



WHAT'S NEW

SOLIDWORKS PDM 2024



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SOLIDWORKS PDM

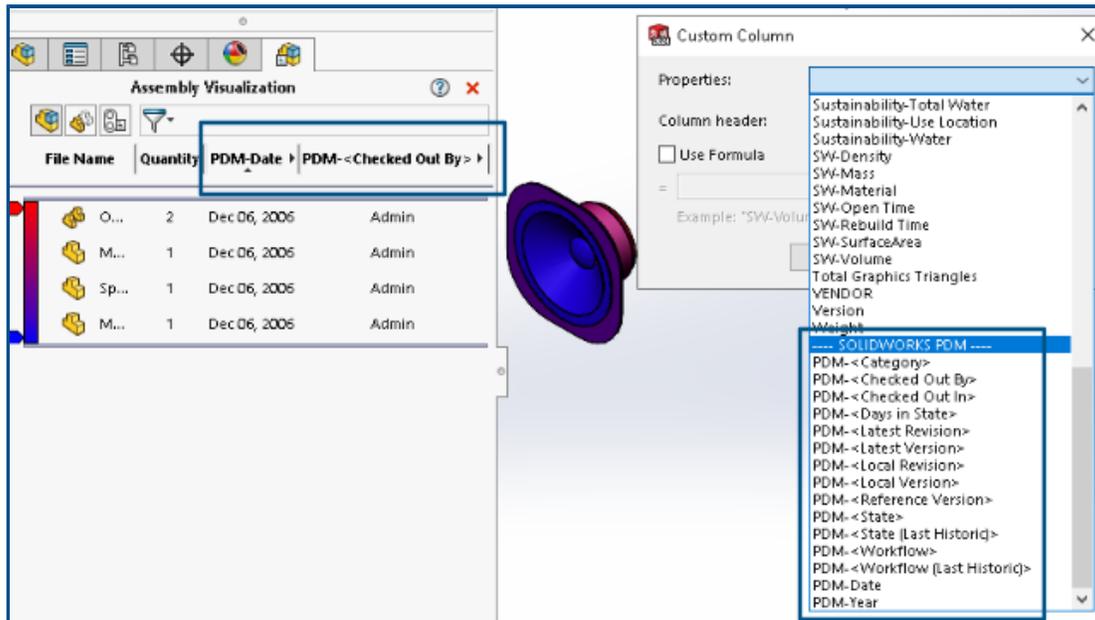
Topics:

- **Assembly Visualization**
- **Downloading Specific Versions of a File in Web2**
- **File Type Icons**
- **Check Out Option in Change State Command**
- **Progress Dialog Boxes (2024 SP1)**
- **Viewing Check-Out Event Details**
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- **Folder Card Variables in Web2 (2024 SP1)**
- **SOLIDWORKS PDM Add-in Enhancements (2024 SP1)**
- **Displaying the Preview Tab for Search Results (2024 SP2)**
- **Bill of Materials (BOM) View - Flattened Type (2024 SP2)**

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.

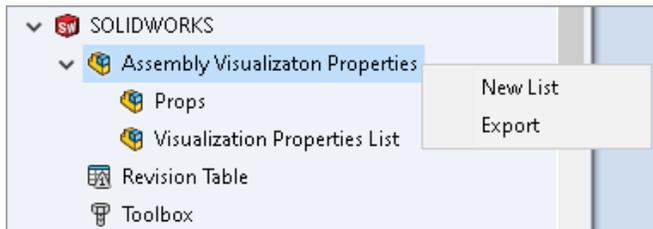
SOLIDWORKS PDM Professional is a full-featured data management solution for a small and large number of users, and is available as a separately purchased license.

Assembly Visualization



You can access SOLIDWORKS PDM variables in the SOLIDWORKS Assembly Visualization tool.

The SOLIDWORKS PDM variables are listed under **Properties** in the **Custom Column** dialog box of the Assembly Visualization tool. You can select variables, for example, **PDM-Checked Out By** or **PDM-Date** under the **SOLIDWORKS PDM** section in **Properties** and then view them in the Assembly Visualization panel.



To view SOLIDWORKS PDM custom variables in Assembly Visualization:

1. In the SOLIDWORKS PDM Administration tool, right-click **SOLIDWORKS > Assembly Visualization Properties** and click **New List**.
2. In the Customize Assembly Visualization Properties - Visualization Properties List dialog box, create a property list from the available variables. You can create multiple lists of properties and view them in Assembly Visualization depending on the permissions.

Customize Assembly Visualization Properties Dialog Box

You can use this dialog box to specify variables for specific users or groups that they can view in the SOLIDWORKS Assembly Visualization tool.

To open this dialog box:

1. In the Administration tool, expand **SOLIDWORKS**.
2. Right-click **Assembly Visualization Properties** and select **New List**.

Name

Specifies the name of the new properties list.

Variables

Variable	Displays the selected variable.
Name	Displays the name of the selected variable.
Add	Adds the selected variable.
Delete	Deletes the selected variable.
Up and down arrows	Moves the selected variables up or down.

Selected Variable

Variable	Displays the list of available variables and lets you select a variable from the list.
Name	Displays the name of the selected variable and lets you update the name.

