



Infor PLM for Discrete SolidWorks User Guide

Release 2022.x

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Contacting Infor

If you have questions about Infor products, go to Infor Concierge at <https://concierge.infor.com/> and create a support incident.

The latest documentation is available from docs.infor.com or from the Infor Support Portal. To access documentation on the Infor Support Portal, select **Search > Browse Documentation**. We recommend that you check this portal periodically for updated documentation.

If you have comments about Infor documentation, contact documentation@infor.com.

Chapter 2: Introduction

PLM interfaces directly with SolidWorks, facilitating direct transfer of product designs from the SolidWorks environment into PLM and then ERP. This eliminates the inherent risks and errors associated with data entry from multiple sources. SolidWorks files can be managed according to configuration management methodology, while facilitating the direct transfer of complete design Bills of Material (BOMs) to the production environment. Main features of the integration

Viewing design files is no longer a privilege afforded only to engineering departments. All authorized users are provided with the tools to search, browse, view and manipulate documents from a secured central database, enhancing collaborative teamwork.

Using the embedded PLM menu and toolbar, the PLM integration for SolidWorks provides access to PLM functionality from within the native working environment.

This integration connects engineering workgroups to the entire enterprise in the native SolidWorks environment. SolidWorks and PLM exchange product information, and update files and documents throughout the design process.

Main features of the integration

The PLM integration for SolidWorks includes the following features:

- Files can be saved from SolidWorks to PLM.
- Document link management.
- Unique file names for all new SolidWorks files.
- SolidWorks files can be revision controlled.
- Concurrent engineering features enables you to manage ownership of the files.
- Files can be retrieved from the PLM using the Download Manager.
- Bill of Material (BOM) is transferred to PLM when you save files to PLM.
- Support for design variants enables you to assign an item to the configuration. As a result, you can transfer the BOM of a design variant(configuration) to PLM.
- Advanced query mechanism for searching objects in PLM.
- Role-based authorization allows users to perform operation based on their roles.
- Files can be automatically saved to PLM in neutral formats such as PDF, BMP and so on. Neutral files can be viewed using applications that are not CAD specific
- Built-in file viewer.

- Thumbnails of CAD files can be generated and stored in PLM database for snapshot view of the design files.
- The design process can be controlled using the workflow functionality.
- Preferences to control the behavior of the integration. Example: integration preference, Set Object to Match File Name enables you to save the files using a specific file naming convention.
- Mapping functionality enables you to transfer values, such as, document properties, custom properties, and so on, from CAD file to PLM and vice versa.
- Graphic representation of product trees.

Chapter 3: Getting Started

The following topics are available to help you getting started with the PLM integration for SolidWorks:

- [Installing the Integration](#) on page 13
- Requirements
- [PLM embedded menu and toolbar](#) on page 14

Note: It is essential that the administrator sets up the integration correctly before users start to work with it.

Software Configuration

For the latest PLM Server, Infor PLM and SolidWorks Integration Kits, refer to the latest release notes.

To install the integration kit

To install the integration kit, extract the Integration kit zip file to the local system. It is suggested to install the integration kit from the local setup files for all the components.

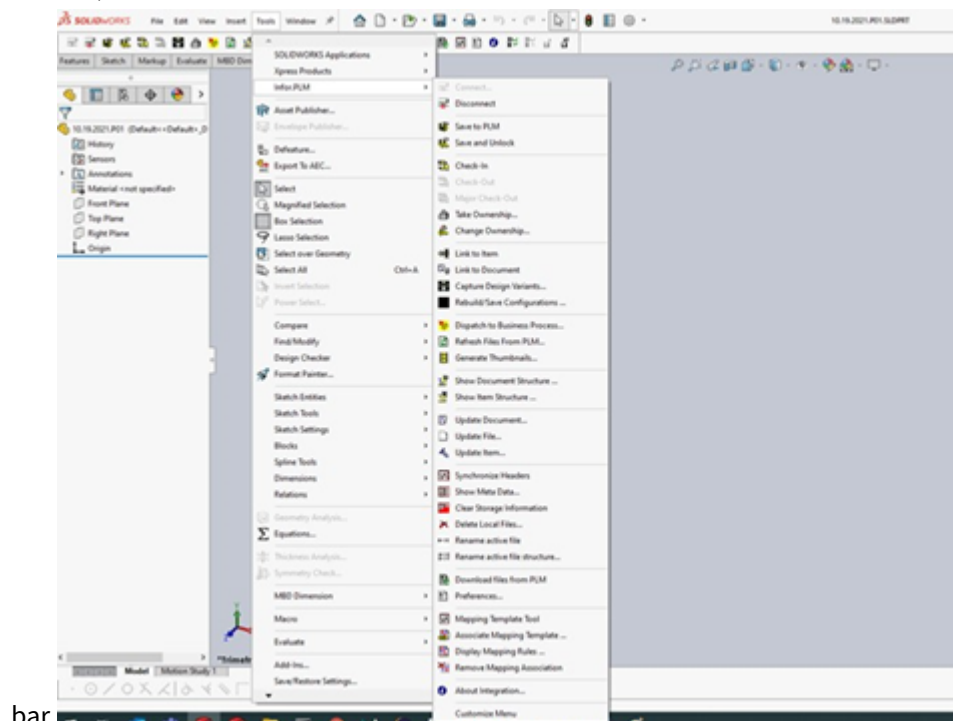
Installing the Integration

To install the integration:












- 1 Before installing the PLM integration for SolidWorks, the SolidWorks must be installed.
- 2 To install the SolidWorks integration, double click the .msi file, and follow the instructions of the install shield.




















PLM embedded menu and toolbar


After the PLM integration for SolidWorks has been installed, the PLM toolbar is added to your SolidWorks toolbar, and a menu is added to the SolidWorks menu



bar.

	Connect	Connecting to PLM on page 17
	Disconnect	Disconnecting from PLM integration on page 17
	Capture Design Variants	Managing design variants/ configurations on page 44
	Link to Item	Link to Item on page 24
	Save to PLM	To save a file to PLM on page 19
	Save and Unlock	Saving and Unlocking Files on page 27
	Check In	Checking in a File on page 28
	Check Out	Checking out a File on page 30
	Check Out Major	Checking out a File on page 30
	Take Ownership	Taking Ownership on page 32
	Change Ownership	Changing the ownership of a file on page 32

	Dispatch to Business Process	Dispatch to business process on page 49
	Refresh Files from PLM	Refresh Files from PLM on page 43
	Show Document Structure	Opening a file in PLM on page 36
	Show Item Structure	Opening a file in PLM on page 36
		Using Infocards on page 37
	Update Document	Using Infocards on page 37
	Update File	Using Infocards on page 37
	Synchronize Headers	Synchronize Headers on page 39
	Show Meta Data	Viewing PLM Data on page 34
	Clear Storage Information	Clear Storage Information on page 41
	Delete Local files	Delete Local Files on page 42
	Display Mapping Rules	For information on mapping, see: <ul style="list-style-type: none"> • Display Mapping on page 65 • General Mapping on page 58
	Associate Template Mapping	The Associate Mapping Rules menu option on page 63
	Remove Mapping Association	Removing mapping associations on page 64
	Preferences	Integration Preferences on page 97
	About	Contains product and system information as well as additional sources for professional assistance.
	To Toggle the Integration Preference (Create Document + Item)	Creates both the document and item when you click Save to PLM .
	To Toggle the Integration Preference (Create Document Only)	Creates only document when you click Save to PLM .
	To Toggle the Integration Preference [Save Only Modified File] to False	Disables the Save Only Modified File integration preference

	To Toggle the Integration Preference [Save Only Modified File] to True	Enables the Save Only Modified File integration preference
-----------------------------------------------------------------------------------	------------------------------------------------------------------------	-------------------------------------------------------------------

Chapter 4: Working with PLM

This section contains the following topics that describes the tasks to be executed while working with PLM:

- [Connecting to PLM](#) on page 17
- [Opening a file in PLM](#) on page 36
- [Disconnecting from PLM integration](#) on page 17

Connecting to PLM

To use PLM Integration for SolidWorks, you need to establish connection between SolidWorks and PLM. The connection gives you access to the PLM database and projects that you need to work with.

To connect to Infor PLM Discrete:

From the SolidWorks application, do one of the following:

- Click the **Connect** icon in the PLM integration toolbar.
- Select **Connect** in the PLM drop-down menu.

The functions in the PLM menu and toolbar are enabled.

Disconnecting from PLM integration

Disconnecting is a global operation for all SolidWorks integration applications; disconnecting from one SolidWorks application disconnects all connected SolidWorks application from PLM.

To disconnect from the PLM integration, do one of the following:

- Select **Disconnect** from the PLM integration drop-down menu.
- Click **Disconnect** in the PLM integration toolbar.

Chapter 5: Saving Files to PLM

When any SolidWorks file, such as a drawing, is saved to PLM, it is represented by two PLM objects that are linked to each other:

- A PLM file, which is revision-controlled, and which specifies the location of the SolidWorks file, together with additional data about it. This PLM file (also known as a data set) has the same name as the SolidWorks file.
- A PLM document, which is also revision-controlled, and which contains attributes that describe the characteristics of the file. The file's name is held in the document's Description attribute.

If the active SolidWorks file is a model (component or assembly), an item can be created and linked to the document. This depends on your preference settings. If the file has children (components), the child objects are also saved in PLM. Files, documents and items are created for each component.

Document links are created representing the assembly-component relationships to the newly saved files as well as to the components that were previously saved in PLM.

When you perform the Save to PLM operation on a SolidWorks file, the document is saved with the status **Draft**. Each consecutive save updates the latest changes performed on the **Draft** revision of the document in PLM.

You can only save a file if its related PLM document has a status of **Draft**. If the document has the status **UNDER CHANGE** or **RELEASED**, the file cannot be saved to PLM.

To save your changes on the server (not only locally), first save the file locally, and then perform the PLM action, **Save to PLM**.

Saving an existing file to PLM also saves the file in SolidWorks. If you save the file only to SolidWorks, the file is saved only locally and the associated documents and/or items in PLM are not saved.

The assignment of attributes during the save is determined by the parameters set up for you by the administrator in your preference settings. If the **Set Object Attribute During Save** check box is selected in the preferences, you can define attributes for the files, documents, and items that you save to PLM.

If you are working with design variants, known as configurations in SolidWorks, you must capture the data of the design variants before you save the item to PLM. For more information, refer to [Managing design variants/configurations](#) on page 44.

To save a file to PLM

Before you can save a file to PLM, you must save it in SolidWorks with a unique name.

Note:

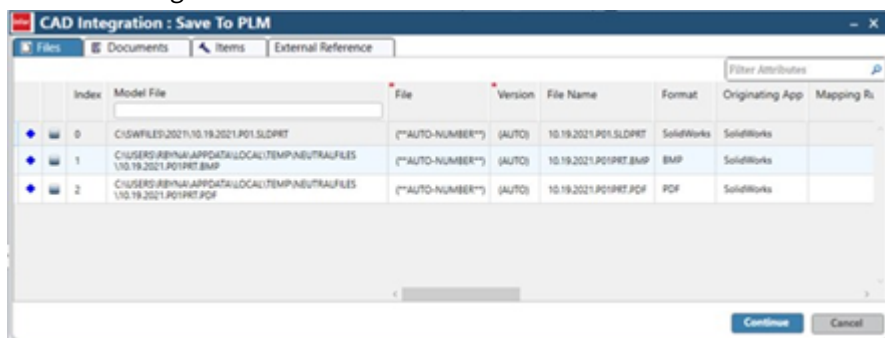
- Do not use & < > ' " symbols in the file names that you save to PLM, this will cause errors to occur.
- If you are working with design variants and you want to save the design variants as well, complete the Capture Design Variant process before you save a file to PLM.

To save a file to the PLM, do one of the following:

- Click **Save to PLM** in the PLM toolbar.
- Select **Save to PLM** from the PLM menu.

Clicking **Save to PLM** in the menu or the toolbar results in one of the following:

- If all the IDs are generated automatically by PLM, which you can specify in the preferences, the file is saved.
- If the **Set Object Attribute During Save** check box is selected in the integration preferences, the Set Object Attributes dialog box



appears.

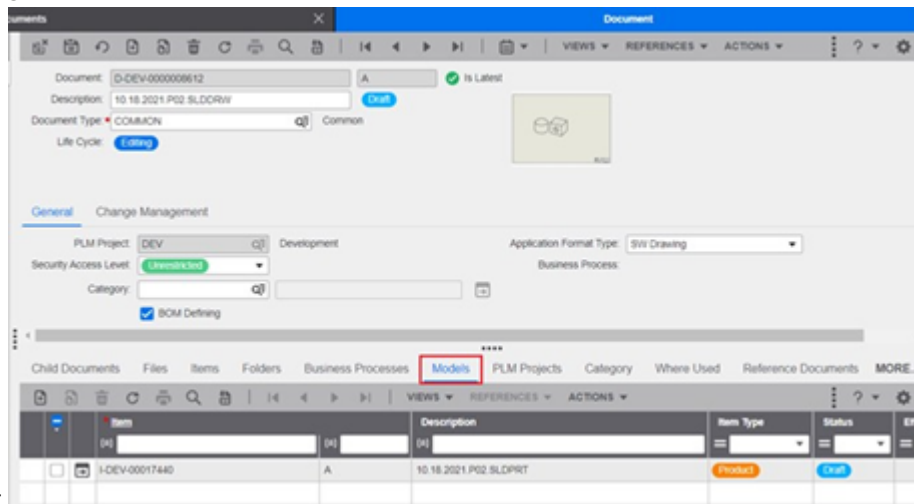
In this dialog box, you can switch between the **File**, **Document**, and **Item** tabs and for each tab you can update any attribute. After you finish updating attributes, click **OK** in this dialogbox.

For attributes that you do not update, the PLM default values are automatically inserted. For further information, see [Initial Save Option](#) on page 99.

While saving files to PLM, the integration checks whether you own the files being saved. Editable files that are not owned by you or by any other users are displayed in the Taking Ownership on dialog box. In this dialog box, select the files that you want to save to PLM. See [Taking Ownership](#) on page 32

The Integration saves the files, documents, and items to the PLM database and creates hierarchical links between the documents, files, and items for the assembly documents and the bill of material.

When you create a drawing for a part and click **Save to PLM**, the integration creates files and documents for the drawing, and in the hierarchy, links the item of inserted part as a model to the drawing



document.

Saving a file and creating a document

To save a file to PLM and create only a document.

It is assumed you have the correct preference settings selected to create a document only. For more information, see [Integration Preferences](#) on page 97 .

- 1 Select **Save to PLM** from the PLM drop-down menu or click **Save to PLM** icon on the toolbar. As a result, the Attributes for File-Documents-Items dialog box appears. The **Item** tab displays no data and the item fields are not accessible since your preferences are set up to create a document only.
- 2 After updating the relevant attributes, click **OK**. As a result, only the documents and files are created in PLM, and not the items.

Chapter 6: Download Manager

The download manager retrieves the latest files from the PLM database and saves them to the local work directory of the user. The download manager is the basis of the following PLM preferences:

- [File Name Uniqueness](#) on page 21
- [Download Additional Drawings](#) on page 21
- [Indicator for Locally Changed Files in Download Manager dialog](#) on page 22
- [Skip Mapping for Released Files](#) on page 22
- [Skip Meta Data Comparison During Download](#) on page 23

File Name Uniqueness

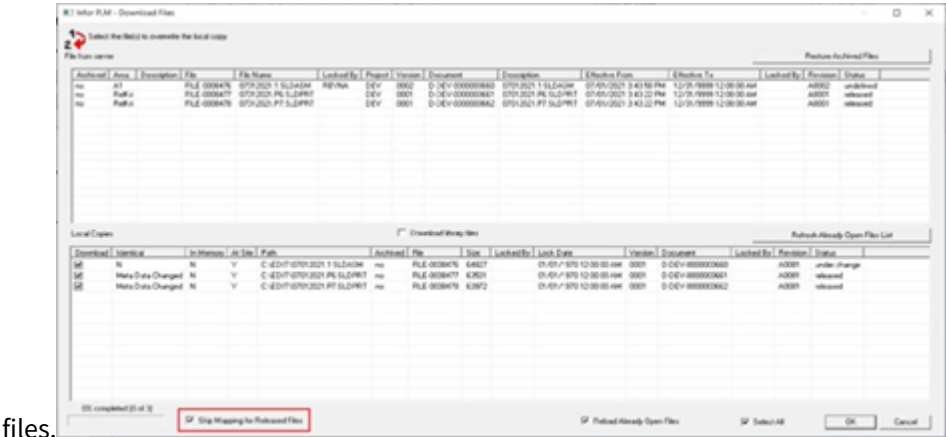
If this check box is selected, in the **General Option** of PLM preferences, the names of the files stored in PLM are maintained as unique.

If this check box is cleared, you can store multiple files with identical file names in PLM but in different projects. It is not recommended to clear this check box, since it might cause problems if you want to download a file while a file with an identical name already exists locally. For example, you cannot use multiple files which have identical file names; from different folders; in one assembly.

Download Additional Drawings

If this check box is selected, in the **General Option** of PLM preferences, the **Edit/View file in Integration** option enables the user to download the related drawings of all the models in the current Document structure.

Additionally, it is also possible to select the **Skip Mapping for Released Files** during the download process. Select **Skip Mapping for Released Files** check box to skip the mapping for the released



The default value for this check box is defaulted based on the integration preference **Skip Mapping for Released Files**.

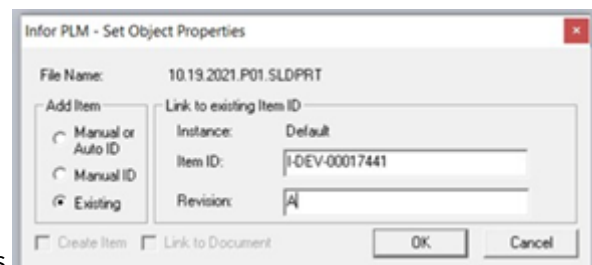
Skip Meta Data Comparison During Download

When **Skip Meta Data Comparison During Download** check box is not selected in the General Option of PLM preferences, the download manager indicates the changes if any, in the PLM database.

The Download Manager indicates if there is any change in the PLM data of Document/ Item/File. In case the Download Manager indicates the data change, it is recommended to download the indicated files. Hence the Download option for the specific changed file is selected by default.

Chapter 7: Link to Item

If the preferences specify that no items must be generated for part files on initial save to PLM, you must use the **Link to Item** option from the PLM menu to create items for the part file that you are working on, or to link the part file to an existing item in the PLM database.



If you click this option, the **Link to Item** dialog box appears.

In this dialog box, you must enter item **ID** data as required or specify the item to which the part file must be linked.

If you select the **Manual** or **Auto ID** radio button and leave the Item ID: field blank, an item and an item ID is generated in PLM. If you enter an ID in the Item ID:, an item is generated with the ID that you entered in the Item ID: field. The **Revision** field is optional.

If you select the **Manual ID** radio button, you must enter an item ID in the Item ID: field. As a result, PLM generates an item with the ID that you entered in the Item ID: field.

If you select the **Existing** radio button, you must specify an item ID in the Item ID field. This ID and revision must match the Item that is existing in PLM Database.

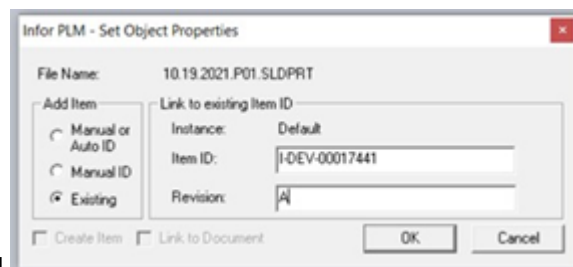
If you entered incorrect item ID data in the Link to Item dialog box, before you save to PLM, you can click this menu option again to correct the item data.

After you enter the ID data as required, you must save the part file to PLM.

Note that the properties set in the PLM Integration Properties for Integration dialog box determine which item data you can specify in the Link to Item dialog box. For further information, see [Initial Save Option](#) on page 99.

Link Item to More than one Document

If the preferences specify that no items must be generated for part files on initial save to PLM, you must use the Link to Item option from the PLM menu to create items for the part file that you are working on, or to link the part file to an existing item in the PLM database.

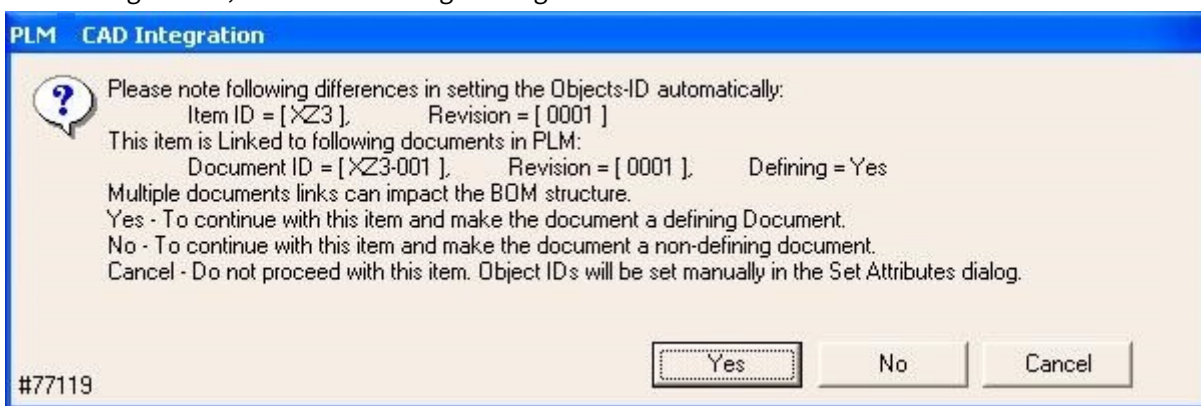


If you click this option, the Link to Item dialog box is displayed.

