What's New in SolidWorks Enterprise PDM 2009

SolidWorks

Contents

Notices

1 What's New for Users

SolidWorks® Enterprise PDM replaces the previous product name, PDMWorks® Enterprise.

This chapter includes the following topics:

Bills of Materials

Enterprise PDM bills of materials are more tightly integrated with SolidWorks BOMs than in earlier releases. Named BOMs have additional editing and update features.

You can:

- Display SolidWorks BOMs in assemblies and drawings
- Display cut lists and weldment BOMs for weldment parts
- Display BOMs that reflect SolidWorks BOM exclusions

In named BOMs, you can:

- Update the BOM to reflect changes in Solidworks sources
- Drag and drop columns and rows
- Hide and unhide columns
- Add position numbers

You must convert SolidWorks files to Solidworks 2009 to use the BOM features. An administrator can convert all SolidWorks files in the vault to SolidWorks 2009. See File Conversion and BOM Features on page 13.

SolidWorks BOMs

Enterprise PDM displays SolidWorks assembly and drawing BOM tables. You can edit a SolidWorks BOM, check it in, and see the changes to the BOM in Enterprise PDM.

On the Bill of Materials tab, you select either a computed BOM or a SolidWorks BOM. The SolidWorks BOM has the name used in the SolidWorks FeatureManager design tree. You can save the SolidWorks BOM as a named BOM.

Cut Lists and Weldment BOMs

You can display a cut list or a weldment BOM for a weldment part.

This feature requires configuration by an administrator. See Cut List and Weldment BOM Templates on page 16.

You must manually update the part in SolidWorks 2009 to display a weldment cut list. Right-click the cut list in the FeatureManager design tree and select **Update.**

To display a weldment cut list or weldment BOM:

1. In the File Explorer, select a weldment part.
2. On the **Bill of Materials** tab, select the type of BOM:
   - **Weldment Cut List**
   - **Weldment BOM**

SolidWorks BOM Exclusions

Computed BOMs reflect your SolidWorks exclusions automatically. In earlier releases, computed BOMs showed all assembly components, regardless of the exclusion.

Enterprise PDM respects these SolidWorks BOM exclusions:

- Components excluded from a configuration using **Exclude from bill of materials** in the component properties.
- Sub-assembly components excluded using **Don't show child components in BOM when used as a sub-assembly**.

BOM Part Number

The Part Number column on the Bill of Materials tab lists part numbers from the CAD file. Computed BOMs are now more similar to SolidWorks BOMs in content and appearance. For Pro/ENGINEER and other CAD programs that use table instances, Enterprise PDM uses the table instance name in this column.

Enterprise PDM uses the SolidWorks part configuration property **Part number displayed when used in a bill of materials** for the Part Number column. You can set this property to the document name, a configuration name, or a user-specified name.

Sort BOM by Column

You can sort the contents of any type of BOM by any column.

To sort BOMs by column, double-click a column header on the **Bill of Materials** tab in the File Explorer for an assembly or drawing. The sort preserves the assembly structure.

Updates for Named BOMs

When you create a named BOM in Enterprise PDM 2009, the BOM is associated with a source. You can update the named BOM when a new version of the source file is checked in.

You can create a named BOM from a computed BOM or a BOM in a SolidWorks 2009 drawing or assembly.

To create and update a named BOM based on a SolidWorks BOM:

1. In the File Explorer, select a SolidWorks drawing or assembly that contains a BOM.

   The drawing or assembly must be a SolidWorks 2009 file. You can open, rebuild, and save a file in SolidWorks 2009 to convert it.
2. Click **Bill of Materials**.
3. Under **BOM**, select a SolidWorks BOM.

The default name is **Bill of Materials1**. The BOM list shows the name given in SolidWorks FeatureManager.

4. Click **Save BOM** and select **Save As**.
5. Click **Save**.
   The named BOM has the default file extension SWBOM and a number. Named BOMs based on the computed BOM have the default extension BOM.
6. Check out the associated drawing or assembly and open it in SolidWorks.
   a) Edit the BOM in the SolidWorks file. For example, add a new description to the BOM table.
   b) Rebuild and save the file.
   c) Check in the file.
7. Under **BOM**, select the named BOM you created.
8. Click **Update Source Version** and select the new version of the SolidWorks file.
   The updated BOM information is in green.
9. Click **Save** to save the updated information.

Enhanced Tab for Named BOMs

You can add rows, columns, and position numbers to named BOMs on the Bill of Materials tab. You can also check in and check out named BOMs using new icons.

