



Infor PLM for Discrete Creo User Guide

Release 2022.x

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About this guide

This document describes the configuration and usage of the Infor PLM Discrete for Creo.

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 1: Introduction

Infor PLM Discrete interfaces directly with Creo, facilitating direct transfer of product designs from the Creo environment into Infor ERP LN. This eliminates the inherent risks and errors associated with data entry from multiple sources. Creo 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.

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 Infor PLM Discrete menu and toolbar, the Infor PLM Discrete Integration for Creo provides access to Infor PLM Discrete functionality from within the native working environment, such as:

- Searching and retrieving files from the database.
- Saving files to the database.
- Creating hierarchical relationships between various documents and items.

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

Main features of the integration

The Infor PLM Discrete integration for Creo includes the following main features:

- Files can be saved from Creo to PLM.
- Document link management.
- Unique file names for all new Creo models.
- Creo 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.
- Ballooning information can be transferred from Creo to part list table in PLM. The values can also be retrieved from PLM to Creo.
- Utility to convert older version of Creo file to the latest version.
- The Batch Register Utility enables you to register all the Creo files to PLM. The utility can be used to migrate legacy data to PLM.
- Legacy files prepared outside the PLM environment can be imported with automatic updates of links, reflecting the relationships between all models and files.
- Graphic representation of product trees.

Chapter 2: Getting started

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

- [Installing the integration](#) on page 12
- [Requirements](#) on page 12
- [PLM embedded menu](#) on page 13

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

Requirements

The minimum hardware and software requirements for the Infor PLM Discrete integration for Creo are the same as those required for Infor PLM Discrete and for the Creo suite.

Refer to the latest release notes for the minimum software and hardware requirements.

Software configuration

For the latest Infor PLM Discrete and Creo 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 Infor PLM Discrete integration for Creo, the Creo must be installed.
- 2 To install the Creo integration, double click the setup.exe file, and follow the online instructions of the install shield.

PLM embedded menu

After the Infor PLM Discrete integration for Creo has been installed, the Infor PLM Discrete menu is added to your Creo menu bar.

Options displayed in the toolbar:

Option	Description
Connect	Connecting to PLM on page 15
Disconnect	Disconnecting from the PLM integration on page 16
Verify Items	Generates a report of the files that are not yet linked to items in PLM.
Capture Design Variants	Capturing design variants/configurations on page 44
Link to Item	Link to item on page 26
Save to PLM	Saving files to PLM on page 17
Save and Unlock	Saving and unlocking a file on page 28
Check In	Checking in a file on page 31
Check Out	Checking out a file on page 33
Check Out Major	Checking out a file on page 33
Take Ownership	Take ownership on page 35
Change Ownership	Changing ownership of a file on page 35
Dispatch to Business Process	Dispatch to business process on page 51
Document Structure	Opening a file in an SSA PDM workspace on page 37
Item Structure	Opening a file in an SSA PDM workspace on page 37
Item	Using infocards on page 38
Document	Using infocards on page 38
File	Using infocards on page 38
Create Balloon Attribute	Map Balloon Numbers from ProEngineer to PLM on page 65
Generate Thumbnails	Generating Thumbnails on page 61

Option	Description
Refresh Files from PLM	Refresh files from PLM on page 43
Sync StorageInformation	Synchronize headers on page 39
Show Metadata	Viewing PLM Data on page 36
Clear Storage Information	Clear storage information on page 41
Delete Local Files	Delete local files on page 42
Display Mapping	Display mapping on page 60
Associate Mapping	Associate Mapping on page 58
Remove Mapping	Remove Mapping on page 59
Import Mapping Template	To create a new mapping template.
Edit Mapping Template	To edit an existing mapping template (add/delete/modify rules).
Preferences	Setting Preferences on page 69
Download files from PLM	Use this option to download files from PLM. Users can search files by document, item or file and select them to download to Creo.

Note: The functions that you can perform is based on the user type and authorization.


Chapter 3: Working with PLM

This chapter describes the tasks to be executed while working with PLM:

- [Connecting to PLM](#) on page 15
- [Disconnecting from the PLM integration](#) on page 16

Connecting to PLM

To use Infor PLM Discrete Creo connector, you must establish connection between Creo and Infor PLM Discrete. The connection gives you access to the PLM database and projects that you need to work with.

- 1 From the Creo application, do one of the following:
 - a Click the **Connect** icon  in the PLM integration toolbar.
 - b Select **Connect** in the **PLM** drop-down menu. If the Infor PLM Discrete is already connected, an automatic silent connection occurs.



The image shows a 'Sign In' dialog box with the Infor logo. It contains the following fields and controls:

- Authentication Type:** A dropdown menu currently set to 'Basic'.
- PLM URL:** A text field containing 'https://lndevelopment.infor.com:7445/inui-'
- Environment:** A text field containing 'nllbawappsdev2'.
- Company:** A text field containing '0422'.
- User ID:** An empty text field.
- Password:** A text field with a placeholder 'Password'.
- Buttons:** 'Ok' and 'Cancel' buttons at the bottom.

- 2 Specify the Company, User ID, Password and other details.
- 3 Click **OK**.


When you open another Creo application, it will automatically have an enabled PLM toolbar and menu ready to use.

Note: On connecting to Infor PLM Discrete using a different user name, you must do a synchronization using Synchronize PLM Info command, else the PLM commands will not be enabled or disabled properly.

Disconnecting from the PLM integration

Disconnecting is a global operation for all Creo integration applications; disconnecting from one Creo application disconnects all connected Creo application from Infor PLMDiscrete.

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

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

Chapter 4: Saving files to PLM

The Save to PLM process saves your Creo file in the Infor PLM Discrete database. The document is saved with the status Draft. Each consecutive save updates the latest changes performed on the Draft revision of the document in Infor PLM Discrete.

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 Infor PLM Discrete.

To save your changes on the server (not only locally), perform Save to PLM. However, to save system performance, do not save to PLM too often. For more information, refer to "Saving files to PDM - Best practices".

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

The assignment of the file ID is determined by the parameters set up by the administrator as well as by the selected preference settings. The IDs for documents and items can be assigned manually or automatically by PLM, this is also set up in the preferences. For more information, refer to [Introducing PLM preferences](#) on page 69.

The ID of an item can be assigned in one of the following ways:

- Automatically, with an PLM item ID
- Automatically, with a Creo item ID. For every new part or assembly created, a new Creo parameter must be added.
- Manually

The IDs for documents and files can be assigned either manually or automatically by PLM.

If you are working with design variants, known as family tables in Creo, you will need to save the design variants before you save the item to PLM. For more information, refer to [Capturing design variants/configurations](#) on page 44.

If you are working with business processes, you may need to acquire ownership of the item from the previous owner. For more information, refer to [Saving and unlocking a file](#) on page 28.

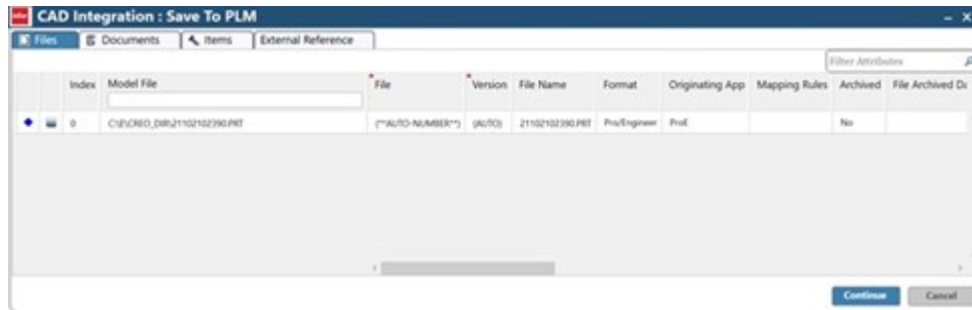
To save a file to PLM

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

Note: Do not use & < > ' “ symbols in the file names that you save to Infor PLM Discrete, this will cause errors to occur.

To save a file to Infor PLM Discrete, select **Save to PLM** from the PLM menu. Click **Save to PLM** in the menu:

- If all the IDs are generated automatically by PLM, which you can specify in the preferences, the file is saved. For further information on setting preferences, see [Introducing PLM preferences](#) on page 69.
- If the **Set Object Attribute During Save** check box is selected in the integration preferences, the **Cad Integration: Save To PLM** screen is displayed.



In this screen, 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 **Continue**.

For attributes that you do not update, the PLM default values are automatically populated. Attributes for File-Documents-Items - Commands.

Note: You only have to enter ID and/or revision numbers the first time that you save your file to PLM.

Note: No Item is created during initial Save To PLM if all the following conditions apply:

- The Integration preference Set as PHANTOM Item must be selected.

Saving 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, refer to [Introducing PLM preferences](#) on page 69.

- 1 Select **Save to PLM** from the PLM menu. The Cad Integration: Save To PLM screen is displayed. The item tab displays no data and the item fields are not accessible since your preferences are set to create a document only.
- 2 After updating the relevant attributes, click **Continue**. As a result, only the documents and files are created in PLM, and not the items.

Saving a file to PLM and link the file to an existing item

It is assumed you have the correct preference settings selected to create a document only. For more information, refer to [Introducing PLM preferences](#) on page 69.

- 1 Select **Save to PLM** from the **PLM** menu. The **Set Object Properties** dialog box is displayed.
- 2 Click **OK**, the document is saved to PLM.
- 3 Click **Link to Item** in the menu to link the document to existing item.

Infor PLM - Set Object Properties

File Name: 21102102390.PRT

Add Item

☐ Manual or Auto ID
☒ Manual ID
☐ Existing

Enter Item Manual ID

Instance: 21102102390

Item ID: (**AUTO-NUMBER**)

Revision: (AUTO)

☐ Create Item ☐ Link to Document

OK Cancel

- 4 Enter the item ID and revision number for an item that currently exists in the PLM.
- 5 Click **OK**.
- 6 Save file to PLM using Save to PLM.
The saved document is linked to the item that you specified.

Save to PLM checks Pro Model Check result

Creo has a utility called Model Check. This performs various checks on the models in Creo. This creates a parameter on the file to indicate if the Check has found any errors. Parameter Name is MC_ERRORS and it contains the integer value. 0 (Zero) indicates no errors. > 0 indicates errors. During Save To PLM of the Part or Assembly file, integration should read this value and save the part file only if the value of this parameter is Zero. If this is non-zero, then give a message and stop the process.

To implement this feature, select **Action to be taken when MC_ERROR has non zero value** preference.

If MC_ERRORS parameter has value other than “zero” the behavior for these three options is as follows:

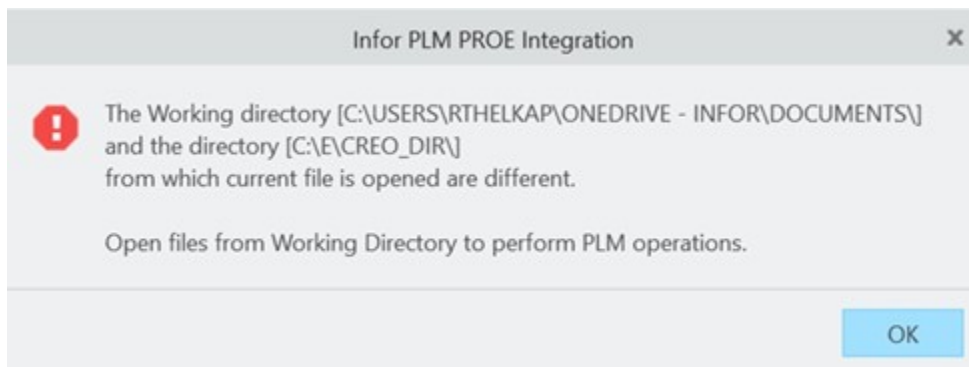
- Ignore: Save/Check-in operations continue without any error messages even if MC_ERRORS has non-zero value.
- Allow SaveToPLM only: Only “Save To PLM” and “Save and Unlock” operations are allowed. Check-in is not allowed.
- Restrict all: All the three operations Save/Save and Unlock/Check-in are restricted.

The error message is displayed and process is stopped if any part/assembly in the structure is found to have MC_ERROR value not equal to zero and if the integration preference is set Restrict All.



Check for the current Working Directory

In order for the correct execution of PLM operations on the Creo files, it is needed that files are open from the Working Directory that is set in the current session of the Creo. In order to avoid the errors a check is now added in the integration which will stop the PLM operations if the files are not from the working directory.



Action for objects locked by Business Process

When attaching objects to a Business Process, it is possible that those objects are already locked. What action needs to be taken by the program under these circumstances can be controlled by an integration preference.

Following are the list of possible values for this preference:

While attaching objects to a Business Process, integration performs these actions based on the preference:

- Attached as unlocked: 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 unlocked: All the objects are attached to the newly created Business Process without taking lock for any of the objects.

Creating Object IDs based on Item ID

When the integration preference Generate Object IDs based on Item-ID is enabled, during Save to PLM process of new files, DOC-ID, FILE-ID, FILE NAME are created based on the Item-ID.

