



***SOLIDSOLUTIONS***  
SUPPORTING EXCELLENCE

SOLIDWORKS PDM

# Recommended User-Controlled Settings

## INTRODUCTION

This document is a compilation of best practice recommendations for settings that we believe every user should be aware of, and set themselves on 'day 1' when initially beginning to use SOLIDWORKS PDM. The use of the term 'user-controlled' (in this context) refers to a variety of local environmental settings that affect the behaviour of the PDM Client, and specifically those settings that cannot be centrally managed or enforced globally by a vault administrator. In other words, these are the key settings that users are individually responsible for on a 'per-machine' basis.

## OUTLINE

### Display Options

- Card Preview (Turn Off)
- Set focus to Active Configurations (Personal Preference)
- Show full UI in SOLIDWORKS preview (Turn On)
- Include SOLIDWORKS Simulation results in Check-In (Turn Off)
- Full Row Select (Personal Preference)

### Windows Folder Options

- Navigation Pane (Turn On)
- Details display mode (all folders)
- Hide extensions for known file types (Turn off)

### Column Displays

- Contains (customise)
- Where Used (customise)
- Check In/Out (customise)
- Change State (customise)

### Within SOLIDWORKS

- PDM Add-in Options>'Prompt to checkout...' (Turn off)
- SOLIDWORKS System Options>'Don't prompt to save...' (Turn on)
- Toolbox (customise)
- SOLIDWORKS System Options>Default Templates (PDM PRO Only) (customise)

## DISPLAY OPTIONS

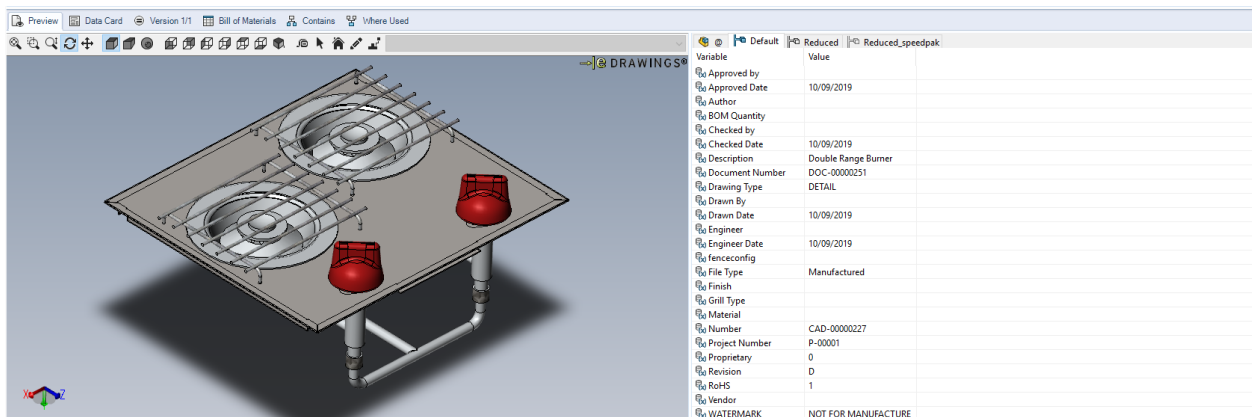
Within the PDM Vault folder (Windows Explorer UI) users will find a drop-list style menu bar called the 'PDM menu bar'. This menu contains Drop lists 'Actions', 'Modify', 'Display', 'Tools' as well as direct buttons for 'Search' 'Items' and 'logoff'. The 4 options we are interested in here are in the Display>Options menu:



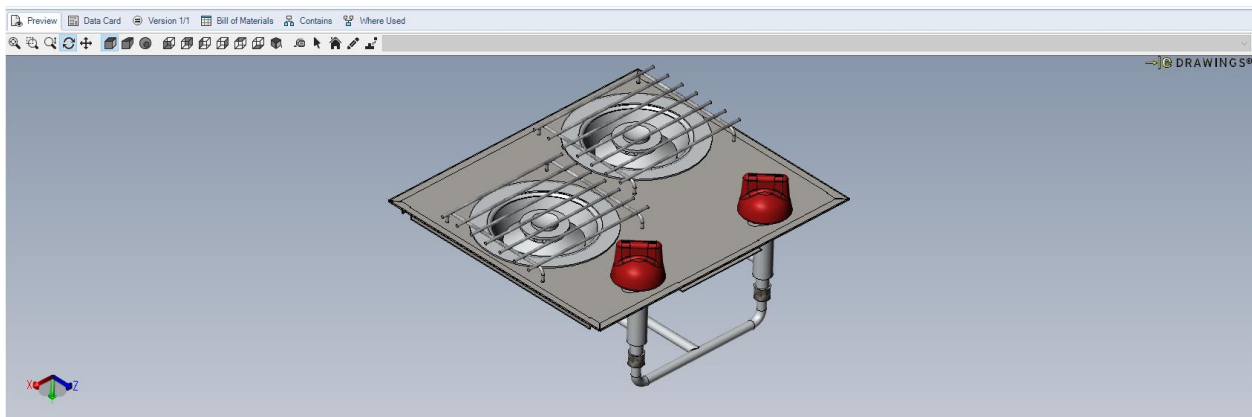
### Display>Card Preview

The card preview is on (Checked) by default and is typically good to turn off (Uncheck). The default setting will split the preview into two segments with one displaying a preview of the file and the other a tabulated list of the associated metadata. Turning this option off will provide a larger preview of the selected file whereby the metadata is still accessible via data card tab.

