What's New in SolidWorks Enterprise PDM

Version 2010

SolidWorks

Contents

Legal Notices

About this Guide

This guide describes new and changed SolidWorks® Enterprise PDM capabilities in the 2010 release.

Intended Audience

This guide is for experienced users of the SolidWorks Enterprise PDM software.

1 Managing Tasks

The new **Tasks** feature in the Administration tool makes it easy to perform tasks such as converting and printing SolidWorks documents.

The task framework lets administrators and users execute tasks on demand, based on a schedule, or triggered by a SolidWorks Enterprise PDM workflow. Administrators can configure tasks to run on a specific computer or distribute them to both client computers and dedicated servers. A SolidWorks **Print** or **Convert** task can be initiated on any computer that is running Enterprise PDM, as long as the computer where the task is executed has SolidWorks installed.

Administrators can define tasks to be interactive, letting users initiate the task and choose task options. Alternatively, they can set all options so that the task runs silently, without requiring user input.

Using the task API and templates provided in the Enterprise PDM software, programmers can also extend the functionality to support custom tasks.

This chapter includes the following topics:

Setting Up Tasks

You use the Administration tool to install and configure Enterprise PDM tasks.

To enable tasks:

- Include the **Convert** and **Print** task options when you create a new vault and specify the vault's configuration details.

  See *SolidWorks Enterprise PDM Administration Tool Help: Creating a Vault*.

- Enable the execution of tasks on host computers.

  See *SolidWorks Enterprise PDM Administration Tool Help: Permitting Task Execution*.

- Configure the **Convert** and **Print** tasks.

  These tasks, which are supported by the SolidWorks Task Add-in, are listed under **Tasks** in the Administration tool.

- Create new tasks using the Task wizard.

  If you use the Enterprise PDM API to program a task add-in, you add tasks supported by the add-in to the **Tasks** feature by creating new tasks.

- Include task execution in workflow transitions.

For example, you can create a transition action that converts parts and drawings to `.pdf` files when an assembly is ready for review.

See *SolidWorks Enterprise PDM Administration Tool Help: Launching a Task Using a Workflow*.

- Initiate and monitor tasks using the Task List dialog box.

  Use **Add Task** in the Task List dialog box to immediately launch a task.

  Use the Task List dialog box to monitor tasks and to view information about completed tasks.

Convert and Print Tasks

The **Tasks** feature in the Administration tool lets you configure task properties. The **Convert** and **Print** tasks are available if you select them when you create the vault.

You can copy an existing task and modify it so that you have multiple task configurations. For example, you can create a silent configuration that does not require user input and a second user-initiated configuration that lets users modify task settings.

Convert tasks lets you convert SolidWorks files using the output formats and options supported by SolidWorks. For example, you can:

- Specify how converted files are named, for example, using the source file name and revision number.
- Save converted files to the vault or another destination such as a directory used by an Enterprise Resource Planning (ERP) application.
- Map variables from the source file data card to the output file data card to transfer attributes such as Description and Part Number.
- Convert a multisheet drawing to a multipage `.pdf` file.
- Convert only specifically named drawing sheets such as sheet metal flat patterns.

Print tasks are also performed on SolidWorks files. For example, you can:

- Print drawings to a print server, so that user computers are not tied up.
- Direct output to printers based on paper size requirements.
- Set permissions so that users can print files to a printer near them.
- Specify the following for source files with references:
  - Which references to print - none, drawings, parts, or assemblies
  - Which versions to print - the latest version or the version referenced the last time the source file was built

You can initiate either a **Print** or **Convert** task by adding it to a workflow transition. For example, you can:

- Print all drawings in an assembly when the assembly state changes to Reviewed.
- Convert all drawings in an assembly to `.pdf` format when the state changes to Released.

For instructions on configuring tasks, see *SolidWorks Enterprise PDM Administration Tool Help* topics:

- *Configuring the Convert Task*
- *Configuring the Print Task*

Executing Tasks

The task infrastructure provides multiple ways to initiate tasks.

As an administrator, you can:

- Trigger tasks using a workflow transition

  For example, you can automatically generate `.pdf` files when SolidWorks files reach a state such as Released. As part of task setup, you can specify options for the newly created files such as file names, properties, and destination folders.

- Use the Task List dialog box to execute an on-demand task

  On-demand task execution lets you initiate the printing of SolidWorks files from an Enterprise PDM client that does not have the SolidWorks software installed.

- Configure tasks so that they can be initiated by users

  With File Explorer, users with permissions to perform tasks can select files and use right-click menu options to convert or print them.

