3DEXPERIENCE Connector for Creo Parametric

User Guide

3DEXPERIENCE R2022x

DASSAULT SYSTEMES | The 3DEXPERIENCE Company

Prepared by International TechneGroup Inc.

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Overview

Welcome to the *3DEXPERIENCE Connector for Creo Parametric User Guide*. This guide is intended for users who need to become quickly familiar with the 3DEXPERIENCE Connector for Creo Parametric product.

3DEXPERIENCE Connector for Creo Parametric in a Nutshell

3DEXPERIENCE Connector for Creo Parametric provides a multi-site Creo Parametric design data management solution for the extended enterprise. It allows designers to access and share each other's designs from within the native Creo Parametric user interface by leveraging the design team collaboration capabilities of X-CAD Design.

3DEXPERIENCE Connector for Creo Parametric puts access to 3DEXPERIENCE capabilities in the Creo Parametric user interface, allowing designers to effortlessly access, manage, share, and store CAD design data without leaving their preferred environment. 3DEXPERIENCE Connector for Creo Parametric facilitates process workflow, increases data integrity, and improves configuration management.

From within 3DEXPERIENCE Connector for Creo Parametric, users connect to the database using X-CAD Design engine predefined access setup to search and browse, lock and check-out designs including associated drawings to their local drives. After modification, users again utilize capabilities provided by X-CAD Design to store changes in the 3DEXPERIENCE database.

Before Reading this Guide

You may also like to read *3DEXPERIENCE Connector for Creo Parametric Install and Administration Guide* and *3DEXPERIENCE Connector for Creo Parametric Readme.*

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading and performing the step-by-step user tasks, which cover all product functionalities.

The *Getting Started* section describes common tasks that are used each time you use 3DEXPERIENCE Connector for Creo Parametric.

The *Using Templates* section describes the procedure to create new designs using predefined templates.

The *Validate Name* section describes the feature which can be used to validate the name for any invalid or unsupported characters used for newly created CAD models.

The *Saving Designs in 3DEXPERIENCE* section describes the procedure to save a design to the 3DEXPERIENCE database from Creo Parametric.

The *Using 3DEXPERIENCE Open or Insert Option* sections describe the procedure to search, lock or unlock, and open or insert designs from 3DEXPERIENCE database to Creo Parametric.

The *Open Examine* section describes the feature to examine the structure of selected iteration of a design based the configuration chosen during checkout process.

The *Open Partial* section describes the feature to process partial structures without affecting the structure integrity.

The *Lock* section describes the procedure to lock Creo Parametric design in 3DEXPERIENCE.

The *Baselines* section describes the procedure to capture and store a structure at any point in time during the design process using baselines.

The *Viewing Properties* section describes the procedure to view the properties of a design in 3DEXPERIENCE from Creo Parametric.

The *Using Workspaces* section describes the procedure to manage designs using Integration Exchange Framework Client.

The *Viewing Design Details in X-CAD Design* section describes the procedure to view CAD Portal View page of the design from Creo Parametric.

The *Global Refresh* section describes the procedure to view the properties of an active structure in 3DEXPERIENCE from Creo Parametric and refresh the models with the latest designs.

The *Local Working Directory Refresh* section describes the procedure to view the properties of designs present in a local working directory and refresh the designs with the latest designs from 3DEXPERIENCE.

The *Design Export* section describes the procedure to create a CAD designs package for export out of 3DEXPERIENCE.

The *Update PLM Parameters During Save* section describes PLM attributes getting updated in the CAD properties as a part of Save so that these attributes values can be reflected in the title block of the drawing if configured.

The *Disconnect From 3DEXPERIENCE* section describes the procedure to disconnect from 3DEXPERIENCE.

The *3DEXPERIENCE Interface Description* section describes the commands that are available in Creo Parametric and also within other 3DEXPERIENCE dialog boxes that are specific to 3DEXPERIENCE Connector for Creo Parametric.

The *Design Classification During Save* section describes the feature which allows Creo designer to classify the objects right inside the CAD session using Integration's 3DEXPERIENCE Save UI.

The *Appendix: Configure settings.ini* section provides details of various client side settings.

What's New?

This section provides information about the new and enhanced functionalities in this release.

Functional Capabilities

- Creo Parametric 6 Support(New)
- Transfer Material Property to 3DEXPERIENCE
- New Bookmark UI and Syntax
- User Credentials

Getting Started

This section introduces first time users to 3DEXPERIENCE Connector for Creo Parametric. The following sections will show you how to connect from 3DEXPERIENCE Connector for Creo Parametric to 3DEXPERIENCE.

Connecting to 3DEXPERIENCE from Creo Parametric

This task shows you how to connect from Creo Parametric to 3DEXPERIENCE.

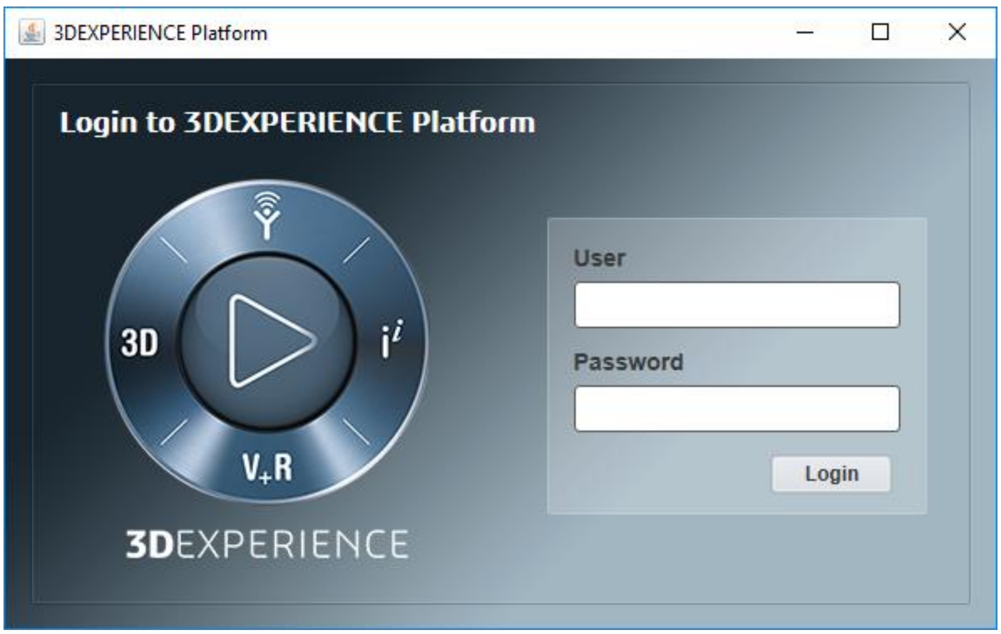
The ProENGINEER User role must be assigned to you by the Administrator. For more details, see *3DEXPERIENCE Connector for Creo Parametric Install and Administration Guide*.

1. Click **3DEXPERIENCE** > **Login** in the Creo Parametric ribbon interface.

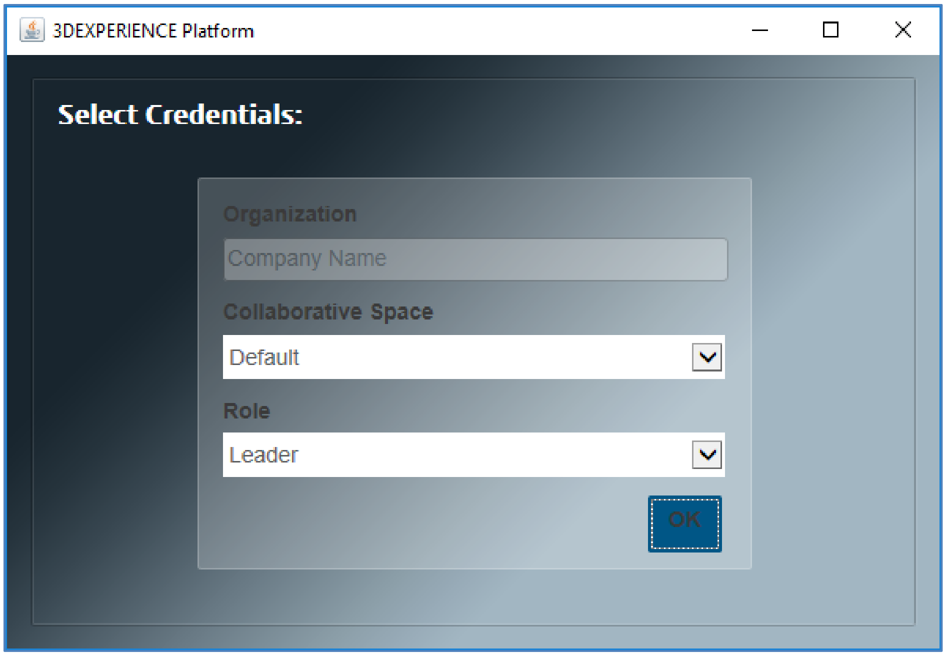
   Note: Creo Parametric connects to 3DEXPERIENCE if an active session exists.

   3DEXPERIENCE Login dialog box opens.

2. Enter your User Name and Password and click **Login.**



3. **Select Credentials**:
   When the User enters the login credentials User name and Password, the 3DEXPERIENCE Select Credentials dialog will appear for the User to choose a specific Organization, Collaborative Space, Role to continue with the login process. User will need to choose a desired 'Organization/Collaborative Space/Role and click 'OK' to complete the Login process.



Based on the 'Credentials' set during login, the User will get the respective privileges to perform various operations.

You are logged in to 3DEXPERIENCE. To check whether your log in is successful, click 3DEXPERIENCE tab from Creo Parametric ribbon interface. All 3DEXPERIENCE commands except Login are enabled.

On successful login to the 3DEXPERIENCE, the integration will change the working directory of the Creo session to the checkout directory specified in the local preferences.

Note: The system tray displays the icon when connected.

You are logged in to 3DEXPERIENCE database.

Using Templates

This section describes the methods to create new designs using pre-defined templates. The following sections will show you how to create designs of type part, assembly or drawing using templates.

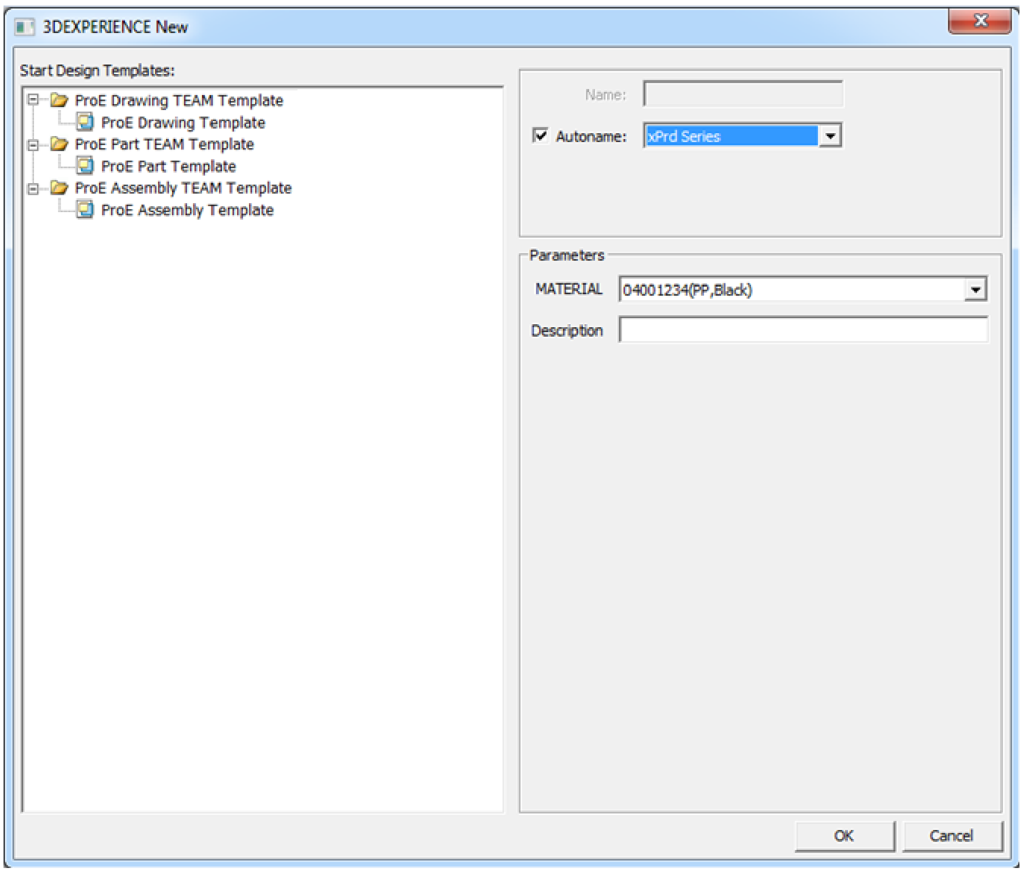
Creating Designs Using Templates

This task shows you how to create designs using predefined templates.

Required templates must exist. Templates are created by the Integration Administrator or a user who has appropriate access privileges. See *X-CAD Design Install and Administration Guide* for details on creating templates.

1. Click **3DEXPERIENCE** > **New** from Creo Parametric ribbon interface.

   The **3DEXPERIENCE New** dialog box opens.



2. Enter details for the new design including the following:

   - **Start Design Templates**. Select a template listed for the ProE design to be created. The Attribute section lists the attributes of the selected template.
   - Name of the design can be specified using one of the following methods:

i. **Name**. Enter a name for the design.

ii. **Autoname**. Select the **Autoname** check box and select the naming series from the drop-down list. Selecting the **Autoname** checkbox disables the **Name** field.

Parameters. The integration administrator specifies the parameters to be mapped for templates in GCO. The parameters can also be made mandatory and non-mandatory. Mandatory parameters are displayed in red font and non-mandatory parameters are displayed in black font in the dialog. Mandatory parameters and non-mandatory parameters are alphabetically sorted. Mandatory parameters values must be filled.

- Enter desired parameter values.

3. Click **OK** when done.

The template file is downloaded to the Checkout directory defined in the Preferences and opened in Creo Parametric as the active session.

Note: The design is not saved to 3DEXPERIENCE until you save the design using **3DEXPERIENCE** > **Save** option.

The new design is created with the selected template and is loaded in Creo Parametric.

Creating a Drawing Using Template

This task shows you how to create drawings using predefined templates. The in-session designs are automatically associated to the drawing which is created using templates.

• Drawing template must exist. Templates are created by the Integration Administrator or a user who has appropriate access privileges. See *X-CAD Design Install and Administration Guide* for details on creating templates.

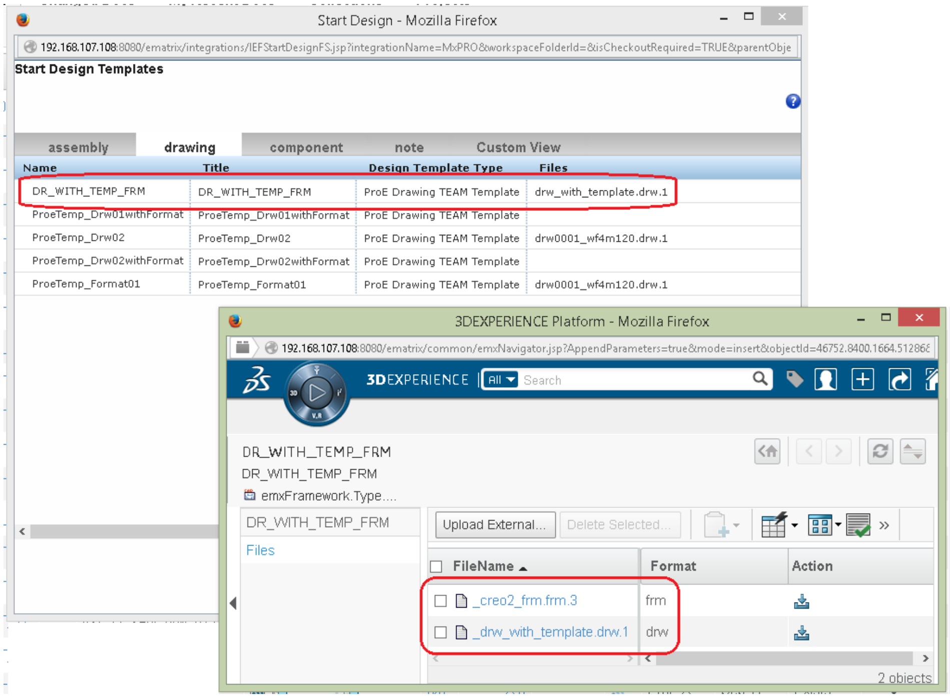
Integration allows the drawing templates to be configured in one of the below two ways,

- Option 1 : The ProE Drawing template can be configured with self-contained ProE Drawing File with the required information
- Option 2 : The ProE Drawing template can be configured with explicit ProE Drawing and ProE Format files

The native ProE Drawing file is checked in to the format ‘drw’.

The native ProE Drawing Format file needs to be checked in to the format ‘frm’.

