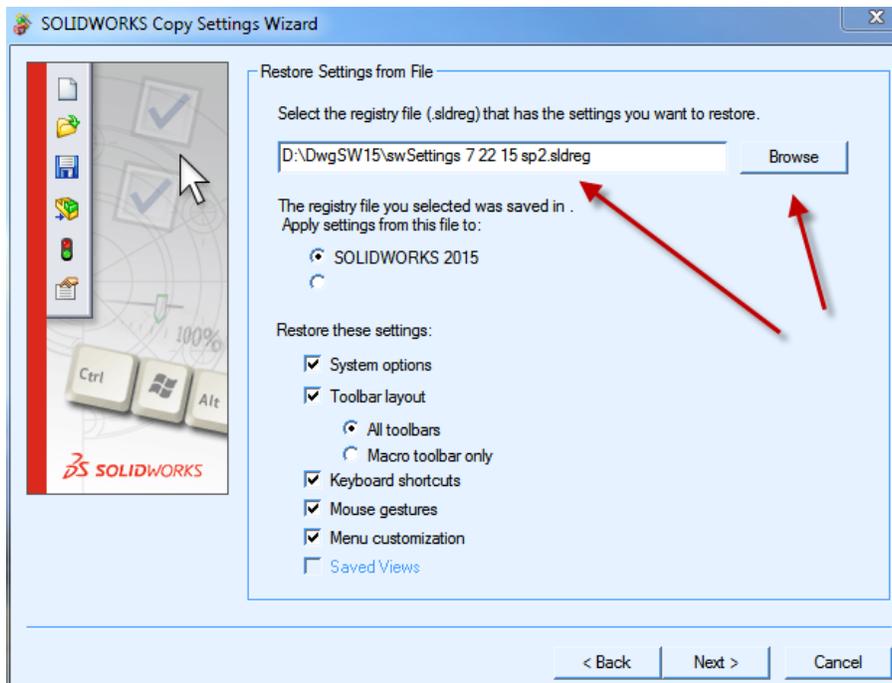
The directory structure should be displayed (as shown below). Under the latest version of SW, and under "Solidworks Tools" the "Copy Settings Wizard" should be available.



When the Copy Setting Wizard is started, "Restore Settings" should be selected before clicking on <Next>:

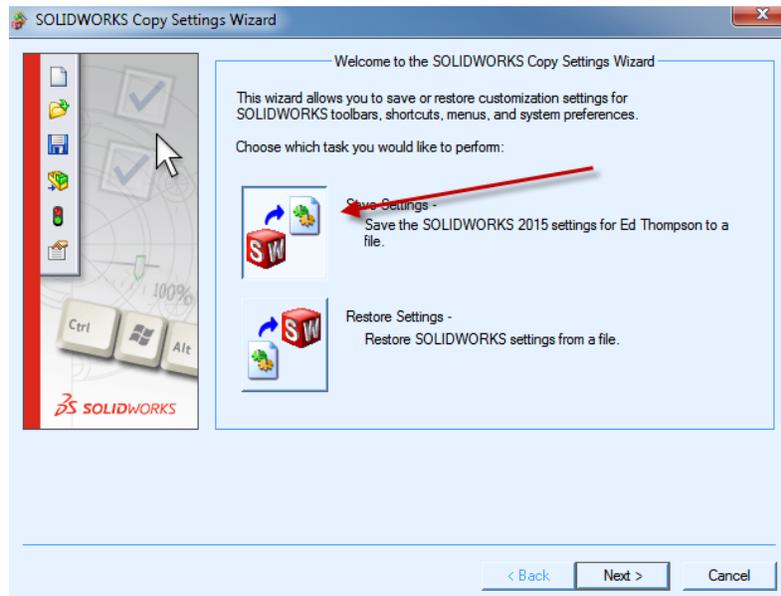


The first time that the Copy Setting Wizard is ran for a new version there will be no saved file. The "Browse" button should be selected and the latest file for the previous version should be selected.