During the Save to PLM and when you use the PLM menu option Link to Item, you can specify if the document must update the BOM structure or not.

When you click Save To PLM:

- If a user saves a file with preference Set object to match filename selected, and the filename is matching an existing item id, then the following message is



- If a user saves a file with preference Set object to match filename selected and links the existing item in the Set Attributes screen and the item is already linked to another document, the following message is



displayed:

Chapter 8: Saving and Unlocking Files

If a file that you are working on is supposed to be used in a business process, the business process cannot be launched while the file is locked. To avoid this situation, you should save such files using the **Save and Unlock** command. This automatically updates your data, while relinquishing ownership of the file. The business process can then proceed.

If you have finished working on a file and the file must be passed on to an unknown new user, you should also use the Save and Unlock command. The new user who needs to work on the file can now acquire ownership of the file. If you know who will be the new owner of the file, you can transfer ownership to the new owner. See [Changing the ownership of a file](#) on page 32.

To save and unlock a file:

- Select **Save and Unlock** from the PLM menu.

The file is saved in PLM with no owner. For further information on saving files to PLM, see [Saving Files to PLM](#) on page 18.

Chapter 9: Checking in a File

When you have finished working with a file, it can be vaulted. The PLM integration for SolidWorks enables you to check in any SolidWorks file and its linked documents to the PLM vault.

The check-in operation accomplishes the following:

- Confirms the changes you made in the SolidWorks file.
- Changes the file's status from **Draft** to **RELEASED**.
- Transfers the SolidWorks file to the PLM Released area from the users work area.

The integration verifies that you have authorization to perform this operation and that the document linked to the file has the **Draft** status. If the file is new, the system prompts you with the Save As dialog box to save the file locally and perform the check-in operation.

After the file has been checked in, the file can be modified only after check out. To check in a file:

1 Do one of the following:

- Click check in icon in the PLM toolbar.
- Select **Check In** from the PLM menu.

2 If prompted to save locally, save the file to SolidWorks first. The file is checked in.

You can specify how to handle files after they have been checked in. See Integration Preferences.

Note: If you check in a parent assembly, components that were checked out individually, and that belong to sub-assemblies of the parent assembly, are not checked in.

For example, the situation is as follows:

- Parent assembly is **RELEASED**.
- Sub-assembly is **RELEASED**.
- Child of sub-assembly is **RELEASED**.

The user does the following:

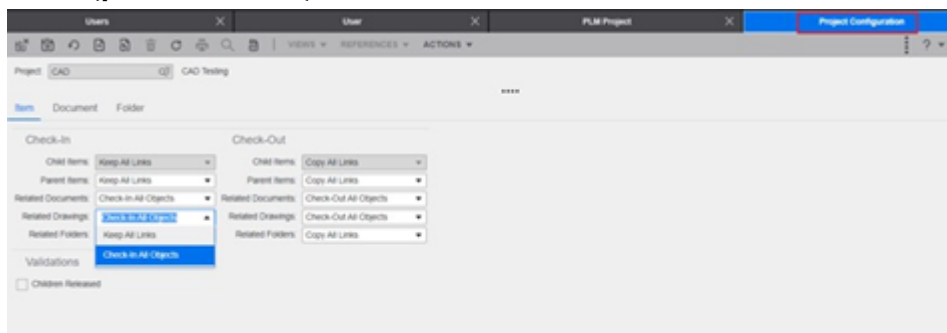
- Checks out the parent assembly.
- Checks out the child of the sub-assembly and make changes.
- Checks in the parent assembly.

As a result, only the parent assembly is checked in. The child of the subassembly is not checked in because the sub-assembly was never checked out and was therefore released at the time the parent assembly was checked in.

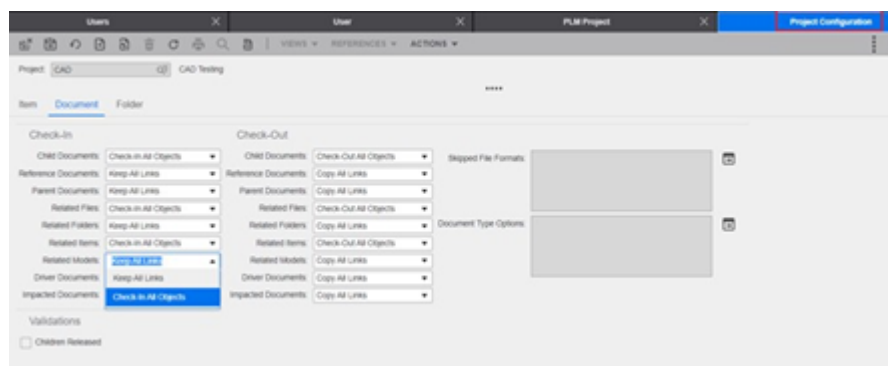
Check-in related drawing files

When user performs Check-In operation on a Drawing, the integration will Check-in its related model. Also, if Check-in operation is performed on a model, the integration will Check-in its related drawings.

To enable this functionality, PLM Administrator needs to set the Projects' Vault Parameter in **Project Configuration (pdadm3101s000)**



session.



Chapter 10: Checking out a File

You must check a file out of the vault in order to change it. The Check Out option is available after the file has been retrieved from PLM tool.

To check out the file, you need to open it for editing in the integration. The file is opened in read-only mode and cannot be modified until it is checked out. If you try to perform any modification in it, you will be prompted to check it out first. This is done to protect the integrity of your data.

Types of check out:

Check Out Minor

This option is usually used for minor design revision changes. A minor check-out results in a higher sequence number within the current revision for the checked out file, for example from A0001 to A0002.

Check Out Major

This option is usually used for significant changes with major impact on the form, fit or functionality of the product. A major check-out results in a higher version number for the checked out file, for example from A0001 to B0001.

To check out a file:

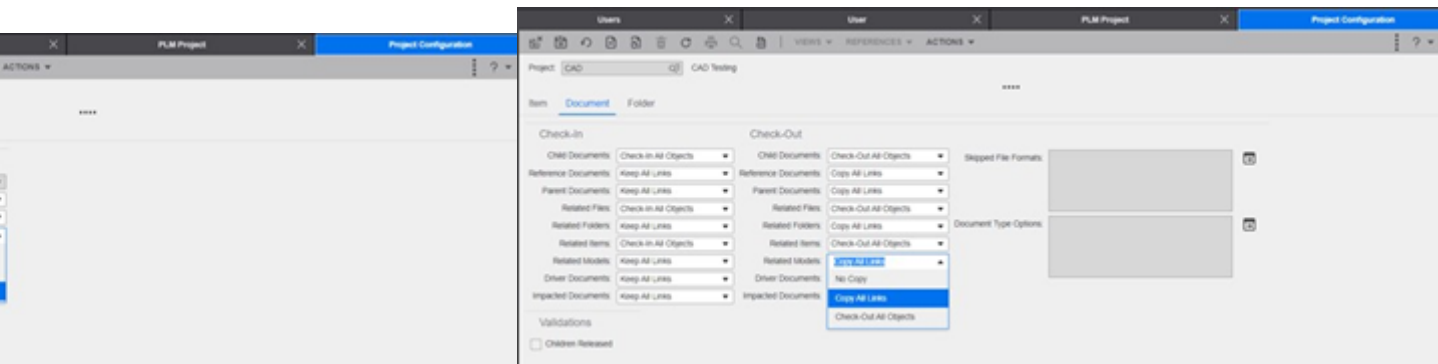
- Select the assembly or component part that you want to check out and select **Check Out** from:
 - The PLM Toolbar
 - The PLM menu
 - The right-click menu available on the objects in the design tree.

As a result, the assembly or component part is checked out. After the check-out, the file is no longer in read-only mode and can be modified as required.

Check-out related drawing files

When user performs check-out operation on a drawing file, the integration will check-out its related model. Also, if check-out operation is performed on a model, the integration will check-out its related drawings.

To enable this functionality, PLM Administrator needs to set the Projects' Vault Parameter in **Project Configuration (pdadm3101s000)**



Note:

- Limitations
 - Drawing must contain views of only one Model. But, if the drawing contains views of multiple models, then the check-out operation from one Model will not impact the other Models.
 - The integration user can check-out only the latest revision of the document. If a user wants to check-out previous revisions of any document, that can be done from the PLM Client. In this case the user must perform the Edit File after performing the check-out from the PLM Client.

Chapter 11: Taking Ownership

While working on a large assembly, you may need to update a particular part. To prevent other users from making changes to this part simultaneously, you must take ownership of the file.

To take ownership of a file within an assembly:

- 1 In the SolidWorks Feature Manager, select the part of which you want to take ownership.
- 2 Do one of the following:
 - Click save and unlock icon in the PLM toolbar.
 - In the PLM menu, select **Take Ownership**.

As a result, you are the owner and the file is locked for other users.

Note: When **Take Ownership during Edit File** preference check box is not selected, the users do not take the ownership of the files even when they execute **Edit File** command.

Changing the ownership of a file

The owner of a file is determined by one of the following:

- The user who created it and saved it to PLM.
- The user who checked it out of PLM.
- The user to whom the ownership has been transferred.
- The user who performs the Edit File process on a saved and unlocked file.
- Project administrators are not the owners of all files, but they are given access to modify the files owned by users in their projects

When you are registered as the owner of a file in PLM, you can edit the file as required, while other users can view but not modify the file. You can choose to transfer the ownership to another user when the user needs to work on that file.

Note: Ownership can only be changed if the file has been saved to PLM.

After a file has been checked in (and has **RELEASED** status), it does not have a specific owner. You cannot change the ownership of a file with **RELEASED** status.

To change the ownership of a file:

- 1 In SolidWorks, for the part file with which you are working, select **Change Ownership** in PLM menu or toolbar.

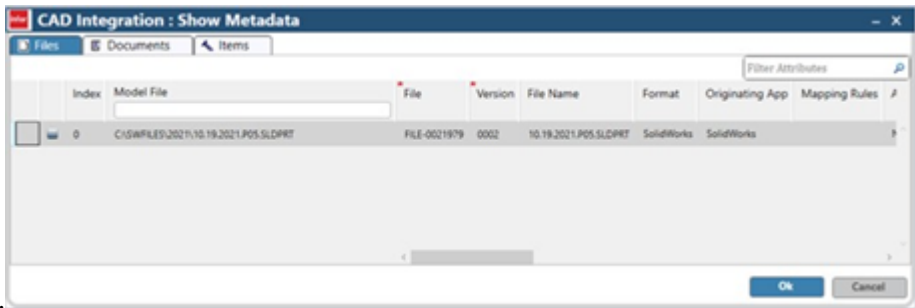
- 2 In the Select User dialog box that appears, select the user you want to transfer the ownership to and click **OK**. The selected user now owns the file.

Chapter 12: Viewing PLM Data

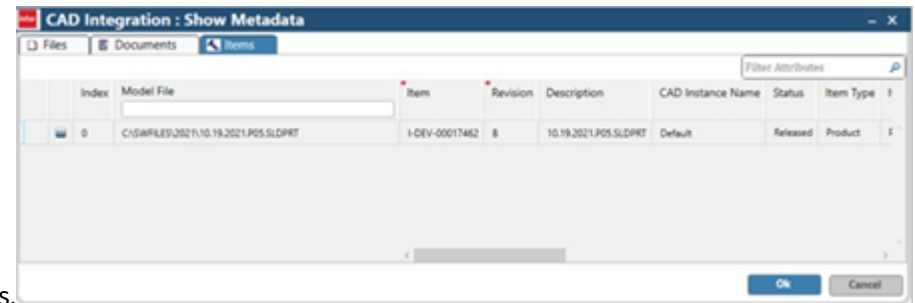
While working on a SolidWorks file that already exists in PLM, you may want to look into the PLM data related to the file and its components.

In SolidWorks, to view the relevant PLM data, proceed as follows:

- 1 On the PLM menu, select **Show Meta Data**. The Set Object Attributes dialog box is displayed. The default tab, **File**, displays the PLM data of all the files that are part of your SolidWorks structure.
- 2 Click the **Document** tab to display the PLM document data related to your SolidWorks part



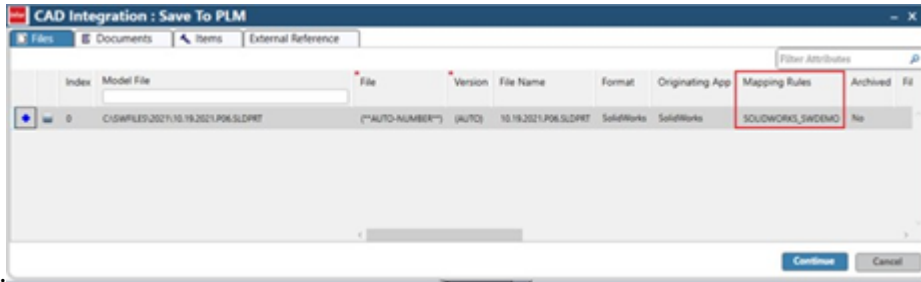
- 3 Click the **Item** tab to display the PLM Item data related to your SolidWorks



Assigning Mapping Template

Mapping template can be associated to SolidWorks files. The files is either already available in PLM or yet to be saved to PLM. You can assign a mapping template using the **Associate Mapping Template** menu.

Click the **File** tab on the Show Meta screen. View the template in the **Mapping Rules**



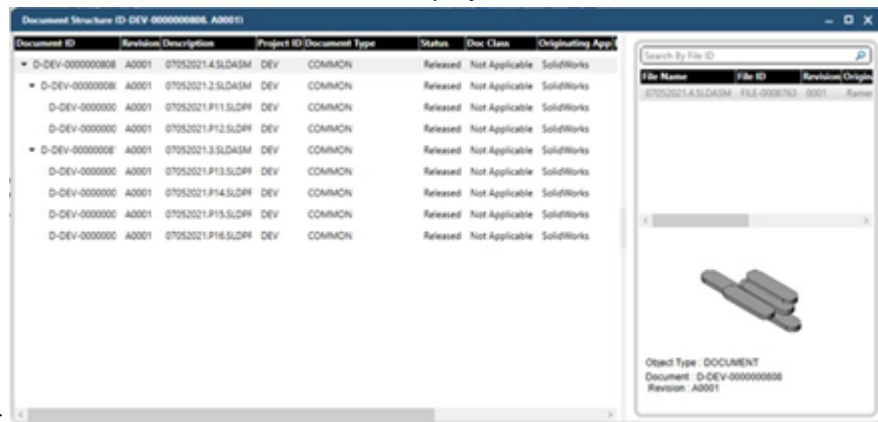
field.

Chapter 13: Opening a file in PLM

You can open a part file directly from the integration in a PLM item or document pane. To open a document in a PLM:

- 1 In the SolidWorks Feature Manager, select the root object.
- 2 Select Show Document Structure icon or Show Item Structure icon in the PLM menu or toolbar. The document is opened in the appropriate PLM structure. The Document Structure displays the document that was created in PLM. The Item Structure displays the items created for that

document.

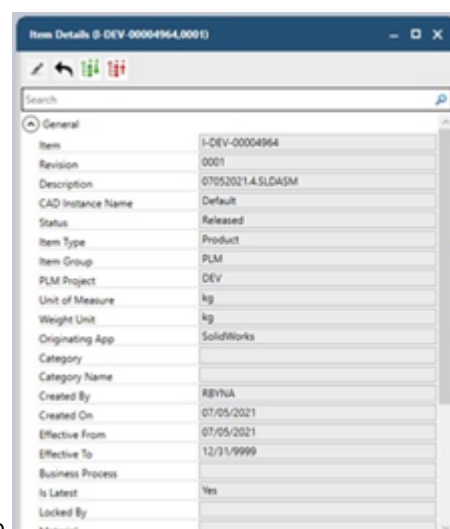


Chapter 14: Using Infocards

An infocard is a dialog box that enables you to update meta data of items, documents or files generated from part files created in SolidWorks. There are three types of infocards, one for items, one for documents, and one for files. You must use infocards to make changes to the attributes of item, document or file from SolidWorks rather than in PLM. You can update only those items, document, or files that you own.

To update items, documents or files generated from the SolidWorks object that you are working on, proceed as follows:

- 1 Select either an assembly or a part from the SolidWorks Feature Manager Design Tree.
- 2 From the PLM main menu or the right-click menu, select the relevant update option. You can select **Update Item**, **Update Document** or **Update File**. As a result, the relevant pane is opened in Web PLM.
- 3 Make the required changes.
- 4 Click **Save**.



You can update the meta data of a file using the Details pane.

Note: To update a component of an assembly, in SolidWorks, open the Assembly Navigator and double-click one of the components. The title bar displays the selected component-assembly relationship. To open the infocard, select **Update Item**, **Update Document**, or **Update File** from the PLM menu.

If the part file whose item, document or file you want to update is not saved to PLM, a message appears informing you that the selected part does not exist in the PLM database.

If the part file is not editable because you have no editing rights, the part file is locked by another user, or because the object is released, the infocard dialog box appears in display only mode.

If you select **Update Item**, but no item is created for the part file with which you are working, the following error message appears:

The file [filename] you have selected is not associated with an Item in PLM.

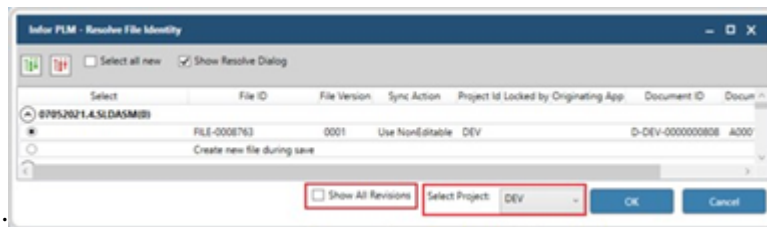
This occurs if your preferences specify that no PLM items must be created for the part files that you save to PLM

Chapter 15: Synchronize Headers

Header information is the PLM data that is stored in your local environment. This information is used by the PLM Integration for SolidWorks to verify whether a specific local file can safely overwrite the PLM file. The Synchronize Headers operation updates the headers of the SolidWorks files currently opened with the relevant PLM information.

If you are using PLM data to be displayed in your title box drawings, **Synchronize Headers** retrieves the latest data to be displayed. If you receive a file from an external developer, and you must replace the PLM file with the new file, you can use Synchronize Headers to identify the local file. In this case the PLM file will be overwritten by the local file while saving to PLM.

When you click Synchronize Headers, the PLM Integration compares the metadata (version, revision and status) in the local system with the metadata of the file in the PLM database. In case the PLM Integration detects a discrepancy in the metadata of the files compared, the following Resolve File Identity screen is



displayed:

By default, the PLM Integration displays only latest revisions / versions of the file. The PLM Integration selects the most recent revision / version.

File ID	The name of the file.
Version	The version of the file.
File Name	The name of the file.
Overwrite File	The permission to overwrite the file.
Project ID	The ID of the project in which the file is saved presently.
Locked By	The ID of the entity which locked the file.
Originating App	The application which created the file.
Document ID	The ID of the document linked to the application.
Revision	The present revision number of the file.
Status	The present status of the file.

Description	The description of the file.
Effective From	The date from which the file is effective.
Effective To	The date to which the file is effective.
Business Process ID	The business process ID of the file.
Show All Revisions	Displays all the available revisions of the file.
Select Project	The name of the present project in which the file is saved. In case you select a new project, the PLM Integration automatically selects the option Create New File During Save .
Save	Saves the selected file in the local system.
Print	Prints the selected file.
OK	Synchronizes the Local storage with the data stored in PLM.
Cancel	Stops the synchronization process.

The following are the salient features of the Synchronize Headers process:

- All the files are displayed in a single screen.
- By default the selected records are synchronized to the latest revision in PLM. Users can select or deselect the files displayed.
- By default only the latest revisions are displayed. Select Show All Revisions to view all the revision for the project.
- You can change the project and synchronize the data from the selected project.
- You can save all the data displayed in the Resolve File Identity screen or only the selected data.
- By default the file is saved in the Edit location with the name as <RootFileName.extn>_<date>_<time>.txt.

Chapter 16: Clear Storage Information

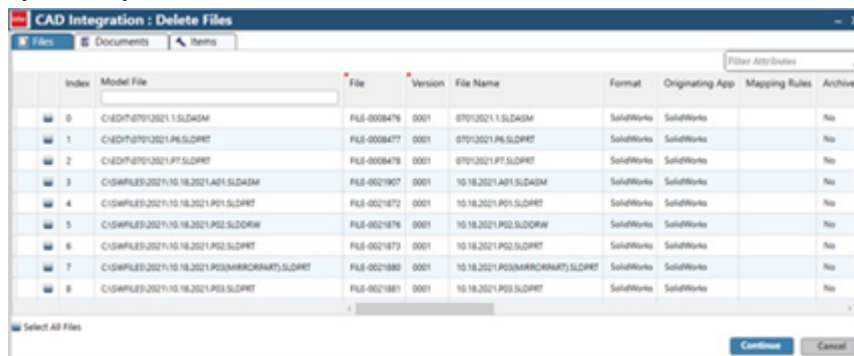
Clears the storage information for the active/selected file and its dependents. When you click **Clear Storage Information**:

- Clears the data for the active/selected file and its dependents from the storage data tables. The Infor PLM treats the part as new. The Clear Storage Information functionality is designed to enable the user to quickly save the active/selected.

