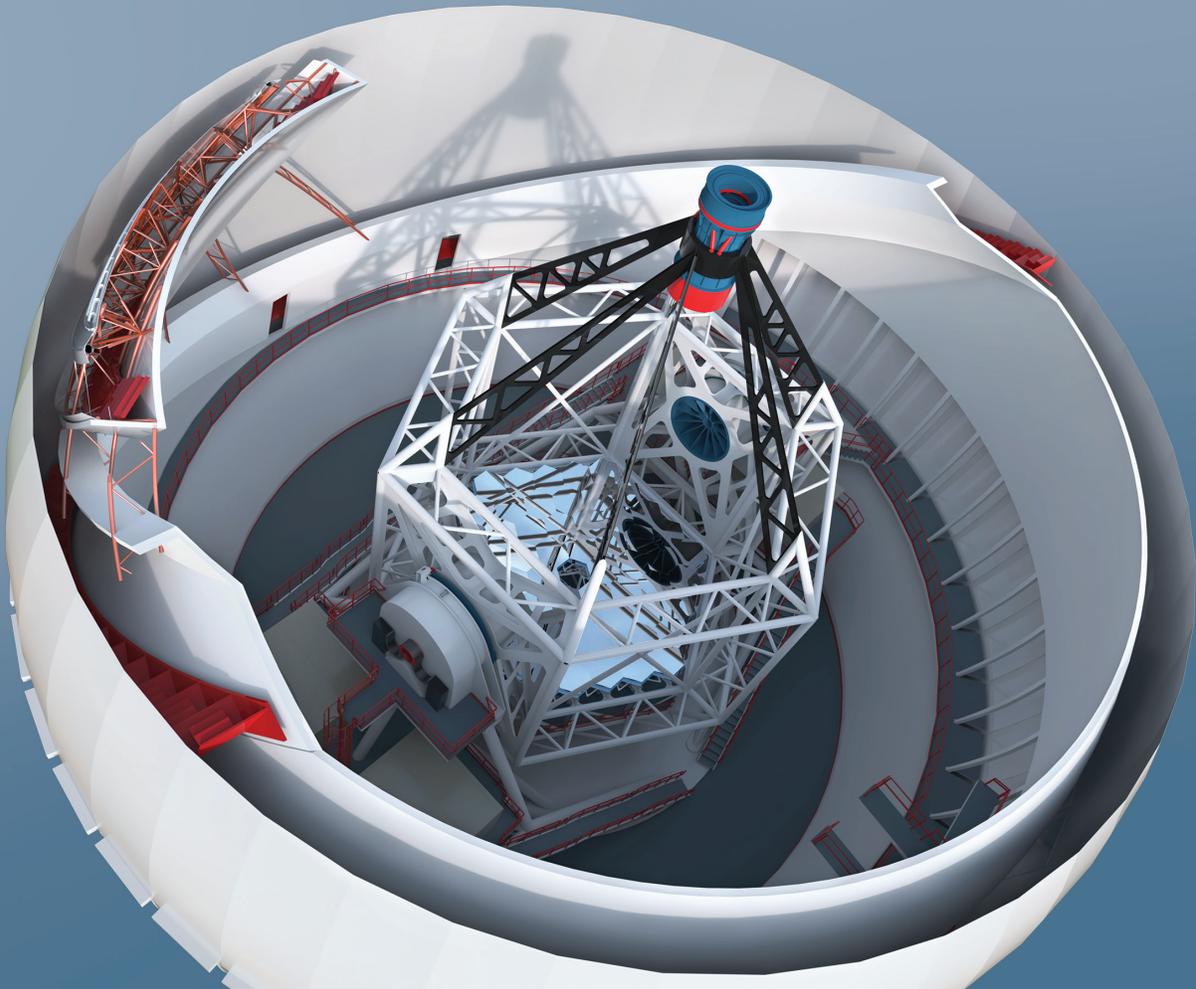


WHAT'S NEW

SOLIDWORKS PDM 2019



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SOLIDWORKS PDM

This chapter includes the following topics:

- **Conditional Notifications**
- **Default Search Variable in Search Cards**
- **Default Values Generation for New Sheets in Drawings**
- **Editing History Comments**
- **File Shortcut Menu**
- **Integrated Installation of Web API Server**
- **Mixed Authentication Support**
- **Performance Improvements - SOLIDWORKS PDM**
- **Responsive Redesign in Web2**
- **Selection of Referenced Drawing Nodes During Change State**
- **SOLIDWORKS Inspection Integration**
- **Support for Configurations of Weldment Cut Lists**
- **Support for DXF/DWG File Format in Convert Task**
- **Support for eDrawings Functionality**
- **Upload and Check in File Structure and Download with References**
- **User Comments**

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.