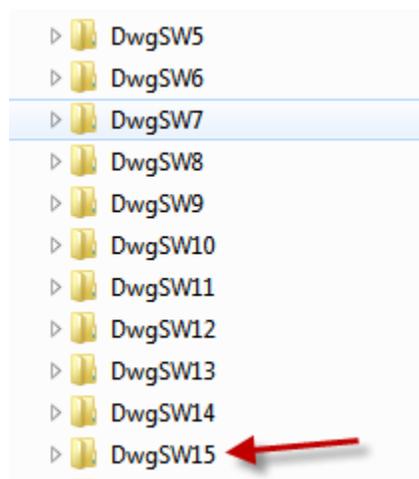


There may be some warning messages. It has been acceptable to ignore these messages in previous versions. Not all of the customization will transfer over, and some of the settings will need to be manually corrected, but most of the custom settings will be saved with previous versions.

When the settings are transferred over, SW should be opened and some of the customization should be checked to see if they translated properly. If all looks "close", close SW and re-run the Copy Settings Wizard. However, this time "Saved" should be selected before clicking <Next>:



The new save setting file should be saved for the current version at a file location that can be easily recalled. MSI has generally used the "version" directory for the location of the version settings file:

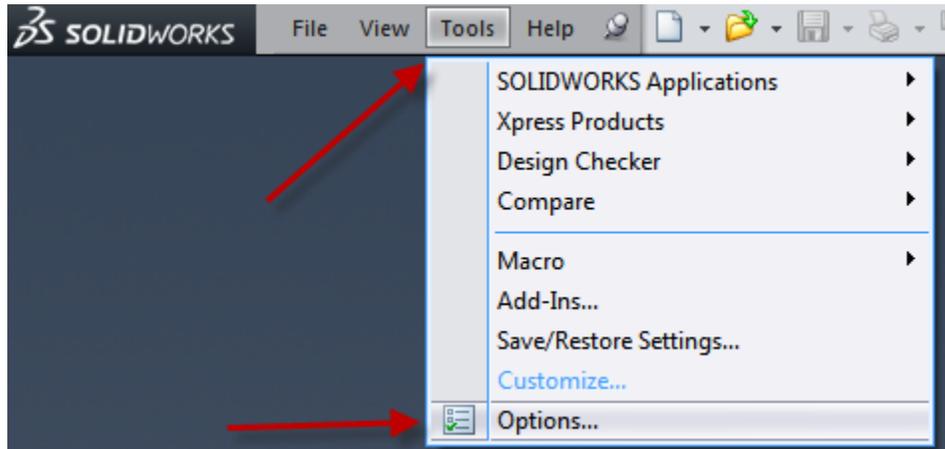


When the new location is located, selecting <Finish> will complete the process and a saved settings file will be established for the new version.