### Before

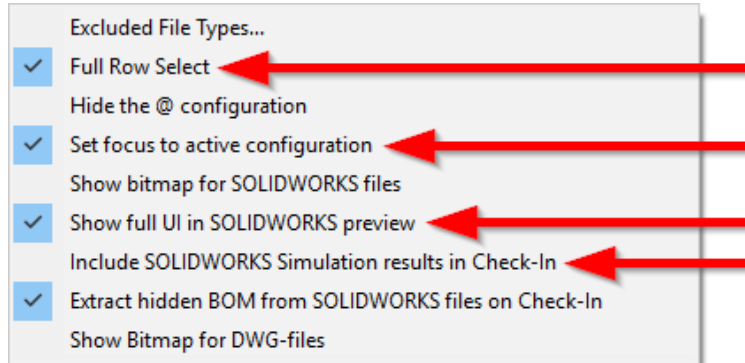


### After



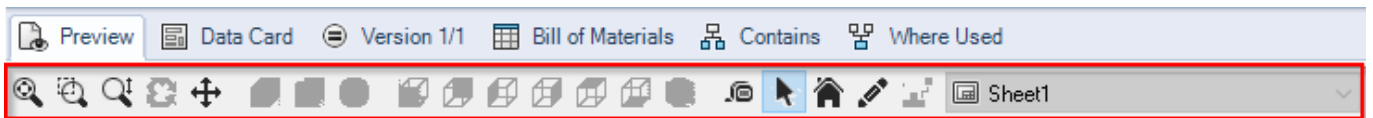
### Display>Options>Set focus to Active Configuration

This setting is on (Checked) by default and is typically good to turn off (uncheck). This setting affects the behaviour of the data card configuration tab focus and causes the card (when checked) to always adjust to the 'active configuration' tab when selecting a SOLIDWORKS Part or Assembly. Usually it's preferred keep the focus on the '@' tab which represents the Custom Property block data. Once turning this off, the same tab will be kept (i.e. @) from file to file. Play with this setting to see which tab focus you prefer.



### Display>Options>Show full UI in SOLIDWORKS preview

This Setting is off (un-checked) by default and should be turned on (checked). This setting affects the behaviour of the preview tab when showing eDrawings previews for CAD files. This setting turns on the eDrawings preview toolbar, which includes pan/zoom/mark-up type of controls, as well as the direct sheet selection dropdown list (direct sheet selection – PDM PRO only).



### Display>Options>Include SOLIDWORKS Simulation results in Check-In

This Setting is on (checked) by default and should typically be turned off (un-checked). This setting affects the behaviour of the check-in dialogue for SOLIDWORKS Parts and Assembly models that contain simulation study information. When on it will try and include external study results as file references in the same way as an assembly would look for part files as reference during check-in. It does this to show associativity between the model and the results data. Typically, though it's not a good idea to actually store Simulation study results (CWR) data in the vault. Often times the result files are lost or not available to the user checking in the files, and depending on user permissions this condition may entirely block the assembly or part from being checked-in when the results are 'Not found'. Its best practice not to include results, so turning off this setting will tell the check-in to not look for them and ignore.

## Display > Options > Full Row Select

This Setting is off (un-checked) by default and should be turned on (checked). This setting affects how the selection of files is displayed in the explorer interface. Without full Row Select, only the file name itself is highlighted when selected. With Full Row Select, the file name and its associated metadata are all highlighted, making it easier to read which metadata is associated with a selected file.

