



WHAT'S NEW SOLIDWORKS PDM 2018



Contents

Legal Notices	3
Preface: Introduction	6
1 SOLIDWORKS PDM	7
Automated Drawing PDF Creation in SOLIDWORKS PDM Standard 🔀	7
Configuring the Convert Task (For SOLIDWORKS PDM Standard only)	8
Converting a SOLIDWORKS Drawing File to PDF	8
Data Card Editor Undo 📩	9
Design Branching and Merging (For SOLIDWORKS PDM Professional only)	9
Branching Settings	10
Merging Settings	17
Detailed Warning Messages 📩	24
Enhanced Assignment of Explicit Folder Permission	25
Enhanced File Version Upgrade Tool	25
Upgrade Tool Behavior for Checked out Files	25
File Version Upgrade Summary	26
Monitoring the Upgrade Process	26
Notifying Users About the Upgrade Process	27
Enhanced Permission Control	27
Quality Enhancements in SOLIDWORKS PDM	27
Revision Table Integration	28
Configuring Revision Table Integration	
Configuring the Revision Table Node	29
Defining Variable Mapping	30
Revision Table Dialog Box	30
Setting Dynamic Variable Values in Copy Tree	31
SOLIDWORKS PDM Application Programming Interface	32
SOLIDWORKS PDM Support for Non-SOLIDWORKS CAD File References 🔀	33

Legal Notices

© 1995-2018, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systemes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

Patent Notices

SOLIDWORKS[®] 3D mechanical CAD and/or Simulation software is protected by U.S. Patents 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028; 8,910,078; 9,129,083; 9,153,072; 9,262,863; 9,465,894; 9,646,412; 9,870,436; 10,055,083; and foreign patents, (e.g., EP 1,116,190 B1 and JP 3,517,643).

eDrawings[®] software is protected by U.S. Patent 7,184,044; U.S. Patent 7,502,027; and Canadian Patent 2,318,706.

U.S. and foreign patents pending.

Trademarks and Product Names for SOLIDWORKS Products and Services

SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

CircuitWorks, FloXpress, PhotoView 360, and TolAnalyst are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of HCL Technologies Ltd.

SOLIDWORKS 2018, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Simulation Standard, SOLIDWORKS Simulation Professional, SOLIDWORKS Simulation Premium, SOLIDWORKS Flow Simulation, eDrawings Viewer, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical Schematic Standard, SOLIDWORKS Electrical Schematic Professional, SOLIDWORKS Electrical 3D, SOLIDWORKS Electrical Professional, CircuitWorks, SOLIDWORKS Composer, SOLIDWORKS Inspection, SOLIDWORKS MBD, SOLIDWORKS PCB powered by Altium, SOLIDWORKS PCB Connector powered by Altium, and SOLIDWORKS Visualize are product names of DS SolidWorks.

Other brand or product names are trademarks or registered trademarks of their respective holders.

COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY

The Software is a "commercial item" as that term is defined at 48 C.F.R. 2.101 (OCT 1995), consisting of "commercial computer software" and "commercial software documentation" as such terms are used in 48 C.F.R. 12.212 (SEPT 1995) and is provided to the U.S. Government (a) for acquisition by or on behalf of civilian agencies, consistent with the policy set forth in 48 C.F.R. 12.212; or (b) for acquisition by or on behalf of units of the Department of Defense, consistent with the policies set forth in 48 C.F.R. 227.7202-1 (JUN 1995) and 227.7202-4 (JUN 1995)

In the event that you receive a request from any agency of the U.S. Government to provide Software with rights beyond those set forth above, you will notify DS SolidWorks of the scope of the request and DS SolidWorks will have five (5) business days to, in its sole discretion, accept or reject such request. Contractor/Manufacturer: Dassault Systemes SolidWorks Corporation, 175 Wyman Street, Waltham, Massachusetts 02451 USA.

Copyright Notices for SOLIDWORKS Standard, Premium, Professional, and Education Products

Portions of this software $\ensuremath{\mathbb{G}}$ 1986-2018 Siemens Product Lifecycle Management Software Inc. All rights reserved.

This work contains the following software owned by Siemens Industry Software Limited:

D-Cubed® 2D DCM © 2018. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed[®] 3D DCM © 2018. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed[®] PGM © 2018. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed® CDM © 2018. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed[®] AEM © 2018. Siemens Industry Software Limited. All Rights Reserved.

Portions of this software © 1998-2018 HCL Technologies Ltd.

Portions of this software incorporate PhysX[™] by NVIDIA 2006-2010.

Portions of this software © 2001-2018 Luxology, LLC. All rights reserved, patents pending.

Portions of this software © 2007-2018 DriveWorks Ltd.

© 2011, Microsoft Corporation. All rights reserved.

Includes Adobe[®] PDF Library technology.

Copyright 1984-2016 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents.5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382; Patents Pending.

Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries.

For more DS SolidWorks copyright information, see **Help** > **About SOLIDWORKS**.

Copyright Notices for SOLIDWORKS Simulation Products

Portions of this software © 2008 Solversoft Corporation.

 $\mathsf{PCGLSS} \ensuremath{\,^{\odot}}\xspace$ 1992-2017 Computational Applications and System Integration, Inc. All rights reserved.