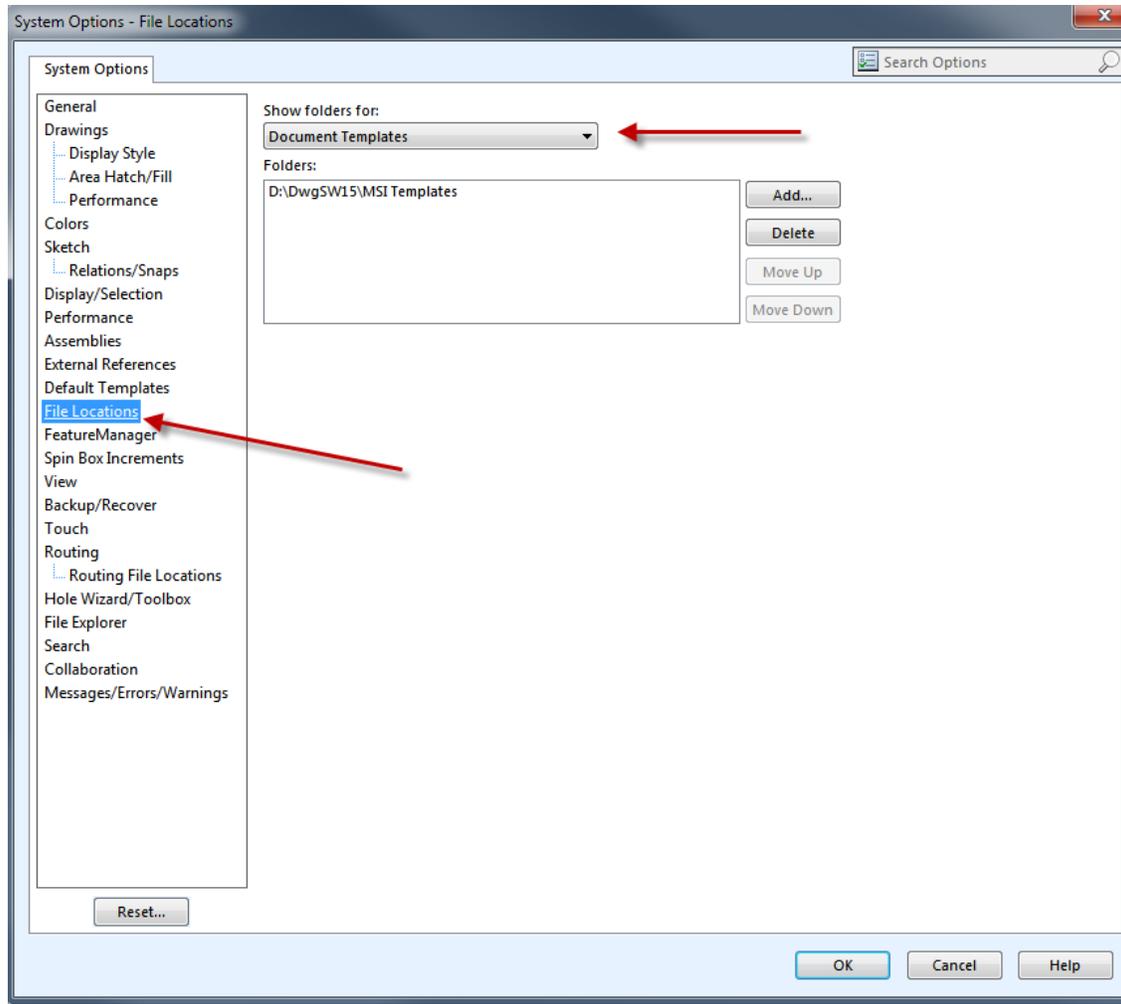
Setting up Solidworks working directories:

While many of the custom settings will be transferred, the file locations will all be for the previous version of SW. Therefore it is necessary to edit each of the settings. This is a little tedious but once completed, the new version will usually operate quite nicely.

To begin to setup the file directories, start the new version of SW and open the "Options" directory:



Under the System Options, select "File Locations". There will be a drop down box, (shown below) where each of the file locations definitions can be updated.

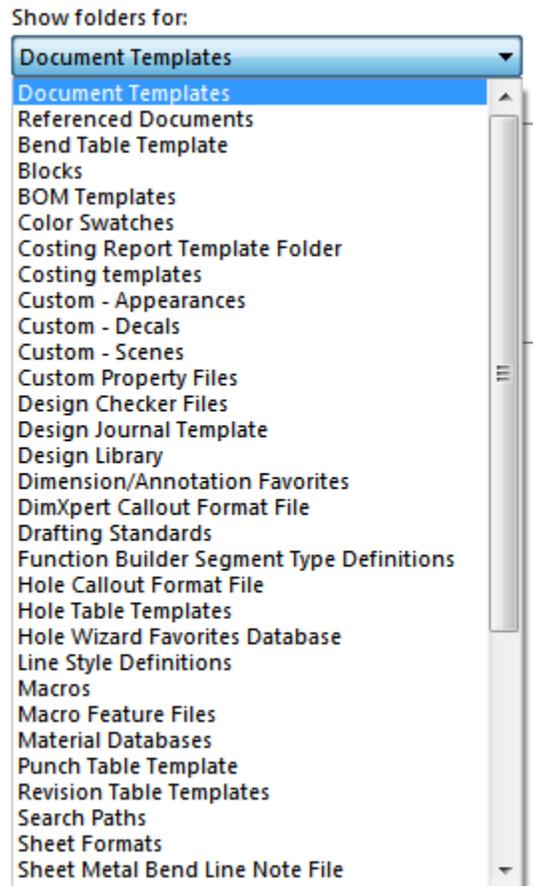


For most of the locations, the new file can be added before the old one is deleted. It has been helpful to add the new file location by using the existing location as a "pattern". Most of the locations will be identical except for the basic version directory change.

