



Infor PLM for Discrete AutoCAD User Guide

Release 2022.x

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Publication Information

Release: Infor LN 2022.x

Publication Date: May 6, 2022

Document code: ln_2022.x_plmacadug__en-us

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About this guide

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

Intended audience

This guide is intended for users of the PLM and AutoCAD.

Contacting Infor

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Chapter 1: Introduction

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

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

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

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

Main features of the integration

The PLM integration for AutoCAD includes the following main features:

- AutoCAD drawings can be vaulted from within the AutoCAD environment.
- Automatic update of engineering BOMs according to the sub-assembly structure.
- Direct transfer of product configurations from AutoCAD into ERP LN via PLM.
- Document link management.
- Unique file names for all new AutoCAD models.
- Foolproof authorization maintains security of AutoCAD files.
- Files can be automatically saved to PLM in neutral formats.
- Legacy files prepared outside the PLM environment can be imported with automatic updates of links, reflecting the relationships between all models and files.
- Automatic attribute mapping.
- Graphic representation of product trees.
- Built-in file viewer.

Product structure management

PLM provides product structure management for tight control over complex product structures. PLM assists the engineer in managing the product structure information, as well as the relationship between the product and the describing documents, thus creating the product definition.

Document management and file management

PLM enables users to manage all the enterprise's design, manufacturing and other engineering-related documents and files in a secure environment. PLM enables the user to control complex relationships between documents, items and related changes.

All documents can be linked to one or multiple PLM objects, such as other documents, items and files.

Revision control

PLM's advanced revision control ensures that only the correct product information is accessed, and that design changes are only entered to production after they are approved. Previous revisions are securely stored for future access.

Assigning IDs manually

There are two ways of assigning an ID, either automatically, according to the mask defined in PLM, or manually. Manual IDs can be entered for both documents and items or for each separately. Similarly, object revisions can also be specified manually or can be generated automatically according to the mask.

Neutral file formats

Data and files that have been vaulted with the original model can be exported in neutral formats, to be used in popular viewers and other CAD applications.

Chapter 2: Getting started

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

- Installing the integration
- [Requirements](#) on page 11
- [PLM embedded menu and toolbar](#) on page 15

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

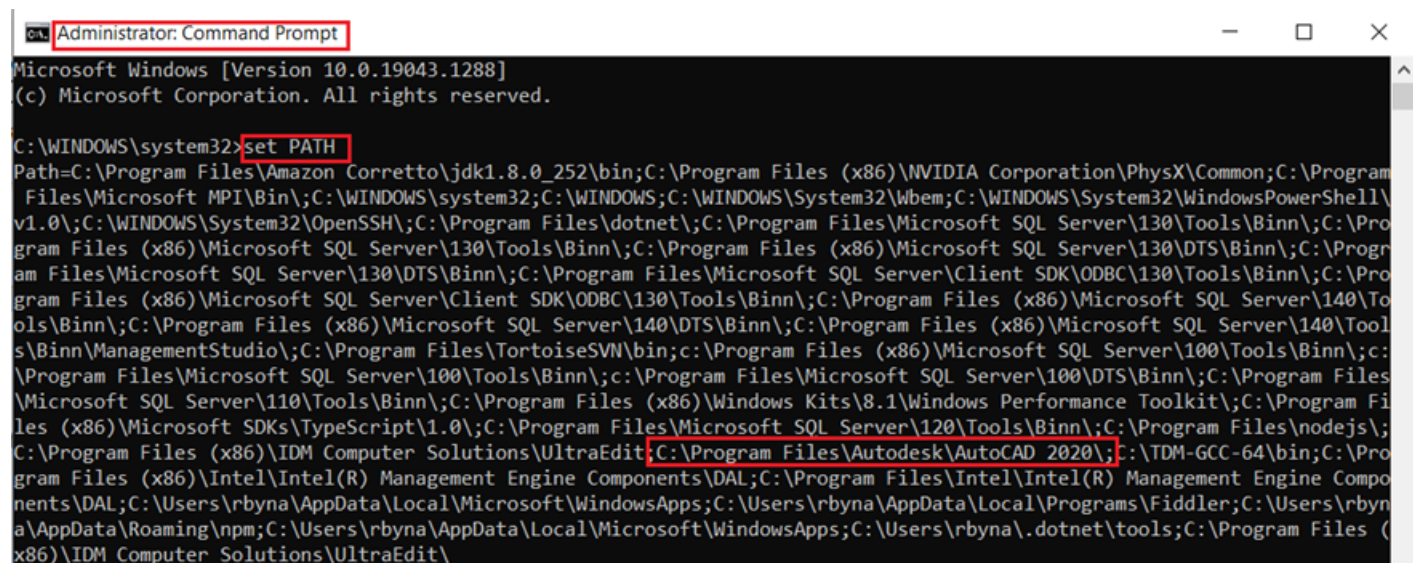
Requirements

The minimum hardware and software requirements for the Infor PLM Discrete integration for AutoCAD Is AutoCAD application and AutoCAD installation patch should be update in the system environment variable PATH.

Example:

AutoCAD installation Path: C:\Program Files\Autodesk\AutoCAD 2020

System Environment PATH variable:

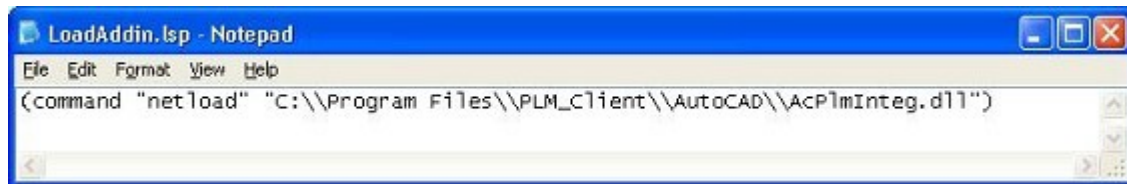


```
Administrator: Command Prompt
Microsoft Windows [Version 10.0.19043.1288]
(c) Microsoft Corporation. All rights reserved.

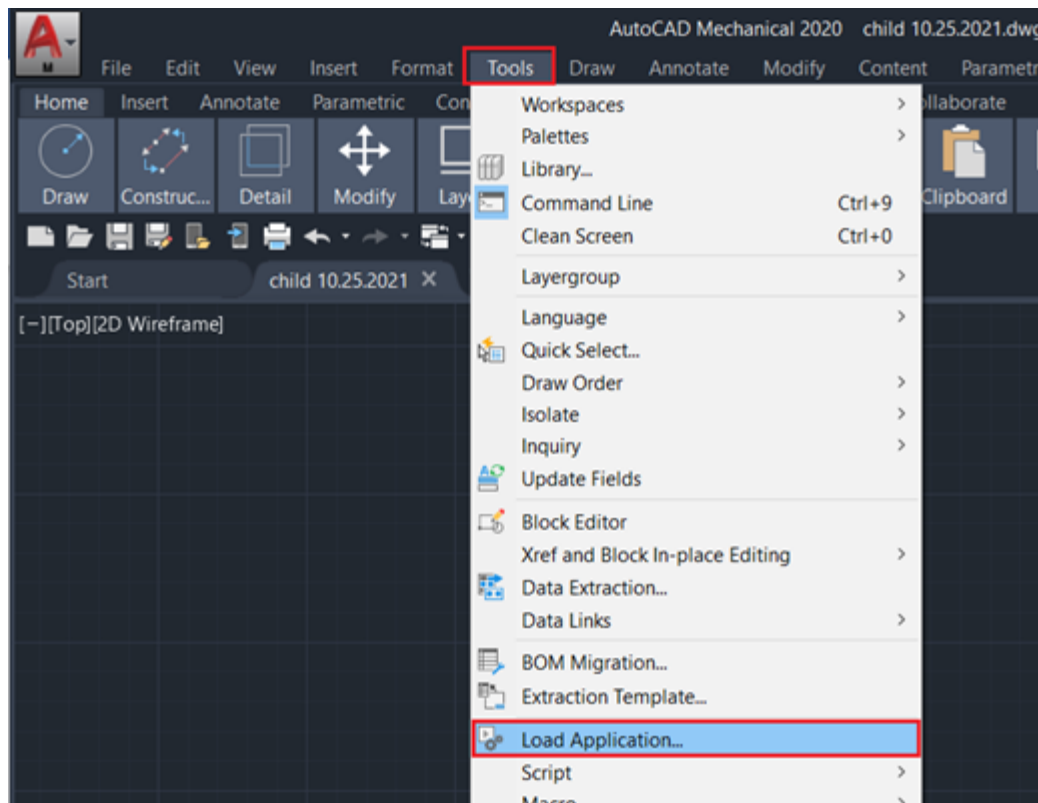
C:\WINDOWS\system32>set PATH
Path=C:\Program Files\Amazon Corretto\jdk1.8.0_252\bin;C:\Program Files (x86)\NVIDIA Corporation\PhysX\Common;C:\Program Files\Microsoft MPI\Bin\;C:\WINDOWS\system32;C:\WINDOWS;C:\WINDOWS\System32\Wbem;C:\WINDOWS\System32\WindowsPowerShell\v1.0\;C:\WINDOWS\System32\OpenSSH\;C:\Program Files\dotnet\;C:\Program Files\Microsoft SQL Server\130\Tools\Binn\;C:\Program Files (x86)\Microsoft SQL Server\130\Tools\Binn\;C:\Program Files (x86)\Microsoft SQL Server\130\DTS\Binn\;C:\Program Files\Microsoft SQL Server\130\Tools\Binn\;C:\Program Files\Microsoft SQL Server\130\Tools\Binn\;C:\Program Files (x86)\Microsoft SQL Server\140\Tools\Binn\;C:\Program Files (x86)\Microsoft SQL Server\140\Tools\Binn\;C:\Program Files (x86)\Microsoft SQL Server\140\Tools\Binn\ManagementStudio\;C:\Program Files\TortoiseSVN\bin;c:\Program Files (x86)\Microsoft SQL Server\100\Tools\Binn\;c:\Program Files\Microsoft SQL Server\100\Tools\Binn\;c:\Program Files\Microsoft SQL Server\100\DTS\Binn\;C:\Program Files\Microsoft SQL Server\110\Tools\Binn\;C:\Program Files (x86)\Windows Kits\8.1\Windows Performance Toolkit\;C:\Program Files (x86)\Microsoft SDKs\TypeScript\1.0\;C:\Program Files\Microsoft SQL Server\120\Tools\Binn\;C:\Program Files\nodejs\;C:\Program Files (x86)\IDM Computer Solutions\UltraEdit\;C:\Program Files\Autodesk\AutoCAD 2020\;C:\TDM-GCC-64\bin;C:\Program Files (x86)\Intel\Intel(R) Management Engine Components\DAL;C:\Program Files\Intel\Intel(R) Management Engine Components\DAL;C:\Users\rbyna\AppData\Local\Microsoft\WindowsApps;C:\Users\rbyna\AppData\Local\Programs\Fiddler;C:\Users\rbyna\AppData\Roaming\npm;C:\Users\rbyna\AppData\Local\Microsoft\WindowsApps;C:\Users\rbyna\dotnet\tools;C:\Program Files (x86)\IDM Computer Solutions\UltraEdit\
```

Configuring the Integration in AutoCAD

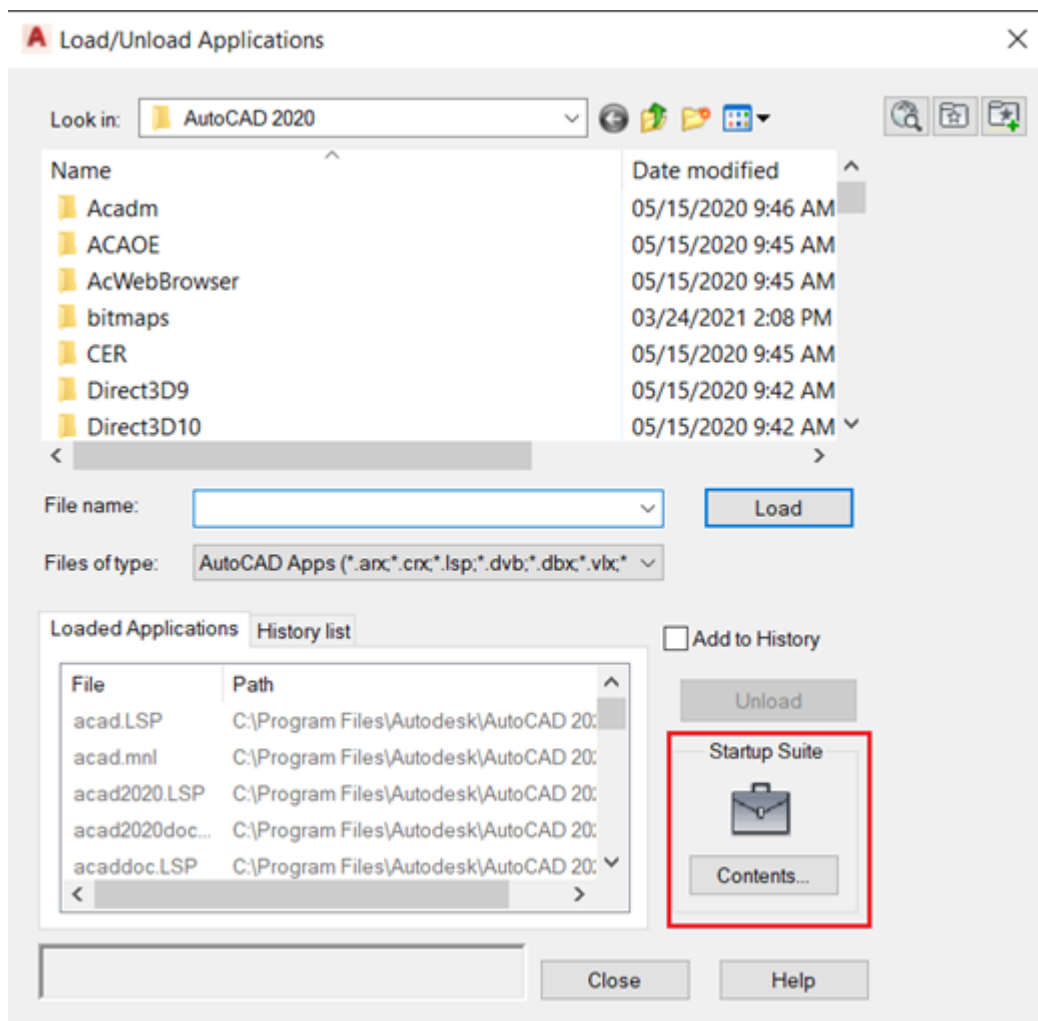
- 1 Install the integration kit.
- 2 Go to the folder %cfe_client_home%\AutoCAD and modify the file LoadAddin.lsp as shown below. The path should contain "\\".



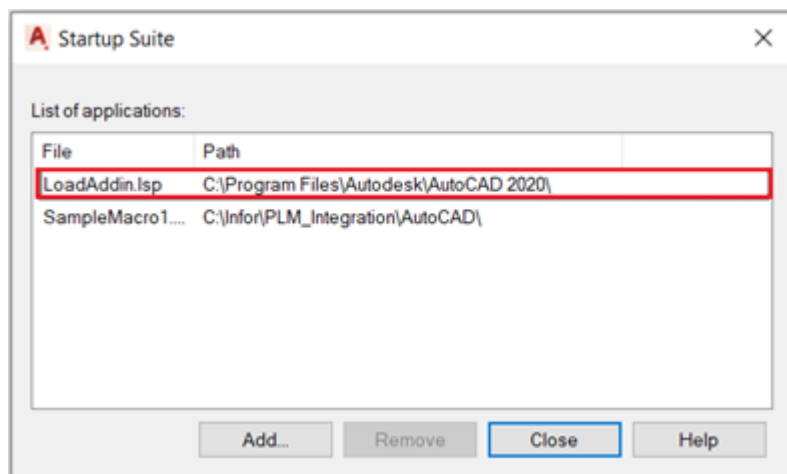
- 3 Start the AutoCAD application.
- 4 In the AutoCAD application menu click **Tools > Load Application** as shown in the following screen.



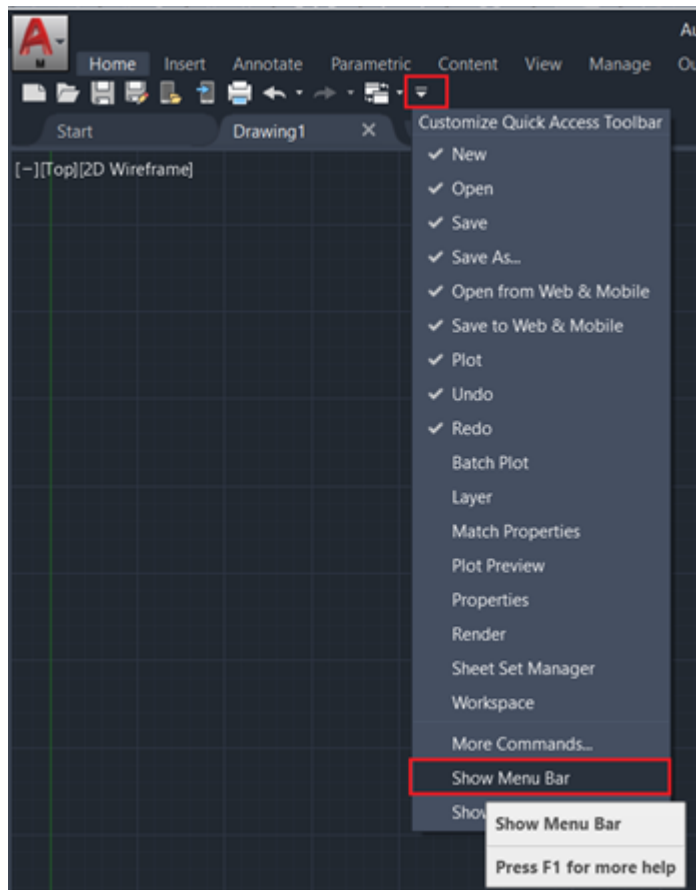
- 5 Click **Contents** in the **StartUp** suite control box.