Copyright Notices for SOLIDWORKS PDM Professional Product

Outside In[®] Viewer Technology, © 1992-2012 Oracle © 2011, Microsoft Corporation. All rights reserved.

Copyright Notices for eDrawings Products

Portions of this software © 2000-2014 Tech Soft 3D.

Portions of this software © 1995-1998 Jean-Loup Gailly and Mark Adler.

Portions of this software $\ensuremath{\mathbb{C}}$ 1998-2001 3D connexion.

Portions of this software © 1998-2017 Open Design Alliance. All rights reserved.

The eDrawings $^{\mbox{\tiny 6}}$ for Windows $^{\mbox{\tiny 8}}$ software is based in part on the work of the Independent JPEG Group.

Portions of eDrawings[®] for iPad[®] copyright © 1996-1999 Silicon Graphics Systems, Inc.

Portions of eDrawings[®] for iPad[®] copyright © 2003 – 2005 Apple Computer Inc.

Copyright Notices for SOLIDWORKS PCB Products

Portions of this software © 2018 Altium Limited.

Preface Introduction

This guide describes new and changed SOLIDWORKS[®] PDM capabilities in the 2018 release.

Intended Audience

This guide is for experienced users of the SOLIDWORKS PDM software.

1 SOLIDWORKS PDM

This chapter includes the following topics:

- Automated Drawing PDF Creation in SOLIDWORKS PDM Standard
- Data Card Editor Undo
- Design Branching and Merging (For SOLIDWORKS PDM Professional only)
- Detailed Warning Messages
- Enhanced Assignment of Explicit Folder Permission
- Enhanced File Version Upgrade Tool
- Enhanced Permission Control
- Quality Enhancements in SOLIDWORKS PDM
- Revision Table Integration
- Setting Dynamic Variable Values in Copy Tree
- SOLIDWORKS PDM Application Programming Interface
- SOLIDWORKS PDM Support for Non-SOLIDWORKS CAD File References

SOLIDWORKS[®] PDM is offered in two versions. SOLIDWORKS PDM Standard is included with SOLIDWORKS Professional and SOLIDWORKS Premium, and is available as a separately purchased license for non-SOLIDWORKS users. It offers standard data management capabilities for a small number of users.

SOLIDWORKS PDM Professional is a full featured data management solution for a small and large number of users, and is available as a separately purchased product.

Automated Drawing PDF Creation in SOLIDWORKS PDM Standard

SOLIDWORKS PDM Standard can convert a SOLIDWORKS Drawing file to PDF during a workflow transition using the same task technology used in SOLIDWORKS PDM Professional.

To provide access to the **Convert** task, you must add it to the vault by:

- Selecting the task when you create a new vault.
- Importing the task into an existing vault.

Configuring the Convert Task (For SOLIDWORKS PDM Standard only)

Administrators can configure the Drawing to PDF Convert task provided by the SOLIDWORKS Task Add-in to convert a SOLIDWORKS Drawing file to PDF. The task can map data card variables, and set the naming convention and destination for converted files.

To configure the Drawing to PDF Convert task:

- 1. In the Administration tool, expand **Tasks** and double-click **Convert**.
- 2. In the dialog box, in the left pane, click **Execution Method**.
- 3. Select the client computers that can be used to execute the task and how task execution is initiated.

You must configure each client computer as a task host to appear in the list and run the Drawing to PDF Convert task on the computer where you initiate the workflow transition.

Each client computer must have a licensed version of SOLIDWORKS to initiate the task.

 Click Conversion Settings > Conversion Options to display the output file format's Advanced Conversion Options (Adobe PDF) dialog box.
 The only output file format available is Adobe PDF (default).

The only output file format available is Adobe PDF (default).

- 5. Under **Source file references**, specify the version of referenced files to convert.
- 6. Click **File Card** to map variables from the source file's data card to the output file's data card.
- 7. Click **Output File Details** to specify the output file name format and destination. You can specify only the output paths that are inside the vault.
- 8. Click **OK**.

Converting a SOLIDWORKS Drawing File to PDF

To convert a SOLIDWORKS Drawing file to PDF:

- 1. In the Administration tool, configure the Convert task.
- 2. Edit a workflow transition to add an action to execute the Drawing to PDF Convert task.

When you change the state of a SOLIDWORKS Drawing file using a transition on which the Drawing to PDF Convert task is set, SOLIDWORKS PDM runs the task. The task:

- a. Launches a SOLIDWORKS session on the client computer.
- b. Opens the SOLIDWORKS Drawing file.
- c. Saves the file as PDF.

Data Card Editor Undo

SOLIDWORKS PDM lets you undo the changes done in the Card Editor for an active card.

Previously, to undo the changes, you had to close and reopen the card without saving.

To undo the card editor changes, do one of the following:

- Click **Edit** > **Undo**.
- Click **Undo** in the Main toolbar.
- Ctrl + Z.

You can undo the data card changes only if you have not saved the changes.

SOLIDWORKS PDM supports undo for:

- Modifying cards using control properties and card properties.
- Adding and deleting card controls.
- Moving and resizing card controls.

You cannot redo the data card changes that Undo discards.

Design Branching and Merging (For SOLIDWORKS PDM Professional only)

SOLIDWORKS PDM Professional has improved handling of various design change scenarios including multiple design improvement approaches, engineering change requests, and working with external design consultants.