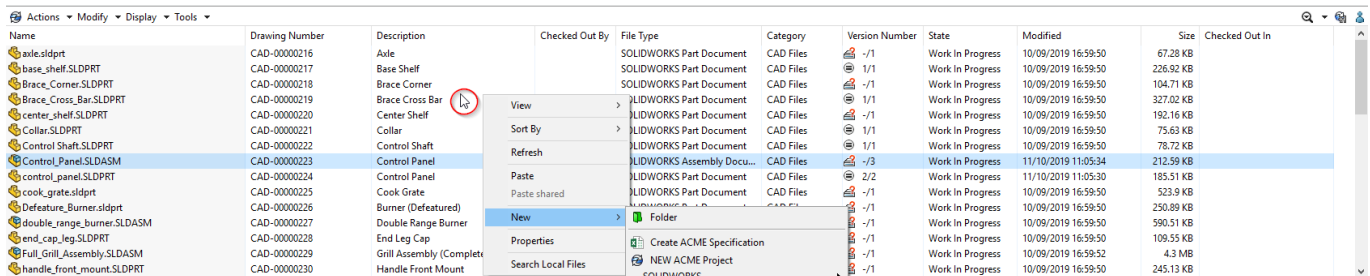
### Without Full Row Select

00008.SLDPRT	B	CHECK VALVE BODY	Work in Progress CAD	SOLIDWORKS Part Document	143.18 KB	24/04/2019 15:08:02	00008	3/3	Work in Pr...	
00009.SLDPRT	B	CHECK VALVE CENTER SHAFT	Work in Progress CAD	SOLIDWORKS Part Document	146.31 KB	24/04/2019 15:08:02	00009	3/3	Work in Pr...	
00010.SLDPRT	B	WASHER	Work in Progress CAD	SOLIDWORKS Part Document	69.25 KB	24/04/2019 15:08:02	00010	3/3	Work in Pr...	
00011.SLDPRT	B	WASHER-OUTER	Work in Progress CAD	SOLIDWORKS Part Document	73.81 KB	24/04/2019 15:08:00	<SSMTECH86...	00011	3/3	Work in Pr...
00012.SLDASM	B	CHECK VALVE	Work in Progress CAD	SOLIDWORKS Assembly Document	772.19 KB	24/04/2019 15:08:02	00012	3/3	Work in Pr...	
00013.SLDASM	B	CONTROL VALVE	Work in Progress CAD	SOLIDWORKS Assembly Document	924.97 KB	24/04/2019 15:08:02	00013	3/3	Work in Pr...	

### With Full Row Select

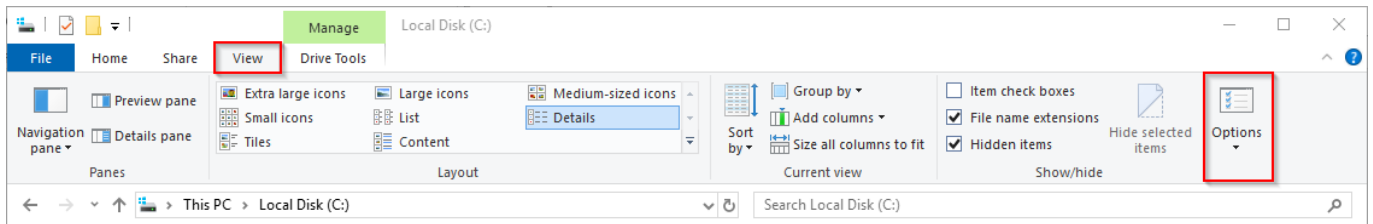
00008.SLDPRT	B	CHECK VALVE BODY	Work in Progress CAD	SOLIDWORKS Part Document	143.18 KB	24/04/2019 15:08:02	00008	3/3	Work in Pr...	
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00013.SLDASM	B	CONTROL VALVE	Work in Progress CAD	SOLIDWORKS Assembly Document	924.97 KB	24/04/2019 15:08:02	00013	3/3	Work in Pr...	

However the full row select option does effect the behaviour of the RMB (Right Mouse Button) click. When needing to access the folder background menu (templates etc.) with full row select activated the RMB needs to be performed in white space on an unselected item (as shown below).



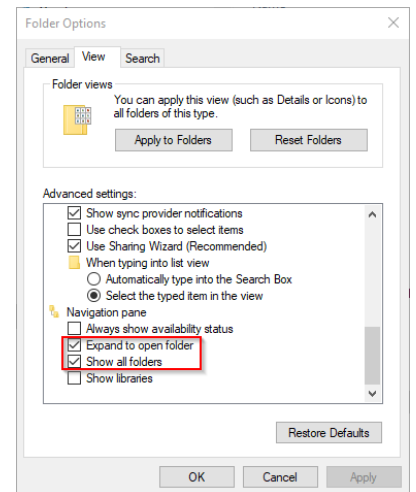
## WINDOWS FOLDER OPTIONS

As PDM integrates to windows explorer there are native Windows Explorer Folder Settings that can be modified to optimise the vault interface. The 3 settings we'll highlight are found in the Folder Options dialogue. These are settings that could be controlled by Windows Group Policy and so could be controlled centrally by the IT department via the domain, but most often these are left to the local user's discretion.



### Folder Options>View

In Windows Explorer the navigation pane settings for 'Show all Folders' and 'expand to open folder' are typically off (un-checked) by default. For PDM, it's good to encourage the use of the navigation pane in general (for efficient folder navigation) and these settings will cause the navigation pane to perform more optimally for this purpose. Turn these options both on (checked) Respectively: these settings will cause the navigation pane to show any available subdirectories at the current folder level, and then also cause the navigation pane to keep up with folder navigation from the file view (otherwise the navigation pane stays static)

