



Infor PLM for Discrete Autodesk Inventor 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 Autodesk Inventor.

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

This section introduces the *Autodesk Inventor - PLM integration*.

Infor PLM Discrete Integration interfaces directly with Autodesk Inventor, facilitating direct transfer of product designs from the Autodesk Inventor environment into Infor ERP LN environment. Autodesk Inventor files can be managed according to the 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 Autodesk Inventor provides access to Infor PLM Discrete functionality from within the native working environment. It provides functionality 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 Autodesk Inventor environment. Autodesk Inventor and Infor PLM Discrete exchange product information, and update files and documents throughout the design process.

The Infor PLM Discrete client must be installed before you can install and use this integration.

For instructions on installing and starting to use the Infor PLM Discrete Integration for Autodesk Inventor, see [Getting Started](#) on page 12.

Main features of the integration

The main features of the Infor PLM Discrete Integration for Autodesk Inventor include the following:

- Files can be saved from Autodesk Inventor to Infor PLM Discrete.
- Document link management.
- Unique file names for all new Autodesk Inventor models.
- Autodesk Inventor 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.
- Top-Down design of CAD assemblies is supported.
- The design process can be controlled using the workflow functionality.
- Preferences to control the behaviour of the integration.
- 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 Autodesk Inventor to part list table in PLM.
- Legacy files prepared outside the PLM environment can be imported with automatic updates of links, reflecting the relationships between all models and files.

Chapter 2: Getting Started

This section contains the following topics:

- [Installing the integration](#) on page 13
- [Software Configuration](#) on page 12
- [PLM embedded menu and toolbar](#) on page 13

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

Requirements

The minimum hardware and software requirements for the *PLM - Autodesk Inventor* Integration are the same as those required for Infor PLM Discrete and the Autodesk Inventor suite.

The minimum software and hardware requirements are as follows:

Software Requirements

For the software requirements, refer to the latest release notes.

Hardware Requirements

For hardware requirements, refer the following:

- Autodesk Inventor 2012 Installation CD.
- Infor PLM Discrete sizing documentation.

Software Configuration

For the latest Autodesk Inventor 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, perform the steps below:

- 1 Before installing the Infor PLM Discrete Integration for Autodesk Inventor, you must install the Autodesk Inventor application.
- 2 To install the Autodesk Inventor integration, double click the setup.exe file, and follow the online instructions of the Install Shield.

PLM embedded menu and toolbar

After installing the Infor PLM for Discrete Integration for Autodesk Inventor, the PLM toolbar menu is added to your Autodesk Inventor toolbar, and a drop-down menu is added to the Autodesk Inventor menu bar.

These menus and toolbar options are available:

Option	Refer
Connect	Connecting to PLM on page 15.
Disconnect	Disconnecting from the PLM integration on page 15.
Capture Design Variants	Managing Design Variants on page 44.
Link to Item	Link to Item on page 22.
Save to PLM	Saving to PLM on page 16.
Save and Unlock	Saving and unlocking a file on page 25.
Check In	Checking in a file on page 26.
Check Out	Checking out a file on page 29.
Check Out Major	Checking out a file on page 29.
Change Ownership	Changing ownership of a file on page 31.
Take Ownership	Enables ownership of a file.
Open in Document Structure	Opening a file in an PLM workspace on page 34.
Open in Item Structure	Opening a file in an PLM workspace on page 34.
Update Item	Using Infocard Option on page 38.
Update Document	Using Infocard Option on page 38.
Update File	Using Infocard Option on page 38.
Associate Mapping	Associate mapping rules on page 57.
Display mapping	Display mapping on page 58.
Remove Mapping	Removing mapping associations on page 57.
Show Meta Data	View PLM data on page 32.

Option	Refer
Delete Local Files	Delete Local Files on page 43.
Clear Storage Information	Clear Storage Information on page 42.
Generate Thumbnails	Thumbnails on page 62.
Synchronize PLM Info	Sync PLM Info on page 39.
Preferences	Setting preferences on page 87.
About	Contains product and system information as well as additional sources for professional assistance.

Note: All the PLM operations are performed on the active "document". The PLM commands are enabled only when the current user is authorized to perform those operations.

Chapter 3: Working with PLM

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


- Selecting a server
- [Connecting to PLM](#) on page 15
- Opening PLM
- [Disconnecting from the PLM integration](#) on page 15

Connecting to PLM

To use Infor PLM Discrete Integration for Autodesk Inventor, you need to establish connection between Autodesk Inventor and Infor PLM Discrete. The connection gives you access to the Infor PLM Discrete CE database and projects that you need to work with.

To connect to Infor PLM Discrete:

From the Autodesk Inventor application, do one of the following:


- Click the **Connect** icon  in the PLM integration toolbar.
- Select **Connect** in the PLM drop-down menu. If the PLM client is already connected, an automatic silent connection occurs.

The functions in the PLM menu and toolbar are enabled. When you open another Autodesk Inventor application, it will automatically have an enabled PLM toolbar and menu ready to use.

Disconnecting from the PLM integration

Disconnecting is a global operation for all Autodesk Inventor integration applications; disconnecting from one Autodesk Inventor application disconnects all connected Autodesk Inventor application from Infor PLM Discrete.

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

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

Chapter 4: Save Files to PLM

The Save to PLM process saves your Autodesk Inventor file in the PLM 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 PLM.

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

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.

Saving an existing file to PLM also saves the file in Autodesk Inventor. If you save the file only to Autodesk Inventor, 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.

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 25.

Saving a file to PLM

Before you can save a file to PLM, you must save it in Autodesk Inventor with a unique name. If the file is not saved in your local disk, the Save As dialog box prompts you to save the file, with specific save as type, such as *.ipt, *.iam, or *.idw.

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

- Select **Save to PLM** from the PLM menu.

Clicking **Save to PLM** in the in one of the following:

- If all the IDs are generated automatically by PLM, which you can specify in the preferences, the file is saved.
- If you select either **Specify Manual ID for Documents** or **Specify Manual ID for Items** preferences, to override the system-generated IDs for documents and items, the CAD Integration - Save to PLM dialog box is displayed.
- In the CAD Integration - Save to PLM dialog box, you can insert IDs for the documents and/or items that are generated when saving the Autodesk Inventor file to PLM Server.

CAD Integration - Save to PLM - Commands

- **Save**

Saves the meta data details that are displayed in the local system. By default the files are saved in the **Edit Location** defined in the integration parameters.

Whenever you click **Save**, PLM creates separate .txt files for: Item, Document and File metadata. The file names are in the following format:

- ITEM_<RootFilename.extn_ Metadata_<date>_<time>.txt
- DOC_<RootFilename.extn_ Metadata_<date>_<time>.txt
- FILE_<RootFilename.extn_ Metadata_<date>_<time>.txt

Saving an assembly to PLM

During the Save to PLM process, the Integration considers the Item type and the Quantity values defined for each component in the BOM table of an Assembly, in the Inventor application. Otherwise, the behavior is normal. The functionality is explained in the following sections.

Phantom Item

When a part/assembly is marked as Phantom or Reference, that part/assembly is not included in the BOM structure (Item structure) in PLM. The Document and Item are created for the part/assembly. If the part/assembly is marked as Phantom, the corresponding Item's "attribute" Phantom is set to Yes, in PLM.

Consider the assembly with three parts.

In the BOM table, one of the parts is set to Phantom, second to Reference and the third to Normal. Documents are created for all the components.

Items are created for Phantom/Reference components also, but they are not included in the BOM structure (Item structure).

The component set to Phantom has the Item "attribute" Phantom set to Yes. Quantity

Quantity of an item is retrieved from the BOM table of the Inventor Assembly and stored in PLM. The quantity for each component can be specified in the BOM table.

In the Item structure for the above assembly, the quantity is set appropriately.

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 Setting preferences.

- 1 Select **Save to PLM** from the PLM menu. As a result, the CAD Integration -Save to PLM dialog box is displayed. The item ID fields have no values since your preferences are set up to create a "document" only.
- 2 If prompted, complete the "document" ID information and click **OK**. As a result, a "document" is created in PLM, but no items are created.

Saving a file to PLM and link the saved 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 Setting preferences.

- 1 Select **Save to PLM** from the PLM menu. As a result, the CAD Integration - Save to PLM dialog box is displayed. The item ID fields are not accessible since your preferences are set up to create a "document" only.
- 2 In the **Add Item** section, select **Existing**.
- 3 Enter the item ID and revision number for an item that currently exists in the PLM database.
- 4 Click **OK**. The saved "document" is linked to the item that you specified.

Handling assemblies

If an assembly is having a part P1 from two different locations for two different occurrences, then the **Save to PLM** process will popup a message asking user to use the part from the same path for both occurrences and the process will stop. User must correct the CAD files and perform the **Save to PLM** process.

Note: AutoDesk Inventor provides a facility to suppress the components of an assembly that are not needed to reduce memory consumption as well as simplify the modeling environment. You can save the representation with a name and activate it for modeling tasks, or select it for creating drawings and presentations. We the can save the representation of the suppressed component giving it a proper name. You can choose among several level of detail representations when you open a new or updated assembly. In the Open dialog box, click the Options button to select a system-defined level of detail representation: Master (the default state of the assembly), All Parts Suppressed, All Components Suppressed, or All Content Center Suppressed. Please note that PLM- AutoDesk Inventor Integration Vault operation Menu Options will be enabled only when the active level of representation is "Master".

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. The following preferences control the behaviour of the Download Manager:

Download Additional Drawings

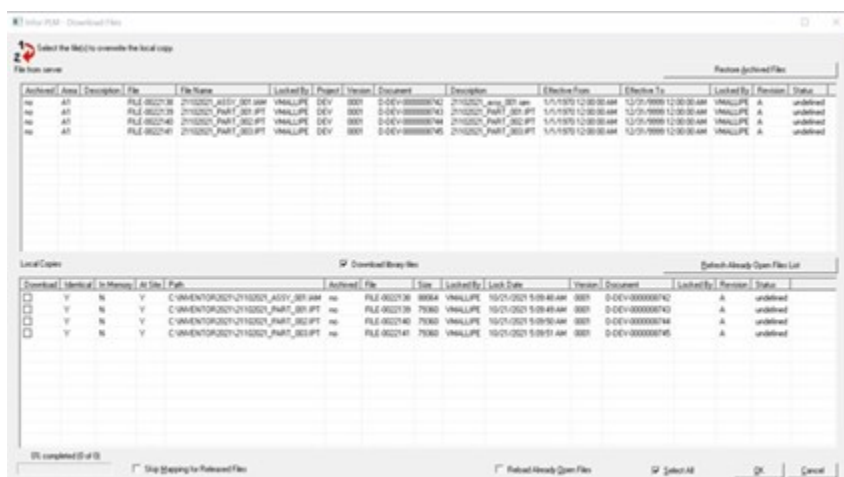
If this check box is selected, in the [General Category](#) on page 89 of PLM preferences, the **Edit/View file in Integration option** enables the user to download the related drawings of all the models in the current Document structure.

Indicator for Locally Changed Files in Download Manager dialog

If this check box is selected, in the [General Category](#) on page 89 of PLM preferences, 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 Category](#) on page 89 of PLM preferences, 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** in integration 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 Category](#) on page 89 of PLM preferences, the download manager indicates the changes if any, in the PLM database.

The Download Manager indicates if there is any change in the PLM data of Document/ Item/File.

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

Download of Large Structures

The Download of Large Structures category in the **PLM Integration Preferences** dialog box provides the following preferences:

- Resolve missing Content Center files during Download.
- Skip replace references of component files During Download.
- Update CAD structure which are out of date with respect to their driving entities.
- Skip Save of root file after Download.
- Defer automatic updates to drawings during Download.
- Skip CAD and PLM structure comparison.

Resolve missing Content Center files during Download

If a Content Center file is downloaded from PLM using the Edit File option instead of the Edit File in Integration process, used in an assembly, and the assembly is saved to PLM, the assembly is linked to the Content Center file with FamilyName_FileName.ipt name. During the Edit File in Integration process, the assembly may display these content center files as Unresolved, because these files are moved to Content Center library location, and file FamilyName_FileName.ipt is no longer available.

If this preference is selected during the Edit File in Integration process, PLM identifies all the missing content center files in the assemblies and includes the actual files from Content Center library.

To improve the performance, do not select this preference when Content Center files are downloaded using the Edit File in Integration process.

Skip replace references of component files During Download

When you download a structure from PLM during the Edit File in Integration process, without selecting this preference, PLM updates the reference links (pointing to the files in original location) with the new links referring to the files in download directory.

If this preference is selected, PLM does not update the reference links. As long as there is only one copy of a file in the current Inventor project and the work directory and Edit/View directory are the same, you can select this preference to improve the performance during the Edit File in Integration process.

Update CAD structure which are out of date with respect to their driving entities

If this preference is checked during Edit File in Integration, Integration updates all the files that are outdated with respect to their driving entities, in the structure. If this preference is not selected, PLM does not update the files.

Skip Save of root file after Download

If this preference is not selected, at the end of “Edit File in Integration” process, after root file is opened in Inventor, PLM Integration saves the root file if it is dirty. If this preference is checked during, Integration does not perform save on root file even if is the file is dirty.

Defer automatic updates to drawings during Download

When a drawing is opened in Inventor, if drawing is not up-to-date with the associated model, Inventor automatically updates the drawing and this process can take long time to open drawing file.

If this preference is checked, at the end of Edit File in Integration process, integration opens the drawing by deferring updates to it. Since drawing is opened by deferring updates, the Defer Update icon appears in the browser after drawing file is opened in Inventor.

To Enable automatic updates in a drawing the following steps need to be performed.

- 1 With the drawing active, click **Tools** tab Options panel Document Settings to access the Document Settings dialog box.
- 2 On the **Drawing** tab of the Document Settings dialog box, clear the **Defer Updates** check box.
- 3 Click **OK** to close the Document Settings dialog box.


Skip CAD and PLM structure comparison

If this preference is checked, as part of Edit File in Integration process, Integration compares local CAD structure and PLM structure to check if there are any differences between both the structures. A structure in PLM can vary from the CAD structure, as a result of Copy Product Structure.

If this preference is not selected, PLM integration does not compare CAD and PLM structures and are assumed to be same.

Chapter 6: Link to item

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

To link the part file to an existing item in the PLM database, In the Infor PLM Discrete , perform Select Source on an item. In Inventor, click this icon  , the Link to Item dialog box appears. In the **Set Object Properties** dialog, click (...) to paste the item that was selected in the Infor PLM Discrete. Click **Ok**.

If you select the **Manual** or **Auto ID** check box and leave the **Item ID:** field blank, an item and an itemID 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.

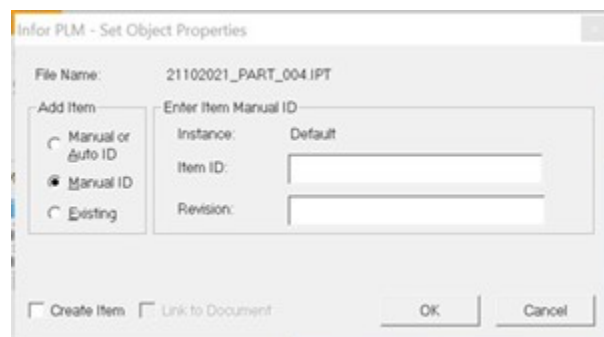
If you select the **Manual ID** radio button, you must enter an item ID in the Item ID: field. As a result, PLM generates an item with the ID that you entered in the Item ID: field.