Note: For Option 2 mentioned above, It is required that the drawing format file should be checked in and “Released” into 3DEXPERIENCE prior to uploading it into drawing template in 3DEXPERIENCE.



• Ensure that the designs that you want to associate with the drawing are open in Creo Parametric.

1. Launch Creo Parametric Application
2. Navigate to **3DEXPERIENCE** > **New** from Creo Ribbon.

   The **3DEXPERIENCE New** dialog box opens
3. Enter details for the new design including the following:
   - **Templates**. Select a template from the templates listed under ProE Drawing Template for the ProE drawing to be created. The Attribute section lists the attributes of the selected drawing template.
   - Name of the design can be specified using one of the following methods:
     i. **Name**. Enter a name for the drawing
     ii. **Autoname**. Select the Autoname check box and select the naming series from the drop-down list. Selecting the Autoname checkbox disables the Name field.
4. Click **OK** when done.

   The template file along with drawing format file is downloaded to the Checkout directory defined in the Preferences and opened in Creo Parametric as the active session. ProE Drawing Format file will be referenced by the new Drawing file. The created drawing is automatically related to the designs active in Creo Parametric.

Saving Designs in 3DEXPERIENCE

The Save function lets you save the design information into the 3DEXPERIENCE vault for sharing across the enterprise. The following sections describe some concepts of the Save function and steps to save new designs, modified designs, configurations, and also save designs without using user interface.

About Saving in 3DEXPERIENCE

The design information consists of the physical files created in Creo Parametric plus metadata or descriptive characteristics. You must save the designs often to keep the collaborative group informed of any changes. When saving Creo Parametric objects to 3DEXPERIENCE, 3DEXPERIENCE Connector for Creo Parametric provides several options that affect the behavior of the Save process. The default settings for these parameters are specified in the Global preferences which are set by your Integration Administrator. See *Setting Preferences* section of *X-CAD Design Install and Administration Guide* for details on setting global preferences. See *Saving Designs to 3DEXPERIENCE* for information of the settings that the users may modify while saving designs to 3DEXPERIENCE.

Note: Ensure that the family table instances are verified before 3DEXPERIENCE Save operation.

The following topics are discussed:

Attribute Synchronization

Attribute synchronization ensures that the attributes in the design file on your computer match the attributes of the associated 3DEXPERIENCE object.

During the design process, you can enter or change attributes or properties of the design. When you save the design, you want those attributes and properties to become part of the metadata for the 3DEXPERIENCE object.

Your Integration Administrator specifies which attributes and properties can be synchronized. Attributes will be synchronized from the CAD data to 3DEXPERIENCE during save.

The integration will now add the default values of mapped attributes in the Mx-To-CAD GCO option to the PROE Models on the checkin. In the past, the integration added those attributes to the models on the first checkout, and hence, the models were modified in the Save dialog of the second checkin.

Note: See *X-CAD Design Install and Administration Guide* for details on attributes that are mapped and mapping attributes.

Note: You can also synchronize attributes when you open a design from 3DEXPERIENCE.

Lock Status While Saving Designs

Locking designs in 3DEXPERIENCE ensures that no other user can modify the designs you are working on. Lock status of a design indicates whether the design that exists in 3DEXPERIENCE is locked by you or any other user. The designs in **3DEXPERIENCE Save** dialog box can be selected for save only if the designs are locked by you.

> **Note:** Lock status is valid only for designs that exist in 3DEXPERIENCE vault. Lock status is not displayed when you save the designs the first time.
>
> 3DEXPERIENCE Collaboration for Creo Parametric provides a Retain Lock option while saving designs. By selecting the Retain Lock option during a save the lock is maintained by the user so that subsequent edits and saves may be performed.
>
> Applying a Lock indicates the intent to modify the design.

New Revision Stream

Revision is a controlled release of a design that replaces previous releases. Each release is usually identified by an associated letter code, termed the "revision level" or simply "revision". The purpose of revisioning is to maintain the history of a design in order to be able to return to a previous release at any point in time.

Using 3DEXPERIENCE Connector for Creo Parametric you can create new revision stream while saving designs using right-click on selected designs and selecting “Revise” in the **3DEXPERIENCE Save** dialog box. The new revision stream that is created is next in sequence to the current stream.

Obsolete Designs

Any design is considered Obsolete if a newer design exists in the 3DEXPERIENCE database.

Your design is considered obsolete because it is based on an earlier design than the latest design in 3DEXPERIENCE. To resolve obsolete designs and continue working with the previous iteration, you can update your designs in your local workspace using X-CAD Design Client. See Working with X-CAD Design Client section of *X-CAD Design User Guide* for details on updating designs from local workspace folders.

For example, a design has been stored with multiple iterations: 1 & 2 in 3DEXPERIENCE database. User A opens iteration 1 wanting to ignore the modifications made in iteration 2. When user A tries to save iteration 1, then the design is displayed as obsolete in **3DEXPERIENCE Save** dialog box.

Saving New and Modified Designs to 3DEXPERIENCE

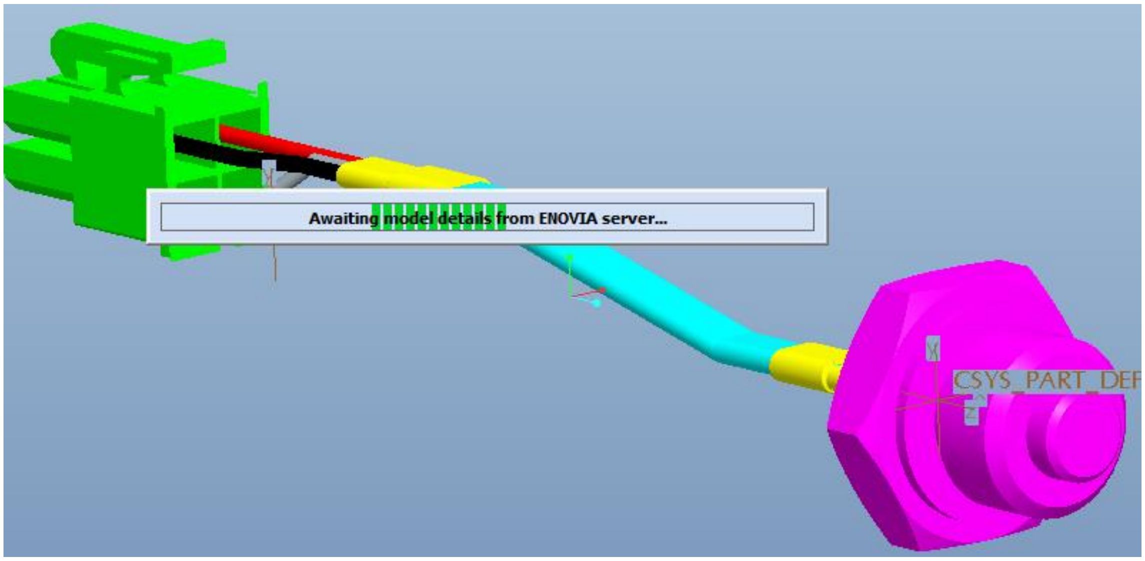
This task shows you how to save new and modified designs to 3DEXPERIENCE database which are opened in Creo Parametric.

Ensure that the designs you want to save to 3DEXPERIENCE are open in Creo Parametric.

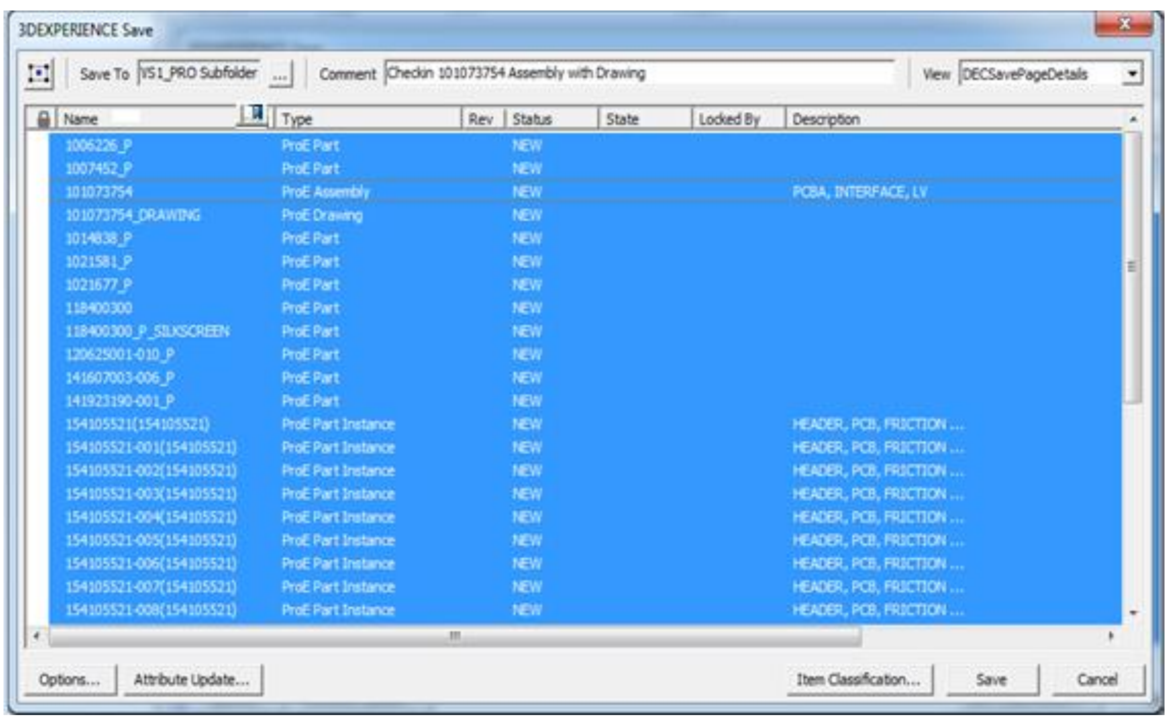
1. Click **3DEXPERIENCE** > **Save > Active/All** from Creo Parametric ribbon. Use the 3DEXPERIENCE>Save>Active menu to save (checkin) the models present in the active window to 3DEXPERIENCE. The 3DEXPERIENCE>Save>All menu will save all models present in session, including models from the hidden windows, to the 3DEXPERIENCE.

Note: A message is displayed and **3DEXPERIENCE Save** dialog box does not open if there are no changes to be saved for the designs open in Creo Parametric.

The user will notice a progress bar that contains details of the background process. It provides valuable feedback to user during the checkin process.



The **3DEXPERIENCE Save** dialog box opens.



All the designs open in Creo Parametric that are new or modified are listed in the **3DEXPERIENCE Save** dialog box. For each design, the **3DEXPERIENCE Save** dialog box lists,

• . Displays the lock status of the design in 3DEXPERIENCE. icon is displayed if the design is locked by you or any other user. The icon is displayed if the design is not locked in 3DEXPERIENCE. For a new design, no icon is displayed.

• **Name**. Name of the design.

• The icon is displayed for designs that are saved to 3DEXPERIENCE Bookmark Workspace. Roll-over mouse to view the name of the workspace where the design is saved. For a new design, this is blank.

• **Type**. The type of the design. ProE types are: ProE Part, ProE Part Family Table, ProE Part Instance, ProE Assembly, ProE Assembly Family Table, ProE Assembly Instance, ProE Drawing, ProE Format, ProE Manufacture, ProE Layout, ProE Diagram, and IDF Model

• **Rev**. Current revision of the sequence. This is blank for a new design.

• **Status**. Status of the design in 3DEXPERIENCE. One of the following is displayed for each design:

- **New** is displayed as the status of the design in 3DEXPERIENCE. This is displayed if the design does not exist in 3DEXPERIENCE.
- **Modified**. Displayed if the design is modified in Creo Parametric since the design was last saved or opened.
- **Obsolete**. Displayed if newer iteration of the design exists in 3DEXPERIENCE database.
- **Duplicate**: A part with same name and type exists in 3DEXPERIENCE

• **State**. Current state of the design in 3DEXPERIENCE lifecycle. This is blank for a new design. The OOTB policy defines the following states: Under Global Design, Design Frozen, Design Released

• **Locked By**. If the Creo Parametric design being saved already exists in 3DEXPERIENCE, and is locked by any user, then this column indicates the name of this user. In the case of saving a new design, this is set to default value '--'. You can only select locked designs.

• **Description**. Description entered for the design by the user.

• If a column name of the Save dialog matches the parameter name of a PROE model present in the Save dialog, the integration will show the CAD value (Tools->Parameters of the PROE Model) in the Save dialog.

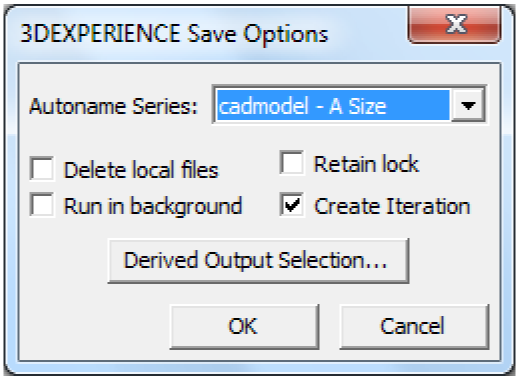
2. You can enter the following details for saving the design.

   - **Save To**. The 3DEXPERIENCE workspace where the designs will be saved. For selecting a workspace, see *Selecting a Workspace to Save Designs*

- **Comment**. Enter comment for the design you are saving. The comment is stored as an attribute in the design in 3DEXPERIENCE.

- **View**. Click and select a table defined by the Integration Administrator from the drop down list. The selected table is applied to the **3DEXPERIENCE Save** dialog box.

3. Click **Options** to open the **3DEXPERIENCE Save options** dialog box:



Note: To support your business processes, your Integration Administrator may have defined some options so those cannot be changed. If the options are modified during a save operation, these parameters will return to their default setting during the next save operation. To permanently change these settings, it must be set using the Design Central web user interface. See *X-CAD Design Install and Administration Guide* for details on setting these preferences using X-CAD Design. Select the options to be executed when you save selected designs to 3DEXPERIENCE

- **Autoname Series.** Select a naming series from the drop down list. Only new designs can be named using Autoname. See *Using Autoname*.

- **Delete Local Files**. Select to delete the files of the design on your local disk upon saving.

Note: Delete Local Files option cannot be selected if **Background** option is selected.

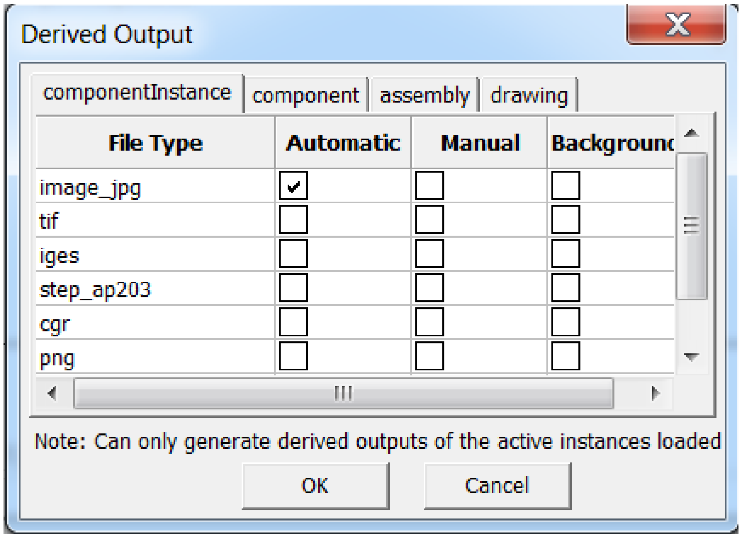
- **Run in background.** Select to save designs without affecting Creo Parametric operations. Copies of the files to be saved to 3DEXPERIENCE are made and control is returned back to Creo Parametric, so you can continue to use Creo Parametric even when the save process is in progress.

- **Retain Lock.** Select to lock the design. It is recommended that you lock your designs while saving to prevent other users from modifying your designs. If not selected, the design is saved to 3DEXPERIENCE in the unlocked state.

- **Create Iteration**. Select to create a new iteration of the design in 3DEXPERIENCE. If not selected, the existing iteration of the design in 3DEXPERIENCE is overwritten by the design being saved.

Click **OK**.

- **Derived Output Selection** button. Select to display Derived Output dialog box.



The allowable derived output file types are listed for each available CAD type. The CAD types can be selected by picking the tabs near the top of the dialog:

- assemblyInstance
- componentInstance
- component
- drawing
- assembly

Select or de-select check boxes to generate the desired derived output files during the Save operation.

By default, the integration will set the image_jpg derived output option for all 3DEXPERIENCE types.

The default settings that must be applied and derived outputs that must be generated while saving designs to 3DEXPERIENCE are controlled by your preferences settings.