You can do these editing tasks on named BOMs:

- Drag and drop columns and rows
- Insert columns and rows
- Copy and paste rows
- Insert child (indented) rows
- Hide and unhide columns and rows
- Assign detailed position numbers
- Filter rows by column values

You can insert, hide, and unhide rows and columns by right-clicking a row or a column header.

There are Bill of Materials tab icons for these functions:

Filter

Position Numbers

Check Out

Check In

You can check in and check out named BOMs in the File Explorer view using the icons. In earlier releases, check in and check out were only available from a Bill of Materials view.

When you click **Compare BOMs** to compare different versions, the Bill of Materials tab shows added, modified, and deleted rows in different colors. A legend at the bottom of the tab identifies the colors for each type of change.

SolidWorks Add-In

The Solidworks Enterprise PDM Add-In has additional toolbar icons, file information icons, and thumbnail previews. You can also compare versions of vault documents in SolidWorks.

Enterprise PDM Task Pane Toolbar

The tool bar at the top of the Enterprise PDM task pane contains new icons.

	State Change	Changes the workflow state of selected components.
	Where Used	Opens the **SolidWorks Enterprise PDM File Viewer** with information for the selected component.
	Search	Opens the **SolidWorks Enterprise PDM Search** window.

File Tree Information Icons

Information icons now appear next to file names, for example:

myAssembly

myPart

	The file does not exist in the vault.
	The local file is newer than the file in the vault.
	The file is edited in SolidWorks but not saved.
	The local file is older than the file in the vault.
	The local file is the same version as the file in the vault.
	The file has been modified by another user.

Check Out Prompt

When you open a vault file in SolidWorks that is not checked out, the Enterprise PDM Add-In prompts you to check out the file.

Vault Properties for Lightweight Assemblies

The Enterprise PDM task pane shows full vault information for all lightweight components, including the version number, check-out status, and workflow state. Earlier releases of Enterprise PDM displayed the component name for lightweight components.

Add-In Options Properties

The View Setting tab in the Enterprise PDM Options dialog box lets you:

- Add display items to the tree in the task pane or the preview pane. For example, you can add configuration names.
- Enable or disable display of part instances.
- Set options for information icons in the file tree.

Compare Documents

The Compare Documents command supports comparing files inside and outside the vault.

You can compare:

- A version of a file with another version.
- A revision of a file with another revision.
- Two vault files.
- A vault file with a file outside the vault.

To compare versions of a vault file:

1. In the CommandManager, click **Office Products** > **SolidWorks Office** > **SolidWorks Utilities**.
2. Click **Compare Documents**.
3. Under **Document 1**, select a vault document.
4. Select **Compare Document within Enterprise Vault for** and **Versions**.
5. Under **Version 1** and **Version 2**, select the versions to compare.
6. Click **Compare.**
   The selected versions open in separate windows.
7. Use the tools in the **Compare Documents** pane to complete the comparison.

   See the SolidWorks online help for more information about comparing documents.

Workflow and Productivity

The enhancements in this section are changes to workflow or user interface improvements.

Workflow Notifications

You can select recipients and include comments in workflow-based notifications.

An administrator must set up dynamic notifications in your workflow before you can use this feature. See Dynamic Notifications on page 18.

Notifications for State Changes

A state change operation that affects multiple files produces a single notification message. The message lists all files that changed state. In earlier releases, changing the state of an assembly created a separate notification message for each part in the assembly and another message for the assembly itself.

Workflow and Named BOMs

When you change the state of a named BOM, Enterprise PDM can:

- Send automatic notifications.
- Export the BOM to an XML file.

  You can then import the XML file into an ERP system.

Both features require an administrator to configure a workflow transition. Export to XML also requires an export rule. See Export Rules for Named BOMs or SolidWorks BOMs on page 17 and Notifications for Named BOMs on page 18.

Drawings Included in Check-Ins

When you check in an assembly or part, the **Check In** dialog box automatically includes related drawings.

The related drawing can be in the current folder or in a folder elsewhere in the vault. By default, Enterprise PDM checks the entire vault for related drawings you have checked out. The scope of the search depends on administrative settings. See Drawing Check-Ins on page 14.

References to Parts Update Automatically

If you copy, move, or rename a part or assembly, Enterprise PDM records the new relative location of the files when you check them in.

While Enterprise PDM supported copying, renaming, and moving parts and assemblies in earlier releases, it did not record their relative locations. As a result, dialog boxes did not show related files at check in or check out after the copy or other operation.

Setting the Configuration Tab View

The Preview tab and File Data Card now show the tab for the active configuration of a part or assembly by default. In earlier releases, the @ configuration was the default configuration tab.