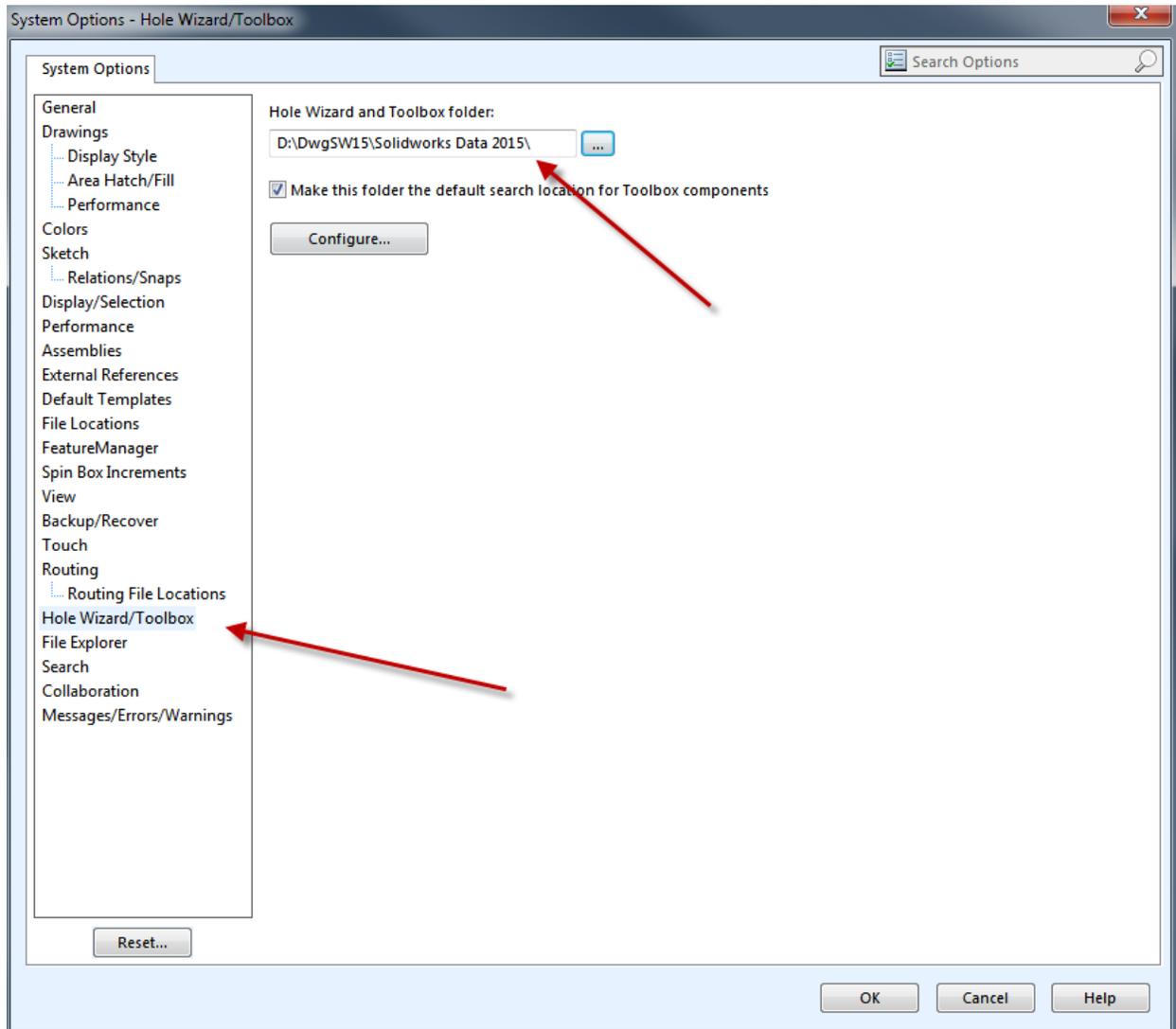
Some file defections will not allow more than one directory at a time. In these cases it may be necessary to record the "basic" file structure.