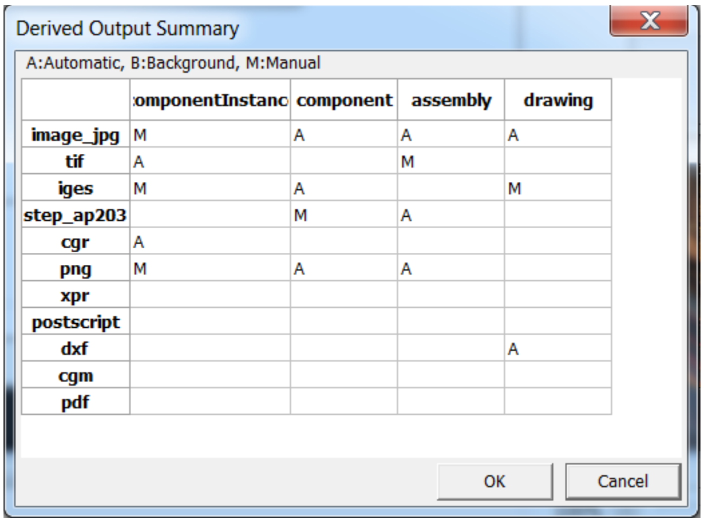
The columns indicate how the derived outputs will be generated.

- **Automatic** indicates that the 3DEXPERIENCE Connector for Creo Parametric will generate the specified derived output file during the Save operation
- **Manual** indicates that the 3DEXPERIENCE Connector for Creo Parametric will look for an existing file in the same directory with the same basename as the file being saved, but with the appropriate extension (e.g., ‘.jpg', tif', ps', dxf', igs', stp', cgr', png', cgm', ‘pdf’)

- **Background** indicates that a request will be sent to generate the specified derived output file(s) via a background process. The background derived output option is NOT support in the OOTB setup; it requires custom programming on the server side.

Click **OK**.

On clicking OK, the user will get the below summary dialog.



	:omponentInstanc	component	assembly	drawing
image_jpg	M	A	A	A
tif	A		M	
iges	M	A		M
step_ap203		M	A	
cgr	A			
png	M	A	A	
xpr				
postscript				
dxf				A
cgm				
pdf				

4. Optionally, right-click the design and select:

- **Revise**. Selecting Revise will set the target revision of the selected designs as Next and will display the setting under the Target Revision column. For new designs, the first revision in the revision sequence is applied as the revision by default. For example, Rev A.

- **Revise Related Instances**: This option is available when there are some family members in the Save dialog. It allows to Revise all the related family members together. This helps in achieving the together mode revision of instances.

- **Current Revision**. Grayed out unless Revise has been selected previously. When selected, the designs Target Revision column will be set back to blank and any ensuing Save will be to the current revision.

- **Properties**... Click to see properties for a selected design. See *Viewing Properties*

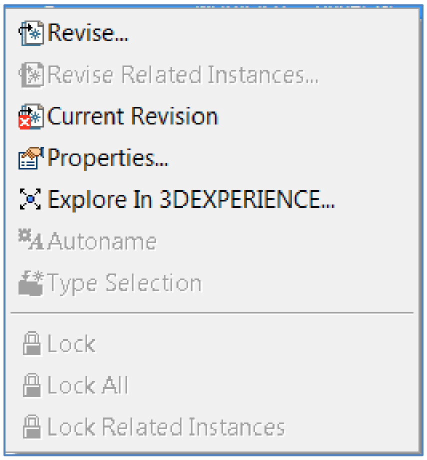
- **Explore In 3DEXPERIENCE**… Opens CAD Portal view of the design in the browser. See *Viewing Design Details in X-CAD Design*

- **Autoname**. Click to assign a name automatically to the selected designs. See *Using Autoname*

- **Lock**. If the design is unlocked, then it will be grayed out in Save dialog. You can select Lock on any unlocked design to acquire lock. After lock is successful, you will be able to select and checkin the design.

- **Lock All.** Lock All can be used to lock all of the unlocked designs in Save dialog at once.

- **Lock Related Instances:** This option is available when there are some family members in the Save dialog. It allows to lock all the related family members in one go.



5. Click **OK** when done.

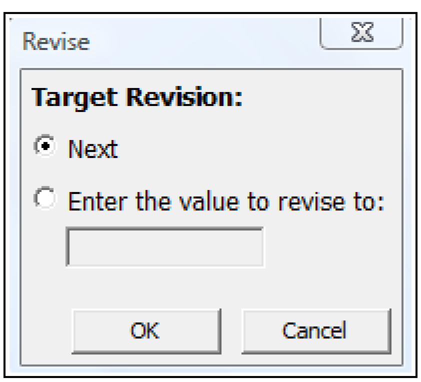
The new and modified designs are saved in 3DEXPERIENCE with the options selected for save applied.

Note: An integration user will get a warning dialog if he or she didn't select all models, including grayed out rows, of the Save dialog.

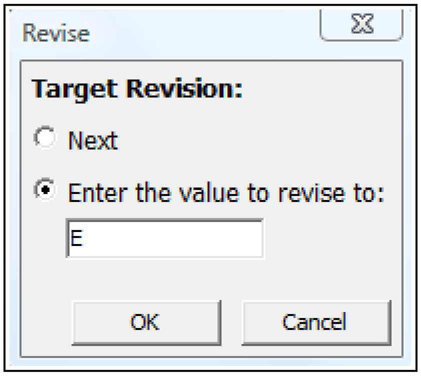
Revise Design

User can specify Revision string to Creo Parametric design in 3DEXPERIENCE Save dialog.

1. Right-click the design, and click Revise to revise design to next revision. Select 'Next' to revise Creo Parametric design to next revision string and click OK. The Save dialog will display the next revision in target revision column. For new designs, the first revision in the revision sequence is applied as the revision by default. For example, Rev A.



2. Right-click the design, and click Revise to revise design to specify revision. Select “Enter the value to revise to” radio button, and specify revision. The selected design will be checked in to specified revision.

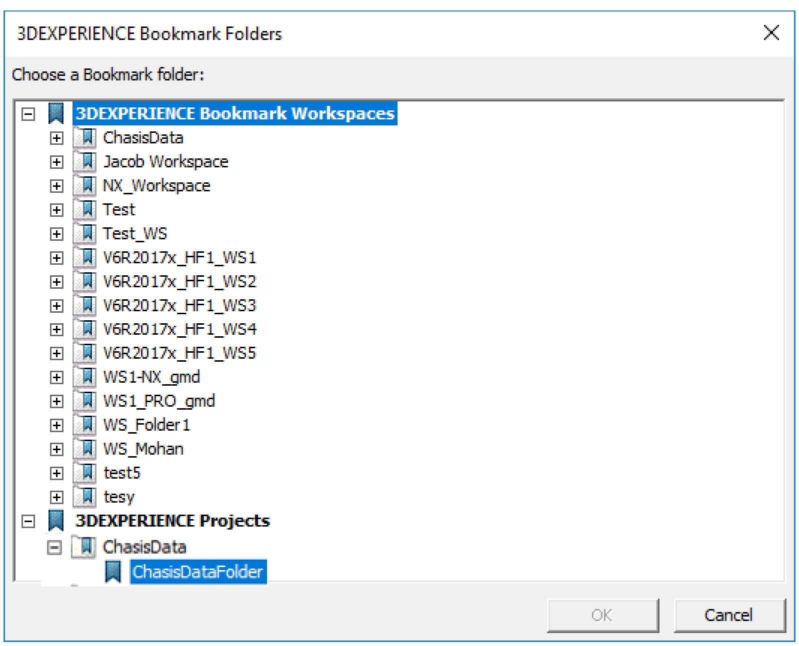


3. “Current Revise” is grayed unless Revise has been selected previously. Right-click the design and click Current Revise. When selected, the design Target Revision column will be set back to blank and the selected design will be checked in to the current revision.

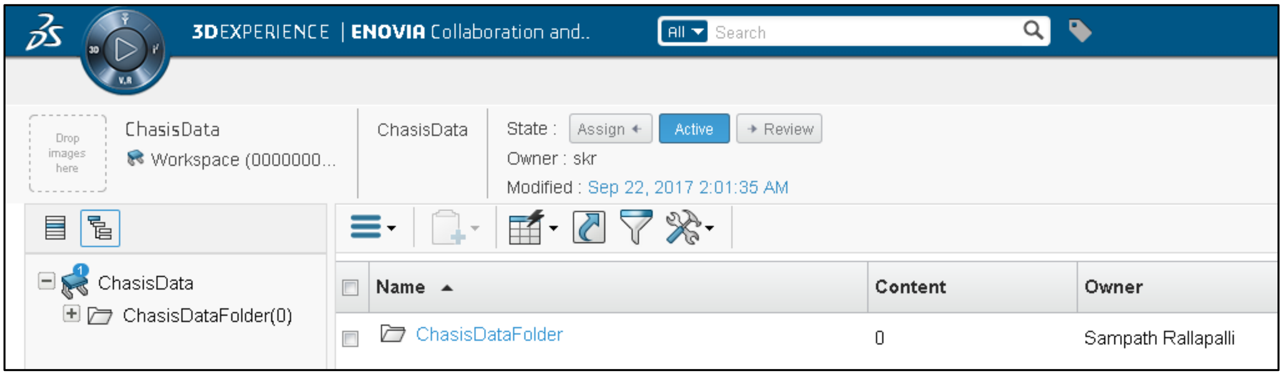
Selecting a Bookmark Workspace to Save Designs

- Click **[...]** in the **Save To** area in **3DEXPERIENCE Save** dialog box.

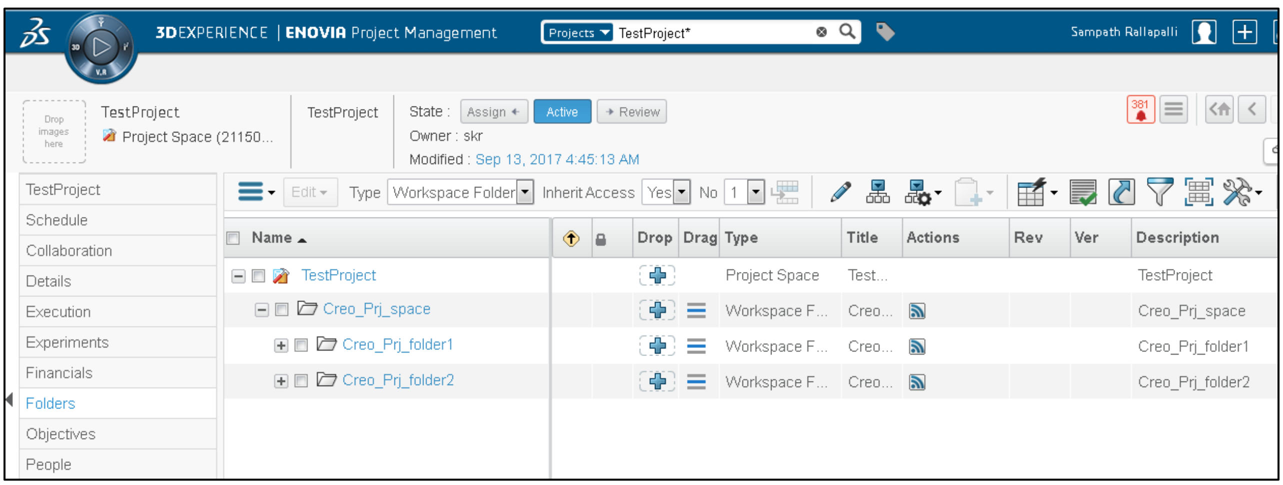
  The **3DEXPERIENCE Bookmark Folders** dialog box opens. It displays Workspaces and Projectspaces.



The Workspace from ‘3DEXPERIENCE Folders’ dialog can be seen in 3DEXPERIENCE Web Interface below



The Project Space from ‘3DEXPERIENCE Folders’ dialog can be seen in 3DEXPERIENCE Web Interface below



- Select a Bookmark folder from the list of 3DEXPERIENCE Bookmark folders and click OK.
- If the 3DEXPERIENCE Admin sets the GCO option “MCADInteg-ForceWorkspaceOnSave” to TRUE, then the integration users need to select a 3DEXPERIENCE Bookmark folder on checkin.
- The integration allows user to checkin designs of an 3DEXPERIENCE bookmark folder to another workspace folder.
- The selected bookmark workspace is displayed in the Save To: field in the 3DEXPERIENCE Save dialog box.
- Bookmark Workspaces and Bookmark folders can also be created in the above dialog, please refer *create Bookmark Workspace/ Bookmark folders during Save operation* section for more details.

Block save operation if the dependent objects are not selected in save dialog

Some of the customer prefer to block the save if dependent objects are not selected in save dialog. Creo Parametric Integration will provide an enhancement that will be make available only to specific customer's business requirement. The OOTB behavior of the Creo Parametric Integration will continue save if dependent objects not selected in Save Dialog by showing the warning message, which shows the list of objects not selected in save dialog.

This enhancement will come into picture when user will click on 'Save' button of save dialog. It will block the 'Check-In' operation if user is trying to Check-in with some dependent objects deselected in save dialog.

It will list all the objects, which are deselected, so that user can take corrective action in save dialog.

Block Save Enhancement will work with Modified designs as well.

Note:

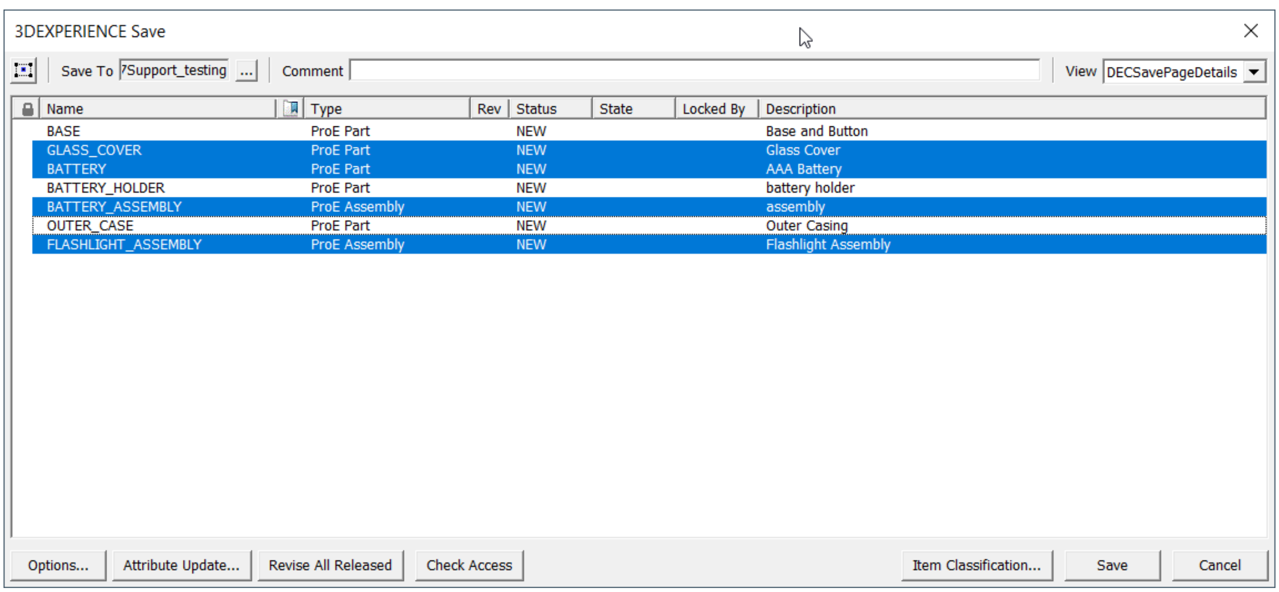
This enhancement will only work with '3DEXPERIENCE>Save>Active' menu

This enhancement will get activated once following setting is added to "settings.ini" file located in bin directory of CreoParementric connector installation.

CreoForceSelectableUnloadedChildRelationships = 1 ;

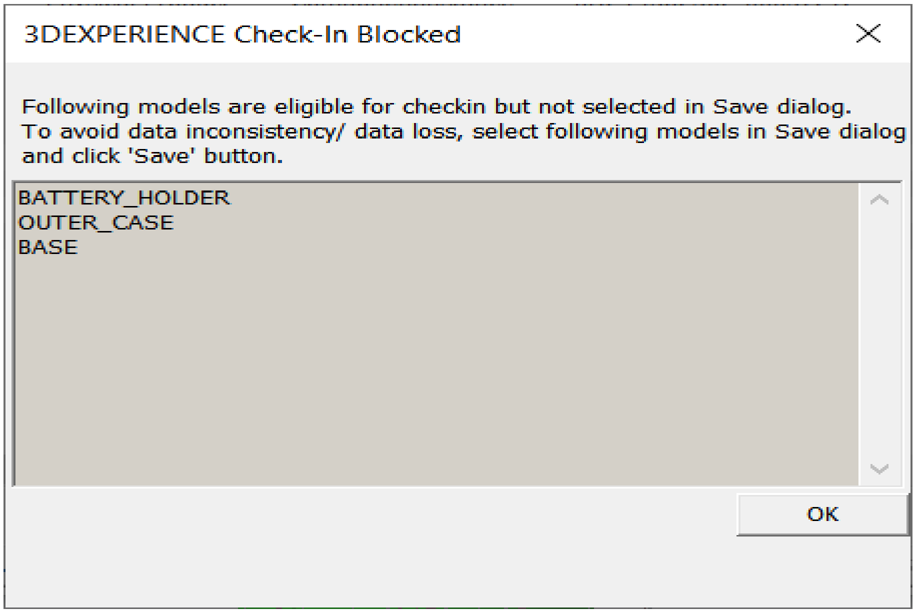
This feature will not be available if above setting is not present or set '0' value

In Save Dialog, do not select some child components under assembly and start Save. In below Example child components BATTERY_HOLDER, OUTER_CASE and BASE not selected.