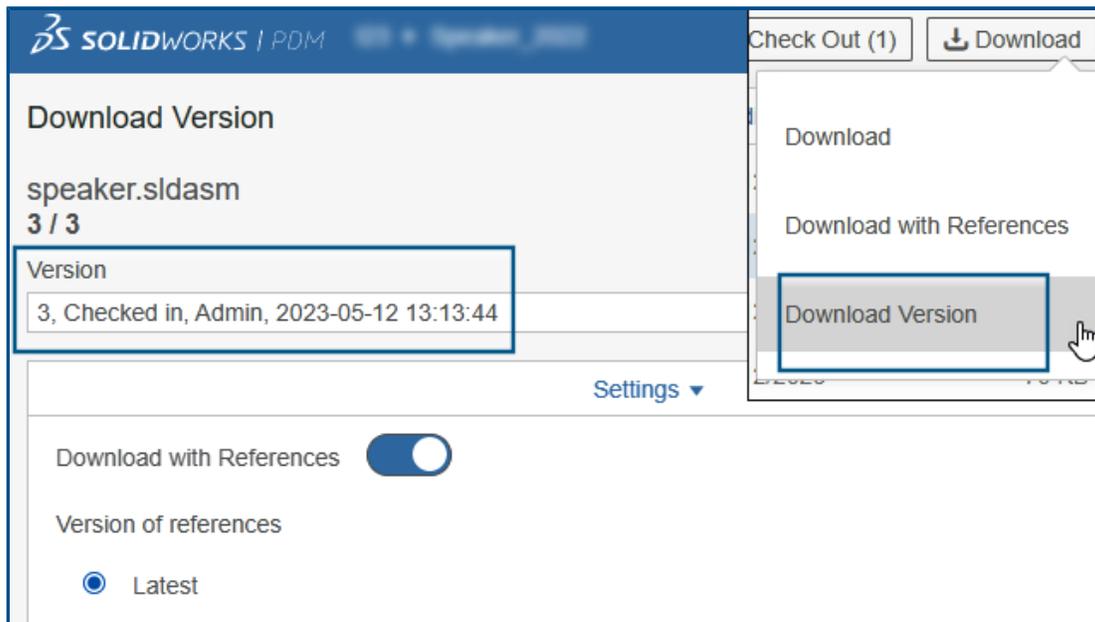
Users

Lists users and lets you specify users who can select the variables and view the list.

Groups

Lists groups and lets you specify groups whose members can select the variables and view the list.

Downloading Specific Versions of a File in Web2



SOLIDWORKS PDM Web2 lets you download a specific version of a file and its references.

You cannot select and download multiple files in a single operation.

The Download Version dialog box lets you select the version and settings for download.

To access this dialog box:

1. In the File list, select a file:
 - Large screen layout. Click **Download** > **Download Version**.
 - Small screen layout. Touch **Download** and then touch **Download Version**.

Download Version Dialog Box

You can use the Download Version dialog box to download a specific version of a file and its references.

To open this dialog box:

- Select a file and click **Download** > **Download Version**.

Version

Select the version of the file to download.

Settings

The collapsible option that displays the download settings options for files.

Download with references	Downloads the file with its references.	
Version	Latest	Downloads the latest version.
	Referenced	Downloads the referenced versions.
Preserve relative paths	Preserves the paths of references relative to the parent file and creates a folder structure as required. When cleared, the folder hierarchy is flattened, and all referenced files are uploaded to the same destination folder as the parent file.	
Include drawing	Downloads the drawing files associated with the file selected to download.	
Include simulation	Downloads the SOLIDWORKS Simulation results associated with the selected files.	

Files

Lists the file references to download. The file list includes customizable columns such as **State**, **Version**, **Size**, and **Path**. Click **Show More**  and specify the columns to display.

Total Files to Download

Displays the total number of files and the count of individual files to download.

Download

Downloads the selected files. When the download is complete, a message appears with the number of downloaded files on the upper bar. If Web2 cannot download any references, a warning message appears.

Download Version Dialog Box - Small Screen Layout

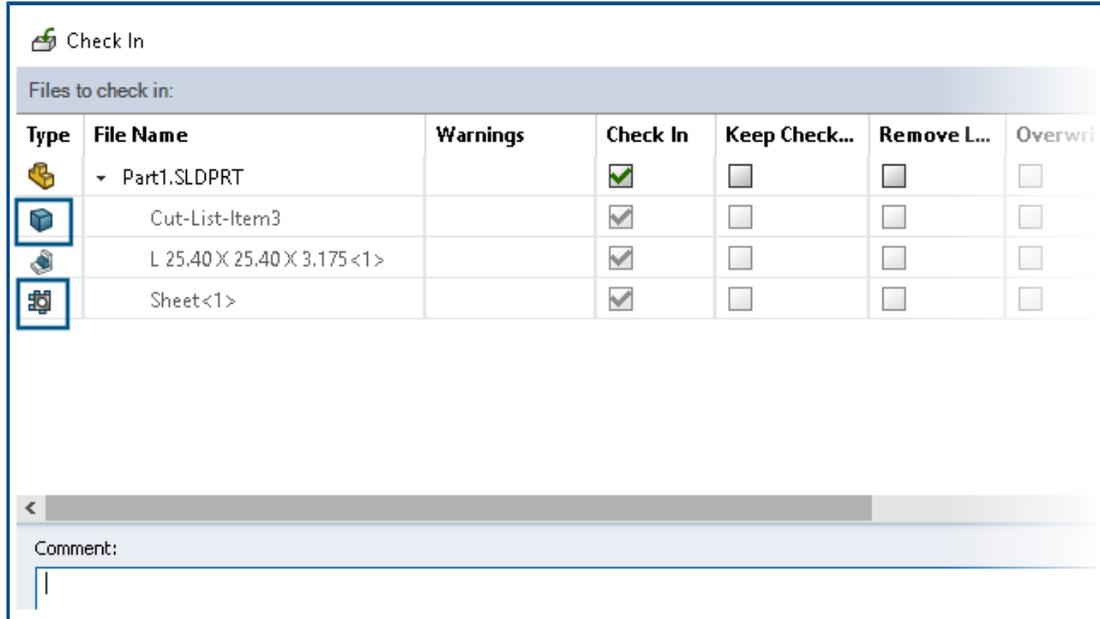
You can use the Download Version dialog box to download a specific version of a file and its references.

To open this dialog box:

1. Select a file and touch **Download**.
2. Touch **Download Version**.

Filename and latest version	Displays the version list and where you can select a version to download.
Settings	Lets you specify options.