If you select the **Existing** radio button, in the **Item ID:** field, you can enter or browse for 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.

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.



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**.

When you click **OK**, the following confirmation message is displayed:



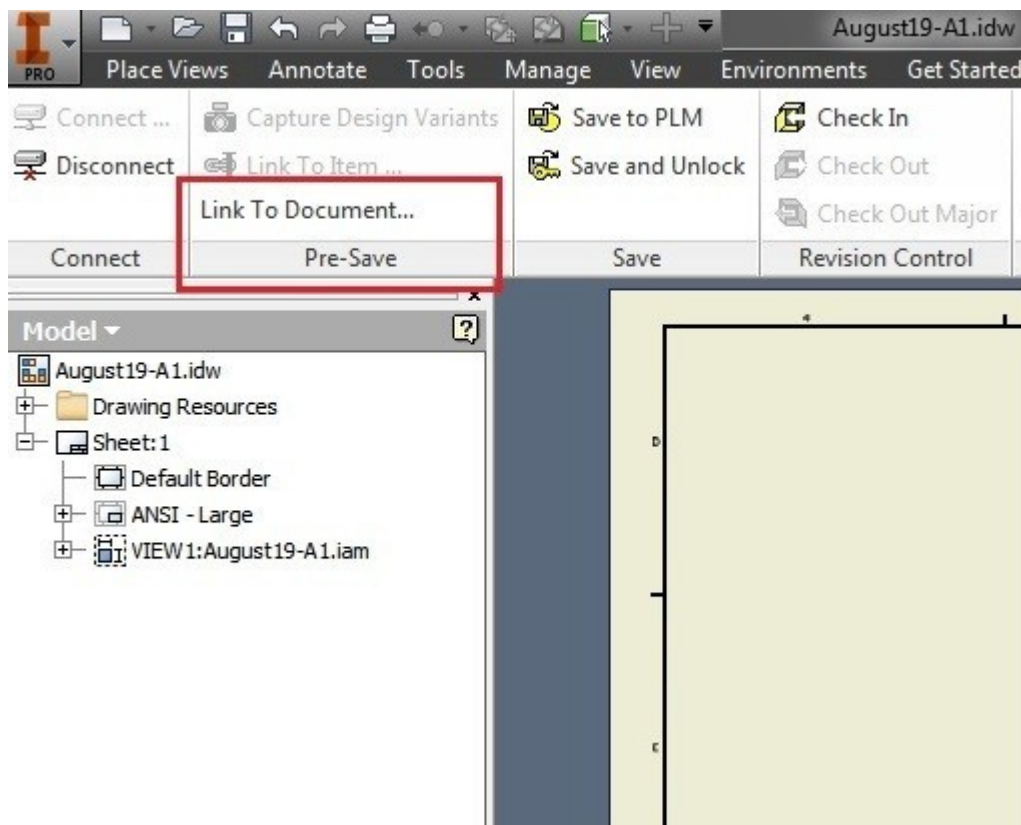
When the current file is already in PLM and when you click **Link to Item**, the **Link to Document** check box is selected indicating that when user clicks **OK**, the item is linked to this document in PLM, without the need to perform **Save To PLM** operation. The **Link to Document** check box is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

In **Link to Document** also, the user is asked to confirm this linking.

Chapter 7: Link to Document

To link existing document:

- In the Infor PLM Discrete, perform Select Source on any document.
- In the Inventor, on the **integration** menu, click **Link To Document** option.



- In the **Set Object Properties** dialog, click (...) to paste the document that was selected in the Infor PLM Discrete.
- Click **OK**. Save to PLM should be performed after this step for the file to be linked to this document and transferred to PLM.
- Perform **Save to PLM** for the file to be linked to this document.

Note: You can use this option for the files that are not saved to PLM.


Note: You cannot link new files to RELEASED Documents.

Chapter 8: Saving and Unlocking Files

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

If you have finished working on a file and the file must be passed on to an unknown new user, you should also use the **Save and Unlock** command. The new user who needs to work on the file can now acquire ownership of the file. If you know who will be the new owner of the file, you can transfer ownership to the new owner. See [Changing ownership of a file](#) on page 31.

To save and unlock a file:

- Do one of the following:
 - Click **Save and Unlock**  in the PLM toolbar.
 - Select **Save and Unlock** from the PLM menu.

The file is saved in PLM with no owner.

To save the file again to PLM, user must perform the take ownership task.

- 1** Select **Take Ownership of file** from the **PLM** menu.
The PLM - Take Ownership List dialog box is displayed.
- 2** Select the required file and click **OK**.

Chapter 9: Checking-in a File

When you have finished working with a file, it can be released. The PLM - Autodesk Inventor Integration enables you to check in any Autodesk Inventor file and its linked documents to the PLM vault.


The check-in operation accomplishes the following:

- Confirms the changes you made in the Autodesk Inventor file.
- Changes the file's status from Draft to RELEASED in PLM.
- Transfers the Autodesk Inventor file to the PLM released area from the PLM work area.

The integration verifies that you have authorization to perform this operation and that the "document" linked to the file has the Draft status. If the file is new, the system prompts you with the Save As dialog box to save the file locally and perform the check-in operation. You can only access this file in Autodesk Inventor via the PLM Integration Query tool. See [PLM integration query tool](#).

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

To check in a file:

- 1 Do one of the following:
 - Click  in the PLM toolbar.
 - Select **Check In** from the PLM drop-down menu.
 - 2 If prompted, save the file to Autodesk Inventor first. The file is checked in.
- You can specify how to handle files after they have been checked in. See [Setting preferences](#) on page 87.

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:

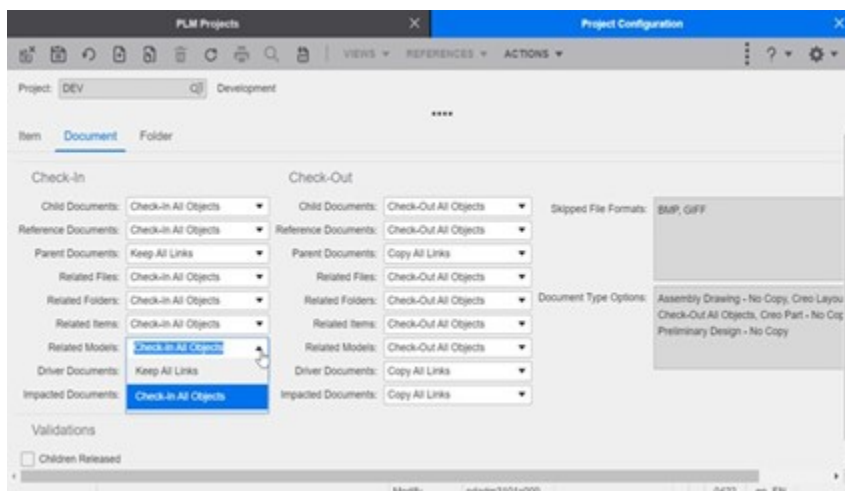
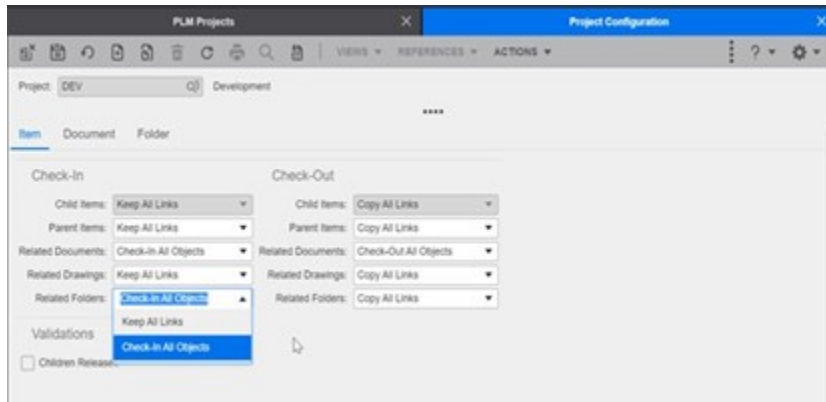
- a Checks out the parent assembly: ASM1.
- b Checks out and changes the child of the sub-assembly: ASM3.
- c Checks in the parent assembly: ASM1

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 as shown below:



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



When user clicks **OK**, the drawing files missing on the local machine are checked-in and their status is set to **RELEASED**.

To terminate the check-In process click **No** and then click **Cancel** on the PLM - Select Properties to Save To PLM dialog box.

Chapter 10: Checking out a file

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

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

In the **CAD Parameters** tab of the **Web PLM Preference** pane, you can specify how PLM will handle reference documents and/or items that are linked to the file that you are checking out.

Two types of check out mechanism 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:

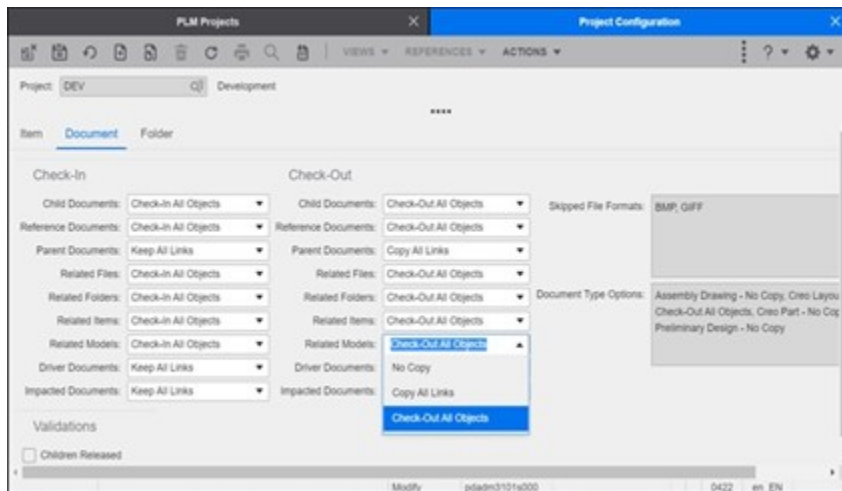
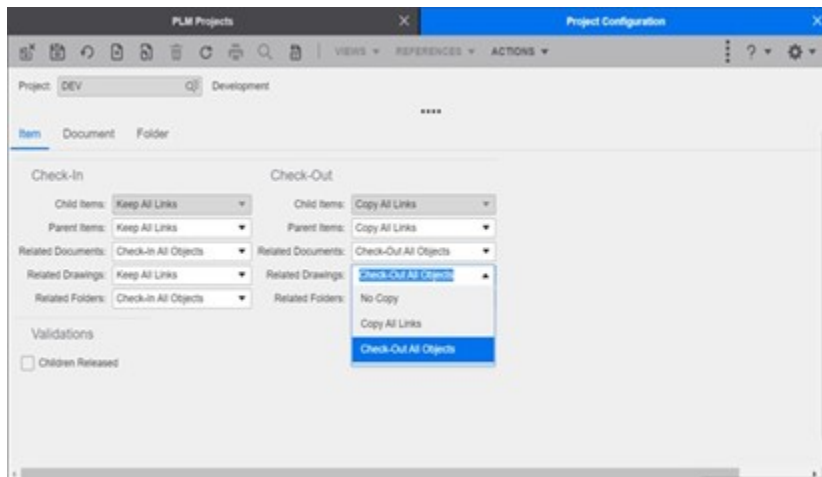
- 1 In Autodesk Inventor, select **Query** from the **PLM** menu and in the query tool that opens, run a search for the file you want to check out.
- 2 In the query **Results** panel, select the required file and click **Edit File in Integration** in the right-click menu. The file is opened in Autodesk Inventor.
- 3 Select the assembly or component part that you want to check out and select **Check Out** icon  from:
 - The PLM toolbar
 - 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 as shown below:




Note: Limitations

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

Chapter 11: 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 Autodesk Inventor Feature Manager, select the part of which you want to take ownership.
- 2 Do one of the following:
 - Click  in the PLM toolbar.
 - In the **PLM** menu, select **Take Ownership**.

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

Note: When **Take Ownership during Edit File 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.

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 PLM.
- The user who checked it out of PLM.
- The user to whom the ownership has been transferred.
- The user who performs the **Edit File in Integration** 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 "project".

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 Autodesk Inventor, for the part file with which you are working, select **Change Ownership** in **PLM** menu or toolbar.
- 2 In the **Select User** dialog box that appears, select the user you want to transfer the ownership to and click **OK**. The selected user now owns the file.

Chapter 12: Viewing PLM Data

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

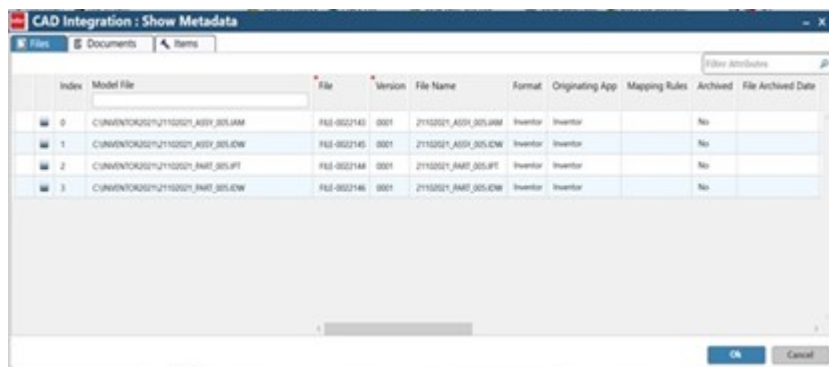
In Autodesk Inventor, to view the relevant PLM data, proceed as follows:

- 1 On the **PLM** menu, select **Show Metadata**.

The Attributes for File-Documents-Items Objects dialog box is displayed.

The default tab, File, displays the PLM data of all the files that are part of your Autodesk Inventor structure.

- 2 Click the **Documents** tab to display the PLM document data related to your Autodesk Inventor part file.
- 3 Click the **Items** tab to display the PLM Item data related to your Autodesk Inventor files.



Attributes for File-Documents-Items - Commands

- Save

Saves the meta data details that are displayed in the local system. By default the files are saved in the **Edit Location** defined in the integration parameters.

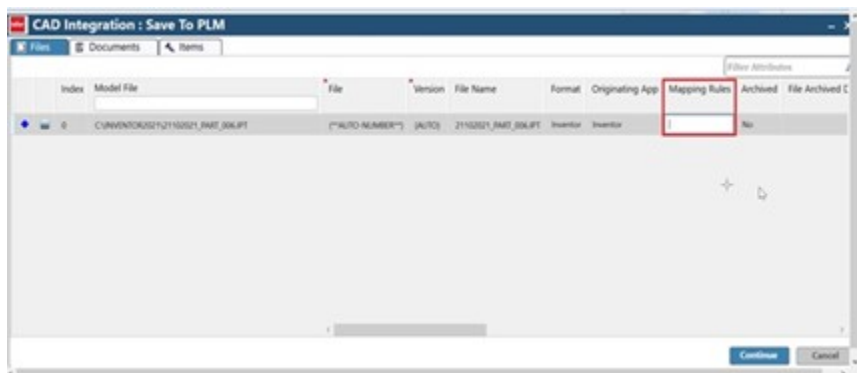
Whenever you click **Save**, PLM creates separate .txt files for: Item, Document and File metadata. The file names are in the following format:

- ITEM_<RootFilename.extn_ Metadata_<date>_<time>.txt
- DOC_<RootFilename.extn_ Metadata_<date>_<time>.txt
- FILE_<RootFilename.extn_ Metadata_<date>_<time>.txt

Assigning Mapping Template

Mapping template can be associated to inventor file which is either already available in PDM or yet to be saved to PDM. This can be done in Show Meta Data Dialog as shown below.