In case of files for which Items are not created (Example drawing), FILE-ID and FILE NAME are created based on DOC-ID.

This is applicable only to the files that are being saved newly into PLM. This is not applicable to the files that are already present in PLM.

A new ProE file (assembly and part), during its initial SaveToPLM will be renamed based on newly generated Item-ID (for its default/generic configuration). Item-ID is generated based on mask defined in PLM.

At the time when Set Attributes dialog is displayed (during Save to PLM), the Item-IDs (for the new files) are displayed as (**Auto-number**). When you click **Ok** in this dialog box, IDs are generated for the files.

When user specifies an Item-ID in the Set Attributes dialog box (during Save to PLM), the file names (and other object ID's) are defaulted based on the Item-ID specified.

If Link to Item/Capture DV is performed for a new file, the Filename and other Object IDs are set based on the ItemID in the set attributes dialog (during SaveToPLM). DOC-ID, FILE-ID, Filename are generated based on the Item-ID. The object revisions also match the revision of Item. This requirement (of renaming files) will be applicable only for new files.

When the new assembly is saved to PLM, the components are renamed according to Item-ID. Therefore, parts are renamed based on the Item-ID.

While renaming the files, Family Table instances are also renamed. Instances are renamed according to its Item-ID. You can change only the **Instance Name** column but not the **Common Name** column. This column is used by windchill.

If you perform Save to PLM operation with the **Create Document Only = True** preference enabled, then Object IDs are created based on Doc-ID. File is also renamed based on Doc-ID.

Drawing files must be renamed based on Item-ID of the model file name (with some naming convention) and also the CDI_TYPE parameter defined on the Model of the Drawing.

Neutral files will be renamed based on the new name of the files.

You must be aware of the implications of defining mapping rules from (CAD to PLM) for file-name attributes, especially during Save to PLM (of new files).

If ProCabling assembly is renamed according to Item ID, .mfg, and .rsd files which are linked as slave files to assembly document, are also renamed.

Note: Ensure that the **let_proe_rename_pdm_objects** option is enabled in the current ProE session for this functionality to work correctly.

File Name Uniqueness

If this check box is selected, in the [General Option](#) on page 55 of [PLM preferences](#) on page 69, 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.

Chapter 5: Retrieving Files from PLM

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

To download files from PLM, click Infor PLM a Download files from PLM. A dialog box will be displayed and user can create search conditions based on attributes like file name, description, item id etc and files matching the search conditions are displayed in results grid.

For example, below search statements look for items that originated in Creo and item id containing value "23". Files linked to these items are displayed in Files grid. Select a file and click on **Download & Open** and file along with its dependents are downloaded into local system and the selected file is opened in Creo.

Download files from PLM

Search: ☒ Use Wildcard (*) ☐ Open for Editing ☐ Show Thumbnails ☒ Ignore Case

Define search statement:

Attribute Name: Add Remove Clear

Value:

List of search statements:

Attribute	Value
originating_app	proe
item	23

Query:

Items...

Item	Revision	Description	Business Process	CAD Instance Name
05082112320	0001	05082112320.PRT		05082112320
123_ASM	A	123_ASM.ASM		123_ASM
123_ASM_2	A	123_ASM_2.ASM		123_ASM_2

Documents...

Document	Revision	Description	Application Format Type	Approval Date	BOM Defining	Business Process
D-CADTEST-000620	A	05082112320.PRT	PE Part		Yes	

Files...

File	Version	File Name	Archived	Area	Base Area	Base Version	CAD Environment	Category List
FILE-0014409	0001	05082112320.PRT	No	A1	A1	0001	Not Applicable	Not Applicable
FILE-0014409	0001	05082112320.PRT	No	A1	A1	0001	Not Applicable	Not Applicable

Download Additional Drawings

If this check box is selected, in the [General Option](#) on page 55 of [PLM preferences](#) on page 69, the **Edit/View file in Integration** option enables the user to download the related drawings of all the models in the current Document structure.

Indicator for Locally Changed Files in Download Manager dialog

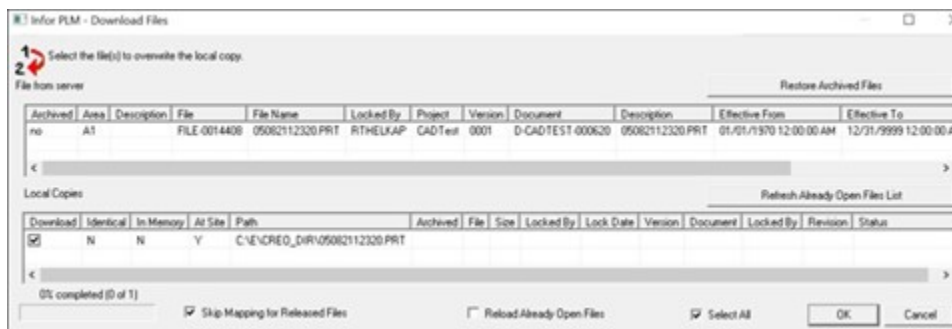
If this check box is selected, in the [General Option](#) on page 55 of [PLM preferences](#) on page 69, the value Y for the Identical field of the PLM - Download Files dialog box distinguishes between the files that are changed in PLM and locally by the user.

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.

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 check box **Skip Mapping for Released Files** 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.



Skip mapping for released files

When you select the preference **Skip Mapping for Released Files** during Download in the [General Option](#) on page 55 of [PLM preferences](#) on page 69, 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.



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](#) on page 55 of [PLM preferences](#) on page 69, the download manager indicates the changes if any, in the PLM database.

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 set to true by default.

Chapter 6: Link to item

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

If you click this option, the Link to Item dialog box is displayed. In this dialog box, you must specify the item to which the part/assembly file must be linked or enter an ID for the item that you want PLM to generate and link to the part/assembly file.

Infor PLM - Set Object Properties

File Name: 21102102390.PRT

Add Item

☐ Manual or Auto ID

☒ Manual ID

☐ Existing

Enter Item Manual ID

Instance: 21102102390

Item ID:

Revision:

☐ Create Item ☐ Link to Document

OK Cancel

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.

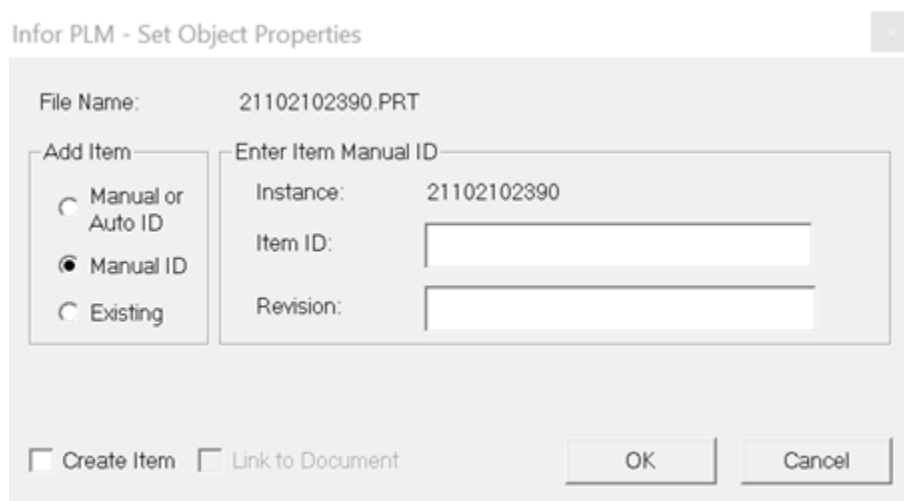
If you select the **Existing** radio button, in the **Item ID:** field, you can enter an item from the PLM database. As a result, the part file is linked to the item that you entered or selected in the **Item ID:** field.

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 preferences set in the PLM Preferences dialog box determine which item data you can specify in the Link to Item dialog box. For further information, see [PLM preferences](#) on page 69.

When the present file is not yet saved in PLM and you click Link to Item on the PLM menu, the following screen is displayed.



The dialog box is titled "Infor PLM - Set Object Properties". It contains the following fields and controls:

- File Name:** 21102102390.PRT
- Add Item:** A group box containing three radio buttons:
 - Manual or Auto ID
 - Manual ID** (selected)
 - Existing
- Enter Item Manual ID:** A group box containing three text fields:
 - Instance:** 21102102390
 - Item ID:** (empty field)
 - Revision:** (empty field)
- Buttons:** "Create Item" (unchecked), "Link to Document" (unchecked), "OK", and "Cancel".

If you select **Create Item** check box, the item is created in PLM when user clicks **Ok**, without waiting for the Save to PLM operation. Default setting of the Create Item is defined by the integration preference **Create Item Checkbox Default** under **Link to Item Dialogue** group in **Preferences**.

Click **OK**.



The dialog box is titled "Infor PLM CAD Integration". It contains the following elements:

- Icon:** A question mark icon in a blue circle.
- Text:** "Create the Item in PLM for this Item now?"
- Buttons:** "Yes" and "No".
- Footer:** "#77020"

When the current file is already in PLM and when you click **Link to Item**, the **Link to Document** check box is selected indicating that when user clicks OK, the item is linked to this document in PLM, without the need to perform **Save To PLM** operation. The **Link to Document** check box is enabled based on the options selected in the **Link to Item Dialogue** group in **Preferences**.

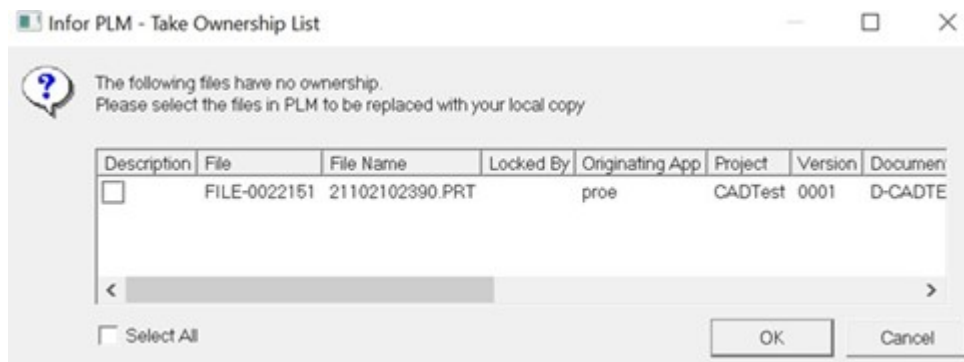
Chapter 7: Saving and unlocking a file

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 ownership of a file](#) on page 35.

To save and unlock a file:

- 1 Select **Save and Unlock** from the **PLM** menu. As a result, the file is saved in PLM with no owner.
- 2 To save the file, first you have to take ownership of the file. Click the **Take Ownership** option in the menu. The Take Ownership List dialog box is displayed.



- 3 Select the required file and click **OK**.
- 4 Save the file again in Creo and then to PLM.

Item Creation during initial save

Item ID creation during Save to PLM process can be controlled by new preferences. The preferences are grouped as Set-Item ID during Initial Save category.

The Set-Item ID during Initial Save preference category, contains the following preferences.

- [Set Item-ID based on](#) on page 29

- [Action to take if Item-ID exists in PLM database](#) on page 29
- [Action to take when same Item-ID is generated for multiple files](#) on page 29

Set Item-ID based on

Item-ID is created during initial save depending on the value selected by the user for this preference. Allowed values

None:

The functionality is not enabled.

File Name

The generated Item-ID is same as file name.

Part Number

The generated Item-ID is same as part number.

Action to take if Item-ID exists in PLM database

Allowed values

Cancel Save

The save process is canceled with the message that multiple files resolve to the same Item-ID.

Generate Auto ID

Generates the Item-ID based on mask when multiple files resolve to the same Item-ID.

Link to Item (Set Defining Manually)

A dialog box is displayed to the user in which he can decide to set the document as BOM defining or not.

Link to Item (Set Defining to No)

Action is automated as if user has selected **No** in the above dialog. Document created for this file will not be BOM defining document.

Link to Item (Set Defining to Yes)

The action is automated as if user has selected **Yes** in the above dialog. Document created for this file will be BOM defining document.

Action to take when same Item-ID is generated for multiple files

Allowed values

Cancel Save

Cancel the save process and a message is displayed indicating that multiple files resolve to the same Item-ID.

Generate Auto ID

Generate the Item-ID based on mask when multiple files resolve to the same Item-ID.

Allow Link to existing Item

If the item-ID is resolved to the existing Item, then this values indicate that file can be linked to the existing Item. But this is based on certain conditions and is dependent on Action to take if Item-ID exists in PLM database preference.

Chapter 8: Checking in a file

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

The check-in operation accomplishes the following:

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

After the file has been checked in, you can only change the file by checking it out. To check in a file:

- 1 Select **Check In** from the **Infor PLM** menu.
- 2 If prompted, save the file to Creo first. The file is checked in.

You can specify how to handle files after they have been checked in. See [PLM preferences](#) on page 69.