File Type Icons



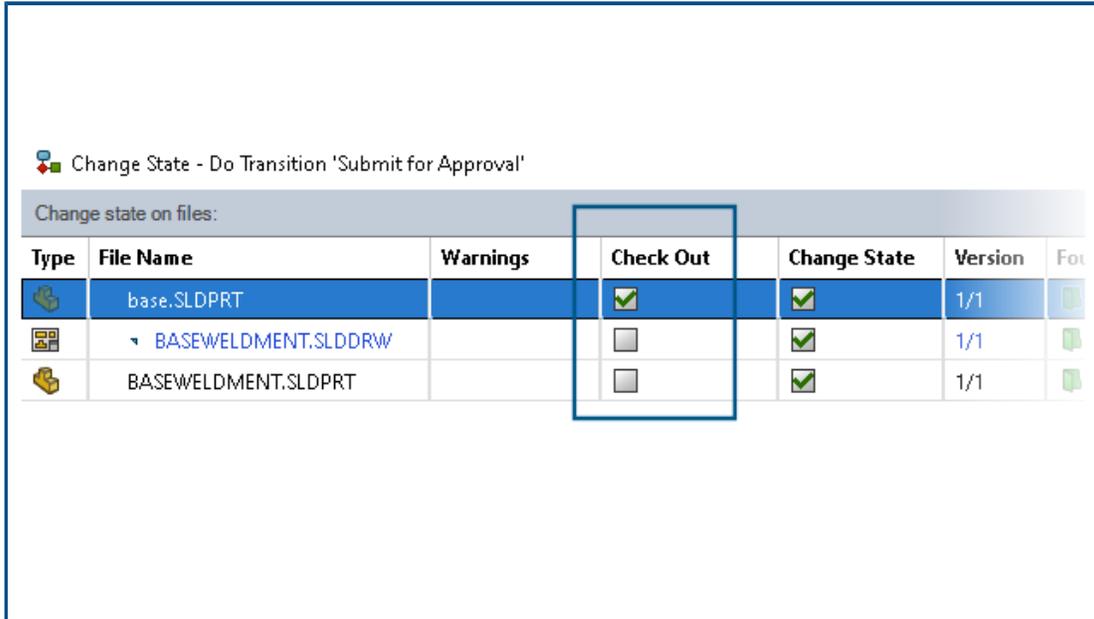
You can view the file type icons for weldment cut list items and the files that were shared using pasted shared overlays.

These icons are available in the dialog boxes for:

- File Details
- File Operations
- Web2

The type icons for cut list items are not available for SOLIDWORKS BOMs.

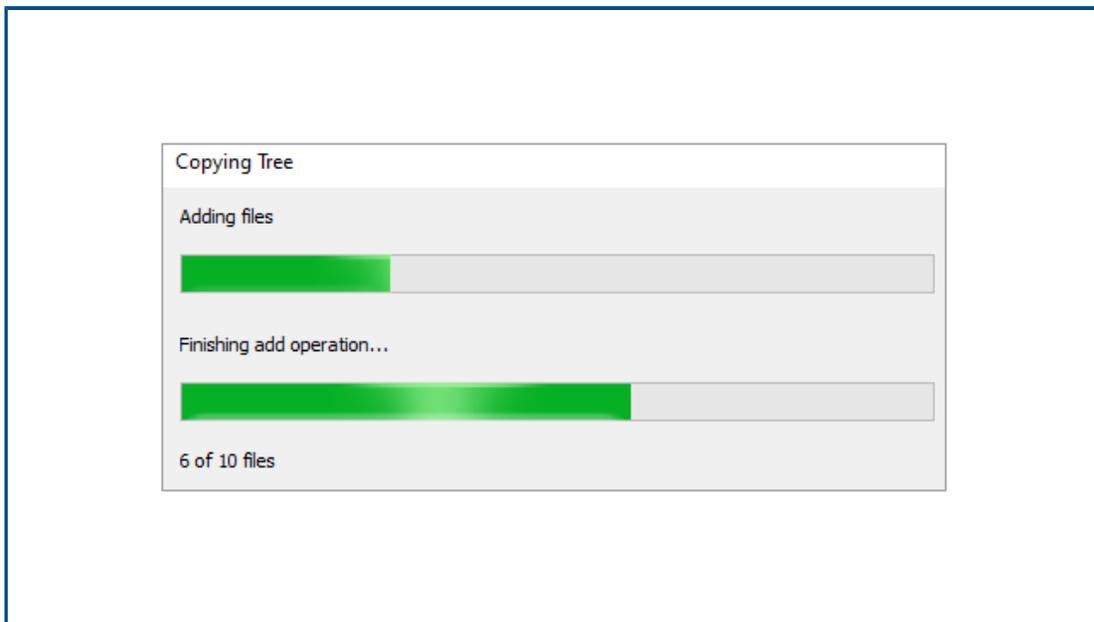
Check Out Option in Change State Command



You can check out a file after the change state operation completes.

You can customize the column set of the Do Transition dialog box to include the **Check Out** system variable. If you select **Change State** and **Check Out** for a file, the file is checked out after its state changes.

Progress Dialog Boxes (2024 SP1)



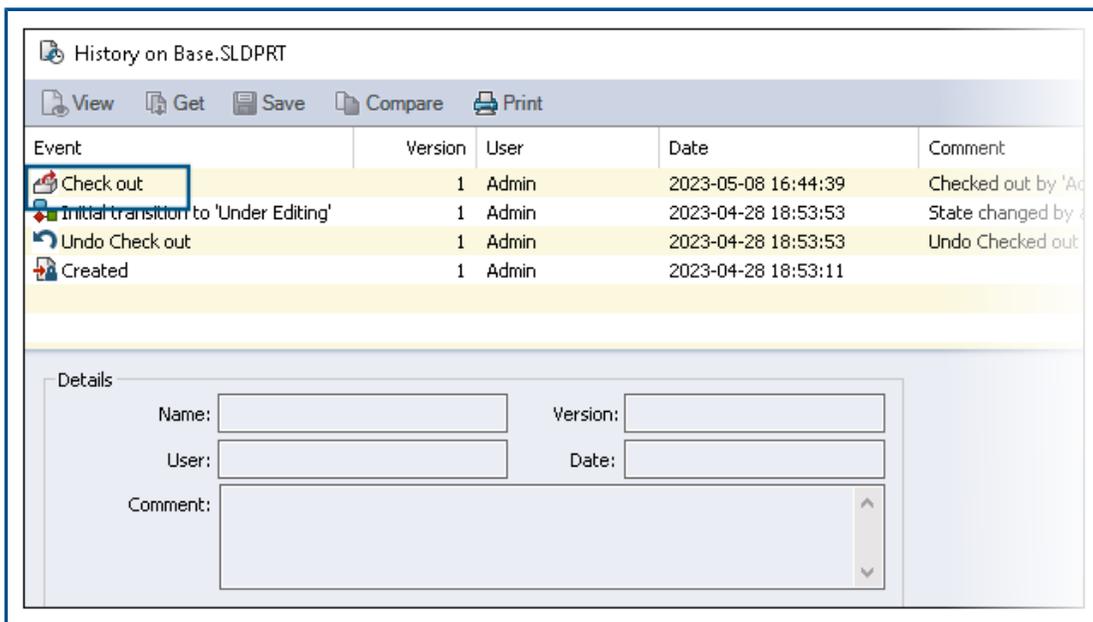
In the SOLIDWORKS PDM File Explorer, the progress dialog box of certain operations displays more information.