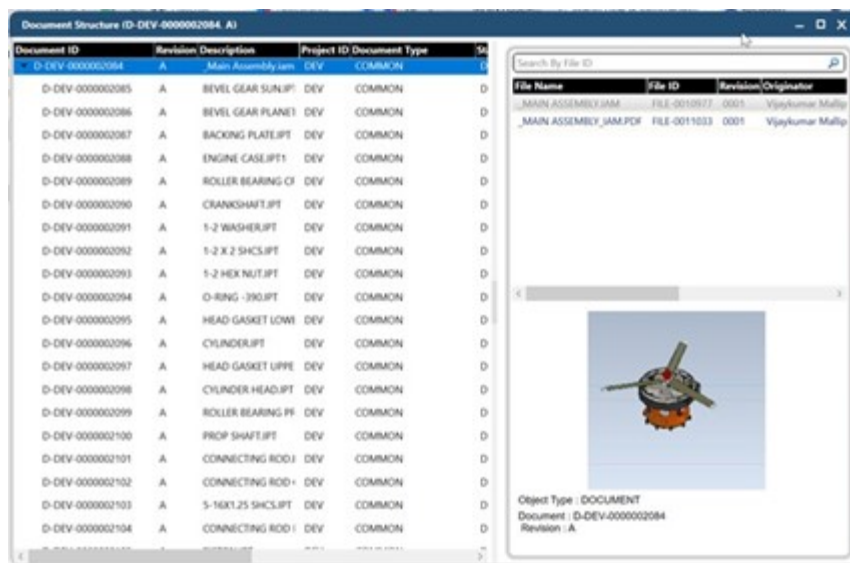
This can be done in the **Files** tab of the Show Meta Dialog. A text box is available to specify mapping template name.



Chapter 13: Opening a File in PLM Structure

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

- 1 Create a part file and save it to PLM. See [Saving to PLM](#) on page 16.
- 2 Select **Show Document Structure** or **Show Item Structure** in the PLM menu or toolbar.



Chapter 14: Batch Save Process

The utility saves files from the specified Inventor project (.ipj) directory and its subdirectories, to PLM.

- 1** Before you Register Files, ensure that only PLM is open. When the Register Files process starts, Inventor opens automatically with the input project as the active project and connects to WebPLM.
- 2** Set the preferences **Check In cleanup** and **Save and Unlock Cleanup** to **Make Files Read-Only**.
- 3** For drawings the document ID is generated using File Name. If a document ID already exists in PLM with the same name as a drawing file being saved by batch utility, a new auto-number is assigned to the document ID of the drawing being saved.
- 4** Currently neutral files (PDF and DWF) are supported only for Drawings. For other files (parts/assemblies) neutral files are not supported. Uncheck DWF preference for drawings if DWFs are not required.
- 5** Thumbnails can be generated for all types of files.
- 6** Set integration preferences as required. For example, if no items are required for the files, select the preference **Create Documents Only**.
- 7** All library files/ content center files will also be saved if used in an assembly/ drawing. Any library file/content center file that is not used in an assembly/ drawing will be saved only if its location falls under the Inventor .ipj directory or one of its subdirectories.
- 8** All the iAssembly members/ iPart instance files, available under the cache folders are saved to PLM during batch save process.
- 9** If Create Document Object for Presentation File is selected, saves a presentation file under its own document. In this case all the presentation files available under the input project directory will be saved to PLM as part of the batch save process.

If the Create Document Object for Presentation File is not selected, only those presentation files that are used in a drawing are saved to PLM. In this scenario a presentation file is linked to the same document as that of the drawing in which it is used. It means that a presentation file not used in any drawing is not saved to PLM as part of batch save process.

Save process

The save process comprises of two section:

- Generate Input Data
- Register Files

Generate Input Data

This action generates an input file that contains information about the Inventor files under the specified Inventor project directory and its subfolders.

It is not required to open the Inventor at this stage.

To generate the input file:

- Specify Inventor project (.ipj) file path.
- Click **Generate Input data**.

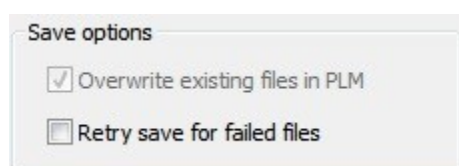
Register Files

This action saves the files to PLM. The list of files to save to PLM is read from the input file generated during Generate input data.

To save files to PLM:

- Ensure Inventor is not open.
- Specify the Inventor project path. This should be the same project that was selected to generate input file during the step Generate input data.
- Click **Login**. Inventor opens and the input Inventor project is set as the active project in Inventor.
- If the connection to PLM exists Inventor automatically connects to PLM or Login to PLM from Inventor.
- Specify the mapping templates for different file types.
- Select **Save and unlock** or **Check in**.
- In **File types** select the type of files to save to PLM.
- Set the Retry save for failed files option as required.
- Click **Register Files** to start the save operation.

Save Options



- If **Overwrite existing files in PLM** is selected and a file exists in PLM with the same name as that of a file being saved to PLM as part the current batch save operation, the file in PLM will be overwritten if the PLM file is editable by current user. This option cannot be modified.
- If **Retry save for failed files** is not selected, batch utility will not retry to save the files that were failed to save during earlier batch save operation.

During the batch save operation the following files are generated under the directory %CFE_CLIENT_ HOME%\Inventor\BatchLogs.

BatchGen.Input.xml - Input file containing list of files to save, generated during “Generate input data.”

BatchGen.Output.xml - This file is generated during “Register Files” action and contains the information on the save status of the files.

In case of any issues provide above two xml files and the following log files.

- %tmp%\toolkit
- %tmp%\logwininet
- %tmp%\Toolkit\InventorPLMAdd\«current date» folder
- %CFE_CLIENT_HOME%\Inventor\Batchlogs

Chapter 15: Using Infocards

An infocard is a dialog box that enables you to update metadata of items, documents or files generated from part files created in Autodesk Inventor. There are three types of infocard, one for items, one for documents, and one for files. You must use info card to make changes to the item, "document" or file in Autodesk Inventor 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 Autodesk Inventor object that you are working on, proceed as follows:

- 1 From the PLM main menu, select the relevant update option. You can select **Update Item**, **Update Document** or **Update File**. As a result, the relevant update dialog box is displayed.
- 2 Make the required changes.
- 3 Click **Update**.

Note: To update a component of an assembly in Autodesk Inventor, right -click on the component in the browser bar and select the **Open** option.

To open the infocard, select **Update Item**, **Update Document** or **Update File** from the PLM menu.

To view your changes to the item, "document" or file, you can open the appropriate workspace. For further information, refer to [Opening a file in an PLM workspace](#) on page 34.

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 16: Synchronize PLM Info

You can synchronize a working PLM file with Autodesk Inventor.

There exists few scenarios that result in PLM information for the local file to be desynchronized. For example:

- If a file, which exists in PLM, is copied to a local workstation from another location in the network (instead of being opened from PLM) or
- If a new file is created that uses the name of a file that already exists in PLM or
- If the status of file located both locally and in PLM (changes in PLM) due to, for example: A manual or WorkFlow Check In/Out action. Not having authorization to a WorkFlow step. Change project action, leading to a permission change for the file. Change Ownership operation performed by another user.
- If a file becomes desynchronized, (depending on the exact de-synchronization) the user may receive warnings during the PLM Save process and may have to cancel the save process and rectify the problem before continuing with the PLM Save process. Alternatively, if users are aware that de-synchronization has occurred, this option can be used to manually re-synchronize and correct any problems that maybe highlighted.

To Synchronize PLM Information:

- 1** Select **Synchronize PLM Info** option from the PLM menu. The PLM - Inventor Integration dialog box appears.**
- 2** Click **Yes** to continue, else click **No** to cancel the synchronization process.

Note: ** For an open file in Inventor, the synchronization is done for that file and all the components. For an open file, the synchronization process does not ask any question whether to continue or cancel the process. It by default, process the synchronization.

But, if no file is opened in Inventor, the synchronization happens for all the files listed in the local storage files. This command is enabled when the user is connected to PLM.

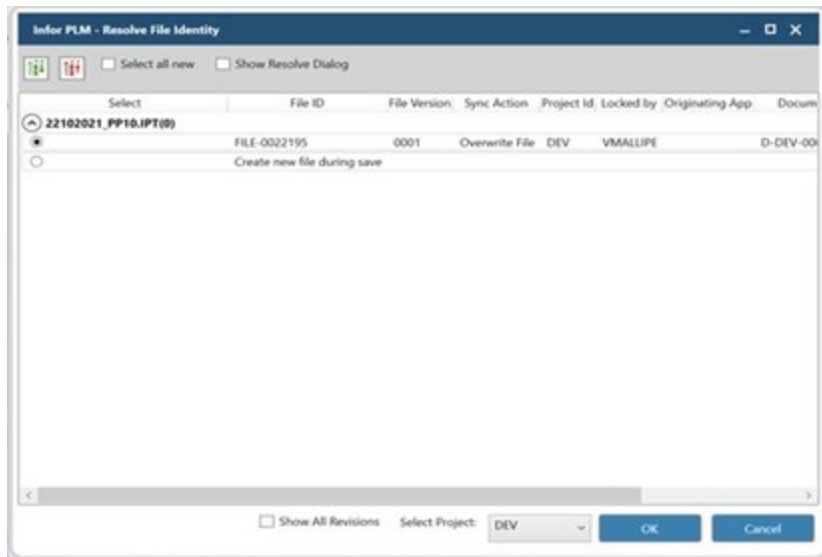
Synchronize headers

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

If you are using PLM data to be displayed in your title box drawings, **Synchronize Headers** retrieves the latest data to be displayed. If you receive a file from an external developer, and you must replace the PLM file with

the new file, you can use **Synchronize Headers** to identify the local file. In this case the PLM file will be overwritten by the local file while saving to PLM.

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



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

File ID	The name of the file.
Version	The version of the file.
File Name	The name of the file.
Overwrite File	The permission to overwrite the file.
Project ID	The ID of the project in which the file is saved presently.
Locked By	The ID of the entity which locked the file.
Originating App	The application which created the file.
Document ID	The ID of the document linked to the application.
Revision	The present revision number of the file.
Status	The present status of the file.
Description	The description of the file.
Effective From	The date from which the file is effective.
Effective To	The date to which the file is effective.
Business Process ID	The business process ID of the file.

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

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

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

Chapter 17: Clear Storage Information

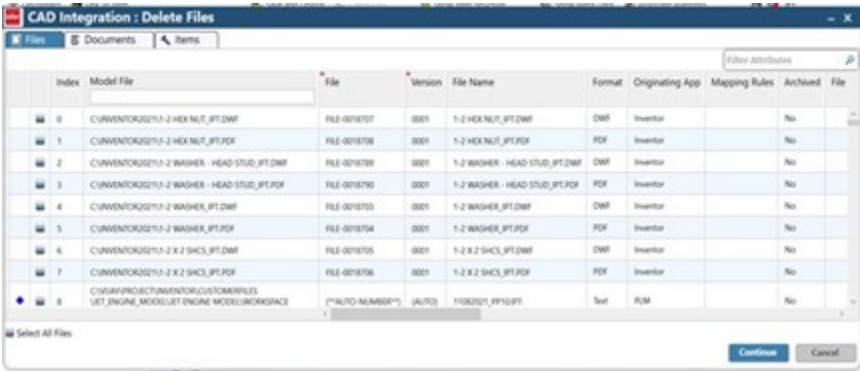
The **Clear Storage Information** functionality is designed to enable the user to quickly save the active/selected part to a new project.

When you click **Clear Storage Information**, integration clears the data for the active/selected file and its dependents from the storage data tables. The Infor PLM Discrete treats the part as new.

Chapter 18: Deleting 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.

When you click the **Delete Local Files**, the following screen is displayed:



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

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

Chapter 19: Capturing and Saving Design Variants

The Capture Design Variants option enables you to automatically or manually link items to "design variant" or configurations for parts. This option is available in the PLM menu.

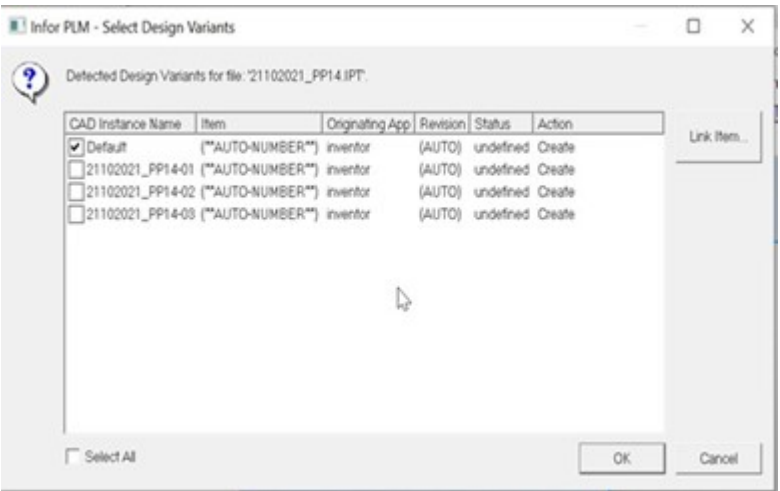
Managing design variants

The menu item **Capture Design Variants** is enabled only for iPart factory and iPart instances. The **Capture Design Variants** operation is applicable when:

- You create a new item, or
- use existing items

Selecting design variants

- 1 Select **Capture Design Variants** from the PLM menu. The **PLM -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** to create the new items.
- 4 Save the file to PLM to link the items to the part. For further information, see [Saving to PLM](#) on page 16.

When you save to PLM, links are formed between the "document" and the items created by means of the **Capture Design Variants** and **Link to Item** 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.

CAD "instance" name for a configuration of a Part or Assembly should not be given as "Default". Because this name is given to the Part Factory file when Capture Design Variants is performed

Each item's CAD instance name in PLM matches the name used in Autodesk Inventor. If you change the CAD instance name for an item in Autodesk Inventor that will be treated as new "instance" and it has to be captured before save to PLM. So, once a variant is captured, user should not change it's file name.

Note: The option Capture Design Variants is enabled only for ipart factory and ipart instance files.

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.

Linking new items to design variants

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

- 1 Select **Capture Design Variants** from the PLM drop down menu. The PLM-Select Design Variants dialog box is displayed.
- 2 Select a specific row of design variants you want to link to item.
- 3 Click **Link Items**. The PLM - Set Object Properties dialog box is displayed.
The behavior of the **PLM - Set Object Properties** dialog is determined by the Manual Id for Items preference. If the Manual ID is selected, the user will be able to enter values for the **Item ID** and **Revision** fields while the radio button is set to New as shown in the above figure.
- 4 Select the **New** radio button, and enter the required item ID and revision.
Note that if the **Manual ID for Items** option is selected in your preferences, the Item ID and Revision fields are available. For further information, see [Setting Preferences](#) on page 87.
- 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 PLM-Select Design Variants dialog box.
- 8 Save the file to PLM. See Saving to PLM.