Since Enterprise PDM shows SolidWorks custom file properties on the @ configuration tab, you might want to change the default view.

To change the default tab:

1. In the File Explorer, right-click in a blank area of the right pane and select **View** > **Set focus to active configuration.**

   This toggles your view of the configuration tab.

   To show the @ configuration by default, ensure that **Set focus to active configuration** is cleared.

2. Select a part or assembly and click the data card.
   The @ configuration tab appears.

SolidWorks Simulation File Support

Enterprise PDM automatically includes simulation results files when you check in a SolidWorks file with a reference to a simulation study.

Microsoft Office 2007 Files

The File Explorer view lists properties for Microsoft® Office 2007 files.

AutoCAD and Inventor Add-ins

The AutoCAD Add-In and Inventor Add-In for Enterprise PDM support AutoCad® 2009 and Autodesk Inventor® 2009.

Validations

The Enterprise PDM performs additional validations for check-ins and other operations. The validations require set-up using the Enterprise PDM Administration tool.

Warnings Can Block Operations

You may not be able to check in files if there are warnings for one or more files. An administrator can specify that a warning block a check-in or other operation.

For configuration information, see Operation Blocking for Warnings on page 15.

Warnings about Obsolete Drawings

Enterprise PDM checks that you have rebuilt drawings or assemblies to reflect changes to parts. The validation is performed when you check in, increment the revision number, or change the state of a part or assembly.

The **Check In**, **Change State**, and **Increment Revision** dialog boxes now list the warning **The file is not regenerated** for assemblies or drawings that have not been rebuilt. If an administrator has enabled operation blocking for files that are not rebuilt, you cannot perform the operation.

References to File Serial Numbers

Files are not supported as a source for serial numbers in this release. As a result, you may see an error when you attempt to generate a serial number on a data card. If you see this error, ask your administrator to update the serial number generator to a supported type.

For configuration information, see Serial Number Generation from Files on page 15.

Prompts for Passwords

You may see a prompt for an electronic signature when you change the state of a file. When you see the prompt, enter your Enterprise PDM password.

For configuration information, see Electronic Signatures on page 18.

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What's New for Administrators

This chapter describes administrative features and set-up procedures in the Enterprise PDM Administration tool.

This chapter includes the following topics:

- Web-based License Activation
- Platform Support
- File Conversion and BOM Features
- Online Help
- Users and Groups
- Data Cards and Templates
- Workflow

Web-based License Activation

You activate Enterprise PDM by pointing to a license file on your system. You can obtain the license file over the Web.

Browsing for a License File

1. Double-click **License**.
2. Click  and locate the license file.

Obtaining a License File over the Web

1. Double-click **License**.
2. Under **License file**, click **Request license file**.
3. On the SolidWorks Customer Portal welcome page, enter the email address and password for your customer account.
4. Under **My Support**, click **My Products**.
5. Click **Get License**.
   The license file is sent to your customer account email address.
6. Open the email and save the license file on your system.

Platform Support

Support for Enterprise PDM servers extends to Microsoft® Windows Server 2008.

File Conversion and BOM Features

You must convert SolidWorks files to Solidworks 2009 before users can take advantage of many BOM features in this release.

Install and run the **SolidWorks Enterprise PDM File Version Upgrade Tool** to convert SolidWorks files in the vault to SolidWorks 2009. See the conversion utility online help for more information about running the utility.

Online Help

Administrative dialog boxes now have help buttons and associated help topics.

Users and Groups

This section describes changes to managing users and groups.

Full Names in History Records

You can display full user names in the **History** dialog box.

To show full user names in history records:

1. Right-click **Users** and select **Settings**.
2. On the **Explorer** tab, under **Miscellaneous**, click **Show full user names in the History dialog box**.
3. Click **OK.**

Group Folder Permissions

The new permission **Assign group membership** provides flexible control for group folder access.

Users with this permission see the Group Membership tab in the properties dialog for vault folders. You can grant the permission to an administrator or a user who does not have other administrative privileges.

Using the Group Membership tab, a user or administrator can set access rights to the selected vault folder.

Granting Assign Group Membership

To grant Assign Group Membership:

1. Double-click **Users** and then double-click the name of a user.
2. Click **Permissions per folder** and select a folder.
3. Under **Permissions**, click **Assign group membership**.
4. Click **OK**.
   The user can see the **Group Memberships** tab in the folder properties dialog box.

Assigning Folder Access

This procedure uses the **Group Membership** tab to grant a user access to one of the folders assigned to a group.

In this example, a user group called Designers has access to the folders Project1 and Project2. You want to make the new user JSmith a member of the Designers group with access to the folder Project2 but not the folder Project1.