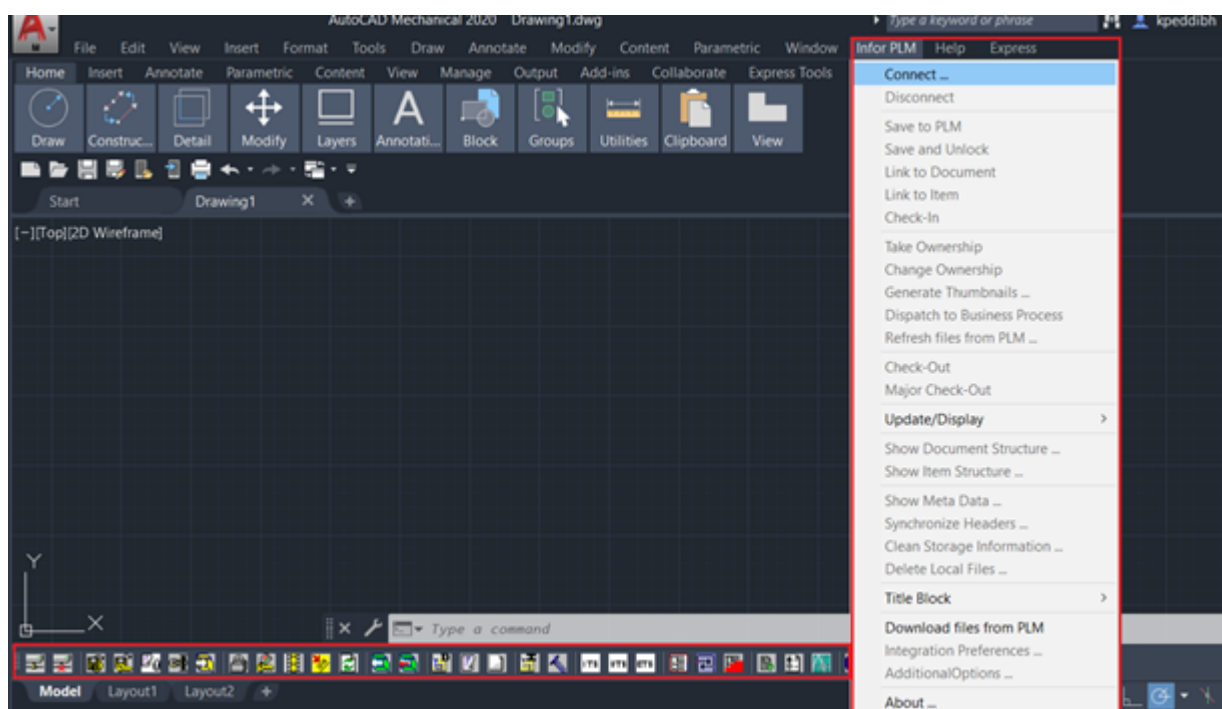
- 6 Select %cfe_client_home%\AutoCAD\LoadAddin.lsp file in the Startup Suite.



- 7 Close all the dialog box and return to the application.




























- 8 Close the application and restart the AutoCAD application. When the application is up and running, integration toolbar and Menu are visible. The following screen shows the Integration toolbar and menu.



PLM embedded menu and toolbar

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

Icon	Option	See/Description
	Connect	Connecting to PLM on page 17
	Disconnect	Disconnecting from the PLM integration on page 17
	Save to PLM	Saving Files to PLM on page 18
	Save and Unlock	Saving and Unlocking Files on page 26
	Check In	Checking in a File on page 27
	Take Ownership	Take ownership on page 34
	Change Ownership	Ownership on page 34
	Generate Thumbnails	Thumbnails on page 31

Icon	Option	See/Description
	Dispatch to Business Process	Dispatch to Business Process on page 42
	Refresh Files from PLM	Refresh Files from PLM on page 41
	Check Out	Checking out a File on page 28
	Check Out Major	Checking out a File on page 28
	Show Document Structure	Opening File in a PLM on page 35
	Update Document	Using Infocards on page 36
	Update File	Using Infocards on page 36
	Show Item Structure	Opening File in a PLM on page 35
	Update Item	Update/ Display Item Attributes
	Insert Title Block	How to insert a title block
	Update Title Block	Working with PLM on page 17
	Edit Title Block	How to edit a title block
	Show Meta Data	Viewing PLM Data on page 29
	Synchronize Headers	Synchronize Headers on page 37
	Clear Storage Information	Clean Storage Information on page 39
	Integration Preferences	Setting preferences
	Additional Options	Additional Options on page 19
	About	Contains product and system information as well as additional sources for professional assistance.

Some of the above options may also be available in the right-click menu.

Note: The functions that you can perform may depend on your user type and authorization.

Chapter 3: Working with PLM


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

- [Connecting to PLM](#) on page 17
- [Disconnecting from the PLM integration](#) on page 17

Connecting to PLM

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

To connect to PLM:


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

The functions in the PLM menu and toolbar are enabled. When you open another AutoCAD 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 AutoCAD integration applications; disconnecting from one AutoCAD application disconnects all connected AutoCAD application from PLM.

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

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

Chapter 4: Saving Files to PLM

The Save to PLM process saves your AutoCAD 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 status of **draft**. If the document has the status **UNDER CHANGE** or **RELEASED**, the file cannot be saved to PLM.

To save your changes on the server (not only locally), 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 AutoCAD. If you save the file only to AutoCAD, 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.

In case of concurrent engineering, you may need to acquire ownership of the file from the previous owner. For more information, refer to [Saving and Unlocking Files](#) on page 26.

Saving files to PLM- best practices

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

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

To save a file to PLM

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

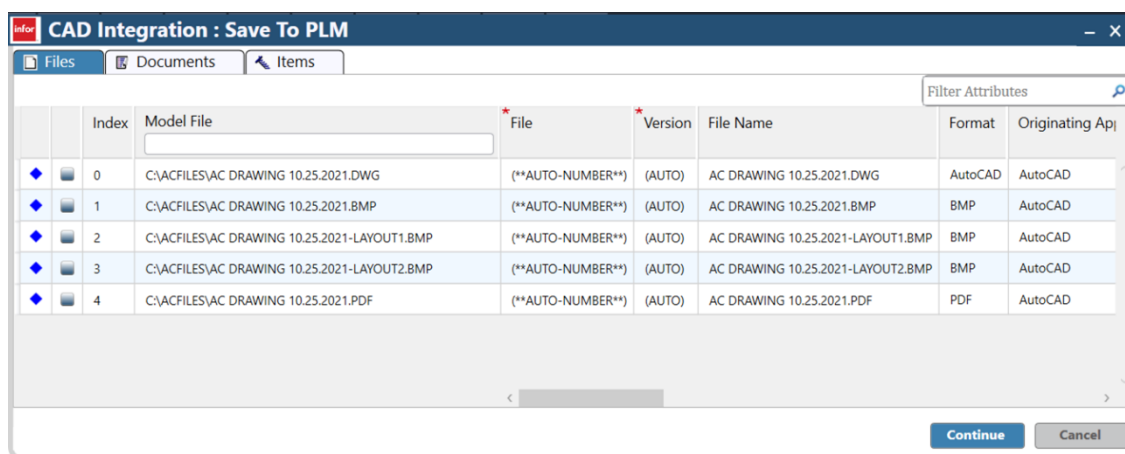
Note: Do not use & < > ' " symbols in the file names that you save to PLM, this will cause errors to occur.

Do one of the following:

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

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

- If all the IDs are generated automatically by PLM, which you can specify in the preferences, the file is saved. For further information on setting preferences, see **General** tab.
- If the **Set Object Attribute During Save** check box is selected in the integration preferences, the Set Object Attributes dialog box appears.



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

For attributes that you do not update, the PLM default values are automatically inserted.

Additional Options

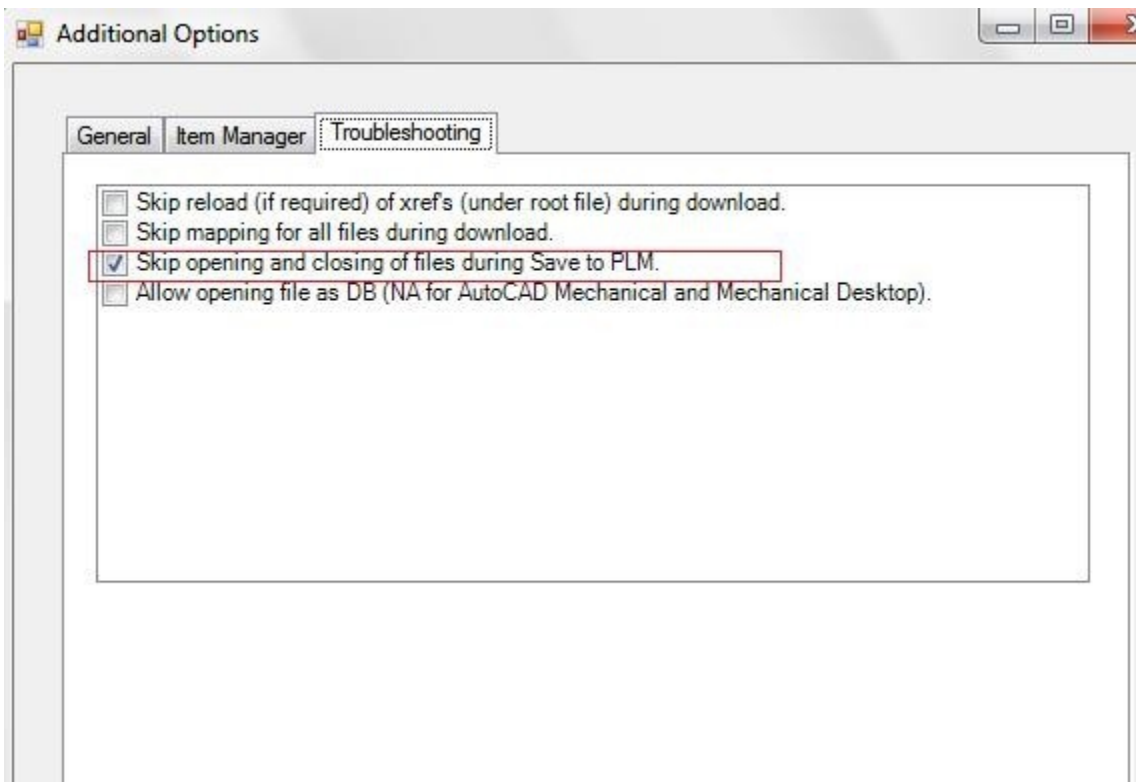
Before performing SaveToPLM on the drawing files, you must be aware of the preference in Additional options.

You can access **Additional Options** preference from Menu or Toolbar as shown below:



Click **Additional Options**. The Additional Options dialog box opens. Select the **Troubleshooting** tab and set the preference as shown below. Setting this preference avoids the opening and closing of the files during SaveToPLM process.

Note: When you perform SaveToPLM of the large drawing (Drawing with say > 100 children), you may need to uncheck this preference.



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 PLM preferences controls the functions of download manager:

File Name Uniqueness

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

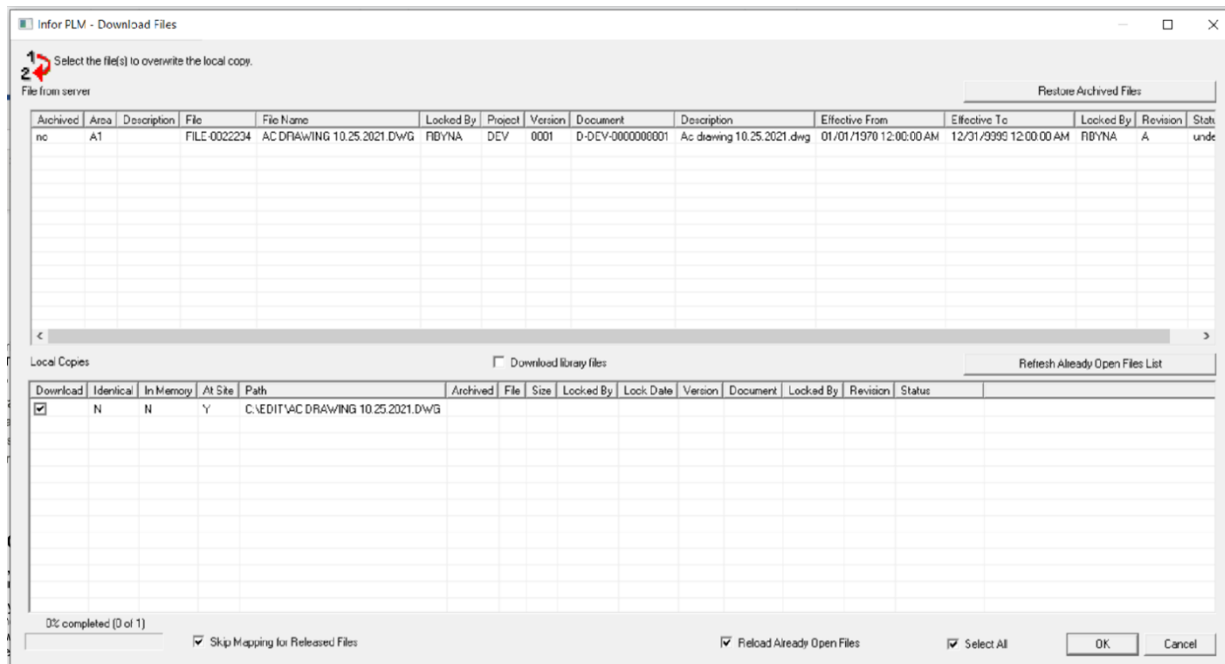
If this check box is cleared, you can store multiple files with identical file names in PLM but in different projects. It is not recommended to clear this check box, since it might cause problems if you want to download a file while a file with an identical name already exists locally. For example, you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

Indicator for Locally Changed Files in Download Manager dialog

The identical column under the **PLM - Download Files** dialog box indicates whether the file is changed in PLM and/or 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.

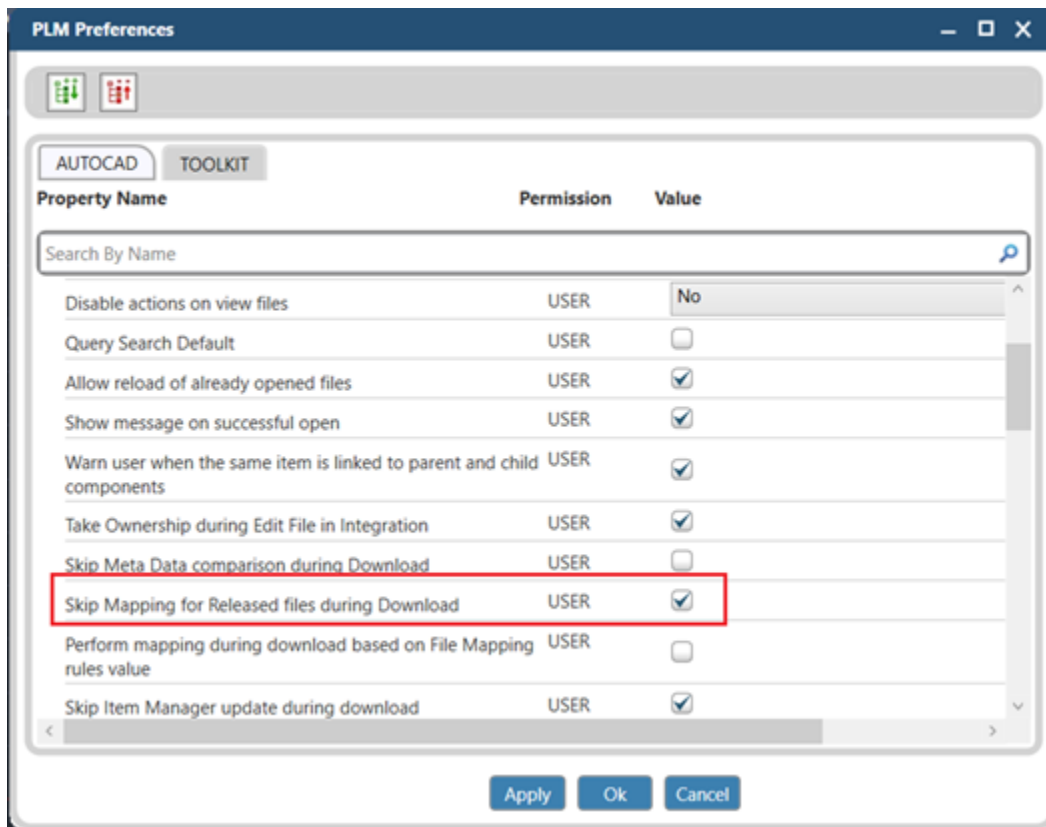


Download Files - Commands

- 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 value for this check box is defaulted based on the integration preference Skip Mapping for RELEASED files during Download. While downloading large assemblies, you can check this check box in order to improve the performance of the download operation.
- Refresh Already Open Files: Refreshes the information under the In Memory column of the download manager dialog.
- Restore Archived Files: Retrieves the file from the archive area.

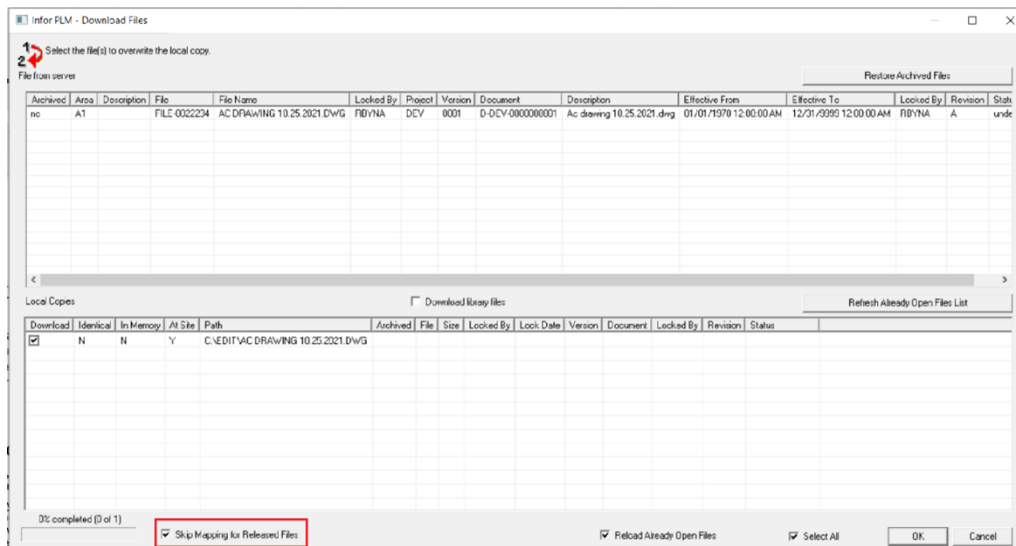
Skip Mapping for Released Files

When you select the preference **Skip Mapping for Released Files during Download** in the General Option 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 one or more file properties may not be up to date with PLM.