Linking existing items to a design variant

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

- 1 Select **Capture Design Variants** from the PLM menu. The PLM -Select Design Variants dialog box is displayed.
- 2 Select a specific row of "design variant" s you want to link to item.
- 3 Click **Link Item**. The Add Design Variants dialog box is displayed.
- 4 Select 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 PLM -Select Design Variants dialog box.
- 8 Save the file to PLM. See [Saving to PLM](#) on page 16.

Viewing the design variants in a document structure

To view the "design variant" s in a document workspace, proceed as follows:

- 1 Select **Open in Document Workspace** from the PLM menu.
- 2 In PLM, the document workspace is displayed.
- 3 Click the **Items** tab in the **Document Properties** section of the workspace.

As a result, the items that were created and linked to the "document" as part of the "design variant" process are displayed. You can create any number of "design variant" s for the same item.

Note: After capture the "design variant" s, user must save the "design variant" s to PLM to link the captured variants items to the "document".

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 Inventor file, to a PLM business process. As a result, PLM items and/or documents of Inventor file are linked to a workflow, which is distributed to the users associated to the workflow template.

Dispatching documents and/or items to a business process

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

- 1 Select **PLM > Preferences**; the Integration Properties for Integration is displayed.
- 2 In the **Attached to Business Process** field located under the **Attached to Workflow** category, select the PLM objects that can be dispatched to the business process.
- 3 Click **OK**.
- 4 In the Inventor 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**.

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 Inventor application, save the file to PLM.
- 5 Click **Dispatch to Business Process** on the PLM menu. The Infor PLM Inventor Integration dialog box is displayed.
- 6 Click **OK**.
- 7 To save and unlock the file, click **Yes**. Otherwise, click **No**.

Example

Part11.ipt is the model file and Part11.idw is the drawing file associated with Part11.ipt. 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.

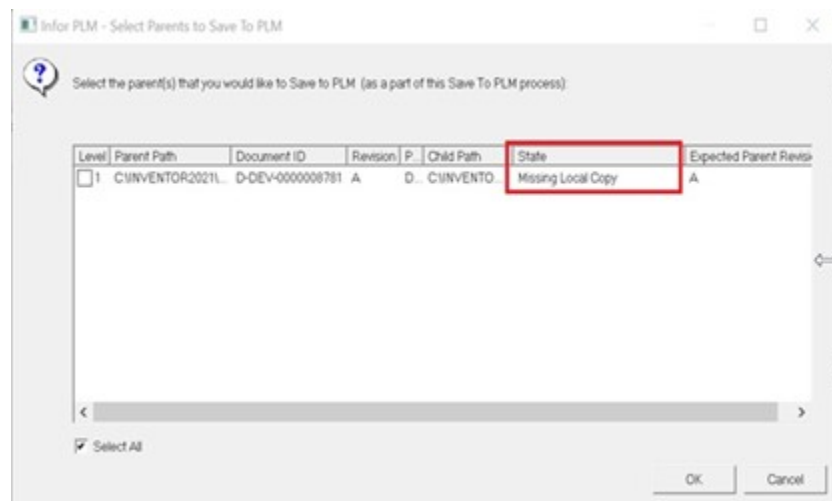
Note:

- 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 Inventor, before you dispatch the model file to business process.

Troubleshooting Scenarios

The drawing file is missing on the local system

If you dispatch a model file to a business process when the related drawings are deleted from the local system, the following warning is displayed:



Drawing file not open in Inventor Application

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

Dispatch to BP – Multilevel

When you perform the Dispatch to BP operation on the selected file and the Attach All Related Objects preference is enabled, it is checked whether the selected file exists in PLM and is with the status Draft.

The dialog box contains the following information:

- Files that are within the structure.
- Files that must be unlocked.
- Possible problems in performing Save and Unlock operation on the files.

Note: If the file does not exist in PLM or the status of the file is not Draft, the process is stopped.

The Information column of the above dialog box can have the following values:

- **Missing Locally**
The file is missing locally.
- **Version Mismatch**
The version of the local file differs from version of file indicated in the grid.
- **Dependent File [Missing / Mismatch]**
Child file(s) of the current structure, which is either Missing Locally or has a Version Mismatch.

The Requires Locking column can have the following values:

- **Yes**
The file is locked by the current user and must be unlocked.
- **Locked by Others**
The file is locked by other user.

Click **OK**, the list of files locked by current user and therefore must be unlocked, are displayed. If one or more files are returned in the above step (for performing Save and Unlock operation), then add-in performs Save and Unlock operation on these files.

If Save and Unlock operation fails, you can either continue with creating the business process or can stop the business process.

If you continue the business process, the integration looks for all the files that are Draft and are UNLOCKED, and their corresponding document-id (DOC.DOC_KEY + DOC.REVISION) and associated items-ids (ITEM.ITEM_KEY + ITEM.REVISION) are read. If no objects are obtained, the process is stopped.

When the business process is created and is launched, you are indicated of the progress.

Chapter 21: Save Modified Files Only

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

Example

In the following structure:

```
MA.IAM
|--> SA1.IAM
    |--> P1.IPT
    |--> P2.IPT
|--> SA2.IAM
    |--> P3.IPT
    |--> P4.IPT
```

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

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

```
MA.IAM (changed)
|--> SA1.IAM (changed)
    |--> P1.IPT (changed)
    |--> P2.IPT
|--> SA2.IAM
    |--> P3.IPT
    |--> P4.IPT
```

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

Important Points & Limitations

- The **Save only modified files** preference is applicable only for Save to PLM operation and is not applicable for Save and Unlock and Check-In operations. The reason being that if the files contain mapping, then

even if the files are unchanged, still the Save and Unlock and the Check-In operations might modify the files as a result of the mapping definitions.

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

Integration Preferences to Save Changes to PLM

Use the **Save Changes to PLM** property to control the aspects related to saving the Autodesk Inventor files to PLM.

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

- Save only modified files
- Warn if the selected document is changed but not locally saved
- Do not traverse child files under non-modified files

Save only modified files

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

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

Do not traverse child files under non-modified files

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

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

Example

Consider below structure:

```
A1.IAM (changed)
|--> A2.IAM (changed)
    |--> A3.IAM
        |--> P4.IPT
```

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

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

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

Example

```
A5.IAM (CHANGED)
|--> A6.IAM
    |--> P7.IPT
    |--> P8.IPT
```

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

Chapter 22: General mapping

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

Example

In Autodesk Inventor, create template Template_1. For Template_1, define the following mapping rule: Autodesk Inventor "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.

Creating the template

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

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

- 1 In Autodesk Inventor, select **File > New** to create a new template file.
- 2 Specify the "attribute" properties to be mapped to PLM for the template. To set the "attribute" go to **File > iProperties > Custom option**.
- 3 Click **OK** to save the template under template folder in Autodesk Inventor.
- 4 Open the template in the Infor PLM Discrete. This is described in [Opening the template in the Web PLM](#) on page 54.

Note: To change the properties of a template, open the template file, adjust the properties, and save the template file. The template files created for parts and assembly can also be used for iParts and iAssemblies, respectively.

Opening the template in the Infor PLM Discrete

- 1 In the Infor PLM Discrete, go to **Administration > Integration Setup**.
The PLM Mapping Tool screen is displayed.
- 2 Select **Autodesk Inventor** as the application.
- 3 In the **Template** list, select the template for which you want to define mapping rules.
If the template is new or if changes were made to the template in the CAD application, you must import the template into PLM first. For further information, see [Importing the template into PLM](#) on page 54.
Any previously defined mapping rules for the selected template are displayed in the dialog box.

Importing the template into PLM

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

To import a template:

- 1 From the **Application** list in the upper left of the PLM Mapping Tool dialog box, select **Autodesk Inventor**.
- 2 Click **Import**.
- 3 In the Open dialog box that appears, select the template for which you want to define mapping rules and click **Open**. As a result, the template properties are displayed in the upper right section of the **PLM Mapping Tool** dialog box.

Defining the mapping rules

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

To map Autodesk Inventor attributes to PLM attributes and vice versa

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

- 1 Click the **Item** icon on the left side of the PLM Mapping Tool dialog box. A list of item attributes is displayed. The dialog box below provides an illustration of this process.
- 2 From the list of item attributes, click **Description** from the list.
- 3 From the Autodesk Inventor side of the window, click **Summary**. A list of attributes is displayed.
- 4 Click **Author** from the list of attributes.
- 5 Click the **To PLM** button in the middle section of the dialog box. This maps the value of the **Author** "attribute" to the **Description** "attribute" of the item in PLM. You can add more mapping rules before accepting them. See Mapping options for further information on the available mapping options.

- 6 When you are through specifying mapping rules, click **Apply**.
- 7 Click **OK** to exit the PLM Mapping Tool dialog box.
- 8 For the mapping rules to take effect, in the PLM client, disconnect and then reconnect.

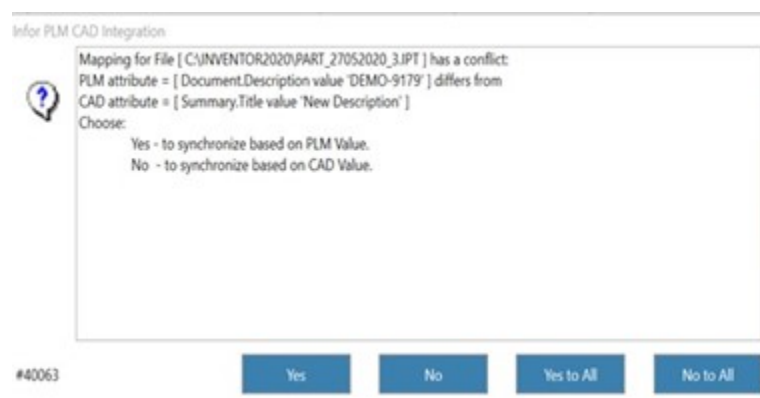
Mapping options

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

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

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

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



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

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 Autodesk Inventor "attribute" to one PLM "attribute", one Autodesk Inventor attribute to many PLM attributes, or many Autodesk Inventor attributes to one PLM "attribute".

For example, if you mapped a second Autodesk Inventor "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 Autodesk Inventor separated by a comma.

Mapping restrictions

Depending on the attributes selected in the PLM Mapping Tool dialog box, mapping to PLM, to Autodesk Inventor, 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, Autodesk Inventor, and to both. For all mapping rules that you want to define, the target "attribute" must be a modifiable field.

PLM

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

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

Table 1: Attribute formats

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

+ (*) According to regional settings.

The target "attribute" must be of type string if more than one "attribute" is mapped to this "attribute". The target PLM "attribute" to which Autodesk Inventor 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 Autodesk Inventor file-specific rule can have no other general mapping.

Relationships

You can map one Autodesk Inventor "attribute" to one PLM "attribute", one Autodesk Inventor "attribute" to many PLM attributes, or many Autodesk Inventor 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 Autodesk Inventor 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 the mapping rules to part files, assemblies, or drawings:

- 1 In Autodesk Inventor, open a template file for which you have created mapping rules 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

Alternatively, you can use the **Associate Mapping Rules** option from the PLM menu to associate mapping rules to files. Note that you can only use this option for files not associated to a mapping template previously by opening the template and saving the file under another name.

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

- 1 In Autodesk Inventor, open the file for which you want to associate mapping rules.
- 2 Click **PLM > Associate Mapping Rules**. The PLM Mapping Tool dialog box is displayed.
- 3 From the **Template** list, select a template you created in Autodesk Inventor. 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.

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

Removing mapping associations

You can remove the rules that are associated with the template. For this purpose, select **Remove Association** from the PLM menu. The owner of the file or "project" administrators who own the file can remove an association.

Display mapping

You can use the **Display Mapping** option in the PLM menu to view the general 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 Infor PLM Discrete as described previously, you must first disconnect and reconnect your CAD application to PLM.

The mapping then takes place depending on the rules that you set up. In the example in [Defining the mapping rules](#) on page 54, the mapping takes place when you:

- Save to PLM
- If the file is new and has not been saved to PLM, you must save the file to PLM to apply the mapping. Once the mapping is applied, you can select Update Item, Update Document or Update File from the PLM menu to see the mapped values. For further information, see [Using Infocard Option](#) on page 38.
- To edit the files, use the info card options: Update Item, Update Document or Update File.

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 other mapping rules, the "attribute" mapping is omitted without displaying a warning message.

Chapter 23: Neutral Files

Use this option to specify an additional format to save the local Autodesk Inventor 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 Autodesk Inventor files:

- Parts
- Assemblies
- Drawings
- iParts

Following properties are relevant to the neutral files:

- Generate Neutral Files
- Show Neutral files during save
- Generate Neutral File during

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.

For more information, refer to [Integration Preferences for Neutral Files](#) on page 92.

Chapter 24: Presentation Files

Presentation files provide a multi-dimensional view of an assembly. You can use the presentation files to develop exploded view, animations, and other stylized views of an assembly to help "document" your design; where as a drawing, provides only a static view of the assembly. Presentation views are saved in a separate file called a presentation file (.ipn). Each presentation file can contain as many presentation views as needed for a specified assembly. When changes are made to an assembly, the presentation views are updated automatically. A presentation can contain only a single assembly.

Presentation files can be saved into PLM, in the following ways:

- Presentation file as neutral file to the drawing
- Presentation file as separate document

Presentation file as neutral file to the drawing:

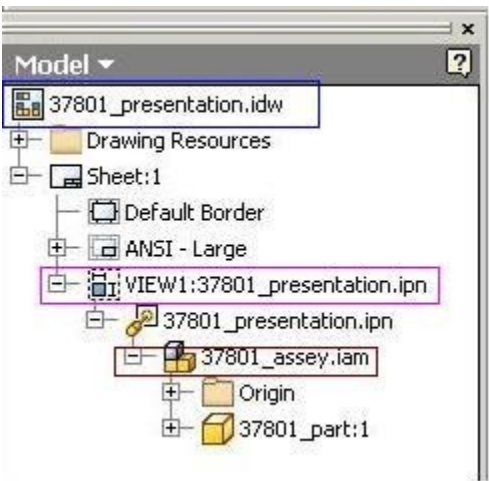


Table 2: Legend

Blue color	Drawing
Pink color	Presentation file
Brown color	Assembly file

For example, in the above figure, 37801_presentation.idw is a drawing, 37801_presentation.ipn is a presentation and 37801_assey.iam is the assembly linked to the presentation.

The assembly in the *.ipn will be linked as the child of the drawing which holds the ipn file. The .ipn file is saved as a neutral file to the drawing.

Document object for the presentation files

To create a document object for the presentation file, the Create Document Object for Presentation file preference must be enabled. When the preference is enabled and the presentation file is saved to PLM, a separate document is created for this file in PLM. Items are not created for the presentation files in PLM. Presentation document is saved as a parent to the assembly document in the PLM structure. If this presentation is used in a drawing, in the PLM structure Presentation document is saved as child to the drawing document.