Branching an existing file structure is similar to Copy Tree, but SOLIDWORKS PDM Professional maintains the history of source files and branched files. You can create new versions of the source files from the modified versions of branched files regardless of whether they have been moved or renamed.

🔒 View 🛛 🕼 Get	📄 Sar	ve 🗅	Compare	e 👌 Print
Event	Ver	User	Date	Comment
🔁 Initial transit	1	Ad	2017	State changed by automatic transition.
A Branched 'br1'	1	Ad	2017	Branched From C:\Vault_631\Tool Vise\tool vise.SLDASM
📄 Created	1	Ad	2017	
		Н	istory	of branched file
B History on too	I vise.SL	H	istory	of branched file
🗟 History on too	ıl vise.SL t 🗐 Si	H DASM ave	istory Compar	of branched file
B History on too D View D Ge Event	ıl vise.SL t 🔚 Si Ver	H DASM ave D User	istory Compar Date	of branched file re Print Comment
History on too History on too Get View Get Event A Branched 'br1'	I vise.SL t 📳 Si Ver 1	H DASM ave () User Ad	Compar Date 2017	of branched file re Print Comment Branched To C:\Vault_631\Branching\tool vise.SLDASM
History on too History on too History @ Ge View @ Ge Event S Branched 'br1' Initial transit	I vise.SL t 📄 Si Ver 1 1	H DASM ave User Ad	istory Compar Date 2017 2017	of branched file Te Print Comment Branched To C:\Vault_631\Branching\tool vise.SLDASM State changed by automatic transition.

Branching functionality lets you create complete or partial copies of a file structure with different names and place them in one or more folders. Placing the files into multiple folders lets you edit the branched files, retaining the source files in their original state and permission status. If the edits are approved, you can create new versions of the source files. Otherwise, you can delete the branched files.

Branching Settings

You can branch a single file or a file with its references. After branching, the source file history and branched file history are updated.

Administrators can also set user and group properties to exclude selected folders from branching. The files in the excluded folders are not available for branching.

In the Administration tool, on the Settings - Branch page, you can select the following predefined variables from the list to update the variable values.

Branch File Name
Branch File Name without extension
Branch File Path
Branch Name
Current time
Logged in user
Source File name
Source File name without extension
Source File path
Today's date
User - Full name
User - initials
User - User data

In the file list, right-click a branched file and select **Associated Branches** to list all the branches the file is a part of. Click a branch from the list to see the branch details.

Branch Dialog Box

The Branch dialog box is similar to the Copy Tree dialog box and lets you copy selected file references, or retain the existing references.

If your administrator has excluded a folder from branching, a message appears in the **Warnings** column and the **Branch** option is unavailable.

To display the Branch dialog box:

• Select a file and click **Tools** > **Branch**.

Branch Name

Enter a branch name.

You must enter a unique branch name.

Default Destination

Displays the current folder location of the selected file. You can enter a new path or browse to specify the destination path for the folder in the vault.

Settings

This group is a collapsible interface that includes:

Version to Use	
Version to ose	Latest Uses the latest version of the references for branching.
	Referenced Uses the attached version of the references for branching.
Options	
	Include simulation Branches SOLIDWORKS Simulation results associated with the selected files.
	Preserve relative paths Keeps the paths of references relative to the branched parent file, creating a folder structure as needed. When you clear Preserve relative paths , the folder hierarchy is flattened and all referenced files are branched to the same destination folder as the parent.
	Include drawings Shows any related drawing files in the file list so you can include them when branching the reference tree.
	Regenerate serial number in cards If serial numbers are used in data cards, assigns the next number in the sequence. For Rename with serial number transformations, the same serial numbers used to name the files are used in data cards.
	Name drawings after their models Sets drawing file names to the same as their associated assembly or part file names.

Transform Operations Modifies the file names of selected files using the transformation you select.

Add Prefix

Adds the prefix you type to the file names.

Add Suffix

Adds the suffix you type to the file names.

Rename with serial number

Replaces file names with serial numbers.

Only available when serial numbers are defined in the Administration tool.

Replace

Replaces strings in file or folder names. In the Replace dialog box:

- 1. For **Find what**, type the string to replace.
- 2. For **Replace with**, type the replacement string.
- 3. Select whether to apply the change to all files or only the selected files.
- 4. Select to apply the change to file names, folder names, or both.

Filter Display

Enter text to refine the list of files displayed. Use the column selection drop-down to limit the filtering to a specific column. For example, limit the columns to **File name**, or **All Columns**.

By default, the filter is applied to **All Columns**. You can expand the **All Columns** list and select a specific column to apply the filter to. The software searches for the text that you enter, displays rows that contain the text, and highlights the text for identification.

The search field supports these wildcard characters: *, ?, %, -, ., "".

*string	Displays rows that contain text that ends with the entered string. The string in the filtered rows is highlighted in pink.
string*	Displays rows that contain text that starts with the entered string. The string in the filtered rows is highlighted in pink.
	Instead of *, you can use ? or %.