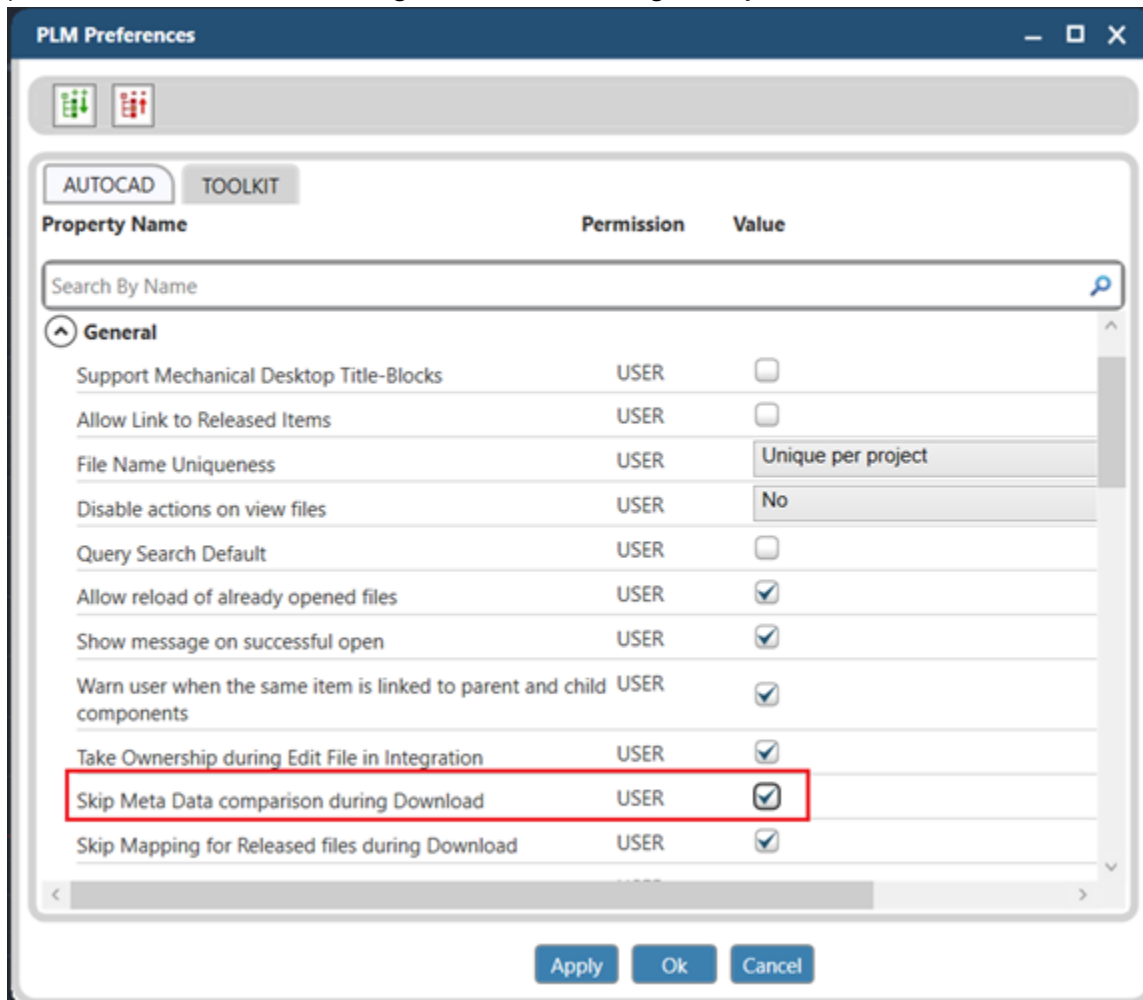
Additionally, it is also possible to select the **Skip Mapping for Released Files during the download** process. Select **Skip Mapping for Released Files** check box to skip the mapping for the released files.

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



Skip Meta Data Comparison During Download

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



The **Download Manager** also indicates if there is any change in the PLM data of Document/ Item/File as shown below.

Infor PLM - Download Files

1 Select the file(s) to overwrite the local copy.

2 File from server

Restore Archived Files

Archived	Area	Description	File	File Name	Locked By	Project	Version	Document	Description	Effective From	Effective To	Locked By	Revision	Status
no	A1		FILE 0022234	AC DRAWING 10.25.2021.DWG	RBYNA	DEV	0001	D-DEV-000000001	Ac drawing 10.25.2021.dwg	01/01/1970 12:00:00 AM	12/31/9999 12:00:00 AM	RBYNA	A	unde

Local Copies

Download library files

Refresh Already Open Files List

Download	Identical	In Memory	At Site	Path	Archived	File	Size	Locked By	Lock Date	Version	Document	Locked By	Revision	Status
<input checked="" type="checkbox"/>	N	N	Y	C:\EDIT\AC DRAWING 10.25.2021.DWG										

0% completed (0 of 1)

☒ Skip Mapping for Released Files

☒ Reload Already Open Files

☒ Select All

OK Cancel

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

Chapter 6: 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 [Ownership](#) on page 34.

To save and unlock a file

Do one of the following:

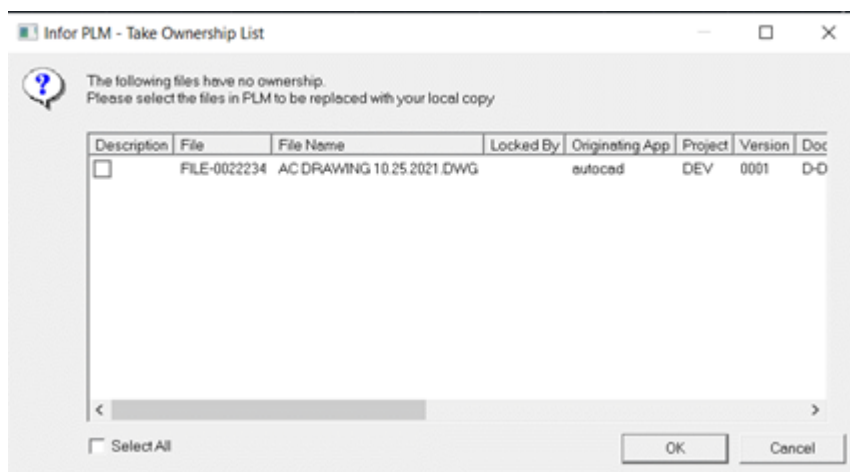
- Click **Save and Unlock**  in the PLM toolbar.
- Select **Save and Unlock** from the PLM menu.

As a result, the file is saved in PLM with no owner.

To acquire ownership of a saved and unlocked file:

- 1 If you do not have ownership of the current file, click **Take Ownership** and acquire the ownership of the file. You can also take ownership for one or dependent files as part of **Save to PLM** operation.

The Take Ownership List dialog box is displayed.



- 2 Select the required file(s) and click **OK**.
- 3 Save the file again in AutoCAD.

Chapter 7: Checking in a File

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


The check-in operation accomplishes the following:

- Confirms the changes you made in the AutoCAD file.
- Changes the AutoCAD file's status from **draft** to RELEASED.
- Transfers the AutoCAD file to the PLM Release area from the work area.

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

After the file has been checked in, 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 menu.
- 2 If prompted, save the file to AutoCAD first. The file is checked in.

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

Chapter 8: 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 Download files from PLM menu option.

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

Two types of check out are available:

- **Check Out Minor**
This option is usually used for minor design revision changes. A minor check-out results in a higher sequence number within the current revision for the checked out file, for example from A0001 to A0002.
- **Check Out Major**
This option is usually used for significant changes with major impact on the form, fit or functionality of the product. A major check-out results in a higher version number for the checked out file, for example from A0001 to B0001.

To check out a file:

- 1** Select the assembly or component part that you want to check out and select **Check Out** 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.

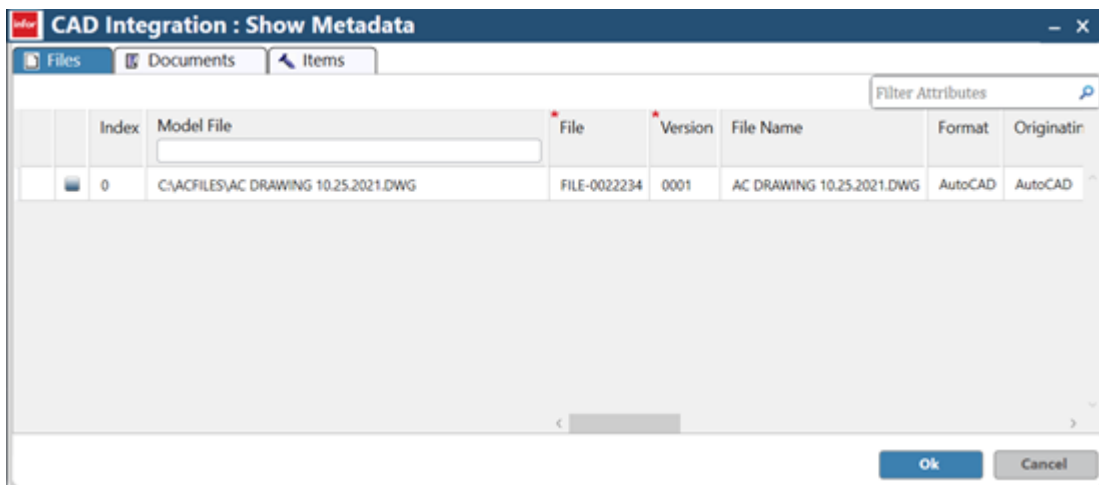
Chapter 9: Viewing PLM Data

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

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

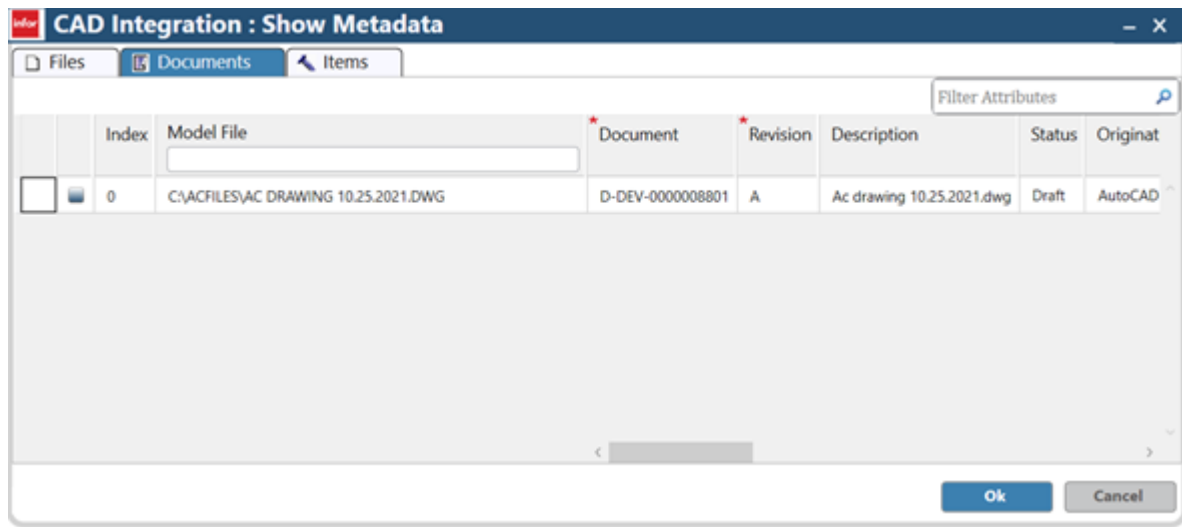
- 1 On the **PLM** menu, select **Show Metadata**.

The Set Attribute dialog box is displayed.

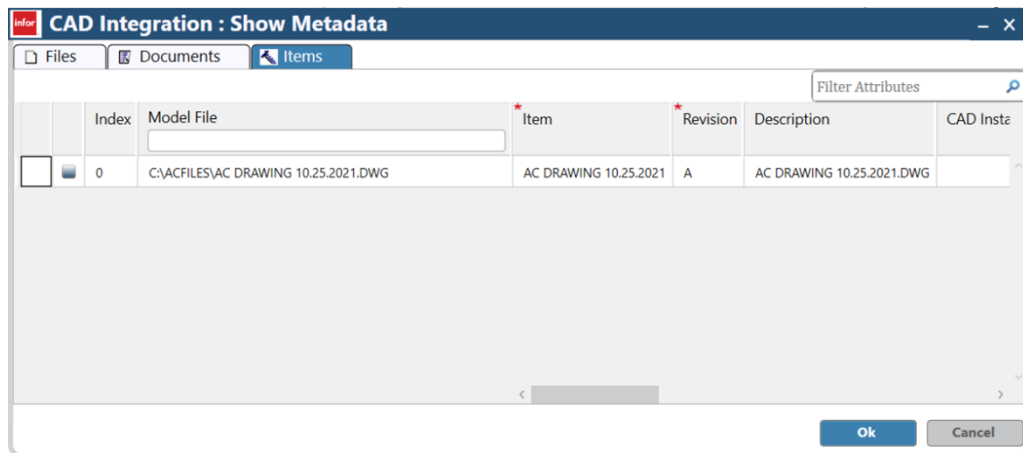


The default tab, **Files**, displays the PLM data of all the files that are part of your AutoCAD structure.

- 2 Click the **Documents** tab to display the PLM document data related to your AutoCAD part file.



- 3 Click the **Items** tab to display the PLM Item data related to your AutoCAD files.



Chapter 10: Thumbnails

Thumbnail is a miniature representation of a AutoCAD file.

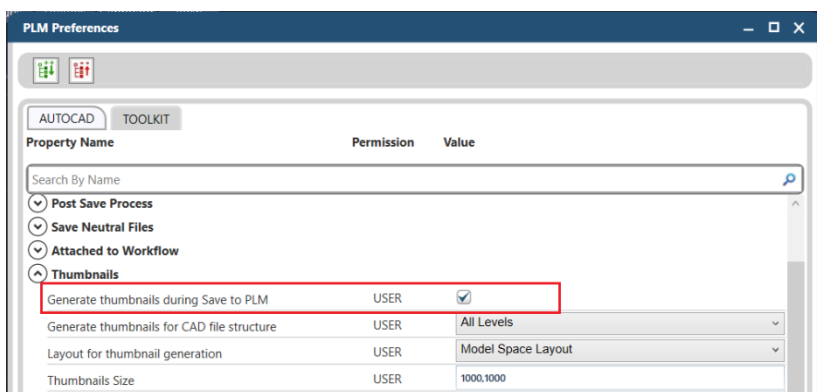
Thumbnails can be generated in the following ways:

- During the [Saving Files to PLM](#) on page 18, thumbnails are created for editable files.
- Using the **Generate Thumbnails** option on the PLM menu, users can also create thumbnails for the non-editable files in the structure.

Generating Thumbnails

To generate thumbnails during Save to PLM operation, perform the following steps:


- 1 In AutoCAD 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).

To generate thumbnails for editable/non-editable files, perform the following steps:

- Open the AutoCAD file.
- Click **Generate Thumbnails**  on the toolbar.

Thumbnail Locations

By default, thumbnails are saved in the same model file location (or) %CFE_CLIENT_HOME%\temp\Thumbnails folder.

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

Integration Preferences for Thumbnails

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


- Whether to generate thumbnails during Save to PLM operation.
- Level to which the thumbnails can be generated in a CAD structure.
- Layouts of AutoCAD files for which thumbnails must be generated.

The **Thumbnails** option includes the following properties:

Generate thumbnails during Save to PLM

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

Generate thumbnails for CAD file structure

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

Allowed Values:

- **All Levels**
Thumbnails are generated for the files under the structure of 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, where in you can select the files for which thumbnails needs to be generated.

Layout for thumbnails generation

Select the layout for thumbnail generation.

This field can have the following values

- Model Space Layout
- Active LayoutInfor

Chapter 11: Ownership

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

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

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

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

After a file has been checked in (and has RELEASED status), it does not have a specific owner.


To change the ownership of a file:

- 1 In AutoCAD, 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.

Take ownership

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

To take ownership of a file within an assembly, 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.

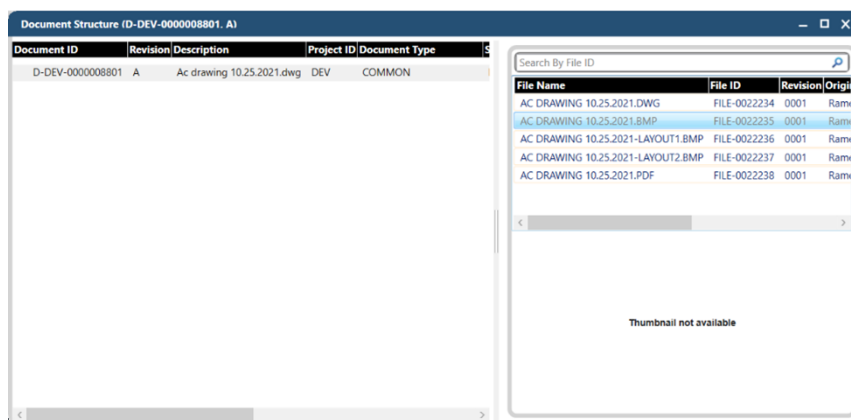
Note: When **Take Ownership during Edit File** preference check box is not selected, the users do not get the ownership of the files even when they execute Edit File command.

Chapter 12: Opening File in a PLM

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

- 1 Create a file in AutoCAD and save it to PLM. See [Saving Files to PLM](#) on page 18.
- 2 Select **Show Document Structure**  in the PLM menu or toolbar. The document is opened in the appropriate PLM Structure.

The document shows the document that was created in PLM.



Chapter 13: Using Infocards

An infocard is a dialog box that enables you to view/update meta data of documents or files created in AutoCAD. There are three types of infocards, one for documents, one for items and one for files. You can update only those files that you own.

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

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

If the part file whose 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.

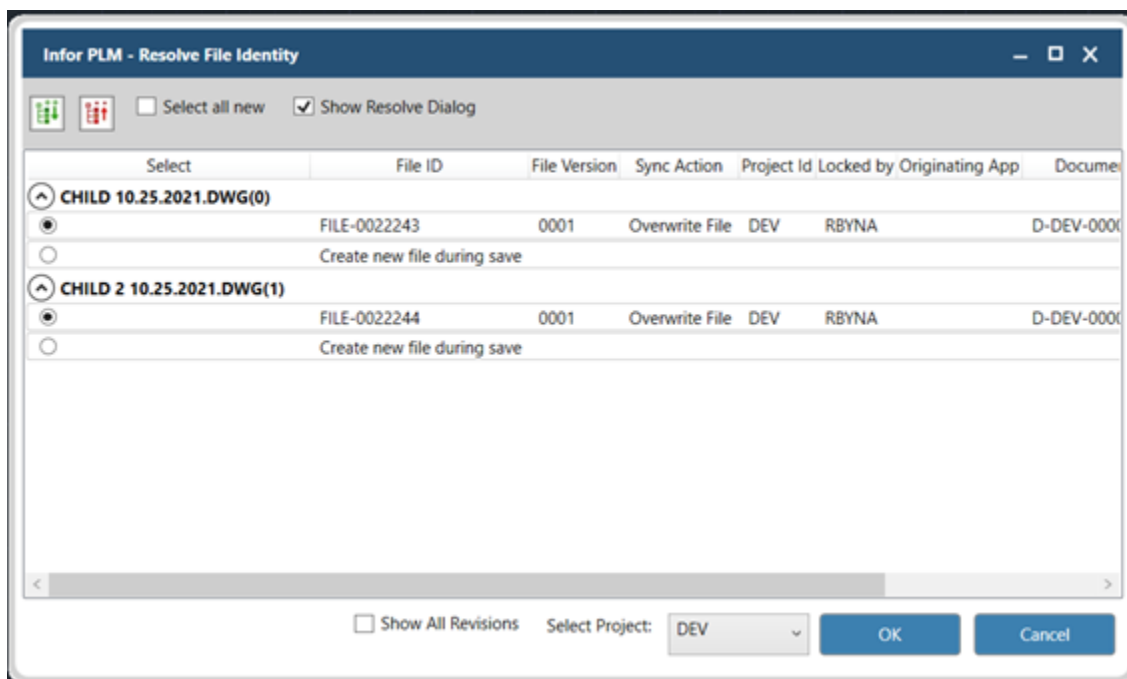
If the part file is not editable because you have no editing rights, the part file is locked by another user, or because the object is released, the infocard dialog box appears in view mode.