Chapter 25: Thumbnails

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

Thumbnails can be generated in the following ways:

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


Generating Thumbnails

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

- 1 In the Autodesk Inventor application, select **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.
- 3 Click **OK**.
- 4 Click **Save to PLM** on the toolbar.

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

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

- a Open the Inventor file.
- b Click  on the toolbar.

Thumbnail Locations

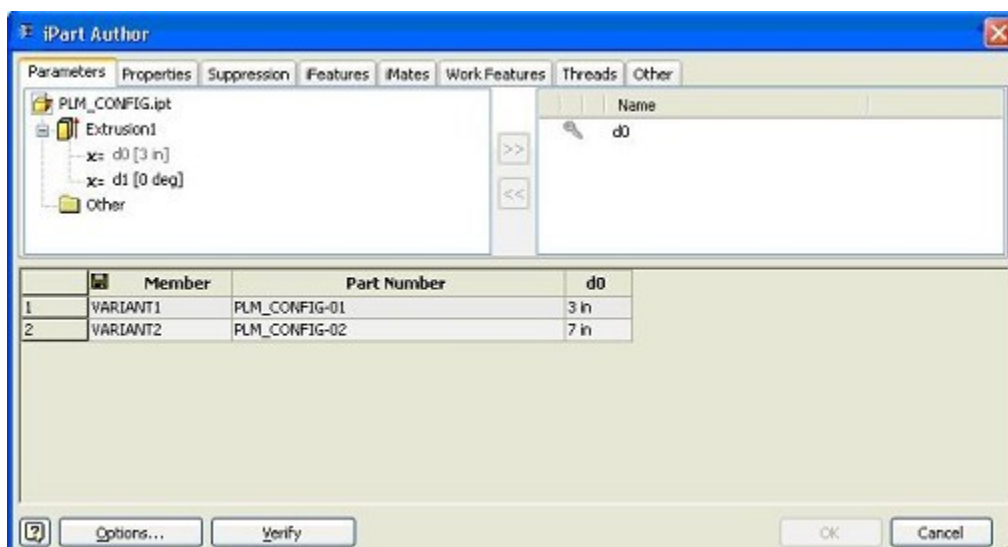
By default, thumbnails are saved in the %CFE_CLIENT_HOME%\inventor\Temp folder. To change the folder, modify the variable value of the PLM_THUMBNAIL_GEN_DIR environment variable.

Thumbnails for Item Configurations

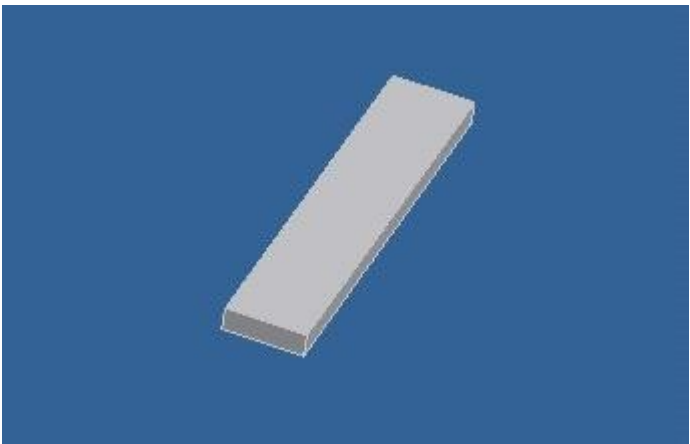
Thumbnails are generated for the default configuration, that is for the originating items, when the design variants are not captured. The thumbnails are linked to the document.

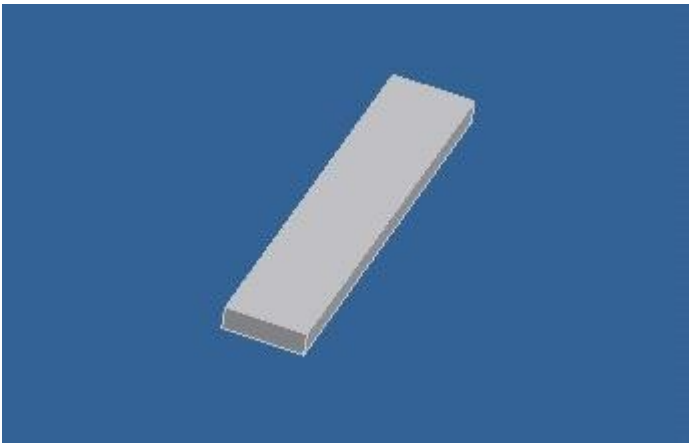
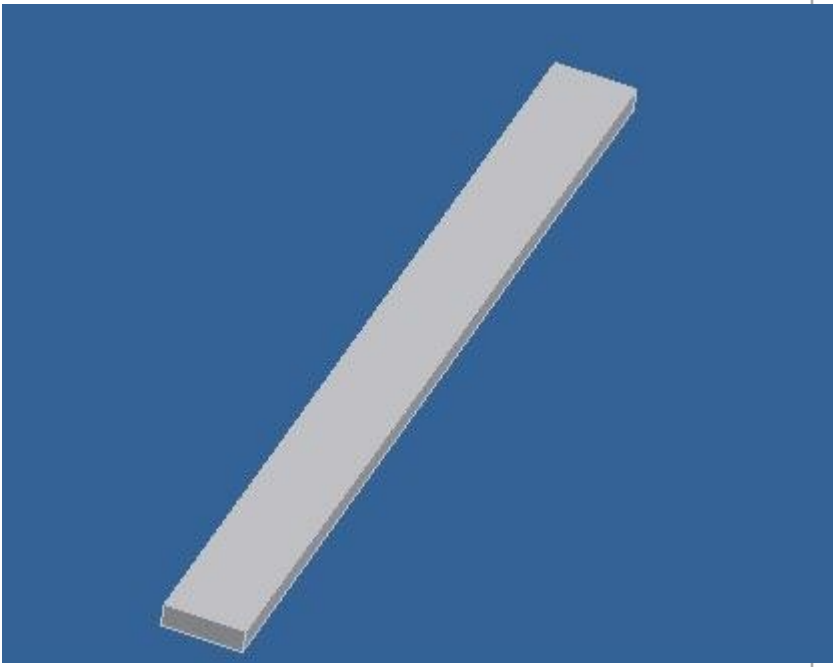
Example

In the following configuration PLM_CONFIG.ipt the default part configuration and Variant 1 and Variant2 are the design variants.



The following table indicates Inventor parts linked to various design variants of the configuration.

Inventor Configuration	Image
Default[PLM_Config]	

Inventor Configuration	Image
Variant1[PLM_Config]	
Variant 2 [PLM_Config]	

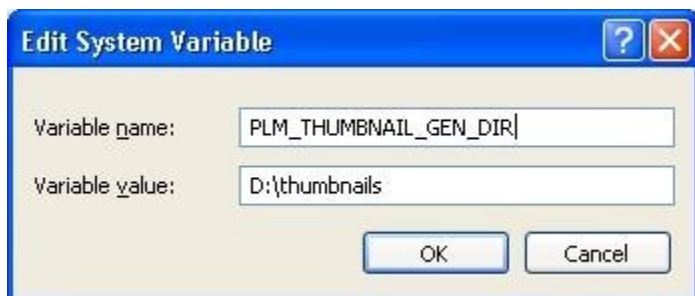
When the design variants are captured for the configuration and the corresponding instance files are generated, thumbnails of all the variants are attached to the document and to individual items.

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

The thumbnails are generated in the %CFE_CLIENT_HOME%\inventor\temp\\ folder. However you can specify a new location; to specify a new location, change the value of the PLM_THUMBNAIL_GEN_DIR variable, as shown below:



Integration Preferences for Thumbnails

The properties comprising the **Thumbnails** control the following aspects:

- Autodesk Inventor 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 Autodesk Inventor files are saved to PLM. This option generates thumbnails only for the editable components of a Autodesk Inventor 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 Autodesk Inventor part files, that is, files with .ipt extension.

Generate thumbnails for assembly file types:

If this check box is selected, thumbnails are generated for the Autodesk Inventor assembly files, that is, files with .iam extension.

Generate thumbnails for drawing file types:

If this check box is selected, thumbnails are generated for the Autodesk Inventor drawing files, that is, files with .idw extension.

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

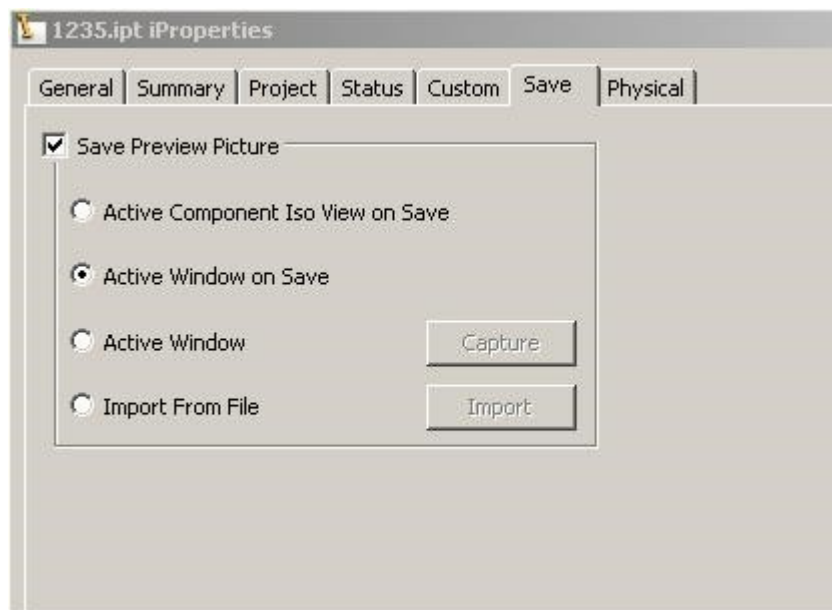
Missing Thumbnail Images

If the integration preferences are set to generate thumbnails and if the thumbnails images are missing on the inventor files, then during save to PLM, integration will create the thumbnail images on inventor and this will have impact on the performance of save to PLM process.

It is observed that to create thumbnail images for 4 inventor files, over all save to PLM process time is increased by about 60 seconds. To avoid this users can manually create the missing thumbnails on the inventor files.

To generate the thumbnails that are missing for an Inventor file, complete the following steps:

- 1 Open each Inventor file for which thumbnail images are missing and access the iProperties dialog.



- 2 Select the **Active Window on Save** option.
- 3 Click **Apply**.

- 4 Click **OK**.
- 5 Save file locally in Inventor. A thumbnail preview is generated in Inventor (e.g. From “fileopen” dialog).
- 6 On the PLM menu, click **Save to PLM** or **Generate thumbnails**.

Chapter 26: Balloon Mapping

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

Constraint

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

Solution

During **Save to PLM**, integration reads the Ballooning information defined on the active sheet of the drawing for the assembly and transfers the values to the PLM.

Integration Preferences for Balloon Mapping

Use the Balloon Mapping property to control the transfer of balloon numbers from AutoDesk Inventor 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

Transfer Ballooning Information:

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

Field in Part List table:

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

Warn for duplicate Balloon IDs:

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

Warn for incorrect Ballooning:

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

Chapter 27: Content Center

Content center files are standard library components generated from the content center database.

Saving Content Center files

The save process of any "content center" file goes through the following phases:

- For any of the **Save to PLM** operations, the system checks if the file is a "content center" file and located under the content center folder pointed to by the current "project".
- If the file is determined as a content center file, the file name given in PLM will be the actual file name with a prefix made up of the directory path location of the file in the Content Center.

Example

For example: The following "content center" file: "C:\Documents and Settings\abeiser\My Documents\Inventor\Content Center Files\R10\en-US\DIN ISO 1481 - C\ST2.2-4. 5.ipt" will be saved as: "DIN ISO 1481 - C_ST2.2-4.5.ipt".

The content center files are treated as RO files, which you can not change in any case during the **Save to PLM** process.

Note: The "Delete on Check In" Integration preference does not apply to these files.

The files are considered as RO in PLM after the initial save. This implies that any subsequent changes and save of a "content center" file does not update the copy in PLM.

- There are two reasons for this limitation, the first is that theoretically these files never change, the second is to improve performance (i.e. we will not save files to PLM that never change).

Note: An item cannot be linked to a "content center" file if during the initial save, the Integration Preference "Create Documents Only" is set and File is Checked In. This is a limitation because Content Center Files cannot be checked-Out.

Attributes for File-Documents-Items Objects

The Attributes for File-Documents-Items Objects feature for "content center" files can be configured when you add any item or document manually or link to an existing item.

The possible scenarios are:

- During first time **Save to PLM** process, **Manual ID** and **Revision** creation will be supported for content center files.
- During first time **Save to PLM**, it will be possible to link existing items to content center files.

Note: The Attributes for File-Documents-Items objects dialog is controlled by the PLM Integration preferences dialog with the activated General category. If either of the “Specify Manual ID for Documents” or the “Specify Manual ID for Items” option is selected, the Attributes for File-Documents-Items objects dialog appears and assume that the model is not yet associated with this PLM information.

Impact of Rename

For the integration purpose, you can alter a "content center" file which is moved and renamed. Once the content center file is moved and renamed, that is no longer considered as a RO. Instead, it is treated as a standard inventor part file for the purposes of the integration.

Once the content center file is moved, you must rename the file locally, because the integration process depends on the internal file details, such as the original file name to determine if the file is a real content center file or not. If the internal file name and the local file name matches, then the integration considers the file to be a "content center" file.

Chapter 28: Derived Components

Derived parts are new parts that use an existing part or assembly as their base component.

You can add features to a derived part, which is considered as the base component, and update it to incorporate changes made to the original part or assembly.

Example

Examples of "derived parts" are:

- Mirror parts
- Joined parts

In addition, a base component can be an "instance" of an iFactory, which is basically an iPart.

Note: PLM integration supports a derived part from assembly or from a normal iPart. You can derive a whole part as well as a standalone part.

Working with Derived Parts

To work with derived components you must understand the following concepts:

- Saving derived parts to PLM
- Suppression and Break Link
- Opening a derived component

Saving a derived parts to PLM

Derived parts are represented in the PLM similar to assemblies. This representation is required to provide:

- Where Used, for one level in the **WorkSpace** tab, which will show both Parent Assembly /Drawing and Driven (Impacted) Parts.
- A report showing the impact of any change to a component on the complete hierarchy (i.e. all parent and/or all child parts).

Note: The integration will attempt to identify the unaccepted or unapplied pending changes to parts, assemblies or drawings (flagged by Inventor with a lightning bolt). If such cases can be identified, the following warning will appear:

One or more of the components in the structure you are saving has been impacted by a modification which has not been accepted. Click **OK** to continue with the Save Process or Cancel.