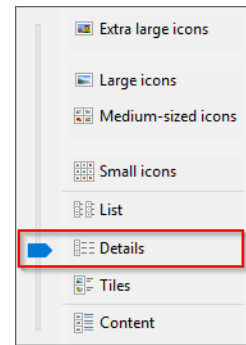


### Folder Options>View>Folder views>Apply to Folders

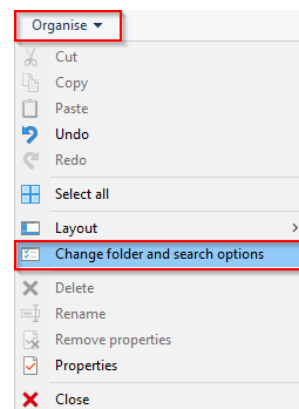
Because PDM makes use of Windows column information to display file meta-data (e.g. Checked Out By, workflow status, Description, Revision etc...) it is vital that the vault folders always be viewed in 'Details' view display mode (as opposed to Icon or Tiles etc...). Many users find that they set Details view mode one day but the next time they access vault the folders revert back to another non-preferred display mode. The secret to enforce Details on all folders is to...

1. Navigate to the root file Vault view 'Blueberry' folder (which is usually up to the C:\VAULT NAME)

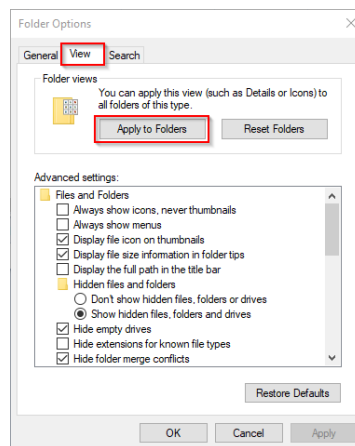
2. Make sure this folder is displayed in Details mode



3. Open the Folders Options >Organise>'Change folder and search options'



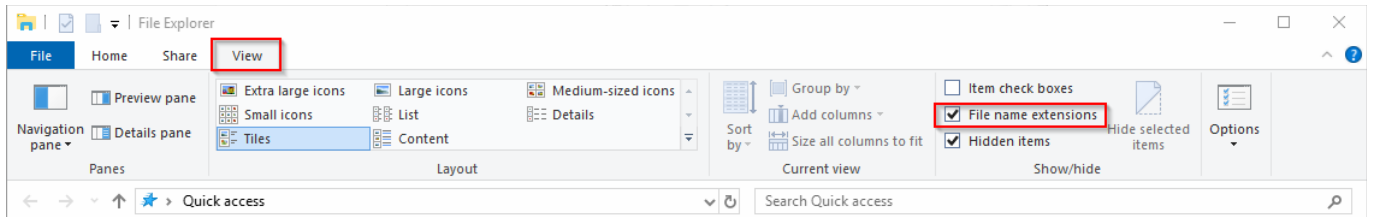
4. Press the button labelled 'Apply to Folders' and then



5. 'Yes' to the prompt: "Do you want all folders to match [these] view settings". This enforces the current folder's display style on all sub-directories. Since the PDM file view folder is a subdirectory, it will now permanently remember to use Details view mode.

## Folder Options>View>Advanced Settings>Hide extensions for known file types

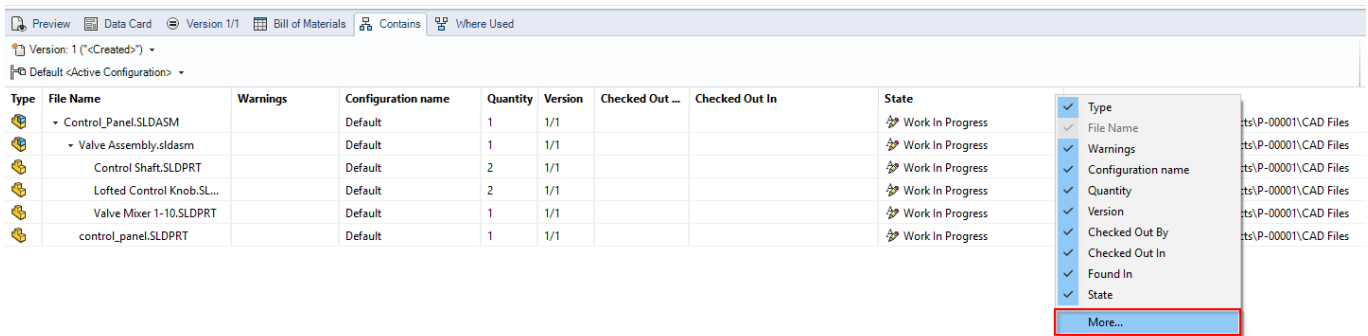
This is another classic Windows setting (above) in Folder options that is turned on (checked) by default. Turn off this setting to ensure that the full file name (including extension) is always displayed even for recognised file types. It is always best practice to display the full file name (including extension) in the name column of Windows Explorer. Once turned off in order to have this setting take effect usually requires ending the explorer process and restarting from a Task Manager, or rebooting the machine.