Chapter 14: Synchronize Headers

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

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 15: Clean Storage Information

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

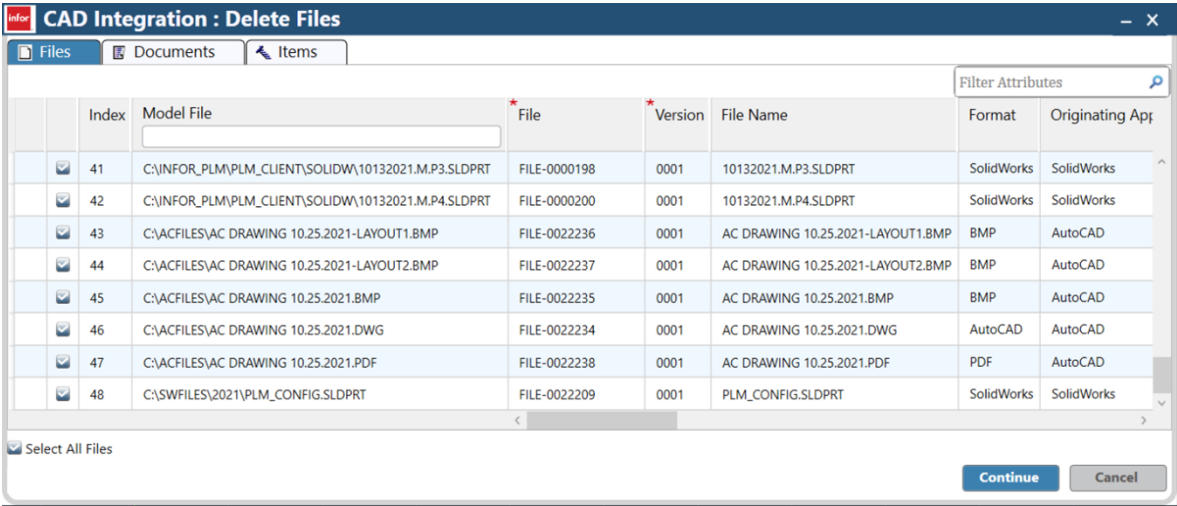
When you click **Clean Storage Information**:

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

Chapter 16: Deleting Local Files

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

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




The screen above lists the files that are registered in the storage files and saved in the local system. By default the files are selected for deletion. When you click **Delete**, PLM prompts you to confirm the file deletion.

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

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

Chapter 17: Refresh Files from PLM

You can click **Refresh files from PLM** () to update the currently open files from Infor PLM.

When you click this option integration will display the Download Manager dialog box with the list of only those files that are changed under the structure of current file in PLM. When you click ok in the Download Manager dialog, it will retrieve the changed files from PLM. For more information, refer to Download Manager section.

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

Chapter 18: 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 file, to an PLM business process. This will link the PLM items and/or documents of the part file to a workflow, which will be distributed to the users associated to the workflow template.

In the preferences you can specify whether documents, items, or both must be linked to a business process. For this purpose, click **Preferences** on the PLM menu and on the **Attach to Work Flow** group, select the relevant option under the Attached to Business Process preference.

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

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

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

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

For example, this can happen if you specified that items must be attached to the business process while your preferences specify that no PLM items must be created for the files that you save to PLM.

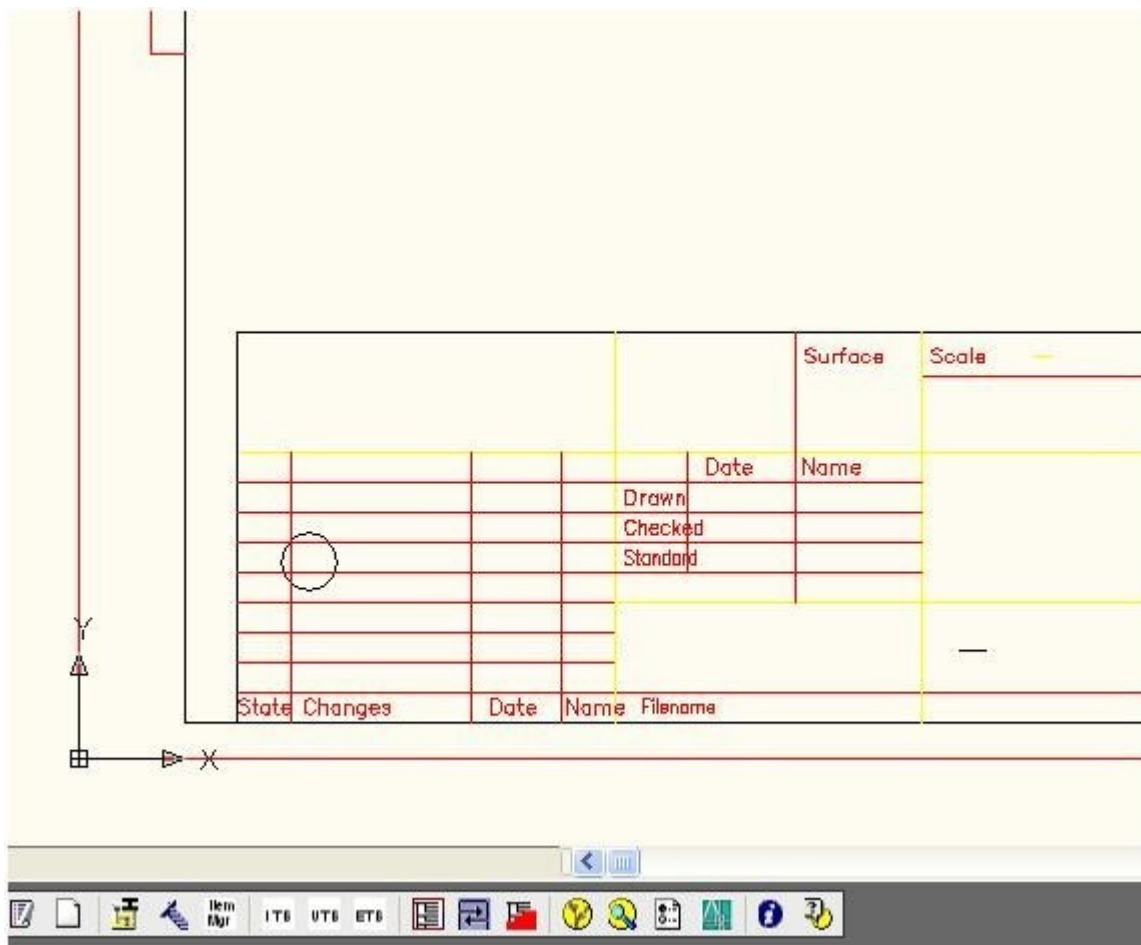
If the check box **Attach all related objects** is selected, then during this operation, the PLM objects (items, documents) of all the components in CAD structure of a AutoCAD file will be attached to the business process.

Chapter 19: Mapping Rules

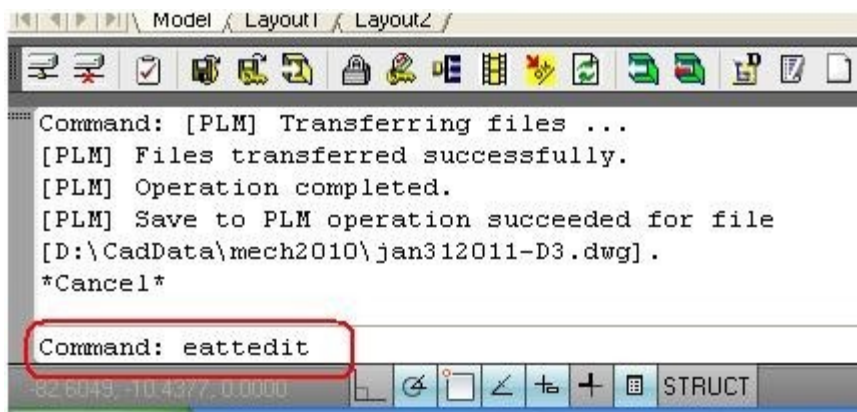
We can use Mapping to transfer the values of Title-Block attribute tags from AutoCAD to PLM and also can use to update the Title-Block attribute tags values based on values from PLM.

Steps to define mapping rules in AutoCAD

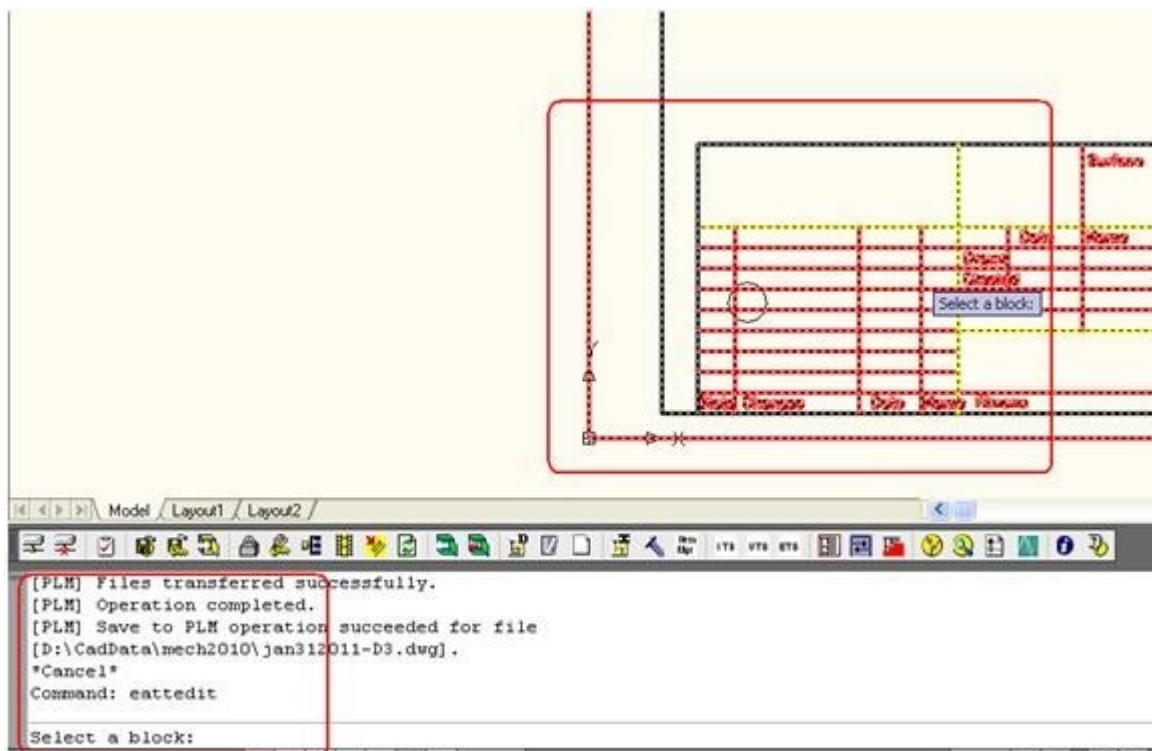
- 1 When we insert title block in a new or an existing drawing and perform SaveToPLM, it results in a drawing with title block without mapping, as shown below.



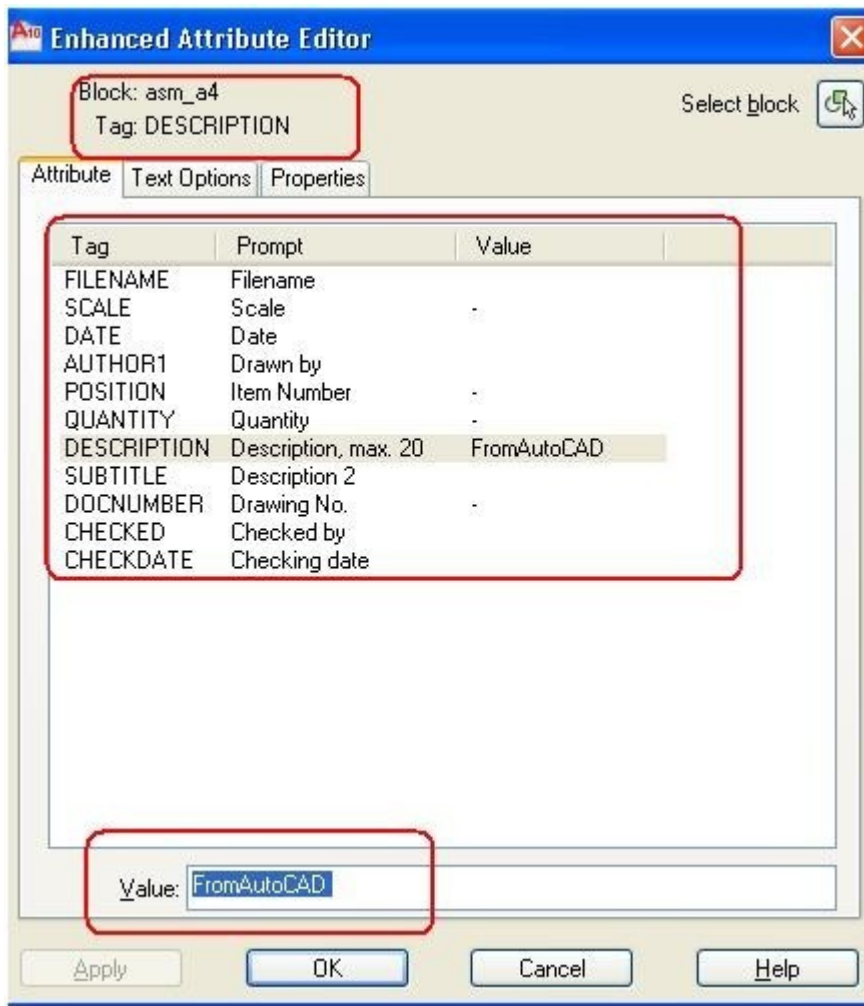
- 2 In order to know the list of tags under the Title-Blocks, enter **eattedit** in the AutoCAD command prompt and press enter, as shown below.



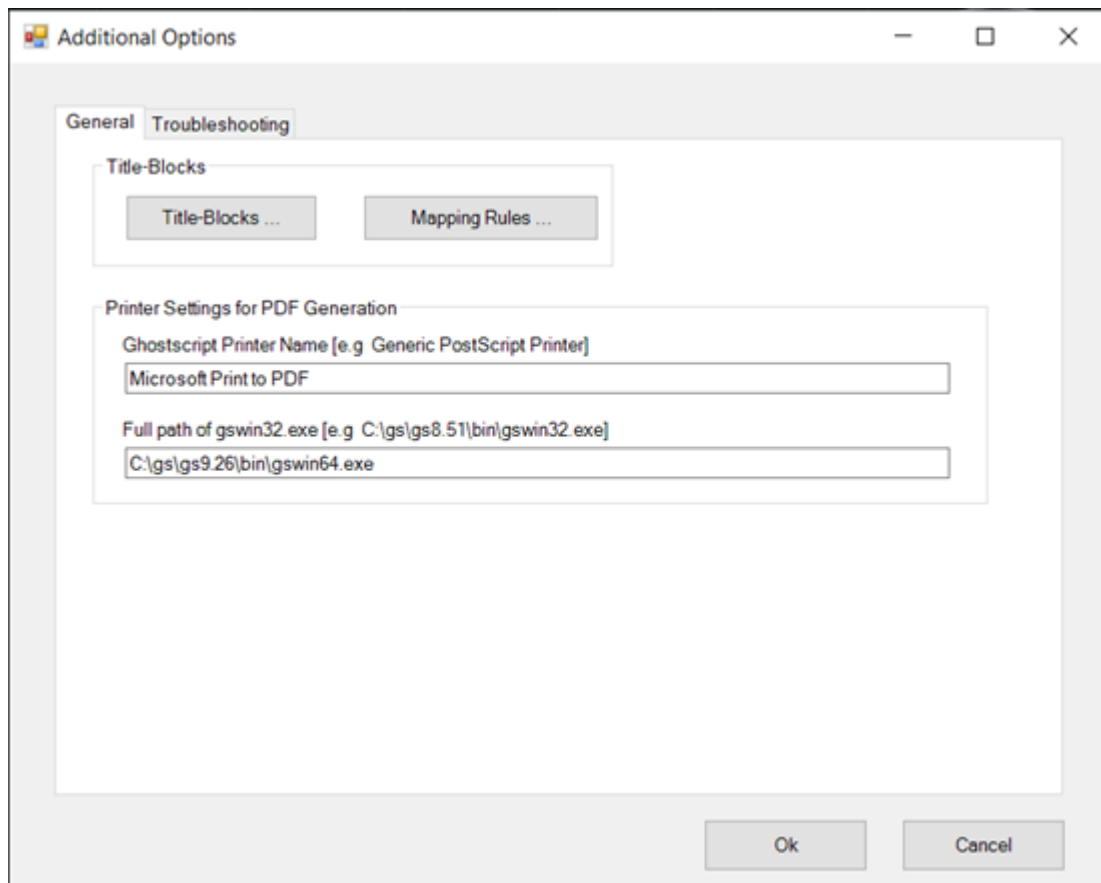
- 3 Select the **Title Block**, as shown below.



- 4 Click the selected block. The following Enhanced Attribute Editor dialog box opens.



- 5 In the Enhanced Attribute Editor dialog, values listed under “Tag” are the fields in the title block. You can enter the value for these field. In the above example, value for DESCRIPTION tag is specified as "From AutoCAD". Note the names of the tags (field in title blocks) for which you want to define mapping. Now, click **Additional options** on the integration menu/toolbar. The following dialog box opens.



- 6 Click **Title- Blocks** on the Additional options dialog box. The Add/Edit Title Block Mapping Rule dialog box opens as shown below. Clicking “Mapping Rules...” in the above dialog will open the dialog to define mapping rules. On this click on “Add” button Add/Edit Title Block Mapping Rule dialog box appears. Specify the Tag (title block field name), Direction of mapping (To_CAD, To_PLM or ToBoth), PLM Table, and PLM table field.



Note: You might see an updated dialog instead of above, the details of which are provided in this section at the end.

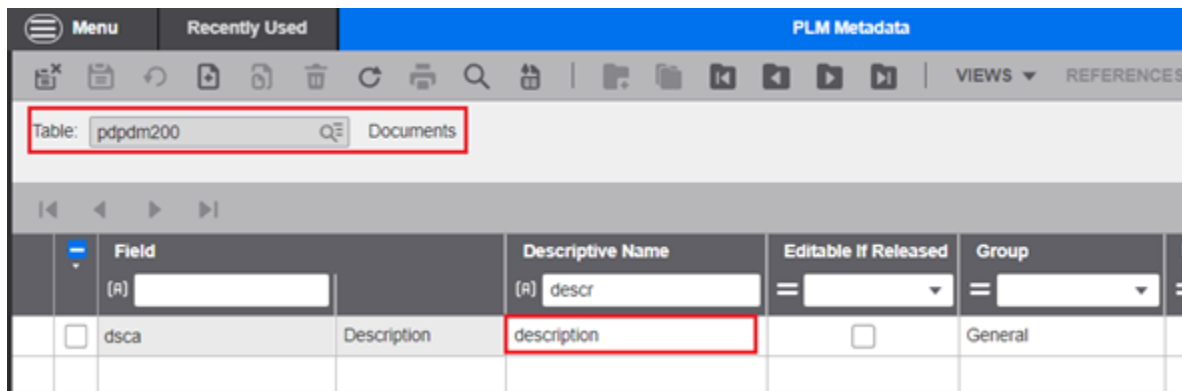
Note the following in Add/Edit Title Block Mapping Rule dialog box:

- Tag [CAD] value is DESCRIPTION. This is same as the name displayed in the dialog of `eattedit` command.
- Direction: ToCAD means that the direction of mapping is from PLM to CAD.
- PLM Table is the table name in PLM. For example, DOC, ITEM, FILE etc.
- PLM Field: DOC_NAME is the description field in the table DOC in the PLM.