To assign a group folder:

1. Log in as a user with the **Assign group membership** permission.
2. In the File Explorer, right-click the folder **Project2** and select **Properties.**
3. On the **Group Memberships** tab, under **Designers**, select the user name **JSmith** and click **OK**.

   This adds JSmith to the Designers group, but only grants access to the current folder, Project2.
4. In the Administration tool, double-click **Users.**
5. Double-click **JSmith**.
   Under **Members of** on the **General** tab, the row for **Designers** shows the Project2 directory.

When JSmith logs in to the vault, the File Explorer shows Project2 as a green folder that is accessible in Enterprise PDM. Project1 is gray and contains no files.

The **Group Memberships** tab for the folders and the user information in the Enterprise PDM Administration tool reflect the folder assignment.

Drawing Check-Ins

You can now configure whether related drawings checked out with a part or assembly are listed in the **Check In** dialog box.

In earlier versions, related drawings were available for check-out but not listed for check-in.

To configure finding drawings at check-in:

1. Right-click **Users** and select **Settings**.
2. Click **Check In**.
3. Select settings:
   - **Include drawings automatically when checking in the model** - Select to enable finding drawings at check-in or clear to disable finding related drawings.
   - **Look for drawings in the entire vault** - Select for the broadest search scope, or clear to define a limited search.
   - **Look for drawings in subfolders of the folder containing the model** - You can combine this with finding in parent folders.
   - **Look for drawings in parent folders of the folder containing the model** - You can combine this with finding in subfolders.
   - **Only look for drawings with the same file name as the part/assembly** - You can combine this with other selections.

Explorer Column Views for Groups

You can create a column view and assign the view to a group. The columns you add supplement the default columns in the File Explorer.

To create and assign a column view:

1. Right-click **Columns** and select **New Column Set**.

2. Under **Column set name**, enter a name for the new column view.
3. Click **New Column**.
   The **Selected column** area is now active.
4. Select a **Variable**, **Name**, and other column properties.
   The selected column appears in the **Columns** list.
5. Click **New Column** again to specify column properties for an additional column.
6. Under **Groups**, select a group name.
7. Click **OK**.

   You can also assign the column view from the user's or group's **Properties** dialog box.

Operation Blocking for Warnings

Administrators can block checking in, checking out, and other operations for selected warning conditions.

For example, you can block a check-in if Enterprise PDM raises the warning "The file is not rebuilt." You specify operation blocking using the **Warnings** tab on the user's or group's **Properties** dialog box.

Data Cards and Templates

This section contains new or changed administrative tasks and features in the Card Editor and the Template Manager.

Serial Number Generation from Files

The File type is no longer supported for serial number generation. You must modify any serial numbers that use the File type, or users will see an error when they attempt to generate a serial number that has a file source.

To modify serial number settings:

1. Double-click **Serial Numbers**.
2. Double-click a serial number.
3. For File types, click **Yes** at prompt to convert the File type into a List type.
4. Check all the remaining serial numbers and convert any other File serial number generators.

Serial Numbers for Configurations

Enterprise PDM can generate a different serial number for each configuration of a part or assembly. This feature is available for files added to the vault for the first time in Enterprise PDM 2009.

The data card property **Updates all configurations** controls whether a single value or multiple values are generated.

In earlier releases, the Enterprise PDM generated a single serial number for all configurations.

To configure separate serial numbers for configurations:

1. Double-click **Cards**.

2. In the **Data Card Editor**, select **File** > **Open** and select a data card with a serial number field.

   The field should have **Serial Number** selected under **Default Value**.
3. Under **Flags**, clear **Updates all configurations** to generate a different serial number when a user right-clicks the field.
4. Save and close the card file.

Cut List and Weldment BOM Templates

An administrator must assign access rights to the templates so that users can see cut lists and weldment BOMs.

To configure Weldment BOM and Weldment Cut List templates:

1. Double-click **Bills of Material**.
2. Double-click **Weldment Cut List**.
3. Under **Columns**, review the column list and add or edit columns.

   You can rename the initial columns but the variables are required to display the cut list.
4. Assign rights to the template:
   a) Under **Users and Groups**, select the users or groups that need to use the template.
   b) Under **Rights**, select **Activate computed BOM** and **See Computed BOM**.
5. Click **OK**.
6. Repeat Steps 2 through 5 for the **Weldment BOM** template.

Computed BOM Template

The default BOM template for computed BOMs has a Part Number column. The new column makes the content of a computed BOM closer to the content of a SolidWorks BOM.