Chapter 17: Delete Local Files

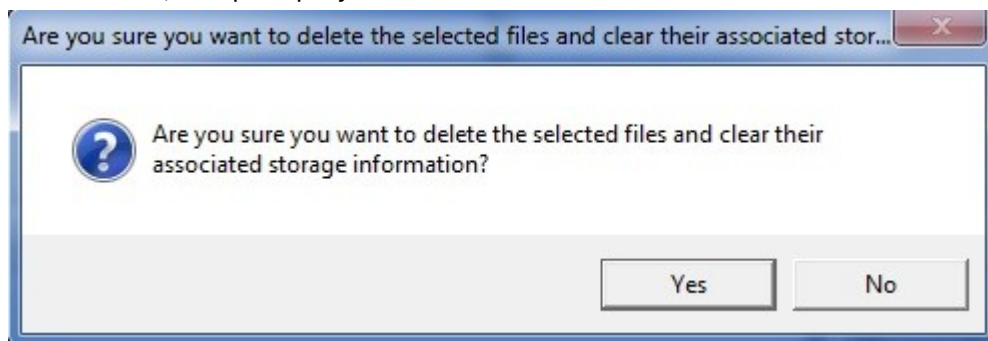
Use the PLM menu option Delete Local Files icon to delete the selected local files. PLM also clears the information about the deleted files from the storage files.

The CAD Integration: Delete Files screen displays the files that are registered in the storage files and saved in the local system. By default, the files are selected for



deletion.

When you click **Delete**, PLM prompts you to confirm the file



deletion.

Click **Yes** to delete the selected local files. PLM also clears the information about the deleted files from the storage files.

Note: You cannot delete the files that are open. The selection box for the open files is disabled.

Chapter 18: Refresh Files from PLM

You can click Refresh files from PLM icon to update the currently open files from Infor PLM.

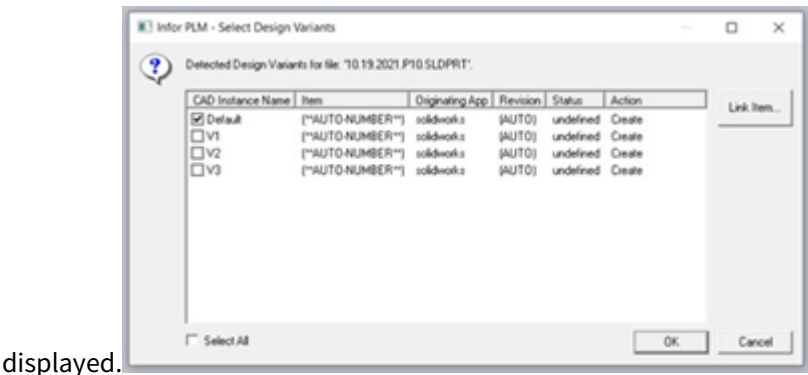
Note: Refresh files from PLM is only applicable for the files present in Edit/View directories.

Chapter 19: Design Variants

The **Capture Design Variants** feature enables you to automatically or manually link items to design variants or configurations for parts and assemblies.

Managing design variants/ configurations

To capture configurations and manage the links between model configurations and PLM items, select **Capture Design Variants** from the PLM menu. The PLM Select Design Variants dialog box is



In this dialog box, you can define how each configuration must be linked to an Item. To select configurations, select the check boxes of the configurations to be linked or select the **Select All** checkbox. The data that you define in this dialog box are saved locally, until you save the items linked to the configurations to PLM.

Note: Item IDs can be set as CAD Instance Name. For more information, refer to "Set CAD Instance Name as Item-ID" on page 125.

Generate Items for all Configurations

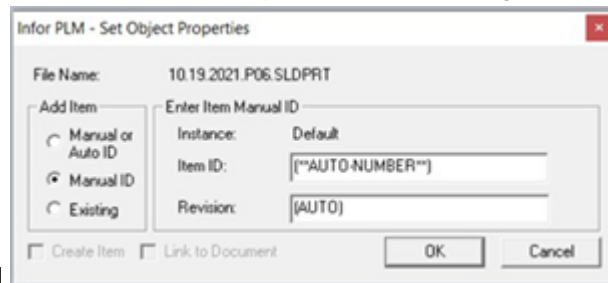
If this check box is selected, in PLM preferences, items for all the configurations are generated and linked to document in PLM when you perform the Save to PLM operation, regardless whether Capture Design Variants steps are performed or



Linking configuration to existing item

To link a configuration to an item present in PLM, proceed as follows:

- 1 Select the line with the required configuration.
- 2 Click **Link Item**. The PLM Set Object Properties dialog box is



displayed.

- 3 Select **Existing**.
- 4 Specify the item ID and the revision number in the Item ID and Revision fields.
- 5 Click **OK**. If you save the linked items to PLM, the configuration data is also stored in PLM.

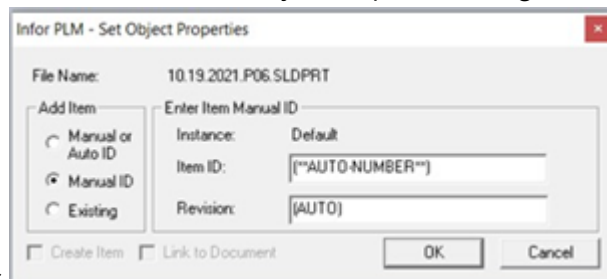
Automatically create item for configuration

If item IDs are automatically created according to the preference settings, in the PLM Select **Design Variants** dialog box, just select the required configuration. When you save the file to PLM, the item for the configuration is automatically created. For further information on preference settings, see Integration Preferences.

Manually create item for configuration

If item IDs are manually created according to the preference settings, to create an item for a configuration:

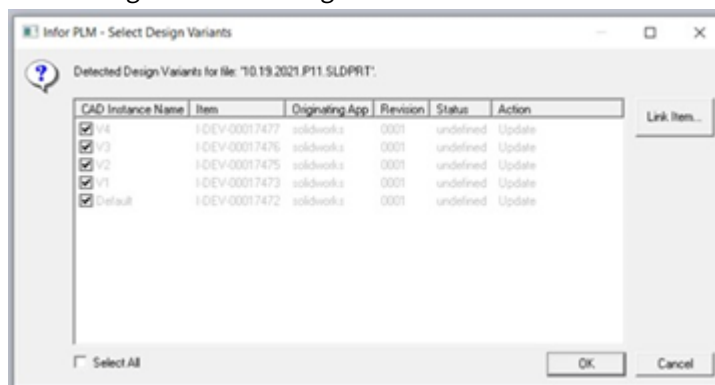
- 1 Select the line with the required configuration.
- 2 Click **Link Item**. The PLM Set Object Properties dialog box



appears.

- 3 Specify the **Item ID**.
- 4 To have PLM define the first revision, click **Manual** or **Auto ID**.
- 5 Click **OK**.

As a result, the PLM Select Design Variants dialog box shows the details of the existing item with the link



to the configuration.

If you click **OK** in the PLM Select Design Variants, the data is saved locally. If you save the linked item to PLM, the linked configuration data is also stored in the PLM database.

After saving the data locally or to the PLM database, you can use the PLM Select Design Variants dialog box to view the links between the model configurations and the PLM items.

Best practices

To change or update design variants in the CAD application, do not manually update the SolidWorks Design Table (the internal Excel table). The reason is, that the Design Table includes details that are also stored in the PLM items associated with the design variants.

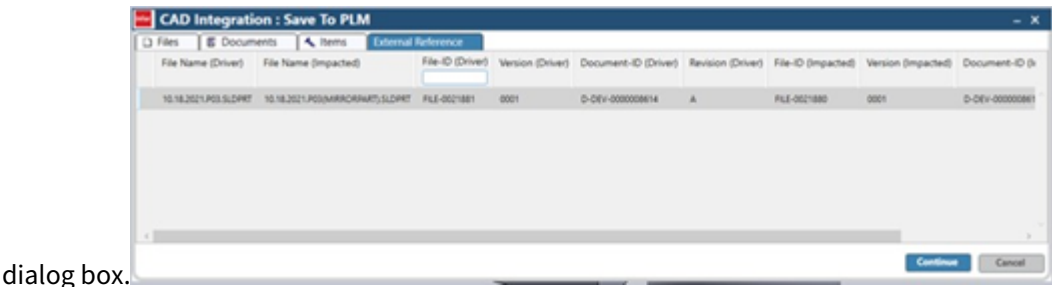
Such updates must be performed in PLM. For example, to remove a design variant, you must unlink the variant item from the related document. When the variant is next opened in SolidWorks from PLM, you are prompted about the changes.

To modify any SolidWorks configuration, including the Default name, use the SolidWorks **Configuration Manager** tab. When you save to PLM after modifying the configuration, the related item's CAD instance name attribute will be updated.

Note: Configuration name modifications to the Default name must never be performed using the SolidWorks Design Table (the internal Excel file). Using the SolidWorks Design Table for this purpose results in the creation of an additional, redundant configuration.

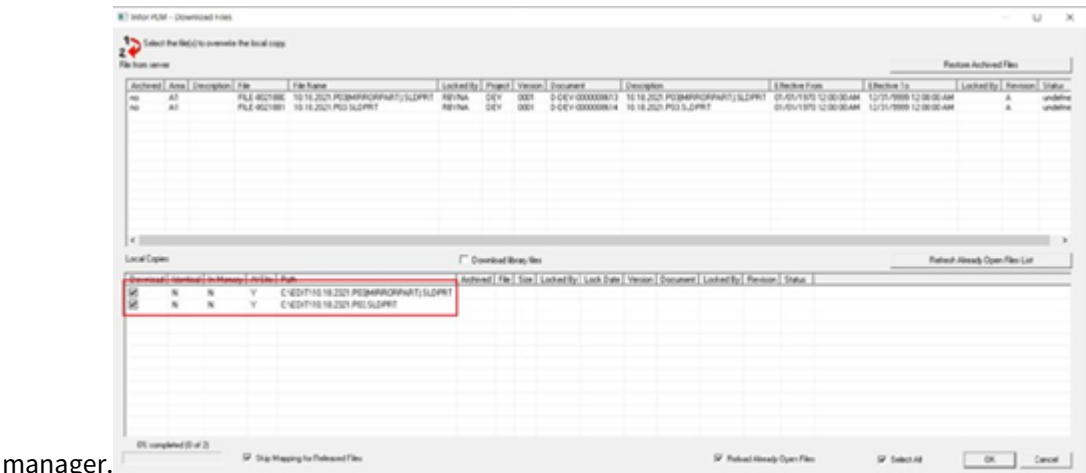
Chapter 20: Associative Links

SolidWorks part or an assembly can be dependent on another part or assembly for its design. The associative links functionality, saves the associations between the SolidWorks files and PLM when you perform the Save to PLM operation. To actualize this functionality, the **Reference Links** tab is included in the Set Attributes



Edit file

When you download a SolidWorks file (Part or Assembly), all the drivers and the impacted files associated with the SolidWorks file are also downloaded. These files are listed in the download



Check-In

When you perform a Check-In operation on a file from the Integration or from the PLM, the application check-in all the drivers of the file.

Check-out

When you perform the Check-Out operation on a file from the Integration, the application checkouts all the impacted files.

Integration Preference

To enable the associative links functionality, you must enable the **Save External References** check box, in **General** tab of the Solidworks Preference session.

Chapter 21: Dispatch to business process

The **Dispatch to Business Process** option on the PLM menu enables you to link the PLM items and/ or documents, which are generated for a SolidWorks file, to a PLM business process. As a result, PLM items and/or documents of SolidWorks file are linked to a workflow, which is distributed to the users associated to the workflow template.

For further information, see [Attach to Workflow](#) on page 108.

Dispatching documents and/or items to a business process

To dispatch documents and/or items to a business process, complete the following steps:

- 1 Select **PLM > Preferences**. . . . The Integration Preference screen is displayed.
- 2 In the **Attached to Business Process** field located under the **Attached to Workflow** category, select the PLM objects that can be dispatched to the business process. For further information, see [Dispatch items to a business process](#).
- 3 Click **OK**.
- 4 In the SolidWorks application, save the file to PLM.

- 5 Click **Dispatch to Business Process** on the PLM menu. The Business Process Details screen is

displayed.

- 6 Specify the business process data and select a workflow template.
7 Click **Save**. The business process launches after you save the template.

Dispatching related models or drawings

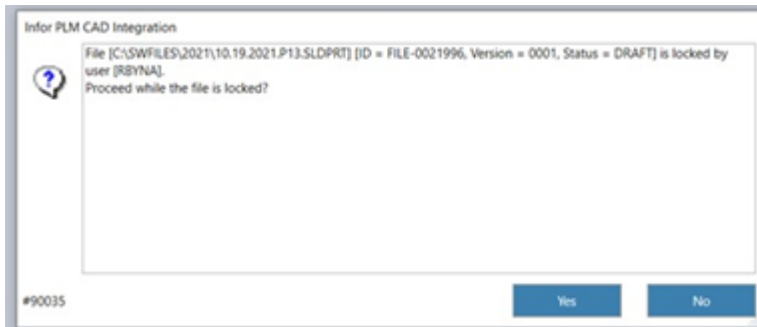
You can attach drawings of a model file to the business process. The model files are either part files or assembly files. To dispatch a drawing to the business process, enable the Attached related Models\Drawings check box. When you dispatch a model file to the business process, the drawings and the PLM objects of the models are linked to the same business process.

If you dispatch a model file to the business process, related drawings are attached to the business process, to which the models are linked. Similarly, if you dispatch a drawing file to the business process, related models are attached to the business process, to which the drawings are linked.

To dispatch drawings to a business process

- 1 Select **PLM > Preferences.....** The Integration Preference screen is displayed.
- 2 Select the **Attached related Models\Drawings** check box located under the Attached to Workflow category.
- 3 Click **Ok**.
- 4 In the SolidWorks application, save the file to PLM.

5 Click **Dispatch to Business Process** on the PLM

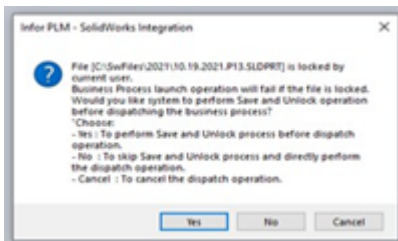


menu.

6 Click **Yes**. The Business Process Details pane is

displayed.

7 Specify the business process data and select a workflow template.



8 Click Save icon.

9 To save and unlock the file, click **Yes** or click **No**. On a successful completion of the business process, confirmation message is displayed.

Note:

- Save all the drawing attached to the model file, before you dispatch a model file to a business process.

- Ensure that the model files and the drawing files are open in SolidWorks, before you dispatch the model file to business process.

Controlling Locked Objects

If you select a value for **Action for Objects locked by Business Process in Integration Properties for Integration**, it controls the behavior of the integration when the objects that are part of current business process are locked.

The following screen shows the options in Attached to Work Flow preference for



Attached to Workflow		
Attached to Business Process	USER	Both
Attach related Models/Drawings	USER	<input checked="" type="checkbox"/>
Attach all related objects	USER	<input type="checkbox"/>
Allow edit Workflow	ADMIN	<input type="checkbox"/>
Action for objects locked by Business Process	ADMIN	Cancel action

These options are available for Action for Objects locked by Business Process:

Attached as Unlock

If any object is locked by any other business process, then integration will attach that object to the newly created Business Process without taking lock for that object.

Do not attach

If any object is locked by any other business process, then integration will not attach that object to the newly created Business Process.

Cancel action

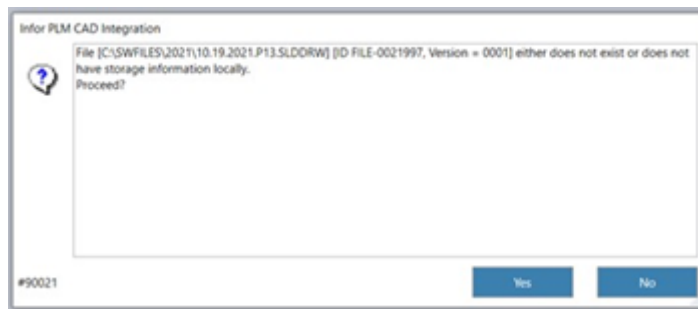
If the object is locked by any other business process, then integration will cancel the current dispatch to Business Process operation.

Always attached as Unlock

All the objects are attached to the newly created Business Process without taking lock for any of the objects.

Troubleshooting Scenarios

- The drawing file is missing on the local system:
If you dispatch a model file to a business process when the related drawings are deleted from the local system, the warning message is displayed.

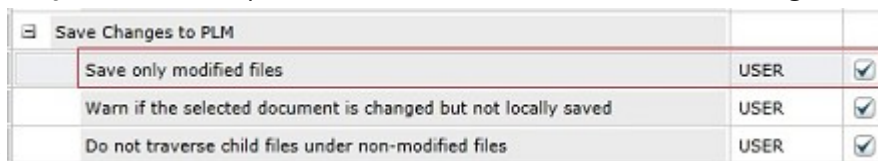


- The local version of drawing different from the version linked to the model in PLM.
- Drawing file not open in SolidWorks Application:

If you dispatch a model file to a business process when the related drawings are not open in SolidWorks application, a warning is displayed.

Chapter 22: Saving Modified Files Only

Generally, the Save to PLM process performs the operation on all the editable files within the structure of current file and transfers all these editable files to PLM. However, it is possible that one or more files within the structure of the current file are unchanged even though they are editable; therefore, it is not required to save them into PLM. To save only the modified files, enable the **Save only modified files** integration preference. The **Save only modified files** preference is made available in the **Save Changes to PLM**



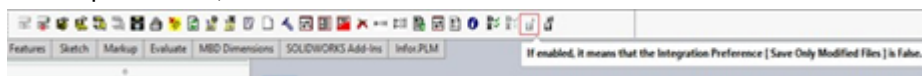
property.

You can also enable or disable the Save only modified files from the add-in

toolbar.

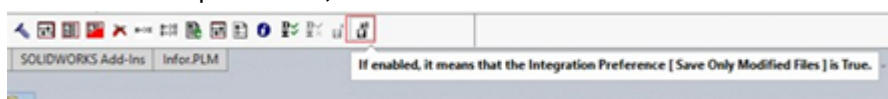


The first button, when enabled indicates that the integration preference **Save only modified files** is enabled. To disable the preference, click the



button.

The second button, when enabled indicates that the integration preference **Save only modified files** is disabled. To enable the preference, click the



button.

For Example,

```

MA.sldasm
|--> SA1.sldasm
    |--> P1.sldprt
    |--> P2.sldprt
|--> SA2.sldasm
    |--> P3.sldprt
    |--> P4.sldprt
  
```

In the following structure:

Assume that all files A1, SA1, SA2, P1, P2, P3 and P4 are new. Normally, when you perform Save to PLM for assembly A1, all the files are saved to PLM.

Now, assume that you have made changes to MA, SA1 and P1 files in the A1

```
MA.sldasm (changed)
|--> SA1.sldasm (changed)
    |--> P1.sldprt (changed)
    |--> P2.sldprt
|--> SA2.sldasm
    |--> P3.sldprt
assembly: |--> P4.sldprt
```

When you perform the Save to PLM operation on the above file structure, not only the changed files, even the unmodified files are also saved to PLM. However when you enable the Save only modified files preference, only MA, SA1 and P1 files of the A1 assembly are saved to PLM.

Important Points and Limitations

These are the limitations and important points to be considered for Saving modified files only preference:

- The Save only modified files preference is applicable only for Save to PLM operation and is not applicable for Save and Unlock and Check-In operations. The reason being that if the files contain mapping, then even if the files are unchanged, still the Save and Unlock and the Check-In operations might modify the files as a result of the mapping definitions.
- Operations such as Link to Item, Capture Design Variants, Associate/Remove Mapping Template mark the file as locally modified. Therefore, you must save the files to PLM.
- During Save to PLM operation, if you take ownership of any file, then those files must also be saved to PLM.
- Changing/setting the values of any of the integration preference values does not consider the file to be requiring saving into PLM. Example setting/changing of neutral file generation options, setting thumbnail generation preferences, usage of preference **Generate items for all configurations** and so on, is not considered for treating the file as requiring saving into PLM.
- If a model/assembly is changed, and user performs Save to PLM from the model and the **Save only modified files** check box is selected, then it is assumed that the associated drawing must also be saved to PLM. In the dialog PLM - Select Parents to Save to PLM, system will list the drawings for only those models that are changed and require saving into PLM.

Integration Preferences to Save Changes to PLM

Use the Save Changes to PLM property to control the aspects related to saving the SolidWorks files to PLM.

The **Save Changes to PLM** property comprises of the following options:

- Save only modified files
- Warn if the selected document is changed but not locally saved

- Do not traverse child files under non-modified files

Save only modified files

If this check box is selected, only the modified files are saved to PLM when you perform the Save to PLM operation.

The **Save only modified files** preference is applicable only for Save to PLM operation and is not applicable for Save and Unlock and Check-In operations. The reason being that if the files contain mapping, then even if the files are unchanged, still the Save and Unlock and the Check-In operations might modify the files as a result of the mapping definitions.

Warn if the selected document is changed but not locally saved

If this check box is selected, when you perform the Save to PLM operation on a document, which is modified in the application session, then you will be prompted to save the file on a local system and then try Save to PLM.

If this check box is cleared, you are not prompted to save the file on a local system, the Save to PLM process is executed. However, it is suggested that you save the file on a local system before saving it to PLM, when Save only modified files option is enabled.