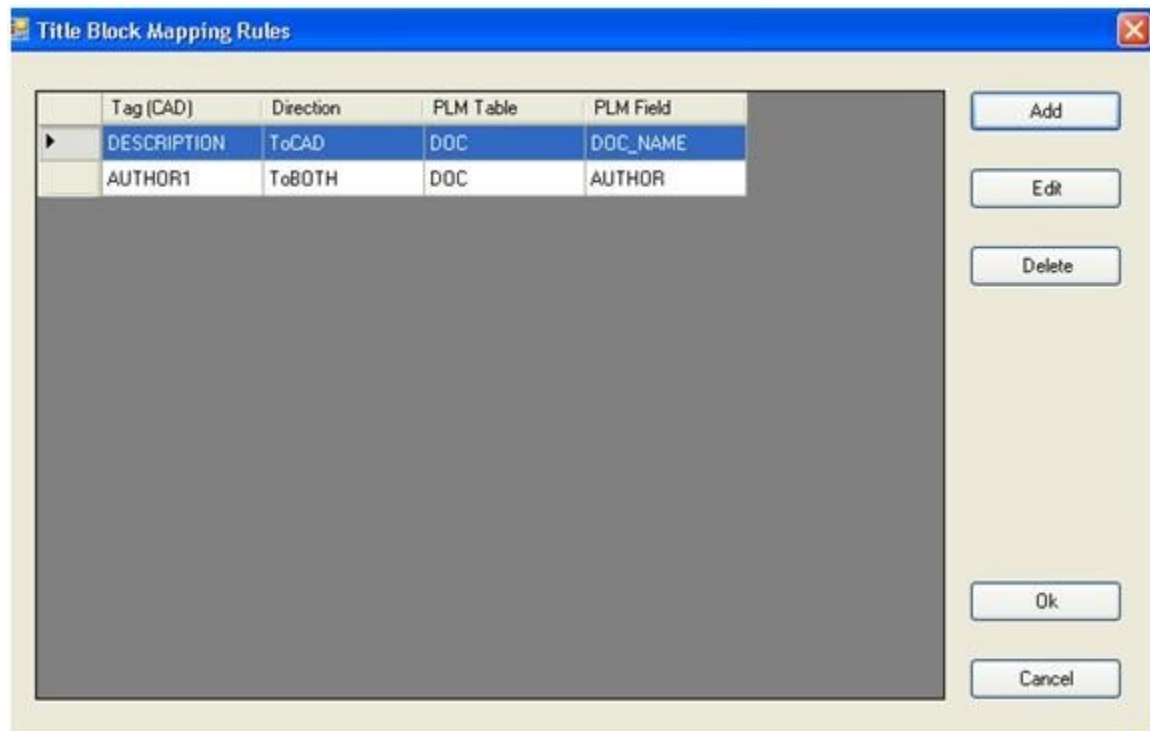
Note: While entering values for PLM Table and PLM Field it is recommended to take the field names from the customization tool. You should be careful in specifying the direction for mapping the field values, as the integration does not check if any rule is valid or not.

Example

To perform mapping from Description field of document table in PLM,
Select the exact descriptive name for the mapping fields from PLM Metadata.



Note the PLM Table and attribute name. Use the same value in the Define Mapping rule dialog in AutoCAD integration. The following screen shows the Typical mapping rules.

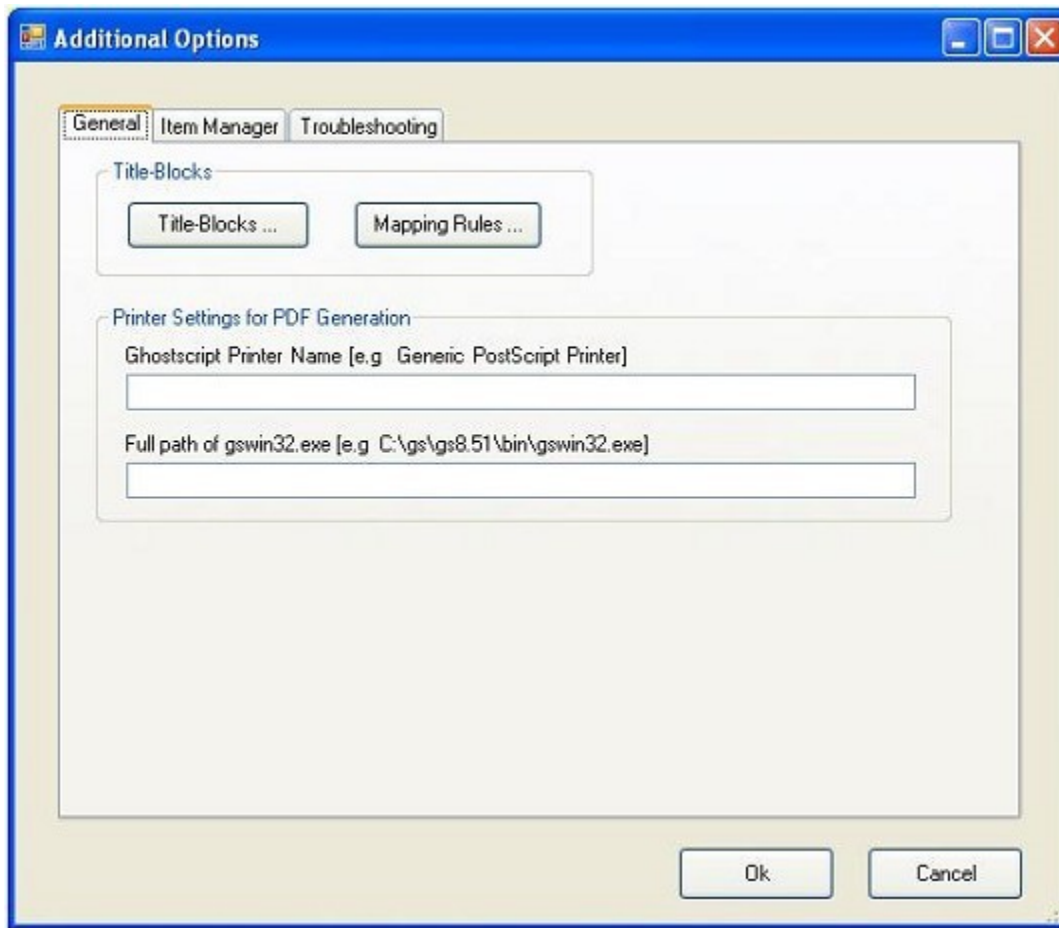


After the mapping rules are defined, mapping is performed during Save to PLM and/or Edit File operations.

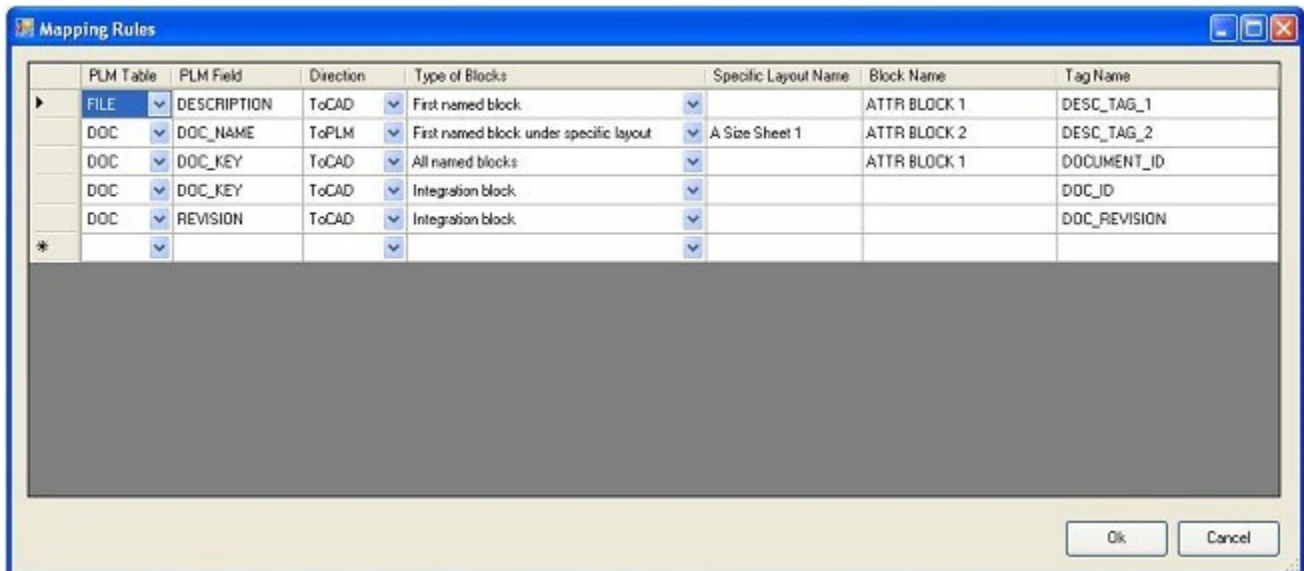
Note: Defined mapping rules are applicable to all Title Blocks.

Latest Changes in Mapping Functionality

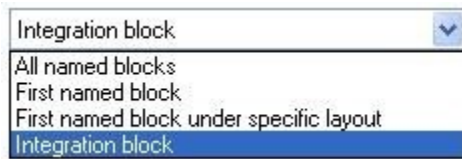
Following changes are implemented to merge the mapping rules information into a single dialog:



When you click **Mapping Rules**, the following screen is displayed:



Following are the options in the Type of Blocks Combo-box:

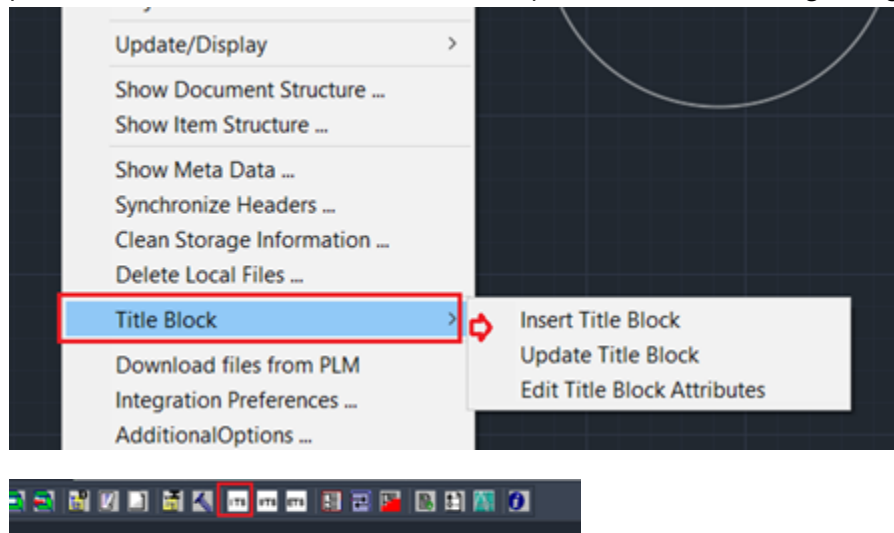


Depending on the Mapping direction, the tag values under the specified blocks of the CAD files will be either read or will be updated.

- First named block: If the mapping direction is To-CAD, then integration will update the tag value of the first instance of the block. If the mapping direction is To-PLM, then integration will read the tag value from the first instance of block.
- First named block under specific layout: In this case, you must specify the Specific Layout Name. If the mapping direction is To-CAD, then integration will update the tag value of the first instance of the block in the specified layout. If the mapping direction is To-PLM, then integration will read the tag value of the first instance of the block from the specified layout.
- Integration block: This refers to the mapping rules for the title-blocks that are inserted via integration menu Insert title-Block. Set the Block Name to empty for all the mapping rules.
- All named blocks: If the value is All named blocks, then integration will update tag values of all the blocks. Set Mapping Direction Name as ToCAD for all the mapping rules.

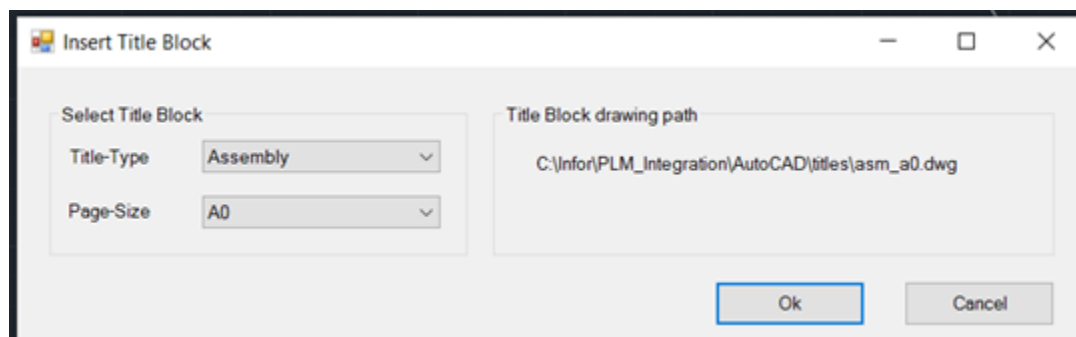
Chapter 20: Title blocks

A collection of AutoCAD Title Blocks used for a particular design purpose is called a Title Block Type. Integration provides features to Insert Title-Blocks and update the Title-Block tags using mapping rules.



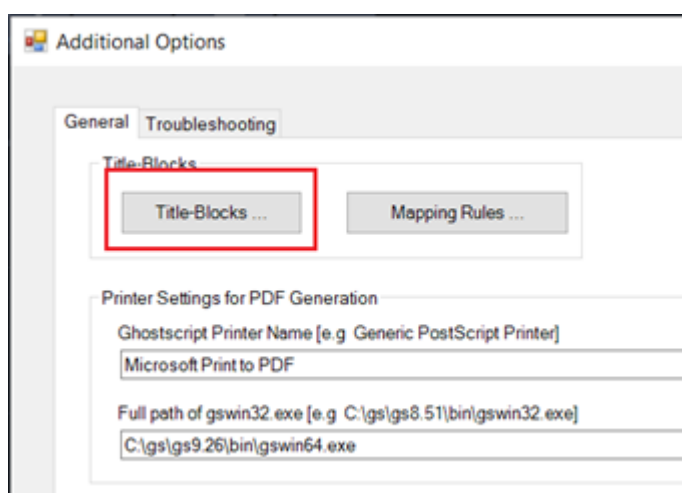
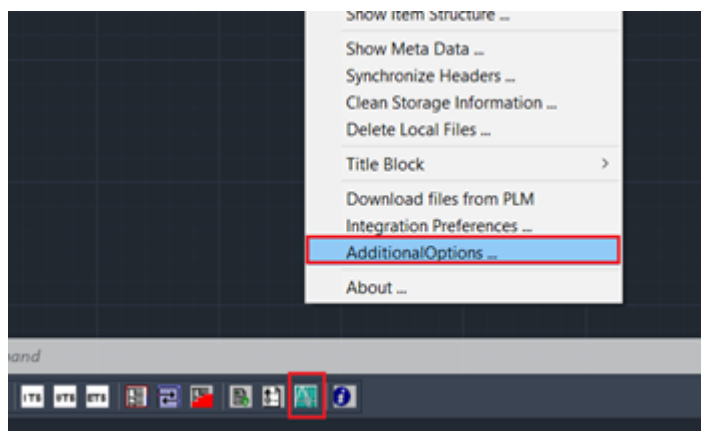
How to Insert Title-Block

Click on integration menu "Insert Title-Block", The Insert Title Block dialog box opens. If you are using the integration for the first time, the Title Type and Page Size will have no values (as shown below).

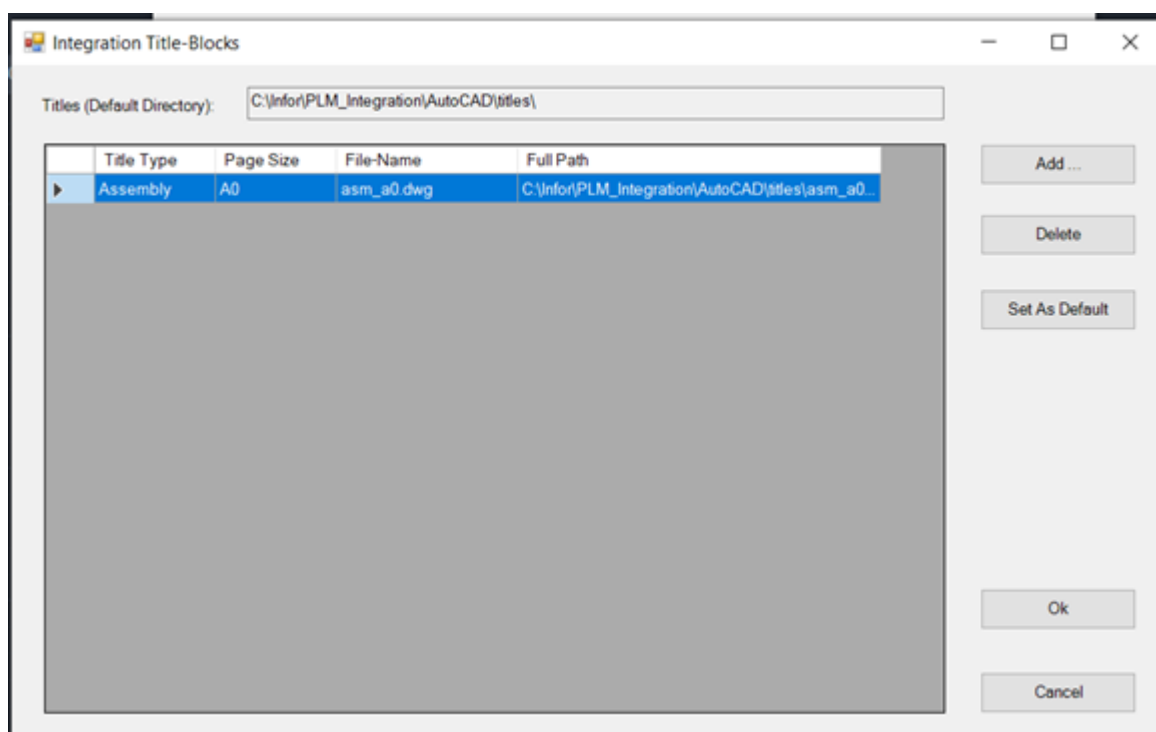


For this, you need to first configure the title type and page size values which can be used in the drawings. Below are the steps to configure the same:

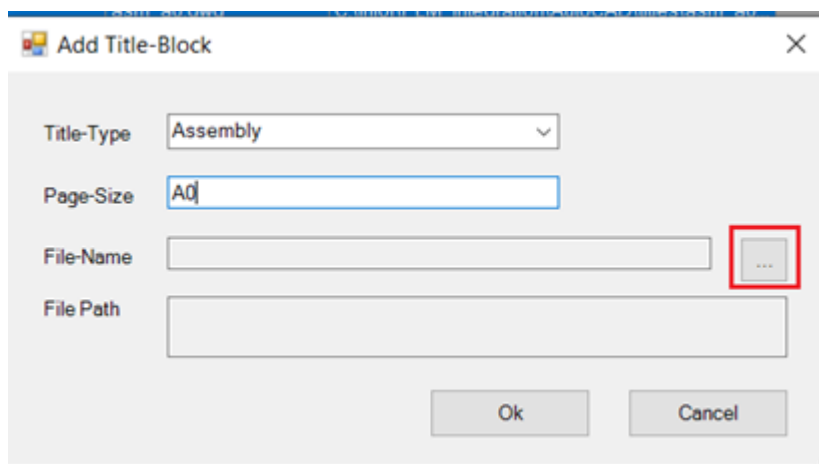
- 1 In the PLM menu / Toolbar, click **Additional Options** as shown below.
- 2 The Additional Options dialog box opens as shown below.