- Schedule task execution

  If you create custom tasks using the APIs, you can include the task scheduling infrastructure so that tasks can be performed when task hosts are available.

Monitoring Tasks

The primary tool for monitoring tasks is the Task List dialog box. You can also use notifications to inform users of the success or failure of tasks.

The Task List dialog box, which you display from the **Tasks** feature in the Administration tool, lets you identify tasks that are pending, monitor tasks that are running, and view information about completed tasks. You can also initiate tasks from this dialog box.

- The **Pending tasks** list shows the task queue, including tasks that are currently running and tasks that are scheduled or waiting to run. You can suspend and resume pending tasks, or cancel them completely.

  If you select a pending task or a task that is executing and click **Details**, you can see the task progress, who initiated the task, and the files on which the task is being executed.

- The **Completed tasks** list shows the status of completed tasks. If a task fails, you can view its details to see an error message and error code. These codes are also reported in error logs that are created for failed tasks.

  Task List Options let you specify the number of task records to retain in the **Completed Tasks** list.

You can set up notification messages to be sent to users and task initiators when a task succeeds or fails.

- If a conversion task succeeds, the notification contains links to the converted files, so that you open a file in the Enterprise PDM File viewer, view the file's properties, and see the file's history.
- If a task fails, the notification contains a link to the error log.

Task Add-ins

Tasks are defined by task executor add-ins. Enterprise PDM includes the task executor add-in SWTaskAddIn, which enables conversion and printing of SolidWorks files.

Programmers can use the Enterprise PDM API to create an add-in to execute tasks unique to their environment. After administrators install the add-in to the vault, they add the tasks it defines to the **Tasks** node by creating new tasks.

To create a task add-in, you need to program in Visual Basic® .Net, C#, or C++. See the description of the API in the *SolidWorks Enterprise PDM Programmer's Reference Guide*.

A task add-in extends the capabilities of the task executor. For example, a task add-in could open files in Microsoft Word and save them to another format such as `.html`.

You can program the task add-in to:

- Send parameters and data to the task executor as part of the task instruction
- Execute any MS-DOS®/Windows command
- Start up any Windows program with an arbitrary number of parameters
- Give commands to any Windows program
- Access and send command to installed resources like printers and plotters

2 Enterprise PDM and Toolbox Integration

Enterprise PDM can now manage SolidWorks Toolbox part files and the Toolbox database entirely in an Enterprise PDM vault. Users point SolidWorks to the Toolbox vault folder and use Toolbox as before. Enterprise PDM automatically:

- Checks Toolbox parts out and in as needed so the latest versions are used
- Adds missing Toolbox parts to the vault
- Redirects assembly references to Toolbox parts in the vault
- Replicates the Toolbox files and database (if replication servers are configured)

Enterprise PDM still supports the SolidWorks pre-2010 Toolbox integration, where the master Toolbox library is outside the vault and Enterprise PDM creates copies in the vault as parts are used. If you have SolidWorks 2010, the new integration is recommended.

This chapter includes the following topics:

Configuring Enterprise PDM and Toolbox

An administrator must configure Enterprise PDM and Toolbox before users access Toolbox.

For details about this procedure, see *SolidWorks Enterprise PDM Administration Tool Help*: *Configuring Toolbox*.

To configure the 2010 Enterprise PDM and Toolbox integration:

1. Check the Toolbox root folder (`SolidWorks Data` by default) into the Enterprise PDM vault.
2. From the Enterprise PDM Administration tool, double-click **Toolbox** and set configuration options.
3. From SolidWorks, click **Tools** > **Options**  > **Hole Wizard/Toolbox**:
   a) Set **Hole Wizard and Toolbox folder** to match the setting from the Enterprise PDM Administration tool.
   b) Click **Configure** to configure Toolbox using the Toolbox configuration tool.
4. Have all users set their SolidWorks **Hole Wizard and Toolbox folder** to the Toolbox vault folder.

Using Toolbox with Enterprise PDM

Enterprise PDM's management of Toolbox is mostly transparent to users, but there are some details to be aware of.

Local Cache

When you first insert a Toolbox component into an assembly, Enterprise PDM caches the latest version of the part file in your local vault view. As you continue to use Toolbox, Enterprise PDM manages your cache automatically.

To reduce disk space, you can clear the Enterprise PDM cache, which removes local files. Enterprise PDM caches files again when needed, which affects performance.

When you clear the cache from the vault root folder (from File Explorer, right-click the vault and select **Clear Local Cache**), the Clear Local Cache dialog box appears. You can remove cached Toolbox files by clearing **Do not remove Toolbox files**.