After Clicking on save button "3DEXPERIENCE Check-In Blocked" dialog will be displayed with

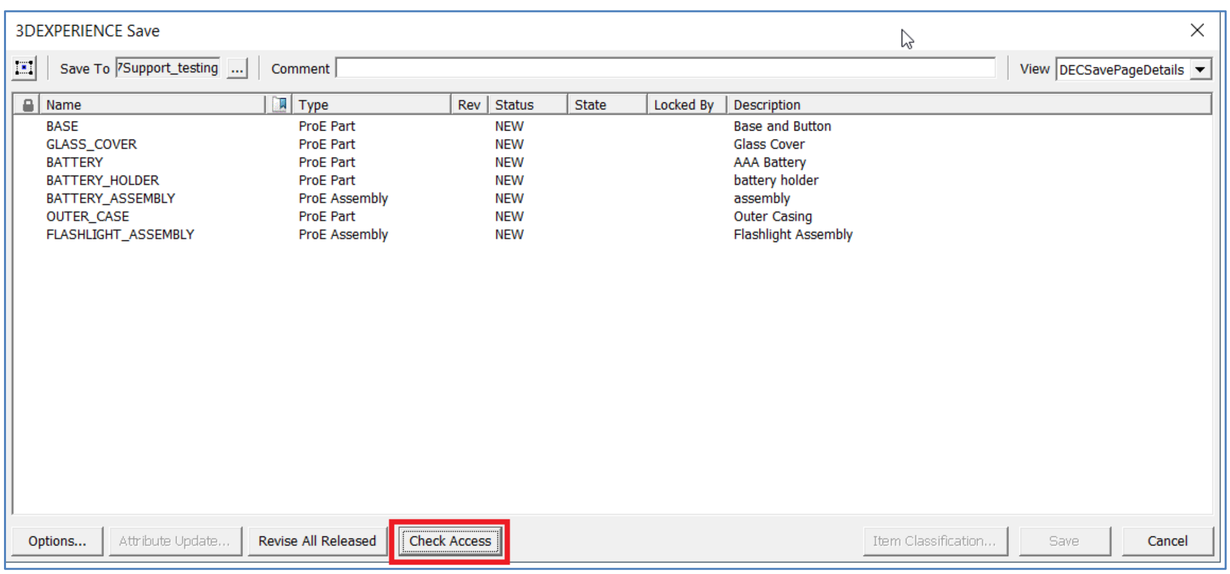
List of all the dependent objects to take corrective action.



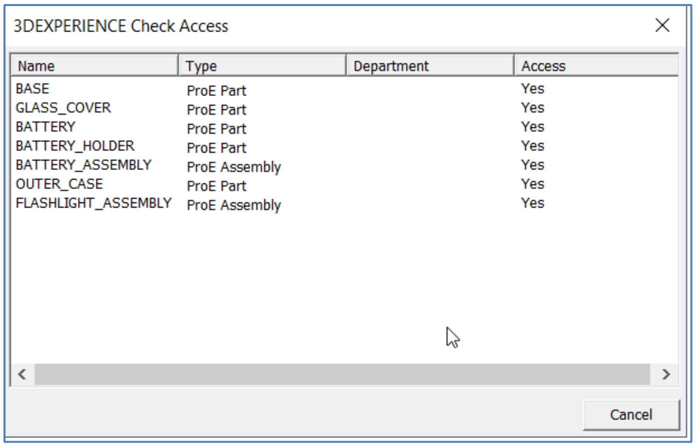
If there are parts, which are greyed out in the save dialog, means these parts are already present in the server. So save is blocked. If user wants NOT to block the save operation even if such parts are not selected while doing save operation then following entry must be added in ‘GENERAL’ section of ‘settings.ini’ file located in ConnectorforCreoClient installation directory. Default value of this variable is 0.

doNotBlockModifiedUnlockedDesigns = 1

- Check Access



When user clicks on Check Access button, he can see whether he has access to the designs From Collaborative Space assigned to the same user. Department Column shows collaborative Space assigned to the design listed in the dialog.

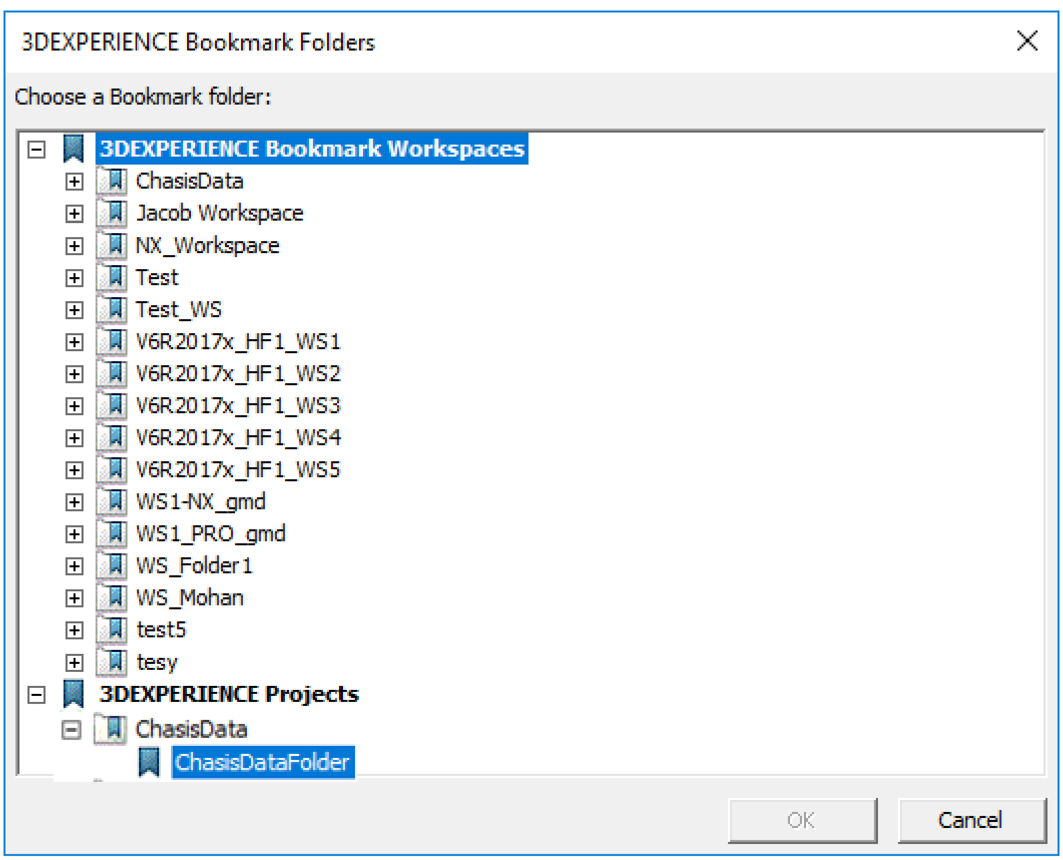


Selecting a Projectspace folder to Save Designs

Note: Projectspace folders can be accessed only if the User is a member of Program Central Projects and appropriate privileges to access folders.

1. Click **[...]** in the **Save To** area in **3DEXPERIENCE Save** dialog box.

   The **3DEXPERIENCE Folders** dialog box opens.



- Select a Projectspace from the list of 3DEXPERIENCE Projects and click **OK**.
- If the 3DEXPERIENCE Admin sets the GCO option “MCADInteg-ForceWorkspaceOnSave” to TRUE, then the integration users need to select a 3DEXPERIENCE Bookmark folder or a 3DEXPERIENCE Projectspace folder on checkin.
- The integration allows user to checkin designs of an 3DEXPERIENCE Projectspace folder to another Projectspace folder.

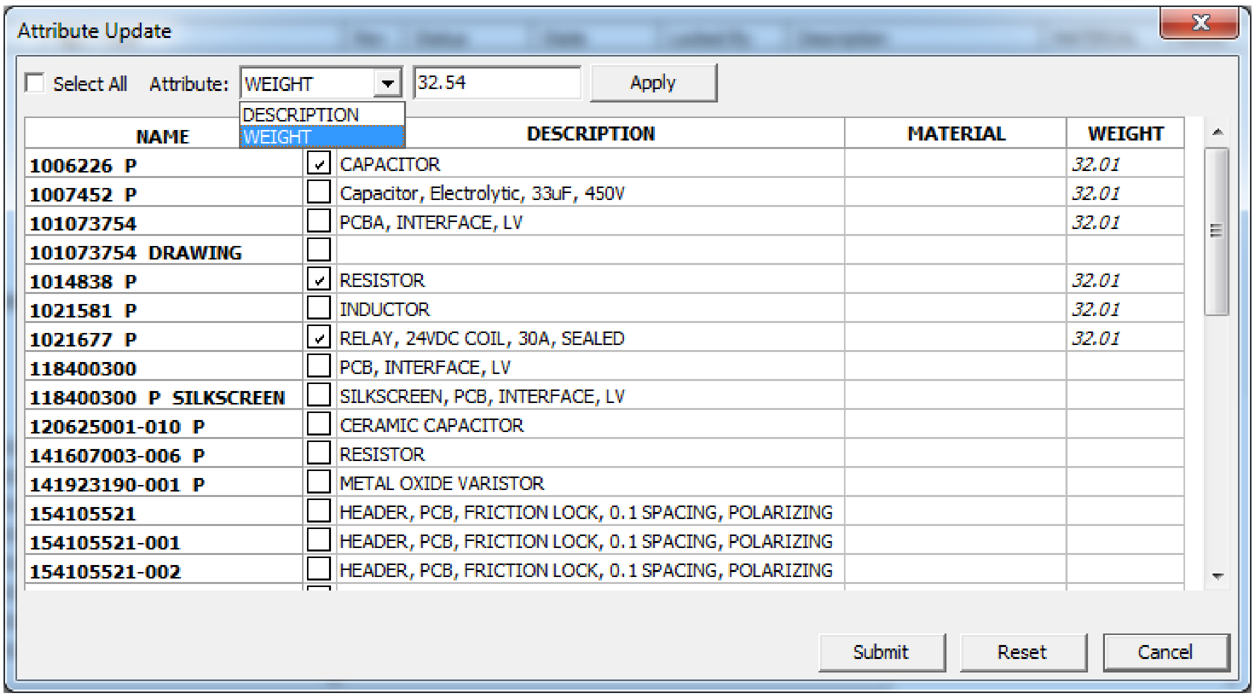
  The selected Projectspace folder is displayed in the **Save To:** field in the **3DEXPERIENCE Save** dialog box.

Using Attribute Update

The Attribute Update option allows the user to view and update specified CAD parameters from a dedicated GCO setting for all selected models during save process.

In **3DEXPERIENCE Save** dialog box, select models and click **Attribute Update** button. The **Attribute Update** dialog will appear showing all selected CAD object names in the left most column.

The list of CAD parameter to 3DEXPERIENCE mapping and the sequence of parameters names in Attribute Update Dialog are specified in the MCADInteg-UpdateAttribMapping GCO option.



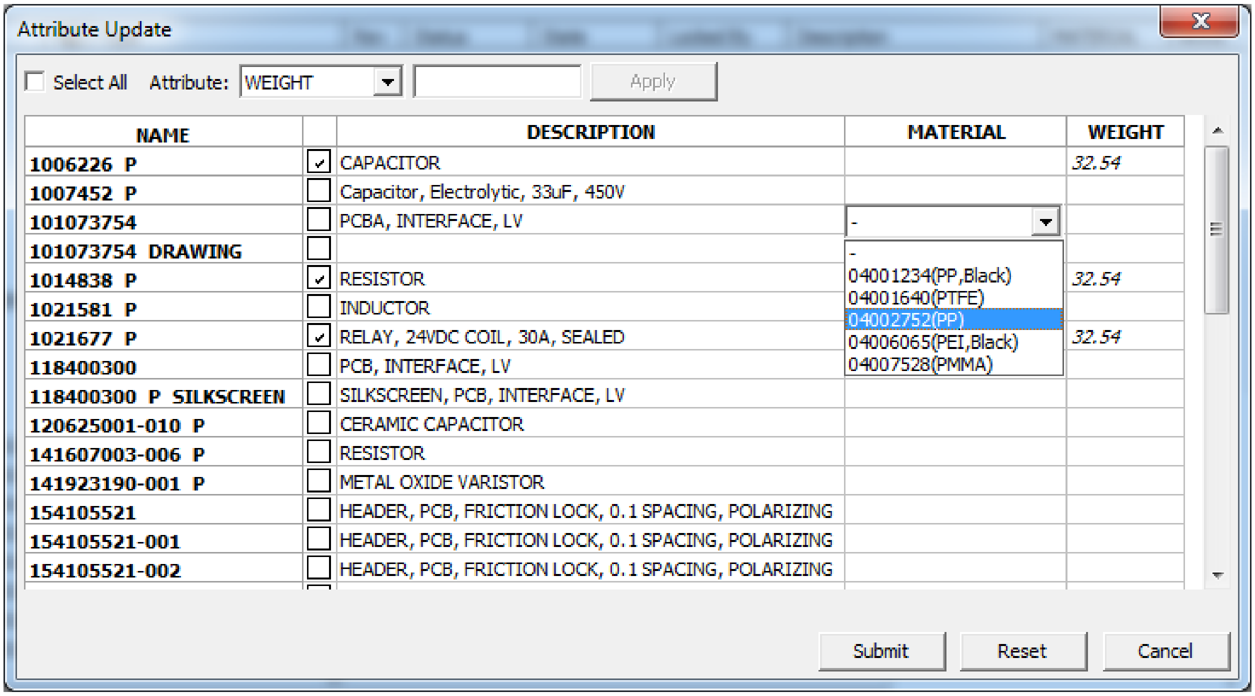
Clicking **Select All** will enable the checkbox in the row of every object. This is useful for adding the same value to all objects via the Apply button.

Clicking **Attribute** Drop-down will allow you to select an attribute if there are multiple attributes to be updated to one or more selected CAD objects

Clicking **Apply** will apply the value set in the textbox to the attribute specified in the dropdown to all models that have a check in the checkbox.

Note: Value of a parameter for an object can also be changed by editing value in the cell with row of the object and column of parameter.

Range Value Support for Creo Parametric Attributes:



For a Creo design, if the range value is specified in MCADInteg-ENOVIANewAttributeMapping GCO setting for a parameter then a pull-down option with specified range values is shown on selecting the field of this parameter column.

Note: CAD attributes with the range values specified in MCADInteg-ENOVIANewAttributeMapping GCO setting, will not be available in ‘Attribute:’ drop-down option.

Clicking **Reset** will revert all changes made since the dialog was opened.

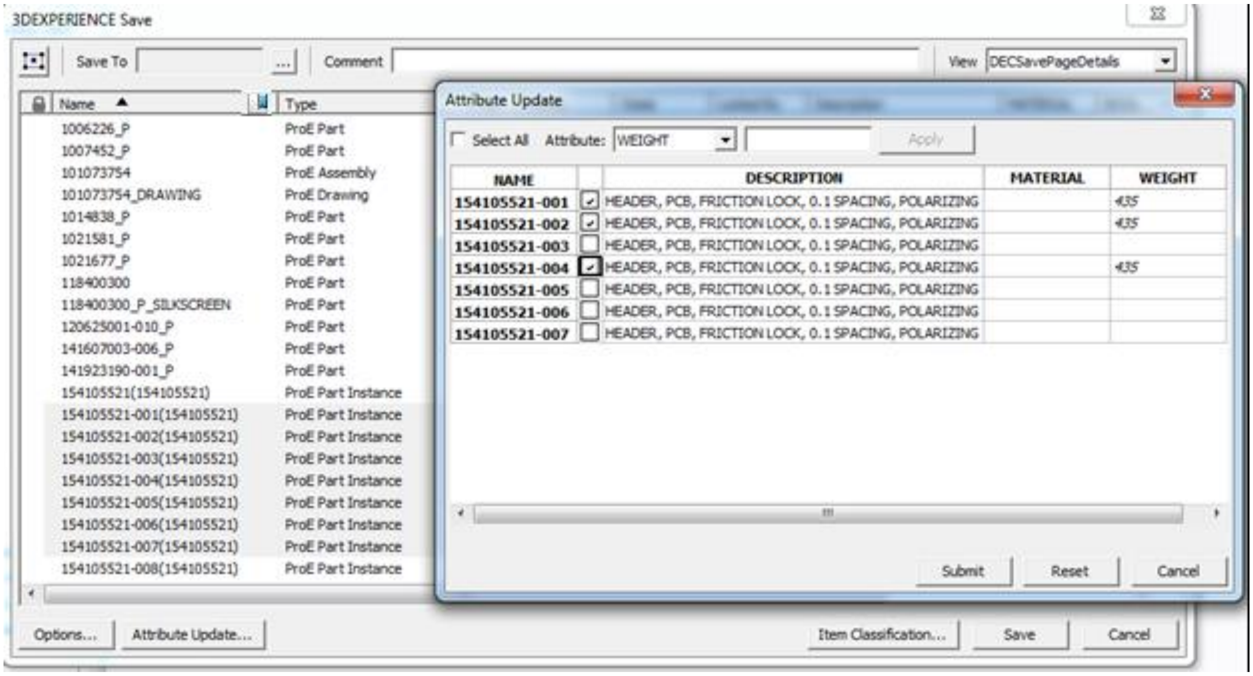
Clicking **Submit** will show a warning message to confirm the changes. On confirmation the integration will apply all the attribute changes made in the dialog to the CAD Models and then return the user to the **3DEXPERIENCE Save** dialog.