-string or string-	Displays rows that contain the entered string.
	If the hyphen is at the beginning, the text ending with the string is highlighted in pink. If the hyphen is at the end, the text starting with the string is highlighted.
"string or string"	Displays rows that contain the entered string. The string in the filtered rows is highlighted in pink.
.string OF string.	Displays rows that contain the entered string and highlights the string with a preceding or a succeeding character depending on the position of the dot.
The filter options include	e:
Invert Filter or !	You can invert the filter by clicking or by entering the ! character as a prefix to the text in the search field.
	Files that do not include the entered text are displayed.
Filter	 File Type Assemblies Parts Drawings Others Selected For Branch Yes, Selected No, Not Selected Path/Name Changed Yes, Changed No, Not Changed

All Columns	Filters the file list based on the text you enter and the column you select. You can select one of the following columns: • Source File Name • Warnings • Version • Checked Out By • Checked Out By • Checked Out In • Source File Found In • Branch File Found In • Branch File Name • State • All Columns
	This filter supports all custom columns that appear in the file list.
Clear Filter Text 🗳	Removes all filters. This appears when you select an option in Filter .

Toolbar Buttons

<u>A</u> t <u>A</u> t	Next Warning/Previous Warning	In the file list, changes the focus to the next file with a warning or the previous file with a warning.
III Show All Levels ▼ III Top Level Only IIII Show All Levels	Levels	Displays file references for the entire file hierarchy (Show All Levels) or just the top level of the file hierarchy (Top Level Only).
		The setting defaults to the most recent setting you used.
	Open File List	Open All opens a list of files in Microsoft Excel.
		Open Visible opens a list of visible files in Microsoft Excel.
	Save File List	Export All exports the file list of all files as comma-separated .txt file.
		Export Visible exports the list of visible files as a comma-separated .txt file.

File List

To change the columns that are displayed, right-click any column heading and select columns to display them or clear columns to hide them. You can add up to 10 columns based on variables by clicking **More** and choosing variables from the Choose Columns dialog box.

You can sort the default and custom columns in tables in ascending, descending, or default order by clicking column headers. Column sorting is useful within large datasets. If you sort a column in ascending or descending order, the column is highlighted in green and an arrowhead appears. When you sort a column in a table, SOLIDWORKS PDM removes the file structure hierarchy. Sorting turns off **Show Tree Lines** and **Show Reference Selection Control**.

Туре	Displays a thumbnail preview of the file when you hover over the file type icon.
Source File Name	Displays the source file name.
Warnings	Displays the warnings.
	To quickly locate files with warnings or errors, use the Next Warning $\stackrel{\text{A}}{\longrightarrow}$ and Previous Warning $\stackrel{\text{A}}{\longrightarrow}$ toolbar buttons or Ctrl + the up-arrow or down-arrow on the numeric keypad.
Branch	Lets you select the files to branch.
Version	The first number is the local (cached) version of a source file or a hyphen (-) if locally modified. The second number is the latest version in the vault.
Checked Out By	User who has the source file checked out, or blank if not checked out.
Checked Out In	Path to computer and local folder where the source file is checked out, or blank if not checked out.
Source File Found In	Path to the folder that contains the source file.
Branch File Found In	Lets you change the path to a folder for the branched file.
	The folder must be in the vault.
Branch File Name	Lets you change the branched file name
State	State of the source file.

Drag a column head to change the position of a column.

Check-in after branching

Checks in files on branching. You can optionally enter the check-in comments.

Total to Branch

Displays the number and type of files you have selected for branching.

The warning \triangle indicates that other files are selected for branching but do not appear in the file list because of the current filter.

Reset All

Resets all the changed file names and changed destination folder paths to default.

Merging Settings

In the Administration tool, on the Settings - Merge page, you can select the following predefined variables from the list to update the variable values.

Branch File Name
Branch File Name without extension
Branch File Path
Current time
Logged in user
Source File name
Source File name without extension
Source File path
Today's date
Use from Branch File
Use from Source File
User - Full name
User - initials
User - User data

Merge Dialog Box

The **Merge** dialog box lets you merge the branched file with the source file using different merging options.

To display the Merge dialog box:

• Select a branched file and click **Tools** > **Merge**.

Settings

Branch Boforoncoc			
	Latest Uses the latest version of the branched file for merging.		
	Referenced Uses the attached version of the branched file for merging.		
New File Options	Applicable for files that are newly created using merge option.		
	Include simulation Merges SOLIDWORKS Simulation results associated with the selected files.		
	Include drawings Shows any related drawing files in the file list so you can include them when merging the reference tree.		
	Regenerate serial number in cards If serial numbers are used in data cards, assigns the next number in the sequence. For Rename with serial number transformations, the same serial numbers used to name the files are used in data cards.		
	Name drawings after their models Sets drawing file names to the same as their associated assembly or part file names.		

Transform Operations Modifies the file names of selected files using the transformation you select. Applicable for files that are newly created using merge option.

Add Prefix

Adds the prefix you type to the file names.

Add Suffix

Adds the suffix you type to the file names.

Rename with serial number

Replaces file names with serial numbers.

Only available when serial numbers have been defined in the Administration tool.

Replace

Replaces strings in file or folder names. In the Replace dialog box:

- 1. For **Find what**, type the string to replace.
- 2. For **Replace with**, type the replacement string.
- 3. Select whether to apply the change to all files or only the selected files.
- 4. Select to apply the change to file names, folder names, or both.