Once the proper location is defined the old location should be deleted.

Yes, it is tedious to edit each of these file location definitions but this only needs to be done once for every new version and can usually be done in 20 to 30 minutes.



When all of these file locations have been corrected, the Hole Wizard/Toolbox location needs to be edited:



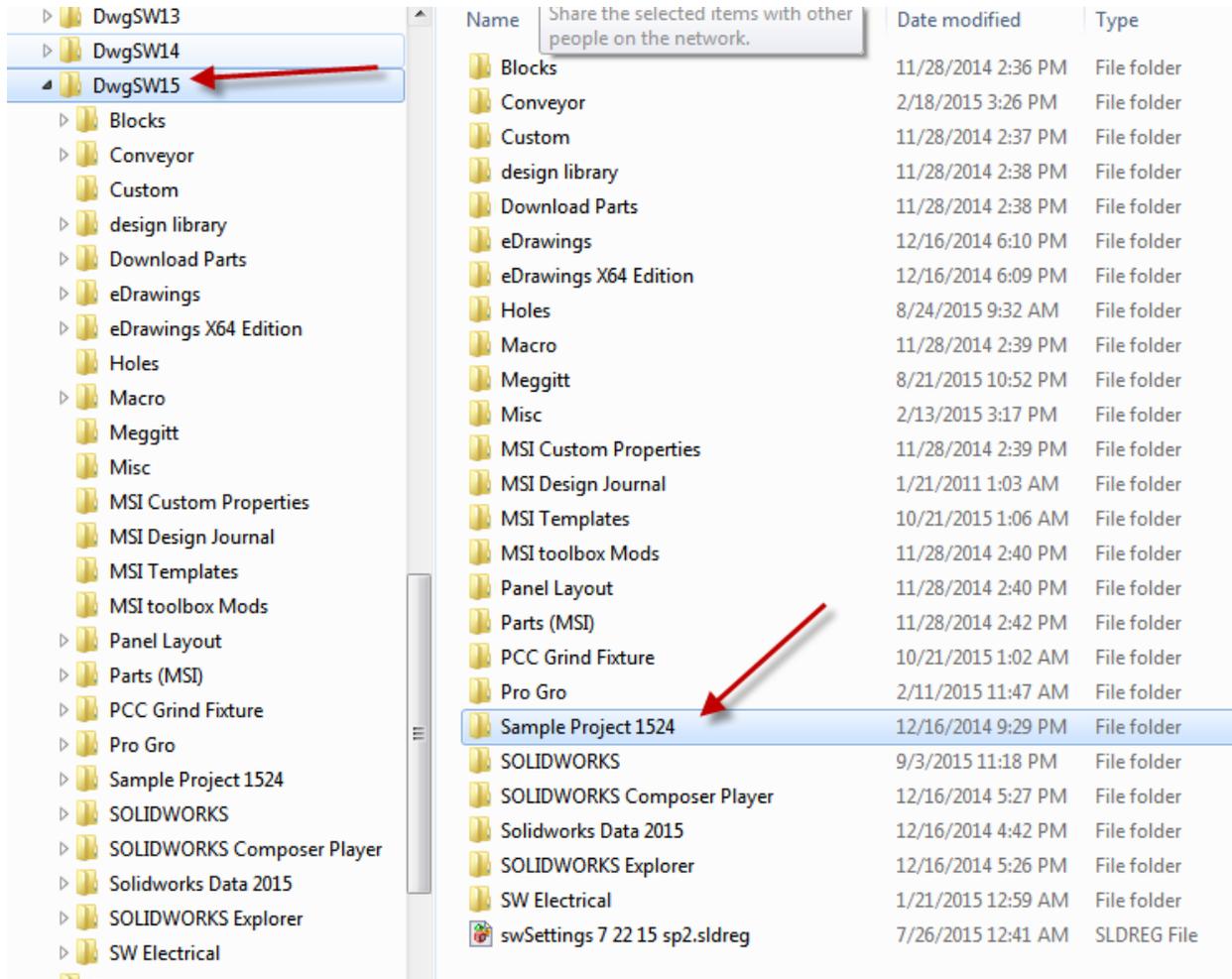
If the Hole Wizard or Toolbox files have been customized, these modifications can be updated in the new version by selecting the "Configure..." button (shown above) and following the appropriate procedures.