Do not traverse the child files under non-modified files

If this check box is selected, the child files attached to the unmodified files are not traversed. By default, the entire structure is checked for the editable files. If this preference is enabled, integration will not traverse CAD files for children under the sub-structure of non-modified files. Enabling this preference improves the performance and minimizes application memory usage. It shall be noted that, then even if this preference is checked, integration will still look for the modified child files (within the structure that are tagged as modified) by using the information under Toolkit local storage (StorageTableChildren.txt).

In order to know which files within the structure are modified and requires saving to PLM, integration traverse the structure of the CAD file downwards (within the CAD Tree). This causes loading of the files (even if those files require saving into PLM or not).

```
A1.sldasm (changed)
|--> A2.sldasm (changed)
    |--> A3.sldasm
        |--> P4.sldprt
```

For Example, Consider below structure:

For example, in the above case, when you perform Save to PLM operation for A1.sldasm, the files A3.sldasm and P4.sldprt are also loaded into application memory (even though these files are not transferred to PLM, and are unchanged).

When Do not traverse child files under non-modified files is enabled, integration does not traverse downwards within the structure of the non-modified files.

There are situations where one of the child file is changed, but its parent is not changed.

```
A5.sldasm (changed)
|--> A6.sldasm
    |--> P7.sldprt (changed)
    |--> P8.sldprt
```

For Example,

In this case, if the Do not traverse child files under non-modified files is enabled, then although integration will not traverse downward within the structure of non-modified files to find out the child files that are changed. However, the integration can identify the file P7.sldprt under the structure as requiring saving into PLM. This is possible by using the information from Toolkit storage file %CFE_CLIENT_ HOME%\Toolkit\data\StorageTableChildren.txt.

Chapter 23: General Mapping

General mapping enables you to define attribute-based mapping rules for a template. When you create parts, assemblies, or drawings in SolidWorks using a template, the mapping rules are carried over to the parts, assemblies, or drawings based on how you set up the rules.

The mapping rules determine how values of SolidWorks attributes are mapped to PLM attributes and vice versa for part files that use the template containing the mapping rules.

In SolidWorks, create template Template_1. For Template_1, define the following mapping rule: SolidWorks attribute Weight goes to PLM item attribute **Estimated Weight**.

Save Template_1 as part file A0001. As a result, the value calculated for the Weight attribute of part file A0001 is mapped to the **Estimated Weight** field of the item that is created when you save part file A0001 to PLM.

For any following part files for which you need the mapping rules defined in Template_1, open

Template_1 and save Template_1 under the desired part file name.

Note: To use general mapping, you must have administrator's rights.

Creating the Template

To use general mapping, templates must exist for which mapping rules have been defined. You can only create mapping rules if you have administrator's rights.

To create a template and define mapping rules for the template, proceed as follows:

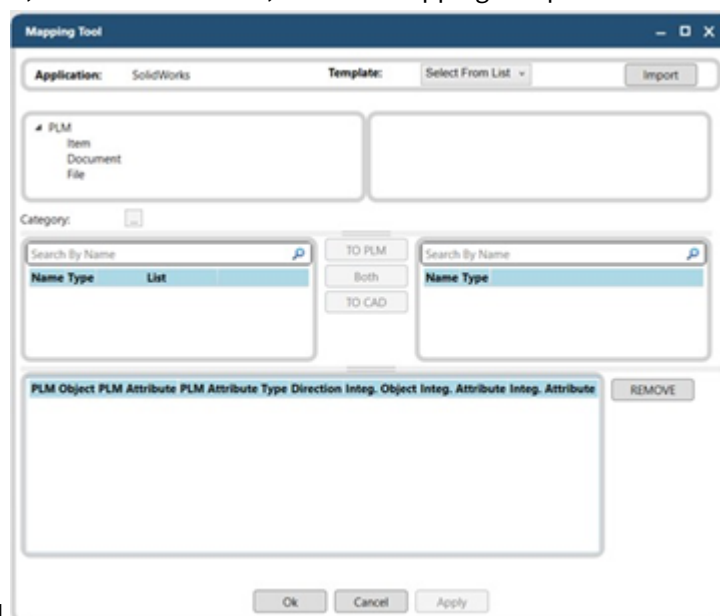
- 1 In SolidWorks, select **File > New** to create a new template file.
- 2 Specify the attribute properties to be mapped to PLM for the template.
- 3 Click **OK**.
- 4 Open the template in the SolidWorks.

Note: To change the properties of a template, open the template file, adjust the properties, and save the template file.

For general mapping to work, make sure that the assemblies, parts, and drawings that you use as templates for general mapping are stored in the <SolidWorks installation path>\Solid Works\data\templates folder.

Opening the template in SolidWorks

- 1 In SolidWorks, from the PLM menu, click Use Mapping Template tool. The PLM Mapping Tool dialog box



is displayed.

- 2 Select SolidWorks in the **Application** field.
- 3 In the **Template** list, select the template for which you want to define mapping rules.
If the template is new or if changes were made to the template in the CAD application, you must import the template into PLM first. For further information, see [Importing the template into PLM](#) on page 59. Any previously defined mapping rules for the selected template are displayed in the dialog box.

Importing the template into PLM

Before you can define mapping rules for new or changed templates, you must import them in PLM. Re-importing a template will not remove any previously defined rules for this template.

To import a template:

- 1 From the Application list in the upper left of the PLM Mapping Tool dialog box, select **SolidWorks**.
- 2 Click **Import**.
- 3 In the Open dialog box that appears, select the template for which you want to define mapping rules and click **Open**.

As a result, the template properties are displayed in the upper right section of the PLM Mapping Tool dialog box.

Defining the mapping rules

In the PLM Mapping Tool dialog box, you can define the mapping rules that are used to map attributes between PLM and SolidWorks. This dialog box is divided into two sections, one for the PLM attributes and one for the SolidWorks attributes.

To map SolidWorks attributes to PLM attributes and vice versa.

To map, for example, the value of the **Author** attribute in the Summary groups in SolidWorks to the **Description** attribute of an item from PLM, proceed as follows:

- 1** Click the Item icon on the left side of the PLM Mapping Tool dialog box.
A list of item attributes is displayed on the left side of the middle section of the screen.
- 2** From the list of item attributes, click **Description** from the list.
- 3** From the SolidWorks side of the window, click **Summary**.
A list of attributes is displayed.
- 4** Click **Author** from the list of attributes.
- 5** Click the **To PLM** button in the middle section of the dialog box. This maps the value of the **Author** attribute to the **Description** attribute of the item in PLM.
You can add more mapping rules before accepting them. See [Mapping options](#) on page 60 for further information on the available mapping options.
- 6** When you are through specifying mapping rules, click **Apply**.
- 7** Click **OK** to exit the PLM Mapping Tool dialog box.
- 8** For the mapping rules to take effect, in the PLM client, disconnect and then reconnect to the server.

Mapping options

There are countless variations of the attributes you can map. You can select an attribute associated with an object in PLM and map it to an attribute in SolidWorks or you can reverse the direction and map an attribute value in SolidWorks to an object's attribute in PLM.

For this purpose, make selections from the PLM and SolidWorks sections and click the button that describes the direction you want the mapping to take place.

- The **To CAD** button maps the selected attributes from PLM to SolidWorks.
- The **To PLM** button maps the selected attributes from SolidWorks to PLM.
- The **Both** button maps the selection to both PLM and SolidWorks. This mapping rule goes in both directions, which means the values remain in sync.

In SolidWorks, if you are working with a part file and you enter a value for an attribute that is mapped to both, the following dialog box appears when you save the part file to



PLM.

This dialog box prompts you to indicate which value you want to keep the value just entered in SolidWorks or the value entered in PLM.

Note: Depending on the selected type of attribute, the **To CAD**, **To PLM**, and/or **Both** buttons can be unavailable. For more information on mapping restrictions. See [Mapping restrictions](#) on page 61.

You can map one SolidWorks attribute to one PLM attribute, one SolidWorks attribute to many PLM attributes, or many SolidWorks attributes to one PLM attribute.

For example, if you mapped a second SolidWorks attribute, to the item's description in PLM and you save to PLM, the value of the item's description in PLM will be the two values you mapped from the Author and Date attributes in SolidWorks separated by a comma.

Note: You cannot map more than one attribute to a target attribute in a format other than string. If you do, the system regards the two or more concatenated values (separated by a comma) that are mapped to the target attribute as one value in string format and thus generates an error message. For example, map the values of the **Mass Properties** field and the **Mass Properties** field of the Configuration Specific group, which are in double format, to the **Estimated Weight** field of an item in PLM, which is also in double format. As a result, when mapping is performed, an error message appears informing you that the mapped values must be in double format.

Mapping restrictions

Depending on the attributes selected in the PLM Mapping Tool dialog box, mapping to PLM, to SolidWorks, or both can be unavailable. The mapping restrictions are described in the following sections. Attributes not included in the following lists are available for mapping to PLM, SolidWorks, and to both. For all mapping rules that you want to define, the target attribute must be a modifiable field.

The following PLM attributes are only available for mapping from PLM to SolidWorks:

- PLM Effective From
- PLM Effective To
- Item:

Lifecycle

All attributes related to an item's life.

Business Process

The attributes related to an item business process or a business process.

Keywords

Keyword attributes.

Business Partner

Manufacturer, supplier, customer, and subcontractor attributes.

- Document:

Lifecycle

All attributes related to an item's life.

Business Process

The attributes related to an item business process or a business process.

Keywords

Keyword attributes.

Project

All project attributes.

Attribute format restrictions

To enable you to define mapping rules, the PLM attributes and the SolidWorks attributes must have the same format or must have a conversion in order to facilitate the following matches:

Table 1: Attribute formats

From:	To:	String	Integer	Date	Real
String		+			
Integer		+	+ (*)		+ (*)
Date		+		+ (*)	
Real		+			+ (*)

+ (*) According to regional settings.

The target attribute must be of type string if more than one attribute is mapped to this attribute. The target PLM attribute to which SolidWorks dimensions are mapped is of type string, because the value and the unit must be concatenated to a string. The target PLM attribute in a SolidWorks file-specific rule can have no other general mapping.

SolidWorks

Attributes from the SolidWorks Configuration Specific group can only be mapped to PLM item attributes. The reason is, because configuration specific attributes have different values for each configuration, these

values must be mapped to the item representing a particular configuration and not to the document that describes all configurations for a particular assembly.

Relationships

You can map one SolidWorks attribute to one PLM attribute, one SolidWorks attribute to many PLM attributes, or many SolidWorks attributes to one PLM attribute.

Associating the mapping rules to part files, assemblies, or drawings

Now that you have created mapping rules and saved them to a template, you can create parts, assemblies, or drawings in SolidWorks using this new template. The part, assembly, or drawing acquires the values of the attributes you defined in the mapping rules. Only the owner of a file or the project administrator can perform associations.

To associate mapping rules to a file:

- 1 In SolidWorks, open a template file for which you have created mapping rules from the `<Solid Works installation path>\SolidWorks\data\templates` folder as described in the previous sections
- 2 Save the template file under another name (that is, under the name of the part, drawing, or assembly that you are going to work on and to which you want the mapping rules of the template to apply). As a result, the new part file (or drawing or assembly, as the case may be) has obtained the attributes and the mapping rules of the template.
- 3 Save the part file with the new name to PLM.

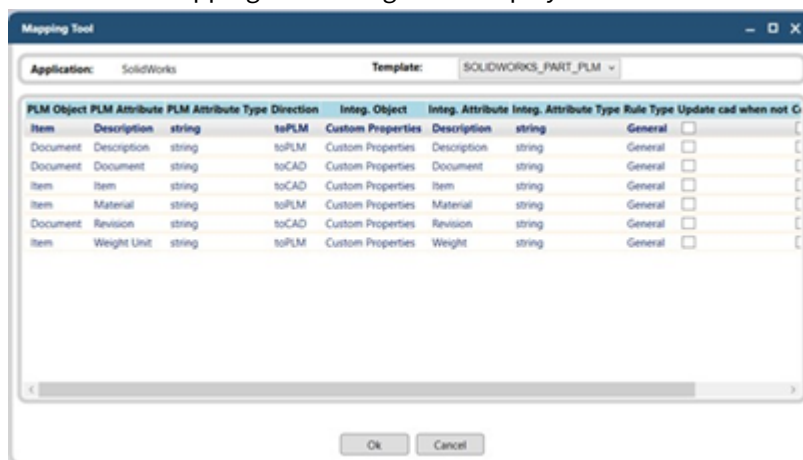
The Associate Mapping Rules menu option

In addition to the methods described in Associating the mapping rules to part files, assemblies, or drawings, you can use the **Associate Mapping Rules** option from the PLM menu to associate mapping rules to files. Note that you can only use this option for files not associated to a mapping template previously by opening the template and saving the file under another name.

To associate a file to a template using the **Associate Mapping Rules** option, proceed as follows:

- 1 In SolidWorks, open the file for which you want to associate mapping rules.
- 2 Click `PLM > Mapping > Associate Template`.

The PLM Discrete Mapping Tool dialog box is displayed in the



PLM.

- 3 From the Template list, select a template you created in SolidWorks. The mapping rules for this template are displayed.
- 4 Click **OK**.

The mapping is completed when you save to PLM.

After the mapping rules are associated, you can click **Display Mapping** from the PLM menu to display the current mapping rules.

Removing mapping associations

You can remove the rules that are associated with the template. For this purpose, select **Remove Association** from the PLM menu. Only project administrators who own the file can remove an association. If you remove an association, any file-specific mapping rules are kept.

Using configuration-specific mapping

For each template, you can define mapping rules for attributes that only relate to a specific configuration. Mapping can take place between the parts and the PLM items.

To perform configuration-specific mapping, proceed as follows:

- 1 Define a mapping rule using a configuration-specific attribute in SolidWorks. For this purpose, on the PLM Mapping Tool dialog box, select **Configuration-specific** under the SolidWorks section.
- 2 Define a rule to map, for example, map the value Mass in the Mass properties in SolidWorks to the item's Estimated Weight in PLM.

Note that users must avoid defining configuration-specific mapping for the following attributes:

- Item_ID

- Revision
- Status

These attributes are used to create design variants and serve as links between design variant instances and PLM items. For further information, see [Managing design variants/configurations](#).

- 3 Create a file in SolidWorks.
- 4 Specify configurations. For example, specify a value for Mass.
- 5 Capture the design variants for the configuration. For further information, see [Managing design variants/configurations](#).

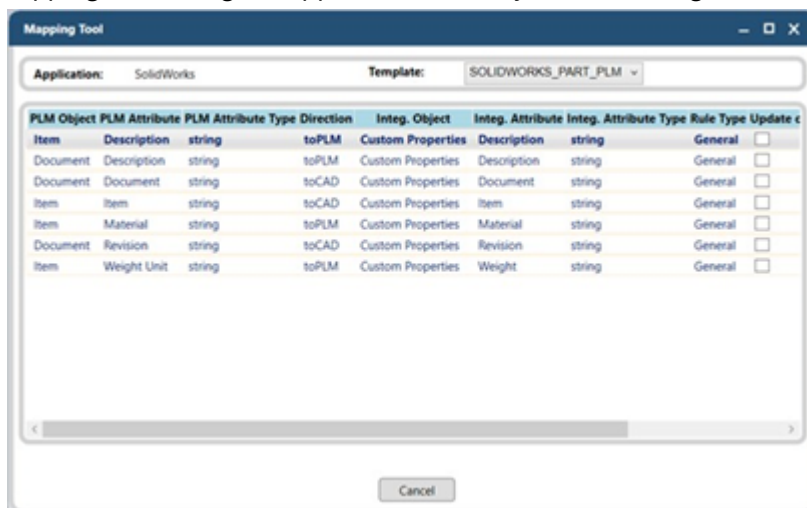
This procedure is applicable for all parts, assemblies and drawings. The mapping groups may be different for these components. The mapping groups are listed in the following table.

Parts	Assemblies	Drawings
Summary	Summary	Summary
Custom Properties	Custom Properties	Custom Properties
		Notes
Material Properties		
Configuration-specific Properties	Configuration-specific Properties	Sketch Dimensions
Notes	Notes	Blocks
Specific Attributes	Specific Attributes	
Feature Dimensions	Feature Dimensions	

Display Mapping

You can use the **Display Mapping** option in the PLM menu to view the general or file-specific mapping rules defined for the template associated to the part file with which you are working. If you click this option, the

PLM Mapping Tool dialog box appears in view-only mode showing the associated template and the mapping



rules.

Applying the mapping rules

For the mapping rules to take effect after you define or update them in the Web PLM as described previously, you must first disconnect and reconnect your CAD application to PLM.

The mapping then takes place depending on the rules that you set up. In the example in "Defining the mapping rules" on page 69, the mapping takes place when you:

- Save to PLM.
If the file is new and has not been saved to PLM, you must save the file to PLM to apply the mapping. Once the mapping is applied, you can select Update Item, Update Document or Update File from the PLM menu to see the mapped values. For further information, see "Using infocards" on page 41.
- Use the infocard options:
 - Update Item
 - Update Document
 - Update File
- Edit the files.

Chapter 24: Save Neutral Files Option

Use this option to specify an additional format to save the local SolidWorks files to PLM. As a result, if you save a part file to PLM, the part file is also saved in the additional format. The additional file is used for viewing

To enable the additional save, select the check box for the required file type and the radio button for the required format.

You can define formats for the following types of SolidWorks files:

- Parts
- Assemblies
- Drawings

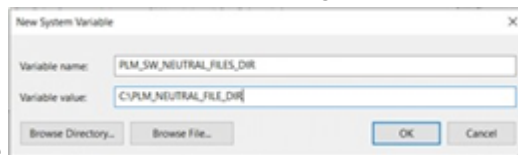
Following properties are relevant to the neutral files:

- Generate Neutral Files
- Show Neutral files during save

If this check box is selected, the information about the Neutral Files is displayed in the Set Attributes dialog box, before you save the files to PLM.

Location of generating neutral files

By default, the neutral files are created in the %TMP%/NeutralFiles or <PLM_Client>\Solid Works\Neutral Files folder. However, you can change the folder location by specifying the new folder path in the environment



variable.

If you set the environment variables as shown above, the neutral files are created in C:\PLM_ NEUTRAL_FILE_ DIR. Ensure that no other files are manually created in the folder, because the files in the folder are deleted when you close SolidWorks.

Generate neutral files during

Specify during what actions the neutral files should be generated. Available options are:

On All Actions

Neutral files will be generated during Save to PLM, Save and Unlock and Check-In actions.

On Save and Unlock

Neutral files will be generated only during Save and Unlock.

On Check-In

Neutral files will be generated only during Check-In.

Save Neutral Files Parts

Specify the different formats in which neutral files are to be generated; applicable for part files. The supported formats to generate neutral files:

- BMP
- PDF
- IGES
- ParaSolid
- ACIS
- Stereo Lithography

Save Neutral Files Drawing

Specify the different formats in which neutral files are to be generated; applicable for drawing files. The supported formats to generate neutral files:

- BMP
- PDF
- DWG Format
- DXF Format

Save Neutral Files Assembly

Specify the different formats in which neutral files are to be generated; applicable for assembly files. The supported formats to generate neutral files:

- BMP
- PDF
- ParaSolid

Chapter 25: Thumbnails

Thumbnail is a miniature representation of a SolidWorks part, assembly or a drawing. Solidworks integration generates the thumbnails in the JPEG or PNG format.

Thumbnails can be generated in the following ways:

- During the Save to PLM, thumbnails are created for editable files.
- Using the **Generate Thumbnails** option on the PLM menu, users can create thumbnails for the non-editable files in the structure.

Generating Thumbnails

To generate thumbnails for editable components, that is, to generate the thumbnails when you save the SolidWorks file to PLM, complete the following steps:

- 1 In SolidWorks application, select PLM > Preferences the PLM Preferences for SolidWorks pane is displayed.
- 2 Select the **Generate thumbnails during save to PLM** check box located under the Thumbnails category.
- 3 Click **OK**.
- 4 Click **Save to PLM** on the toolbar.

Note: Thumbnails are generated only for the editable components, that is, components with status Draft, and that are locked by you (current user). You can generate thumbnails for a SolidWorks file, only when you have the ownership for the components.

Generate thumbnails for non-editable files

To generate thumbnails for SolidWorks files saved to PLM and for non-editable files, complete the following steps:

- 1 Open the SolidWorks file. For further information on opening the files that are saved to PLM, see Editing files in the integration.
- 2 Click on the toolbar.

Thumbnail Locations

By default, thumbnails are saved in the %TMP%/ Thumbnails or %CFE_CLIENT_HOME_%\temp\ Thumbnails folder. To change the folder, modify the variable value of the **PLM_SW_THUMBNAIL_GEN_DIR** environment variable.

Note: Thumbnails are deleted from the local system when you close the SolidWorks application.

Thumbnails for Item Configurations

Thumbnails are generated for the default configuration, that is for the originating items, when the design variants are not captured. The thumbnails are linked to the document.


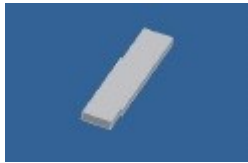

For Example, In the following configuration **PLM_Design_Variant** is the default part configuration and Variant



1 and Variant 2 are the design variants.