Filter Display

Lets you enter text to refine the list of files displayed. A column selection drop-down lets you limit the filtering to a specific column, for example, **File name**, or for **All Columns**.

By default, the filter is applied to **All Columns**. You can expand the **All Columns** list and select a specific column to apply the filter to. The software searches for the text that you enter, displays rows that contain the text, and highlights the text for easy identification.

The search field supports these wildcard characters: *, ?, %, -, ., "".

*string	Displays rows that contain text that ends with the string. The string in the filtered rows is highlighted in pink.
string*	Displays rows that contain text that starts with the string. The string in the filtered rows is highlighted in pink.
	Instead of *, you can use ? or %.

-string or string-	Displays rows that contain the string.	
	If the hyphen is at the beginning, the text ending with the string is highlighted in pink. If the hyphen is at the end, the text starting with the string is highlighted.	
"string Or string"	Displays rows that contain the string. The string in the filtered rows is highlighted in pink.	
.string Or string.	Displays rows that contain the string and highlights the string with a preceding or a succeeding character depending on the position of the dot.	
The filter options include	9:	
!	You can invert the filter by entering character ! as a prefix to the text in the search field.	
	Files that do not include the entered text are displayed.	
Filter	• File Type	
	 Assemblies Parts Drawings Others 	
	Selected For Merge	
	Yes, SelectedNo, Not Selected	
	Path/Name Changed	
	Yes, ChangedNo, Not Changed	

All Columns	Filters the file list based on the text you enter and the column you select. You can select one of the following columns: Branch File Name Branch File Version Branch File Checked Out By Branch File Checked Out By Branch File Checked Out In Branch File State Warnings Merge Merge Option Source File Name Source File Found In Source File Checked Out By Source File Checked Out By
	This filter supports all custom columns that appear in the file list.
Clear Filter Text 🗵	Removes all filters. This appears when you select an option in Filter .

Toolbar Buttons

₩ ¹ ₩	Next Warning/Previous Warning	In the file list, changes the focus to the next file with a warning or the previous file with a warning.
말 Show All Levels ▼ 플 Top Level Only 말 Show All Levels	Levels	Displays file references for the entire file hierarchy (Show All Levels) or just the top level of the file hierarchy (Top Level Only).
		The setting defaults to the most recent setting you used.
	Open File List	Open All opens a list of files in Microsoft Excel.
		Open Visible opens a list of visible files in Microsoft Excel.
	Save File List	Export All exports the file list of all files as comma-separated .txt file.
		Export Visible exports the list of visible files as a comma-separated .txt file.

File List

To change the columns that are displayed, right-click any column heading and select columns to display them or clear columns to hide them. You can add up to 10 columns based on variables by clicking **More** and choosing variables from the Choose Columns dialog box.

You can sort the default and custom columns in tables in ascending, descending, or default order by clicking column headers. Column sorting is useful within large datasets. If you sort a column in ascending or descending order, the column is highlighted in green and an arrowhead appears. When you sort a column in a table, SOLIDWORKS PDM removes the file structure hierarchy. Sorting turns off **Show Tree Lines** and **Show Reference Selection Control**.

Drag a column head to change the position of a column.

Туре	Displays a thumbnail preview of the file when you hover over the file type icon.	
Branch File Name	Displays the branched file name and its references.	
Branch File Version	The first number is the local (cached) version of a branched file or a hyphen (-) if locally modified. The second number is the latest version in the vault.	
Branch File Found In	Path to the folder that contains the branched file.	
Branch File Checked Out By	User who has the branched file checked out, or blank if not checked out.	
Branch File Checked Out In	Path to computer and local folder where the branched file is checked out, or blank if not checked out.	
Branch File State	State of the branched file.	
Warnings	Displays the warnings \triangle specific to source files and branched files. For multiple warnings for a file, a hyperlink appears. You can click the link to see the details of the warnings.	
	To quickly locate files with warnings or errors, use the Next Warning and Previous Warning toolbar buttons or Ctrl + the up-arrow or down-arrow on the numeric keypad.	
	You can complete the merge action only after you resolve the critical warnings.	
Merge	Lets you select files to merge. If the Merge check box is not selected for a file, the references from the branched file are retained as they are, after the merge action.	

Merge Option

Merge

Selected by default for the branched file that is modified. Creates a new version of a source file when a branched file is checked in through the Merge user interface. You can keep the file checked out and check it in later.

You must have read file permission and check out file permission for the source file.

Use Source Reference

Selected by default for the unmodified branched file only if its immediate parent file is selected to merge. SOLIDWORKS PDM repoints the reference to the source file after the merge action.

If **Always work with the latest version of files** is selected, SOLIDWORKS PDM repoints the reference to the latest version of the source file. You must have read file permission for the latest version.

If **Always work with the latest version of files** is cleared, the merge action repoints the reference to the version of the source file that is used during branching.

If this version is not available or in cold storage, or does not provide read access, the merge action repoints the reference to the latest version that has read-file permission.

Merge As New File

Available for a branched file. The parent file refers to the new file after the merge action. You must have the add or rename file permission for the vault folder that contains the new file.

By default, the Source File Name is the name of the branched file and its target location is the parent file location. You can change the file name and its target location.