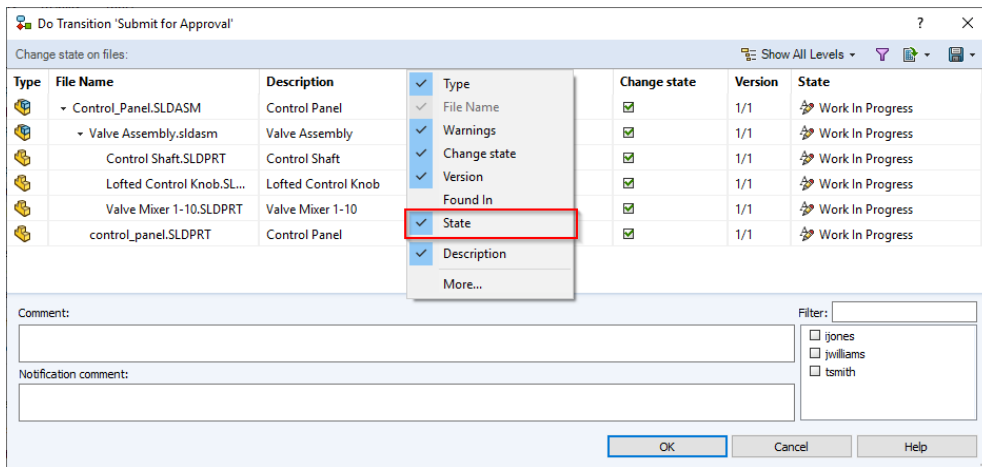
## COLUMN DISPLAYS

### Contains, Where Used, Check in/out, Change State

There are a number of column view displays used within the PDM environment. Some of these are centrally controlled administratively (e.g. BOM, Details view, and Search results columns) however many of the column sets are now accessible and solely within the users preview to control, configure and optimise (e.g. Contains, where Used, Check In/Out, and Change State UI's). For these column sets users are able to add/subtract to and from the available columns as desired. To configure these column views, the process is the same across all of these contents. For example, we'll look at modifying the Contains tab report. To modify the columns here is a very intuitive process: simply navigate to an assembly (or really any file), select it from the file view, and then go to the Contains tab. Within the Contains report, information is presented in a series of default columns:



To add or subtract columns from the default set: right click anywhere on the column header bar to see the available column list. Select 'More...' at the bottom of the list to add columns based on file data card property information. A great setup practice would be to keep the default set in place, and add to it any key universal document identifiers (e.g. Description, Number, Revision, etc.) to help more easily identify the records shown within report. This is also a good opportunity to take a moment to thoughtfully re-order the columns (drag/drop) to arrange them in a preferred order. For example: moving 'Description' to the left side next to the 'File name' column is always useful. The same procedure can be repeated and applied similarly to the **Where Used**, **Check-in/Out** and **Change State** dialogues. Modify these as well as you come across them in daily use.