Card Editor Variable for Search Tabs

You can use the **Controlled by Variable** property to turn search tabs on or off.

In this release, you can conditionally show search tab layouts created in the **Card Editor** based on the value of the selected variable. In previous releases, the **Controlled by variable** property was only available on data cards.

Default Configuration Tabs in Templates

In file creation templates, you can specify the default configuration tab for file data cards created by the template.

When you edit a template in the **Template Manager**, you can set the default configuration tab in the **Edit Template File** dialog box. Under **Default card page**, you can select from:

- **Standard Selection** - The active configuration tab appears on file data cards created from the template.
- **@** - The @ tab appears on data cards created from the template.

Workflow

This section describes changes and features in workflows.

XML Export for BOMs

You can configure XML export for named BOMs and SolidWorks BOMs using export rules and workflow transition actions. The export rules for computed BOMs can create XML in either a new table-based format or in tree format as in earlier releases.

To enable XML export of named BOMs and SolidWorks BOMs, first set up an export rule, and then create a transition action that uses the export rule.

Export Rules for Named BOMs or SolidWorks BOMs

To create an export rule for named BOMs or SolidWorks BOMs:

1. Double-click **Data Import/Export Rules**.
2. Right-click **Export rules** and select **New Export Rule**.
3. In the **Export Rule** dialog box:
   a) Type a **Rule Name**.
   b) Under **Output XML-files to folder**, enter a valid path for the export file folder on the server.
   c) Under **Output XML-file name**, enter a formula for the file name.

      Build the formula by clicking and using the variables and counters in the list as building blocks. Each item you select appears under **Output XML-file name**.
   d) Under **Type of data to export**, select either **Named BOM** or **CAD BOM**.

      CAD BOM exports SolidWorks assembly or drawing BOMs.
4. Click **OK.**

Transition Actions for Named BOMs or SolidWorks BOMs

To create a transition action for exporting named BOMs or SolidWorks BOMs:

1. Double-click **Workflows** and double-click a workflow.
2. Click **Properties** on a transition.
3. Click **New** to add a new action.
4. In the **Transition Action** dialog box:
   a) Enter a **Description** for the action.
   b) Select either **Run on named bills of materials** or **Run for files** for SolidWorks BOMs.
   c) Clear **Run on files** to limit the action to named BOMs.
   d) Under **To**, select a user or group to receive the message.
   e) Under **Select export script to use**, select an export rule for exporting the BOM type.
   f) Click **OK.**
5. Click **OK** to save the transition properties.
6. Save and close the workflow.

Export Rules for Computed BOMs

Export rules for computed BOMs support exporting XML in a table format as well as tree format.

By default, computed BOMs are exported in the tree format used in earlier releases. To use the new format, select **Export bill of materials in table format** in an export rule.

Some export options are not supported for the new format.

Notifications for Named BOMs

You can set up automatic notifications for named BOMs by adding an action to a transition in a workflow.

To set up automatic notifications for named BOMs

1. Double-click **Workflows** and double-click a workflow.
2. Click **Properties** on a transition.
3. Click **New** to add a new action.
4. In the **Transition Action** dialog box:
   a) Enter a **Description** for the action.
   b) Click **Run for named bills of materials**.
   c) Under **To**, select a user or group to receive the message.
   d) Under **Message**, enter the message text.
   e) Click **OK.**
5. Click **OK** to save the transition properties.
6. Save and close the workflow.

Dynamic Notifications

You can set up dynamic notifications that let users specify recipients and enter comments for notifications triggered by workflow transitions.

To set up dynamic notifications for a project:

1. Double-click **Workflows** and open a workflow.
2. Click **Notifications** on a workflow transition.
3. On the **Notifications by Project** tab:
   a) Select a project folder.
   b) Select the **Users** or **Groups** to receive the notifications.
   c) Click **Add**.
   d) Under **Notifications**, select **Dynamic**.
   e) Click **OK**.

   The steps are similar for **Notifications by Assignment**.
4. Repeat steps 2 and 3 for each transition you want to have dynamic notifications.
5. Save and close the workflow.
   When a user performs a state change using a transition you configured, the **Do Transition** dialog box has fields for message recipients and comments.

Electronic Signatures

You can configure workflow transitions to prompt for an electronic signature when users change the state of a file.

If electronic signatures are enabled, the user must enter a Windows or vault password to initiate the state change. You set the **Authentication** property on each workflow transition where a signature is required.

To require an electronic signature for a state change:

1. Click **Workflows** and double-click a workflow.
2. Click **Properties** on a transition in the workflow diagram.
3. Click **Authentication.**
4. Click **OK.**
5. Save and close the workflow.