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: ASM1
- Sub-assembly: ASM2
- Child of sub-assembly: ASM3

The user does the following:

- a Checks out the parent assembly: ASM1
- b Checks out and changes the child of the sub-assembly: ASM3
- c Checks in the parent assembly: ASM1

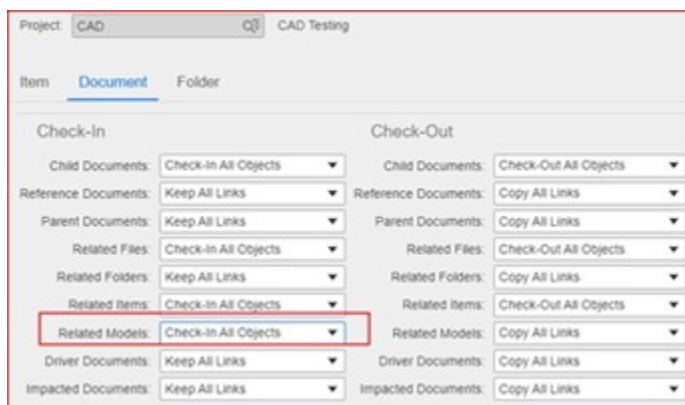
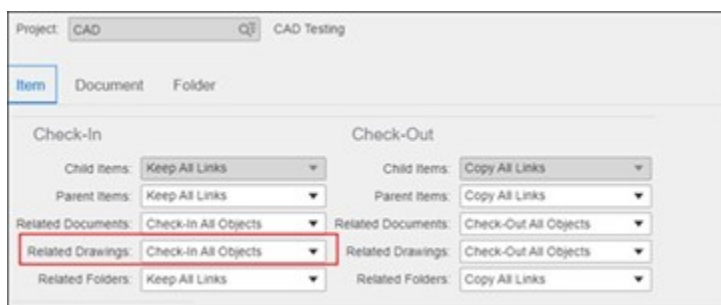
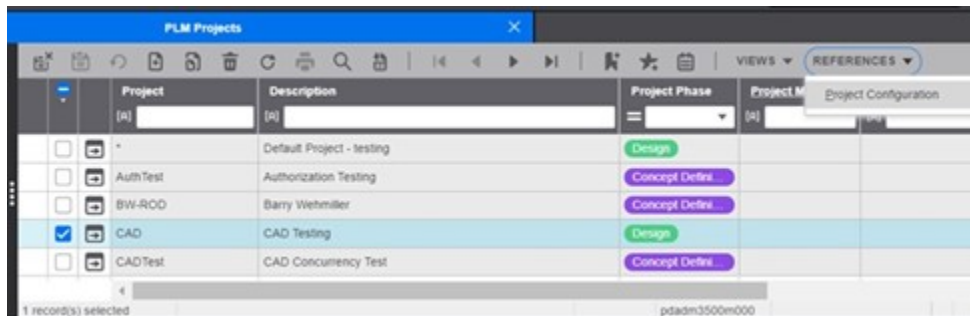
As a result, only the parent assembly is checked in. The child of the subassembly is not checked in because the sub-assembly has not been checked out and therefore remain in released status at the time when the parent assembly was checked in.

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.

Open PLM Projects (pdadm3500m000) session. Select **References** menu and click **Project Configuration**.



If any of the drawing files of the model is missing during the check-In operation, the following dialog box indicates the missing drawings.



Chapter 9: Checking out a file

You must check a file out of the vault in order to change it.

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.

Two types of check out are available:

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:

- 1 In Creo, select Download from PLM or access from the local drive.
- 2 Select the assembly or component part that you want to check out and select **Check Out** from the **PLM** menu.

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.

Project: CAD Q3 CAD Testing *****

Item Document Folder

Check-In		Check-Out	
Child Items:	Keep All Links	Child Items:	Copy All Links
Parent Items:	Keep All Links	Parent Items:	Copy All Links
Related Documents:	Check-In All Objects	Related Documents:	Check-Out All Objects
Related Drawings:	Check-In All Objects	Related Drawings:	Check-Out All Objects
Related Folders:	Keep All Links	Related Folders:	Copy All Links

Project: CAD Q3 CAD Testing

Item Document Folder

Check-In		Check-Out	
Child Documents:	Check-In All Objects	Child Documents:	Check-Out All Objects
Reference Documents:	Keep All Links	Reference Documents:	Copy All Links
Parent Documents:	Keep All Links	Parent Documents:	Copy All Links
Related Files:	Check-In All Objects	Related Files:	Check-Out All Objects
Related Folders:	Keep All Links	Related Folders:	Copy All Links
Related Items:	Check-In All Objects	Related Items:	Check-Out All Objects
Related Models:	Check-In All Objects	Related Models:	Check-Out All Objects
Driver Documents:	Keep All Links	Driver Documents:	Copy All Links
Impacted Documents:	Keep All Links	Impacted Documents:	Copy All Links

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. In this case the user must perform the Edit File after performing the check-out from the PLM.

Chapter 10: Changing 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 Infor PLM Discrete.
- The user who checked it out of Infor PLM Discrete.
- 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.

To change the ownership of a file:

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

Take 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 Creo Feature Manager, select the part of which you want to take ownership.
- 2 In the PLM menu, select **Take Ownership**.

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

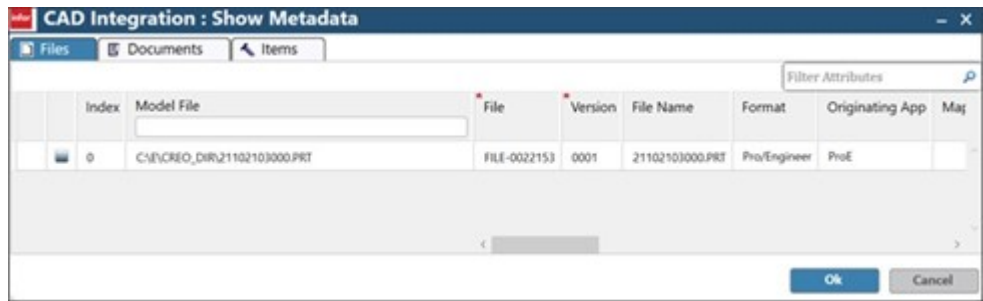
Note: When **Take Ownership during Edit File preference** check box is not selected, the users do not take the ownership of the files even when they execute **Edit File** command.

Chapter 11: Viewing PLM Data

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

In Creo, to view the relevant PLM data:

- 1 On the **PLM** menu, select **Show Metadata**.
The CAD Integration: Show Metadata dialog box is displayed.



- The default tab, **Files**, displays the PLM data of all the files that are part of your Creo structure.
- 2 Click the **Documents** tab to display the PLM document data related to your Creo part file.
 - 3 Click the **Items** tab to display the PLM Item data related to your Creo files.
- Attributes for File-Documents-Items - Commands.

Chapter 12: Opening file in a PLM Structure view

You can open a part file directly from the integration in an PLM item or document structure view.

To open a document structure in PLM:

- 1 Create a part file and save it to PLM. See [Saving to PLM](#) on page 17.
- 2 Select **Document Structure** or **Item Structure** in the **PLM** menu.

The document structure view displays the document and child documents that was created in PLM. The item structure displays the items and BOM that were created for that document with the child items (BOM).

Document Structure (D-CADTEST-000740, A)					
Document ID	Revision	Description	Project ID	Document Type	Status
▼ D-CADTEST-000740	A	21102103261.ASM	CADTest	COMMON	Draft
D-CADTEST-000739	A	21102103000.PRT	CADTest	COMMON	Draft

Chapter 13: Using infocards

An infocard is a dialog box that enables you to update meta data of items, documents or files generated from files created in Creo. 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 item, document or file in Creo 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 Creo object that you are working on, proceed as follows:

- 1 In Creo, select an assembly, a part or a drawing.
- 2 From the PLM main menu select **Item**, **Document** or **File**. The relevant update dialog box is displayed.
- 3 Make the required changes.
- 4 Click **Update**.

Note: To update a component of an assembly, in Creo, 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 **Item**, **Document** or **File** from the **PLM** menu.

To view your changes to the item, document or file, you can open the appropriate Structure. For more information, see "Opening a file in PLM".

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 view mode.

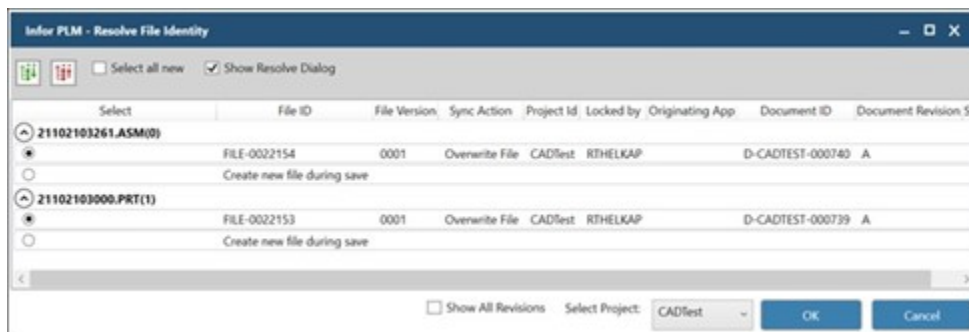
If you select Update Item, but no item is created for the part file with which you are working, the following error message appears: "The file [filename] you have selected is not associated with an Item in PLM." This occurs if your preferences specify that no PLM items must be created for the part files that you save to PLM.

Chapter 14: Synchronize headers

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

Chapter 15: Clear storage information

Clears the storage information for the active/selected file and its dependents.

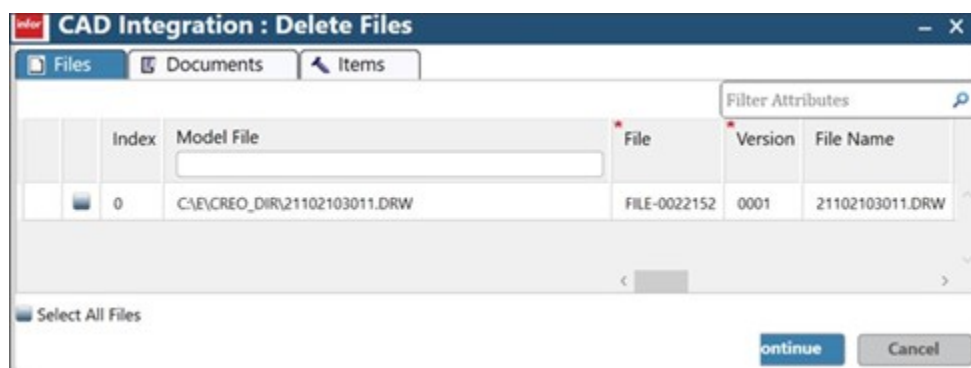
When you click **Clear Storage Information**:

- 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 part to a new project.

Chapter 16: Delete local files

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

Click the **Delete Local Files**, the **CAD Integration: Delete Files** screen is displayed:



Select the file to be deleted and click **Continue**.

Chapter 17: Refresh files from PLM

You can click **Refresh files from PLM** 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.

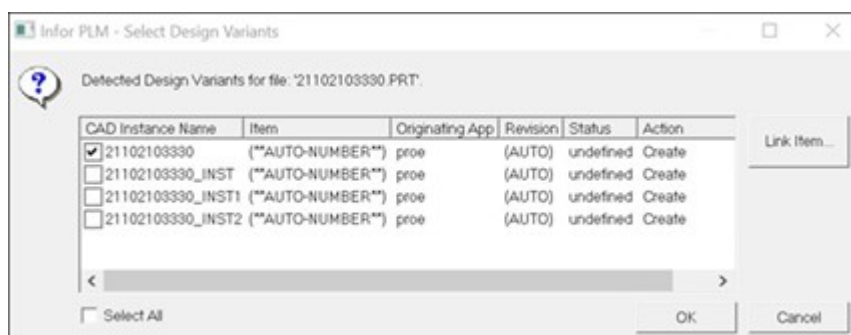
Chapter 18: Design variants

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

Capturing design variants/configurations

To capture design variants, proceed as follows:

- 1 Select **Capture Design Variants** from the **PLM** menu. The **Select Design Variants** dialog box is displayed.



- 2 Select the check boxes of the variants to be linked or select the **Select All** check box.
- 3 Click **OK**. The new items are created.
- 4 Save the file to PLM to link the items to the part. For further information, see [Saving to PLM](#) on page 17. When you save to PLM, links are formed between the document and the items created by means of the Capture Design Variants, Link to Item, and Top Down Design features.

Note: Regardless of manual or automatic preference settings for item generation, the PLM CAD integration will always automatically create a new item for a captured and saved design variant.

Each item's CAD instance name in PLM matches the name used in Creo. If you change the CAD instance name for an item in Creo and save the part to PLM, the name also changes in PLM.

Generate Items for all Configurations

If this check box is selected, in [PLM preferences](#) on page 69, 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.