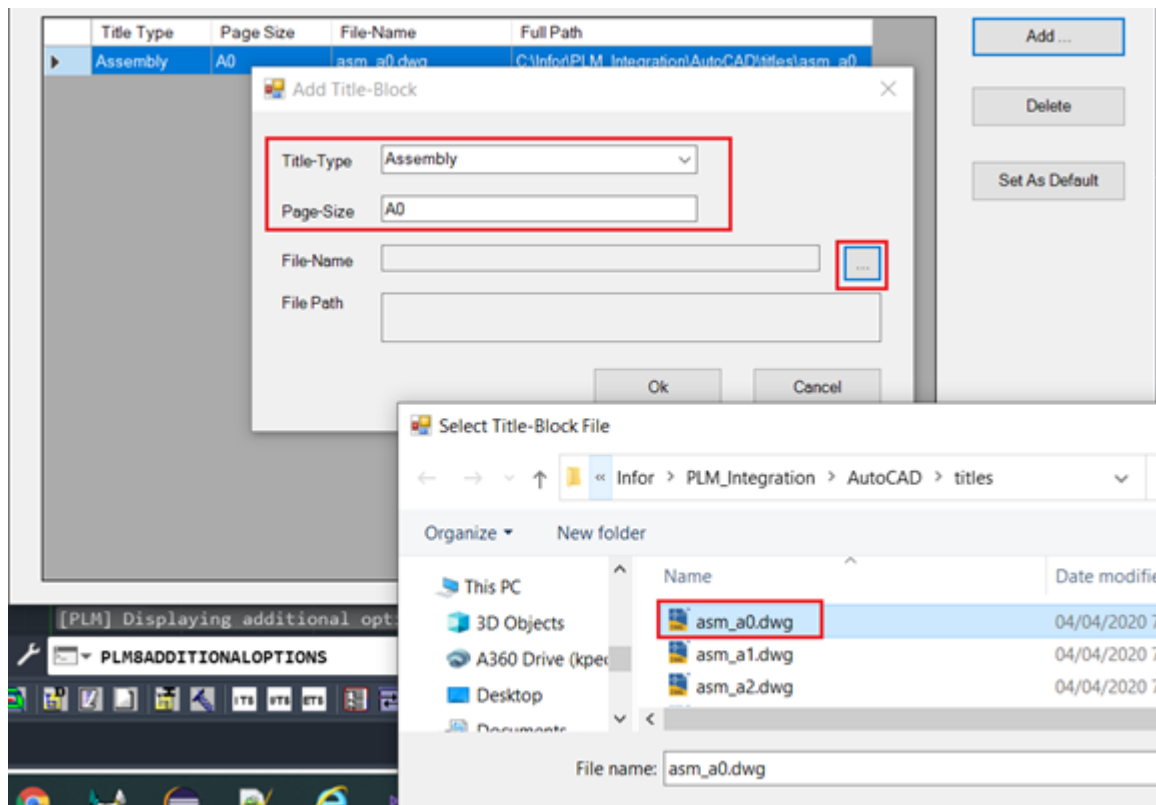
- 3 Click **Title Blocks**. The Integration Title-Blocks dialog box opens.



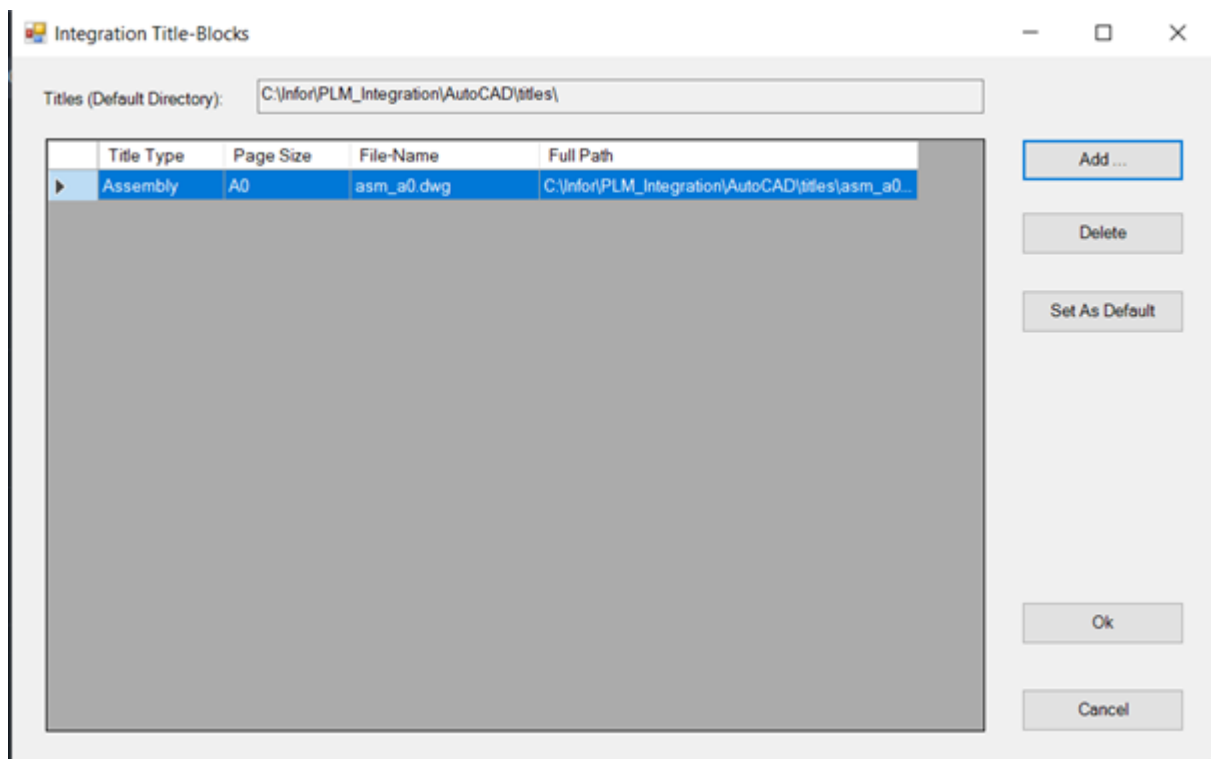
- 4 Click **Add** in Integration Title-Blocks dialog box.



- 5 Enter the values for Title-Type and Page-Size and click Browse. The Select Title-Block File dialog box opens. In this dialog browse to %cfe_client_home%\AutoCAD\Titles folder and select the corresponding Title-Block file. You can create your own Title-Block files and can place here.

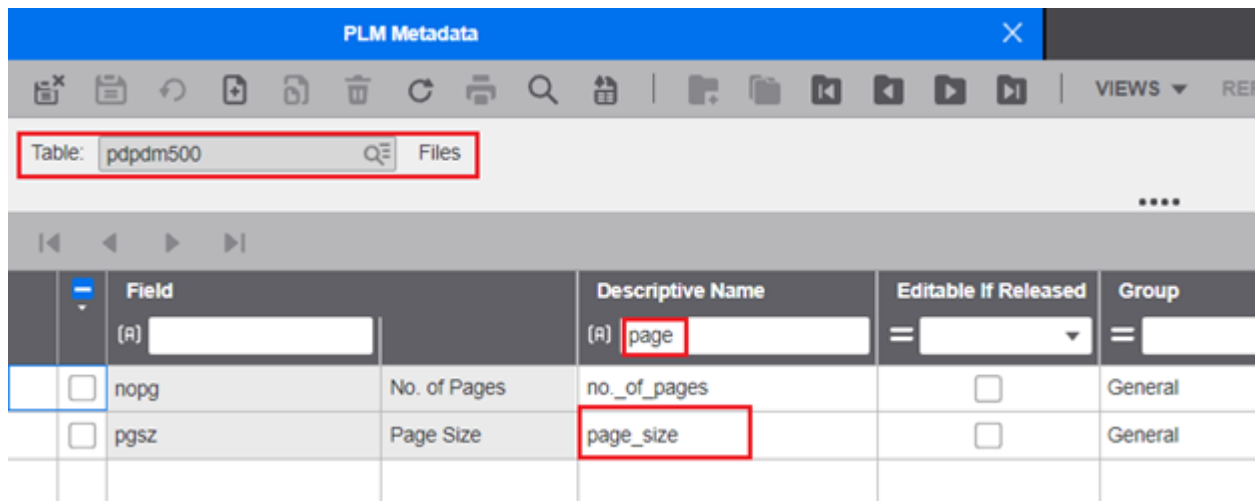


6 Typically the entries in the tables will be as shown below.



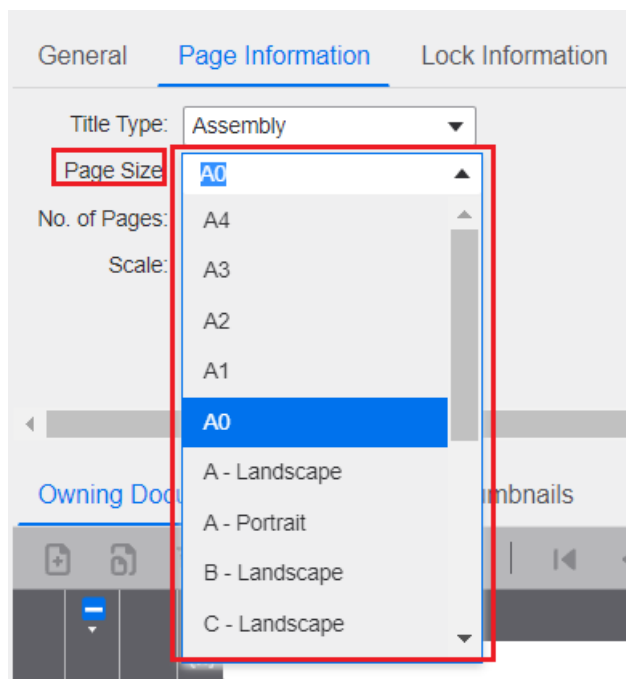
- 7 Values of A0, A1, etc should exist in the PLM, Page Size.

From File PLM Metadata



Field	Descriptive Name	Editable If Released	Group
(A) []	(A) page	= []	= []
<input type="checkbox"/> nopg	no._of_pages	<input type="checkbox"/>	General
<input type="checkbox"/> pgasz	page_size	<input type="checkbox"/>	General

From File session



General **Page Information** Lock Information

Title Type: Assembly

Page Size A0

No. of Pages: A4

Scale: A3

A2

A1

A0

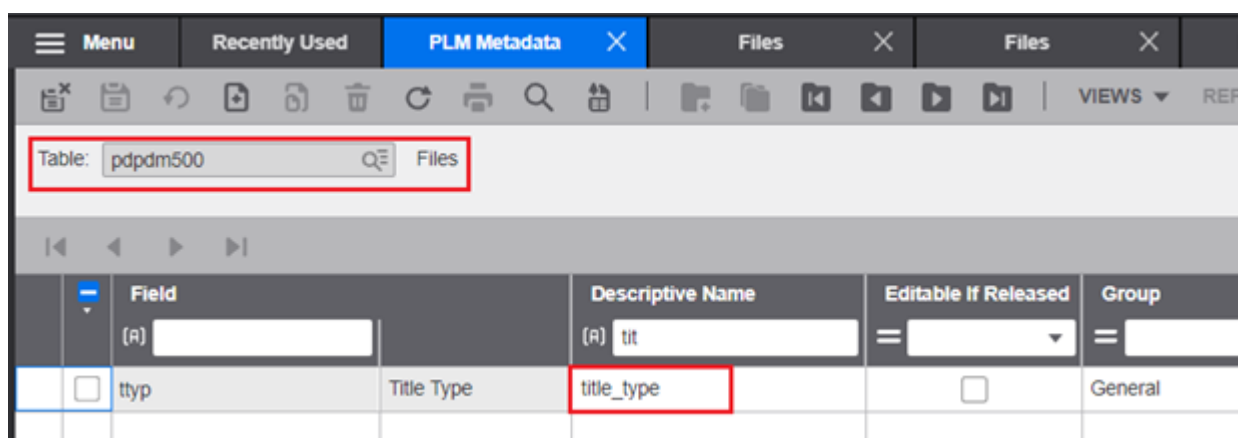
A - Landscape

A - Portrait

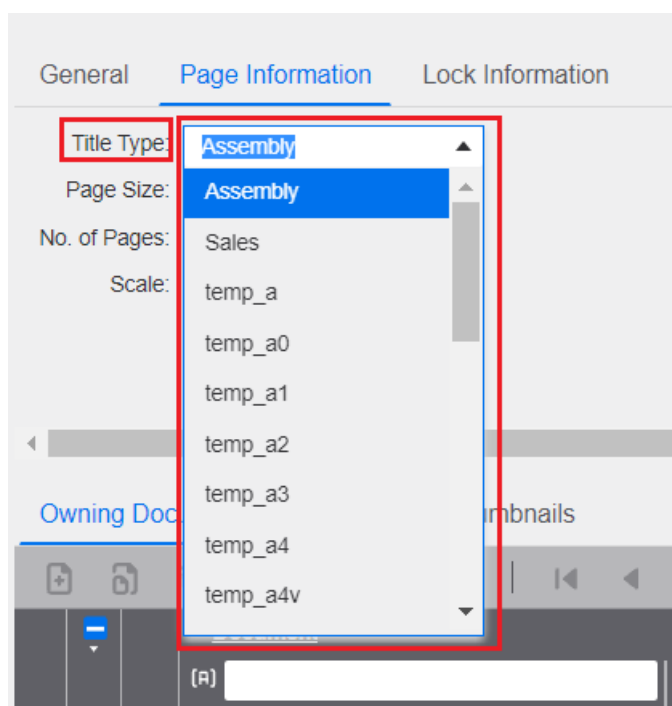
B - Landscape

C - Landscape

- 8 Values of Assembly, Sales, etc should exist in the PLM, Title-Ty.



From File session



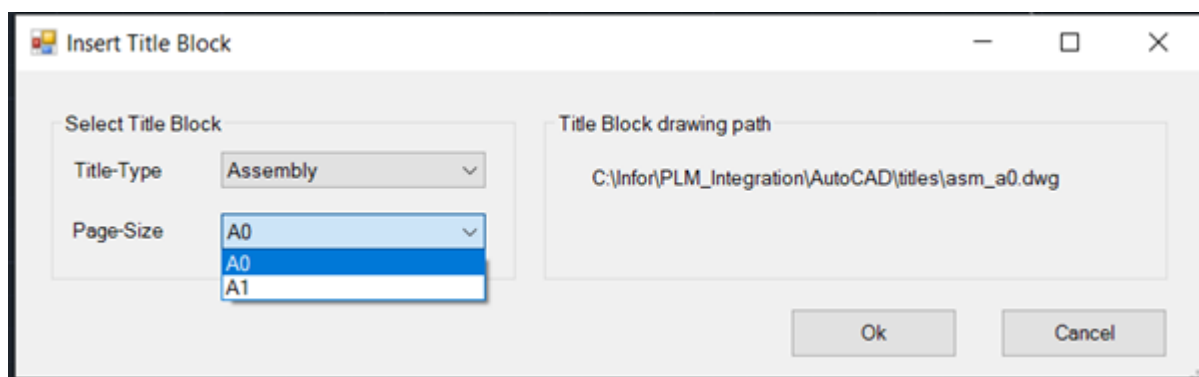
- 9 As part of additional validation open the file %cfe_client_home%\AutoCAD\AcPlmIntegPref.xml in Internet Explorer to ensure that the entries are available as shown below.


```

- <IntegrationPreferences>
- <Titles Location="C:\Program Files (x86)\PLM_Client\AutoCAD\titles\">
- <Assembly>
  <A0>asm_a0.dwg</A0>
  <A1>asm_a1.dwg</A1>
</Assembly>
- <Sales>
  <A0>sale_a0.dwg</A0>
</Sales>
</Titles>
<MappingRules />
- <Annotations>
  <FIND_NO start="0" increment="10" />
  <FACTORS DonutOutsideRadius="0.333" BlockSize="1" CircleRadius="1.5" />
  <LayerColors CircleColor="-32640" LineColor="-128" TextColor="-8323200" />
</Annotations>
<GhostscriptPrinterSettings GhostscriptPrinterName="" GhostscriptPrinterWin32ExePath="" />
- <Troubleshooting>
  <SkipReloadOfChildXrefUnderRootFileDuringDownload>False</SkipReloadOfChildXrefUnderRootFileDuringDownload>
  <SkipMappingForAllFilesDuringDownload>False</SkipMappingForAllFilesDuringDownload>
  <SkipOpenCloseLogicDuringSaveToPLM>True</SkipOpenCloseLogicDuringSaveToPLM>
  <AllowOpeningFileAsDB>False</AllowOpeningFileAsDB>
</Troubleshooting>
</IntegrationPreferences>

```

- 10 After validation click insert title block and you can see the entries to be selected.



- 11 Once you press **OK**, the selected title-Block will be inserted in the Model space layout of the AutoCAD file.

Note: In case of AutoCAD Mechanical and AutoCAD Mechanical Desktop, it is possible to insert/use the Title-Blocks that are provided by these applications through the "AMTITLE" command. for this, you need to set the preference "Support Mechanical Desktop Title-Blocks". when you check this option and click on "Insert title-Block", integration will prompt you with the AutoCAD application dialog to insert Title-Block (i.e. dialog shown using AutoCAD "AMTITLE" command). Once selected, integration will proceed with this Title-Block.

In case a Title-Block (inserted manually using the "AMTITLE" command) already exists in the AutoCAD file, and user clicks on "Insert title-Block", then user will be asked whether to continue with this existing Title-Block.

please refer chapter "[Mapping Rules](#)" on page 43" for details on how to perform mapping for the attribute tags in the Title-Blocks.