Automatic Part Creation

Toolbox detects when assemblies contain configurations that do not exist in the library and can automatically create missing sizes for you.

If Toolbox is configured to create part files for each new size, Enterprise PDM automatically creates a new file in the vault without first checking the local cache. If Toolbox is configured to add configurations to the master part file, Enterprise PDM checks the cache and retrieves the latest file if necessary. If the size does not already exist, Enterprise PDM creates a new version of the master part file containing the new configuration.

Search Paths

SolidWorks always searches for Toolbox parts in the Enterprise PDM vault first. You do not need to add the vault folder to your System Options - File Locations folder list.

Permissions

Depending on how your administrator has configured Toolbox integration, Enterprise PDM operations are performed using your Enterprise PDM permissions or the permissions of a designated user. These permissions determine what operations you can perform. Enterprise PDM displays a message if you do not have permission to perform an operation.

For example, if you have read access to Toolbox but no permission to check out files, you can use existing component sizes but not create new sizes.

Working Offline

You can work with Toolbox parts when you do not have access to the vault by caching the parts locally before going offline. In File Explorer, use **Get Latest** on the Toolbox vault folder. You cannot create new sizes when working offline.

3 File Explorer and SolidWorks Add-in

This chapter includes the following topics:

Naming Files with Serial Numbers Using Copy Tree

The **Copy Tree** command can now automatically name copied files using serial numbers. By default, the serial numbers set up for naming parts, assemblies, and drawings in the SolidWorks add-in options are used, but you can select any serial number.

Right-click a file and click **Copy Tree**. In the Copy Tree dialog box, do one of the following:

- Click **Transform** > **Rename with Serial Number**.
- Right-click a file in the file list and click **Rename with Serial Number**.

See *SolidWorks Enterprise PDM File Explorer Help*: *Copying Files with References*.

Naming Drawings with Model Names Using Copy Tree

The **Copy Tree** command can copy drawings using the same names as their associated assembly or part files.

Right-click a file and click **Copy Tree**. In the Copy Tree dialog box, select **Name drawings after their models**.

See *SolidWorks Enterprise PDM File Explorer Help*: *Copying Files with References*.

Drawings with Multiple References Not Automatically Checked Out

Enterprise PDM no longer automatically checks out or retrieves drawings referencing more than one file when you check out or retrieve a referenced part or assembly. This change addresses issues where drawings remained checked out after the referenced file was checked in.

File Preview Rebuild Warning

The Preview tab now displays a warning if a drawing or assembly needs to be rebuilt. This happens when referenced parts or subassemblies have been modified but the selected drawing or assembly has not been opened and resaved in SolidWorks.

Warning when File Is Open in Another Application

File Explorer now warns you if you check in, check out, or edit the data card of a file that is open in another application. Close the file in the other application and retry the Enterprise PDM operation.

By default, this condition blocks check-ins and check-outs. To make this condition nonblocking, from the Enterprise PDM Administration tool, expand **Users** or **Groups** and double-click a user or group. In the Properties dialog box, click **Warnings**. For **Affected operation**, select **Check In** or **Check Out**, clear **The file is open in another application**, and click **OK**.

If the file is open in SolidWorks and the Enterprise PDM add-in is installed, you can perform operations from the add-in without closing the file.

Changing Column Names and Order in the SolidWorks Add-in

You can now rename and reorder the columns in the Enterprise PDM SolidWorks client.

To rename a column, in SolidWorks, click **Enterprise PDM** > **Options**. In the Enterprise PDM Options dialog box, on the View Setting tab, under **Display Information**, type a new **Caption**.

To reorder columns in the SolidWorks Enterprise PDM task pane, drag a column header (caption) to a new location. **File / Variable** must be the first column.

Cold Storage Versions in Get Version Submenu

The **Get Version** command now lists cold storage versions in a submenu, which shortens the main list to only versions you can retrieve.

From File Explorer, right-click a file and click **Get Versions** > **Versions in Cold Storage**.

From the SolidWorks add-in, click **Get Versions** > **Versions in Cold Storage**.

Versions in Cold Storage is available only if at least one version is in cold storage.

4
Item Management

This chapter includes the following topics:

Generating Item IDs from File Attributes

Enterprise PDM can now generate item IDs from file attributes. An administrator selects which data card variable to map to, such as part number. When generating an item from a file, Enterprise PDM defaults to an existing item if there is an item with the same ID, otherwise the item ID defaults to the mapped value. A user can choose to generate a different ID. If variable mapping is not enabled or if the variable has no value, Enterprise PDM uses the item serial number to generate the ID.