Note: if the attribute are modified for the family table instances then a warning is shown for instances being in unverified is shown. See *Attribute Update for Family Table instances*

Clicking **Cancel** will return the user to the **3DEXPERIENCE Save** dialog without applying any changes to the CAD Models.

Attribute Update for Family Table instances

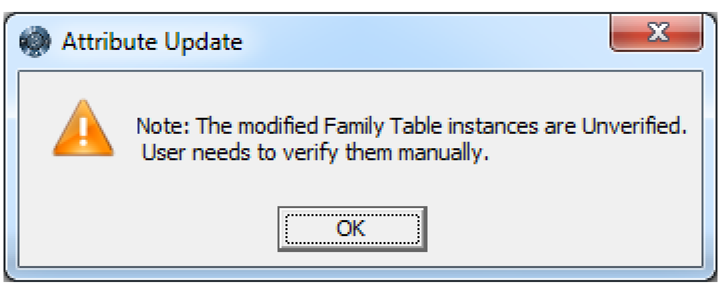
Attribute Update dialog is enhanced to support Creo Parametric attribute update for the selected instances.



If the Family table instances are selected for 'Attribute Update', then the mapped CAD parameters, shown as columns in 'Attribute Update' dialog, will be added as columns or updated as corresponding column's value in instance's Family-Table.

1. Create new CAD parameter on the instance or family table:
   If CAD parameter is not already present on Family-Table, then new column is added for CAD parameter on the Family-Table. This parameter is then assigned the value set in 'Attribute Update' Dialog.
2. Update existing parameter on the instance:
   If Creo parameter column is already present on Family Table, then the corresponding CAD parameter column values are updated with modified values set in 'Attribute Update' Dialog

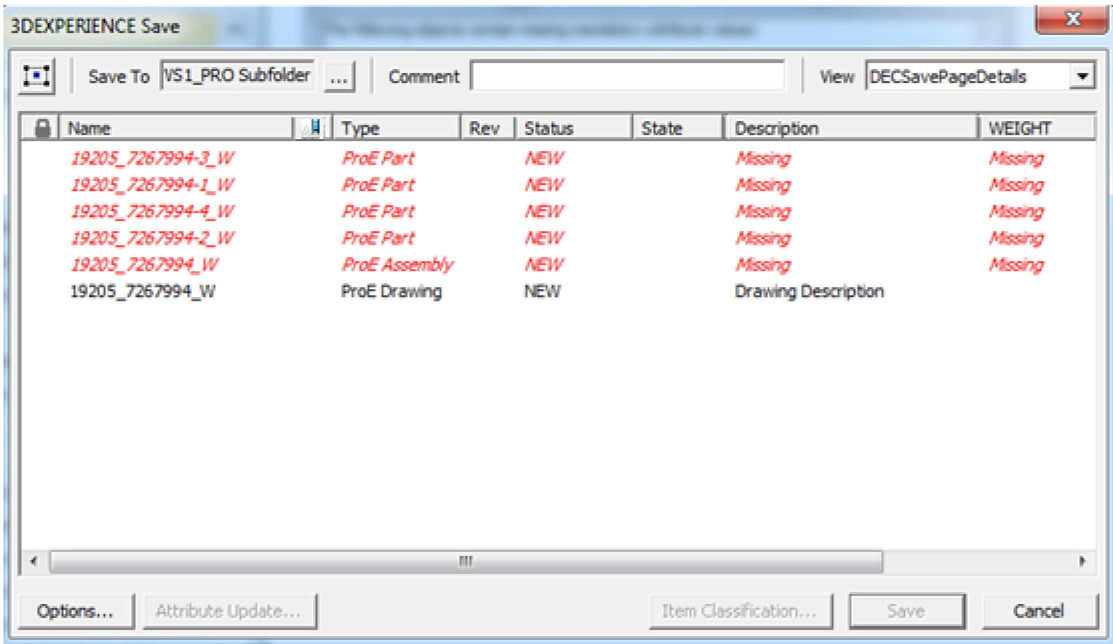
Note: The modified Family Table are shown as unverified in CAD Tools-> Family Table. Following warning message will appear on confirming the attribute value changes for instances in Attribute Update dialog



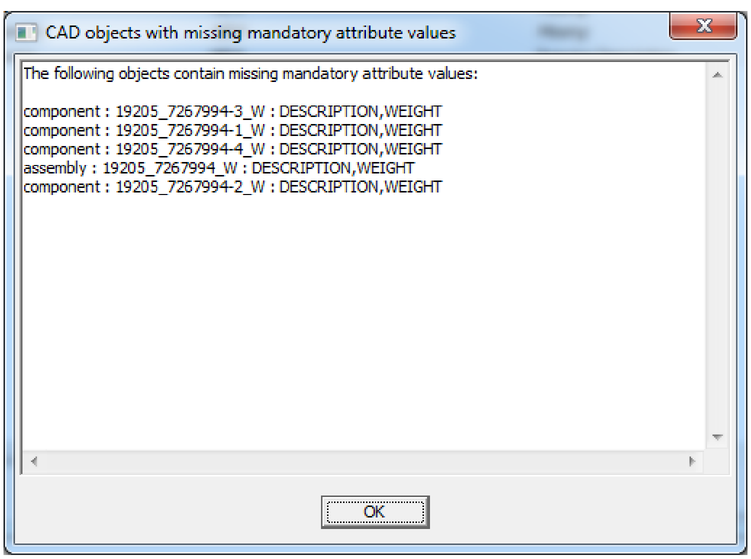
Mandatory Attribute

The mandatory attribute feature ensures that mandatory attributes for the CAD object are assigned with appropriate values before proceeding with checkin operation. The list of 3DEXPERIENCE attributes mapped to corresponding Creo Parametric object types are specified in the GCO attribute IEF=CADToMxMandatoryAttribMapping. For details refer to *3DEXPERIENCE Connector for Creo Parametric Install and Administration Guide's 'The Global Configuration Object'* section.

As per GCO setting, whenever user tries to Save design, the feature will check for the mandatory attributes and if no value is found corresponding to these mandatory attributes then those CAD models are shown in Red italic font and the attribute column will contain "missing" to signify that these mandatory attribute has to be filled with valid values in order to proceed with Save operations.



If user still clicks on the "Save" command, then an error message is displayed to the user as shown:

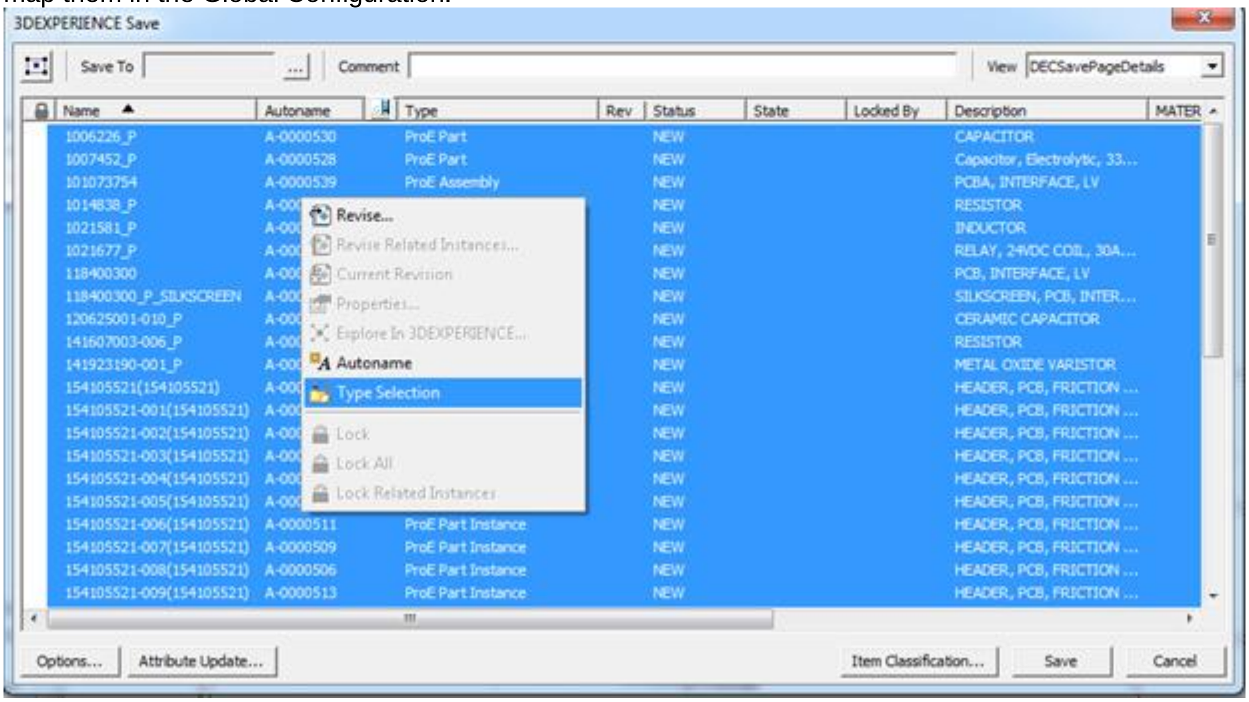


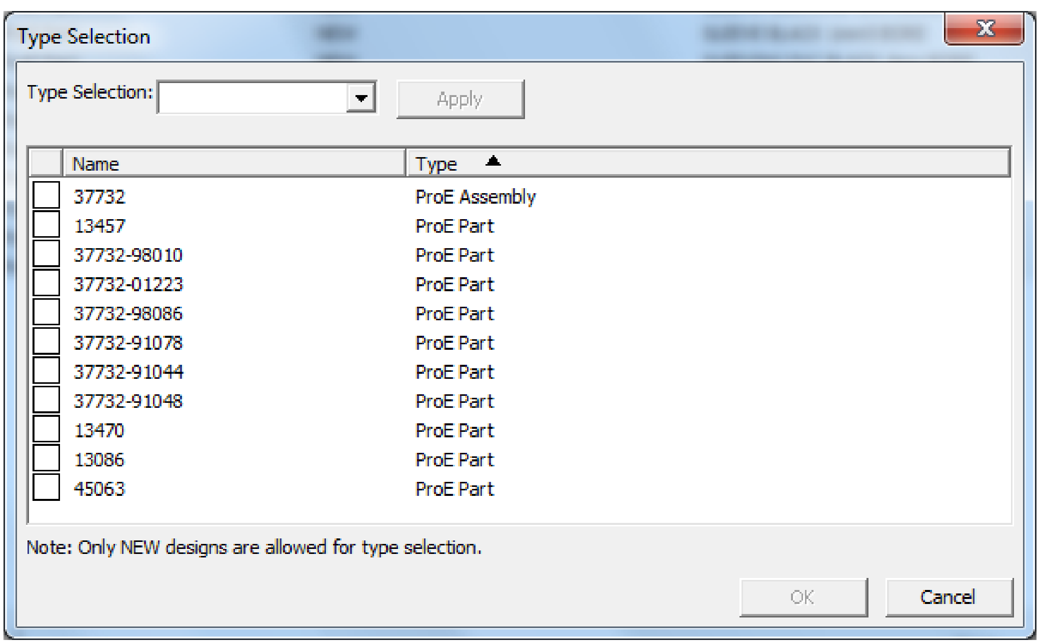
Change 3DEXPERIENCE Type on Checkin

Using the “Type Selection” right-mouse button in the Save dialog, the integration user can change the default 3DEXPERIENCE type of the new designs. This feature supports components, assemblies and drawings.

Note: This feature doesn’t support family tables.

Before using this feature, the 3DEXPERIENCE Admin needs to create new 3DEXPERIENCE types and map them in the Global Configuration.

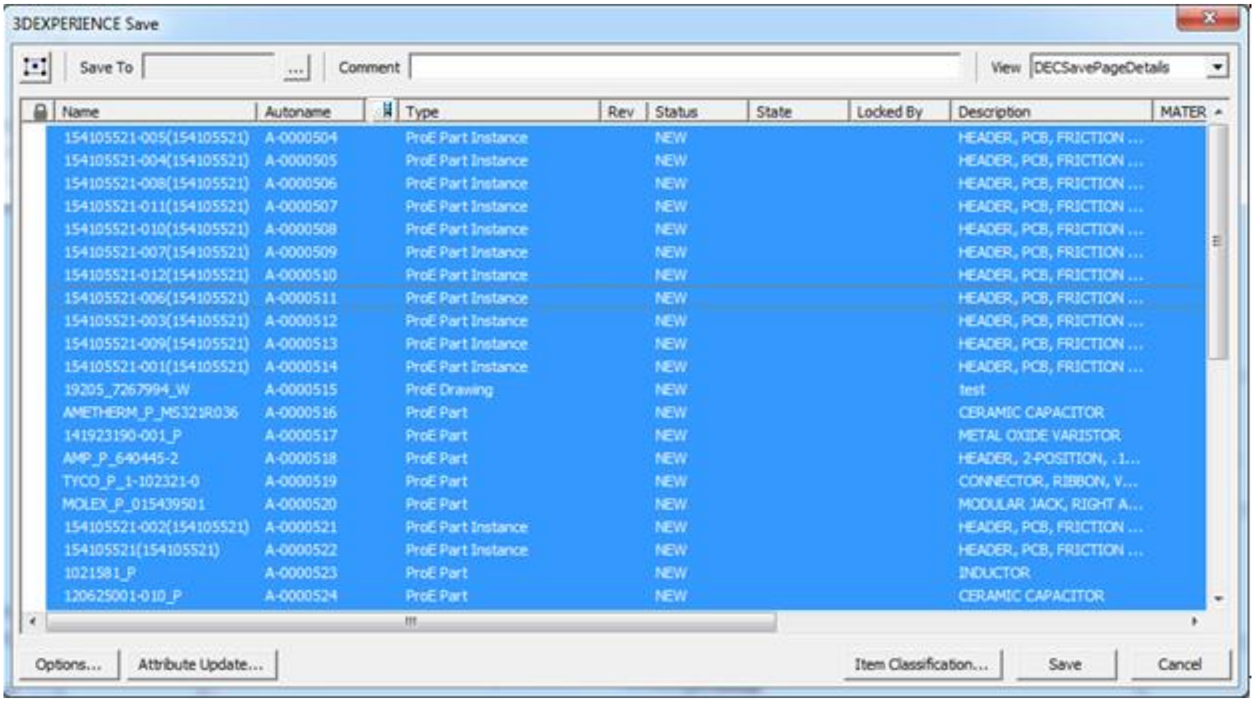




Using Autoname

The Autoname option is available depending on the Preferences set by the Integration Administrator. Only designs with status "New" can be named using Autoname. As a prerequisite, the Autoname series must be set in the 3DEXPERIENCE Save options dialog box.

In the **3DEXPERIENCE Save** dialog box, right-click designs with status "New" and click **Autoname.** Names are generated automatically according to the selected Autoname series in 3DEXPERIENCE Save options dialog box and are displayed under the **Autoname** column in **3DEXPERIENCE Save** dialog box.



On saving, the designs are saved with the names generated automatically.

Note: The integration does not support AutoName feature on instances.

Update PLM Parameters During Save

The PLM parameters such as STATE, REVISION, DESIGNED_BY, MODIFIED_BY are updated in the Creo Parametric parameters so that these parameters can be configured to reflect in to the title block of the Creo Drawing.