Create New File

Available for the file that is not branched but is a part of the reference tree. You must have the add or rename file permission for the vault folder that contains the new file. The parent file refers to the new file after the merge action.

By default, the Source File Name is the name of the branched file and its target location is the parent file location. You can change the file name and its target location.

Source	File	Name	Le
			:-

Lets you change the source file name only if the **Merge Option** is set to **Merge As New File** or **Create New File**.

Source File Found In	Lets you change the path to the vault folder that contains the source file only if the Merge Option is set to Merge As New File or Create New File .
Source File Checked Out By	User who has the source file checked out, or blank if not checked out.
Source File Checked Out In	Path to folder where the source file is checked out, or blank if not checked out.
Source File State	State of the source file.

Check-in on Merge

Checks in files on merging. You can optionally enter the check-in comments.

Total to Merge

Displays the number and type of files you have selected for branching.

The warning indicates that other files are selected for branching but do not appear in the file list because of the current filter.

Reset All

Resets all the changed file names and changed destination folder paths to default.

Detailed Warning Messages

SOLIDWORKS PDM now provides more-detailed warning messages when a transition fails.

If one or more conditions are not met, the **Warnings** column displays:

- A warning in case of one failed condition.
- A hyperlink in case of multiple failed conditions or all failed OR conditions.

You can click the hyperlink to see the details of the conditions that are not met. Based on the conditions defined in the transition, the warning includes details like condition type, comparison criteria, value, and configuration details.

Enhanced Assignment of Explicit Folder Permission

SOLIDWORKS PDM lets you set explicit folder permissions on multiple folders at the same time.

Previously, you could set explicit permissions on one selected folder at a time.

On the Assigned Folder Permissions tab, to select multiple folders, do one of the following:

- Shift + click
- Ctrl + select
- Left-drag

The **Folder Permissions** area displays the assignable permissions. The status of check box changes as the folder permission assigned to selected folder changes.

Check box status	Description
☑ (Selected)	Permission is set to all selected folders.
(Green)	Permission is set to one or more selected folders, but not all.
(Cleared)	Permission is not set to any selected folder.

Enhanced File Version Upgrade Tool

In the SOLIDWORKS PDM File Version Upgrade tool, vault analysis and upgrade are now quicker.

The enhancements in the upgrade tool let you:

- Upgrade the checked out files on other client workstations.
- Add or remove folders to upgrade.
- Sort the file list in Search Results.
- Stop the upgrade process and resume.
- View the summary of the file versions to upgrade.
- Monitor the upgrade process.
- Notify users about the upgrade process.

Upgrade Tool Behavior for Checked out Files

You can upgrade the checked out file and its references if the file is checked out on a machine other than the host machine.

You can upgrade the checked out file when:

• Creating new versions of files

• Overwriting existing versions of files

The following table describes the file handling of checked out file and its references:

Option	Action on checked out file	Action on parent references	Action on child references
Create new versions of files	Skip the file	Upgrade the latest version of the parent file	Upgrade the latest version of the child references
Overwrite existing versions of files	Upgrade all versions of the file	Upgrade all versions of the parent file	Upgrade all versions of the child references

You cannot upgrade the checked out file and its references if the file is checked out on the host machine.

When using **Overwrite existing versions of files** option, the upgrade tool does not check out the file, and the file is accessible to other users.

File Version Upgrade Summary

The upgrade tool shows the summary of the file versions to upgrade on the Ready to Upgrade Files screen.

The summary displays the following:

- Total number of SOLIDWORKS versions to update.
- Number of SOLIDWORKS part file versions.
- Number of SOLIDWORKS drawing file versions.
- Number of SOLIDWORKS assemblies file versions.
- Work instruction file names.
- Number of files in each work instruction file.

Monitoring the Upgrade Process

To monitor the upgrade process, do one of the following:

- On the Welcome screen, click **Monitor upgrade process**.
- Initiate the upgrade process and, on the Ready to Upgrade Files screen, click Monitor.

The Monitor upgrade screen displays the overall upgrade process progress and the upgrade progress of each work instruction file.

Host Name	Machine name on which you initiate the upgrade process
Instruction file name	Names of the work instruction files
Progress	Upgrade completion status of work instruction files

Processed Batch	Number of processed batches / total number of batches
Processed File	Number of processed files / total number of files
Start time	Time when the upgrade process is started
Elapsed time	Time elapsed after the upgrade is started
Status	Upgrade status of work instruction files

Notifying Users About the Upgrade Process

SOLIDWORKS PDM notifies users whenever the upgrade process is started, paused, resumed, completed, or has failed.

To notify users about the upgrade process:

- 1. On the Upgrade Settings screen, select **Notify success/failure to users**.
- 2. Click **Add** and select the users to be notified.
 - SOLIDWORKS PDM sends notification emails to the designated users based on the status of work instruction files.

Enhanced Permission Control

The Admin user can allow users to undo check out or check in the files that are checked out by another user.

The administrative permission, **Can undo check out files checked out by other users**, is selected by default for the Admin user.

With this permission, users can do the following to the files checked out by other users:

- Undo the Check out or Check in of files on the same machine and in the same vault view.
- Undo the Check out of files on a different machine and in a different vault view.

When a user does undo check out to a file checked out by another user, the details are added in history.

Previously, only administrators could undo the check out or check in of files that are checked out by another user.