In the Administration tool, expand a vault and double-click **Items**. On the **Item ID** page, for **Item ID Variable**, select a variable or select **<Do not read the ID from a variable>**.

See *SolidWorks Enterprise PDM Administration Tool Help*: *Configuring Items*.

Named BOMs for Items

Named bills of materials (BOMs) are now supported for items. You can create one or more BOM views and sort, filter, change position numbers, and compare BOMs.

In Item Explorer, on the Bill of Materials tab, click **Save BOM**  > **Save as BOM**.

See *SolidWorks Enterprise PDM Item Explorer Help*: *Named BOMs*.

Display Formats for Item BOMs

You can now control which items display in item BOMs.

In Item Explorer, on the Bill of Materials tab, click **BOM Display**  and click one of the following:

- **Indented** displays all items in the BOM hierarchically.
- **Parts Only** displays only items without child items.
- **Top Level Only** displays only top-level items.

Expanding and Collapsing Item Structure

You can now expand all items displayed in the Item Explorer main pane to display their content. You can expand one, two, three, or all levels of hierarchy, and collapse all levels of hierarchy.

In Item Explorer, click **Expand Levels** (toolbar) and click a command.

See *SolidWorks Enterprise PDM Item Explorer Help*: *Expanding and Collapsing Items and Folders*.

Optional Parent Node Check Boxes

When generating items from files with references, you can now choose whether to display bulk selection check boxes for parent nodes. Use these check boxes to make selections more efficiently, or turn them off to simplify the user interface. Previously, these bulk selection check boxes were always displayed.

In the Generate Item dialog box or Link Files to Item dialog box, right-click in the file list box and select or clear **Show Parent Node Check Boxes**.

See *SolidWorks Enterprise PDM Item Explorer Help*: *Bulk Selecting References in Dialog Boxes*.

API Support for Items

Enterprise PDM now provides an API for items.

See API Support for Items on page 19.

5
Administration Tool

This chapter includes the following topics:

Add-in and File Format Support

The SolidWorks add-in and file formats for Enterprise PDM 2010 have been updated to support SolidWorks 2010.

Extended Export and Import Functionality

The Administration tool export functionality has been expanded for Enterprise PDM 2010 to make it easier for you to move settings from one vault to another.

You can now export all configurable features to administrative export (`.cex`) files.

You can also export all settings in a vault to a single `.cex` file by right-clicking the vault name and clicking **Export**.

To import administrative export files, right-click the vault name, click **Import**, and navigate to the location of the `.cex` file.

Data Card Export and Import

You can export a data card created using the card editor to a `.cex` file and import the `.cex` file into another vault.

When you import a data card, all card dependencies, such as variables, serial numbers, and card lists, are also imported.

You can export data cards for:

- Files
- Folders
- Items
- Searches
- Templates

You can export all cards in the vault, all cards of a specific type, or individual cards.

Add-in Export and Import

You can export an API add-in to a `.cex` file and import the `.cex` file into another vault. You can export a single add-in or all the currently registered add-ins.

When you import an API add-in, all files included with the add-in are also imported.

To export all registered API add-ins in a vault, right-click **Add-in** and click **Export**.

To export an individual add-in, expand **Add-in**, right-click the add-in, and click **Export**.

Template Export and Import

You can export a template created using the template wizard to a `.cex` file and import the `.cex` file into another vault.

When you import a template, all template dependencies such as input forms, variables, and serial numbers are also imported.

Group settings are included in export files for templates. User settings are not.

To export all templates in a vault, right-click **Templates** and click **Export**.

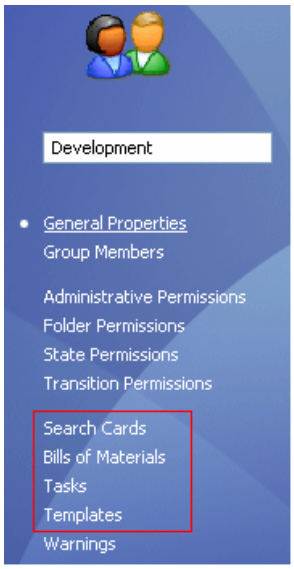
To export an individual template, expand **Templates**, right-click the template, and click **Export**.

Central Assignment of Permissions

The Properties dialog boxes for users and groups now provide a central place to assign all permissions, including permissions for search cards, bills of materials, and templates.

Access to the dialog boxes has not changed. For example, you still expand **Groups** and double-click a group to display its Properties dialog box. To make it easier to locate and display the permissions to set, the tabs used in previous releases have been replaced by a left pane list of controls that link to the permission pages.

The following graphic shows the new Group Properties dialog box controls, highlighting the **Search Cards**, **Bills of Materials**, **Tasks**, and **Template** links.