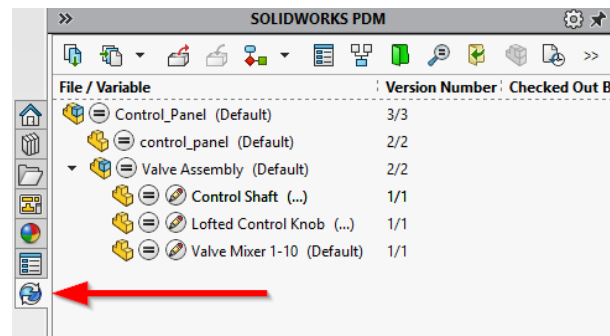


## WITHIN SOLIDWORKS

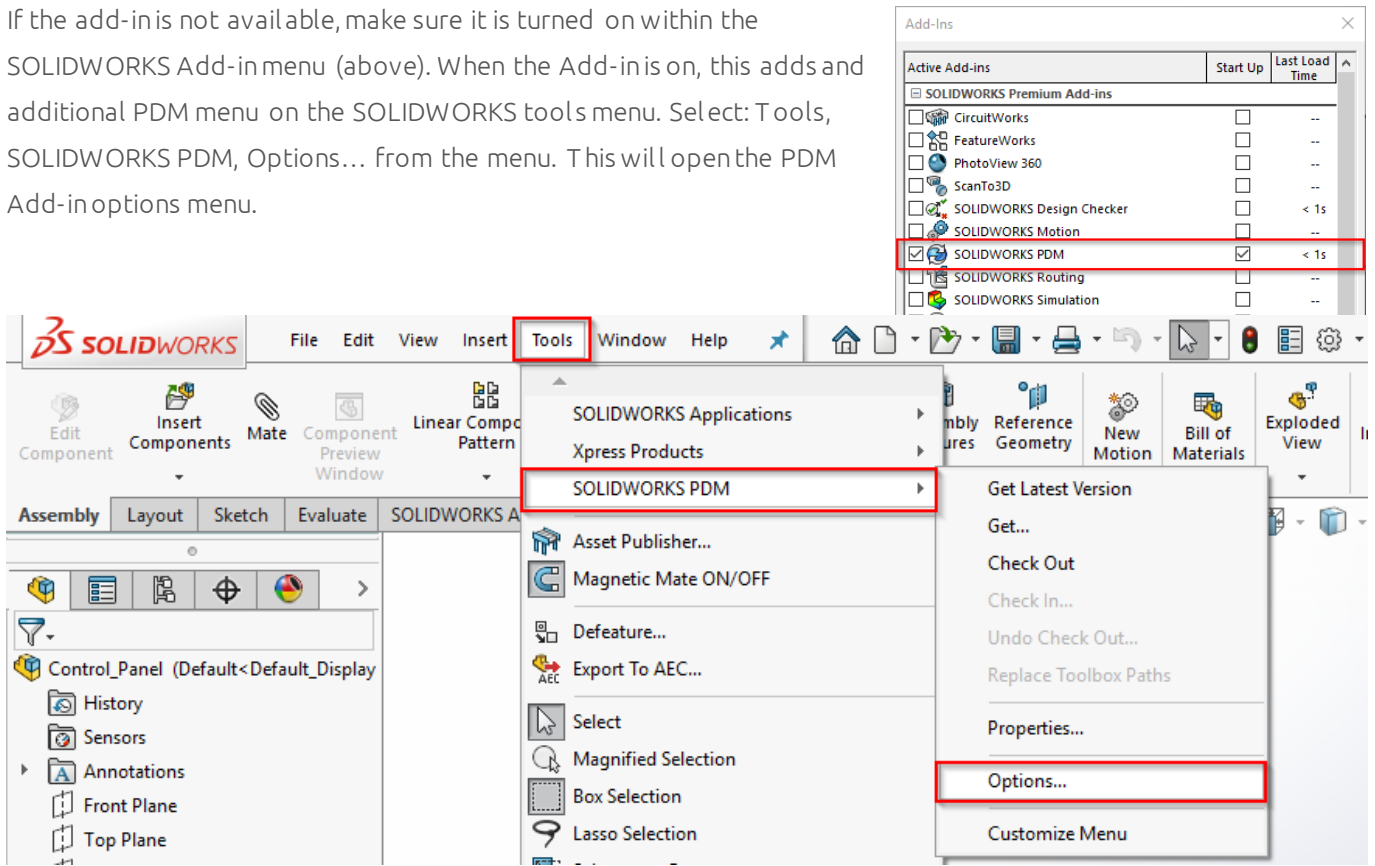
### PDM Add-in Options

When using the PDM Editor Licence within the SOLIDWORKS CAD software: CAD users have access to an additional PDM Interface called an 'Add-in'. By default this add-in is accessed via a tab/tool bar on the right hand side of the Window called the 'TaskPane' (image right).

The PDM add-in is easily identified by the 'blueberry' PDM icon tab.



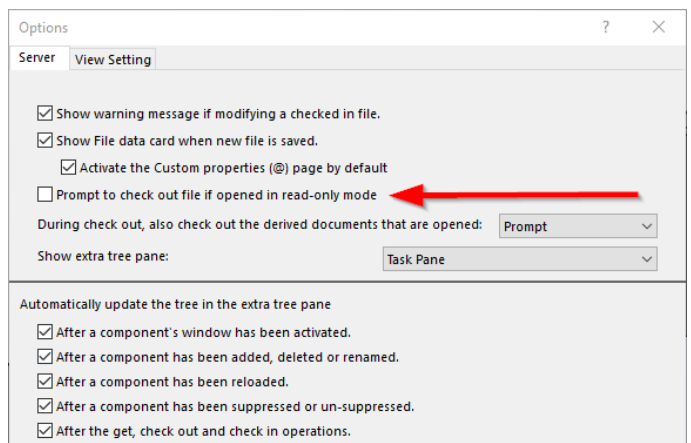
If the add-in is not available, make sure it is turned on within the SOLIDWORKS Add-in menu (above). When the Add-in is on, this adds and additional PDM menu on the SOLIDWORKS tools menu. Select: Tools, SOLIDWORKS PDM, Options... from the menu. This will open the PDM Add-in options menu.