The Change State and Copy Tree progress dialog boxes have two progress bars:

- The first progress bar has the primary steps or actions of the overall operation, such as **Copying Files** and **Copying Variables**.
- The second progress bar has detailed information such as secondary steps, total number of files, etc.

The Check In and Reading File References progress dialog boxes have a single progress bar that displays the current action and file names.

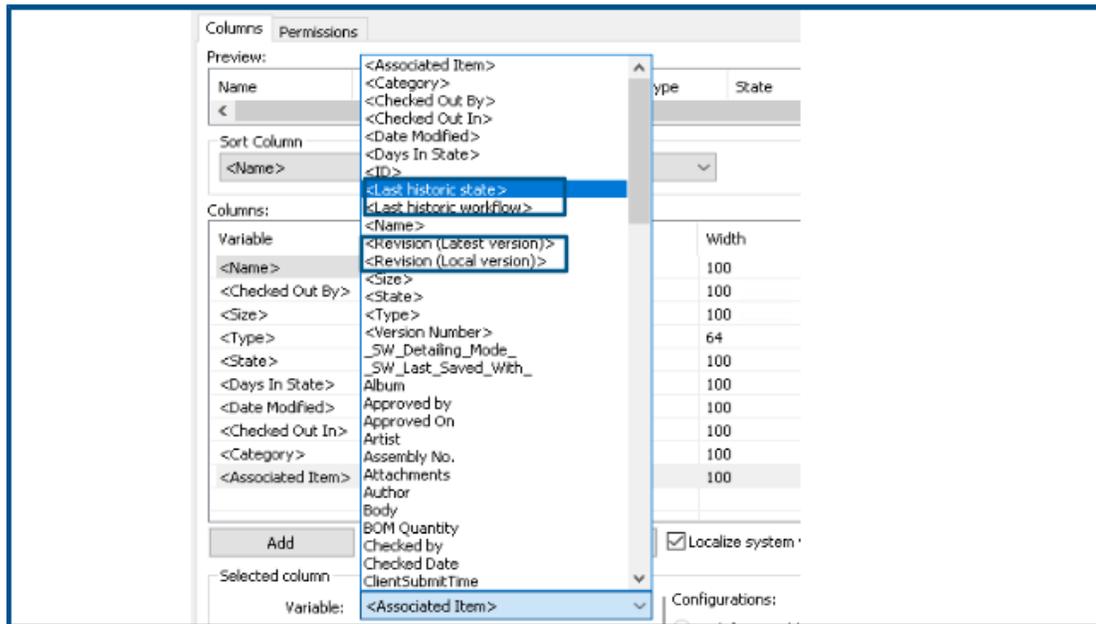
Viewing Check-Out Event Details



In SOLIDWORKS PDM File Explorer, you can view details of check-out and undo check-out events in the History dialog box of a file.

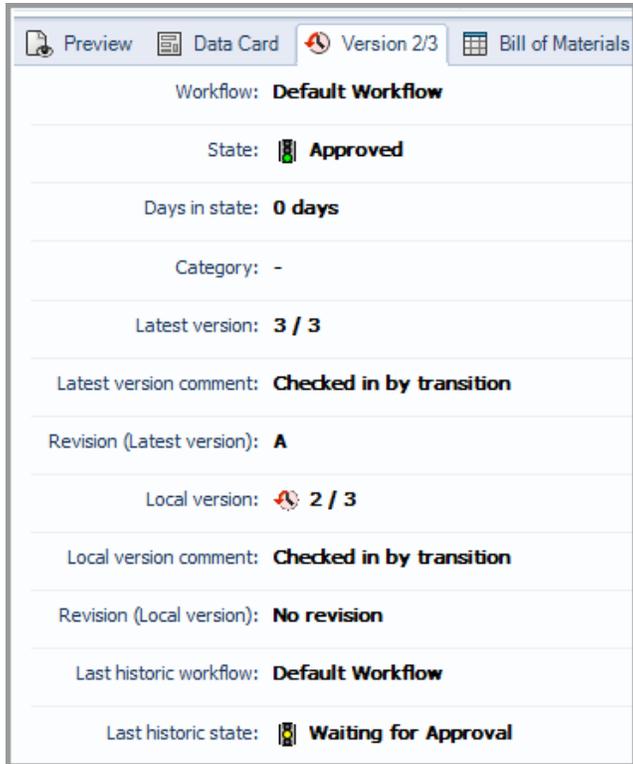
Along with the other details, you can see which user has performed the operation.

System Variables

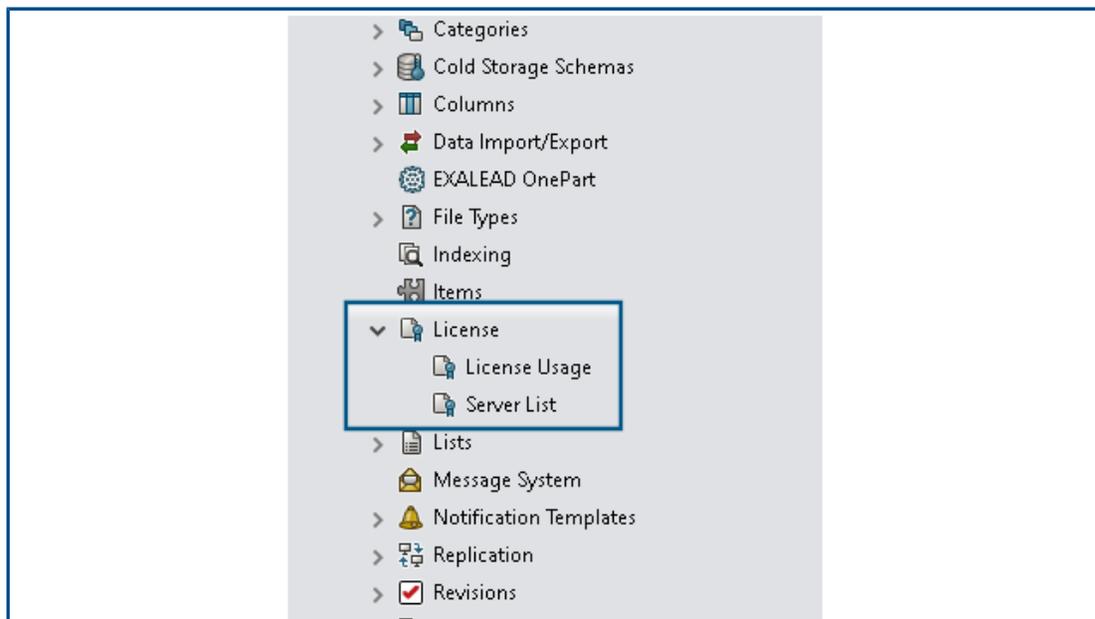


System variables are more available and easier to access.