Manually link new items to design variants

To manually link new items to design variants, proceed as follows:

- 1 Select **Capture Design Variants** from the **PLM** menu. The Select Design Variants dialog box is displayed.
- 2 Select the variants to be linked.
- 3 Click **Link Items**. The Add Design Variant dialog box is displayed.
- 4 Click **New radio** button, and enter the required item ID and revision.
Note that if the Automatic ID Entry option is selected in your preferences, the Item ID: and Revision: fields are unavailable. For further information, see [Introducing PLM preferences](#) on page 69.
- 5 Click **OK**. The item is linked to the selected variant.
- 6 To link more variants, click **Apply** and continue with Step 4.
- 7 Click **OK** in the Select Design Variants dialog box.
- 8 Save the file to PLM. See [Saving to PLM](#) on page 17.

To link existing items to a design variant

To link existing items to a design variant, proceed as follows:

- 1 Select **Save Design Variants** from the **PLM** menu. The Select Design Variants dialog box is displayed.
- 2 Select variants to be linked.
- 3 Click **Link Items**. The Add Design Variants dialog box is displayed.
- 4 Click the **Existing** radio button and enter the required item ID and revision.
- 5 Click **OK**. The item is linked to the selected variant.
- 6 To link more variants, click **Apply** and continue with Step 4.
- 7 Click **OK** in the Select Design Variants dialog box.
- 8 Save the file again in Creo.
- 9 Save the file to PLM. See [Saving to PLM](#) on page 17.

Manually linking new assembly items to design variants

Complete the same steps as those for linking new items to design variants. In PLM, a document is created for the assembly and for each of the parts in the assembly. The documents created for the parts of the assembly indicate that the parts are associated to that assembly.

If you click the **Items** tab in the document workspace, you see the linked parts for the document you selected.

If you open the assembly in an item workspace, you see the primary item for the assembly and sub-items for each configuration.

If you suppress a part in an assembly and save to PLM, the item of the suppressed part is no longer linked to the assembly item.

Automatically link new items to design variants

To automatically link new items to design variants, proceed as follows:

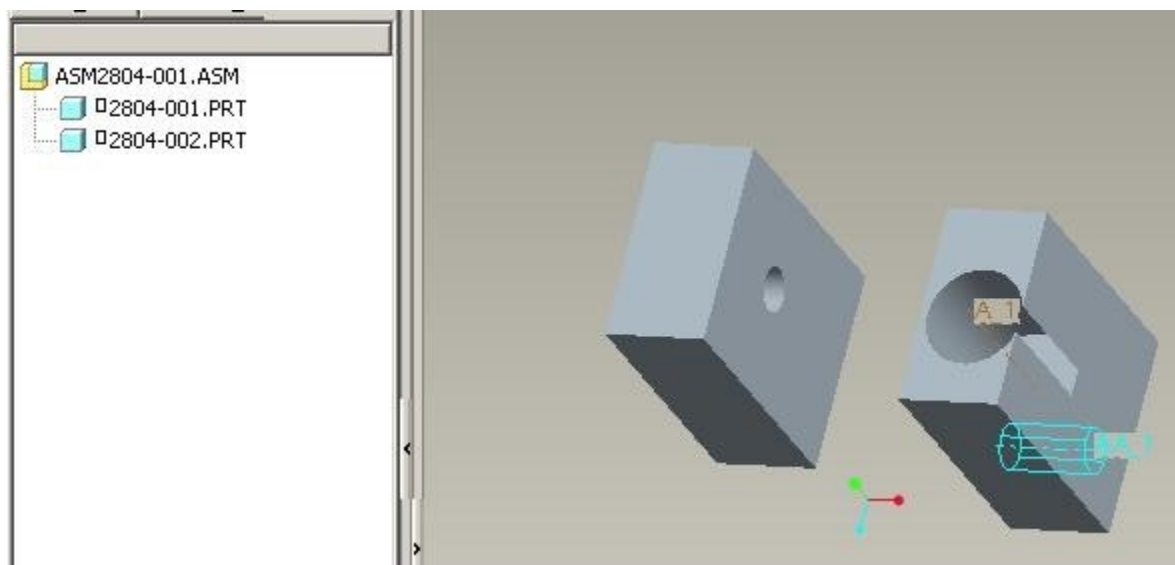
- 1** In Creo, click **Tools > Family Table**.
- 2** Select **New Instance**.
- 3** Specify a name for the configuration and click **OK**.

You will see two configurations for the part. One is the default for the originating item and one is the newly created configuration. For the following steps of the procedure, see [How to capture design variants](#) on page 44.

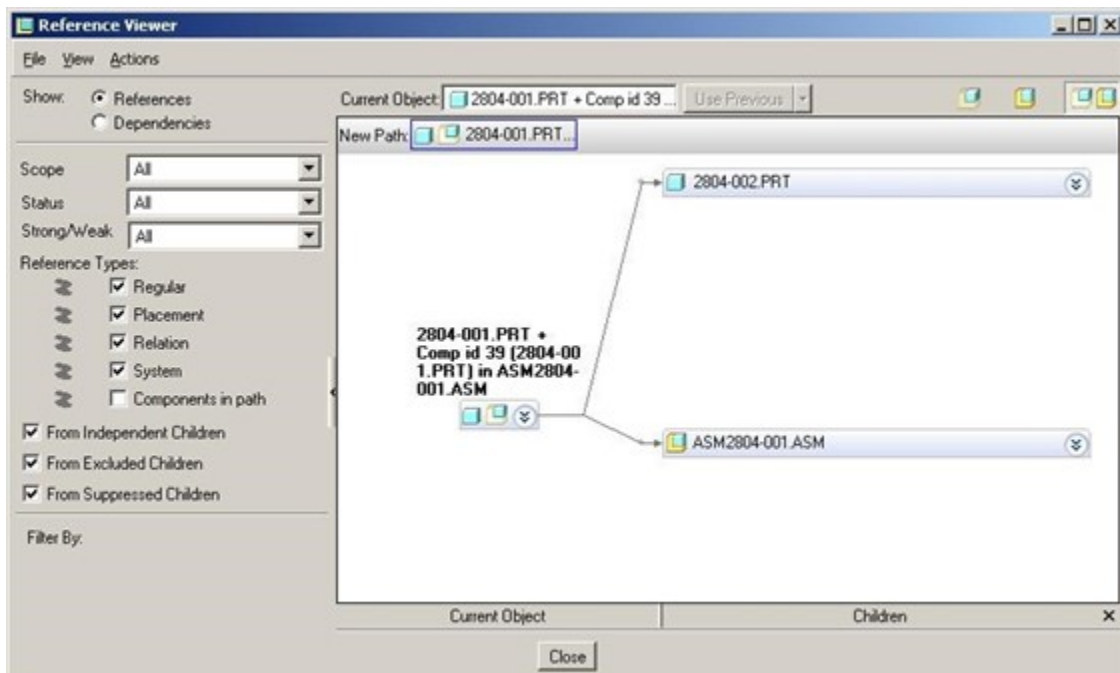
Chapter 19: Associative Links

Creo part or an assembly can be dependent on another part or assembly for its design. The associative links functionality, saves the associations between the Creo files to PLM when you perform the Save to PLM operation. To actualize this functionality, the External Reference tab is included in the CAD Integration: Save to PLM dialog box.

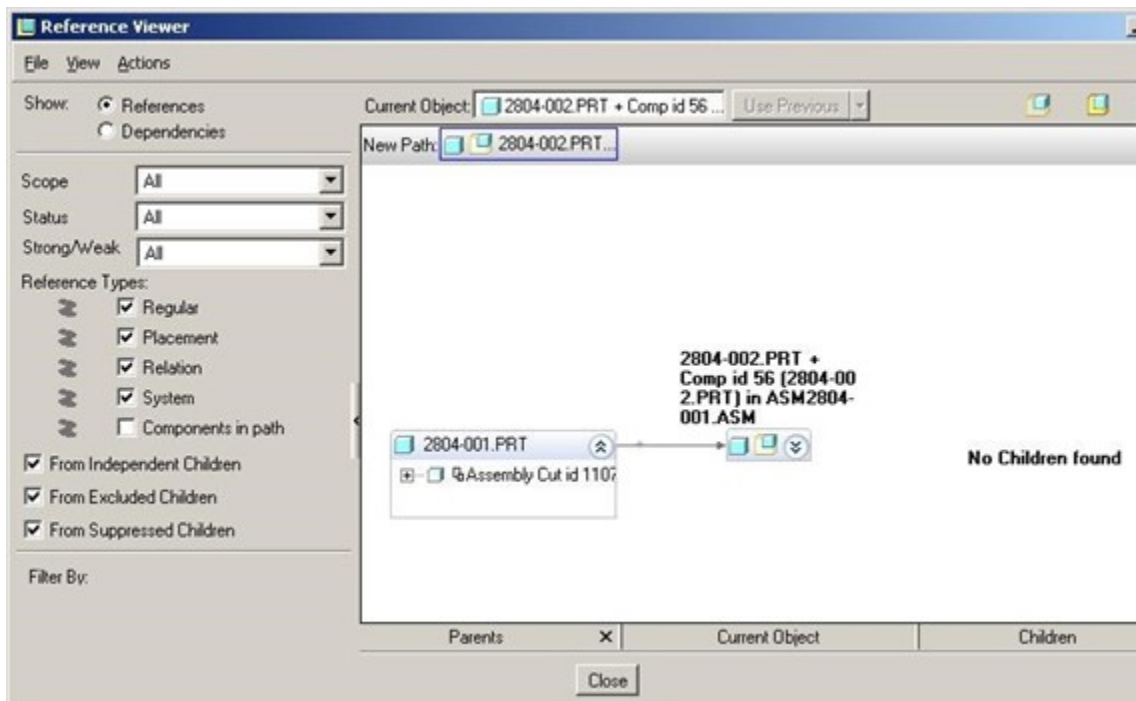
Example



In the reference viewer, the reference information for the 2804-001.prt is displayed as below:



The reference information for the 2804-002.prt part is displayed as below:



CAD Integration : Save To PLM								
Files			External Reference					
File Name (Driver)	File Name (Impacted)	File ID (Driver)	Version (Driver)	Document ID (Driver)	Revision (Driver)	File ID (Impacted)	Version (Impacted)	Document ID (Impacted)
21102103261.ASM	21102103005.PRT	FILE-0022154	0001	D-CADTEST-000740	A	FILE-0022153	0001	D-CADTEST-000739
21102103261.ASM	21102103000.PRT	FILE-0022154	0001	D-CADTEST-000740	A	FILE-0022153	0001	D-CADTEST-000739

When you perform the SavetoPLM operation, the references are displayed on the **External Reference** tab of the CAD Integration: Save to PLM dialog box.

In this dialog, Parent and Child references for each file is displayed. When you click **Ok** on this dialog, the document and the Item structure is created in PLM for the assembly. Additionally, the reference information is also stored in PLM.

You can use the PLM to create links between the files.

When you download a Creo file (Part or Assembly), all the drivers and the impacted files associated with the Creo file are also downloaded. These files are listed in the download manager. In the above example structure, if the 250609-P2.PRT is downloaded, the driver files 250609-P1.PRT and 250609-ASM.ASM are also downloaded to the local machine.

Associative Links Operations

Check in:

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

Check Out:

When you perform the Check-Out operation on a file from the Integration, the system will checkout all the impacted files.

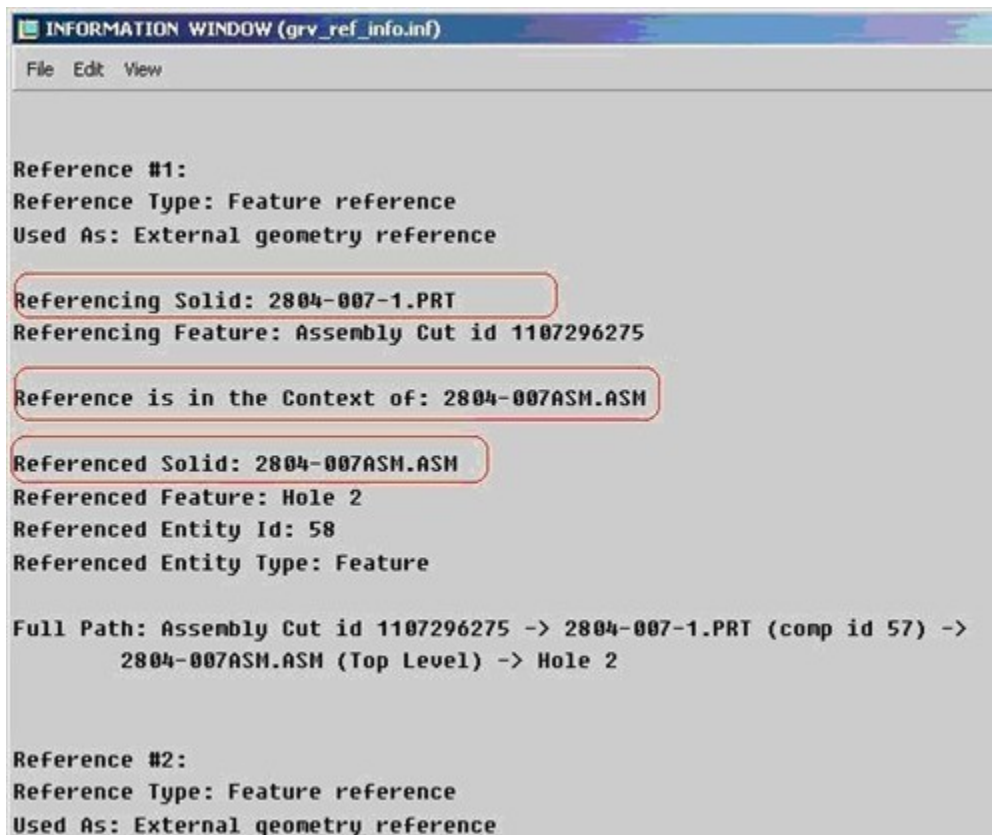
Integration Preference:

To enable the associative links functionality, you must enable the **Save External References** check box, which is part of the General property.