### PDM Add-in Options > Prompt to checkout

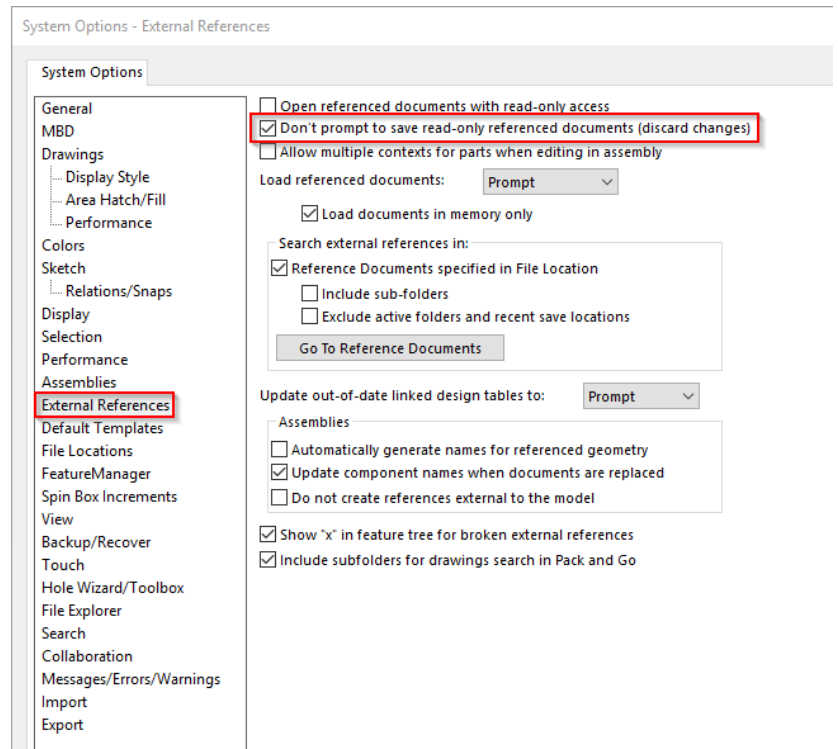
Except those settings labelled as 'Vault Settings', these options are all local user-specific settings that affect the behaviour of the add-in. The 'Vault Settings' in contrast are global settings that must be set centrally by an administrator. Of all the user controlled settings within this dialogue, none of them typically warrant particular attention nor should be changed from their default settings (in most common situations) except the 4th check box from the top: Prompt to check out file if opened in read-only mode. This option is turned on (checked) by default, and this is an important setting that should be turned off (and left off) in most instances. This setting causes a checkout dialogue prompt when opening read-only documents (in fact this is one of the primary functions of PDM: to present most stuff to users as 'read-only'). In

contrast the primary reason the add-in exists is to allow users the ability to checkout files within SOLIDWORKS as needed anytime (permissions permitting). Therefore it's both unnecessary and undesirable to prompt for checkout when merely opening files. This will cause inattentive users to inadvertently check-out many more files (sometimes many more) than they intend or need to checkout



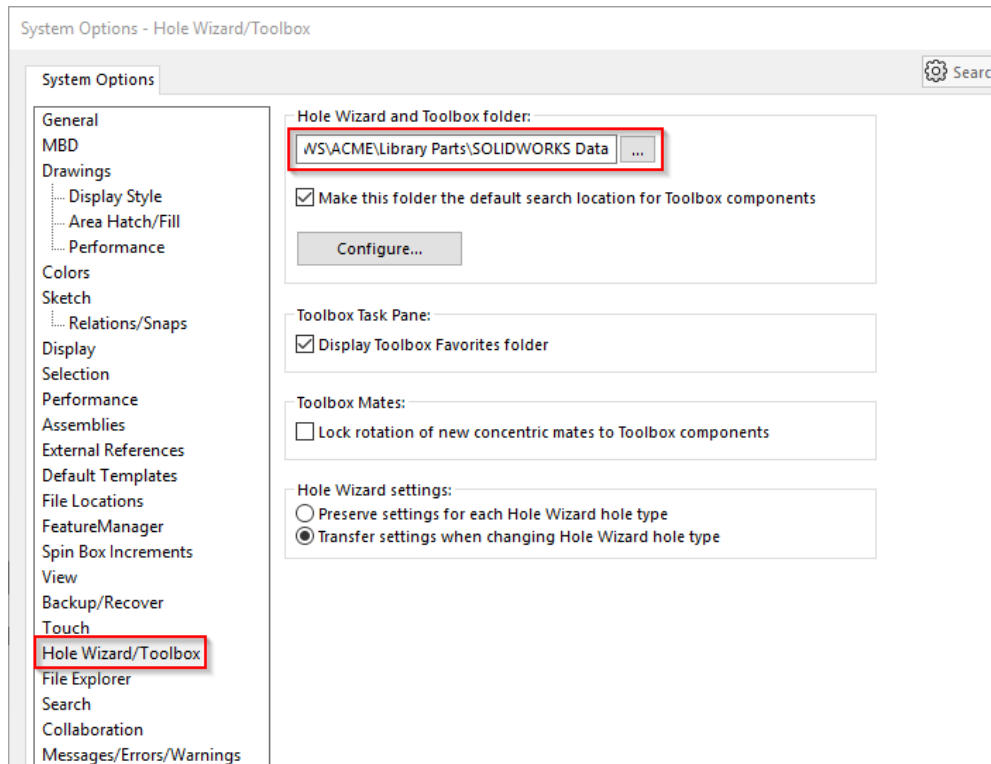
## SOLIDWORKS System Options > External References

SOLIDWORKS has a number of System Options settings that control a multitude of behaviours within the CAD environment. In addition with the recommended Add-In (above) regarding opening read-only files, this SOLIDWORKS setting deals in a similar vein with the behaviour expected when saving parent files referencing read-only components. Given that this is PDM we're talking about: most of the components in an assembly will typically be left read-only during the design process (as items are only checked out if they need to be directly edited). Thus we want SOLIDWORKS to ignore read-only reference files and not prompt to 'save-as' for all the read-only components. This setting is typically off (unchecked) by default, and it's recommended to turn it on (checked) for use with PDM. SOLIDWORKS System Options can be pushed out to all users using the SOLIDWORKS Copy Settings Wizard or direct registry updates. If that is the case, this setting should be turned on for all SOLIDWORKS users with access to PDM.