Conditional Notifications

In SOLIDWORKS PDM Professional, you can define conditional notifications for transitions.

With the **Add Conditional Notification** option, you can:

- Add predefined conditions to send transition notifications to users and groups.
- Send notifications dynamically to users based on the value of data card variables.

Defining Conditional Notifications

To define conditional notifications:

1. In the Administration tool, click a workflow transition.
2. In the transition's Properties dialog box, in the Notifications tab, click **Add Conditional Notification**.
3. In the Conditional Notification dialog box, in the Recipients tab:
 - a) Click **Add Users/Groups**.
 - b) In the Add Users/Groups dialog box, select the users or groups to receive the notification, and click **OK**.
 - c) Click **Add Variable** to select variables to use their values as notification recipients.
 - d) In the Add Variable dialog box, select the variables, and click **OK**.
4. In the Notification Conditions tab:
 - a) Select **<Click here to add a condition>**.
 - b) In the conditions list, select a variable or a condition.
 - c) Under **Comparison**, expand the list and select a comparison operator.
The list of operators depends on the selection you made in the previous step.
 - d) Under **Value**, enter the value to compare with the variable value.

If you selected **User** or **Group** as **Variable**, select the value from the list.
 - e) If you selected a variable as the condition, specify the data card configuration to search for the variable.
If you want to search for all configurations, leave **Configuration** blank.
5. Click **OK** twice and save the workflow.

Conditional Notification Dialog Box

You can use the Conditional Notification dialog box to define conditional notifications for workflow transitions.

To display the Conditional Notification dialog box:

1. Open the transition's Properties dialog box.
2. In the Notifications tab, click **Add Conditional Notification**.

Notification Properties Tab

The Notification Properties tab displays the state fields of the transition. These fields are read-only.

Source state	Displays the initial state of the folder.
Destination state	Displays the final state of the folder after transition.
State change	Displays the transition used to trigger the state change.

Recipients Tab

Use the Recipients tab to define which users and groups receive notification when the condition is met.

Add Users/Groups	Opens the Add Users/Groups dialog box. You can select users and groups to receive the notification.
Add Variable	Opens the Add Variable dialog box. You can select variables to use their values as notification recipients. <div data-bbox="609 552 1424 669" style="border: 1px solid gray; padding: 5px; margin: 5px 0;">The value of selected variable is taken from the data card and is compared with the full name, login name, or group name value in the database vault.</div> <div data-bbox="609 688 1424 779" style="border: 1px solid gray; padding: 5px; margin: 5px 0;">You can filter the list to include only variables whose values contain the filter string.</div>
Remove	Removes selected recipients.

Notification Conditions Tab

Use the Notification Conditions tab to create the conditions for users to receive automatic notifications.

The conditional notifications apply on the vault root and all folders by default. To define them for specific file types or file name, you need to define the file path-based conditions.

Variable

Lists the conditions and variables. You can select a variable or one of the following:

Or

Creates an OR container. By default, all conditions must be true (AND). Use OR containers to group conditions where only one condition must be true.

Category

Compares the category of the file with the **Value**.

Filepath

Compares the file name, extension, or path with the input **Value**. The file path replaces " *.* " in the folder path with the input **Value**.

Object type

Compares the object type (**BOM, File, or Item**) with the **Value**.

Revision

Compares the revision number of files with the **Value**.

User

Compares the name of the user who runs the transition with the **Value**.

Group

Compares the group of the user who runs the transition with the **Value**.

Remove

Removes the selected condition or variable.

Comparison

Lists the comparison operators for each condition.

Expand the list to choose a text, numeric, or date comparison operator.

For example:

	Text Does Not Contain		Date Not Equal To
	Number Less Than		Yes or No Equal To

The comparison operators that are available depend on the variable type selected.

Value

Lets you enter the value to compare with the value of the variable.

Configuration

Specifies the data card Configuration tab to search for the value.

This option is available only when the condition is a variable.

Leave **Configuration** blank to search for all configurations.

Default Search Variable in Search Cards

You can define a logged-in user name as the default value on search cards.