Example

The following picture shows the derived CAD parts only:

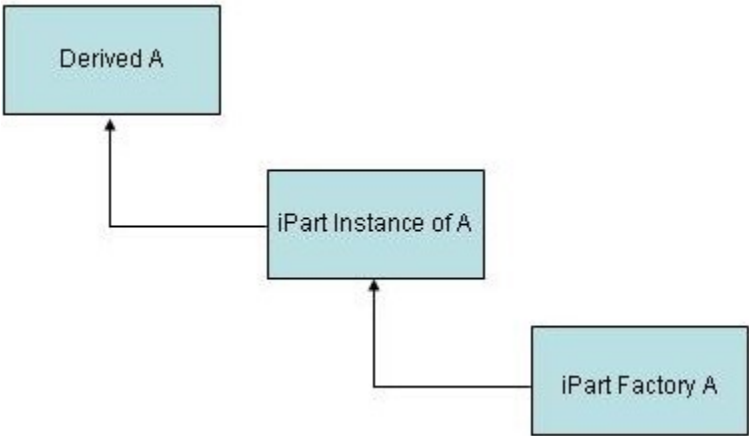


Table 3: Result of PLM structure

Document	Item
Document of Derived B	Item of Derived B
-----> Document of iPart Factory A	-----> Item of iPart Instance A

Document	Item
Document of Derived B ----->	-----> Item of Derived B

Example

Derived parts from CAD assembly

The following dialog box shows the "derived parts" from CAD assembly.

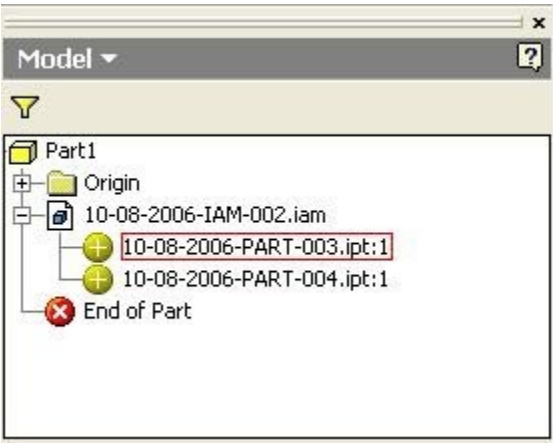


Table 4: Resulting PLM structure

Document	Item
Document of Derived Part1	Item of Derived Part1
----> Document of IAM-002	----> Item of IAM-002
----> Document of Part-003	----> Item of Part-003
----> Document of Part-004	----> Item of Part-004

Table 5: Resulting to PLM Logical Links

Document	Item
Document of Derived Part1	Item of Derived Part1
Document of IAM-002 >	Item of IAM-002 >
Document of Part-003 >	Item of Part-003 >
Document of Part-004 >	Item of Part-004 >

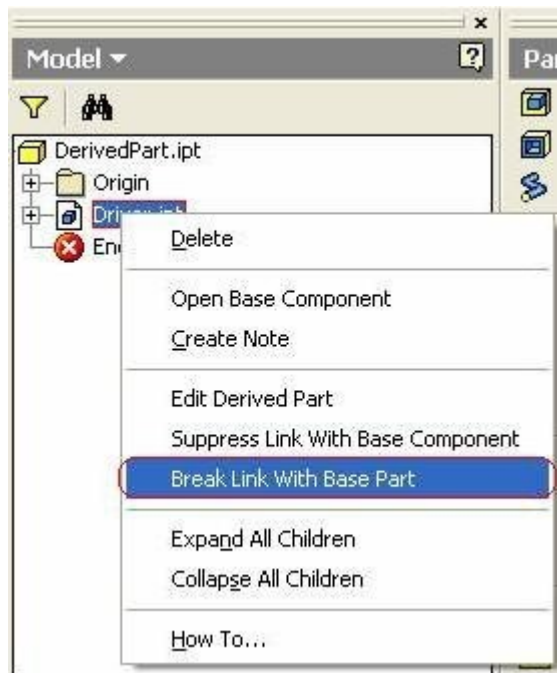
Derived Components and Suppression / Break Link

When you save a derived component in PLM, the integration will automatically save its driver in an identical manner that we do for a parent-child relationship.

The derived components behave as the parent and the driver as the child which implies that saving the derived component will cause a save to the child (if it's editable). The result of the parent-child link also implies that if you check-in the derived component, it's driver will also be automatically be checked in (as a result of the Vault Parameters).

To suppress or break the link for any derived component perform the following steps:

- 1 Create a derived component based on assembly from the Derived Component Tool.
- 2 Right click the assembly and select **Break Link With Base Assembly** option from the context menu as shown in the picture:



Note: While executing the "Break Link With Base Assembly", we do not even create the document with parent child relationship, and the parent child link will be remove once we save/re-save the Derived components. This signifies that in PLM no information about the existence to the link will remain even if it existed before breaking the link and re-saving to PLM.

Open a derived component

When you retrieve a derived (impacted) component, all components that drive it will also be retrieved.

Impact of Changes on Derived Parts

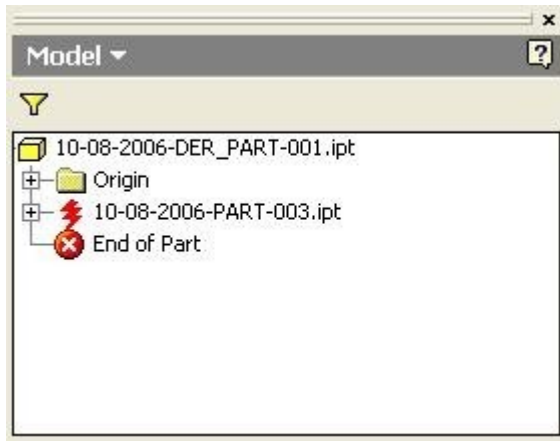
The following scenarios illustrates how the Inventor application behaves when changes are made to components from which "derived parts" are created.

To view the impact of changes for an assembly, perform the steps:

- 1 Create a derived component based on Assembly.
- 2 Close both the assembly and derived part.
- 3 Change the part in the assembly and save the changes locally.
- 4 Re-open the derived part.

Note: The derived part will open with the new option Update.

The assembly will be marked with a lightning bolt that shows that updates are pending for it.



- Opening local copies of files
- Opening files from PLM via “Edit/View file in Integration”

To view the impact of changes for a drawing file, perform the steps:

- a Create a drawing for the derived component.
- b Close the drawing.
- c Update the derived part. For example, increase or decrease the length.
- d Re-open the drawing.

Note: The integration will attempt to preserve this behavior (i.e. the integration will not automatically apply pending updates) both ways when:

- Opening local copies of files
- Opening files from PLM via “Edit/View file in Integration”

Chapter 29: Tools and Utilities

The PLM Integration for Inventor includes a utility that enables you to convert RELEASED Inventor files from previous Inventor versions to the current version of Inventor.

This utility downloads, convert and transfer the Inventor file back to PLM. This is done for each RELEASED inventor file in PLM.

During the conversion process, all the released Drawing/ Assembly files and components are downloaded, opened in Inventor, saved locally and then transferred back to PLM.

Software Configuration

The utility is part of the Inventor integration kit that is developed with Toolkit 404 and later. Download and install the integration kit.

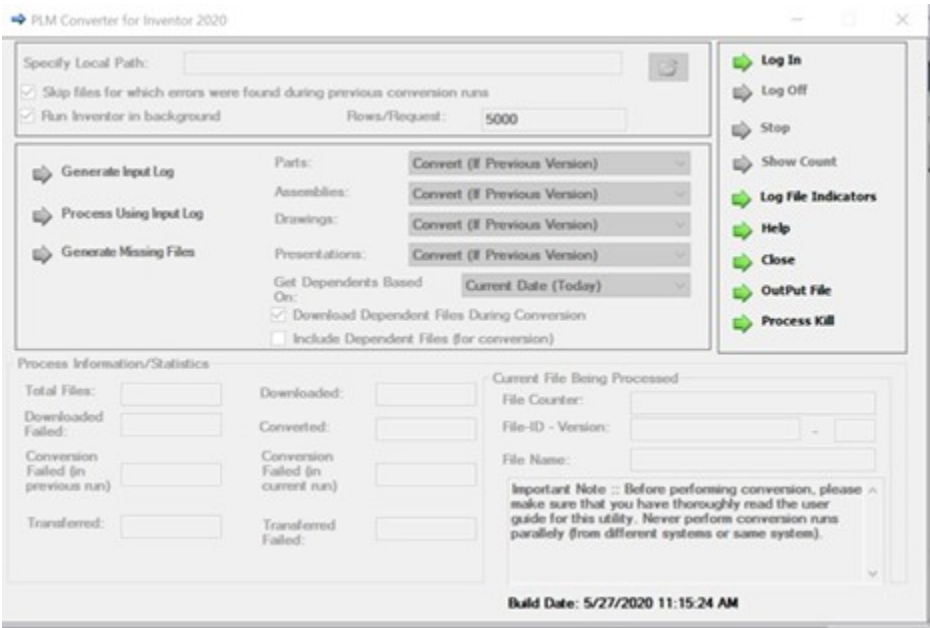
This utility is available from the inventor kit PLM8_Inventor2010_Toolkit402_239.zip onwards.

Integration Kit Installation Notes

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

Accessing the utility

To access the conversion utility, double-click the PLMInventorConverter.exe located at <PLM_Client>\Inventor; the following dialog box appears:

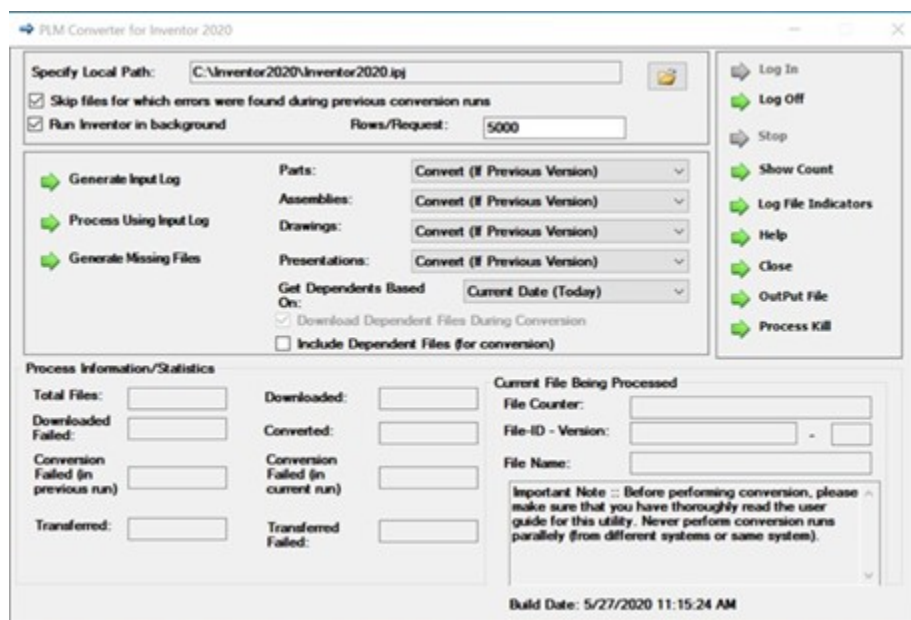


Login into PLM

To start the utility, click **Login**, on the PLM Converter for Inventor 2010 screen. The location specified in the **Specify Local Path** field is defaulted from the **Edit Location** field in the Integration Properties for Integration screen. However, you can change the location. If the location is used by all the users, it is recommended to retain the default value.

Note: Select the **All Projects Mode** check box when you log on to PLM.

On successful login, all the fields in the following screen are enabled:



Note:

- By default, **Run Inventor in background** check box is selected and enabled.
- By default, **Rows/Request** is set to 50000.

Conversion Process

- 1 Click **Generate Input Log** to create the log file in the following location:

<PLM_Client>\Inventor\ConvLogs\input.log

The entries listed in the file are in ascending order of FILE_ID + FILE_VERSION and are used for processing. The file contains entries for only RELEASED files.

- 2 Select the type of operation that must be performed for different file types. Three options are provided for Inventor Part, Assembly, and Drawing file types.
 - Convert (if previous version): Indicates that the utility must convert these file types only if the file is of earlier version.
 - Re-Convert All: Indicates that the utility must perform re-saving of these files (even if the files are in current version). The utility is used when the conversion run is not successful.
 - Skip: Indicates that the utility must skip these file types.

- 3 Download dependent files during conversion: If this check-box is checked. This indicates to the user that the utility will perform conversion for a file by downloading its dependent files also.
- 4 Click **Process Using Input Log** to convert the files that are listed in the input.log. During the processing of a file, the utility downloads that file along with its dependent files. The file is converted and if the conversion is successful, the file is transferred to PLM. During the file conversion, the utility also checks for dependent files in the list. If dependent files exist, the utility converts the files. If the conversion is successful, these dependent files are also transferred to PLM.

During this operation, the utility creates a log file in the following location:

```
<PLM_Client>\Inventor\ConvLogs\output.log
```

- 5 Include dependent files (for conversion): By default, this check-box is not selected. This flag is required only for specific cases. For example, if you want to convert a particular assembly(say FILE-ID = "FILE-01585", Version = "0003") and all its dependents under its structure, you can create a single entry (as below) representing the data set for this assembly in the <PLM_Client>\Inventor\ConvLogs\input.log file as FILE-01585_0003.IAM|0. If this check box is selected and you click Process Using Input Log, the utility searches for the dependent files (that are RELEASED) under the assembly structure, and also considers them (if required based on the above mentioned flags for various file types) for conversion.
- 6 Generate Error Log: During conversion process, status of files is listed in "output.log" file. When user clicks on this button, the utility will create following log file containing only the list of files for which there were errors during conversion operation. <PLM_Client>\Inventor\ConvLogs\ error.log

Restart the Conversion Process

If the process fails and/or the operation stops, run the utility again to resume the operation. The utility uses the following log files to determine the files for which the operations need to be processed:

```
<PLM_Client>\Inventor\ConvLogs\input.log
```

```
<PLM_Client>\\Inventor \ConvLogs\output.log
```

Before initiating a new conversion, ensure that there are no log files in the following folder:

```
<PLM_Client>\Inventor \ConvLogs\
```

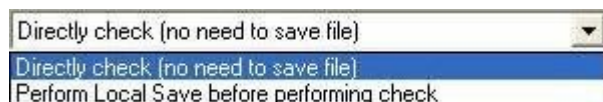
Pre-Conversion Step

Missing components in an assembly/drawing file can result in wrong conversion. To avoid this, perform a pre-conversion step to generate a list of files for which the first level components are missing.

Perform the following pre-conversion steps:

- 1 Click **Generate Input Log**.
- 2 Click **Generate missing child (first-level) log** The following Generate Report of MissingChild (first-level) screen is displayed.
- 3 Select the file-types you want to process.

Following are the options in the Combo-box:



- **Directly check (no need to save file):** Directly checks the first-level dependents, without performing local Save.
- **Perform Local Save before performing check:** Opens the file to save it locally, and checks for the first-level dependents.

Select the **Send Request** to get first-level child's only check box, to move only the first-level child's from PLM. If the check box is not selected, the utility moves the complete structure of the CAD file.

Click **OK** to check the missing first-level child references. The process can be stopped and resumed again, if required.

During the process, the following log file: "<PLM_Client>\\Inventor\\ConvLogs\\MissingChildsReport.log" is generated.

Example Following files are missing (in the first-level document-Tree) from the "FILE-00922_0001.IAM [050510-02-1.IAM]":

050510-02.IPT [D:\\edit\\050510-02.IPT]

050510-06.IAM [D:\\edit\\050510-06.IAM]

050510-14.IPT [D:\\edit\\050510-14.IPT]

The first column above displays only the file names and the second column displays the child path (as stored within the parent file).