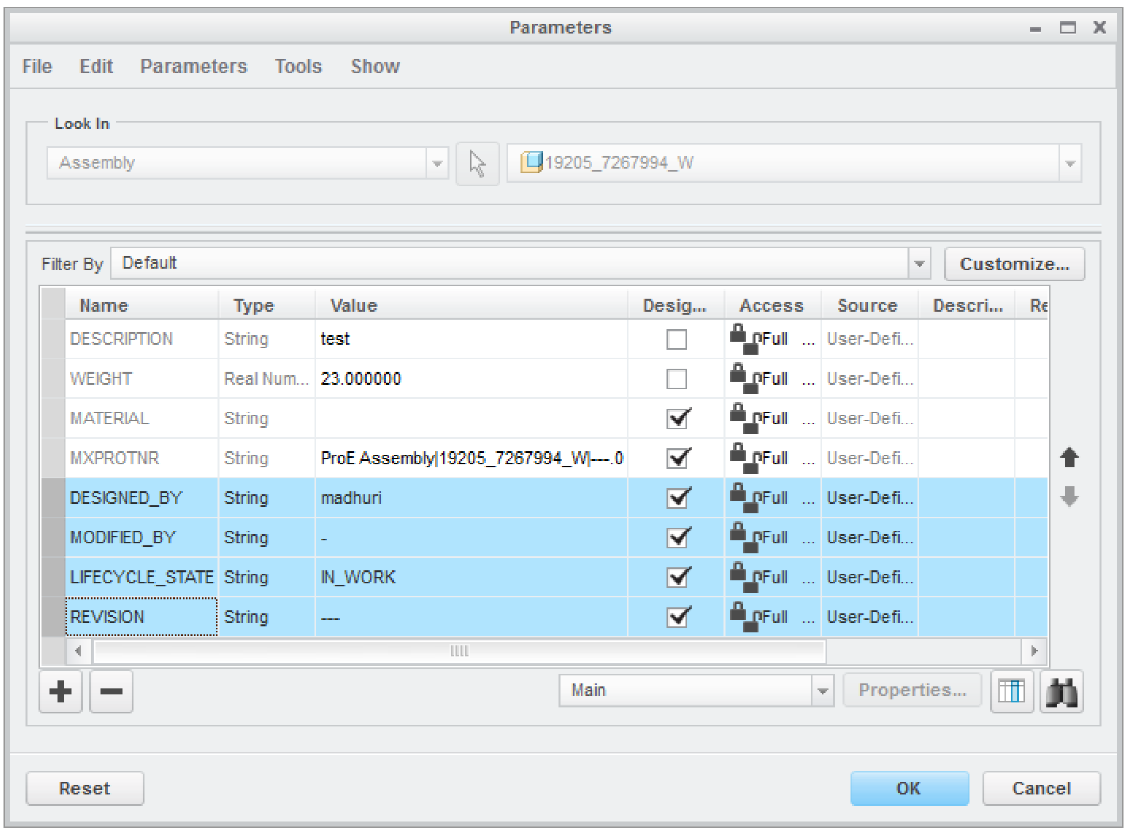
These common PLM parameters will be updated during 3DEXPERIENCE Save operation.

MODIFIED_BY and DESIGNED_BY parameter information will be updated with the User credentials after checkin through integration.

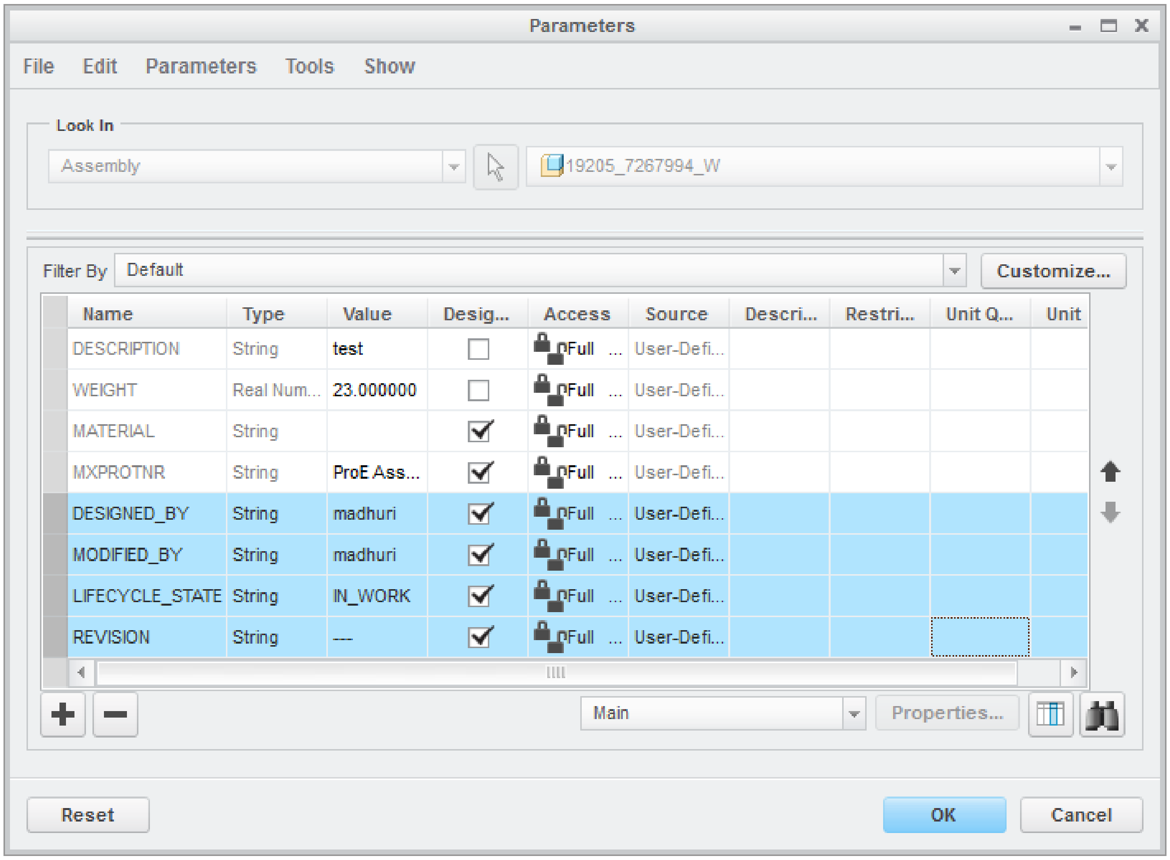
To enable Update PLM parameter functionality you will have to add 'CheckinStart' in GCO attribute IEF-UserExitEventRegistration setting. For details refer to *3DEXPERIENCE Connector for Creo Parametric Install and Administration Guide's 'The Global Configuration Object'* section.

Updating PLM parameters

During first Checkin of Creo Parametric Design to 3DEXPERIENCE, the parameters DESIGNED_BY, LIFECYCLE_STATE, REVISION are populated with appropriate values. MODIFIED_BY will be updated as "-"



During checkin of modified Creo Parametric Design to 3DEXPERIENCE, the parameters DESIGNED_BY, LIFECYCLE_STATE, REVISION will be populated with appropriate values. MODIFIED_BY will be updated as user credentials.



Using Quick Save

This task shows you how to save designs to 3DEXPERIENCE without opening the **3DEXPERIENCE Save** dialog box. The Global preferences set by the Integration Administrator are applied to the designs on saving. If Quick Save happens without Save dialog, then lock of the object checked in is retained with the user.

The Check in preferences must be set by the Integration Administrator.

Click **3DEXPERIENCE** > **Quick Save** from Creo Parametric ribbon/toolbar

3DEXPERIENCE Save dialog box opens if any obsolete nodes are present in the designs being saved.

Note: Some settings in the Global Configuration Object by the Integration Administrator can cause the **3DEXPERIENCE Save** dialog box to open even if obsolete designs are not present in the designs being saved.

The selected designs are saved with the values set by the Integration Administrator in **Check in** Preferences.

- **Delete Local Files**. Integration Administrator selects this option to allow deletion of files of the design on local disk upon saving.

  Note: Delete Local Files option cannot be selected if **Background or Retain Lock** option is selected.

- **Run in background.** Integration Administrator selects this option to save designs without affecting Creo Parametric operations. Copies of the files to be saved to ENVOIA are made and control is returned back to Creo Parametric, so you can continue to use Creo Parametric even when the save process is in progress.

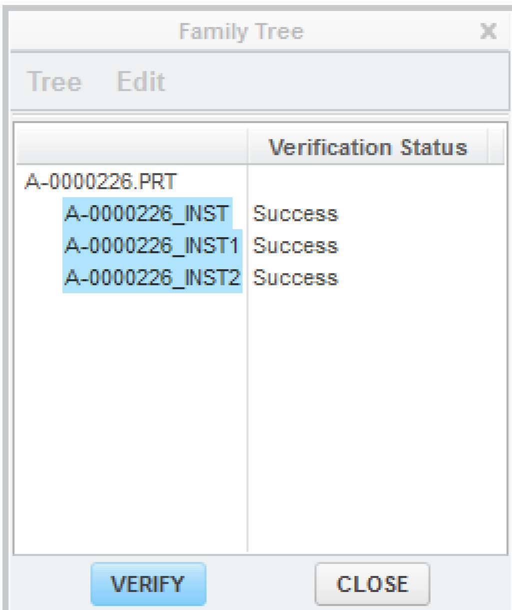
- **Retain Lock.** Integration Administrator selects this option to lock the design. Retain Lock option is shown as selected if user has set user preferences setting called “Retain Lock” option for checkin under PRO tab or set the GCO setting ‘ IEF-Pref-MCADInteg-LockObjectOnCheckin’ to true

- **Create Iteration**. Integration Administrator selects this option to create a new iteration of the design in 3DEXPERIENCE. If not selected, the existing iteration of the design in 3DEXPERIENCE is overwritten by the design being saved.

Saving Family Tables to 3DEXPERIENCE

Family Tables save operation is mainly governed by settings.ini configuration 'ftKeepTogethermode' , by default its value is set to 1, it is called ft together mode turned on, when ftKeepTogethermode value is set to 0 then it called ft together mode turned off.

Note: Ensure that all the instances are verified, before saving to 3DEXPERIENCE.



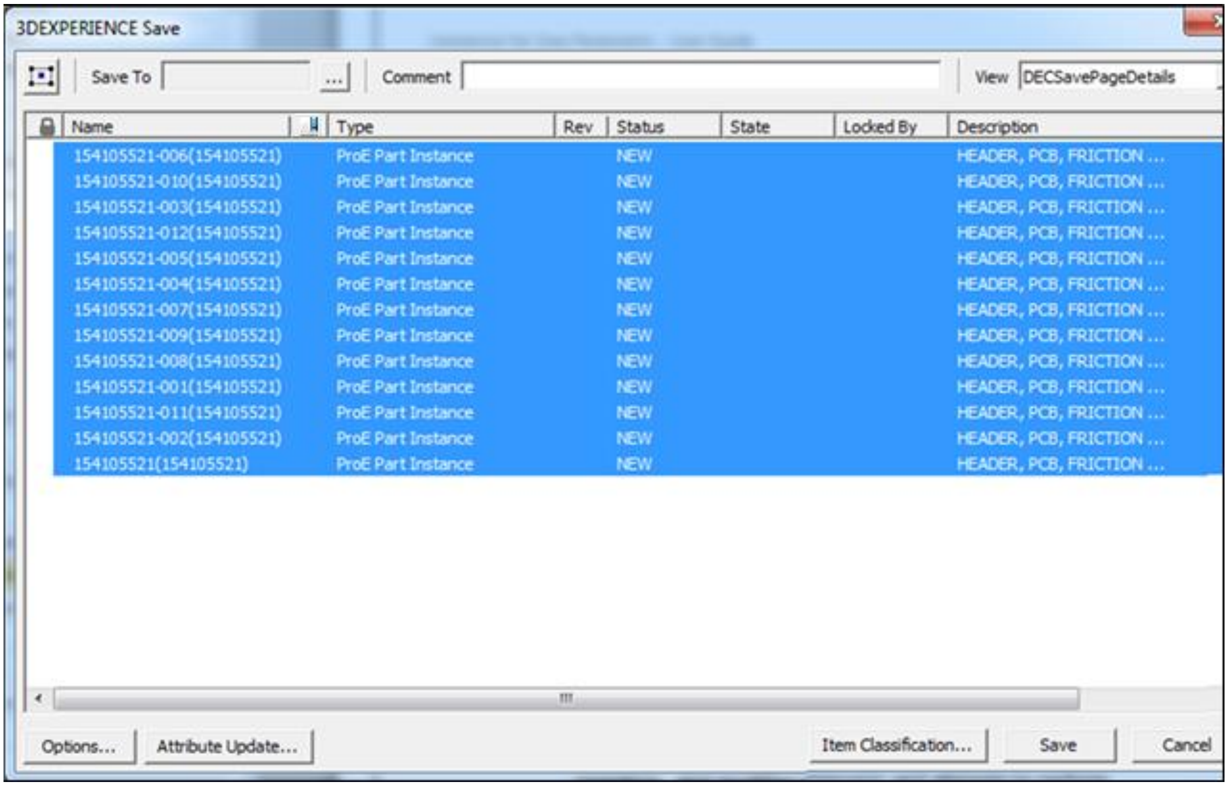
Ensure that the family tables/ required instances are opened in the Creo Parametric.

1. Click **3DEXPERIENCE** > **Save** from Creo Parametric ribbon/toolbar.

   The **3DEXPERIENCE Save** dialog box opens listing the configurations and active instances.

2. Select the family table or instances.

   - If the Family Table is NEW:
     - All instances will be available for selection
     - If any instance or the family table is selected, all instances and the family table will be saved.



Saving Family Table with FT Together mode turned on

- When the User performs the 3DEXPERIENCE->Save of a Family table in the Creo Parametric session, the configuration 'ftKeepTogethermode=1' in settings.ini will make sure that the Generic and all the other instances are selected for continuing with the 'Checkin' operation.

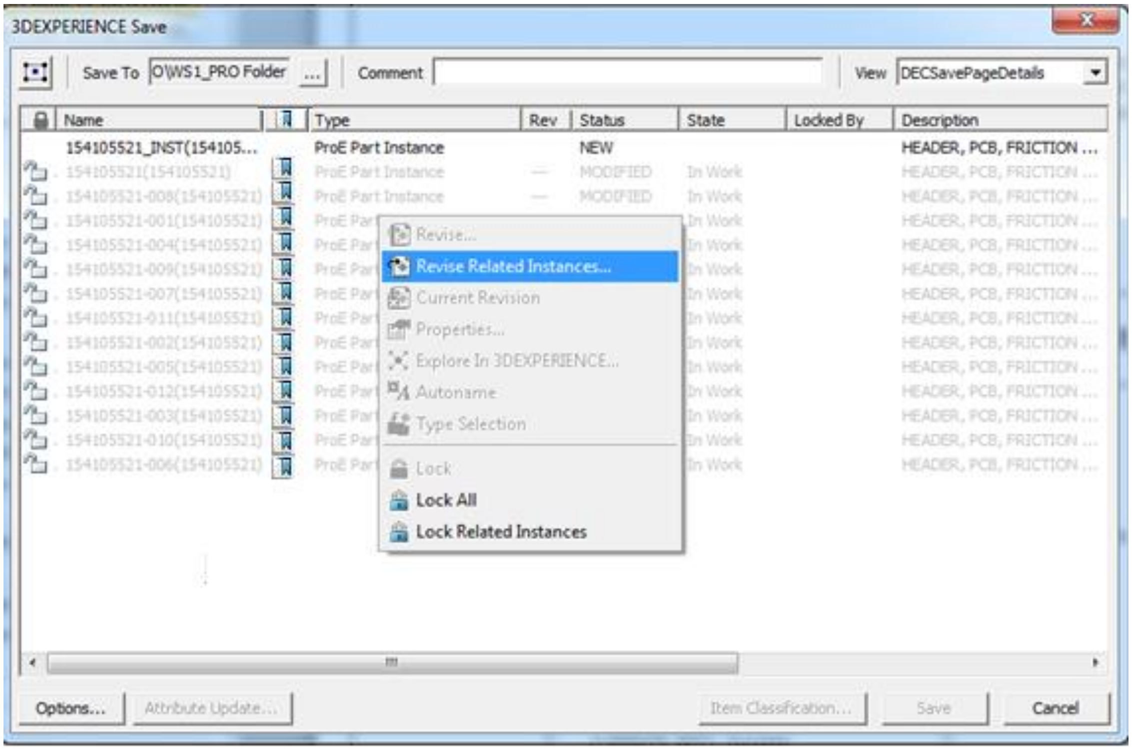
- If the Family Table already exists in 3DEXPERIENCE and an instance has been modified, as the 'ftKeepTogetherMode=1' is enabled in the integration configuration,

    - All the instances of the Family table will be available for selection.

    - Use will have to select all the instances to continue with Save operation.

    - Along with all the instances, the family table will also be saved to 3DEXPERIENCE.

- You cannot select instances with status NEW in **3DEXPERIENCE Save** dialog box if the family table is at a higher state in the object lifecycle. For example, when you attempt to save a family table which is in Review state with instances with status NEW that are in Preliminary state of the lifecycle, then the family table and its existing instances cannot be selected in **3DEXPERIENCE Save** dialog box. For such family tables, the right mouse button commands '***Revise Related Instances***' function must be used.