Chapter 21: AutoCAD and Toolkit Preferences

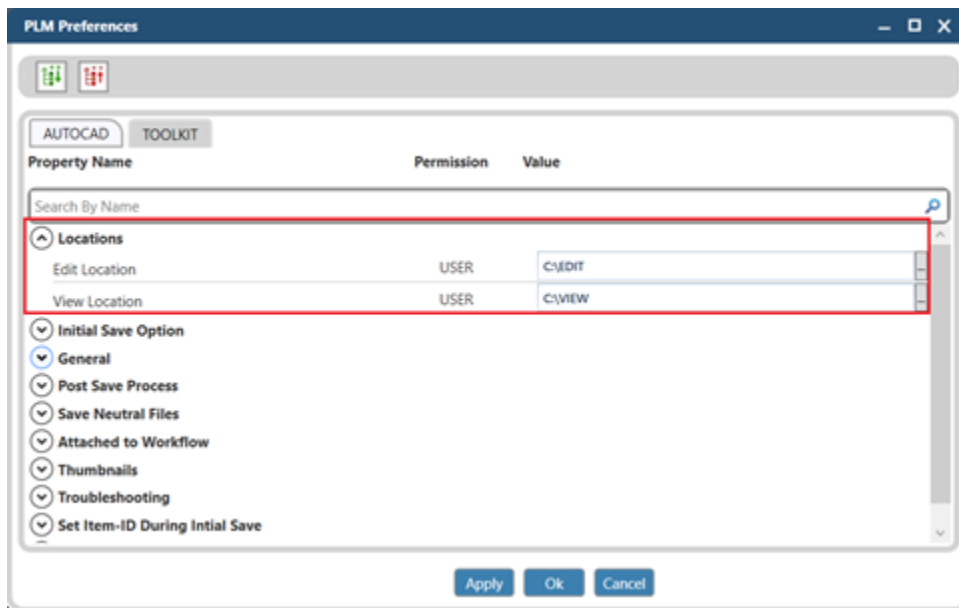
The AutoCAD tab includes properties that control the behavior of the PLM integration for AutoCAD.

This tab includes the following groups:

- [Locations](#) on page 58
- [Initial Save Option](#) on page 59
- [Post Save Process](#) on page 65
- [Save Neutral Files Option](#) on page 66
- [Attach to Workflow](#) on page 68
- [Integration Preferences for Thumbnails](#) on page 32
- [Troubleshooting Option](#) on page 69
- [Set Item-ID During Initial Save](#) on page 69

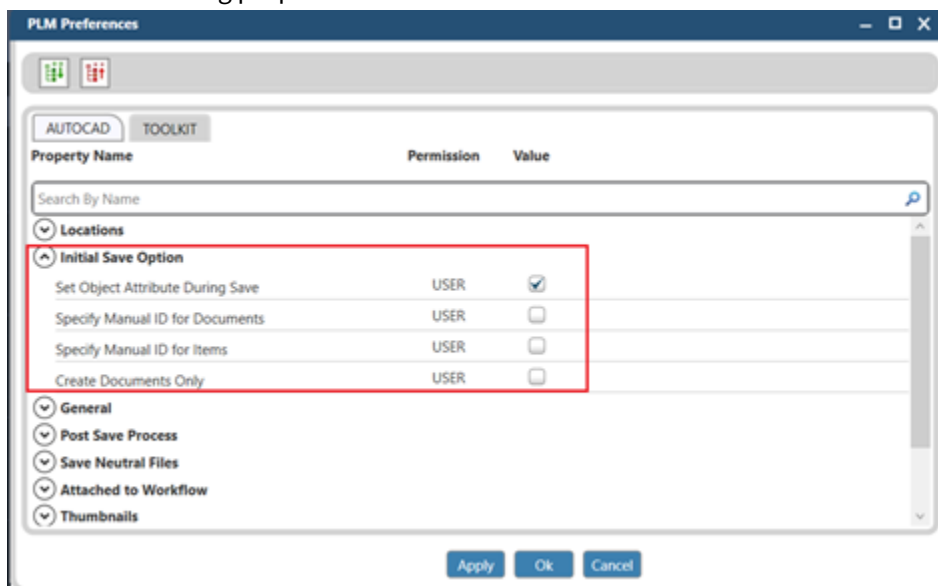
Locations

To carry out various commands, such as View File or Edit File from the Results panel of the integration query, PLM must locally download the files that you want to view or edit in AutoCAD. In the Locations section you can specify the folders in which PLM must locally download the files that you want to view or edit in AutoCAD. If required, you can specify the same folder for both viewing and editing, but if you do this, the Disable Actions on view files property will not work.



Initial Save Option

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



Set Object Attribute During Save

If this check box is selected, you can define attributes for the files, documents and items that are being saved to PLM. For further information, see [Saving to PLM](#) on page 18.

Specify Manual ID entry for Documents and/or Items

When **Manual ID** check box is selected in the preferences, you can specify IDs for the Item and Documents in the Set Object Attributes

Initial Save Option

Set Object Attribute During Save	USER	<input checked="" type="checkbox"/>
Specify Manual ID for Documents	USER	<input checked="" type="checkbox"/>
Specify Manual ID for Items	USER	<input checked="" type="checkbox"/>
Create Documents Only	USER	<input checked="" type="checkbox"/>

window.

Create documents only

If this check box is selected and when you save a CAD file to PLM, a document is created in PLM, but no items are created. For further information, see [Saving to PLM](#) on page 18.

Limitations

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

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

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

General Option

The General Option includes the following properties:

Support Mechanical Desktop Title-Blocks

In case of AutoCAD Mechanical and AutoCAD Mechanical Desktop, it is possible to insert/use the Title-Blocks that are provided by these applications using the "AMTITLE" command. To enable this, you must set the preference "Support Mechanical Desktop Title-Blocks". If this checkbox is selected and click "Insert title-Block" available in Infor PLM toolbar, integration will prompt you with the AutoCAD application dialog to insert Title-Block (i.e. dialog shown using AutoCAD "AMTITLE" command). After you specify, integration will proceed with this Title-Block.

If a Title-Block (inserted manually using the "AMTITLE" command) exists in the AutoCAD file, and user clicks on Insert title-Block", then the system prompts the user to confirm whether to continue with this existing Title-Block.

Allow Link to Released Item

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

File Name Uniqueness

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

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

Disable Actions on View File

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

Note that for the **Disable Actions on View File** option to work, the view and edit locations specified in the Locations option must be different. For further information, see [Locations](#) on page 58.

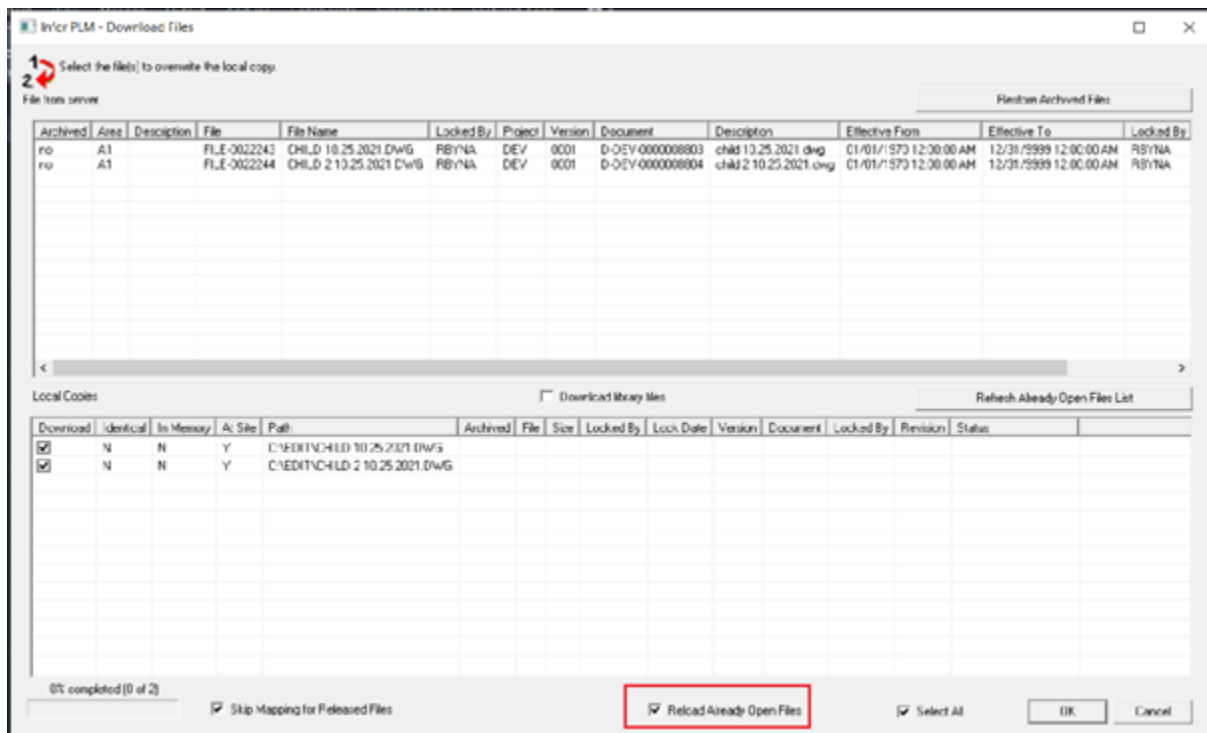
Query Search Default

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

- ITEM
- DOCUMENT
- FILE

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.

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

Show Message on Successful Open

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

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

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

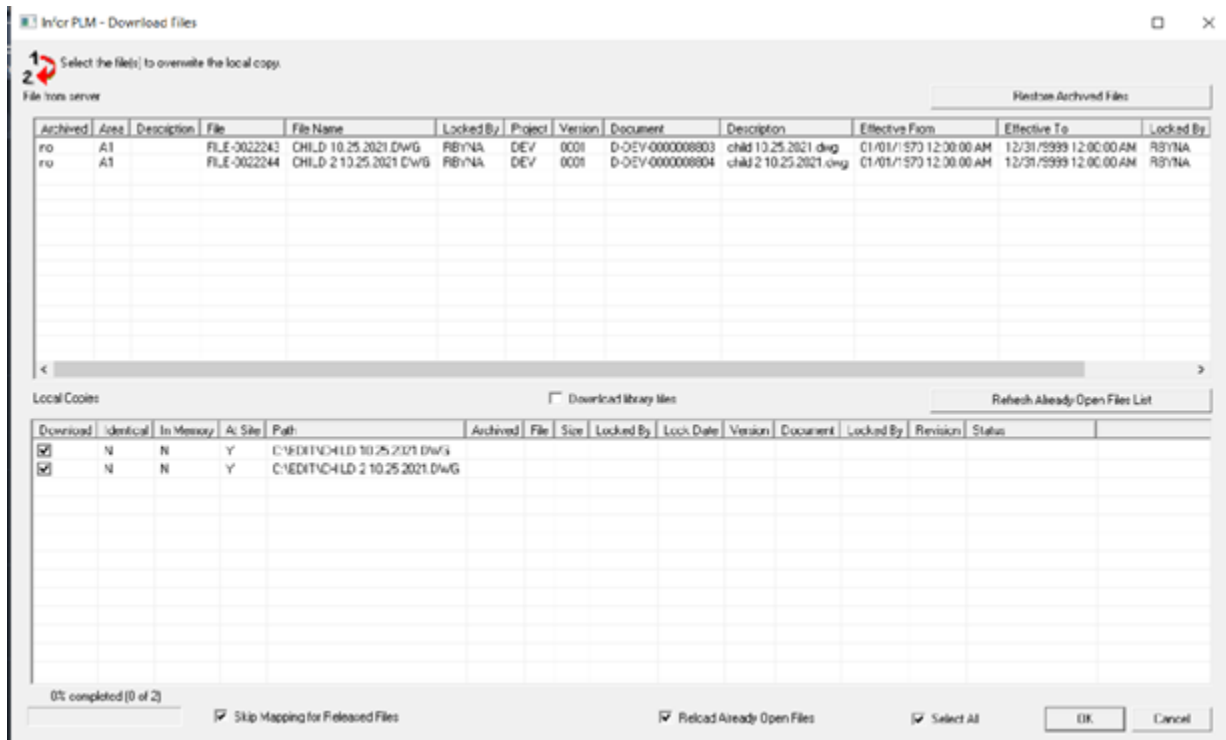
Take Ownership During Edit File in Integration

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

Skip Meta Data Comparison During Download

When **Skip Meta Data Comparison During Download** check box is not selected in the preferences.


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



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.

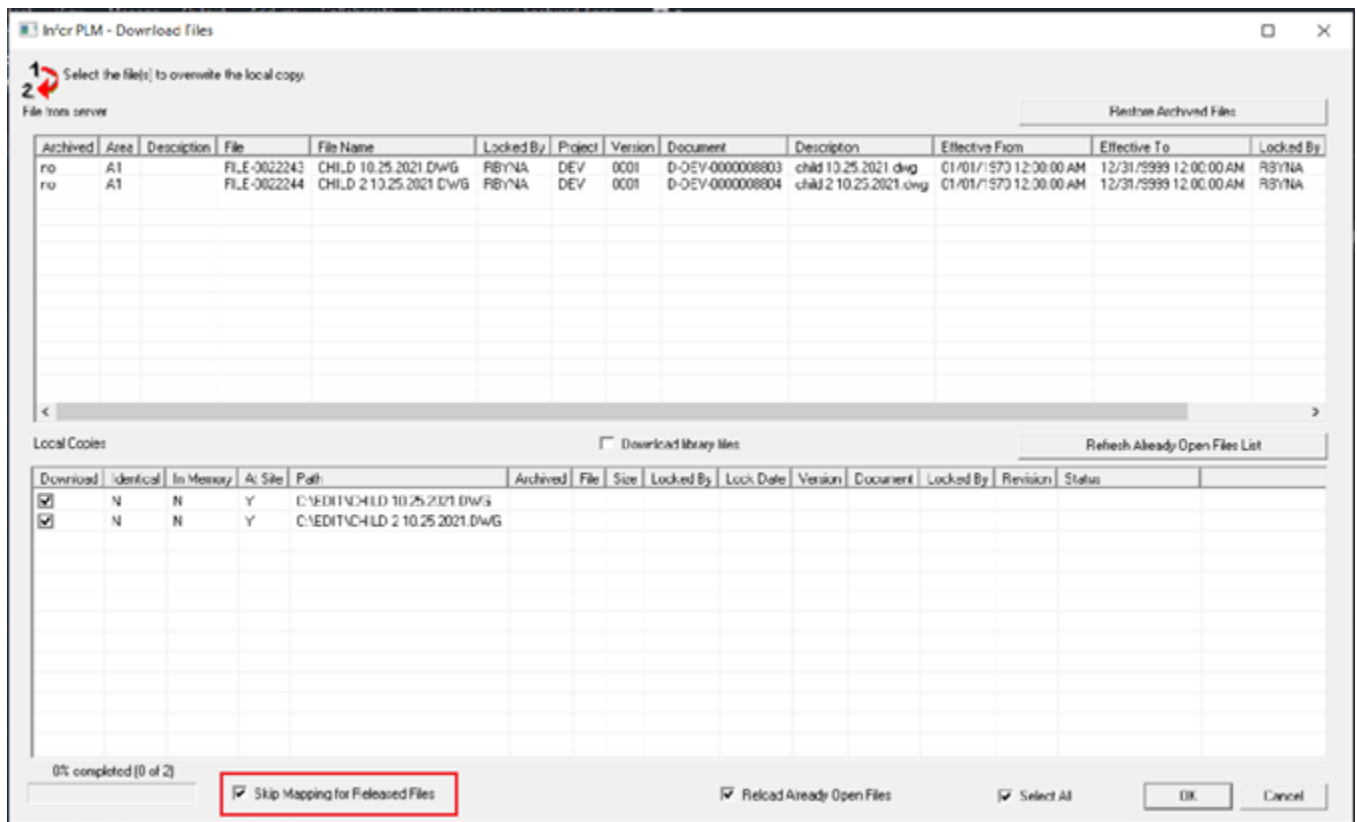
Skip Mapping for Released Files

When you select the preference Skip Mapping for Released Files during Download, the integration does not perform the mapping for the files which are in Released status in the PLM during the download process. It is recommended to select the check box to improve the download performance of large assemblies in the View/Edit File operation. However, you should be aware that some file preferences may not be up to date with PLM.

 **General**

Support Mechanical Desktop Title-Blocks	USER	<input type="checkbox"/>
Allow Link to Released Items	USER	<input type="checkbox"/>
File Name Uniqueness	USER	Unique per project
Disable actions on view files	USER	No
Query Search Default	USER	<input type="checkbox"/>
Allow reload of already opened files	USER	<input checked="" type="checkbox"/>
Show message on successful open	USER	<input checked="" type="checkbox"/>
Warn user when the same item is linked to parent and child components	USER	<input checked="" type="checkbox"/>
Take Ownership during Edit File in Integration	USER	<input checked="" type="checkbox"/>
Skip Meta Data comparison during Download	USER	<input type="checkbox"/>
Skip Mapping for Released files during Download	USER	<input checked="" type="checkbox"/>

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



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

Post Save Process

Show message on successful save

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

Check In Cleanup

The available options are:

- Delete File: After check-in, ONLY the root file will be deleted from local system. All other files (within the structure) will be read-only.
- Make files read-only: After check-in, all the files (within the structure, including the root file) will be made read-only in the local system.

Save and Unlock Cleanup

The available options are:

- Delete File: After save and unlock, ONLY the root file will be deleted from local system. All other files (within the structure) will be read-only.
- Make files read-only: After save and unlock, all the files (within the structure, including the root file) will be made read-only in the local system.

Save Neutral Files Option

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

- Parts
- Assemblies
- Drawings

Following properties are relevant to the neutral files:

- Generate Neutral Files
- Show Neutral files during save

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

Generate Neutral Files during

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

- On All Actions – Neutral files will be generated during Save to PLM, Save and Unlock and Check-In actions.
- On Save and Unlock – Neutral files will be generated only during Save and Unlock.
- On Check-In – Neutral files will be generated only during Check-In.

Save Neutral Files Formats

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

- BMP
- PDF

Layouts for BMP Generation

Specify the layout for BMP generation. The supported layout for BMP generation:

- Model Space Layout
- Active Layout Only
- All Layouts

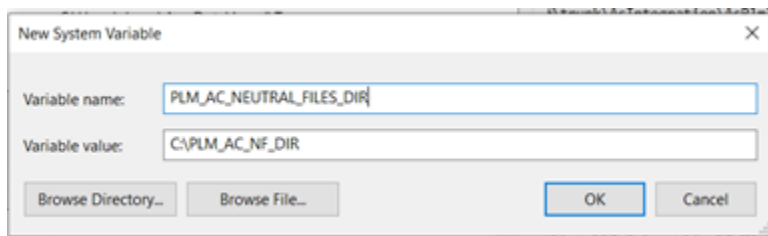
Layouts for PDF Generation

Specify the layout for PDF generation. The supported layout for PDF generation:

- Model Space Layout
- Active Layout Only
- All Layouts (Single PDF)
- All Layouts (Separate PDF)

Location of generating Neutral Files

By default, the neutral files are created in the same model file location (or) <PLM_Client>\AutoCAD\Neutral Files folder. However, you can change the folder location by specifying the new folder path in the environment variable as shown below:



If you set the environment variables as shown above (PLM_AC_NEUTRAL_FILES_DIR), the neutral files are created in H:\PLM_SW_NF_DIR. Ensure that no other files are manually created in the folder, because the folder is cleaned when you close AutoCAD.