Previously, you had to define the logged-in user name as the default value using a search favorite.

Configuring the Default Value in Search Card

You can configure the default value in search card to define the default search variable.

To configure the default value in search card:

1. In the Administration tool, expand **Cards > Search Cards** and double-click a search card.
2. In Card Properties, click **Default Values**.
3. In the dialog box, select a variable and enter a value.

If you select a variable such as **Checked Out by User, Label - by User, Version Created by User, Workflow - Transitions by User, Author**, enter the value as %user%.

4. Click **OK** and save the changes to the Card Editor.

In the File Explorer, when you run the Search tool, the logged-in user name appears as the default search value. For example, in step 3, if you select **Checked Out by User**, the logged-in user name appears on the Checked in/out tab in the Search tool.

In the Administration tool, in the Settings - Admin dialog box, if you select **Show full user names** and if the full name is available, it appears in the Search tool.

Default Values Generation for New Sheets in Drawings

When you add new sheets to a checked-out SOLIDWORKS drawing file, SOLIDWORKS PDM generates default values for the newly added sheets and updates the values in the drawing data card.

Previously, you had to manually update the values of mandatory variables in the drawing data card before check-in.

You must enable the SOLIDWORKS PDM add-in before adding the new sheets.

In the Administration tool, in the SOLIDWORKS Drawing Card, if you select **Updates all configurations**, the variables that use the *\$PRPSHEET* type variable mapping are updated on the tabs for all sheets. Previously, only the values on the as-built (@) configuration tab updated automatically and you had to manually update the active configuration tabs.

Editing History Comments

You can edit your own history comments only if you have the administrative permission, **Can update history comments**.

Previously, you could edit your own history comments without any permission.

Set/Delete Labels Permission

The administrative permission, **Can set/delete labels**, is separated into two permissions.

Permission	Description
Can set labels	Lets users add labels.
Can delete labels	Lets users delete any user's labels.

- A user cannot edit other user's labels.
- To make the history read-only for a user, the administrator must clear **Can update history comments** and **Can delete labels**.

Deleting Labels

You can delete labels with the administrative permission, **Can delete labels**.

To delete labels:

1. Select the file or folder and click **Display > History** or **History** (Microsoft® Windows® Explorer toolbar).
2. In the dialog box, under **Event**, select labels to delete.
3. Click **Delete Labels**.
4. Click **Yes** to confirm deletion.

File Shortcut Menu

The file shortcut menu for a user appears in the Where Used, Contains, and Bill of Materials tabs.

You can more easily perform file operations on parent and child references, multiple references, or references that are spread across vaults.

Previously, this menu was only available for Windows Explorer File View. To perform the file operations in these tabs, you had to manually browse or search the files.

The file shortcut menu on the Contains, Where Used, and Bill Of Materials tabs is available in the:

- File Explorer
- Complete Search and Search tool
- SOLIDWORKS PDM File Viewer interface
- Open and Save dialog boxes where access to these tabs exists.

You must have the required permissions to perform the file operations.

The file shortcut menu is:

- Available for computed BOMs, CAD BOMs, and checked-in named BOMs.
- Not available for checked-out named BOMs, weldment BOMs, and cut lists.

Integrated Installation of Web API Server

EXALEAD® OnePart users can install and configure the Web API server during installation of the SOLIDWORKS PDM Professional server.

Previously, you had to install the Web API server components separately.

EXALEAD OnePart uses Web APIs to access SOLIDWORKS PDM data. With the integrated installation, you can:

- Install the Web API server
- Configure Microsoft Internet Information Services (IIS)
- Add and configure vaults

You can install the Web API server on the same computer with other SOLIDWORKS PDM server components or separately on another computer.

Mixed Authentication Support

An administrator of a vault can set the option to allow users to log in as Microsoft Windows users and SOLIDWORKS PDM users. Previously, the administrator had to select a single login type for all vault users.

This lets you have a mix of login types per vault. The administrator can have users that are not in the Microsoft Windows domain but need to connect to a vault.

Authenticating Users with Mixed Logins

To authenticate users with mixed logins:

1. Click **Start > All Programs > SOLIDWORKS PDM > Archive Server Configuration**.
2. Click **Default Settings** .
3. Select **Windows login**.
4. Select **Allow SOLIDWORKS PDM login**.
5. Click **Apply** and **OK**.