Savings the results:

It is now highly recommended to exit SW (if required) and to Save the settings, using the Copy Settings Wizard. At this point, the new version of SW should be fully functional with most if not all of the custom settings from the previous version.

New Projects:

The directory/files for new projects can be saved in many places. MSI has successfully used the main Solidworks directory to add project file directories:



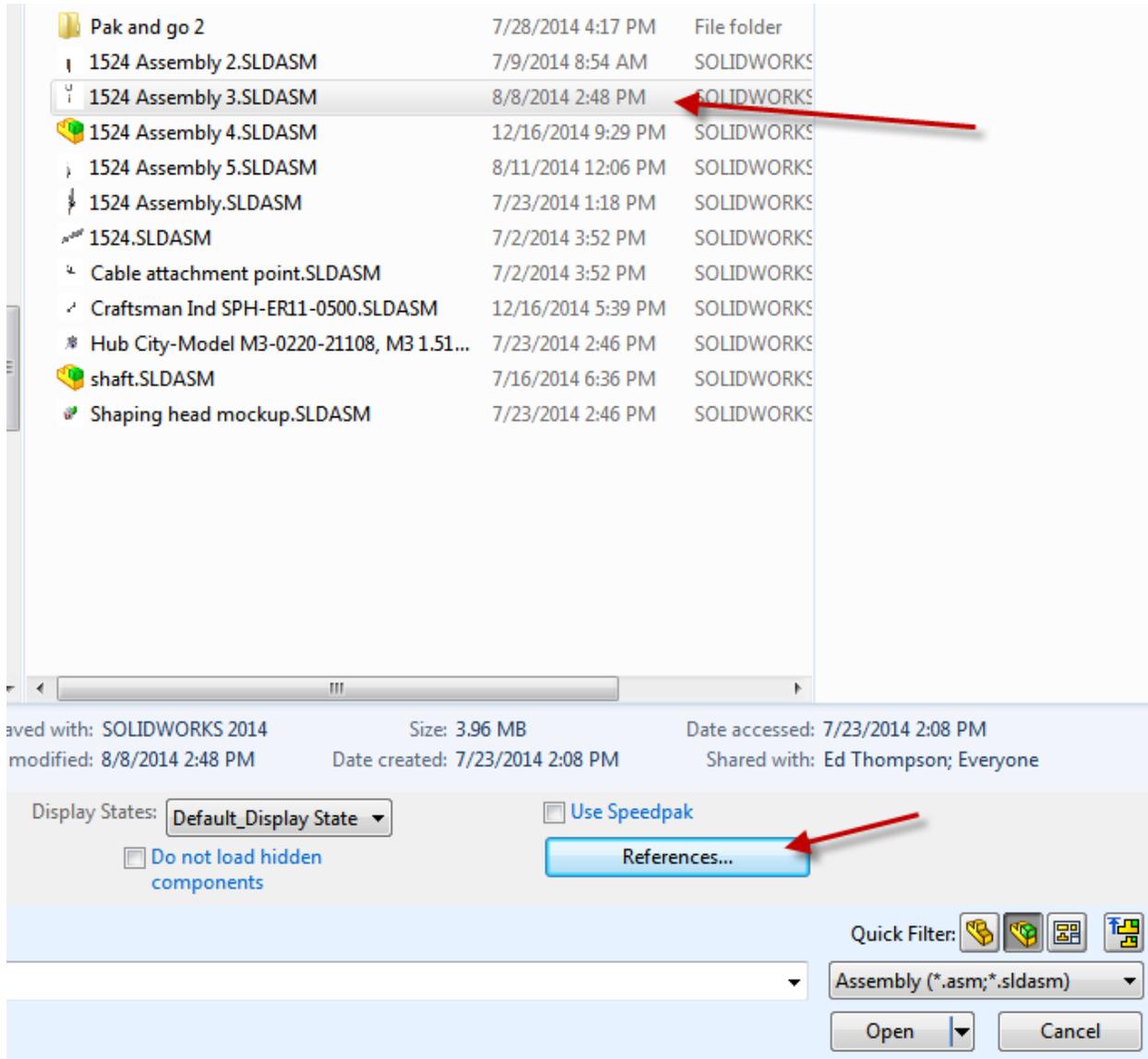
Shown above is a "Sample Project 1524" directory where all of the files and sub-directories are stored for that particular project. (Note: The number "1524" is the MSI project number for this project. Adding the project number to the directory name aids in finding projects files in the future).