- The following system variables are available in the **File List**, **Quick Search Result**, and **Search Result** column set types:
 - **<Last historic state>**
 - **<Last historic workflow>**
 - **<Revision (Latest version)>**
 - **<Revision (Local version)>**
- The **<Days in State>** system variable is available as a default column in **File list**.
- The SOLIDWORKS PDM task pane add-in has more system variables.
- In SOLIDWORKS PDM File Explorer, the addition of more system variables improves the user interface of the Version tab.



Viewing License Usage



You can view license details without any special administrative permissions.

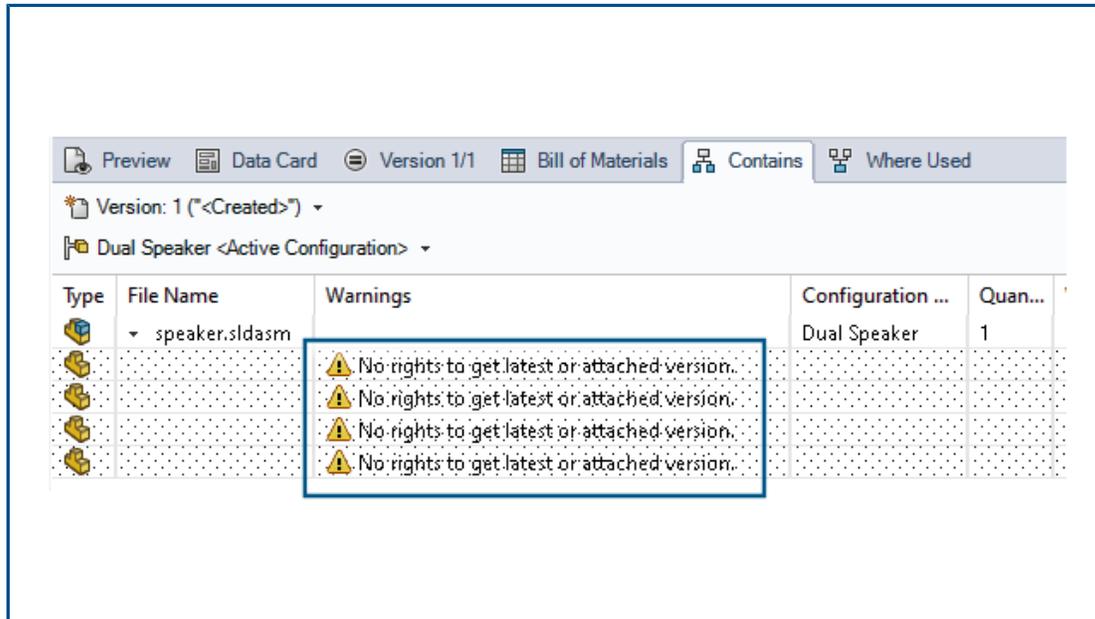
In the Administration tool, the **License** node has the following subnodes:

- **Server List.** Lets you edit license servers.

The administrative permission **Can update license keys** is renamed as **Can update license server**. You need this permission to edit license servers.

- **License Usage.** Lets you view license details. This helps you to ask users to log out if they are not using the tool, request more licenses from the administrator, or decide whether you need to switch to a different license type.

Data Security Enhancements (2024 SP1)



In SOLIDWORKS PDM File Explorer and Web2, unauthorized users cannot view file information in file view tabs or in file operations and file reference dialog boxes.

The warning message **No rights to get latest or attached version** displays for the following:

- File view tabs:
 - Contains
 - Where Used
 - Bill of Materials (Computed BOMs and Named BOMs)
- File operations dialog boxes
- File reference dialog boxes

SOLIDWORKS PDM Performance Improvements

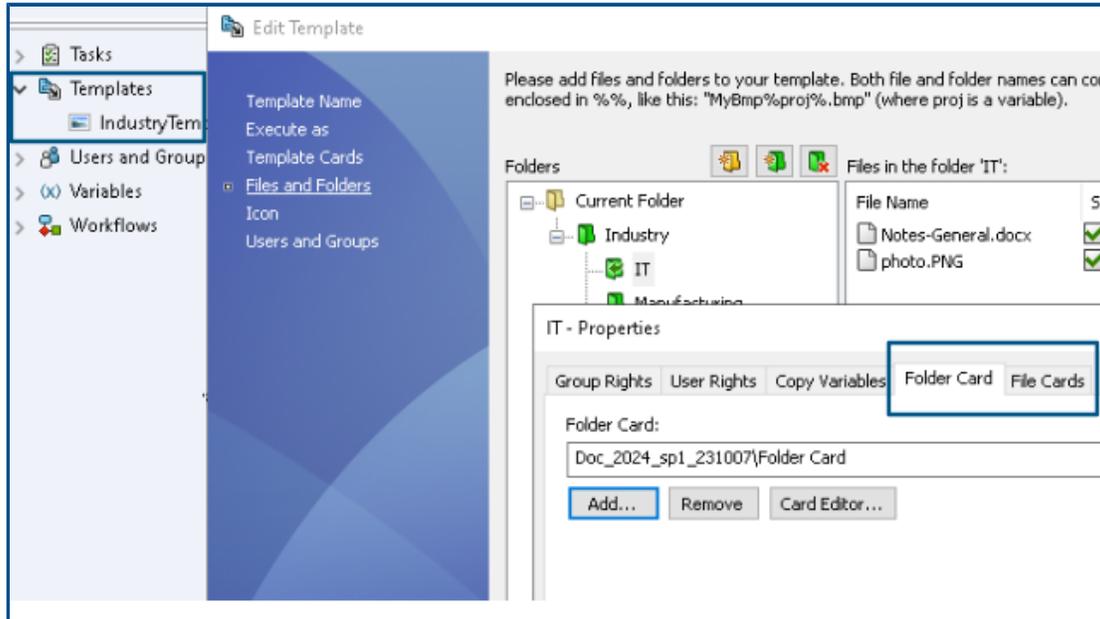
SOLIDWORKS PDM 2024 has improved the performance of file-based operations.