Note: When you perform this operation for the first-time, ensure that the files under the "<PLM_Client>\\Inventor\\ConvLogs\\" directory are clear to avoid the possibility of a mix-up of results generated from the "Process Using Input log" and the "output.log".

Usage of Commands

The Converter utility comprises of various commands, as shown below. This section explains the commands and the functionality.

- **Login:** Connects to PLM when the All Projects Mode check box is selected.
- **Log off:** Disconnects from the PLM server.
- **Stop:** Stops the current operation.
- **Show count:** Shows the number of released Inventor files in PLM. Click Show Count.
- **Log file indicators:** Displays the meaning of different values in the log file.
- **OK:** Allows you to exit from the converter utility.

Processing Information

The following screen displays the processing information of the converter utility:

Specify Local Path: C:\Inventor2020\Inventor2020.apj

☒ Skip files for which errors were found during previous conversion runs

☒ Run Inventor in background Rows/Request: 5000

Generate Input Log

Process Using Input Log

Generate Missing Files

Parts: Convert (If Previous Version)

Assemblies: Convert (If Previous Version)

Drawings: Convert (If Previous Version)

Presentations: Convert (If Previous Version)

Get Dependents Based On: Current Date (Today)

☒ Download Dependent Files During Conversion

☐ Include Dependent Files (for conversion)

Log In

Log Off

Stop

Show Count

Log File Indicators

Help

Close

OutPut File

Process Kill

Process Information/Statistics

Total Files:	Downloaded:	Current File Being Processed
Downloaded Failed:	Converted:	File Counter:
Conversion Failed (in previous run)	Conversion Failed (in current run)	File-ID - Version:
Transferred:	Transferred Failed:	File Name:

Total number of RELEASED Inventor files in PLM is [347]

304 = Part Files (.ipt)

43 = Assembly Files (.iam)

0 = Drawing Files (.idw or .dwg)

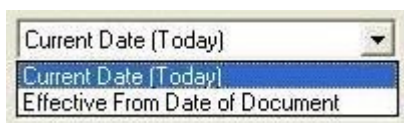
0 = Presentation Files (.ipn)

Build Date: 5/27/2020 11:15:24 AM

Recommended Settings for Conversion

This section describes the recommended setting for the conversion process. You can change the settings according to the requirements.

- **Run Inventor in background:** If this check box is selected, Inventor performance is not hindered during the conversion process. This setting improves the performance of the conversion process. In order to identify specific issues, you can clear this check box.
- **Rows/ Request:** The number of records that have to be fetched, per request. If there are a huge number of records in the PLM server, the request to fetch the records simultaneously, can fail. To avoid this scenario, the converter utility sends multiple requests. It is recommended to set this number to 50,000. For example, if the number of records in PLM is 160,000 and the value of Rows/Request is 50,000, the utility sends four requests during the generation of input log.
- For Part, Assembly, Drawing and Presentation files, the recommended setting is Convert (if previous version).
 - **Convert (if previous version):** Indicates that the utility must convert the file types only if the file is of earlier version.
 - **Re-Convert All:** Indicates that the utility must perform re-saving of the files (even if the files are in current version). The utility is used when the conversion run is not successful.
 - **Skip:** Indicates that the utility must skip the file types.
- **Include dependent files (for conversion):** By default, the check box is cleared. This checkbox is selected for specific cases. For example, if you want to convert a particular assembly(say FILE-ID = "FILE-01585", Version = "0003") and the dependents in the structure, you can create a single entry (as below) representing the data set for this assembly in the "<PLM_Client>\Inventor\ ConvLogs\input.log" file as FILE-01585_0003.IAM|0 file as FILE-01585_0003.SLDASM|0
- **Get Dependents Based on:** Specify the criteria based on which a request is sent for the structure. Following are the possible values:
 - **Current Date (Today):** Sends request using current date.
- **Effective from Date of document:** Sends request using Document Effective From date. This is the recommended option.



- Action for assembly/drawing files with missing references (at first-level) has the following options:
 - **Skip File:** The utility allows you to skip the assembly/drawing file, wherein the first-level dependents (as per information from CAD file) are missing.
 - **Convert File:** The utility does not check the existence of first-level dependents (as per information from CAD file), and directly converts the CAD file.
 - **Prompt (to Convert/Skip File):** If one of the first-level dependent files (as per information from CAD file) are missing, the utility provides the option to convert or skip the file.



Chapter 30: Package Assemblies

The Package Assembly feature is to allow a completely new assembly or drawing to be represented in PLM as a single object. This feature can be applied to new files which have never been stored in PLM database or files that are already flagged as package.

An option called Toggle Flagged as Package has been provided in the PLM menu and command bar to set the Package flag to "True" or "False" for the current CAD assembly or drawing files. You can link all the assembly and drawing files and their component to a root "document" using this feature.

Note: This feature is applicable to assemblies and drawing files.

Working with Package Assemblies

The package assembly feature is applicable to new assembly or drawing files which have never been stored in PLM database or files that are already flagged as package.

Functionality

The package assembly feature allows an assembly or drawing files to be represented in PLM as a single object. Only the root assembly or drawing is saved as a unique object with all components linked as files to the assembly or drawing document.

To save the assembly or drawing files in Infor PLM, perform the following steps:

- 1 Select **Preferences** from the PLM menu.
- 2 The Integration Properties For Integration dialog box appears. Click **General** preferences category and select the **Enable PLM Command 'Toggle Flagged as Package'** check box.
- 3 Click **OK**.
- 4 Execute the Toggle Flagged as Package operation and set the flag as **True**.
- 5 Select **Save to PLM**.

Only the assembly or drawings are saved as a unique object, with all its components linked as files to the assembly or drawing "document". The "document" 'Application Format Type' field will have a value of 'IN Package Assembly/Drawing'.

Example

The following picture shows an example how a package can be assembled, where only the assembly is saved as a unique object, with all components linked as files to the assembly "document".

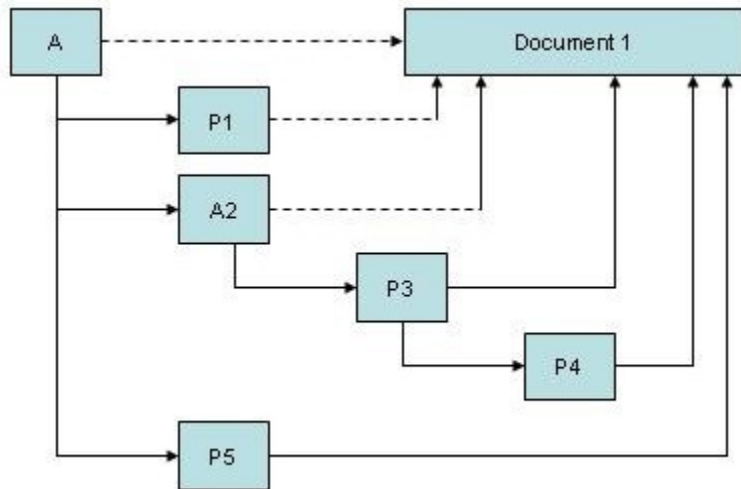


Table 6: Legend

A..	Assembly
P..	Part

Once an assembly or drawing file is saved to PLM as package assembly, user can not save the same file as normal assembly or drawing and vise-a-versa. If any assembly or drawing structure has a component file, which is already saved to PLM as a normal file, the system will not allow that assembly or drawing file to be saved as package assembly, unless the file is removed from the structure. The user get an error message showing that 'The file <File Path> cannot be stored as a package assembly because it already exists in PLM. To resolve this, you must remove the component file from the structure being stored or toggle the package assembly flag for the structure.

Note:

- The Toggle Flagged as Package option in the PLM menu controls the features of package assembly and drawing files. User can save an assembly or drawing file as a package assembly, if it is already saved as package assembly or if it is new. The limitation for package assembly is listed in the Limitations page.
- The content center files that are saved as normal CC files in PLM can be part of package assembly and Vice Versa, because the design of the CC file cannot be modified.

Chapter 31: iAssemblies

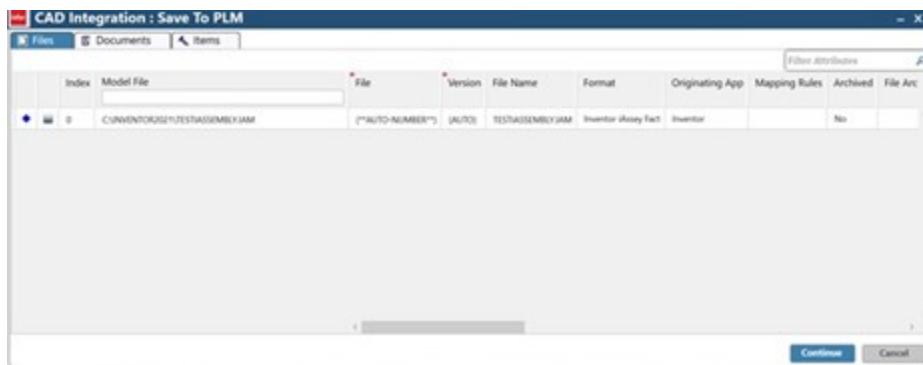
An iAssembly is a collection of various members and individual parts.



For example, in the above figure, **testiassembly.iam** is the factory and the entities under **Table** are the members.

Saving a factory to the PLM Server

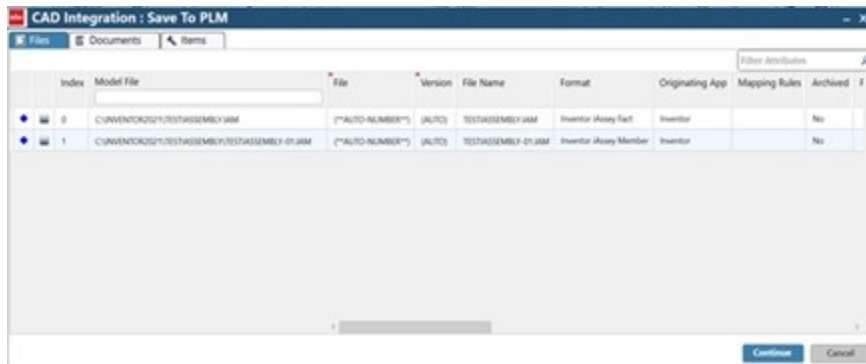
When you save a factory to the PLM Server, a document is created only for the factory and not for the members. Example when you save the **testiassembly.iam** factory to the PLM Server, the files are displayed as follows:



Documents are created only for the **testiassembly.iam** factory along with its neutral files; however, documents are not created for the members.

Saving a member to the PLM Server

When you save a member to the PLM Server, documents are created for the member and also for the factory. Example when you save the **testiassembly-01.iam** member to the PLM Server, the documents are displayed as follows:



Documents are created for the **testiassembly.iam** and **testiassembly-01.iam** factory along with the corresponding neutral files.

Note:

- If the members of the factory contain the child documents, the links are retained when you save the factory to the PLM Server.
- You cannot save a factory, that contains a member file only on the PLM server and not on the local machine; the following error message is displayed.



Chapter 32: Setting preferences

The PLM preferences control the way the PLM integration for Autodesk Inventor works. To access the PLM preferences, in Autodesk Inventor, click **Open PLM Inventor Integration Preferences** in the PLM toolbar. As a result, the PLM Integration Preferences dialog box appears.

The PLM Integration Preferences dialog box is comprised of the various properties grouped under following categories:

- [Locations Category](#) on page 88
- [Initial Save Category](#) on page 88
- [General Option](#) on page 99
- [Save Neutral Files Option](#) on page 59
- Save Neutral Files Parts
- Save Neutral Files Assembly
- Save Neutral Files Drawing
- Save Neutral Files iParts
- [Post Save Process](#) on page 95
- [Attach to Workflow/Business Process Option](#) on page 95
- [Troubleshooting Option](#) on page 97
- [Link to Item Dialog](#) on page 97
- Downloading Multiple Revisions of a File
- [Integration Preferences for Thumbnails](#) on page 51
- Set Item-ID During Initial Save
- [Integration Preferences for Balloon Mapping](#) on page 68
- Download of Large Structures
- [Integration Preferences to Save Changes to PLM](#) on page 51

In these tabs, you can perform the following:

- Specify whether IDs must be system-generated or manually entered.
- Specify neutral files to be created.
- Specify whether to create only documents, and not items.
- Specify whether to link a document to an existing item.
- Specify the location of folders in which to open files.
- Specify the location of folders in which to view files.
- Specify how to handle files after they have been checked in.

The admin level preference can be set only by the administrator, using the All Users Preferences pane. The user level preference, can be set by the individual users. The user level preferences are specific to each user,

that is, settings of a user do not affect the preferences of another user. However, the admin preference are common for all the users and cannot be changed by the users.

Note: The administrator can use the All Users Preferences to specify the Permission of a preference, that is, admin or user.

Locations Category

To carry out various commands, such as View File or Edit File from the object pane of the integration query, PLM must locally download the files that you want to view or edit in Autodesk Inventor.

In the **Locations** category, you can specify the folders in which PLM must locally download the files that you want to view or edit in Autodesk Inventor. If required, you can specify the same folder for both viewing and editing. If you do so, the **Disable PLM Commands** on “View in Integration” command will not work. For further information, see [Viewing files in the integration](#) and [Editing files in the integration](#).

The edit and view directory feature allows you to view and edit the selected directory of Autodesk Inventor file after it has been copied to the PLM vault during the check-in procedure.

The available options are:

- **Edit Location**
Select a file using the browse option to edit the directory.
- **View Location**
Select the browse option to view directory.

Note: The edit and view locations must be in the current inventor "workspace".

Note: You can also select the **Use Inventor Project Location** check box to set the Inventor project location as Edit and/or View directory.

Initial Save Category

Use the Initial Save Option category, to specify your preferences such as assigning manual IDs for items and objects, specifying object "attribute" when saving an item, and so on.

Following are the properties grouped under the Initial Save Option category:

- **Create Document Only**
If this check box is selected, a document is created in PLM, but no items are created when you save to PLM. For further information, see [Saving to PLM](#) on page 16.
- **Must assign an Item**
If this check box is selected, you must assign an item to the objects.
- **Set Object Attribute During Save**

If this check box is selected, you can specify or change the "attribute" of the objects when you save them. If you set this preference, the Attributes for Files-Documents- Items objects dialog box appears.

In the dialog box, you can specify the attributes such as, File IDs, Document IDs, Item IDs, description and so on.

- **Specify Manual ID for Items**

If this check box is selected, the PLM Options dialog box appears when a part file is saved to PLM for the first time. In the PLM Options dialog box, the user can manually enter item IDs.

- **Specify Manual ID for Documents**

If this check box is selected, the PLM Options dialog box appears when a part file is saved to PLM for the first time. In the PLM Options dialog box, the user can manually enter document IDs.

General Category

Use the **General** preferences category, to specify your preferences such as, default query such object,file name uniqueness, and so on.

Following are the properties grouped under the **General** category:

- **Allow Link to Released Items**

If this check box is selected, you can link the objects or files to the items, with Released status.