Quality Enhancements in SOLIDWORKS PDM

You can open the search results from the embedded Windows Explorer search tool in Microsoft Excel or export them as a comma-separated value (.csv) file. The **Open search result** and **Export search result** commands are available on the Search toolbar. Click is to export the search results as a comma-separated value (.csv) file. Click to open the search results in a Microsoft Excel spreadsheet.

You can copy links of files and paste them to notification messages to your colleagues. Right-click a file, select **Copy Link**, and paste the link in the new message. When you click the link, SOLIDWORKS PDM navigates you to the appropriate file in the vault.

In the Administrative Export File dialog box, you can expand or collapse specific nodes or all nodes. To expand or close all nodes, right-click a node or in the dialog box and select **Expand All Nodes** or **Close All Nodes**. When you remove the specific setting or variable, the expanded or collapsed structure is retained.

SOLIDWORKS PDM provides the _*SW_Last_Saved_With_* variable that provides mapping between the **Summary** block name and **Last Saved With** attribute name for slddrw, sldasm, and sldprt files. SOLIDWORKS PDM updates the value of the variable whenever you check in a file. In the Administration tool, in the Customizable Columns dialog box, you can select this variable to add a column in the file list or search result for specific users. You cannot delete this variable.

Revision Table Integration

SOLIDWORKS PDM can now read and write values from or to a SOLIDWORKS revision table. You can configure SOLIDWORKS PDM variables to automatically add information such as revision date, description, and approver to a new row in the table or to update the last row.

When you change the mapped variable values in the file's data card, the values in the latest row of the table are updated and vice versa.

The revision table row can be updated automatically by using:

- Set variable transition actions
- Set Revision command

Configuring Revision Table Integration

To configure revision table integration:

- Configure the mapping between SOLIDWORKS PDM variables and revision table columns in the SOLIDWORKS drawing.
 For details, see SOLIDWORKS PDM Administration Guide: Mapping Variables to File Properties.
- 2. Configure the **Revision Table** node.
- 3. Set a variable that holds the custom property revision.

You must set the Revision variable for the current state of a file during the set revision action and for the target state of a file during the change state action. For details, see *SOLIDWORKS PDM Administration Guide: Set Variables Dialog Box*.

4. Define the Set Variable transition actions for each variable.

You need not define the Set Variable transition action for the Zone variable as it is read-only.

Configuring the Revision Table Node

You must configure the Revision Table node to update the revision table rows.

To configure the revision table node:

- 1. In the Administration tool, expand **SOLIDWORKS** and double-click **Revision Table**.
- 2. Select **Enable Revision Table** to manage the revision table by SOLIDWORKS PDM.
- 3. Set the values on the **Revision Table Settings** and **Set Revision Command Settings** tabs.
- 4. Click **OK**.

Configuring the Revision Table Settings

You can set options on the Revision Table Settings tab.

To configure the Revision Table Settings:

- 1. Open the Revision Table dialog box.
- 2. On the Revision Table Settings tab, set the number of visible rows in the revision table.
- 3. In the **Revision placeholder character**, enter a placeholder character or characters.

Configuring the Set Revision Command Settings

You can set options on the Set Revision Command Settings tab to update the revision table when you use the Set Revision command.

In the File Explorer, in the Set Revision dialog box, you must select **Update Variable** to update the revision table.

To configure the Set Revision Command Settings:

- 1. Open the Revision Table dialog box and select the Set Revision Command Settings tab.
- 2. Select Update Revision Table in SOLIDWORKS Drawing through "Set Revision" Command.
- 3. Select a check box under the **Revision Table Column** for a variable to appear in the revision table.
- 4. Enter a value under **Column Value**.

You can click \geq to select the system variable.

- 5. Select a corresponding variable under **Variable Name**.
- 6. Click **OK**.

Defining Variable Mapping

You can configure the mapping between SOLIDWORKS PDM variables and revision table columns in SOLIDWORKS drawings.

To configure the variable mapping:

1. Expand the Variables node and edit the variables.

You can edit the current variables, **Approved by**, **Description**, **Date**, **Revision**, or create new variables. You must create the **Zone** variable as read-only because the value of this variable is read from the drawing.

- 2. In Variable name, type a name.
- 3. Set the **Variable type** as **Text**.
- 4. In **Block name**, type **SWRevTable**.
- 5. In **Attribute name**, select the attribute name.

For a custom column, you must type the attribute name that is the same as the revision table column header.

When you use the SWRevTable block for the default revision table column, you can select the attribute name from the list.

6. Type the file extension, slddrw, and click **OK**.

Revision Table Dialog Box

To open the Revision Table dialog box:

• Expand the **SOLIDWORKS** node and double-click **Revision Table**.

Enable Revision Table
Select this option to manage the SOLIDWORKS drawing revision table using SOLIDWORKS PDM.
When revision tables are enabled in SOLIDWORKS PDM, the SOLIDWORKS Alpha/numerical control options are not available for Revision Table in Tools > Options > Document Properties > Tables > Revision. A message, Revision driven from SOLIDWORKS PDM appears in the SOLIDWORKS Tables Options dialog box.

Revision Table Settings

Number of visible rows	Sets the number of rows to display in the drawing revision table. When the number of revision table rows in a drawing exceeds the set number, the oldest row is deleted.
	oldest row is deleted.