The following operations are approximately two times faster:

- Add files
- Change state

- Copy tree
The copy tree to compressed archive operation is orders of magnitude faster.

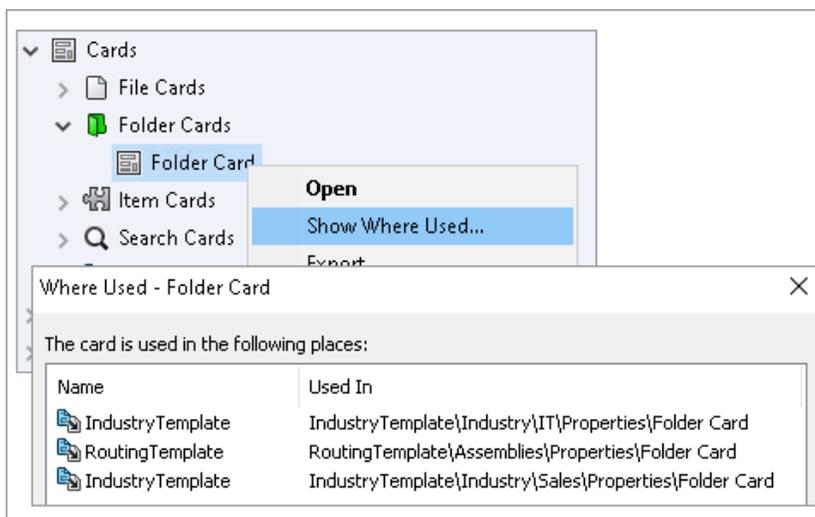
Assigning Data Cards to Files and Folders of a Template (2024 SP1)



In the SOLIDWORKS PDM Administration tool, while creating and editing a template, you can assign a folder card and multiple file cards to a folder.

In SOLIDWORKS PDM File Explorer, right-click and click **New** in the right pane. When the software creates the files and folders structure, the respective data cards are assigned automatically.

Changes to the file extensions for a card, assigned to a template, outside of the template configuration are not recognized.



In the SOLIDWORKS Administration tool, under **Cards** , for each file, folder, and template card, you can right-click and see where the card is used. For example, click **Cards** > **Folder Cards** > **Folder Card** > **Show Where Used**. This option is useful when deleting a file or a folder data card.

Where Used Card Dialog Box

You can use this dialog box to display where a file, folder, or template card is used.

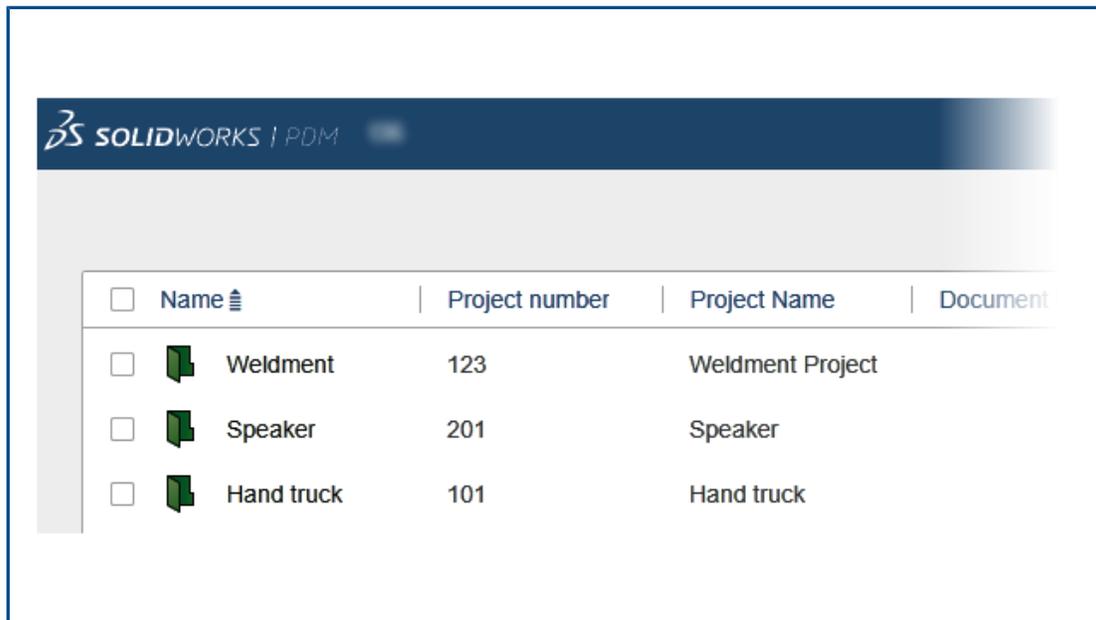
To open this dialog box:

1. In the Administration tool, expand **Cards** .
2. Expand a file, folder, or template card menu, for example **Folder Card**
3. Right-click the card.

You can see a list of all the places where the card is used:

Name	Displays the template using the card.
Used In	Displays where the card is used.

Folder Card Variables in Web2 (2024 SP1)



The screenshot shows the SOLIDWORKS PDM interface with a table displaying folder card variables. The table has four columns: Name, Project number, Project Name, and Document. The 'Name' column is sorted by the 'Name' header. The table contains three rows of data, each with a checkbox, a folder icon, and the variable name.

<input type="checkbox"/>	Name 	Project number	Project Name	Document
<input type="checkbox"/>	 Weldment	123	Weldment Project	
<input type="checkbox"/>	 Speaker	201	Speaker	
<input type="checkbox"/>	 Hand truck	101	Hand truck	