Limitations

Following are the limitations:

- If a reference is created for a part file in the assembly within the context of that assembly, the child refers to the assembly file.
- When the assembly has no references. The part has a child reference which refers to the model assembly. In this case, the part file is the driver and the assembly file is driven. When this structure is saved to PLM, part file will have assembly file as impacted and assembly will have the part as driver.
- If the part file is non-editable, save is performed on the assembly that is editable, the existing driver references of the assembly will be deleted from PLM. The reason being that when the assembly is saved and the part is not editable, the references of the parts are not checked. Since the assembly doesn't contain any references to the part, the link is not re-create and the driver reference is lost. Following screen shot illustrates how you can check the part references to identify a possible problem:



Chapter 20: 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 part file, to an PLM business process. This will link the PLM items and/or documents of the part file to a workflow, which will be distributed to the users associated to the workflow template.

In the preferences you can specify whether documents, items, or both must be linked to a business process. For this purpose, click **Preferences** on the **PLM** menu and on the **General** tab, click the relevant radio buttons of the **Attach to Business Process** group. For further information, see General tab.

Documents and/or items are linked to a new business process. When you click the **Dispatch to Business Process** option on the PLM menu, you are prompted to create a business process.

To dispatch documents and/or items to a business process:

- 1 Save the part file on which you are working to PLM.
- 2 Click **Dispatch to Business Process** on the PLM menu. The Create Business Process dialog box appears.
- 3 Adjust the business process data and select a workflow template.
- 4 Click **Create**. A dialog box appears asking you if you want to adjust the selected workflow template.
- 5 Click **Yes** if you want to adjust the template, otherwise click **No**. The business process is initiated after you have adjusted and saved the template or after you click **No**.

Note: If an PLM document or item should be attached to the business process according to your preferences, but an item or a document was not created when you saved the part file to PLM, an error message appears.

For example, this can happen if you specified that items must be attached to the business process while your preferences specify that no PLM items must be created for the part files that you save to PLM. For further information, see [Introducing PLM preferences](#) on page 69.

Dispatching related models or drawings

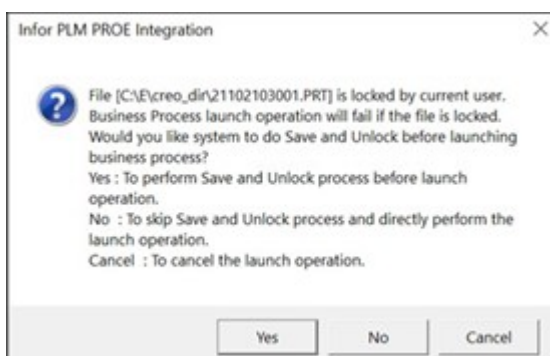
You can attach drawings of a model file to the business process. The model files are either part files or assembly files. To dispatch a drawing to the business process, enable the **Attached related Models\Drawings** check box. When you dispatch a model file to the business process, the drawings and 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 Properties for Integration is displayed.
- 2 Select the **Attached related Models\Drawings** check box located under the **Attached to Workflow** category.
- 3 Click **OK**.
- 4 In the Creo application, save the file to PLM.
- 5 Click **Dispatch to Business Process** on the **PLM** menu. The Business Process Details pane is displayed:

- 6 Enter the business process data and select a workflow template.
- 7 Click **Save**.



- 8 Click **Yes** to save and unlock the file.

Example

Part11.prt is the model file and Part11.drw is the drawing file associated with Part11.prt. The model file and the drawing are linked to ae001 business process. On successful completion of the business process, the model file and the drawing are linked to the ae001.

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 Creo, before you dispatch the model file to business process.

Chapter 21: General mapping

General mapping enables you to define attribute-based mapping rules for a template. When you create parts, assemblies, or drawings in Creo 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 Creo attributes are mapped to PLM attributes and vice versa for Creo part, assembly, or draft files that use the template containing the mapping rules.

Example

In Creo, create template Template_1. For Template_1, define the following mapping rule:

Creo 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 mapping 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.

- 1 In Creo, select **File > New** to create a new template file.
- 2 Specify the attribute properties to be mapped to PLM for the template. C:\Program Files\PTC\Creo 8.0.1.0\Common Files\templates.
- 3 Click **OK**.
- 4 Remove the Creo version number extension from the template parts, assemblies or drawing files. For example, file part1.drw.5 must be renamed as part1.drw.
- 5 Save the file to: C:\Creo\Templates folder.

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 <Creo installation path>\Templates folder.

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 Creo. This dialog box is divided into two sections, one for the PLM attributes and one for the Creo attributes.

To map Creo attributes to PDM attributes and vice versa.

To map, for example, the value of the DOC_IDTFR and DOC_DESCR attributes in the **Parameters** group in Creo to the Description attribute of a document from PLM, take the following steps:

- 1 Click the **Document** 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, as shown in the previous picture.
- 3 From the Creo side of the window, click **Parameters**. A list of attributes is displayed.
- 4 Click **DOC_IDTFR** and **DOC_DESCR** from the list.
- 5 Click the **To PLM** button in the middle section of the dialog box. This maps the value of the DOC_IDTFR and DOC_DESCR attributes to the Description attribute of the document in PLM. The two values of the DOC_IDTFR and DOC_DESCR attributes in the PLM document's description, separated by a comma.
You can add more mapping rules before accepting them. See [Mapping options](#) on page 55 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, disconnect and then reconnect to the server.

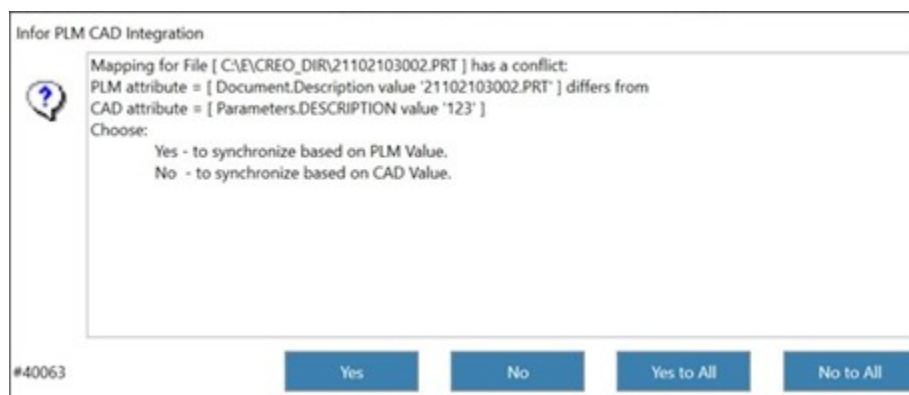
Mapping options

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 Creo. Or, you can reverse the direction and map an attribute value in Creo to an object's attribute in PLM.

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

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

In Creo, if you are working with a part file and you enter a value for an attribute that is mapped to both, a dialog box is displayed 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 Creo or the value entered in PLM.

If you map an attribute for a business process step, the following dialog box appears if you click the **To Integ.**, **To PLM**, or **Both** button.

In this dialog box you can select the relevant business process step.

Note: Depending on the selected type of attribute, the **To Integ.**, **To PLM**, and/or **Both** buttons can be unavailable. For further information on mapping restrictions, see [Mapping restrictions](#) on page 56.

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

For example, if you mapped a second Creo 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 Creo separated by a comma.

Mapping restrictions

Depending on the attributes selected in the PLM Mapping Tool dialog box, mapping to PLM, to Creo, 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, Creo, 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 Creo: The PLM Effective From and Effective To attributes.

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 Creo attributes must have the same format or must have a conversion in order to facilitate the following matches:

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 Creo 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 Creo file-specific rule can have no other general mapping.

Creo

Attributes from the **Creo 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 Creo attribute to one PLM attribute, one Creo attribute to many PLM attributes, or many Creo 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 Creo 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:

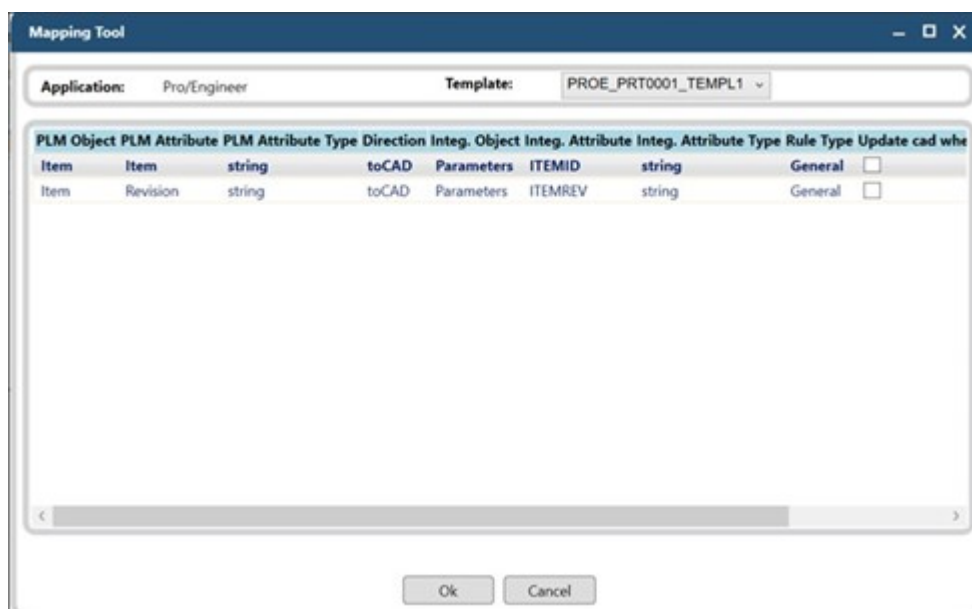
- 1 In Creo, open a template file for which you have created mapping rules from the <Creo installation path>\Templates folder as described in the previous sections.
- 2 Save the template file under another name (that is, under the name of the part, drawing, or assembly that you are going to work on and 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.

Associate mapping rules menu option

In addition to saving the template, for which you have defined mapping rules, under a different name, as described in [Associating the mapping rules to part files, assemblies, or drawings](#) on page 57, you can use the **Associate Mapping** from the Infor 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** option, proceed as follows:

- 1 In Creo, open the file for which you want to associate mapping rules.
- 2 Click **Infor PLM, Associate Mapping**. The PLM Mapping Tool dialog box is displayed, as shown below.



- 3 From the **Template** list, select a template you created in Creo. 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 mapping 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 remain intact.

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.

To perform configuration-specific mapping, proceed as follows:

- 1 Define a mapping rule using a configuration-specific attribute in Creo. For this purpose, on the PLM Mapping Tool dialog box, select **Configuration-specific** under the Creo section.
- 2 Define a rule to map, for example, map the value Mass in the Mass properties in Creo 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 PDM items. For further information, see [Capturing design variants/configurations](#) on page 44.

- 3 Create a file in Creo.
- 4 Specify configurations. For example, specify a value for Mass.
- 5 Capture the design variants for the configuration. For further information, see [Capturing design variants/configurations](#) on page 44.
- 6 Activate the default configuration and save the default configuration to PLM. As a result, the mapping applies only to the item associated with the default configuration.
- 7 Activate the configuration captured through design variants and save it to PLM. This applies to mapping to items associated with the current configuration as well.

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
Parameters	Parameters	Parameters

Parts	Assemblies	Drawings
Mass Properties	Specific Attributes	
Material Properties	Mass Properties	
Configuration-specific	Configuration-specific	Specific Attributes
Specific Attributes		
Dimension		

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 PLM Administrative Console as described previously, you must first disconnect and reconnect Creo 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 55, 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 **Item**, **Document** or **File** from the PLM menu to see the mapped values. For further information, see [Using infocards](#) on page 38.
- Use the Infocard options: **Item**, **Document** or **File**.
- Edit the files.

Note: If a source or a target attribute defined in a mapping rule does not exist for the part file you are working with, the following takes place when you save the part file to PLM:

- For file-specific and configuration-specific mapping rules, the missing source or target attribute is created.
- For other mapping rules, the attribute mapping is omitted without displaying a warning message.