Serial Number Enhancements

You have more control over how Enterprise PDM handles serial numbers for added or renamed files. Previously, if you set up serial numbering, default values were generated for all configurations. You can now configure data cards to specify configurations for which default values will not be generated.

In the Card Editor, when adding most controls, select **Default overwrites** and type the names of the configurations from which to exclude default values.

Enterprise PDM also:

- Generates default values when you create new SolidWorks configurations
- Preserves card values when you rename a configuration

SMTP Mail Setup

You can now quickly verify your SMTP mail settings by performing an email test to validate them.

If there are errors in your settings, Enterprise PDM returns a detailed error message to help you find the problem.

With SMTP selected in the Message System dialog box, on the SMTP tab, click **Test Settings**. In the Test Settings dialog box, type the name of an email recipient and click **OK** to send the test message.

6
API

For information on using the Enterprise PDM API, see the *SolidWorks Enterprise PDM Programmer's Reference Guide*. To access the guide, right-click **Add-ins** and click **Programmer's Reference Guide**.

This chapter includes the following topics:

API Support for Items

Enterprise PDM now provides an API for items.

The API lets you create applications to perform custom tasks such as:

- Synchronizing items between Enterprise PDM and Enterprise Resource Planning (ERP), Materials Requirements Planning (MRP), and other external systems
- Importing items into Enterprise PDM
- Accessing information to generate reports
- Customizing item numbering

Specific API capabilities include:

- Reading, modifying, and creating items
- Generating items from files
- Reading, modifying, and creating item links to child items and files
- Reading and modifying item bills of materials (BOMs)
- Finding items and opening them in Item Explorer

API Methods for Users and Groups

The existing API methods for users and groups have been extended.

You can now use the API to:

- Copy permissions and settings from an existing user
- Assign folder permissions for users and groups
- Add users to and remove users from existing groups
- Delete existing users and groups
- Check folder permissions for users and groups

Dispatch Set Card Variables Command

The **Dispatch** module includes the new command `Set card variables`. This command sets file and folder data card variables to user-specified values. Administrators can select and configure the `Set card variables` command and add it to an action.

To access the `Set card variables` command:

1. Expand **Add-ins**, right-click **Dispatch**, and click **Administrate Actions**.
2. In the Administrate Actions dialog box, click **Add**.
3. In the Edit Action dialog box, click **Add**.
4. In the Select command dialog box, select **Set card variables**.

Use the Set Card Variables dialog box to:

- Configure the command to specify the target file or folder, data card variable, configurations, and variable value
- Specify the variable value as a static string or as a parameterized string constructed using Dispatch variables

7 Installation

This chapter includes the following topics:

Predefined Data Set Configurations

Enterprise PDM provides multiple predefined configurations that consist of data cards, workflows, templates, and bill of materials (BOM) column sets. To make it easier to begin using Enterprise PDM, when you create a vault, you can choose which configuration to import.

Configurations are defined using `.cex` files.

To select a configuration, right-click the server name in the Administration tool and click **Create new vault**. On the Configure vault screen, select one of the standard configurations shipped with Enterprise PDM:

Empty	Creates a data folder with no files, which makes it easier to import or create objects such as data cards and workflows.
Default	Installs the data set from previous versions of Enterprise PDM.
SolidWorks Quick Start	Installs a simple data set with predefined groups and permissions, enabling companies installing Enterprise PDM for the first time to quickly use the software in a production environment.

If your company already has a customized data set based on an administrative export file, you can browse to it and import it.

8 Documentation

This chapter includes the following topics:

Web-based Documentation

Documentation for SolidWorks Enterprise PDM is now available on the Web.

By default, when you access help, the Web version of the documentation is displayed in a Web-based viewer. You can still choose to use local help files (`.chm`) if, for example, your Internet connection is slow or unavailable.

Benefits of Web-based help include:

- Improved search functionality, including improved relevancy ranking, spelling correction, short descriptions in search results views, and guided navigation to help you identify relevant topics.
- Improved topic navigation, including next and previous topic buttons and breadcrumb navigation.
- Ability to provide feedback directly to the documentation team on individual help topics.
- Up-to-date documentation without the need to download large compiled help (`.chm`) files.

Select or clear **Use SolidWorks Web Help** on all SolidWorks Enterprise PDM Help menus to switch between the local and Web-based versions of the help.

The *SolidWorks Enterprise PDM Programmer's Reference Guide* is available as a locally installed `.chm` file only. To access it, in the Administration tool, right-click **Add-ins** and click **Programmer's Reference Guide**.