One of the advantages of this directory structure is that the primary SW version directory (such as DwgSW15 above) can be backed up and easily restored. Following these guidelines, everything is stored in a "single location".

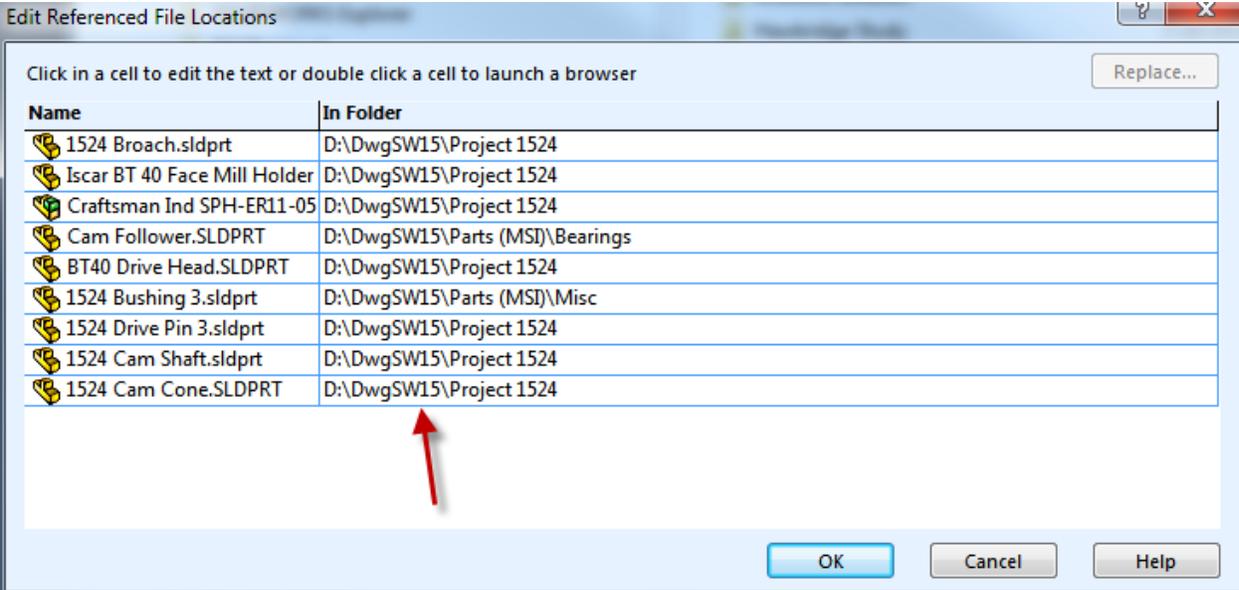
Moving project up to newer versions:

For new projects everything should work very smoothly without any conflicts with older version project files.

One area of caution is when a project directory is copied from a previous version directory. The References can be quickly checked by right clicking on the SW file and then by selecting the <References...> button (shown below).