Chapter 22: Thumbnails

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


Thumbnails can be generated in the following ways:

- During the SavetoPLM, 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 ProE file to PLM, complete the following steps:

- 1 In the Creo application, select Infor **PLM > Preferences**; the Integration Properties for Integration dialog box is displayed.
- 2 Select the **Generate thumbnails during save to PLM** check box located under the **Thumbnails** category.

 Thumbnails		
Generate thumbnails during Save to PLM	USER	<input checked="" type="checkbox"/>
Generate thumbnails for part file types	USER	<input type="checkbox"/>
Generate thumbnails for assembly file types	USER	<input type="checkbox"/>
Generate thumbnails for drawing file types	USER	<input type="checkbox"/>
Generate thumbnails for CAD file structure	USER	All Levels ▾
Format for thumbnails generation	ADMIN	PNG ▾

- 3 Click **OK**.
- 4 Click the **Save to PLM** option from Infor PLM menu.

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 Creo file, only when you have the ownership for the components.

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

- a** Open the Creo file.
- b** Click the **Generate Thumbnails** option on the PLM toolbar.

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

Multiple Document Linked to Item

When multiple documents are linked to the same item, thumbnails of all the documents are linked to the item.

Thumbnail Locations

By default, thumbnails are saved in the %CFE_CLIENT_HOME_%\ProE_Wildfire\Temp folder. To change the folder, modify the variable value of the PLM_THUMBNAIL_GEN_DIR environment variable, as shown below:



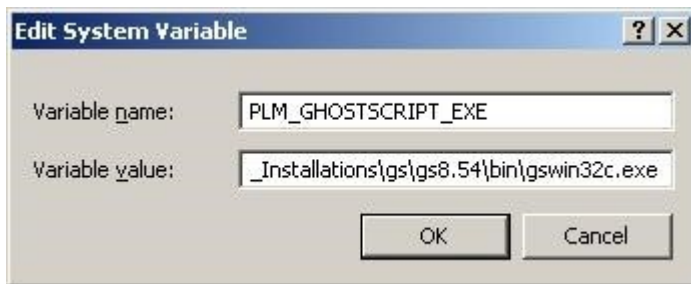
Thumbnail generation for Drawing files

To generate thumbnails for drawing files both during **Save to PLM** and **Generate Thumbnails** operations, complete the following steps:

- Download and install Ghost Script. Ensure that there are no spaces in the installation path.
- After the ghost script installation, include <install_folder>\bin and <install_folder>\ lib locations into PATH variable as shown below:



- Create an environment variable PLM_GHOSTSCRIPT_EXE and point this to the path of the gswinxxc.exe which is in the bin folder of Ghost script installation path. xx indicates the version installed on the system.



- Limitations
If the path %cfe_client_home%\proe_wildfire contains any spaces, then define the environment variable PLM_THUMBNAIL_GEN_DIR. The environment variable must contain a path with no spaces in it.

Integration Preferences for Thumbnails

The properties comprising the Thumbnails control the following aspects:

- Creo 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

If this check box is selected, thumbnails are generated when Creo files are saved to PLM. This option generates thumbnails only for the editable components of a Creo 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 Creo part files.

Generate thumbnails for assembly file types

If this check box is selected, thumbnails are generated for the Creo assembly files.

Generate thumbnails for drawing file types

If this check box is selected, thumbnails are generated for the Creo drawing files.

Generate thumbnails for CAD file structure

This preference is relevant only when you select the **Generate Thumbnails** option from the PLM menu.

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 box.

Format for thumbnails generation

Select the format in which the thumbnails must be generated.

This field can have the following values:

- PNG
- JPG

Chapter 23: Balloon Mapping

You can now transfer balloon numbers from Creo 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. In Creo, when an assembly is save to PLM, the balloon values are automatically transferred to PLM. It is not required to create a separate drawing file to define the balloon values.

Integration Preferences for Balloon Mapping

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

The Balloon Mapping property comprises of the following options:

- Transfer Ballooning Information
- Field in Part List table
- Warn for duplicate Balloon IDs
- Warn for incorrect Ballooning
- Apply Ballooning Values of non-default configuration to default configuration

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.

Component parameter for Balloon mapping

Specify the name of the parameter created on each component in the assembly in Creo. Value of this parameter is read and transferred to PLM.

Warn for duplicate Balloon IDs

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

Warn for incomplete Ballooning

If this check box is selected, you are alerted when a ballooning is performed incompletely for the child components.

Transferring balloon numbers from Creo to Infor PLM

- 1 In the integration preferences, select **Transfer Ballooning Information** check box.
- 2 In the Integration preference, specify the name of the balloon mapping parameter in **Component parameter for Balloon Mapping** field. This preference stores the balloon numbers for an assembly.
- 3 In the Integration preference, specify the Infor PLM Item object attribute in **Field in Part List Table** field. The preference stores the balloon values in Infor PLM. By default this field is set to 'FIND_NO'. This is 'Find No.' attribute of Infor PLM Item object.
- 4 Open a new assembly file in ProE/ Creo application.
- 5 In the ProE/ Creo application, select **Create Balloon Attribute** option from the PLM menu.



The components in the assembly must have a balloon mapping parameter generated which stores the balloon numbers.

The parameter name is same as specified in Step 2.

The starting value of the balloon mapping parameters is 10 and increments of 10, that is 10, 20, 30, and so on.

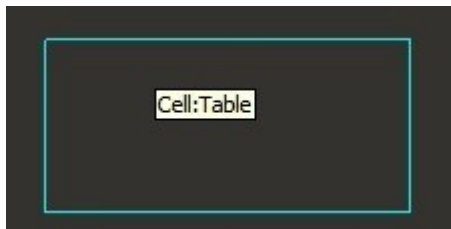
- 6 To view/ change the generated values in ProE/Creo application, select **Tools** and click **Parameters**. The Parameters dialog box opens.

Select the component of the assembly to view/ change the balloon mapping parameter value. In Look In, select Component and select the component in the assembly model tree as shown below.



- 7 Create a drawing file for the assembly.

- 8 Create a table with a single row and a single column. Select **Table** and click **Insert Table**.
- 9 Select **Table** and click **Repeat Region**. The TBL REGIONS menu opens.
- 10 In TBL REGIONS menu, click **Add**.
- 11 Select the single cell, click inside the cell and select **Done** in TBL REGIONS menu. The color of the cell changes to aqua as shown below.



- 12 Double-click inside the cell. The **Report Symbol** menu opens.
- 13 Select **asm**, **mbr**, **cparam** and **User Defined**.
- 14 Specify the balloon mapping parameter name as shown below. The balloon mapping parameter name specified in Step 2.



- 15 Click **Accept Value**.
- 16 Select **Table** and click **Repeat Region**. The TBL REGIONS menu opens.
- 17 Double-click **Switch Syms**. The balloon mapping parameter values created is displayed in the table.
- 18 To display the balloon mapping parameter values, select **View**, select **Update** and click **All sheets**.
- 19 To change value of a mapping parameter value, double-click in the cell and specify desired value.
- 20 Select **Table** and click **BOM Balloons**. The BOM Balloons menu opens.
- 21 Select **Set Region**.
- 22 Select the table that stores the balloon mapping parameter values.
- 23 Select **Set Param** and select the text in the table cell. The message is displayed in ProE/Creo status window as shown below.
- 24 In **BOM BALLOONS** menu, select **Create Balloon**.
- 25 In **BOM VIEW** menu, select **Show All** and click **Done**. The BOM BALLOONS are displayed on the drawings.
- 26 Save the drawing and in Infor PLM Discrete menu, click **Save To PLM**. The Find numbers are transferred to Infor PLM Discrete.

Chapter 24: Support for Pro-Manikin Files

Pro-manikin files are a set of library files provided by PTC. The files depict the human figures and also help the customers in simulation purposes. Manikin files are mainly used for the simulation of vehicles.

Note: The Save to PLM operation for Pro-Manikin files are supported from the integration kits 372_553 (64 bit) and 372_554 (32 bit) onwards.

The Pro-Manikin files installed during the installation of WF4 have the name “manikin” in them.

Assemblies can be created using these library files to suit the needs of the various test environments. For the PLM integration program to consider a file as a manikin file, the file name must contain “manikin”. The naming convention is required because there is no Pro-Toolkit API available to recognize the file as manikin files. When the Assembly file name contains “Manikin”, all the component under this are treated as Manikin files regardless of their names.

When a Pro-Manikin file is saved to PLM, only DOC and FILE objects are created for these files. No Item is created. When the Generate Object-Ids based on Item-ID preference is set enabled, the FILE-ID and the FileName are created same as DOC-ID.

If an assembly file is identified as a manikin file, then all the files under this assembly (all levels) are considered as Manikin files, irrespective of naming convention for the component files.

Chapter 25: Setting Preferences

The Infor PLM preferences control the way the Infor PLM integration for Creo works. To access the Infor PLM Discrete preferences, in Creo, select **Preferences** from the Infor PLM menu. As a result, the PLM Preferences dialog box appears.

In the All Users Preferences dialog box that is accessed from the **Administration** menu, the administrator maintains general values and settings for the integration preferences, including user authorizations. For each property, the administrator can authorize the users to change the setting or the value. For more information, refer to [General integration and authorization property settings](#) on page 70.

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

- 1 In Creo, select the Infor PLM menu.
- 2 Select Preferences or in the toolbar, click . The PLM Preferences dialog box is displayed.

Integration Properties for Integration dialog box

The Integration Properties for Integration dialog box comprises the following tabs:

- Creo
- Toolkit

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

- [Locations](#) on page 71
- [Initial Save Option](#) on page 71
- [General Option](#) on page 72
- [Link to Item Dialog](#) on page 26
- [Save Neutral Files Option](#) on page 74
- [Post Save Process](#) on page 76
- [Attach to Workflow/Business Process Option](#) on page 77
- [Troubleshooting Option](#) on page 78
- [Integration Preferences for Balloon Mapping](#) on page 65
- [Integration Preferences for Thumbnails](#) on page 63

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

- [General Option](#) on page 80
- [Toolkit Extensions to Original Application](#) on page 81

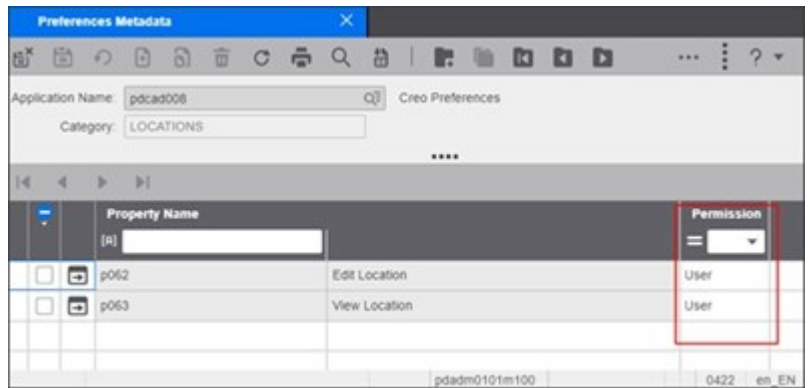
General integration and authorization property settings

The administrator maintains general values and settings for the integration properties, including user authorizations. For each property, the administrator can authorize the users to change the setting or the value.

To access the Integration Properties for Administrator dialog box, in which the administrator maintains the general property settings and authorizations:

In PLM, open the Preference Metadata (pdadm0101m100) session.

- Set the authorization for a property.



A property has either USER or ADMIN authorization. To change the authorization for a property, click the appropriate field in the **Permission** column for the property.

If a property has USER authorization, the value selected in this dialog box is the default value that each user can change locally in the Integration Properties for Integration dialog box.

If a property has ADMIN authorization, the value selected in this dialog box is effective for all users without the ability to change it locally.

To modify the Admin preferences, open All Users Creo Preferences (pdcad0108m100) session.

Note:

The properties listed in the Integration Properties for Administrator dialog box are identical (except for the authorization settings) to the local properties that users can manage in the Integration Properties for Integration dialog box. For further information about the properties, see [Integration Properties for Integration dialog box](#) on page 69.

Chapter 26: Creo Preferences

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

- [Locations](#) on page 71
- [Initial Save Option](#) on page 71
- [General Option](#) on page 72
- [Link to Item Dialog](#) on page 26
- [Save Neutral Files Option](#) on page 74
- [Post Save Process](#) on page 76
- [Attach to Workflow/Business Process Option](#) on page 77
- [Attach to Workflow/Business Process Option](#) on page 77
- Saving Older Revisions
- [Integration Preferences for Balloon Mapping](#) on page 65
- [Integration Preferences for Thumbnails](#) on page 63

Locations

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 Creo. If required, you can specify the same folder for both viewing and editing.

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 available:

Create documents only

If you select this check box, 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](#) on page 17.

Must assign an Item

If this check box is selected, you must assign an item to the file.

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](#) on page 17.