In the Administration tool, when adding a new user in the Add Users dialog box:

- The new option, **New SOLIDWORKS PDM User**, adds a SOLIDWORKS PDM user.
- The renamed option, **New Windows User**, adds a Microsoft Windows user.
- The renamed option, **List Windows Users**, lists the Microsoft Windows users.

Performance Improvements - SOLIDWORKS PDM

SOLIDWORKS PDM has improved performance of loads for large assemblies and complex reference structures in the Contains and Where Used tabs.

You can more quickly complete operations such as moving, dragging, browsing, renaming files, and adding a large number of files to a folder that has many files. You can also check in assemblies with virtual components much faster.

Browsing speed has improved while working on integrated search results.

The elimination of unnecessary interface refreshes speeds up your browsing experience, including folders with many locally cached files.

Responsive Redesign in Web2

The redesign of SOLIDWORKS PDM Web2 makes the website responsive to various device sizes and browser window sizes. The user interface has improved controls plus enhanced support for touch-enabled devices.

Enhancement	Description
Responsive Log In screen	The Log In screen resizes to fit your device and when you resize the browser window.
Responsive and enhanced file list	The file list and file properties also resize with your device. The file list is a continuous scrolling list that lets users browse the vault directory content. Previously, the file list was paginated so users had to navigate through multiple pages.
Navigation bar	The navigation bar lets you: <ul style="list-style-type: none"> • Navigate to different folders in a vault using breadcrumbs. • See the alerts with information on the top bar for file actions such as Change State.
Action bar	The Check Out, Undo Check Out, Delete, Change State, and Download options are available on the action bar.
Choose column	Select or clear the columns to appear in the file list. When you add a column, its width adjusts automatically.
Resize and sort column	Resize the columns on pages with a column view including the Where Used and Contains tabs.
Upload and Check-In	Drag and drop files to upload and check them in.
Search bar	Designate the search location to Current Folder, Current & Subfolders, or All Folders .
File preview	The file preview resizes to fit your device.

Selection of Referenced Drawing Nodes During Change State

You can choose to select the referenced drawing nodes associated with a file when changing the file state.

Previously, you could choose to select or clear all referenced files during the change state operation. You could not specify just the referenced drawing nodes.

The new option, **Do not select references that are defined as drawing nodes during change state**, is cleared by default.

To access this option:

1. In the Administration tool, right-click a user or group and click **Settings**.
2. In the left pane of the user's Settings dialog box, click **Reference Dialog**.

When you change the state of the parent file, the software performs as follows:

Option	Description
Selected	Referenced drawing nodes are not selected (the check boxes in the Change State column are not selected) and are excluded from the change state operation.
Cleared (default)	Referenced drawing nodes are selected and included in the change state operation.

If multiple files are selected for the change state operation, the option selects or clears the check boxes for their referenced drawing nodes in the **Change State** column.

SOLIDWORKS Inspection Integration

SOLIDWORKS Inspection integrates with SOLIDWORKS PDM. The integration covers the SOLIDWORKS Inspection add-in to SOLIDWORKS and the SOLIDWORKS Inspection standalone application.

SOLIDWORKS Inspection standalone supports:

- Variable mapping for inspection project files

SOLIDWORKS PDM lets you map block attributes for the following properties:

- **SWIPrjProperty** defines mapping for Inspection Project Properties.
- **SWICustomProperty** defines mapping for Inspection Custom Properties.

The attribute names available for **SWIPrjProperty** for Project Properties are:

- **Part Name**
- **Part Number**
- **Part Revision**
- **Document Name**
- **Document Number**
- **Document Revision**

- File reference handling

SOLIDWORKS PDM creates references between the inspection project files (.ixprj) and output deliverables (.xlsx, .pdf). In SOLIDWORKS PDM, the Contains tab displays inspection project files as parent files and output deliverables as child references.

In the SOLIDWORKS Inspection add-in, references are created between a drawing file and output deliverables files or between the inspection project file and output files.

- SOLIDWORKS PDM ribbon

SOLIDWORKS Inspection standalone includes the SOLIDWORKS PDM ribbon. You can access SOLIDWORKS PDM features such as Check-Out, Check-In, Search, Get Version, and Get Latest Version using the SOLIDWORKS PDM tab in the CommandManager. This integration helps you manage and centralize the storage of inspection projects plus the files and reports related to them.