The following table indicates SolidWorks parts linked to various design variants of the configuration:

Table 2:

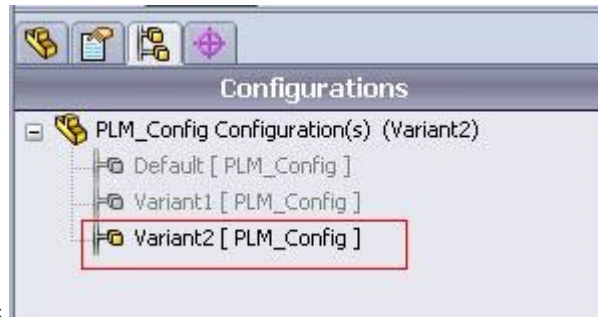
SolidWorks Configuration	Image
Default (PLM_Config)	
Variant1 (PLM_Config)	
Variant2 (PLM_Config)	

Generate Thumbnail for Active Configuration

To generate the thumbnail of the active configuration, enable the Use Thumbnail Information of **Active Configuration** check box in the PLM integration preferences. This scenario is valid when there is only one item or no items linked to the file. When the Use Thumbnail Information of Active Configuration preference is selected, thumbnails are not generated for the default configuration.

For Example,

In the following configuration **PLM_Design_Variant** is the default part configuration and Variant 1 and Variant



2 are the design variants.

When the **Use Thumbnail Information of Active Configuration** option is selected, the thumbnail is generated for the active configuration, that is, Variant 2. However, if the option is not selected, a thumbnail for the Default configuration is generated.

Integration Preferences for Thumbnails

The properties comprising the Thumbnails control the following aspects:

- SolidWorks files for which thumbnails can be generated.
- Format of the thumbnails.
- Level to which the thumbnails can be generated in a CAD structure.

The Thumbnails option includes the following properties:

- [Generate thumbnails during Save to PLM](#) on page 73
- [Generate thumbnails for part file types](#) on page 73
- [Generate thumbnails for assembly file types](#) on page 73
- [Generate thumbnails for drawing file types](#) on page 73
- [Use thumbnail information of active configuration](#) on page 73
- [Generate thumbnails for CAD file structure](#) on page 73
- [Format for thumbnails generation](#) on page 74

Generate thumbnails during Save to PLM

If this check box is selected, thumbnails are generated when SolidWorks files are saved to PLM. This option generates thumbnails only for the editable components of a SolidWorks file. A file is editable when the status is **Draft** and you have the ownership of the file.

Generate thumbnails for part file types

If this check box is selected, thumbnails are generated for the SolidWorks part files, that is, files with .prt or .sldprt extension.

Generate thumbnails for assembly file types

If this check box is selected, thumbnails are generated for the SolidWorks assembly files, that is, files with .asm or .sldasm extension.

Generate thumbnails for drawing file types

If this check box is selected, thumbnails are generated for the SolidWorks drawing files, that is, files with .drw or .slddrw extension.

Use thumbnail information of active configuration

If this check box is selected, thumbnails are generated for the active configuration of a part or an assembly. For further information, see [Generate Thumbnail for Active Configuration](#).

Generate thumbnails for CAD file structure

This preference is relevant only when you select the Generate Thumbnails... option from the PLM menu or when you click button on the toolbar. Allowed values:

All Levels

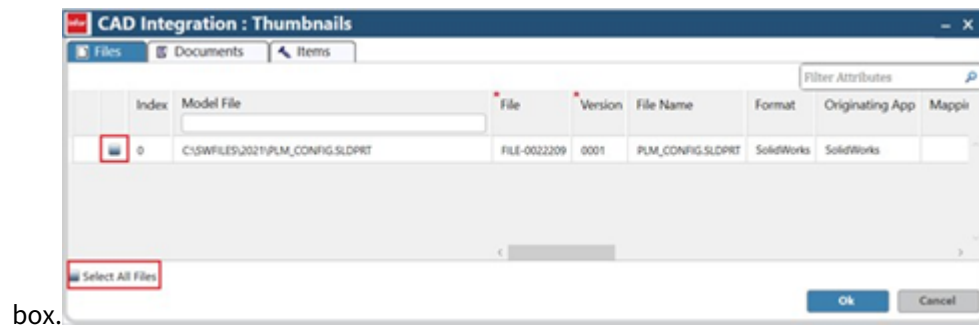
Thumbnails are generated for the files linked the selected file.

Selected document only

Thumbnails are generated only for the selected document.

Prompt

The Select files for Thumbnail Generation dialog box is displayed. To generate thumbnails for a particular document, select the relevant check



Format for thumbnails generation

Select the format in which the thumbnails must be generated. This field can have the following values:

- PNG
- JPG

Chapter 26: Balloon Mapping

You can now transfer balloon numbers from SolidWorks to the part list table in PLM. This information can be stored to any field of the Part List table, in addition to the Find No. field.

- [Constraint](#) on page 75
- [Solution](#) on page 75
- [Assigning mapping template](#) on page 76

Constraint

These are the constraints:

- Ballooning is done only for Top-Level components.
- When there exist multiple drawings in the system for the same [model + configuration], the information in PLM is overwritten with the ballooning information of last saved document.
- If ballooning is not performed for a document, then the information in PLM is not updated.
- You must first create a Top-Level BOM Table and then perform automatic balloon creation.
- Only Balloon information linked to the top level Item is mapped.
- If balloons are mapped to the Find No. field in PLM, user is responsible to prevent duplicate numbers. In case of duplicate Find No, an error message is displayed.

Solution

During Save to PLM, the Ballooning information for a particular [model + configuration] is retrieved from various views across various sheets of a drawing. The ballooning information is read-only from the views that are linked to BOM tables of type **Top-Level**.

Following points must be considered for this solution:

- The parent item [model + configuration] must either be captured in PLM or it is the default configuration of that file. The views must be for this [model + configuration].
- If multiple views are linked to same BOM table, then the BOM information from all the views is retrieved.

- However, if there exist multiple BOM tables in the drawing for the same [model + configuration], then integration will collect information only from those views that are linked to the same BOM table. That is, integration will not mix the information of views linked to different BOM tables.
Example for a particular model “A1.sldasm” and Configuration “Default”, if there are 2 BOM tables (Bom_1 and Bom_2) and 4 views (View_1, View_2, View_3 and View_4). View_1 and View_2 are linked to Bom_1 and View_3 and View_4 are linked to Bom_2, and both the BOM tables are of type “Top-Level”. In this case, the ballooning information from all views linked to BOM table “Bom_1” is retrieved. If no ballooning information is found, then the ballooning information from all views linked to BOM table “Bom_2” is retrieved. In any case, integration will not combine the ballooning information of views from two different BOM tables.
- When multiple drawings exist for the same [model + configuration], ballooning information of the first drawing is retrieved. If ballooning information is not found or if the ballooning information of the drawing is incorrect, then ballooning information is retrieved from the next drawing. When the integration finds the ballooning information in a particular drawing, it uses that for mapping the ballooning information.
- When capture design variant is not performed, the balloon-ids are retrieved in the following order:
 - If the default configuration is also used within the assembly (as Top-Level component), then the Balloon-id of default configuration is retrieved.
 - If the default configuration is not used within the assembly as a Top-Level component, then the Balloon-id of the non-default DV (that is, un-captured DV) that is in use is retrieved. For more information, see Scenarios in Balloon Mapping.
- When multiple documents are linked to same item, the Balloon-id of the “BOM Defining” document is given a priority. If both the documents are either “BOM Defining = True” or both are “BOM Defining = False”, the Balloon-id of the document with the unresolved design configuration is retrieved. However, when both the documents are using unresolved configurations or when both the documents are using the resolved configurations, then integration will pick the Balloon-id of the first one in the collection (it can be random, depending on how it is used in the assembly). See example scenarios for more details.
- When the target field in PLM is **FIND_NO**, then integration checks if the same balloon-id is specified for two different child items. If the balloon-id is specified for two different child items, then the **FIND_NO** field for the parent item is not updated (by using the ballooning data). Old values are retained for all the child items of that parent item, because the field is unique in PLM.

Assigning mapping template

Mapping template can be associated to SolidWorks files. The files is either already available in PLM or yet to be saved to PLM. To assign a mapping template, in the PLM menu select Associate Mapping Template. The CAD Integration - Set Object Attributes screen is displayed.

Click the **File** tab. Select a template from the Mapping Rules list.

Integration Preferences for Balloon Mapping

Use the **Balloon Mapping** property to control the transfer of balloon numbers from SolidWorks to the part list table in PLM.

The **Balloon Mapping** property comprises of the following options:

- [Transfer Ballooning Information](#) on page 77
- [Field in Part List Table](#) on page 77
- [Warn for duplicate Balloon IDs](#) on page 77
- [Warn for Incorrect Ballooning](#) on page 77
- [Apply Ballooning Values of non-default configuration to default configuration](#) on page 77

Transfer Ballooning Information

If this check box is selected, it is possible to map ballooning information from CAD files to PLM.

Field in Part List Table

Specify the target field in PLM Part List table to which the Balloon information must be transferred. This field name must match the database field name.

Warn for duplicate Balloon IDs

If this check box is selected, the user is alerted when two child items (of the same parent) item have the same Balloon-id.

Warn for Incorrect Ballooning

If this check box is selected, the user is alerted if the Ballooning is not properly performed for the child components (under a parent).

Apply Ballooning Values of non-default configuration to default configuration

If this check box is selected, the balloon mapping values of the non standard configuration is applied to the item of the main assembly assigned to the standard configuration.

Scenarios in Balloon Mapping

The section explains various scenarios of mapping the ballooning information from CAD to PLM.

- [Normal ballooning scenario](#) on page 78
- [When various design variants \(configurations\) are used](#) on page 83
- [Same item linked to multiple documents](#) on page 86
- [Multiple Views for the same \[model + configuration \] using the same BOM table](#) on page 88
- [Multiple BOM tables for the same \[model + configuration\]](#) on page 89
- [The Warn for Duplicate Balloon-Ids preference is enabled](#) on page 90
- [The Warn for Incorrect Balloon-Ids preference is enabled](#) on page 91
- [The Warn for Incomplete Ballooning preference is enabled](#) on page 92
- [The Apply Ballooning values of non-default configuration to default configuration preference is enabled](#) on page 92

Normal ballooning scenario

For example, consider the following

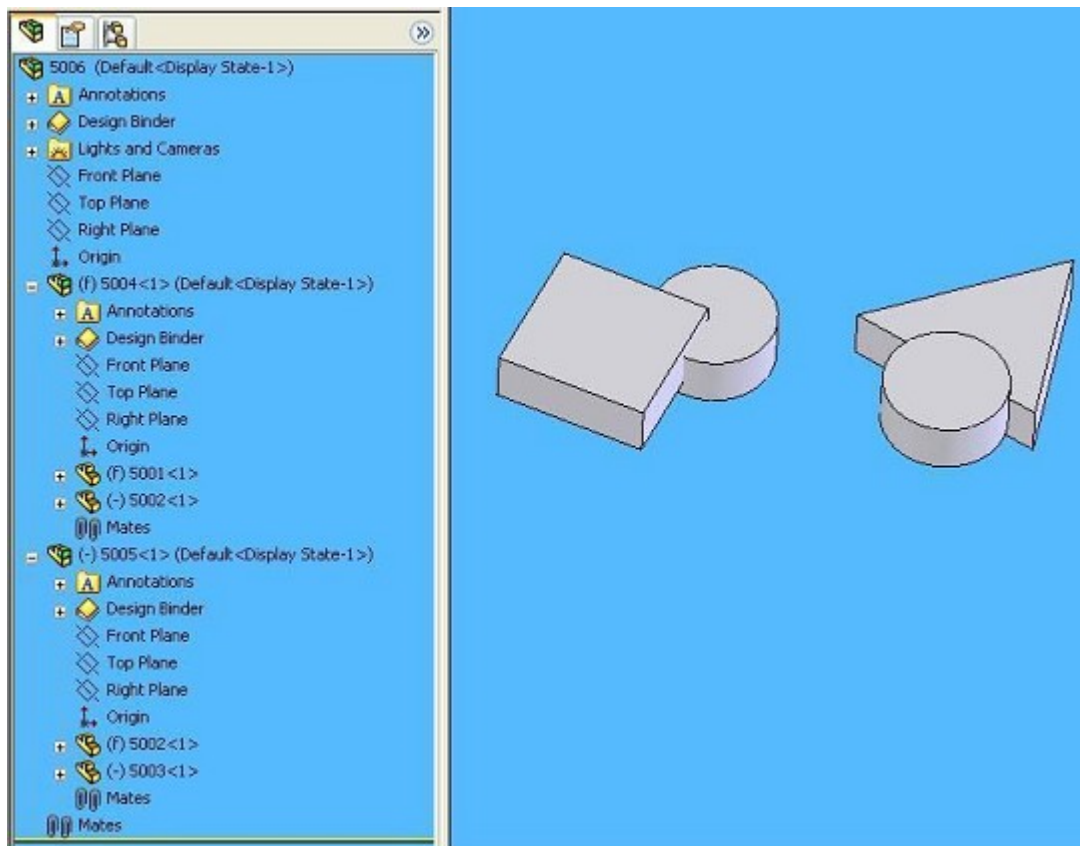
```

5006. sldasm (Configuration = Default)
|
|___ 5004.sldasm (Configuration = Default) x Quantity = 1
|   |___ 5001.sldprt (Configuration = Default) x Quantity = 1
|   |___ 5002.sldprt (Configuration = Default) x Quantity = 1
|
|___ 5005.sldasm (Configuration = Default) x Quantity = 1
|   |___ 5002.sldprt (Configuration = Default) x Quantity = 1
|   |___ 5003.sldprt (Configuration = Default) x Quantity = 1

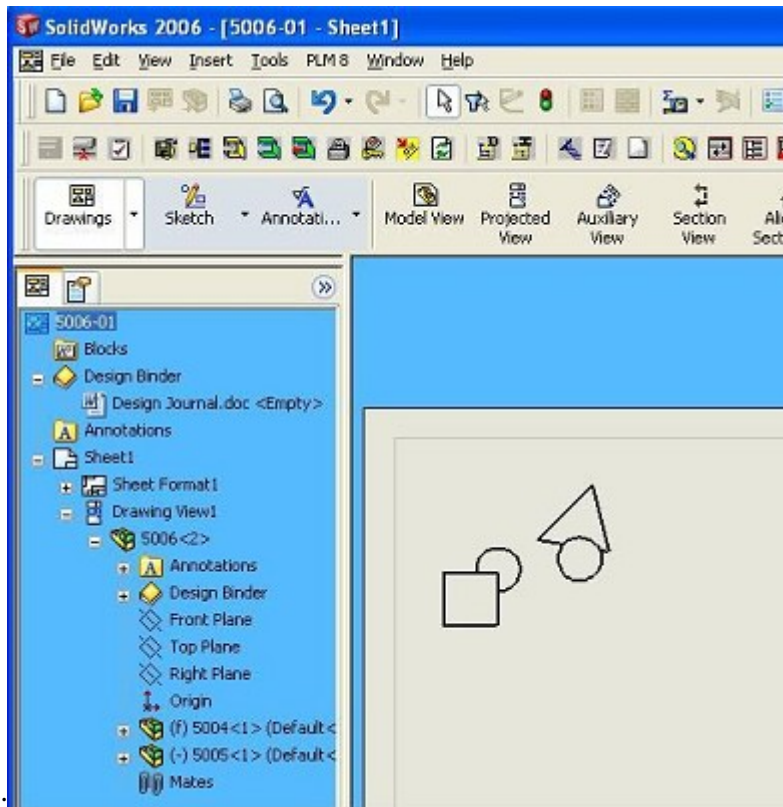
```

structure:

Parent Model	Parent Configuration	Child Model	Child Configuration	Quantity
5006.sldasm	Default	5004.sldasm	Default	1
-do-	-do-	5005.sldasm	Default	1
5004.sldasm	Default	5001.sldprt	Default	1
-do-	-do-	5002.sldprt	Default	1
5005.sldasm	Default	5002.sldprt	Default	1
-do-	-do-	5003.sldprt	Default	1

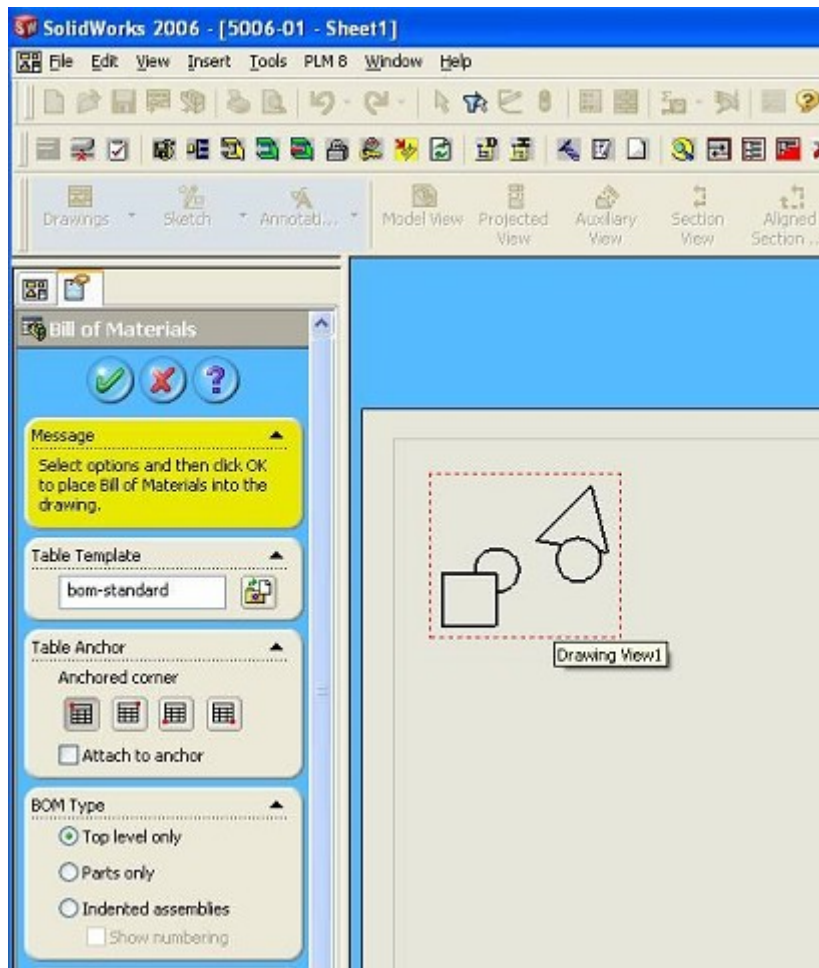


- 1 Create a drawing, Example 5006-01.slddrw for the main assembly 5006.sldasm, as shown



below:

- 2 Select a view.
- 3 To create a BOM table, click **Insert > Tables > Bill of Materials.....**

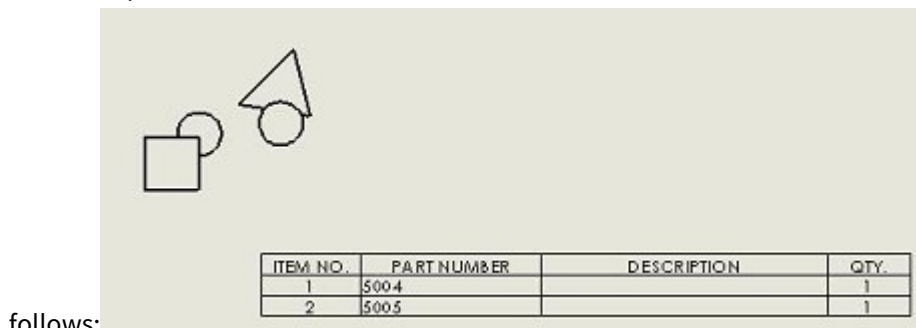


Ensure that in the **BOM Type**

group box, the **Top level** only option is selected.


4 Click **OK**

5 Click on a point to insert the BOM table; the BOM table is inserted as



follows:

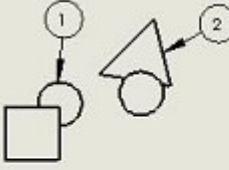
- 6 Change the column description to display the file name and configuration name, as shown below, if



ITEM NO.	SW-File Name	SW-Configuration Name	QTY.
1	5004	Default	1
2	5005	Default	1

required:


- 7 Select the view.
8 To perform auto-ballooning, click **Insert > Annotations > Auto-Balloon.....**
9 Click **OK**. The top-level components are ballooned as shown



ITEM NO.	SW-File Name	SW-Configuration Name	QTY.
1	5004	Default	1
2	5005	Default	1

below:

The properties are updated with the following



values:

- 10** .Perform the Save to PLM operation for the drawing. The target field in PLM (of the Part List table) is updated with the balloon value. In the current example, the target field is **FIND_NO**. The **FIND_NO** is

The screenshot shows a 'PLM Item' table with columns: Item, Name, Revision, Find No., Description, Status, Effective From, Effective To, CAD Instance, PLM Project, and Status. The 'Find No.' column is highlighted in red for the first two rows, indicating the target field for the update.

Item	Name	Revision	Find No.	Description	Status	Effective From	Effective To	CAD Instance	PLM Project	Status
5004	5004	0001	5004	5004	Draft	1/1/1970	1/1/1970	1/1/1970	1/1/1970	DEV
5005	5005	0001	5005	5005	Draft	1/1/1970	1/1/1970	1/1/1970	1/1/1970	DEV

updated with the balloon-ids.

To update the find numbers for the child components of the sub-assemblies, namely, 5004.sldasm and 5005.sldasm, you must create drawings, repeating the above process.