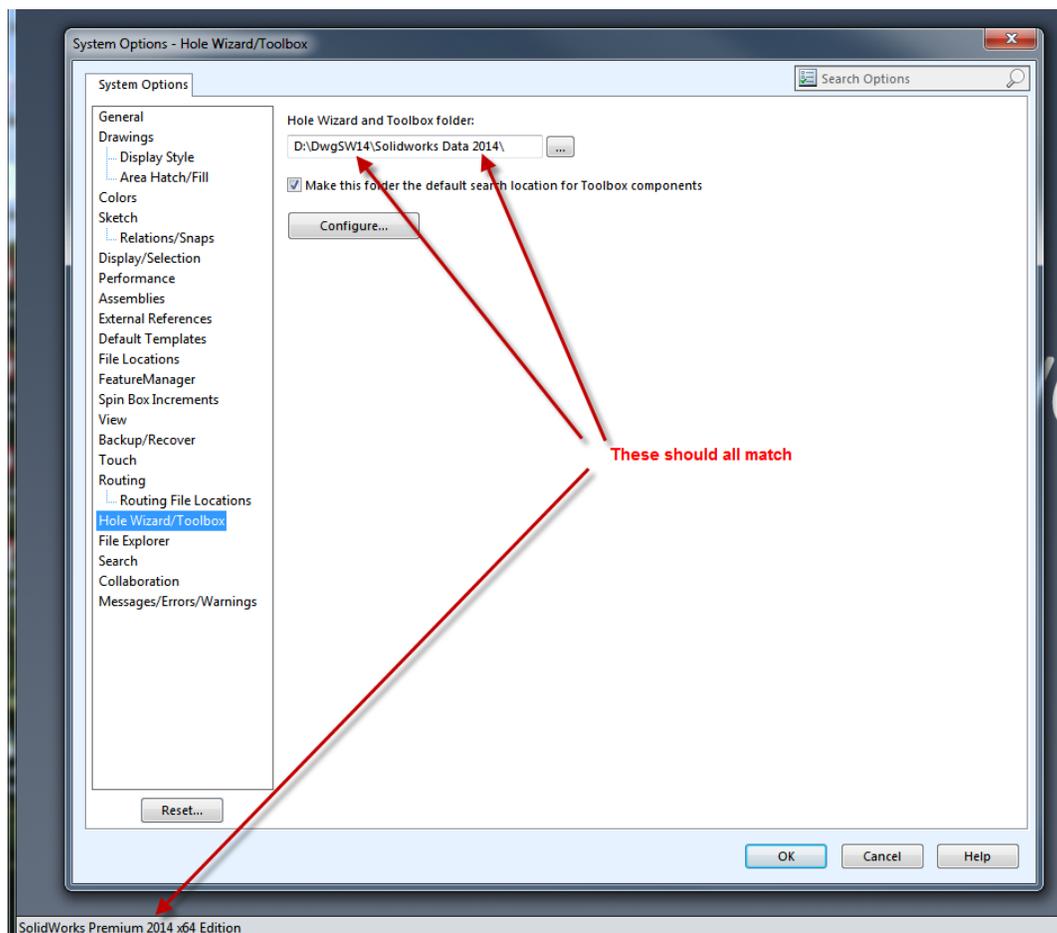


The following dialogue should be displayed. If each of the components folders are located under the correct SW version directory, there should be no problems. If any of the references are incorrect, they need to be edited before opening the file.



Setup Verification:

The ultimate verification is to open/ edit/ save and test parts and files. However, a quick test that is fairly accurate for determining if the desired version of SW is setup properly is to inspect the Hole Wizard Wizard/Toolbox file location. This inspection can be easily performed by selecting Tools/Options.../ and then selecting the Hole Wizard/Toolbox to see if all is consistent as is shown below:



The SW version "path", the version of Solidworks Data and the version of SW should all be the same. If not, then the errors need to be corrected. If reasonable corrections do not solve the issues, perhaps technical support should be contacted.

Note: If everything is set up properly, two different versions of Solidworks can be opened at the same time. This can be very helpful when working with projects that are in different SW versions.

Summary:

Because the files from different versions of Solidworks are not compatible, care must be taken to isolate and manage all of the files. The process of how to setup the file structure that MSI has successfully used for multiple versions of Solidworks has been described.