Support for Configurations of Weldment Cut Lists

SOLIDWORKS PDM extracts and displays weldment cut list properties for all configurations. Previously, you could view the cut list properties for the active configuration only.

You can view configurations of weldment cut list properties in:

- Data cards of cut list items
- Weldment BOMs
- Weldment cut lists

This functionality is supported for parts that you create or save in SOLIDWORKS 2019 or later. You must also add **Rebuild on Save Mark** for all configurations.

Support for DXF/DWG File Format in Convert Task

The Convert task in SOLIDWORKS PDM Professional converts sheet metal flat patterns to the DXF/DWG file format.

SOLIDWORKS 2018 or later supports the conversion.

Advanced Conversion Options (DWG Format/DXF Format)

You can use the Advanced Conversion Options dialog box to set options when exporting sheet metal patterns to the DXF or the DWG file format.

To display the Advanced Conversion Options dialog box:

- When configuring a conversion task, on the Conversions Settings page of the Convert - Properties dialog box, for **Output file format**, select **DWG Format** or **DXF Format** and click **Conversion Options**.
- When initiating a conversion task, in the Convert dialog box, for **Output file format**, select **DWG Format** or **DXF Format** and click **Conversion Options**.

The following options are applied when converting DXF/DWG format files:

Geometry	Exports flat pattern geometry.
Hidden edges	Includes hidden edges (only available if you select Geometry).
Library features	Exports library features (only available if you select Geometry).

Forming tools	Includes forming tools.
Bend lines	Exports bend lines.
Sketches	Includes sketches.
Bounding box	Exports bounding boxes.

Multibody export options

Single file	Converts all flat patterns into a single DXF/DWG format file.
Separate files	Converts each flat pattern into a separate DXF/DWG format file. <div style="border: 1px solid gray; padding: 5px; margin-top: 5px;">The flat pattern name is a suffix on the primary output path.</div>
Enable Map File	Enables the map file and lets you select the location of the map file under Vault Root Folder Path . The file must be accessible on the task host during the task execution.
Reset All	Resets the options to defaults.

Support for eDrawings Functionality

SOLIDWORKS PDM supports the following eDrawings® functionality:

- Ambient occlusion
- Selection of configuration and sheet

Ambient Occlusion

SOLIDWORKS PDM supports ambient occlusion for the 3D file formats supported in eDrawings.

To display the eDrawings toolbar on the Preview tab, on the SOLIDWORKS PDM menu bar, click **Display**, and select **Options > Show full UI in SOLIDWORKS preview**. Click  to see ambient occlusion applied to files.

To enable ambient occlusion in files, in eDrawings, click **Tools > Options**, and select **Fastest speed**.

Selection of Configuration and Sheet

SOLIDWORKS PDM lets you select a configuration and sheet on the Preview tab.

In the Preview tab, on the eDrawings toolbar, you can select:

- Configurations from the **Configurations** list for the .EPRT, .EASM, .SLDPRT, and .SLDASM files.
- Sheets from the list for the .EDRW, .SLDDRW, .DXF, and .DWG files.

Upload and Check in File Structure and Download with References

The SOLIDWORKS PDM Web2 client provides a faster and easier way to upload and download files and folders with their references.

You can:

- Upload a folder and its contents including files and folders to a specific location.
- Download the files and folders with their references.

Previously, to upload or download a file with its references, you had to perform the operation for each file separately.

The supported browsers for **Check In File Structure** functionality are Google® Chrome, Mozilla® Firefox, and Microsoft® Edge.

User Comments

SOLIDWORKS PDM lets you control mandatory comments with more granularity during the check-in or change state operations.

With the improved control, the administrator mandates that users add the check-in or state change comments for specific states and transitions in the workflow. These mandatory comments create a reliable document history.

The enhanced permissions are: **Must enter version comments** for each state in workflow and **Must enter state change comments** for each transition.

Scenario	Must enter version comments	Must enter state change comments
Administrative permission is selected and state or transition permission is selected or cleared.	Enter comments during: <ul style="list-style-type: none">• Initial check-in.• Check in through Copy Tree and Branch operations.• Merge by using the option Merge as new file for at least one file in the reference tree.• Set Revision action when Update Variable is selected.	Enter comments when you change the state of a file regardless of the transition.
Administrative permission is cleared and state or transition permission is selected.	Enter comments when you check in a file that is in a specific state.	Enter comments when you change the state of a file via a specific transition.

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