When various design variants (configurations) are used

Example consider the following model files in structure:

Model File	Configuration	Originating Item	Item-ID	Comments
5020.sldasm	Default	Yes	5020	
5011.sldprt	Default	Yes	5011	
-do-	V1	-	-	Not captured
5012.sldprt	Default	Yes	5012	
-do-	V1	-	-	Not captured
5013.sldprt	Default	Yes	5013.DEF	
-do-	V1	No	5013.V1	
5014.sldprt	Default	Yes	5014	
-do-	V1	-	-	Not captured

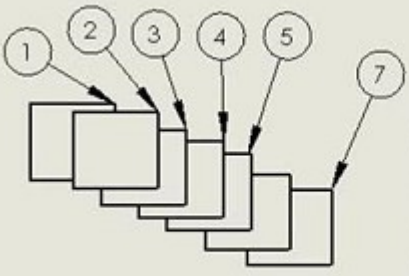
The assembly has the following structure(first-level child):

Parent Model / Configuration	Child Model	Child Configuration	Quantity	Balloon-ID	Comments
5020.sldasm (Default)	5011.sldprt	V1 (not captured)	1	1	Because V1 is not captured, and Default is not used as a first-level child within the parent, so Balloon-id of V1 will be used.

Parent Model / Configuration	Child Model	Child Configuration	Quantity	Balloon-ID	Comments
-do-	5012.sldprt	Default	1	2	Because the configuration has its own item and is used within the structure as first-level child item, so its Balloon-id will be used.
-do-	5012.sldprt	V1 (not captured)	1	3	Because V1 is not captured, and Default is used as a first-level child within the parent, so Balloon-id of this will not be considered
-do-	5013.sldprt	Default	1	4	Because this configuration has its own item and is used within the structure as first-level child item, so its Balloon-id will be used.
-do-	5013.sldprt	V1	1	5	Because this configuration has its own item and is used within the structure as first-level child item, so its Balloon-id will be used.
-do-	5014.sldprt	Default	1	-	Balloon-Id is missing (or balloon deleted).

Parent Model / Configuration	Child Model	Child Configuration	Quantity	Balloon-ID	Comments
-do-	5014.sldprt	V1 (not captured)	1	7	Because V1 is not captured, and even when Default is used. However, the Balloon-id of Default is missing, so the Balloon-id of V1 will be used.

1 Create a drawing for the assembly as shown

ITEM NO.	SW-File Name	SW-Configuration Name	QTY.
1	5011	V1	1
2	5012	Default	1
3	5012	V1	1
4	5013	Default	1
5	5013	V1	1
6	5014	Default	1
7	5014	V1	1

2 Perform the Save to PLM operation on this drawing. The item structure in PLM can be as follows:

Item	Item Revision	Description	Find No.	Status	Effective From	Effective To
5020	0001	5020		Draft	1/1/1970	12/31/9999
5011	0001	5011	0001	Draft	1/1/1970	12/31/9999
5012	0001	5012	0002	Draft	1/1/1970	12/31/9999
5013DEF	0001	5013DEF	0004	Draft	1/1/1970	12/31/9999
5013V1	0001	5013V1	0005	Draft	1/1/1970	12/31/9999
5014	0001	5014	0007	Draft	1/1/1970	12/31/9999

Note:

If a configuration has its own item, and the Balloon-id for the configuration is missing, then:

- If the target field is FIND_NO, its value will be generated automatically by the integration.
- If the target field is other than FIND_NO, its value will not be modified in PLM. Its old value, if any is retained. If this is newly getting created, then its value will be set as empty.

If the target field is FIND_NO, then its value will be generated automatically by the integration. If the target field is other than FIND_NO, then its value will not be modified in PLM. Its old value (if any will retain). If this is newly getting created, then its value will be set as empty.

Same item linked to multiple documents

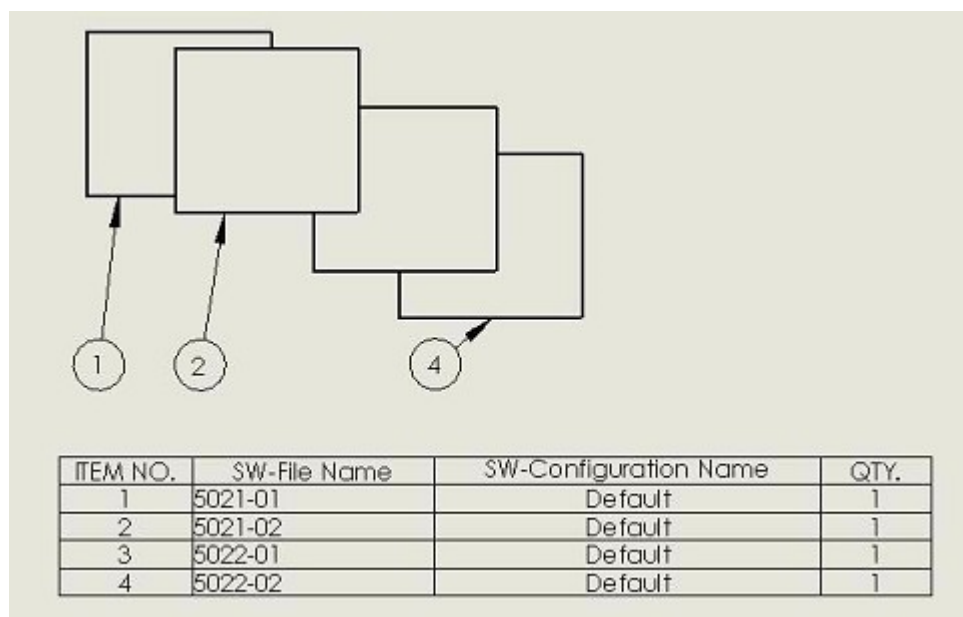
Example consider the following model files in structure:

Model File	Configuration	Originating Item	Item ID	BOM Defining Comments Document
5030.sldasm	Default	Yes	5030	Yes
5021-01.sldprt	Default	Yes	5021	Yes
5021-02.sldprt	Default	Yes	5021	No
5022-01.sldprt	Default	Yes	5022	Yes
5022-02.sldprt	Default	Yes	5022	No

The assembly has following structure (first-level child), with the ballooning done below:

Parent Model	Child Model	Child Configuration	Quantity	Balloon ID	Comments
5030.sldasm	5021-01.sldprt	Default	1	1	Because this configuration has its own item and is used within the structure as first-level child item, the Balloon-id will be used.

Parent Model	Child Model	Child Configuration	Quantity	Balloon ID	Comments
-do-	5021-02.sldprt	Default	1	2	Since this document is not BOM Defining and the associated item is also linked to another document which is BOM defining, the Balloon-id of that document will be used.
-do-	5022-01.sldprt	Default	1	-	Balloon-Id is missing (or balloon is deleted).
-do-	5022-02.sldprt	Default	1	4	Since the balloon-id of the BOM Defining document is missing, the Balloon-id of this document will be used.



When you perform the Save to PLM operation, the item structure in PLM can be as

follows:

Menu

Recently Used

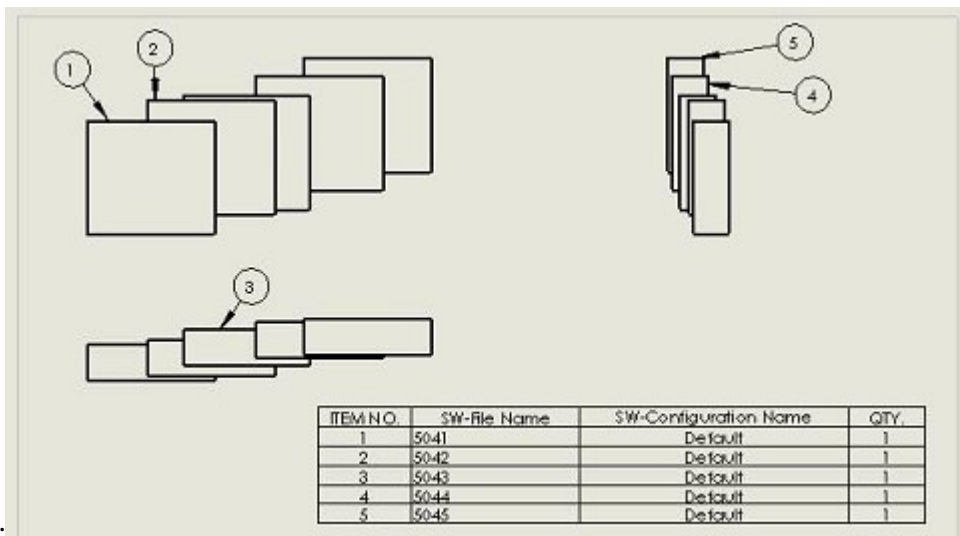
Item Structure Workbench

(5030, 0001, 10/25/2021)

		Item	Item Revision	Find No.	Description	Quantity	Status
		(a)	→a	→a	→a	=	=
	<input type="checkbox"/>	5030	0001		5030		Draft
	<input type="checkbox"/>	5022	0001	0004	5022	1.0000	Draft
	<input type="checkbox"/>	5021	0001	0001	5021	1.0000	Draft

Multiple Views for the same [model + configuration] using the same BOM table

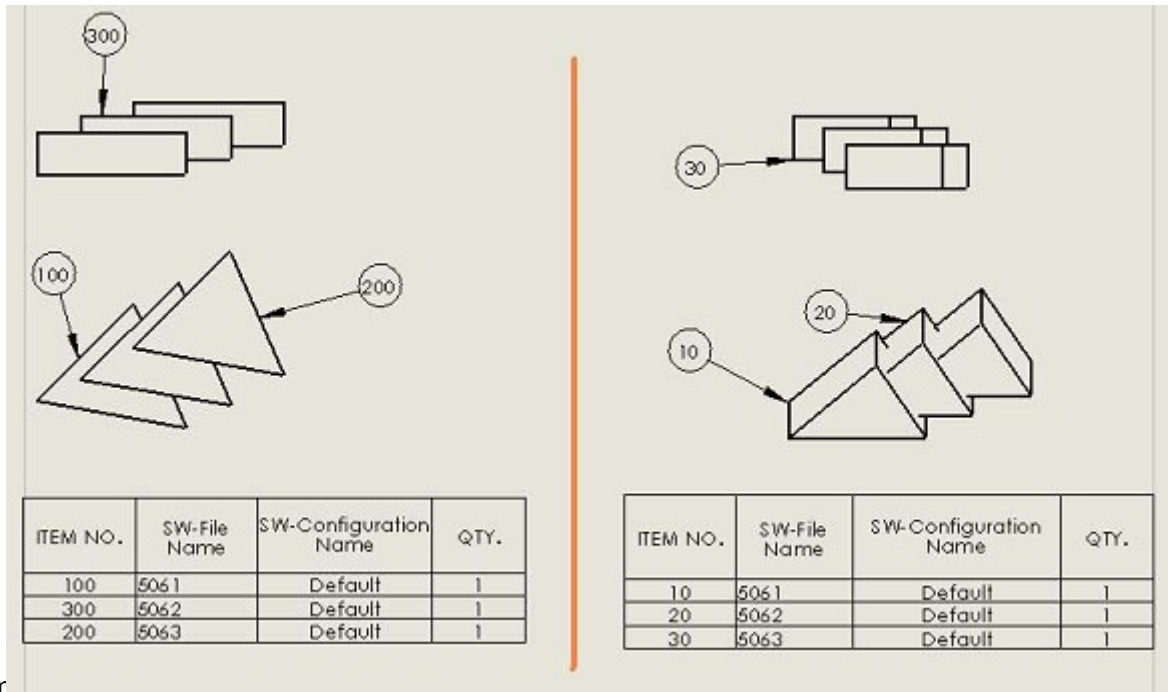
In the following example, there are three views for the assembly. Ballooning is performed for the views using the same BOM table, as shown



below:

Multiple BOM tables for the same [model + configuration]

In the following example, there are four views and two BOM tables. The two views on left refer to the BOM table on the left-bottom, and the two views on the right refer to the BOM table on the



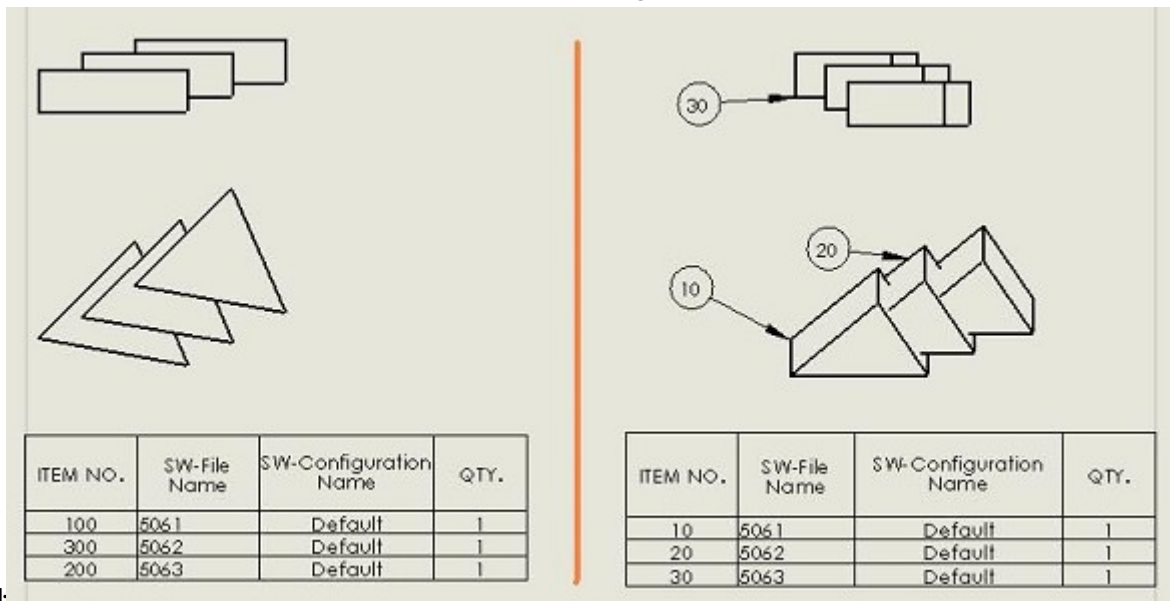
right-bottom

When you perform Save to PLM, views that are linked to first BOM table are checked first. If a view linked to the first BOM table exists, the ballooning information from all the views linked to the BOM table is passed to the integration. The views that are linked to the second BOM table are not checked. The results are as

Item Structure Workbench						
(5070, 0001, 10/25/2021)						
	Item	Item Revision	Description	Find No.	Quantity	Status
<input checked="" type="checkbox"/>	5070	0001	5070			Draft
<input type="checkbox"/>	5063	0001	5063	0300	1.0000	Draft
<input type="checkbox"/>	5062	0001	5062	0200	1.0000	Draft
<input type="checkbox"/>	5061	0001	5061	0100	1.0000	Draft

below:

If the ballooning is not performed for any of the views using the first BOM table, the information from the views linked to second BOM table is retrieved. In the following screen, the balloons of the first view is



deleted:

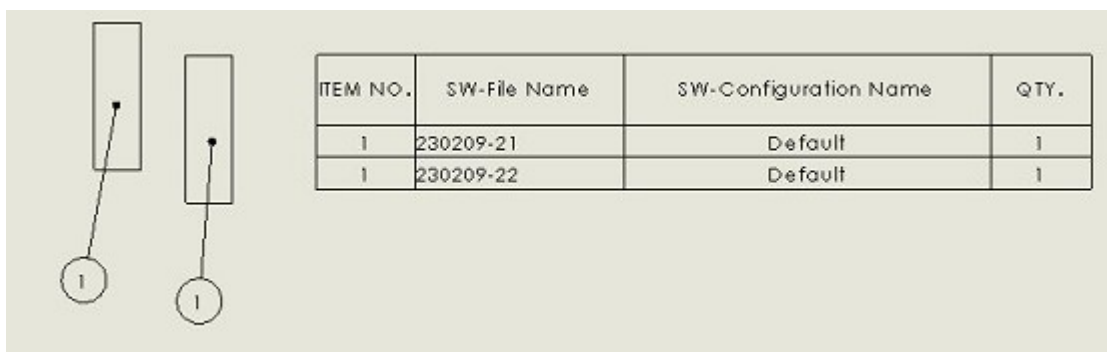
The result after Save to PLM can be as

Item Structure Workbench						
(5070, 0001, 10/25/2021)						
	Item	Item Revision	Description	Find No.	Quantity	Status
<input checked="" type="checkbox"/>	5070	0001	5070			Draft
<input type="checkbox"/>	5063	0001	5063	0030	1.0000	Draft
<input type="checkbox"/>	5062	0001	5062	0020	1.0000	Draft
<input type="checkbox"/>	5061	0001	5061	0010	1.0000	Draft

follows:

The Warn for Duplicate Balloon-Ids preference is enabled

In the following example, the same balloon-id is specified for two component files and both the component files are linked to the same



item.

If the Warn for duplicate balloon-ids preference is enabled, the following dialog box is

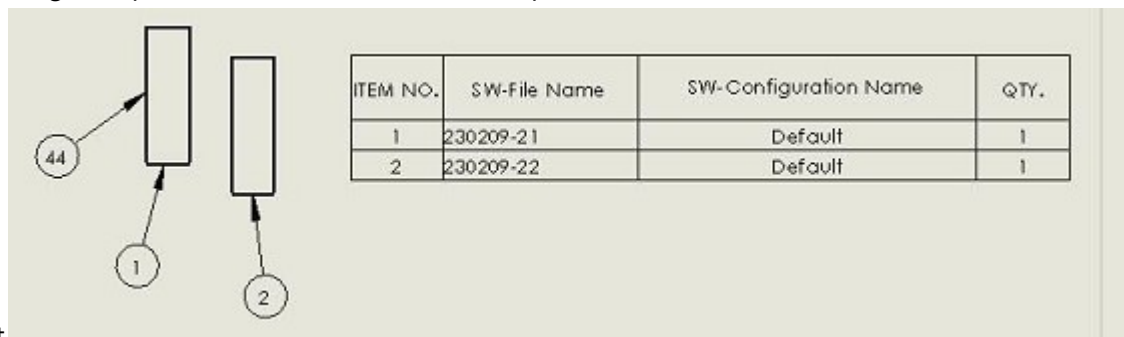


displayed: #77123

This dialog box is displayed only once for a particular parent item (of a model file). It is not repeated, even if the multiple child items have this problem. This information is used for updating the Part List table field (in PLM). If the target field is FIND_NO, ballooning information of the child item of this parent is not updated. Effectively, BOM is updated but not the FIND_NO field.

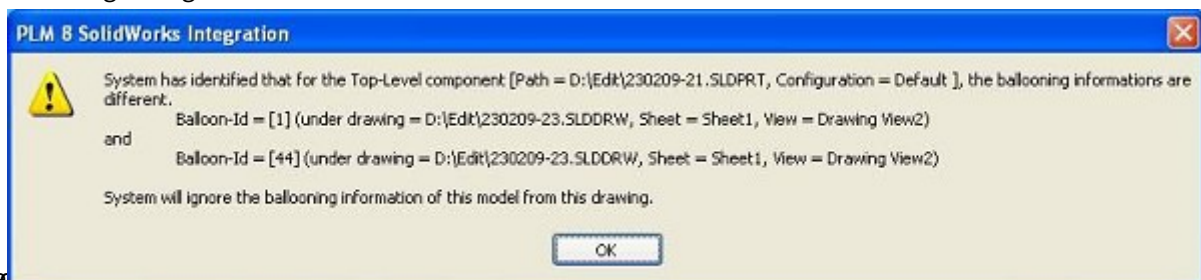
The Warn for Incorrect Balloon-Ids preference is enabled

In the following example, two different balloon-ids are specified for the same child



component.

The following dialog box is

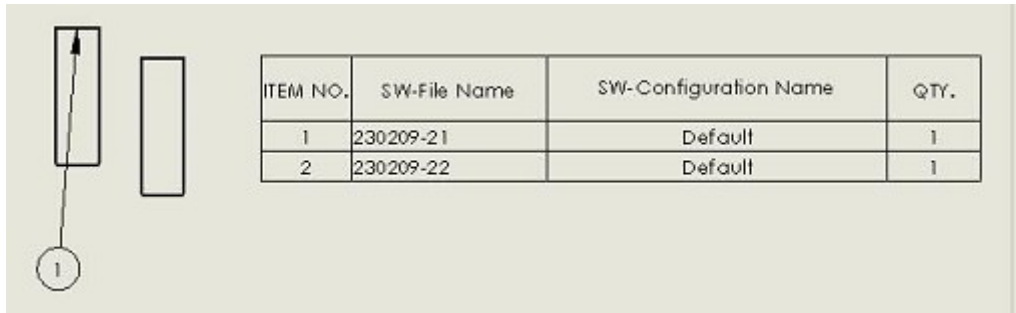


displayed.

Note: The process is not stopped, even though a warning message is displayed.

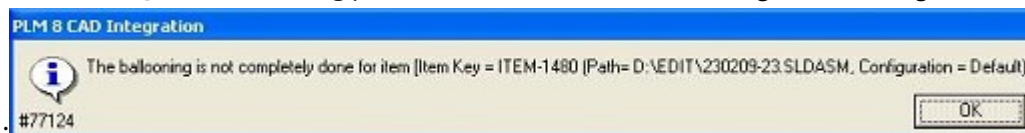
The Warn for Incomplete Ballooning preference is enabled

In the following example, the ballooning information does not exist for the second child



model.

If the **Warn for Incomplete Ballooning** preference is enabled, the following error message is



displayed:

This message is indicative, the ballooning information is updated in the PLM. However, the information is incomplete.