Manual ID entry for documents and/or items

When **Manual ID** check box is selected in the preferences, the following Set Attributes dialog box is displayed regardless of the preference values.

Generate Object-IDs based on Item Id

If this check box is selected, during SaveToPLM files are renamed based on the Item id and also the object Ids are created same as Item ID.

General Option

The **General Option** includes these preferences:

Allow Link to Released Item

If this check box is selected, you can link a document to a RELEASED item. If this check box is cleared, the user who tries to link a RELEASED item receives an error message.

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 using the View File in Integration option. If this check box is selected all the Infor PLM menu options such as, Save to PLM/Save and Unlock/Check-In that can modify data in PLM are disabled.

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 71.

Save Additional Drawings

If this check box is selected, the following operations save the local drawing files of all the editable and 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.

Show message on Successful open

If this check box is selected, a message is displayed upon the successful completion of Edit / View file in Integration.

Query Search Default

This option defines the default search object in Infor PLM > Download files from PLM tool.

- ITEM
- DOCUMENT
- FILE

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 check box, PLM warns the user that the same item is linked to parent and child components.

Take Ownership During Edit File in Integration

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

In case the Take Ownership during Edit File in Integration check box is selected, the user can take ownership for the selected file and its dependents before the save process.

Skip Mapping for RELEASED files during Download

If this check box is selected, 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.

Set as PHANTOM item

If this check box is selected, the item is set as a phantom item.

No Item is created during initial Save To PLM if all the following conditions apply:

- The Integration preference Set as PHANTOM Item is set to true.
- The user created a Creo parameter called Phantom (case insensitive) and has set the parameter to true.

Note: Set as PHANTOM Item functionality is also supported for Assembly.

To set phantom assembly, complete the following steps:

- In the integration, enable the **Set as PHANTOM** Item preference.
- In the assembly file create a string parameter PHANTOM and set the value to **TRUE**.

When you save the file to PLM, item is not created for assembly for which this parameter is set to true. Therefore, BOM is not created for such assemblies. Items are created for the components of the assembly.

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.

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 set to true by default.

Warn for Missing Check In 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 this check box is selected, Item IDs are created with same name as that of file name. This is applicable only for the initial save of the file into PLM.

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, the associative links functionality is enabled. For more information refer to Chapter, [Associative Links](#) on page 47.

Allow reload of already Opened files

If this check box is selected, during download already opened files that are downloaded are closed and opened again to reflect the latest changes.

Create separate BOM lines for component files linked to same item

If this check box is selected, when an assembly containing two different part files linked to same item is saved to PLM, two different BOM lines are created in 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 check box and do not select the Create Documents Only check box in the **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.

Save Neutral Files Option

Use this option to specify an additional format to save the local Creo 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 Creo 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.

Note: The preferences to generate neutral files in PDF format for an assembly and part is applicable only for Wildfire releases starting from 4.0 and above.

Configuration of PDF Writer

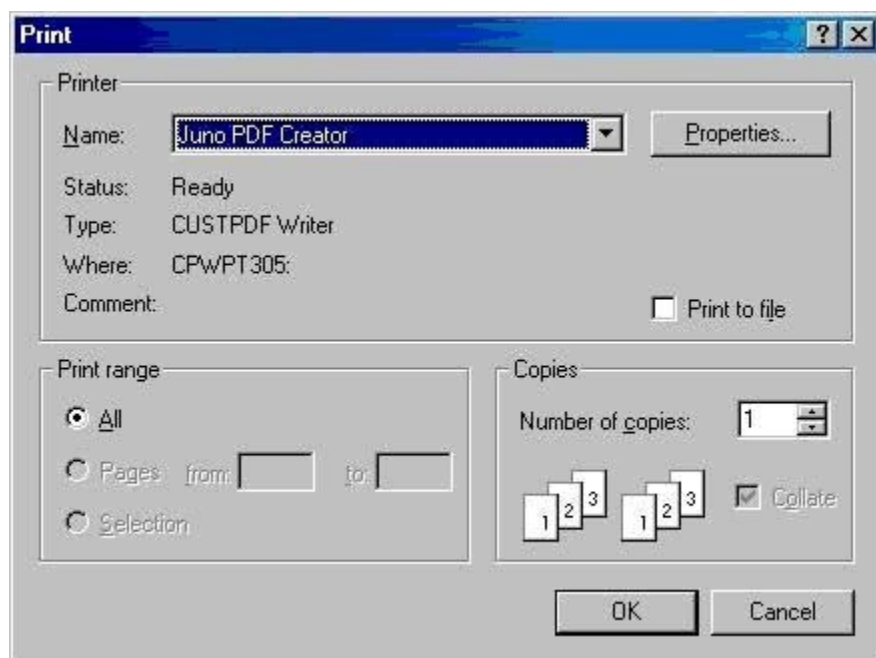
A third party PDF writer is required to generate PDF file for drawing containing OLE object(s) like a bmp, jpg, excel files.

You must configure the PDF writer to point to the file directory in which drawing file is available as the first step using the dialog shown below. The dialog box can be invoked using Preferences of the PDF writer.



Note: Other PDF writers will have their own settings which might differ to the one shown in this document. Users will have to go through the respective documentation.

As part of the Save to PLM process, the Integration will prompt user to select the PDF writer printer as shown below:



To configure the PDF writer, complete the following steps:

- 1 Select the available PDF writer.
- 2 Click **OK**.
- 3 Specify the PDF file name. Example if drawing name is Drw001.drw, PDF name must be specified as Drw0001_drw.pdf. After you specify the PDF file name, PDF file is generated and linked to the drawing.

Configuring a post script printer

Steps to configure a post script printer if system is Windows 7:

- 1 In the control panel, double click **Devices and Printers** and in the dialog, click **Add Printer**.
- 2 In the next dialog select **Add a local printer** and click **next**.
- 3 Under the manufacturer select **Generic** and under printers select "MS Publisher Color Printer".
- 4 Click **Next** until you see the **Finish**.

Post Save Process

Use the Post Save Process category to define the actions that can be performed on the local Autodesk Inventor file when it is copied to the PLM vault.

Following are the properties grouped under the Post Save Process category:

- **Check In Cleanup**
Select the action that must be performed on the file, when it is checked into the PLM Server.

Allowed values

- **Make Files Read-Only:** The file is retained in the Autodesk Inventor directory in the read-only mode. Therefore, the file cannot be updated.
- **Delete File:** The file is deleted from the Autodesk Inventor directory.

- **Save and Unlock Cleanup**

Select the action that must be performed on the file, when the Save and Unlock operation is executed.

Allowed values

- **Make Files Read-Only:** The file is retained in the Creo directory in the read-only mode. Therefore, the file cannot be updated.
- **Delete File:** The file is deleted from the Creo directory.

Show Message on successful save

If this check box is selected, dialog box is displayed when an object saved is successfully saved in to the PLM server.

Attach to Workflow/Business Process Option

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
- Attach related Models\Drawings
- Attach all related objects
- Allow edit Workflow
- Action for objects locked by Business Process
- Attach Draft Objects Only
- Attach All Items

Attached to Business Process

This option controls the PLM objects of a Creo file that can be attached to the business process.

Allowed values

- **Documents Only**
Only the documents associated with the Creo file are attached to the business process.
- **Items Only**
Only the items associated with the Creo file are attached to the business process.
- **Both**
Both the documents and the items associated with the Creo 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 Creo 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 Creo 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 check box 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 Creo. 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 CAD instance name as item-ID

If you select the Set CAD Instance Name as Item-ID check box and perform Save to PLM for an object, these are the three scenarios:

- During Normal Save
During normal save, the PLM sets the name of the CAD instance as the item ID.
- During Capture Design Variants
The PLM sets the filename in upper case (without extension) as the item id of when the **Originating item** field is **Yes** and the **Action** field is **Create**.

The PLM sets the **CAD Instance Name** value as the item ID for all other configuration.

If the item already exists in PLM the (****Auto-number****) (that is the Object default) value is not changed.

- During Link to Item

When you perform **Link to Item**:

You can change the item ID if required.

You can also change the item ID when you perform save to PLM.

Mapping of Customized Fields during Check In

Customized fields which are used in defining the mapping rule, for TO_CAD mapping, and get populated only after Check-In operation, needs to be handled in a special way. This is needed so that their values are correctly updated in the CAD file.

Follow the below steps to set up this feature.

Note: MAPPING TEMPLATE " NAME " and the PLMFIELD " FIELD " must be in Upper-Case.

- 1 Create a file `ModifiedMappingRulesForCheckIn.xml` in the `%CFE_CLIENT_HOME%\Toolkit` directory. The file must contain the following syntax:
- 2 Modify the contents of the `ModifiedMappingRulesForCheckIn.xml` file to match the specific requirement. Perform the following steps to modify the content:
 - a Enter a relevant template name.
 - b Enter a customized field name for the PLMFIELD. The PLMFIELD must have a TO_CAD mapping rule defined in this template.
 - c Type **EVALUATE** indicates the value that can be taken from the system variables like:
 - CURRENT_DATE
 - PLM_USER
 - CURRENT_TIME
 - CURRENT_DATE_TIME
 - d Type **REFERENCE** indicates the value which can be another field from the PLM table.
 - e Type **LITERAL** indicates that during mapping, this field will be assigned with the mentioned constant value.

Note: Only the options specified above are supported by the PLM.

Chapter 27: Preferences - Toolkit tab

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

- [General Option](#) on page 80
- [Toolkit Extensions to Original Application](#) on page 81

General Option

This option includes the following properties:

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 Creo will display the synchronization dialog box during save operations.

Show Top Down Load report

If this check box is selected, the PLM integration for Creo enables the user to open the report related to the **Top Down Load** operation.

Enable Selective Checkout

Use this property to enable or disable the option to operate selective check-out from the PLM integration for Creo 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 .prt, .asm, and .drw should be linked to the ORIGINATING_APP value Creo. This link enables PLM to determine how to open the file.

Chapter 28: Introducing setup and administration

In addition to setting the preferences for individual users in the PLM Preferences dialog box, the administrator must set up various data in order to make the Infor PLM Discrete integration for Creo work in the preferred way.

For further information on the setup data, see

- [Installation and setup](#) on page 82
- [Troubleshooting](#) on page 84

Installation and setup

Please note the following, before installing the PLM integration for Creo:

- Before a new version of the Infor PLM Discrete integration for Creo is installed, the previous installation must be uninstalled.
- Before installing the Creo integration, the Infor PLM Discrete client must be installed.
- Make sure that you have installed the latest Creo service pack.
- During installation you can determine whether only the administrator controls the integration preferences, or whether users can set their own preferences.
- All integration users must be defined in Infor PLM Discrete and assigned to the relevant projects.
- To ensure that Creo assemblies always locate parts correctly, the locations for the Edit and View directories for Infor PLM Discrete must both be set to the Creo working directory.
- Vault parameters must be set for each Infor PLM Discrete project. For more information, refer to [Vault parameter configuration](#) on page 83.

After installing a new version of the Infor PLM Discrete integration for Creo, make sure the correct settings are made for saving files in Infor PLM Discrete and viewing or editing files in Creo. For more information, refer to [Viewing, editing, and saving files in the integration - recommended settings](#) on page 82.

Viewing, editing, and saving files in the integration - recommended settings

To view and edit files in the integration, it is recommended to:

- Specify local download folders
Specify the folders in which you want Infor PLM Discrete to locally download the files that you want to view or edit in the integration using the **Edit Location/ View Location** preferences. It is a best practice to specify the same folder for both viewing and editing the files.
- Disable PLM commands
Disable Infor PLM Discrete commands when using the **View File** option. The area that you define on your disk to view and edit files is used for testing, running "what-ifs," and just plain viewing and not for performing product management.

Vault parameter configuration

In order for the integration to work properly, it is essential that the vault parameters are set correctly for each project.

The settings for CAD projects should not be changed to ensure the integrity of the data.

For details of how to set up a project and define the vault parameters for that project, see Project Settings in the Web PLM User Guide.

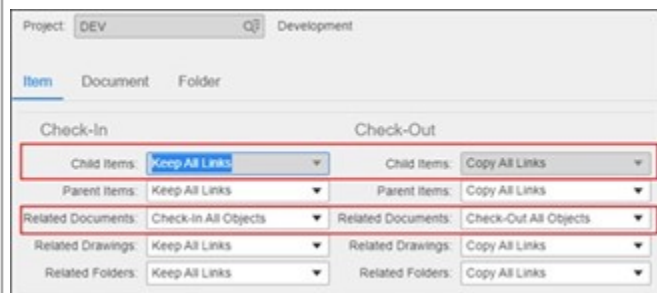
The settings ensure that document and item revisions are correctly synchronized in Infor PLM Discrete when check-in and check-out operations are performed.

Vault parameters for items

The vault parameters for items must have the following settings:

	After item check-in	After item check-out
Child Items	Keep All Links	Copy all links
Related Documents	Check in all objects	Check out all objects

The Vault Parameters dialog box displays the settings for items.