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 all related objects
- Allow edit Workflow
- Action for object locked by Business Process

Attach to Business Process

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

Allowed values

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

Attach all related objects

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

Action for object 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.
- **Always attach as unlocked:** By default object will be attached to Business process and unlocked.

Troubleshooting Option

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

Set Item-ID During Initial Save

The available options are:

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

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

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

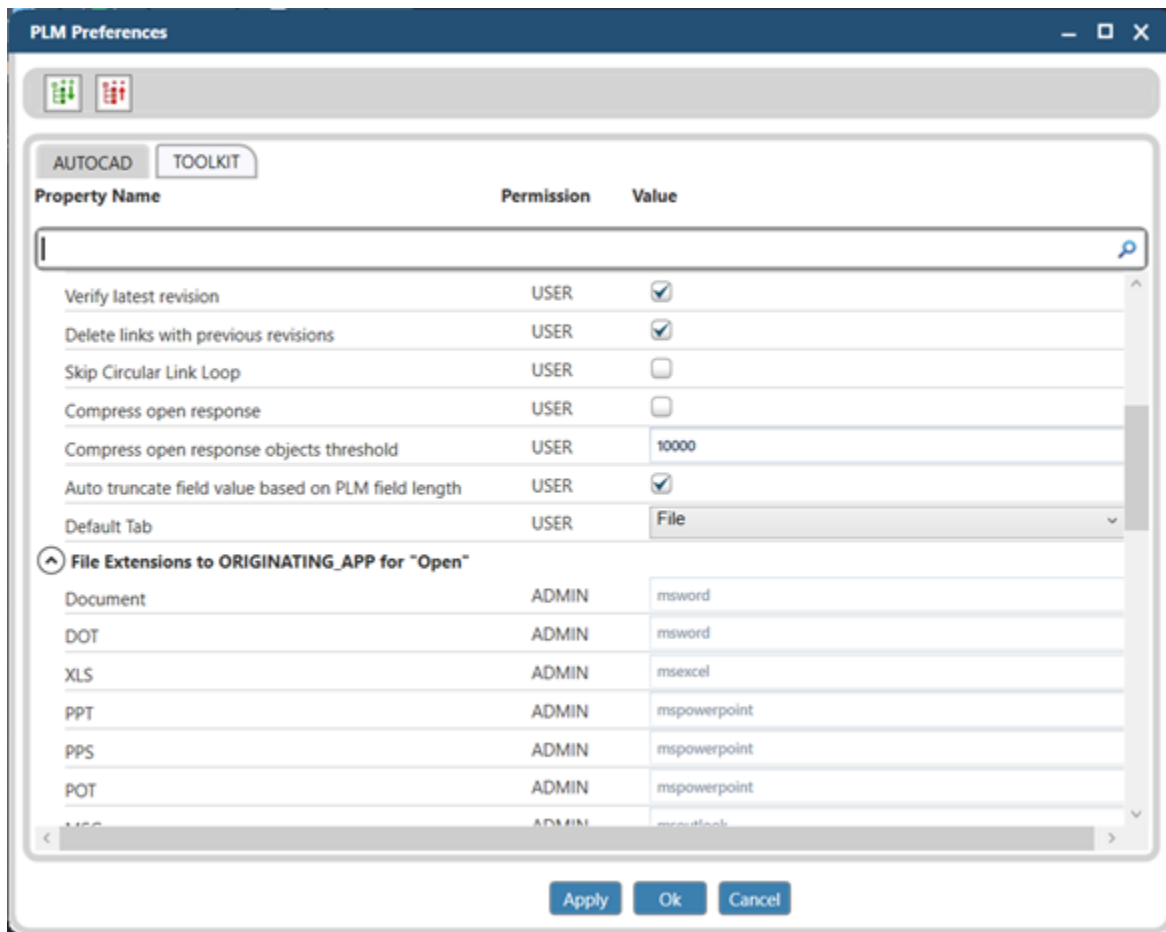
- Specify the “Custom property to use as Item-ID”. For example, specify “CAD Item Number”. This can be the name of any custom property that is defined in the CAD file.
- The Create Documents Only preference must not be selected.

Create a part file. Create a custom property and specify the name as defined in Custom property to use as Item-ID preference. For example, CAD Item Number. Specify a value for the property. For example, specify 1700.11.

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

Toolkit tab - Introduction

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



This tab includes the following options:

- General Option
- Toolkit Extensions to Original Application

Synchronize All Files During "Save"

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

Ignore Items

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

Disable BOM Creation/Modification during "Save"

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

Show synchronize message during "Save"

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

Show Top Down Load report

If this check box is selected, the PLM integration for AutoCAD 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 AutoCAD 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. This link enables PLM to determine how to open the file.

Chapter 22: Setup and administration

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

For further information on the setup data, see

- [Installation and setup](#) on page 72
- PLM development toolkit preferences setup
- Support for PDF format neutral files
- Setting up title block types in PLM
- Title block parameters and PLM attributes mapping
- Special considerations
- [Troubleshooting](#) on page 75

Installation and setup

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

- Before a new version of the PLM integration for AutoCAD is installed, the previous installation must be uninstalled.
- Make sure that you have installed the latest AutoCAD service pack.
- After the AutoCAD integration installation is completed, and before starting to use the integration, check that the path for the AutoCAD executable is correct in the AutoCAD shortcut. For more information, refer to [Checking the AutoCAD shortcut path](#).
- Check that the path of the executable or batch file for the integration is correctly defined, in order for the PLM Edit/View in Integration to work correctly. For more information, refer to [Check AutoCAD executable path in RunCad.properties](#) on page 73.
- Check that the path for title block types is set correctly. For more information, refer to [Checking the title block titles path](#).
- All integration users must be defined in PLM and assigned to the relevant projects.
- Vault parameters must be set for each PLM project. For more information, refer to [Vault parameter configuration](#) on page 73.

After installing a new version of the PLM integration for AutoCAD, make sure the working directory for viewing or editing files in AutoCAD is specified. For more information, refer to **General** tab.

Check AutoCAD executable path in RunCad.properties

To ensure that commands such as the Edit/View File in Integration command from PLM operate correctly, the path of the AutoCAD application executable or batch file must be defined in the RunCad.properties file. This file is located in the following folder:

[INSTALL_DIR] Infor_PLMClients\DeProxy.

The following default entry exists in this file and should be adjusted to match your installation:

```
AutoCAD="C:\Program Files\ACAD2006\acad.exe"
```

In general, this entry is updated automatically during installation, but occasionally the installation process is not able to determine the required path.

If you run the Edit File in Integration command in the integration query tool, a copy of the PLM workspace is stored locally in the \...\<PDM client>\AutoCAD2006\work directory. To modify the file, you need to check it out.

Note: The information in the RunCad.properties file must always be put between quotes.

Note: For the Edit File in Integration and View File in Integration commands to work for AutoCAD, AutoCAD must be running and connected to PLM.

Vault parameter configuration

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

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

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

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-in	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	Check out all objects
Related items	Check in all objects	Check out all objects
Related files Infor	Check in all objects	Check out all objects

Chapter 23: Troubleshooting

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

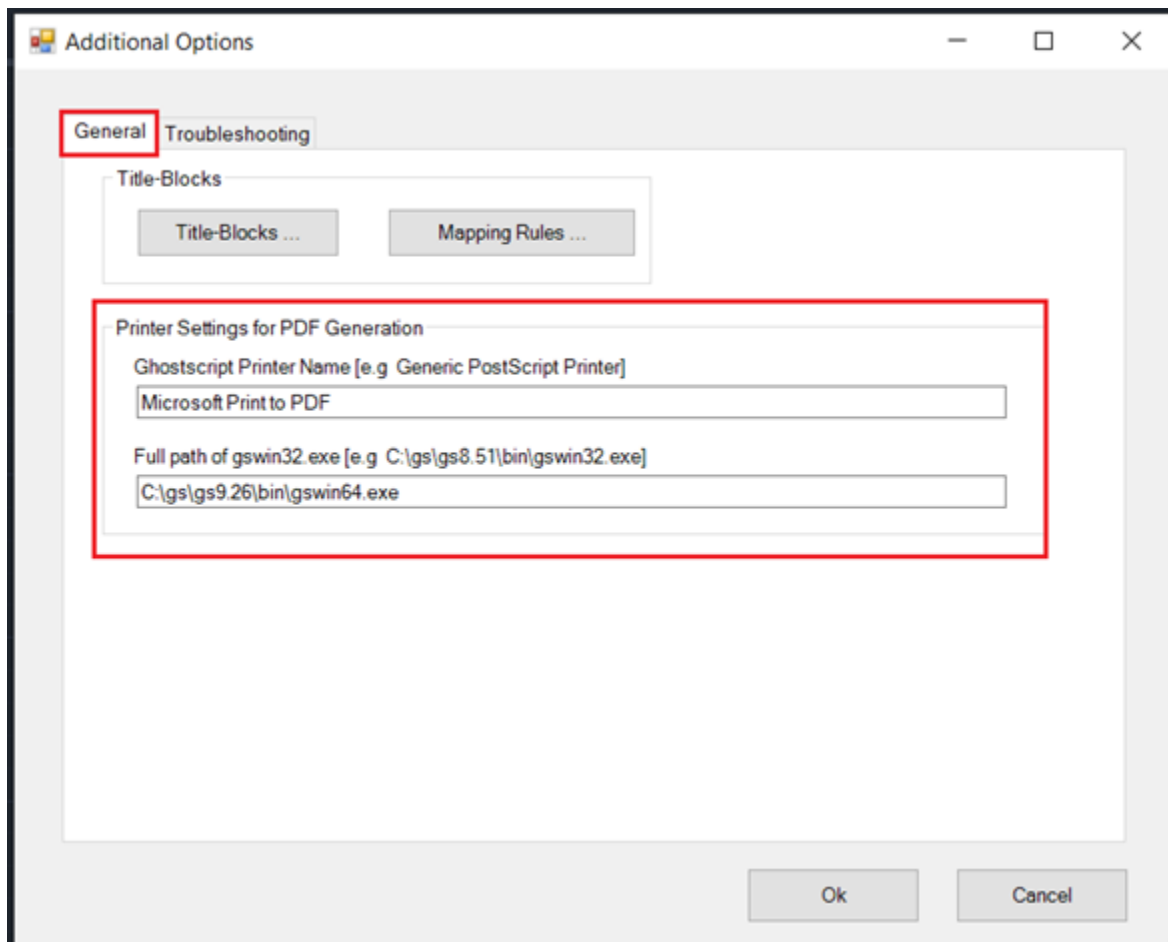
Error Messages

If the user is unable to open the integration, this may be due to one of a number of reasons. The error message displayed to the user should include an error number. The following table lists the error numbers with an explanation of the possible problem:

Number	Meaning
1	Invalid number of input parameters.
2	Failed to create DeProxy.exe instance.
3	Invoke Action Failed in Toolkit.
4	COM Error occurred during invocation.
5	An exception occurred during invocation.
70001	No integration exists for this file format.
70002	Missing originating application parameter.
70003	Development Toolkit Callback not set.
70004	Server Error cannot be identified.
70006	The Open operation failed.

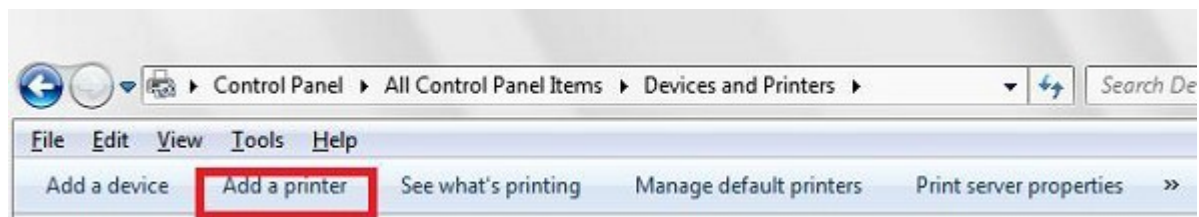
PDF Generation

Install Ghost Scripts to generate PDF neutral files in AutoCAD. Then set the preferences in the additional preferences as shown in the following screen.



Generation of PDF files on Windows 7

- 1 In the control panel, double click **Devices and printers** and in the dialog, click **Add Printer**.



- 2 Select **Add a local printer** and click **Next**.
- 3 Select a printer port as shown below and click **Next**.

← Add Printer

Choose a printer port

A printer port is a type of connection that allows your computer to exchange information with a printer.

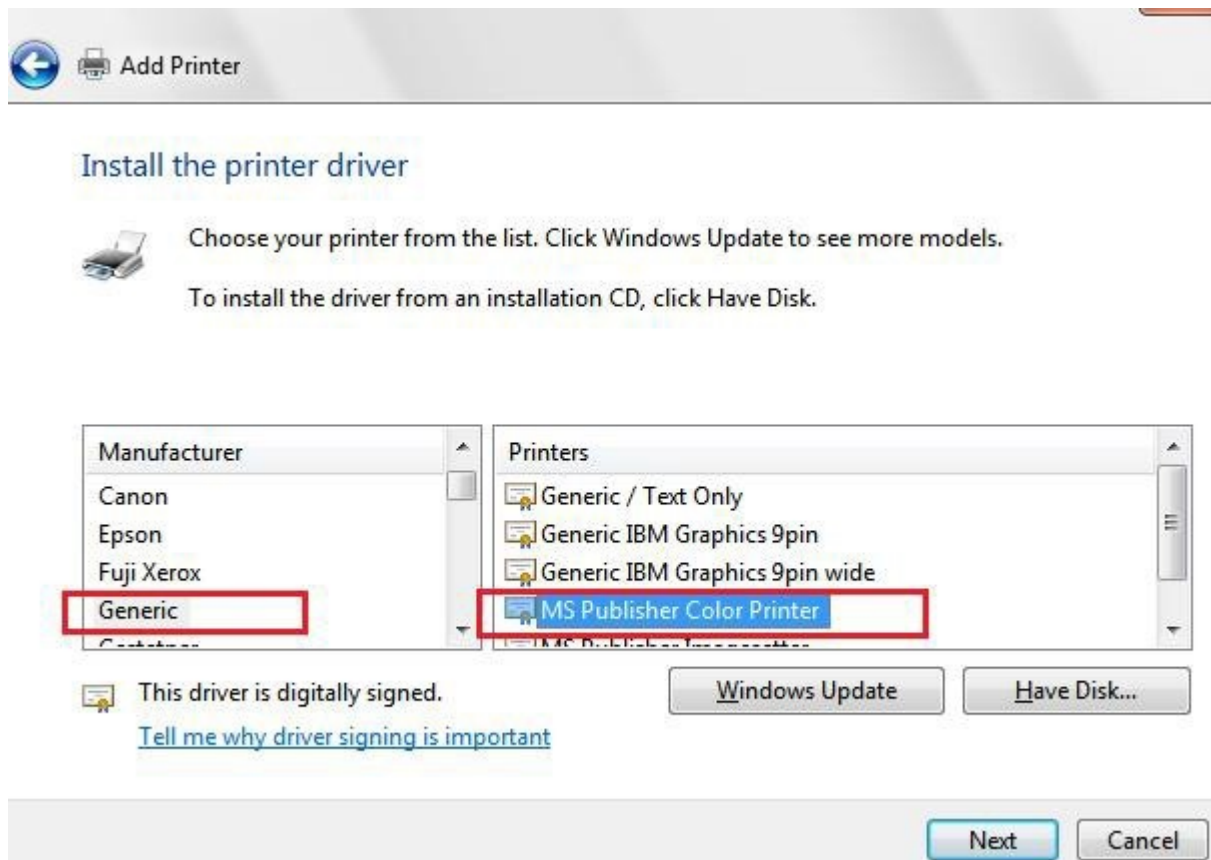
☒ Use an existing port: FILE: (Print to File) ▼

☐ Create a new port:

Type of port: Local Port ▼

Next Cancel

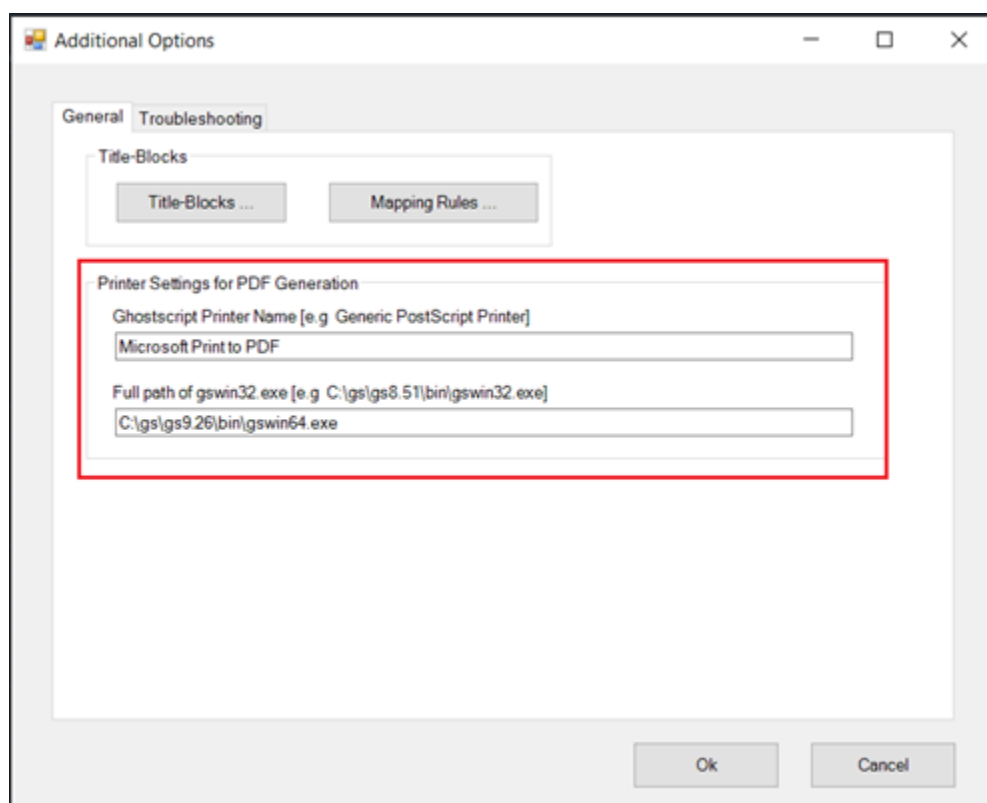
- 4 Under the **Manufacturer** select **Generic** and under **Printers** select **MS Publisher Color Printer** as shown below.



- 5 Click **Next** until you see the Finish.

Installing Ghost Script

Install the appropriate version of Ghost script suitable for the system (32 bit or 64 bit). In the additional preferences of PLM Integration of AutoCAD, the following settings should be made.



Log Files

In case of any error during the PLM operations from integration, perform the following steps:

- Set the integration preference Generate log files to true.
- Create the folder %tmp%\toolkit. If these folders are existing, delete the files in these folders.
- Reproduce the error.
- Provide the following files along with the screen shots of the error messages.
- Files created in the folder %tmp%\toolkit.