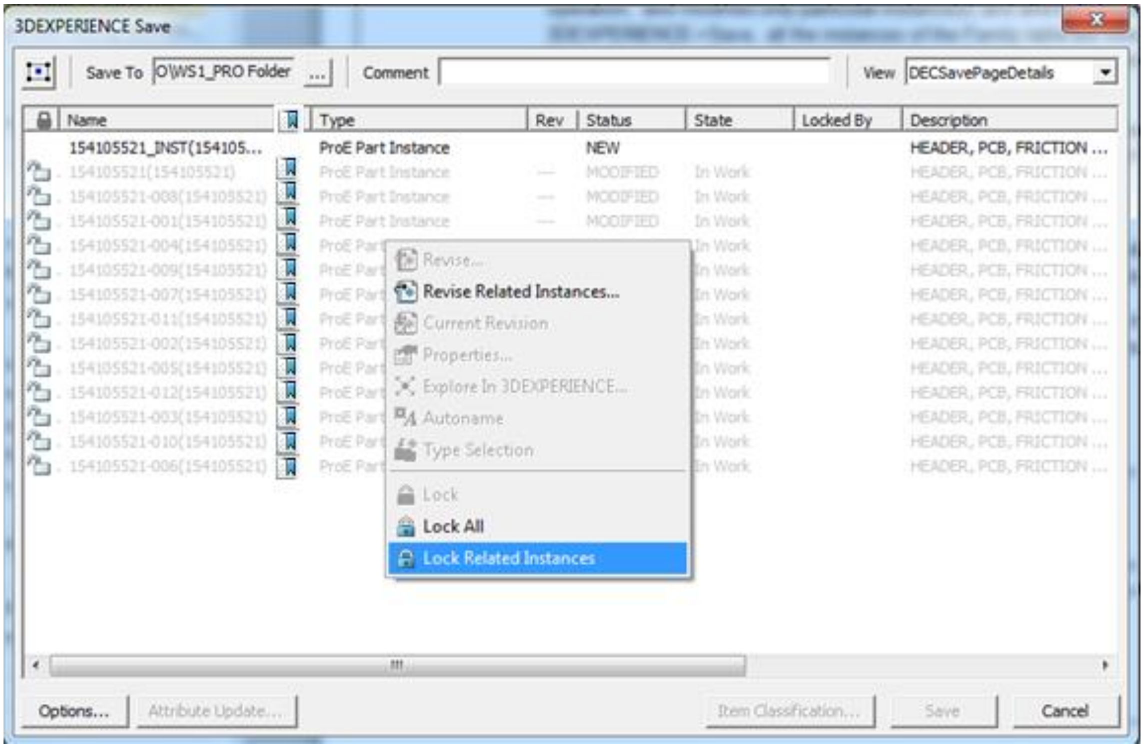


- If the User has not locked all the instances while performing the 3DEXPERIENCE->Open operation, **and** modifies only particular instance(s) and attempts to perform 3DEXPERIENCE->Save, all the instances of the Family table are listed in Save dialog with the instances modified as “Selectable” and the other instances as grayed out (non-selectable).

  In this case, User might choose to use the right mouse button commands, ‘***Lock Related Instances***’ or ‘***Revise Related Instances***’ and selects all the instances and Generic for Save.

  Note: RMB commands ‘***Lock Related Instances***’ and ‘***Revise Related Instances***’ will be available for selection on Family Table Generic and instance objects only.

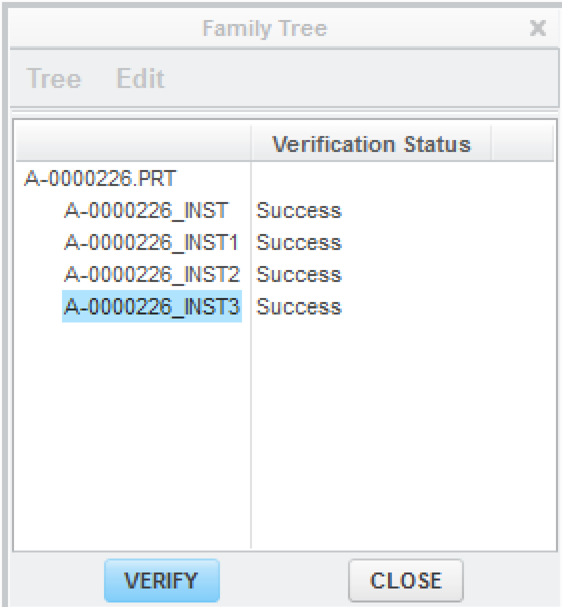
  If User chooses to create a ‘next iteration’ in the same revision stream, the RMB command **‘Lock Related Instances’** needs to be used to perform a ‘Lock’ operation of all the instances.



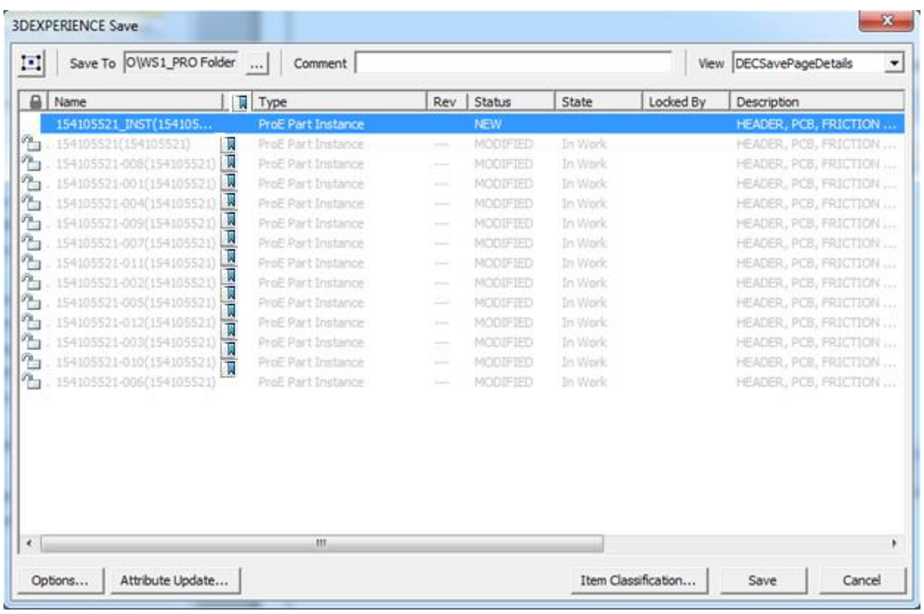
Continuing with the Save operation will create the new iteration in the same revision stream.

- If the User performs the 3DEXPERIENCE->Open of the Family Table Generic to add a new instance to the Family Table, and after adding the new instance if the User attempts to perform 3DEXPERIENCE->Save, all the instances of the Family table are listed in Save dialog with the instances along with the newly added instance.

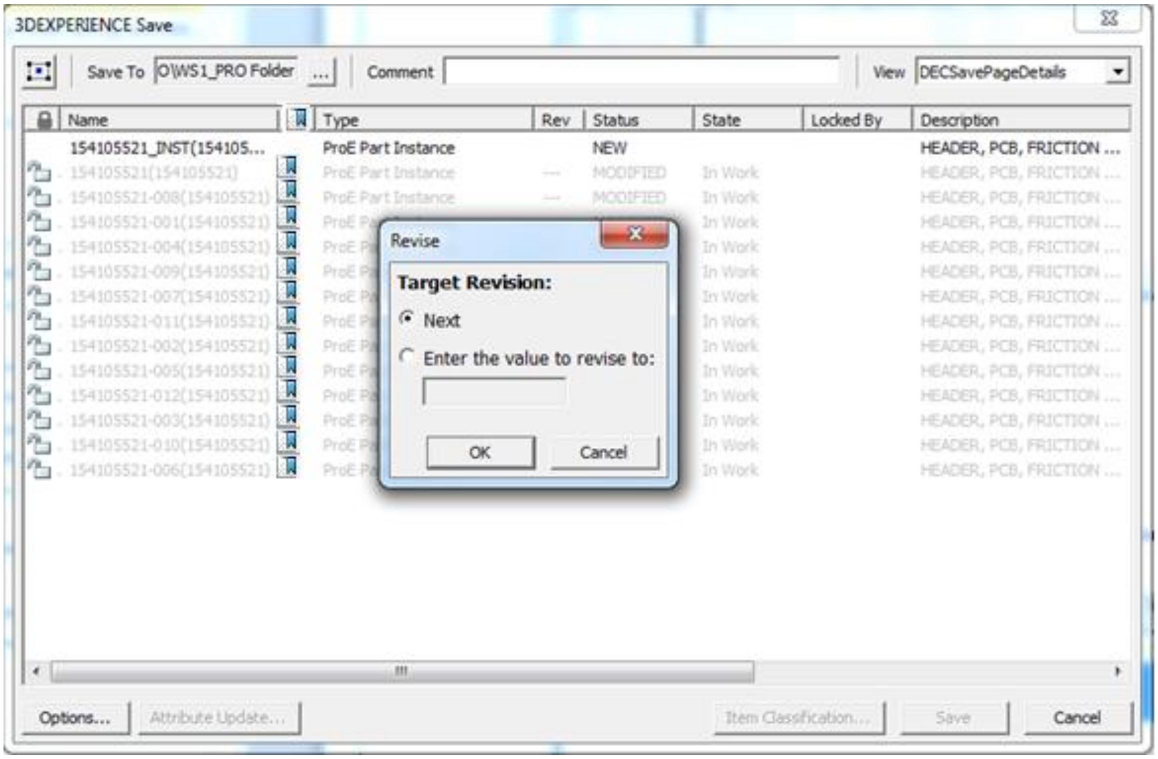
Note: Ensure that all the instances are verified, before saving to 3DEXPERIENCE.

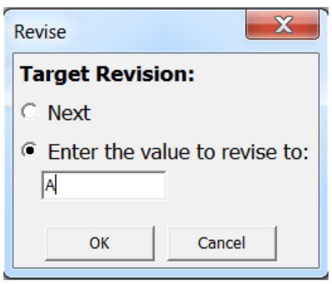


The save dialog displays all the instances along with newly added instance.

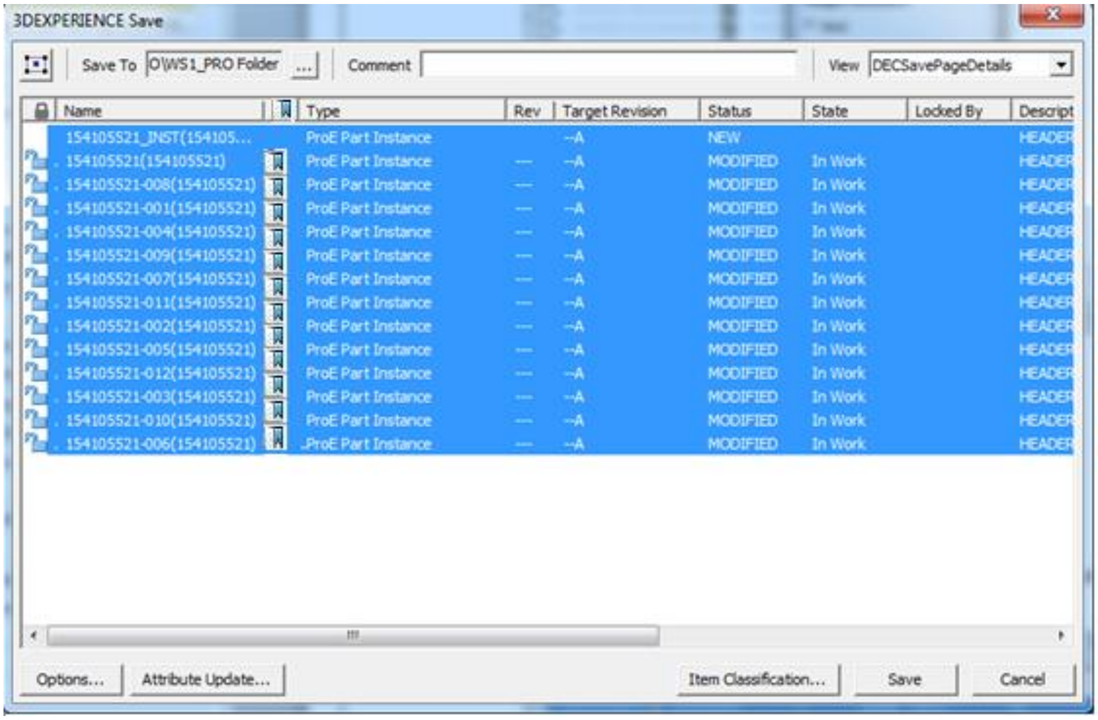


- If User chooses to create a ‘next revision in sequence’ in the same revision stream, the RMB command **‘Revise Related Instances’** needs to be used for choosing the ‘new revision’ to be created upon saving all the instances of the Family Table.





Once user chooses Revise Related Instances the Target Revision column is set with the new revision as chosen the Revision selection dialog.



Continuing with the Save operation will create the new revision in the sequence.

Saving Family Table with FT Together mode turned off

- If the Family Table is NEW:
  - All instances will be available for selection
  - If any instance or the family table is selected, all instances and the family table will be saved.
- If the Family Table already exists in 3DEXPERIENCE and an instance has been modified
  - Only those instances in session will be available for selection.
  - Only those instances selected will be saved.
  - If an instance is selected, the family table will always be saved.
- You cannot select instances with status NEW in **3DEXPERIENCE Save** dialog box if the family table is at a higher state in the object lifecycle. For example, when you attempt to save a family table which is in Review state with instances with status NEW that are in Preliminary state of the lifecycle, then the family table and its existing instances cannot be selected in **3DEXPERIENCE Save** dialog box. For such family tables, the right mouse button Revise function must be used.

Note: Right mouse button **Revise** must be used for family tables in **3DEXPERIENCE Save** dialog box if all the instances with status NEW are inactive.

- Click Save.

The selected configuration and its instances are saved to 3DEXPERIENCE.

Note: If GCO attribute **ProEPartFamilyInstanceHandling** is set to ***PartFamilyInstanceLight*, ** then derived output files will not be generated for family table instances selected for Save unless the instances were in session prior to initiating the Save operation.

Using 3DEXPERIENCE Open or Insert Option

The Open option in 3DEXPERIENCE menu in Creo Parametric is used for a number of functions on designs that exist in 3DEXPERIENCE. The following sections such as; searching for designs, locking and unlocking designs and opening or inserting objects from 3DEXPERIENCE describe the functions available in 3DEXPERIENCE Open dialog box.

Searching Designs

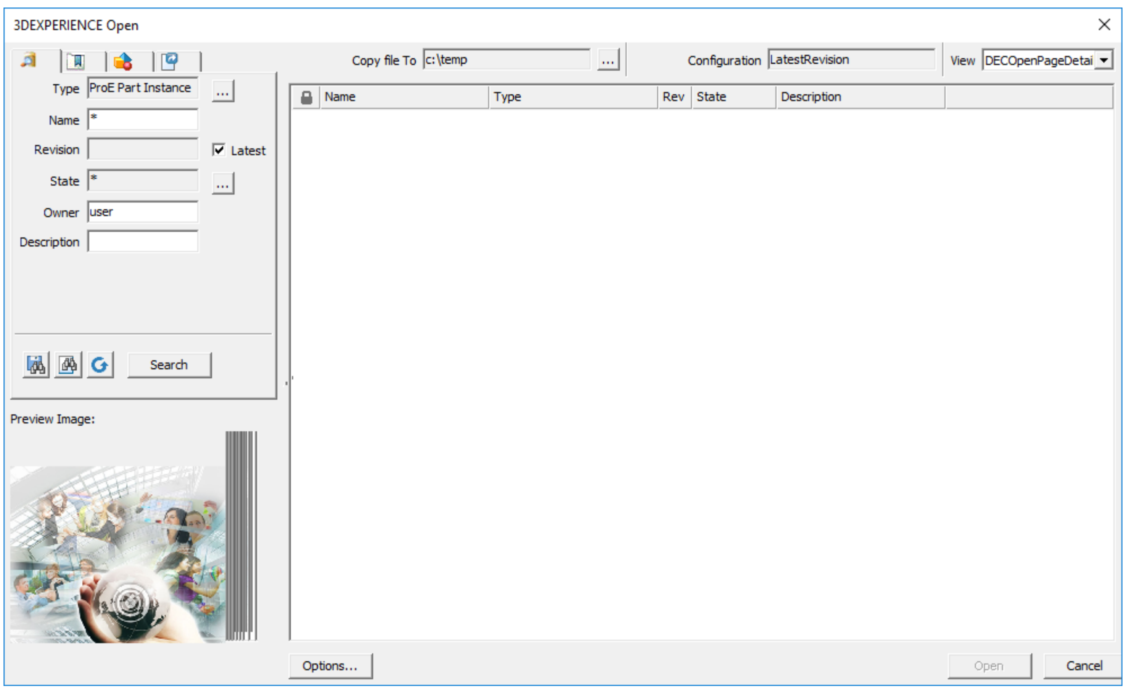
You can search for designs that exist in 3DEXPERIENCE vault from **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box. The following sections describe the steps used to search for designs, saving your query, using the saved query, and removing saved queries.

Searching Designs

This task shows you how to search for designs in 3DEXPERIENCE database from **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box.

1. Click **3DEXPERIENCE** > **Open** or **3DEXPERIENCE** > **Insert** from Creo Parametric ribbon/toolbar.

   The **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box opens depending on your choice.



There are 4 different methods of searching for designs in the 3DEXPERIENCE database:

- Searching via webform driven search parameters.
- Searching user's Workspaces.
- Searching user's Collections.

- Searching user’s recently accessed designs.

Searching Designs Using Webform Driven Search Parameters

To search using webform driven search parameters, click on the tab.

1. Enter criteria as needed; you can use wildcards (*) and enter multiple values (separated by commas) in any text field.