Vault parameters for documents

The vault parameters for documents must have the following settings:

	After document check-in	After document check-out
Parent document	Keep all links	Copy all links

	After document check-in	After document check-out
Child documents	Check in all objects	Check out all objects
Related items	Check in all objects	Check out all objects
Related files	Check in all objects	Check out all objects

Setting the Path for the "Edit/View" Command

To ensure that the Edit/View command from Infor PLM Discrete 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: ProE="C:\Program Files\PLM_Client\ProE_Wildfire\connecttope.exe"

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

Troubleshooting

This topic lists error messages, support log files, and the most commonly encountered issues.

Toolkit 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
- ProE_Integration_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.

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

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

To define mapping rules for a particular Creo template, in the PDM Mapping Tool dialog box, you must first select Creo in the Application list and then select the template for which you want to define the mapping rules. However, sometimes Creo 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="PROE" LABEL="Creo" TEMPLATE_CONVERT_APP="ProE_Wildfire/PE_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 “ProE_Wildfire/PE_ITC.exe” indicated in the applicable TEMPLATE_CONVERT_APP line exists under the PLM_Client 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.

Chapter 29: Working with the PLM Integration for Creo

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.

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.

Saving files to PLM - Recommended practices

It is recommended to enable the **Set Object Attributes** preference to view the files, documents, and items before saving to the PLM Server. You can also change the attribute such as, File IDs, Document IDs, Item IDs, and description.

Note: You can set the file ID, item ID, and document ID during first save.

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 Creo Feature Manager, and then save the entire structure to PLM. If you complete a component in a large assembly, select this component in the Creo Feature Manager, and perform Save and Unlock.

For further information, see:

- [Saving and unlocking a file](#) on page 28
- [Saving to PLM](#) on page 17

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 a file](#) on page 28
- [Changing ownership of a file](#) on page 35

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 a file](#) on page 28
- [Saving to PLM](#) on page 17

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 a file](#) on page 28
- [Taking ownership of a file](#) on page 35

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 Document Structure, or Item Structure options to view the data in a Show Meta Data structure workspace.

For further information, see:

- [Viewing PLM Data](#) on page 36
- [Opening a file in PLM](#) on page 37

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 a file](#) on page 28
- [Take ownership](#) on page 35
- [Checking in a file](#) on page 31

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 Creo file to a different directory and then save it to PLM. For further information, see [Saving to PLM](#) on page 17.

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

To find the PLM information for files that exist locally on your computer:

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

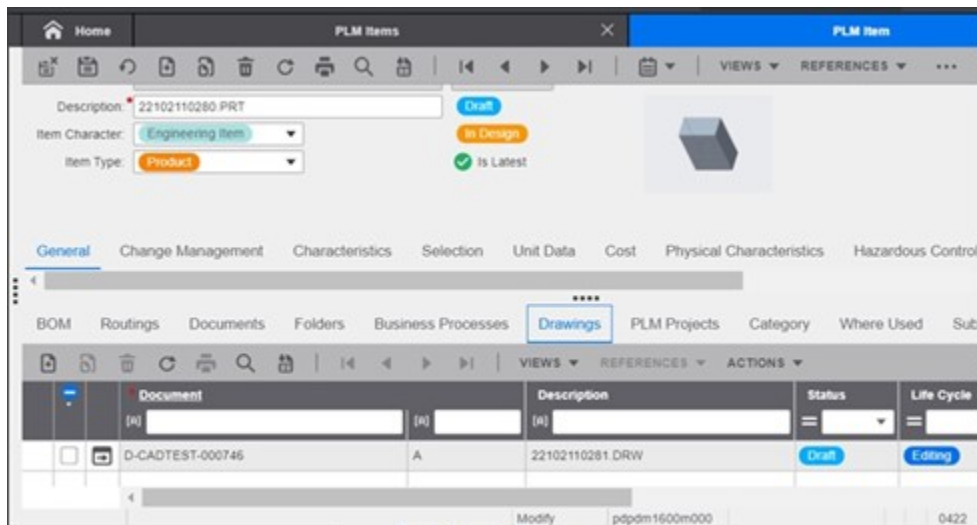
The PLM details are displayed in a dialog box for all the files present in the integration locations on your computer. This dialog box displays only the data saved locally. If data has been changed in PLM, you must perform **Sync Storage Information**.

For more information, see:

- [Viewing PLM Data](#) on page 36
- [Synchronize headers](#) on page 39

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

There are two types of links between Creo 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 Creo. The item structure in the PLM allow users to view this information and make changes to the links, if required.



For further information, see [Saving to PLM](#) on page 17.

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 33.

Chapter 30: Known Issues in Windows 7

The environment variables required for integration to work correctly:

Your Concept

- PROE_INST_DIR: This refers to installation directory of the Creo application.
- PRO_COMM_MSG_EXE: This refers to the exe file that is required for Creo application for its communication for other processes.

These two variables are created during the installation of the integration kit. But on Windows 7 system, it has been observed that these variables are created but not propagated to all users.

To fix this, perform the following steps after completion of integration.

Confirm whether these variables are created or not. Start a command window and type the commands as shown in the screen below. Two variables with values are shown below.



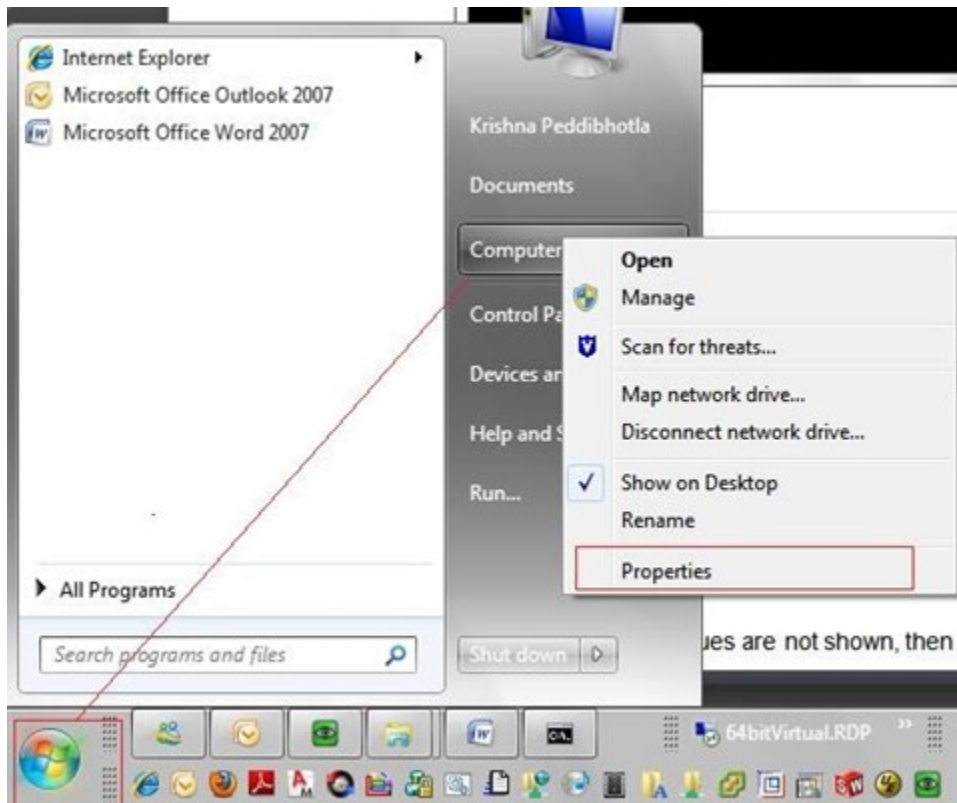
```
Administrator: C:\Windows\system32\cmd.exe

C:\Users\kpeddibh>set proe
PROE_INST_DIR=C:\PLM_Installations\CAD\WF4\

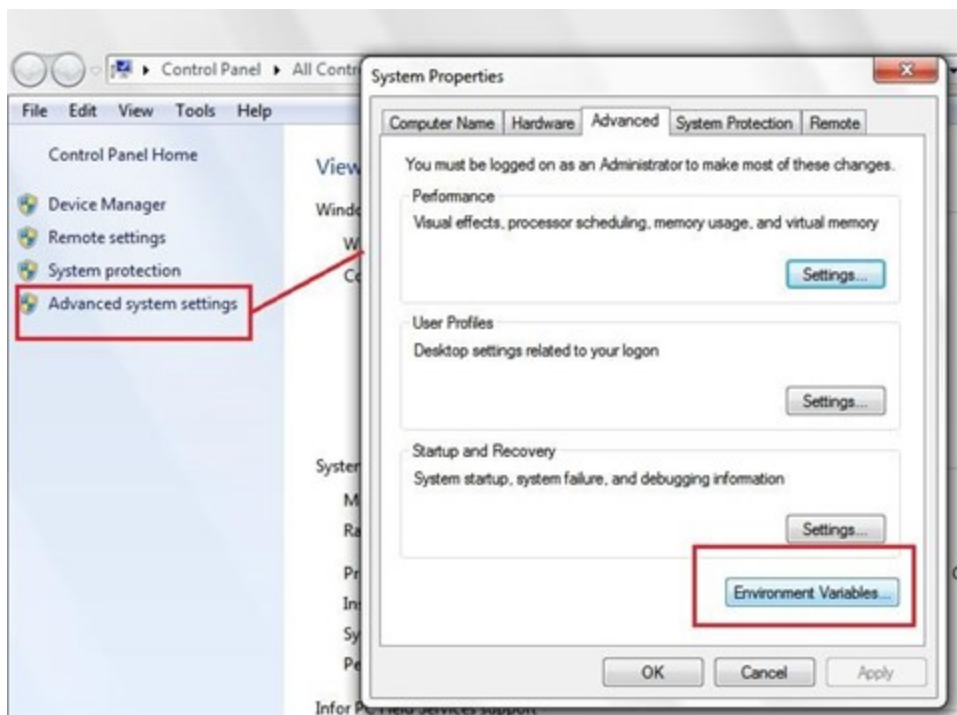
C:\Users\kpeddibh>set pro_c
PRO_COMM_MSG_EXE=C:\PLM_Installations\CAD\WF4\x86e_win64\obj\pro_conn_msg.exe

C:\Users\kpeddibh>_
```

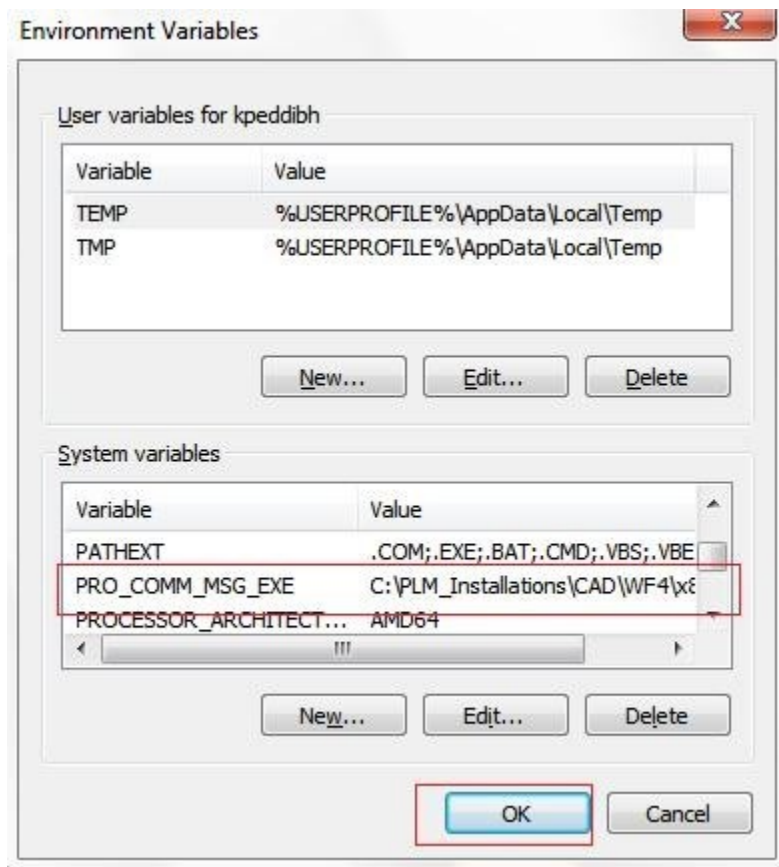
If two variables with values are not shown, then open the environment variables dialog. To open this dialog, first right click on the computer as shown below and select properties.



In the new dialog, click **Advanced system settings** and this will open system properties as shown below. In this dialog click **Environment Variables** button.



In the variables dialog, browse to ensure that the two environment variables are available. After this click **OK**.



Repeat the step # 1 and confirm whether two variables are now visible in the command window. If the variables are not visible, then logout and login again. Before you start PLM 8 integration to Wildfire, ensure that these environment variables are available.

Appendix A: Glossary

Attribute

The attributes of an object are the information fields that describe the object itself. For example, the attributes of a document include the document ID, revision number, description, status, creation date, and many more.