The Apply Ballooning values of non-default configuration to default configuration preference is enabled

Consider a scenario wherein all the following conditions are met:

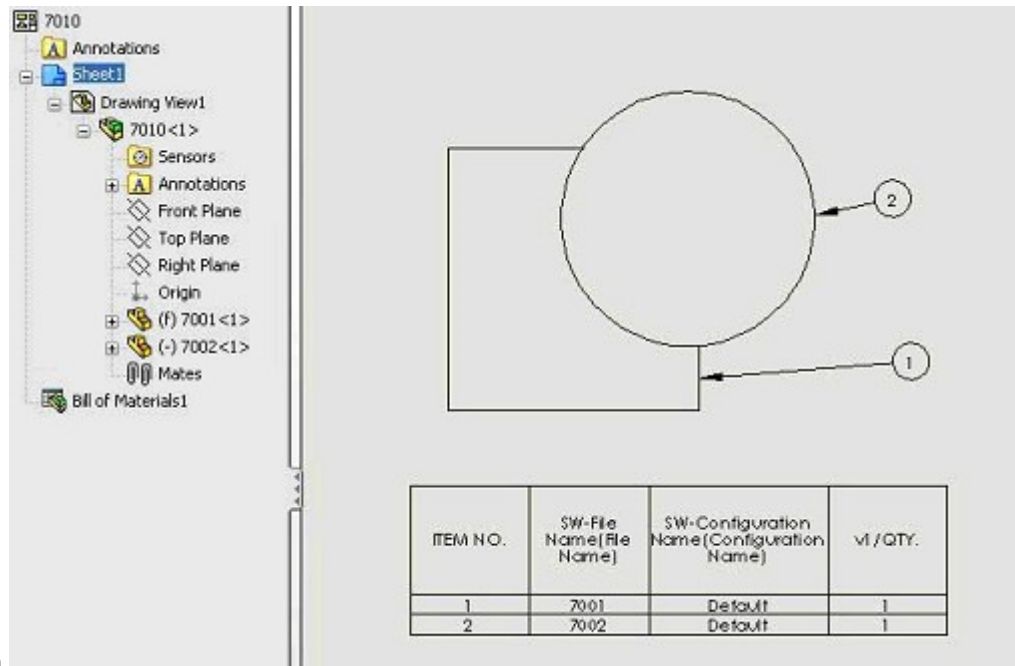
- The Apply Ballooning values of non-default configuration to default configuration preference is enabled.
- The assembly contains the multiple configurations.
- Only one configuration(default) contains the item.
- The ballooning information doesn't exist for the default configuration.

In the above scenario, the ballooning information of the other configuration(s) is applied to the default configuration. When the Apply Ballooning values of non-default configuration to default configuration preference is disabled, the default value is Yes.

For this example, consider the following file details:

- 7010.slddrw (contains view of the V1 configuration of the 7010.sldasm assembly)
- 7010.sldasm (Configurations: Default, V1)
- 7001.sldprt (Configuration: Default)
- 7002.sldprt (Configuration: Default)
- The 7010.slddrw drawing contains view of the V1 configuration of the 7010.sldasm assembly.
- The 7010.sldasm assembly contains only one item (for the default configuration, Default) and the configuration V1 is not captured.

There exists no view that contains the ballooning information of the configuration, Default. Therefore, the ballooning information of other configurations of this file are considered. Since there is ballooning information for configuration V1 of this file, so system will pick the Ballooning information of V1 and will use/apply the same for the Default



configuration.

After the Save to PLM operation is performed, the BOM information in PLM is displayed

Structure Workspace (Structure - 7010, 0001)

DB Filter ☐

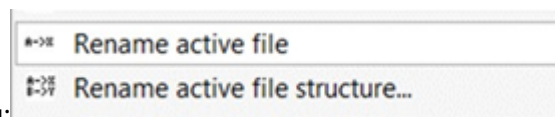
Item ID	Description	Find No.	Quantity	Item Type	Revision	Status	Item Character
7010	7010			MANUFACTURE	0001	UNDEFINED	Engineering It
7002	7002	2	1.00	MANUFACTURE	0001	UNDEFINED	Engineering It
7001	7001	1	1.00	MANUFACTURE	0001	UNDEFINED	Engineering It

Chapter 27: Renaming of SolidWorks Files

Renaming option enables you to rename the SolidWorks files via PLM integration menu in SolidWorks application.

To access the rename option you must connect to PLM from SolidWorks. Select PLM in SolidWorks menu and click Rename. Following are the two options to rename files:

- Rename - Active file
- Rename - Active file structure



Following figure displays rename menu:

Following figure shows Rename option in the



The renaming options performs following operations:

- Renames the files in local folder.
- Renames the files in PLM server.
- Corrects the referenced file names under parent/referenced files and uploading of files into PLM server.

Rename - Active File

If you click **Rename > Active File**, integration checks whether the active file exists in PLM or not. If the active file exists in PLM, then following steps are performed:

- 1 Click **Rename > Active File**. The Save As dialog box in SolidWorks opens. Specify the new filename in the Save As dialog box and click **OK**. SolidWorks renames the file locally and replaces the references of this file under any file that is loaded in application session.
- 2 Integration corrects the references for those files (under the structure of currently open files) that are not loaded in session but has direct reference to the currently renamed file.
- 3 Active file is renamed in PLM. Integration sets the data set field Orig. file name (FILE.ORIG_FILE_NAME) with the value of its original filename.
- 4 The file name of the Neutral files (if any) for this renamed file is also corrected according to new names.

- 5 Since the active file has been modified via application, it is uploaded in PLM.
- 6 The parent files (whose references were corrected or got corrected above and which exist in PLM) are uploaded in PLM.
- 7 Results of renaming active file is displayed.

Rename - Active File Structure

If you click **Rename > Active File Structure**, the following steps are performed:

- 1 Click **Rename > Active File Structure**. The Rename Files dialog box opens. This contains list of all files (under the structure of currently active file which are allowed for renaming) wherein you can specify the new filename for one or more files.
- 2 Click **OK**. Integration prepares a list of all files under the structure of currently open files which have direct reference to the files that are getting renamed.
- 3 Integration closes the files that are open.
- 4 Files that are marked for renaming are first renamed locally (at O/S level) to their new filenames.
- 5 Referenced file names is corrected on the files that have direct reference to these renamed files.
- 6 Files names are updated in PLM with the new filenames. Integration also sets the data set field Orig. file name (FILE.ORIG_FILE_NAME) with the value of its original filename of these files.
- 7 All the files whose references have been modified (and which exist in PLM) are uploaded into PLM.
- 8 File name of the Neutral files (if any) for these renamed files is also corrected according to new names.
- 9 File that was originally active is re-opened
- 10 Results of renaming is displayed.

Corrections of file references during Edit file:

There can be certain files in PLM whose referenced files were renamed, but their referenced paths were not corrected. This is handled when you Edit/View file operation. During Edit file, it checks if any of the files references were renamed. If it is renamed it corrects the referenced filenames for those files and uploads these files into PLM.

Rename Files - Preference

Renaming of active file in PLM using SolidWorks menu **File > Save As**:

If the integration preference Rename file in PLM on **File > Save As** operation is set to **True**, and when you perform **File > Save As** operation on active file, integration will perform the same set of operations as **Rename > Active file**.

Note:

- Before running these operations, you must set the following SolidWorks option to True.
- Only those files which exist in PLM are considered for renaming and/or uploading into PLM.

- If the preference Allow operations on previous revisions is set to true; then during these menu operations, integration allows you to rename and/or correct the file references of previous revision of files also. By default, integration allows to perform rename and upload of latest revisions of files. Note that during Edit/View file in Integration, integration corrects the references and upload the file even if the file is of previous revision irrespective of this preference value.
- During these operations, integration renames only the current version of the file which exists locally. Other versions of the files are not renamed, they retain their original names.
- After renaming the files, ensure that the referenced file names are corrected in all the files that have references to this file. Do not rename a file multiple times unless and until the referenced filenames are corrected in all the files that have references to this file.

Chapter 28: Integration Preferences

The PLM integration preferences control the way the PLM integration for SolidWorks works.

Users locally manage integration properties for which they have been authorized by the administrator. The local integration properties are specific to each user. Your settings do not affect the properties of other users.

Users manage their local integration properties in the PLM Preferences pane. To access the Web PLM Preferences pane:

- 1** In SolidWorks, select the PLM menu.
- 2** From the PLM menu, select Preferences or in the toolbar, click Preferences. The PLM Preferences screen is displayed.

The PLM Preferences screen is displayed.

The PLM Preferences screen includes these tabs:

- SolidWorks
- Toolkit

Chapter 29: SolidWorks Tab

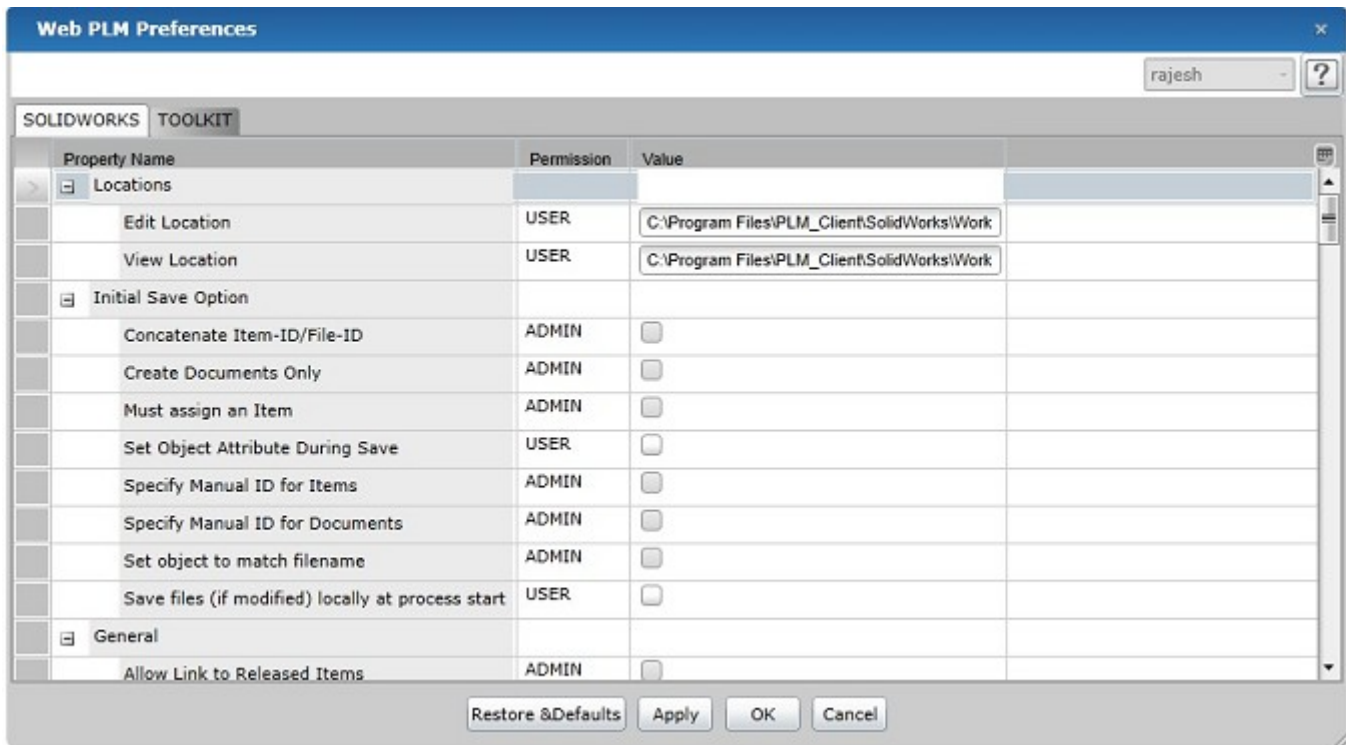
The SolidWorks tab includes properties that control the behavior of the PLM integration for SolidWorks. This tab includes the following options:

- [Locations](#) on page 98
- [Initial Save Option](#) on page 99
- [General Option](#) on page 102
- [Save Neutral Files Option](#) on page 67
- [Post Save Process](#) on page 107
- [After Check In Option](#) on page 110
- [Attach to Workflow](#) on page 108
- [Troubleshooting Option](#) on page 110
- [Integration Preferences for Balloon Mapping](#) on page 77
- [Integration Preferences for Thumbnails](#) on page 72
- [Integration Preferences to Save Changes to PLM](#) on page 55
- [Set Item-ID during Initial Save](#) on page 110
- [Renaming of SolidWorks Files](#) on page 94

Locations

To carry out various commands, such as **View File in Integration** or **Edit File** in Integration from the Results panel of the integration query, PLM must locally download the files that you want to view or edit in SolidWorks. In the Locations section you can specify the folders in which PLM must locally download the files that you want to view or edit in SolidWorks. If required, you can specify the same folder for both viewing and editing,

but if you do this, the **Disable Actions on view files** property will not



Initial Save Option

The properties comprising the **Initial Save Option** determine how files behave when saved to PLM for the first time. The following properties are

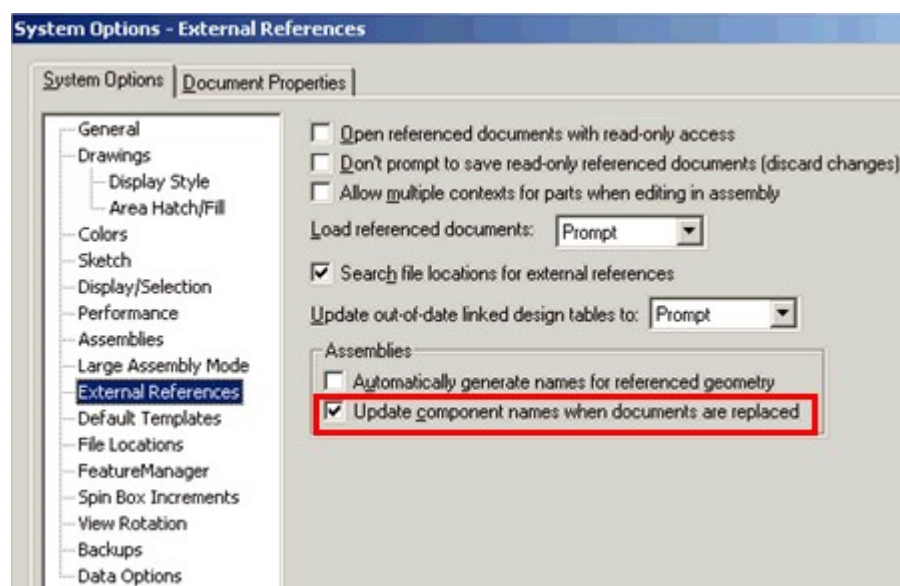
Initial Save Option		
Concatenate Item-ID/File-ID	ADMIN	<input type="checkbox"/>
Create Documents Only	ADMIN	<input type="checkbox"/>
Must assign an Item	ADMIN	<input type="checkbox"/>
Set Object Attribute During Save	USER	<input type="checkbox"/>
Specify Manual ID for Items	ADMIN	<input type="checkbox"/>
Specify Manual ID for Documents	ADMIN	<input type="checkbox"/>
Set object to match filename	ADMIN	<input type="checkbox"/>
Save files (if modified) locally at process start	USER	<input type="checkbox"/>

available:

Concatenate Item/ File ID

If this check box is selected, SolidWorks files are renamed during the first save to PLM. Drawing files are renamed by adding the related PLM file ID as a prefix to the name of the file being saved. For all other files, the related PLM Item ID is used. This means that users do not have to worry about using duplicate file names, because this feature ensures that the name of SolidWorks files saved to PLM are always unique.

Note: To use the Concatenate Item/File-ID option, in the System Options - External References dialog box in SolidWorks, the Update component names when documents are replaced check box must be selected, as shown in the following picture.



In SolidWorks, to access this dialog box, select **Tools > Options > External references**.

If the Concatenate Item/File-ID to all files on first save to PLM check box is selected, but the SolidWorks Update component names when documents are replaced check box is not selected, the PLM integration will automatically select this check box. If this check box has been selected by the PLM integration, it will be cleared when the user logs out from PLM.

In addition, to open renamed files from PLM, make sure that the relative search path for file locations is set correctly in SolidWorks. For further information, see Concatenate Item/File-ID.

Create documents only

If this check box is selected, when you save a file to PLM, a document is created in PLM, but no items are created. For further information, see Saving to PLM.

Must assign an item

If this check box is selected, when you save a file to PLM, you must assign an item to the objects. PLM does not allow you to proceed without assigning an item.

Set Object Attributes During Save

If this check box is selected, you can define attributes for the files, documents and items that are being saved to PLM. For further information, see [Saving to PLM](#).

Manual ID entry for documents and/or items

When Manual ID check box is selected in the preferences, you can specify IDs for the Item and

	Specify Manual ID for Items	ADMIN	<input type="checkbox"/>
Documents.	Specify Manual ID for Documents	ADMIN	<input type="checkbox"/>

Set object to match file name

If this check box is selected, the integration creates the Item ID, Document ID, Filename, and File ID based on the same identifier. When user saves a file with a specific name, it is assumed that this filename (without the extension) is the base identifier. When the file is saved to PLM, the integrations sets the IDs for all components based on that filename.

Set Object to Match Filename preference preference is set

Set Attributes Dialog During Save preference is not set

Limitations

Following is the known limitation when you set objects to match file name:

- If you modify the Item-Id, specified in the Set Attributes dialog for multiple times, the count in the item revisions increases. It is recommended to specify the Item-ID only once in the dialog.

Note: It is recommended not to specify the same Item-Id for two files in the Set Attributes dialog box.

General Option

The General Option includes the following properties:

- [Allow Link to Released Item](#) on page 102
- [File Name Uniqueness](#) on page 102
- [Disable Actions on View File](#) on page 103
- [Query Search Default](#) on page 103
- [Save Additional Drawings](#) on page 103
- [Download Additional Drawings](#) on page 103
- [Allow reload of already opened files](#) on page 103
- [Show Message on Successful Open](#) on page 104
- [Warn User when the same item is linked to parent and child components](#) on page 104
- [Take Ownership During Edit File](#) on page 104
- [Skip Mapping for Released Files during Download](#) on page 104
- [Skip Meta Data Comparison During Download](#) on page 105
- [Warn for Missing CheckIn Specific Modified Mapping XML File](#) on page 105
- [Set CAD Instance Name as Item-ID](#) on page 105
- [Generate Items for all Configurations](#) on page 105
- [Save External References](#) on page 105
- [Resolve light-weight components during Save to PLM](#) on page 106
- [Create Separate BOM lines for component files linked to Same item](#) on page 106
- [Update items with ERP Item Default Data automatically](#) on page 106
- [Link Drawing to all Items of Child Model](#) on page 106
- [Process Impacted Documents during download](#) on page 107
- [Indicator for Locally Changed Files in Download Manager dialog](#) on page 107

Allow Link to Released Item

Your Concept

File Name Uniqueness

If this check box is selected, the names of the files stored in PLM are unique.

If this check box is cleared, you can store multiple files with identical file names in PLM but in different projects. It is not recommended to clear this check box, since it might cause problems if you want to

download a file while a file with an identical name already exists locally. For example, you cannot use multiple files which have identical file names ; from different folders ; in one assembly.

Disable Actions on View File

This option is used to prevent the user from modifying files that he opened from the integration query results using the View File in Integration option. If this check box is selected and the user clicks a PLM menu option, for example, Save to PLM, a message appears informing him that the file is read-only.

Note that for the **Disable Actions on View File** option to work, the view and edit locations specified in the Locations option must be different. For further information, see [Locations](#) on page 98.

Query Search Default

This option defines how the integration query tool is opened from the PLM menu in SolidWorks and determines the query objects. The following options are available:

- ITEM
- DOCUMENT
- FILE

Save Additional Drawings

If this check box is selected, the following operations save the local editable drawing files of all the editable and/or non- editable models of the current CAD structure to PLM:

- Save to PLM
- Save and Unlock
- Check-In

Download Additional Drawings

If this check box is selected, the **Edit/View file in Integration** option enables the user to download the related drawings of all the models in the current Document structure.

Allow reload of already opened files

If this check box is selected, you can replace and reload open files. In the Download Files dialog box, **Reload Already Open Files** check box is included.

To replaces the local file with the files in the PLM, click **OK**. The replaced are automatically opened.

By default, the **Reload Already Open Files** check box is selected, based on the setting of the **Allow reload of already opened files** field in the Integration Properties for Integration dialog box.

When you perform reload operation on drawing files, the drawing files are closed, re-placed, and re-opened, as a result of a SolidWorks limitation.

It is recommended that user save the open documents locally before executing the Edit/View File in Integration process.

Download Files - Commands:

Progress Bar

The progress bar indicates the status of downloading of the files from the PLM to the local system.

Select the **Skip Mapping for Released Files** check box to skip the To-CAD mapping for the files that are in Released status in PLM. The default value for this check box is defaulted based on the integration preference Skip Mapping for RELEASED files during Download. You can select the check box to download large assemblies in order to improve the performance of the download operation.

Refresh Already Open Files

Synchronizes the currently open file with the latest file in PLM.

Restore Archived Files

Retrieves the file from the archive area.

Show Message on Successful Open

If this check box is selected, a message is displayed after downloading (Edit/View file) files from PLM.

Warn User when the same item is linked to parent and child components

If you select the **Warn User when the same item is linked to parent and child components** checkbox, PLM warns the user that the same item is linked to parent and child components.