Revision placeholder character(s)	Lets you enter characters that appear in the revision column of the revision table. The actual revision value replaces these characters when you change the revision in SOLIDWORKS PDM. It is mandatory to enter one or more placeholder characters. The placeholder characters must be between 1 and 5 characters in length.
	characters in length.

Set Revision Command Settings

Update Revision Table in SOLIDWORKS	Updates the revision table when you:
Drawing through "Set Revision"	 Define the Set Revision action.
Command	• Select Update Variable in the File
	Explorer in the Set Revision dialog box.

Select a check box to display a variable as a revision table column in the revision table.

Revision table column	Column value	Variable Name
Revision	By default, set to revision value set with Set Revision action.	Select the mapped variable, Revision .
		The list displays the variables that are mapped using the SWRevTable block name.
Description, Date, Approver	Click b to select a variable to be added to the column. You can also type the text.	Select the variable.

Setting Dynamic Variable Values in Copy Tree

With the enhanced Copy Tree variable settings, it is possible to update variables with dynamic values. You can set dynamic variable values in the Settings dialog box - Copy Tree page.

Previously, you could only use static values.

To set dynamic variable values in Copy Tree:

1. In the Administration tool, in the Settings dialog box - Copy Tree page, under **File types**, click **Add**.

- 2. Click Add Variable and select predefined variables from the list.
- 3. For **Value**, click **>** to select a dynamic value.

The following dynamic values are available:

- Current time
- File Name
- File name without extension
- File Path
- Logged in user
- Target File Name
- Target File name without extension
- Target File Path
- Today's date
- User Full name
- User initials
- User User data

SOLIDWORKS PDM Application Programming Interface

See SOLIDWORKS API Help: Release Notes for late-breaking updates.

SOLIDWORKS PDM 2018 API includes the ability to:

- Add a custom tab to PDM vault views:
 - Customers and third-party users can add custom tabs to PDM vault views in Windows Explorer using a PDM API add-in.
 - API hooks to third-party development code can display special items in these custom tabs.
 - Add-ins can implement custom tabs with custom names and icons.
 - Add-ins can add any number of custom tabs to the vault view in Windows Explorer.
 - Add-ins can programmatically remove custom tabs.
- Programmatically refresh a data card when its card variables change.
- Restore deleted files from the Recycle Bin.
- Select the transition to use when changing a file's state.
- Specify whether to batch update only the variables that are part of the file data card.
- Get a list of values associated with a drop-down control on a data card.
- Provide more IEdmSearch comparators.

SOLIDWORKS PDM Support for Non-SOLIDWORKS CAD File References

SOLIDWORKS PDM supports the handling of references between SOLIDWORKS files and non-SOLIDWORKS CAD data created through 3D Interconnect.

You must add the non-SOLIDWORKS CAD files to the vault at the beginning so that they are referred from the file vault.

SOLIDWORKS PDM recognizes the non-SOLIDWORKS CAD references during check in of the parent SOLIDWORKS part or assembly and lists them as CAD references after check in is complete. You do not have to create any user-defined references. The CAD references are listed in the Contains tab, Bill of Materials tab, Where Used tab, and in any operation that lists the reference tree. This is true for both the parent SOLIDWORKS file and any parent non-SOLIDWORKS file.

For example, when you check in a SOLIDWORKS assembly along with an Inventor sub-assembly that is inserted in it, SOLIDWORKS PDM establishes PDM references between parent and its first level of child references. This also includes references between Inventor subassembly and its parts.

SOLIDWORKS PDM supports the following file formats through 3D Interconnect:

File format	Extension
Autodesk [®] Inventor	IPT, IAM
CATIA [®] V5	CATPART, CATPRODUCT
PTC [®] /CREO	PRT, PRT., XPR, ASM, ASM., XAS
Siemens [™] NX	PRT
SOLID Edge®	PAR, PSM, ASM

This integration does not affect the behavior of a non-SOLIDWORKS CAD file in its native application, that is when working with the file outside 3D Interconnect.

www.solidworks.com

Dassault Systèmes SolidWorks Corp. 175 Wyman Street Waltham, MA 02451 Phone: 1 800 693 9000 Outside the US: +1781 810 5011 Email: generalinfo@solidworks.com

Europe Headquarters Phone: +33 4 13 10 80 20 Email: infoeurope@solidworks.com

Japan Headquarters Phone: +81 3 6270 8700 Email: infojapan@solidworks.com

Asia/Pacific Headquarters Phone: +65 6511 9188 Email: infoap@solidworks.com

Latin America Headquarters Phone: +55 11 3186 4150 Email: infola@solidworks.com

Our **3D**EXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3DEXPERIENCE®** Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 220,000 customers of all sizes in all industries in more than 140 countries. For more information, visit www.3ds.com





Americas Dassault Systèmes 175 Wyman Street Waltham, Massachusetts 02451-1223 LISA

Europe/Middle East/Africa Dassault Systèmes 10, rue Marcel Dassault CS 40501 78946 Vélizy-Villacoublay Cedex

France

Tokyo 141-6020 Japan

Asia-Pacific Dassault Systèmes K.K. ThinkPark Tower 2-1-1 Osaki, Shinagawa-ku,