- **File Name Uniqueness**

Use this option to specify the file name uniqueness. Available options are:

- Unique across projects
- Unique per project
- Duplicate of filename is allowed

- **Disable actions on view files**

If this check box is selected, all the Update PLM options are disabled. To use the Disable PLM Commands on "View in Integration" option, the view and edit locations specified in the Locations tab must be different. For more information, refer to [Locations Category](#) on page 88. This option prevents the user from modifying the files, opened from the integration query results of the PLM client, using the View File option.

- **Query Search Default**

The default query search object displayed in the PLM Integration Query dialog. Allowed values:

- Doc
- File
- Item

- **Save Additional Drawings**

If this check box is selected, any additional drawings of a model in the current CAD structure, that match with the name of the model, are saved to the PLM.

When the drawing is saved to the PLM Server, a parent-child relationship is maintained between the drawing and the object.

- **Download Additional Drawings**

If this check box is selected, any additional drawings of a CAD model are copied from the PLM to the local system. When this preference is enabled, you can download the file along with its drawings, if any. The file and the drawings are copied to the edit location specified in the preferences.

- **Save All iPart Instances**

If this check box is selected, all the iPart "instance" files are saved to PLM. If this check box is cleared, only the iPart Factory file is saved to PLM and the "instance" files are re-generated when you perform the Edit File in Integration action.

Note: If a particular "instance" file exists on the local disk (iPart Factory cache directory) then that "instance" is not generated.

- **Search and Replace files from Content Center**

If this check box is selected, a standard "content center" of an assembly, replaces a non standard Content Center (CC) file, when you perform the Save to PLM operation.

Complete the following steps to search and replace files from CC:

- 1 For each component file which is an .ipt (that is part) and which is not a "content center" file from the "content center" library path, the Content Center (CC) library is searched for the same file name.
- 2 If a match is found in CC library, a message is shown to the user giving the path of the original file, and new file from the CC library file. User will have a choice to replace or continue with the existing file.
- 3 If the user does not want to replace the CC library file, then the .ipt file will be treated as a normal file.
- 4 If the user replaces the CC library file, then it will be treated as CC File.

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

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

- **Take Ownership During Edit File**

When this check box is not selected, the users do not take the ownership of the files even when they execute Edit File command.

- **Skip Mapping for Released Files during Download**

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 in integration operation. However, you should be aware that some file preferences may not be up to date with PLM.

- **Skip Meta Data Comparison During Download**

When this 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 CheckIn Specific Modified Mapping XML File**

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

- **Set CAD Instance Name as Item-ID**

If you select the **Set CAD Instance Name as Item-ID** check box, PLM sets the name of the CAD instance as the item ID.

- **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.

- **Allow reload of already opened files**

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

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

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

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

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

Download Files - Commands

- **Progress Bar:** The progress bar indicates the status of downloading of the files from the PLM to the local system.
- Select the 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.
- **Create Separate BOM lines for component files linked to Same item**

If this check box is selected, when the two different components in an assembly are linked to the same item, then **Save To PLM** operation will create two different BOM lines in the PLM.
- **Create Document Object for Presentation File**

If this check box is selected, integration saves a presentation file under its own document. In this case all the presentation files available under the input project directory will be saved to PLM as part of the batch save process.
- **Allow PLM8 Operations on Non-Master LOD**

If this check box is not selected, and the active assembly file's level of representation is not "Master", PLM operations like "Save to PLM", "Check In" are disabled from Integration. You must select this check box to enable these PLM operations on such assembly file.
- **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 **Autodesk Inventor 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.
- **Show message on successful open**

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

- **Detect BOM Quantity Errors**

If this check box is selected, BOM errors if present in any assembly are reported by the Integration and save to PLM process is stopped.

- **Checkin all Draft Objects in structure**

If this check box is selected, during Check In, Integration will identify all the Draft objects in the structure of the file being checked in. All these UNDEFIEND objects are then sent for Check In.

- **Treat DWG files as Inventor Drawings**

If this check box is selected, DWG files are treated as Inventor drawings. It means File format and Application Format are set to Inventor integration related values. When set to false DWG files saved from Inventor are treated as AutoCAD drawings. File format, Application Format are set to AutoCAD integration related values.

By treating these files as AutoCAD drawings we avoid the need to download these drawings from PLM during edit/view file in integration (as part of “Download Additional Drawings”) which otherwise may impact performance.

Note: If the Save to PLM operation is performed on assembly which is non editable, the following additional steps are performed.

- 1 A folder named REPLACE_IN_PLM is created in the path <%CFE_CLIENT_HOME%>\Inventor\ Work.
- 2 Inside this folder, a folder is created with non editable assembly name. Assembly file is renamed according to the PLM Vault name and copied to this folder.
- 3 A message is provided to the user to check this folder and inform Administrator.

Refer to the Assembly_Report.txt file for more information.

Note: Assumptions & Constraints

- It is mandatory to have a common CC library path which will be accessed by all users in the same location and all users are using the same CC library path.
- Non standard Content Center file can be a normal part which is later published to CC library or a copy of CC library file.

Integration Preferences for Neutral Files

You can specify an additional format to save the local Autodesk Inventor files in the PLM vault during the check-in procedure. This additional save action creates neutral file images in an alternative formats for viewing purpose.

You can create the neutral files for the following objects:

- Parts
- Assemblies
- Drawings
- iParts

Note: Generating neutral files while saving and/or checking in to PLM requires extensive computing resources which directly affect performance. This issue occurs when you save medium to large assemblies. Therefore,

as a best practice, it is recommended that you use neutral file generation selectively and only when required. To disable the additional save, clear the check box for the required file type. You can define formats for the following types of Autodesk Inventor files.

Preference for neutral files are grouped under the following categories:

- Save Neutral Files
- Save Neutral Files Parts
- Save Neutral Files Assembly
- Save Neutral Files Drawing
- Save Neutral Files iParts
- Post Save Process

Save Neutral Files

Following are the properties in this category:

- **Generate Neutral Files**
Select this check box, to generate neutral files for the objects, such as, part, assembly or a drawing.
- **Show Neutral files during Save**
If this check box is selected, neutral files are displayed in the Attributes for File-Documents-Items objects dialog box, when you save an object.
- **Generate Neutral Files during**
Select the operation during which the integration must generate a neutral file. Allowed values:
 - On All Actions
 - On Save and Unlock
 - On Check-In
 - On Save and Unlock and Check-In

Save Neutral Files Parts

Following are the properties in this category:

- **None**
If this check box is selected, neutral file is not generated for a part, even if the "**DWF**" and/or **PDF** check box(es) are selected.
- **DWF**
If this check box is selected, neutral files are generated for a part in the "DWF" format only.
- **PDF**
If this check box is selected, neutral files are generated for a part in the PDF format only.

Save Neutral Files Assembly

Following are the properties in this category:

- **None**

If this check box is selected, neutral file is not generated for an assembly, even if the **DWF** and/or **PDF** check box(es) are selected.

- **DWF**

If this check box is selected, neutral files are generated for an assembly in the DWF format only.

- **PDF**

If this check box is selected, neutral files are generated for an assembly in the PDF format only.

Save Neutral Files Drawing

Following are the properties in this category:

- **None**

If this check box is selected, neutral file is not generated for a drawing, even if the **DWF** and/or **PDF** check box(es) are selected.

- **DWF**

If this check box is selected, neutral files are generated for a drawing in the DWF format only.

- **PDF**

If this check box is selected, neutral files are generated for a drawing in the PDF format only.

Save Neutral Files iParts

Following are the properties in this category:

- **None**

If this check box is selected, neutral file is not generated for an iPart, even if the **DWF** and/or **PDF** check box(es) are selected.

- **DWF**

If this check box is selected, neutral files are generated for an iPart in the DWF format only.

- **PDF**

If this check box is selected, neutral files are generated for an iPart in the PDF format only.

- **Generate for All Instances**

If this check box is selected, neutral files are generated for all the iPart instances of a part.

Note: You must save the part, assembly, or drawing neutral files either in DWF or PDF format. DWF is the "Design Web Format". This is an open, secure file format developed by Autodesk Inventor for the efficient distribution and communication of rich design data to anyone who needs to view, review, or print design files.

To generate PDF files, the following extra setup steps must be performed as explained below:

- 1 Install Post script Printer Driver.

You can download this from: <http://www.adobe.com>.

- 2 Install Ghost Script that will have the utility to covert Post Script file to Portable Document Format (PDF) .AFPL version 8.xx of the Ghost script application.

You can download this from: <http://www.cs.wisc.edu/~ghost/doc/AFPL/index.htm>.

- 3 Add the <Ghost Script installation Path>\bin and <Ghost Script installation Path>\lib to the system environment variable "PATH".

e.g., D:\Progra~1\gs\gs8.51\lib; D:\Progra~1\gs\gs8.51\bin;

- 4 Set PLM_INV_PS_PRN_NAME environment variable to full name of postscript printer name. We can find this name under Control Panel\Printers and Faxes.

Set PLM_INV_PS_PRN_NAME = <PostScript Printer>

- 5 Set PLM_INV_PS2PDF_UTILITY environment variable to the ps2pdf.bat utility path.

Set PLM_INV_PS2PDF_UTILITY = c:\gs\gs8.00\lib\ps2pdf.bat

Note: Ensure that there are no white-space in the installation path mentioned above.

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 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.

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 SolidWorks file that can be attached to the business process. Allowed values:

- **Documents Only**
Only the documents associated with the SolidWorks file are attached to the business process.
- **Items Only**
Only the items associated with the SolidWorks file are attached to the business process.
- **Both**
Both the documents and the items associated with the SolidWorks file are attached to the business process.

Attached related Models\Drawings

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

Attach all related objects

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

Allow edit Workflow

If this check box is selected, you can edit the workflow template when you dispatch a SolidWorks file to business process.

Action for objects locked by Business Process

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

- Attached as unlocked – The object will be attached to Business process and unlocked.
- Do not attach – The objects that are locked will not be considered for dispatch.
- Cancel action – The dispatch operation will be cancelled.

Attach Draft Objects Only

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

Attach All Items

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

Troubleshooting Option

Use this option to specify whether log files must be created for various processes of the PLM integration for Autodesk Inventor. 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.

Link to Item Dialog

Use this option to enable the **Create Item and Link to Document** check box in Set Object Properties pane. The options in the preference:

- **Create Item Checkbox Default:** If this check box is selected, the Create Item check box is selected in Set Object Properties pane. When you click **OK**, the following confirmation message is displayed, Would you like system to create Item in PLM for this item now.
- **Link to Document:** If this check box is selected and when the current file is already in PLM and when you click **Link to Item**, the **Link to Document** check box is selected in Set Object Properties pane, indicating that when user clicks **OK**, the item is linked to this document in PLM, without the need to perform Save To PLM operation.

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.

- Create a file ModifiedMappingRulesForCheckIn.xml in the %CFE_CLIENT_HOME%\Toolkit directory. The file must contain the following syntax:

```
<MODIFIEDCHECKINRULES> <!-- MAPPING TEMPLATE "NAME" and the PLMFIELD
"FIELD" must be in Upper-Case--> <TEMPLATE NAME = "Inventor_PLM">
<PLMFIELD FIELD = "DOC.CS_ATTR_011" TYPE = "EVALUATE" VALUE = "CURRENT_DATE" > </PLMFIELD>
<PLMFIELD FIELD = "DOC.CS_ATTR_012" TYPE
= "REFERENCE" VALUE = "DOC.CS_ATTR_001" > </PLMFIELD> <PLMFIELD FIELD
= "DOC.CS_ATTR_013" TYPE = "LITERAL" VALUE = "Hyderabad">
</TEMPLATE> <TEMPLATE NAME = "Inventor_A4_PLM"> <PLMFIELD FIELD = "DOC.CS_ATTR_011" TYPE =
"EVALUATE" VALUE = "PLM_USER" >
<PLMFIELD FIELD = "DOC.CS_ATTR_012" TYPE = "REFERENCE" VALUE = "DOC. CS_ATTR_002" > </PLMFIELD>
<PLMFIELD FIELD = "DOC.CS_ATTR_013" TYPE
= "LITERAL" VALUE = "Infor"> </PLMFIELD> </TEMPLATE> </ MODIFIEDCHECKINRULES>
```

- Modify the contents of the ModifiedMappingRulesForCheckIn.xml file to match the specific requirement. Perform the following steps to modify the content:
 - 1 Enter a relevant template name.

- 2** Enter a customized field name for the **PLMFIELD**. The **PLMFIELD** must have a TO_CADmapping rule defined in this template.
- 3** Type **EVALUATE** indicates the value that can be taken from the system variables like:
 - CURRENT_DATE
 - PLM_USER
 - CURRENT_TIME
 - CURRENT_DATE_TIME
- 4** Type **REFERENCE** indicates the value which can be another field from the PLM table.
- 5** 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 33: The properties of the Toolkit tab

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 Autodesk Inventor will display the synchronization dialog box during save operations.

Show Top Down Load report

If this check box is selected, the PLM integration for Autodesk Inventor 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 Autodesk Inventor 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. The link enables PLM to determine how to open the file.

Chapter 34: Setup and administration

This section contains the following topics for the administrator:

- [Installation and setup](#) on page 101
- [Troubleshooting](#) on page 97
- Limitations

Installation and setup

Please note the following, before you install the PLM - Autodesk Inventor Integration:

- Before a new version of the PLM Integration for Autodesk Inventor is installed, the previous installation must be uninstalled.
- Before installing the Autodesk Inventor integration, the PLM client must be installed.
- Check that the path of the executable or batch file for the integration is correctly defined, in order for the Edit/View command to work correctly. See Setting the Path for the "Edit/View" Command.
- All integration users must be created in PLM and assigned to the relevant projects, using the **Administration > Roles Management**. For more information, refer to the Roles Management online Help.
- Vault parameters must be set for each PLM project. See Vault Parameter Configuration.

Setting the Path for the "Edit/View" Command

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

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

Vault Parameter Configuration

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

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 Creating Projects in the *Infor PLM Discrete User Guide*.

The settings ensure that document and item revisions are correctly synchronized in PLM 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	After Item Check Out
Child Items		Copy all links
Related Documents	Check In all objects	Check Out all Objects

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
Child Documents	Check In all objects	Copy all links
Related Items	Check In all objects	Check Out all Objects
Related Files	Check In all objects	Check Out all Objects

Chapter 35: Working with the PLM Integration for Autodesk Inventor – Best practices

The following section lists 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.

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

For further information, see:

- [Saving and unlocking a file](#) on page 25

- [Saving Files to PLM](#) on page 16

Can other user 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 25
- [Changing ownership of a file](#) on page 31

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 25
- [Saving to PLM](#) on page 16

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 25
- [Take ownership](#) on page 31

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

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

For further information, see:

- [View PLM data](#) on page 32
- [Opening a file in an PLM workspace](#) on page 34

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 25
- [Take ownership](#) on page 31
- [Checking in a file](#) on page 26

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

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

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

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

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

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

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

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

For further information, see:

- [View PLM data](#) on page 32
- [Synchronize headers](#) on page 39

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

Only check out the components you must change. Note that if you check out a component without checking out the assembly in which the component is used, you must save this component separately.

If a subassembly is RELEASED, the integration does not check the sub components of this assembly, which enhances the performance of the save operation. For further information, see [Checking out a file](#) on page 29.

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

There are two types of links between Autodesk Inventor 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 Autodesk Inventor. The item work spaces in the PLM client utility allow users to view this information and make changes to the links, if required.