Take Ownership During Edit File

When **Take Ownership during Edit File** preference check box is not selected, the users do not take the ownership of the files even when they execute **Edit File** command.

Skip Mapping for Released Files during Download

When you select the preference **Skip Mapping for Released Files during Download**, the integration does not perform the mapping for the files which are in **Released** status in the PLM during the download process.

It is recommended to select the check box to improve the download performance of large assemblies in the View/Edit File operation. However, you should be aware that some file preferences may not be up to date with PLM.

Additionally, it is also possible to select the **Skip Mapping for Released Files**.

The default value for this check box is defaulted based on the integration preference **Skip Mapping for Released Files during the download**.

Skip Meta Data Comparison During Download

When **Skip Meta Data Comparison During Download** check box is not selected in the preferences. The Download Manager indicates if there is any change in the PLM data of Document/ Item/File.

If the Download Manager indicates the data change, it is recommended to download the indicated files. Hence the **Download** option for the specific changed file is set to true by default.

Warn for Missing CheckIn Specific Modified Mapping XML File

If this check box is selected, and if the `ModifiedMappingRulesForCheckIn.xml` is not available in the `%CFE_CLIENT_HOME%` Toolkit directory, the PLM warns the user that the file is missing.

Set CAD Instance Name as Item-ID

If you select the **Set CAD Instance Name as Item-ID** check box, PLM sets the name of the CAD instance as the item ID. For more information, see [Set CAD Instance Name as Item-ID](#) on page 111.

Generate Items for all Configurations

If this check box is selected, items for all the configurations are generated and linked to document in PLM when you perform the Save to PLM operation, regardless whether Capture Design Variants steps are performed or not.

Save External References

If this check box is selected, during the Save To PLM operation, the relations between the files SW files are stored in PLM. For more information, refer to [Associative Links](#) on page 47.

Resolve light-weight components during Save to PLM

SolidWorks supports a mode for assembly components called lightweight. A lightweight component contains only a small subset of its data for display purposes, which increases system performance when loading assemblies. The extra data is not loaded until it is required.

If this check box is selected, while reading BOM of individual configurations, PLM integration activates the desired configuration and initiates this API to resolve all lightweight components during save to PLM.

Note: In case of large assemblies, this API can take a lot of time and also the memory limit of Process can exceed (specific to O/S) and result in exception/ application crash. In such case you can clear this checkbox.

Create Separate BOM lines for component files linked to Same item

If this check box is selected, when the two different components in an assembly are linked to the same item, then Save To PLM operation will create two different BOM lines in the PLM.

Update items with ERP Item Default Data automatically

If this check box is selected, and **Create Documents Only** check box is not selected integration sets the values of certain attributes based on ERP default's data (for the combination of **ITEM_TYPE** & **ITEM_GROUP**). This preference is applicable for the items that are created for the file during Save to PLM.

For example, Select the **Update items with ERP Item Default Data automatically** checkbox and do not select the **Create Documents Only** checkbox in the **SOLIDWORKS Preferences > General**.

Create new part and perform Save to PLM.

Before the Set Object Attributes dialog is displayed, the integration sets the attributes of Items based on Item default data as defined in PLM.

Link Drawing to all Items of Child Model

If this check box is selected, the integration links (as Related Models) all the Items representing different configurations of the model to the Document of the drawing.

If this preference is not selected, only those items for which drawing view is created will be linked to the document of the drawing as Related Models.

Process Impacted Documents during download

If this check box is selected, then when a file is downloaded, all of its impacted documents (for which the file is a driver document) will also be downloaded. This preference is applicable for documents that are linked with each other as Driver Document/ Impacted Document.

For example, create a part file (P1.sldprt) and a mirror part (MP.sldprt). Select the preference **Save External References and Process Impacted Documents during download**. Perform Save to PLM.

The document of P1.sldprt is linked with the document of MP.sldprt as Driver Document and the document of MP.sldprt is linked with the document of P1.sldprt as impacted document

Select the document of P1.sldprt and perform Edit file, the file MP.sldprt will also be considered for downloading.

Indicator for Locally Changed Files in Download Manager dialog

The value Y for the Identical field of the PLM - Download Files dialog box distinguishes between the files that are changed in PLM and locally by the user.

Following are the possible values for the Identical field:

- Y: Local file is identical to the file in PLM.
- N: The file in PLM is different from the file on the local system.
- Y – Locally Changed: Indicates the file in PLM is not changed, but there are some changes made to the file on the local system.

Post Save Process

CheckIn Cleanup

The available options are:

Delete File

After check-in, ONLY the root file will be deleted from local system. All other files(within the structure) will be read-only.

Make files read-only

After check-in, all the files (within the structure, including the root file)will be made read-only in the local system.

Save and Unlock Cleanup

The available options are:

Delete File

After save and unlock, ONLY the root file will be deleted from local system. All other files (within the structure) will be read-only.

Make files read-only

After save and unlock, all the files (within the structure, including the root file) will be made read-only in the local system.

Show message on successful save

If this preference is selected, after Save to PLM/Save and Unlock/Check-In actions are successful, a dialog is displayed to indicate that the process is successful.

Attach to Workflow

Use the Attached to Workflow property to control the Dispatch to Business Process functionality. The Attached to Workflow property comprises of the following options:

- [Attached to Business Process](#) on page 108
- [Attached related Models\Drawings](#) on page 109
- [Attach all related objects](#) on page 109
- [Action for objects locked by Business Process](#) on page 109
- [Attach Draft Objects Only](#) on page 109
- [Attach All Items](#) on page 109

Attached to Business Process

This option controls the PLM objects of a SolidWorks file that can be attached to the business process. Allowed values:

Documents Only

Only the documents associated with the SolidWorks file are attached to the business process.

Items Only

Only the items associated with the SolidWorks file are attached to the business process.

Both

Both the documents and the items associated with the SolidWorks file are attached to the business process.

Attached related Models\Drawings

If this check box is selected, the PLM objects(items, documents) of the models or drawings associated with the SolidWorks file are attached to the business process. The PLM objects attached to the business process are based on the values selected for the **Attached to Business Process** preference.

Attach all related objects

If this check box is selected, the PLM objects (items, documents) of all the components in CAD structure of a Solidworks file are attached to the business process. The PLM objects attached to the business process are based on the values selected for the Attached to Business Process Preference.

Action for objects locked by Business Process

This preference indicates what action must be taken if one or more objects are already locked by another business process. The available options are:

Attached as unlocked

The object will be attached to Business process and unlocked.

Do not attach

The objects that are locked will not be considered for dispatch.

Cancel action

The dispatch operation will be cancelled.

Attach Draft Objects Only

This preference is applicable for both Documents and Item objects. If this preference is selected, only Draft objects (ITEMS/ DOCUMENTS) can be attached. If this checkbox is not selected, no check is made with respect to object status.

Attach All Items

If this preference is selected all the Items linked to the document are considered during dispatch. If this preference is not selected, then only default item is considered for dispatch.

Troubleshooting Option

Use this option to specify whether log files must be created for various processes of the PLM integration for SolidWorks. You are recommended to create these log files if you are experiencing problems with the integration, and, if required, to send the log files to the PLM support group.

Set Item-ID during Initial Save

Set Item-ID based on

The available options are:

- None
- File Name
- Part Number
- Custom Property [Document Level]

If the **Set Item-ID based on** is set to **File Name** and **Create Documents Only** preference is not selected, during save to PLM, integration sets the Item-ID matching to **File Name**.

If the **Set Item-ID based on** is set to **Custom Property [Document Level]**, you must set the following preferences:

- Specify the **Custom property to use as Item-ID**. For example, specify CAD Item Number. This can be the name of any custom property that is defined in the CAD file.
- The **Create Documents Only** preference must not be selected.

Create a part file. Create a custom property and specify the name as defined in Custom property to use as Item-ID preference. For example, CAD Item Number. Specify a value for the property. For example, specify

	Property Name	Type	Value / Text Expression	Evaluated Value
1	CAD Item Number	Text	1700.11	1700.11
2	<Type a new property>			

Save the file to PLM. For the new item that is getting created, integration sets the Item-ID in the Set Object Attributes window as the value specified in custom property.

After Check In Option

Use the properties of the After Check In Option to define what happens to a local SolidWorks file after it has been copied to the PLM vault during the check-in procedure.

The available properties are:

Make Files Read-Only

The files are left in the SolidWorks directory, but cannot be updated.

Delete File

The files only exist in the PLM vault. To work on them in SolidWorks, you must retrieve them from PLM using the **Edit File** or **View File** commands and check out the file either before or after the retrieval. For further information, see [Checking out a file](#).

Note that if the **Deleted** option is selected and the local file that is deleted belongs to an assembly, the following applies:

- If the file that is checked in is the root of an assembly, the integration only deletes the local copy of the root object that is checked in. The integration deletes the root object without checking whether the root object is used in another assembly.
- No local versions of the sub-assemblies or components of the checked in root object are deleted, because the user may be working with other assemblies that use the same components, which therefore must remain on the local machine.

To define the action to be performed on the local SolidWorks files, select the required radio button and click **OK**.

Set CAD Instance Name as Item-ID

If this check box is selected, the integration uses the following naming conventions in creating the new Item-Ids for the configurations of the CAD file.

- If there is a User-specified name for that Configuration in the CAD file, then the User-specified name is taken as the basis of Item-ID (for creating the Item).
- If there is no User-specified name for the Configuration, then the integration uses the configuration name as the basis of Item-ID (for creating the Item).

The Item-ID is automatically selected when the following PLM 8 menu actions are performed:

- [Saving Files to PLM](#) on page 18
- [Design Variants](#) on page 44
- [Link to Item](#) on page 24

Toolkit Tab - Introduction

The **Toolkit** tab includes properties and definitions that can be shared with other integrations for PLM. This tab includes the following options:

- General Option
- Toolkit Extensions to Original Application

Synchronize All Files During Save

If this check box is selected, PLM checks whether the file header is consistent with the data in stored in PLM. Clearing this check box can save time during the save process.

Ignore Items

If this check box is selected, there will be no actions related to items from the integration.

Disable BOM Creation/ Modification during Save

If this check box is selected, there will be no actions related to BOMs (bills of material) from the integration.

Show Synchronize message during Save

If this check box is selected, the PLM integration for SolidWorks will display the synchronization dialogbox during save operations.

Enable Selective Checkout

Use this property to enable or disable the option to operate selective check-out from the PLM integration for SolidWorks integration.

Automatically set resolve filename to New

When you save a new file to PLM and the integration identifies that a file with the same name already exists, the Resolve Filename dialog box is displayed.

This functionality allows the users to save the file with the same names to different projects without responding to the **Resolve Filename** option.

When you set this new preference with the combination of preference **File Name Uniqueness** = false, the integration process checks that a file with this name doesn't exist in the current project, and assumes that the file saved to PLM is the new one. In this case the integration creates a new PLM document without displaying the Resolve Filename dialog.

Toolkit Extensions to Original Application

This option lists the extensions that the integration uses to interact with other CAD applications.

The file extensions are linked to the integrations that will be used to open the file. For example, files with extensions SLDPRT, SLDASM, SLDDRW should be linked to the ORIGINATING_APP value SolidWorks. This link enables PLM to determine how to open the file.

Chapter 30: Troubleshooting

Please note the following, before you install the SolidWorks Integration:

Toolkit and Logwininet log files

The PLM support group may ask you to provide them with additional information about problems you may be experiencing with one of the PLM integrations.

To provide the support group with this additional information, the following log files can be created:

- Toolkit81Log
- Toolkit81_OpenLog
- Logwininet Log

To create these log files, proceed as described below. If you require further assistance creating these log files, please contact the PLM Support group or your local technical support department.

Note: In case you encounter any issues during the PLM operations from integration, do the below steps:

- Create the folders %tmp%\logwininet and %tmp%\toolkit. If these folders are existing, then delete the files in these folders.
- Reproduce the error
- Provide the following files along with the screen dumps of the error messages and the steps to reproduce the error.
- Files created in the folders %tmp%\logwininet and %tmp%\toolkit.

In certain cases, it is possible that the support team might require the CAD files also.

Toolkit log

To create toolkit logs, you must create a folder as "Toolkit" under %TMP%.

The names of the resulting log files are:

- Toolkit81Log_<process id>_<thread id>_[<number>].txt
- Toolkit81_OpenLog_<process id>_<thread id>.txt

Logwininet log file

To create the logwininet log file, create a directory titled Logwininet under the directory specified by your %TMP% environment variable.

The names of the resulting log files are:

- LogTrace_<thread id>.log
- LogTrace_<thread id>.err

Note: Only configure your system to create these logs when requested to by the PLM Support group. After the required logs have been created, logging must be turned off. Failure to turn off the logging will result in a severe reduction of the performance of your system and lead to a considerable use of your disk space.

CAD application not available in Application list of Mapping Tool dialog box

To define mapping rules for a particular SolidWorks template, in the PLM Mapping Tool dialog box, you must first select SolidWorks in the Application list and then select the template for which you want to define the mapping rules. However, sometimes SolidWorks is not available in the Application list.

Recommendation

In file C:\ProgramFiles\PLM Client\config\settings\cfeMappingTool.xml, add the following line to the <APPLICATIONS> list:

```
<APPLICATION ID="SOLIDWORKS" LABEL="SolidWorks"
```

```
TEMPLATE_CONVERT_APP="SolidWorks/SW_ITC.exe" />
```

The IN_ITC.exe convector is the utility used by the CAD applications for converting mapping template information to xml.

In addition, make sure that the directory and executable SolidWorks/SW_ITC.exe indicated in the applicable TEMPLATE_CONVERT_APP line is located in the PLM installation directory.

The entries in the cfeMappingTool.xml file use forward slashes (/) even though the windows standard is to use backward slashes (\) as a path separator.

Setting the Path for the Edit/ View in Integration

To ensure that the Edit/View in Integration from PLM operates correctly, the path of the related CAD application executable or batch file must be defined in the RunCad81.properties file. This file is located in the folder %CFE_CLIENT_HOME%ToolKit.

The following default entries exist in this file and should be adjusted to match your installation:

Note:

SolidWorks= C:\Program Files\PLM_Client\SolidWorks\ConnectToSW.exe

If a path contains a space (such as in Program Files), the path must be put between quotes.

Chapter 31: Recommendations and FAQs

The best practices with regard to the use of the PLM integration for SolidWorks are addressed in the form of a Frequently Asked Questions (FAQ) sheet.

The FAQ includes the following:

- [How to Improve Performance](#) on page 117
- [Saving files to PLM - best practices](#) on page 118
- [How do I introduce a new product to PLM?](#) on page 118
- [Can other people in my group perform changes to the same assembly I work with?](#) on page 119
- [When I work with large assemblies and save them to PLM it takes a long time to complete the operation.](#) on page 119
- [What happens if I have unlocked a component and then realize I must change it?](#) on page 119
- [After saving to PLM, how can I verify the results in PLM?](#) on page 119
- [Can I perform PLM operations on the components of my assembly or only on the root?](#) on page 120
- [Which procedures do you recommend for concurrent engineering?](#) on page 120
- [When I work in a concurrent engineering environment, how can I get the components that other users have changed and that are used in my assembly without closing any files?](#) on page 120
- [How can I operate the download manager without opening the files in SolidWorks?](#) on page 121
- [Why does the Resolve File Identity dialog box appear when I save a file to PLM?](#) on page 121
- [How can I find PLM information for files that exist locally on my computer?](#) on page 121
- [When I modify a complex assembly, which components must be checked out?](#) on page 122
- [How does the integration manage the link between drawings and items?](#) on page 122

How to Improve Performance

The following are the best practices in PLM:

- It is recommended to use the same folder for both EDIT and VIEW operations; for all users. Same EDIT and VIEW folders allow PLM to improve the download performance.
- It is recommended to frequently perform the Synchronization operation when you work with large assemblies. Synchronization operation allows you to identify the files modified in PLM. You can use the command Refresh Files from PLM to download the modified files. See [Refresh Files from PLM](#) on page 43

Saving files to PLM - best practices

Saving to PLM draws heavily on your configuration's resources. To save system performance, you are recommended only to save to PLM when updating the associated PLM documents, items, and files is actually required. For routine saves, use local save options.

In addition, the PLM Check in and Save and Unlock commands also save your data to PLM. Therefore, if the goal of your current session is to check in or save and unlock your part file, you can save your file locally from time to time and complete working on your file by checking in or using Save and Unlock.

- It is recommended to use the same folder for both EDIT and VIEW operations; for all users. Same EDIT and VIEW folders allow PLM to improve the download performance.
- It is recommended to frequently perform the Synchronization operation when you work with large assemblies. Synchronization operation allows you to identify the files modified in PLM. You can use the command **Refresh Files from PLM** to download the modified files.

Open referenced documents with read-only access

To work correctly with the PLM integration for SolidWorks, in the System Options - External References dialog box in SolidWorks, the Open referenced documents with read-only access check box must not be selected, as shown in the following picture.

If you select this check box, files opened from PLM for editing purposes may have read-only instead of read-write access.

In SolidWorks, to access this dialog box, select **Tools > Options > External Preferences**.

How do I introduce a new product to PLM?

If you work on a new large assembly not yet saved to PLM, you can highlight the top model of this assembly in the SolidWorks Feature Manager, and then save the entire structure to PLM. If you complete a component in a large assembly, select this component in the SolidWorks Feature Manager, and perform Save and Unlock.

For further information, see:

- [Saving and Unlocking Files](#) on page 27
- [Saving Files to PLM](#) on page 18

Can other people in my group perform changes to the same assembly I work with?

If you save an assembly to PLM, you own all of its components. Therefore, they are locked from use by other users. To allow other users to work on a subset of the same assembly, perform Unlock for the required components or perform Change Ownership.

For further information, see:

- [Saving and Unlocking Files](#) on page 27
- [Changing the ownership of a file](#) on page 32

When I work with large assemblies and save them to PLM it takes a long time to complete the operation.

After the initial save, perform Unlock on all components you must not change. During the save operation, the Take Ownership List dialog box displays a list of all unlocked writable files. Ensure you only select the files that you changed.

For further information, see:

- [Saving and Unlocking Files](#) on page 27
- [Saving Files to PLM](#) on page 18

What happens if I have unlocked a component and then realize I must change it?

You can perform the Take Ownership operation to ensure you own the component. For further information, see:

- [Saving and Unlocking Files](#) on page 27
- [Taking Ownership](#) on page 32

After saving to PLM, how can I verify the results in PLM?

The Show Meta Data operation displays a dialog box with all the PLM file, document, and item data as it was saved to PLM. You can also use the **Open in Document Workspace**, or **Open in Item Workspace** options to see the data in a Show Meta Data structure workspace.

For further information, see:

- [Viewing PLM Data](#) on page 34
- [Opening a file in PLM](#) on page 36

Can I perform PLM operations on the components of my assembly or only on the root?

You can perform any PLM operation on any component you select in the SolidWorks Feature Manager.

When I work in a concurrent engineering environment, how can I get the components that other users have changed and that are used in my assembly without closing any files?

You can click Refresh files from PLM to update the currently open files from Infor PLM. For more information, refer to [Refresh Files from PLM](#) on page 43.

Which procedures do you recommend for concurrent engineering?

There are two approaches to manage file changes for teams working on assemblies. After users have completed their work on a specific subset of an assembly, they can do one of the following:

Perform Save and Unlock

When the entire assembly is saved to PLM, the integration displays the Take Ownership List dialog box. This dialog box shows the writable files, so the user can select those files that must be saved to PLM. The user should avoid selecting files that were not changed.

Perform Check In

After a file is checked in, you cannot change it until it is checked out (minor check-out in this case). Therefore, each time you check in a file, it is saved to PLM, and later changes are saved in a new version of the file.

For further information, see:

- [Saving and Unlocking Files](#) on page 27

- [Taking Ownership](#) on page 32
- [Checking in a File](#) on page 28

How can I operate the download manager without opening the files in SolidWorks?

Ensure SolidWorks is closed and then perform Edit File in Integration, or View File in Integration from PLM. For further information, see Working with the Results Panel.

Why does the Resolve File Identity dialog box appear when I save a file to PLM?

The integration stores data in a specific location on your computer. This data allows the integration to establish a link between the files stored locally on your machine and the PLM data.

If this information is not available for the integration, and you try to save a file that already exists in PLM, the Resolve File Identity dialog box appears. For example, this dialog box appears if you copy the SolidWorks file to a different directory and then save it to PLM. For further information, see [Saving Files to PLM](#) on page 18.

How can I find PLM information for files that exist locally on my computer?

To find PLM information:

- 1 Open SolidWorks without any file.
- 2 On the PLM menu, click **Show Meta Data**.

The PLM details are displayed in a dialog box for all the files present in the integration locations on your computer. This dialog box displays only the data saved locally. If data has been changed in PLM, you must perform Synchronize Header.

For further information, see:

- [Viewing PLM Data](#) on page 34
- [Synchronize Headers](#) on page 39

When I modify a complex assembly, which components must be checked out?

Only check out the components you must change. Note that if you check out a component without checking out the assembly in which the component is used, you must save this component separately. If a subassembly is RELEASED, the integration does not check the subcomponents of this assembly, which enhances the performance of the save operation. For further information, see [Checking out a File](#) on page 30.

How does the integration manage the link between drawings and items?

There are two types of links between SolidWorks documents and items. The simple **Related Document** link only contains the link between an item and its model document. The **Related Drawing** link only contains the drawings created in the PLM integration for SolidWorks. The item workspaces in the PLM client utility allow users to view this information and make changes to the links, if required.

For further information, see [Saving Files to PLM](#) on page 18.