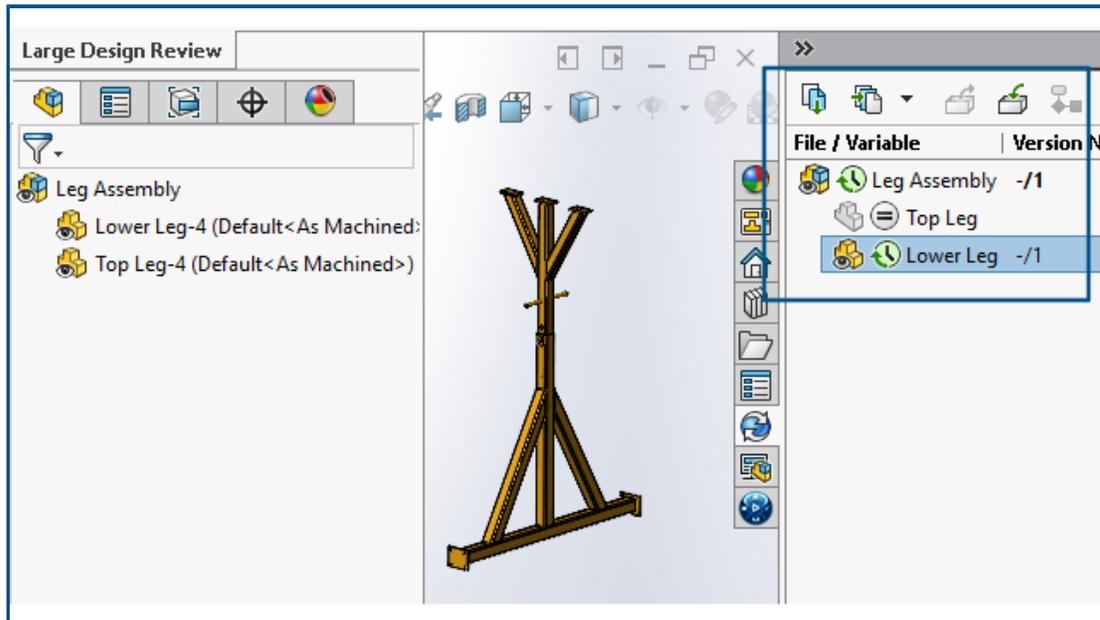
In Web2, you can view data card variables for folders in a folder list. The values for custom columns for the folders are displayed in the list view of the large screen layout.

SOLIDWORKS PDM Add-in Enhancements (2024 SP1)

File / Variable	Value	Version Number	Checked Out By
Assem1 (Default)		-/1	Admin
Checked out by	Admin		
Category			
Part1 (...)		1/1	Admin
Checked out by	Admin		
Category			
countersunk nib_is (...)		2/2	
Checked out by			
Category	-		

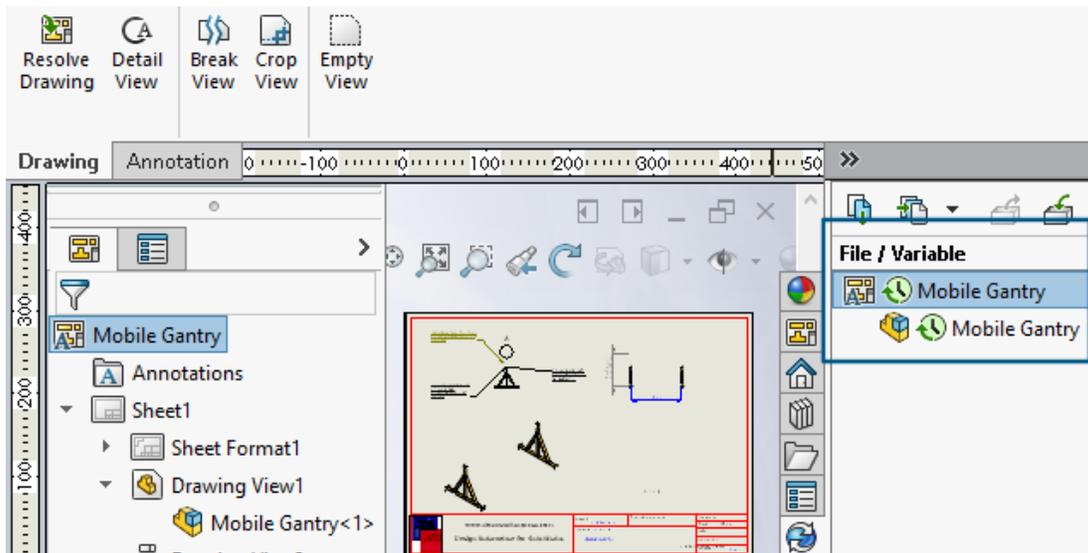
- When you save an assembly file as a part file, an internal component (saved as an external file in the vault), or a mirror component using the **Save as** command, a data card for the new file displays generating serial numbers and default values if set in the card.
- The SOLIDWORKS PDM add-in displays an icon overlay and supports all SOLIDWORKS PDM operations for components that are open in lightweight mode.
- You can enable the **Automatically optimize resolved mode, hide lightweight mode** option even when the SOLIDWORKS PDM add-in is active.

Handling Large Design Review (LDR) and Detailing Mode in the SOLIDWORKS PDM Add-in (2024 SP2)



For assemblies opened in **Large Design Review (LDR)** mode and for drawings opened in **Detailing** mode, you can view the SOLIDWORKS files structure in the SOLIDWORKS PDM Task Pane (along with icons) similar to the FeatureManager design tree.

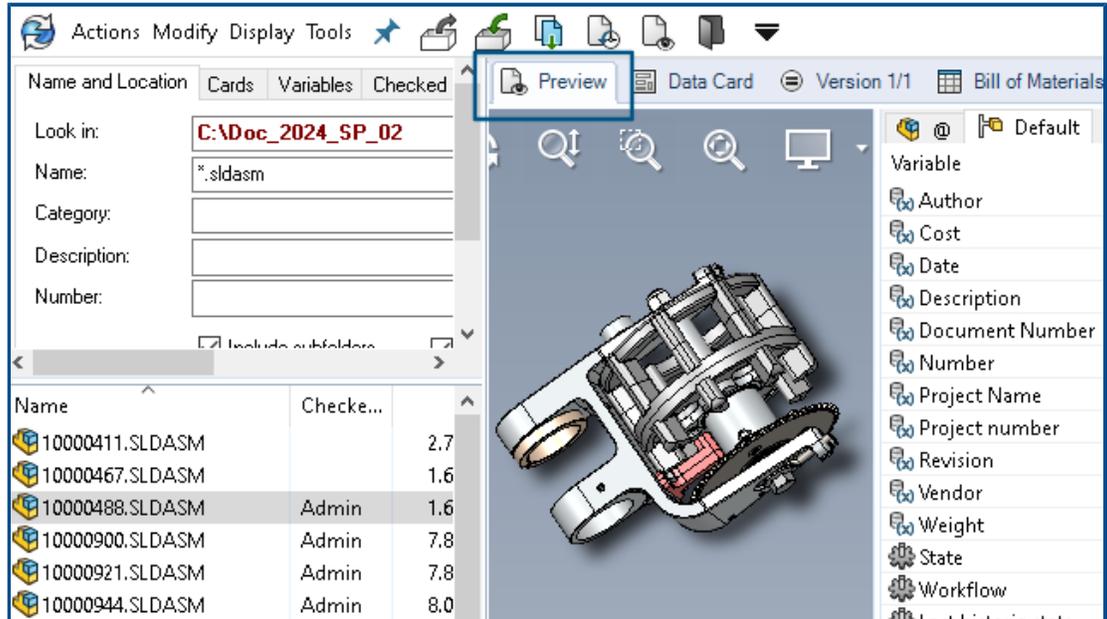
Because the display of both the FeatureManager design tree and the Task Pane tree are identical, you can work on the product structure with more clarity and ease.