## SOLIDWORKS System Options>Toolbox

If the PDM administrator has stored the SOLIDWORKS Toolbox within the vault, then you will be prompted during the first time use to if you want to use the configured PDM toolbox location. **HOWEVER** it is best practice to ensure that the toolbox location is connected to the expected location.

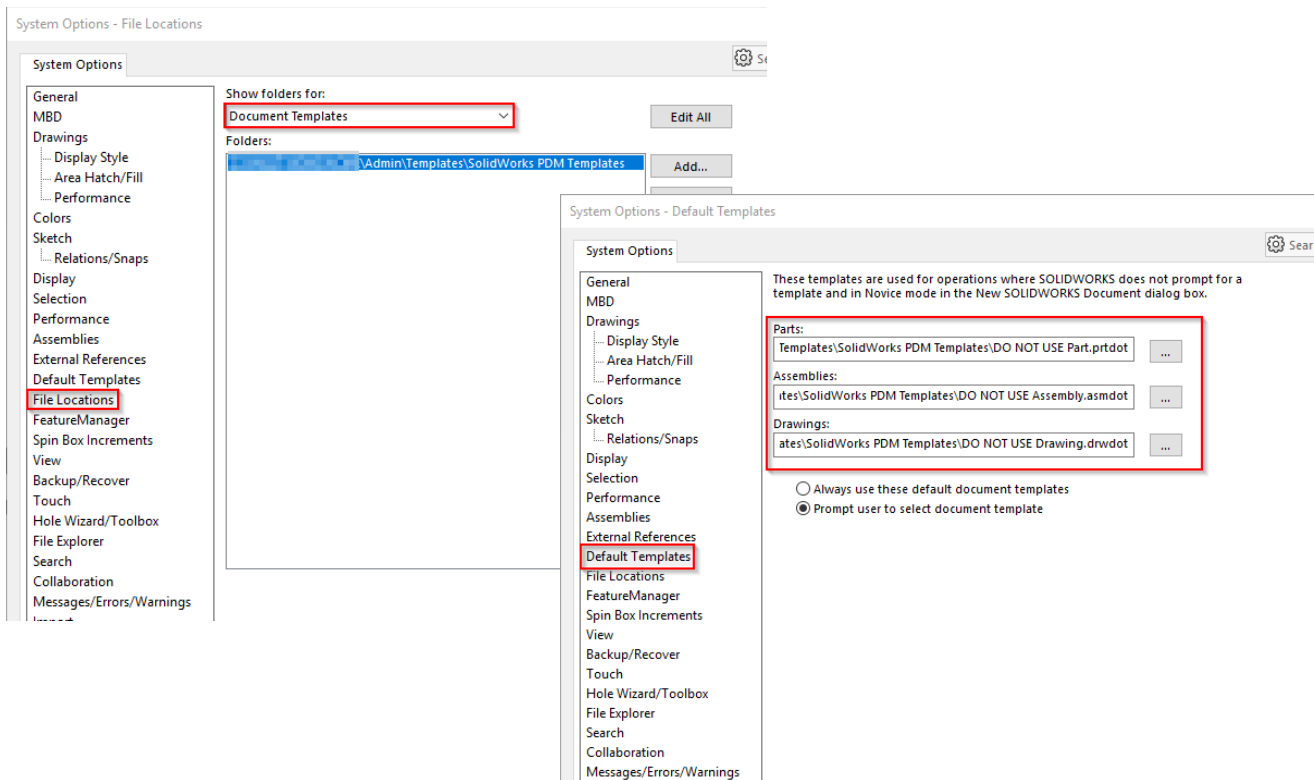


## SOLIDWORKS System Options>Default Templates (PDM PRO Only)

If there are SOLIDWORKS PDM templates (e.g. Automatic Serial) for SOLIDWORKS files defined within the PDM vault, the add-in will try to copy the default SOLIDWORKS file templates to a folder called "SOLIDWORKS PDM" under the user's TEMP location. This is done by the SOLIDWORKS PDM add-in in order to populate the 'New SOLIDWORKS Document' dialog in SOLIDWORKS with the templates from the vault. If the default SOLIDWORKS template source file cannot be found, a warning may show when selecting 'File', 'New' within SOLIDWORKS:

It is therefore best practice to ensure that there are a number of blank default templates available from within the vault. Usually found within – *C:\PDM\Admin\Templates\SolidWorks PDM Templates*

The "Do not Use" naming convention is used to alert the SOLIDWORKS user to select the correct defined templates (e.g. Automatic Serial).



Note that the default SOLIDWORKS templates option above should NOT point to the user's TEMP folder or to the actual template file in the file vault view. The path should point to the default SOLIDWORKS template location outside the vault (or to a shared network location if using a shared network template). If you have regular SOLIDWORKS templates that you want to store in a vault view you should add them to 'File Locations' instead.