For the **Detailing** mode, the PDM Task Pane tree displays child components only to the first level similar to the FeatureManager design tree.

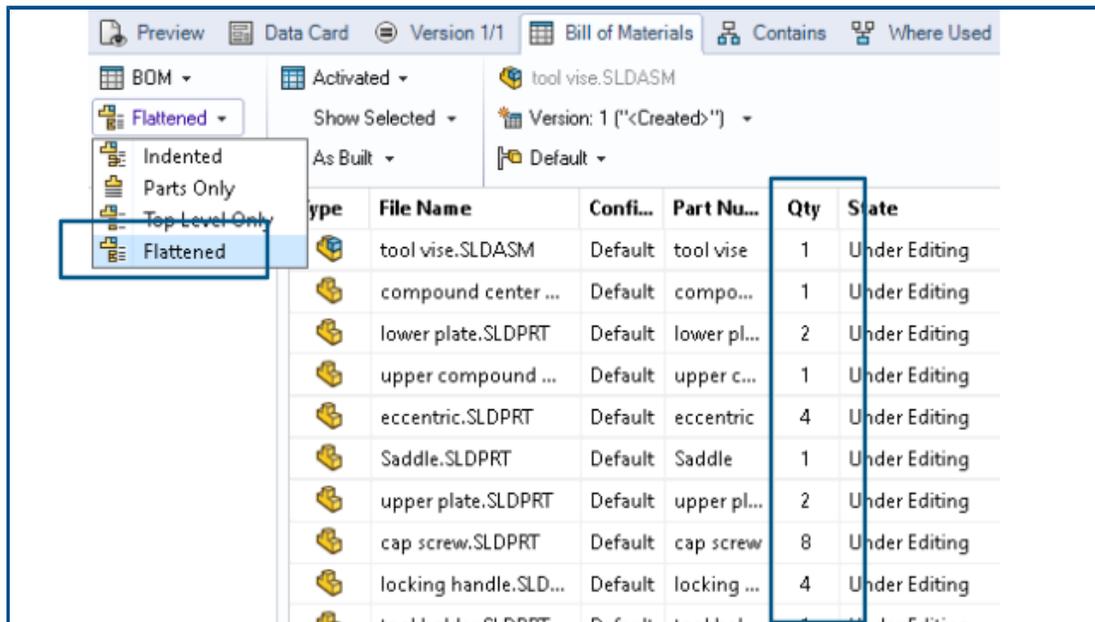
For the **Large Design Review (LDR)** mode, you can perform SOLIDWORKS PDM operations such as **Check in** and **Check out** on the components from both the FeatureManager design tree and the Task Pane assembly tree.

Displaying the Preview Tab for Search Results (2024 SP2)



In the SOLIDWORKS PDM File Explorer, you can display the **Preview** tab for an item in the search result (Quick, Integrated, and Standalone search) at the bottom or to the right side of the window using the existing **Preview Placement** option.

Bill of Materials (BOM) View - Flattened Type (2024 SP2)



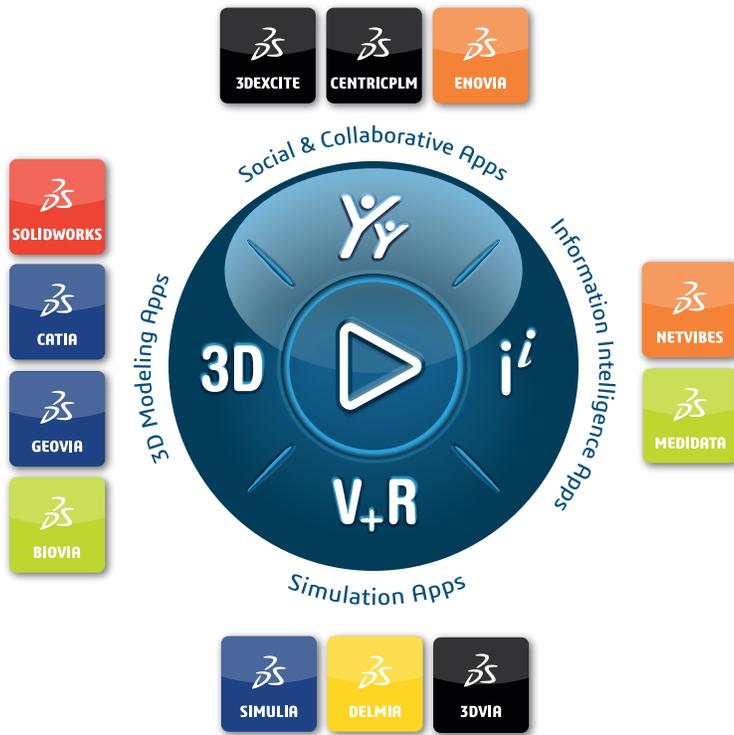
In the SOLIDWORKS PDM File Explorer, in the BOM view of the **Bill of Materials** tab, you can use the new type **Flattened** to view the total number of quantities required of a component present in the product structure.

This option saves time and effort in calculating the total number of quantities of the components.

The **Flattened** BOM view displays:

- The product structure as a list of components without indentation.
- The component only once if it is present at multiple levels of the product structure.
- The quantity of the component by adding the quantities at each level.

The **Flattened** type is available when viewing the computed BOMs in the desktop client and in Web2.



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 10, rue Marcel Dassault
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 France

Asia-Pacific
 Dassault Systèmes K.K.
 ThinkPark Tower
 2-1-1 Osaki, Shinagawa-ku,
 Tokyo 141-6020
 Japan

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

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