   Note: The fields below are the default delivered parameters available for the Search dialogue. However, since this is a customizable form, the actual display may appear different due to your custom environment.

   • **Type**. Click and select the needed type from the Select Type dialog box.

   • **Name**. Enter the name of the object.

   • **Revision**. Enter a revision sequence or select the **Latest** checkbox to search in the latest revision of the design.

   • **State**. Click and select a lifecycle state from the list.

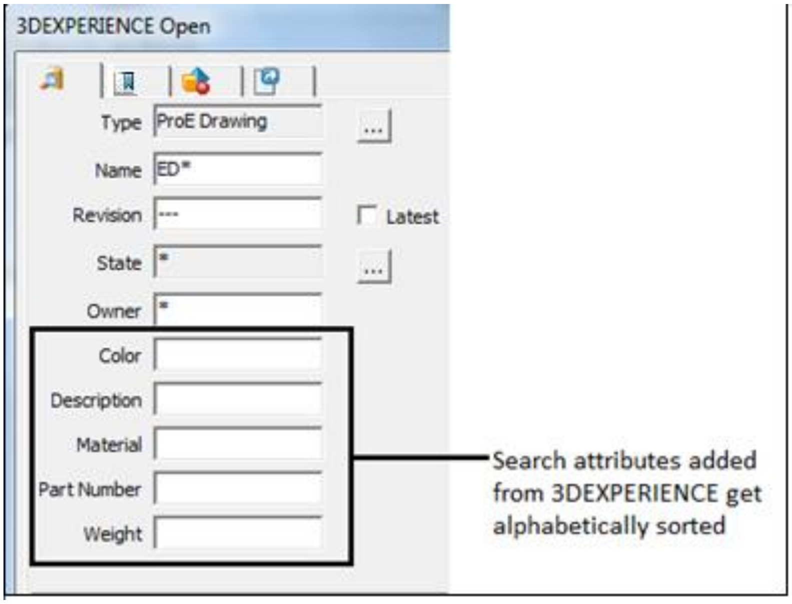
   • **Owner**. Enter the name of the person who owns the design.

   • **Description**. Enter all or part (using wildcards) of the text in the description field.

2. Once you have entered all search parameters Click **Search**.

   Clicking will reset the values of all fields to default values.

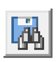
   Note: Following is a search criteria with additional 3DEXPERIENCE attributes added to webform. The attributes are alphabetically sorted in the search form display.



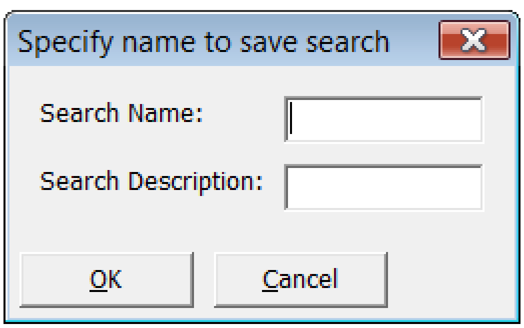
Saving your Webform Driven Search Parameters

This task shows you how to save a query for future use. If you build a query that you need to run periodically, you can save it to have it available for future use.

The search query must be defined.

1. Click  in **3DEXPERIENCE Open** dialog box.

   The **Save Query** dialog box opens.

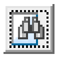
2. Enter the following:
   - **Search Name**. Enter a name for the query.
   - **Search Description**: Enter a description for the query.
3. Click **OK** when done.

The search is saved with the specified name and is listed in the Saved Queries tab in 3DEXPERIENCE Open dialog box.

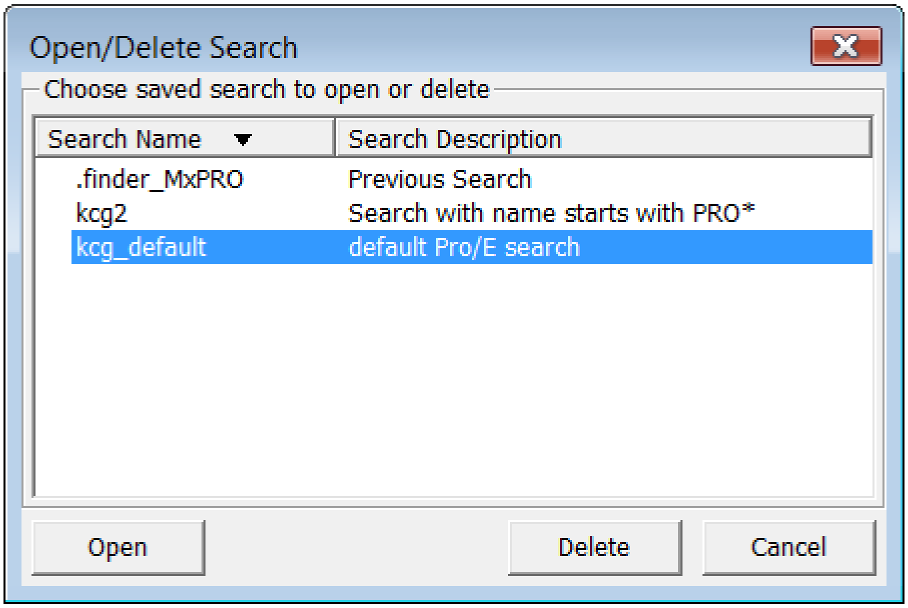
Using Saved Queries

This task shows you how to run/delete a saved query from the 3DEXPERIENCE Open dialog box.

The searches must be saved previously.

1. Click  in **3DEXPERIENCE Open** dialog box.

   The **Saved Queries** dialog box is opened.



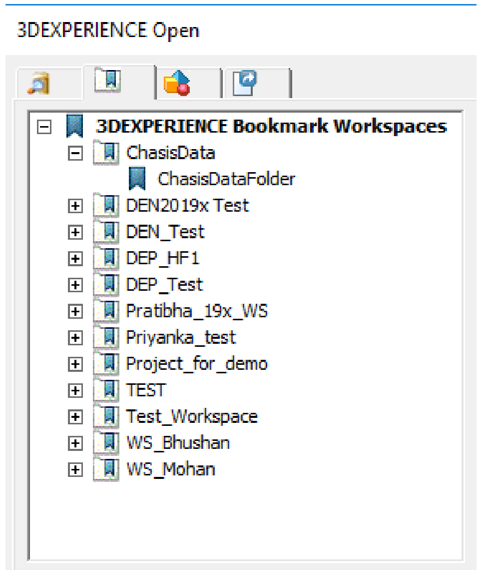
2. Select a query and click Open to populate the search fields.
3. Once all search parameters are set to desired values click **Search**

The search results for the query are listed in the 3DEXPERIENCE Open dialog box.

4. To delete the saved query, select a query in **Saved Queries** dialog box and click Delete.

Searching Designs in User's Workspaces

To search designs in user's Workspaces, click on the  tab. The user's workspaces will be displayed:

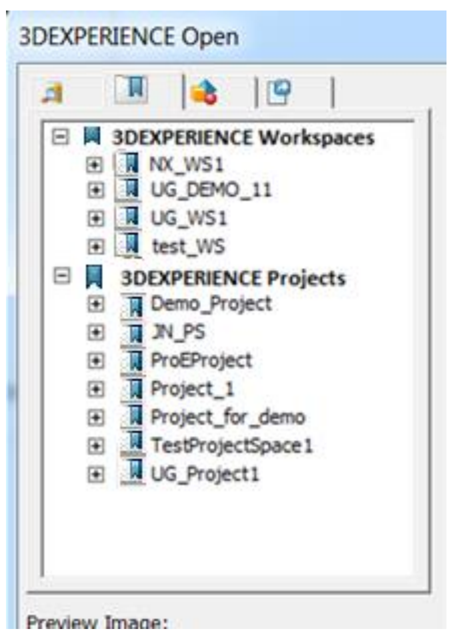


When a workspace is selected, the corresponding search results are displayed in the Open dialog.

Searching Designs in User's Projectspace

If the User is a member of the Program Central Project Spaces with privileges to access the Project folders, The Project folders can also be searched from the 3DEXPERIENCE Open dialog.

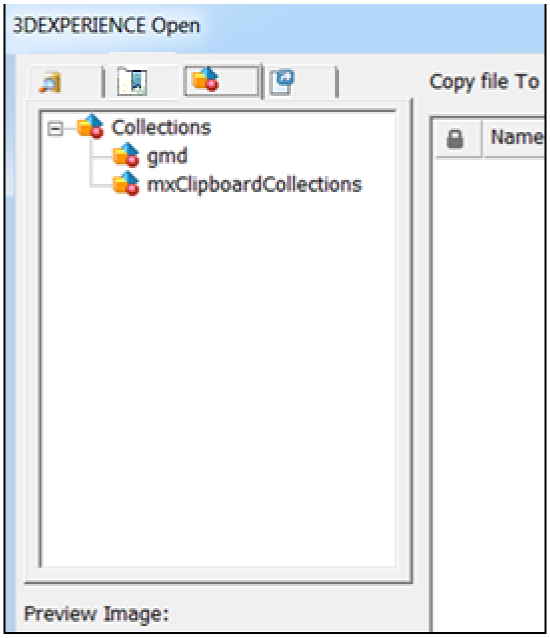
To search designs in user's Projectspace, click on the tab. The user's Projectspace will be displayed:



When a Project space folder is selected, the corresponding search results are displayed in the Open dialog.

Searching Designs In User's Collections

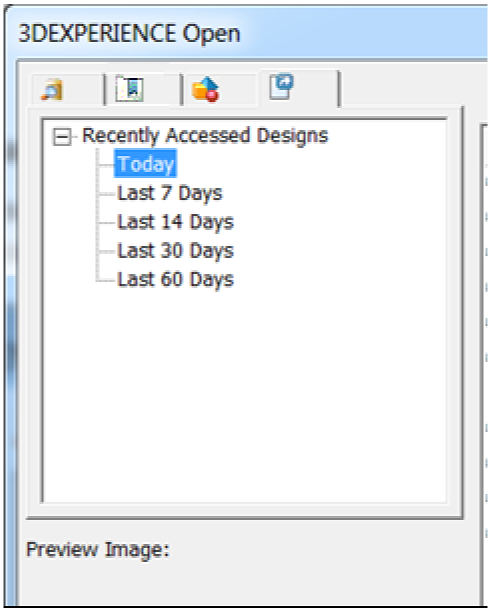
To search designs in user's Collections, click on the tab. The user's collections will be displayed:



When a collection is selected, the corresponding search results are displayed in the Open dialog.

Searching Designs In User's Recently Accessed Designs

To search a user's recently accessed designs, click on the  tab. The following recently accessed classifications will be displayed:



When a recently accessed classification is selected, the corresponding search results are displayed in the Open dialog.

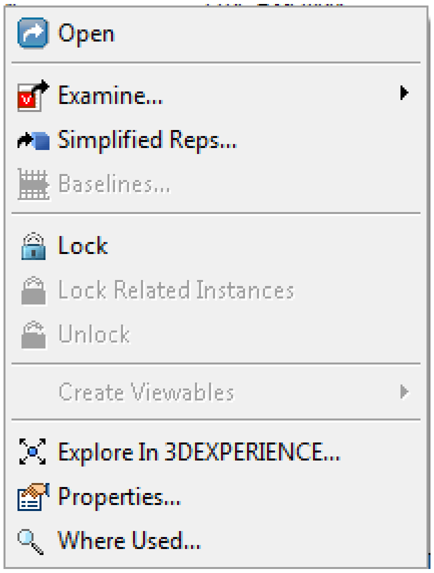
Search Results

Regardless of the search method used, the search results are displayed in **3DEXPERIENCE Open** dialog box. For each design, the **3DEXPERIENCE Open** dialog box lists,

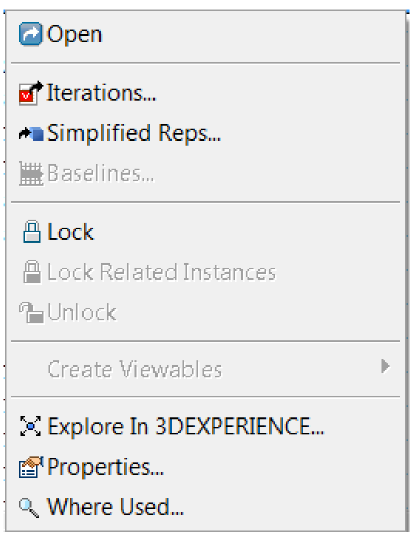
-  **Lock Status.** Whether or not the design is locked.
- **Name**. Name of the design.
- **Type**. Type of the design.
- **Rev**. Revision of the design.
- **State**. Current state of the design in lifecycle.
- **Description**. Description of the design.

Using Context Menu in Open dialog

In 3DEXPERIENCE Open dialog, Right-click the selected design to display the following context menu:



In 3DEXPERIENCE Insert dialog, Right-click the selected design to display the following Context menu:

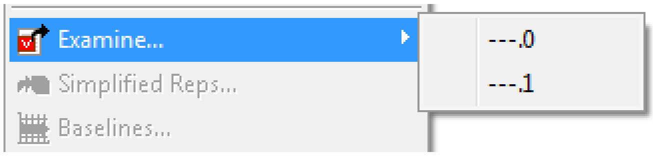


- **Open**. This option is available only if you clicked **3DEXPERIENCE** > **Open** in step 1. Opens selected designs from 3DEXPERIENCE in Creo Parametric. See *Opening Designs From 3DEXPERIENCE in Creo Parametric*

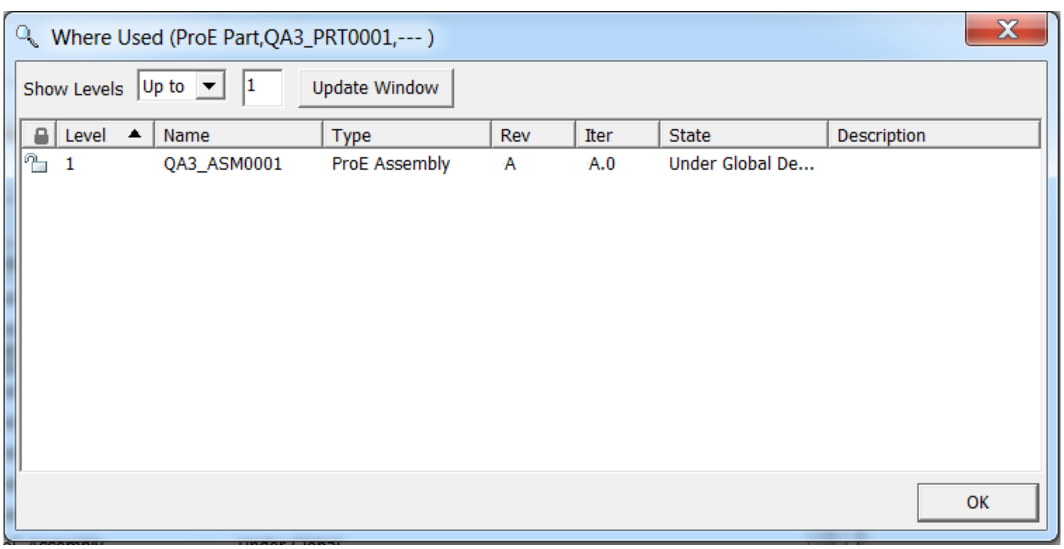
- **Iterations**. Opens the selected iteration of the design in Creo Parametric. This Option is available in Insert Dialog. See Opening Particular Iteration of a Design

Note: 'Iterations' Option is replaced by **Examine** option in OPEN dialog.

- **Examine**. Examines the selected iteration of the design in Creo Parametric. This option is available only in Open Dialog. See *Examine Particular Iteration of a Design*



- **Simplified Reps.** For Creo Parametric assemblies, this option will allow the user to open a specified simplified representation.
- **Baselines**. This option is available only if baselines exist for the selected design. See *Baselining Designs*. The search results are displayed in 3DEXPERIENCE Open dialog box.
- **Lock**. Locks a design. See *Locking and Unlocking Designs*
- **Unlock**. Unlocks a design. See *Locking and Unlocking Designs*
- **Lock Related Instances**: This option is available for family members, (ProE Part Instance) type objects. Using this option user can lock all related instances together.
- **Explore in 3DEXPERIENCE**. Opens CAD Portal view of the design in the browser. See *Viewing Design Details in X-CAD Design*
- **Properties**. Displays properties of a design that already exists in 3DEXPERIENCE. See *Viewing Properties*
- **Where Used**. Displays names of designs that are using the selected model.



Opening Designs from 3DEXPERIENCE

Opening a design copies it from the 3DEXPERIENCE database to your computer and opens or inserts it in another design open in Creo Parametric depending on your choice. The following sections describe the concept of opening designs and the procedure to open designs, insert designs in another design active in Creo Parametric, and open another iteration of the design.

About Opening Designs from 3DEXPERIENCE

Opening a design copies it from the 3DEXPERIENCE database to your computer and opens or inserts it in another design open in Creo Parametric depending on your choice. Your preference settings and global preferences set by Integration Administrator control the open process. You can find the designs you want to open using the Search and select the designs you want to open from the **3DEXPERIENCE Open** dialog box.

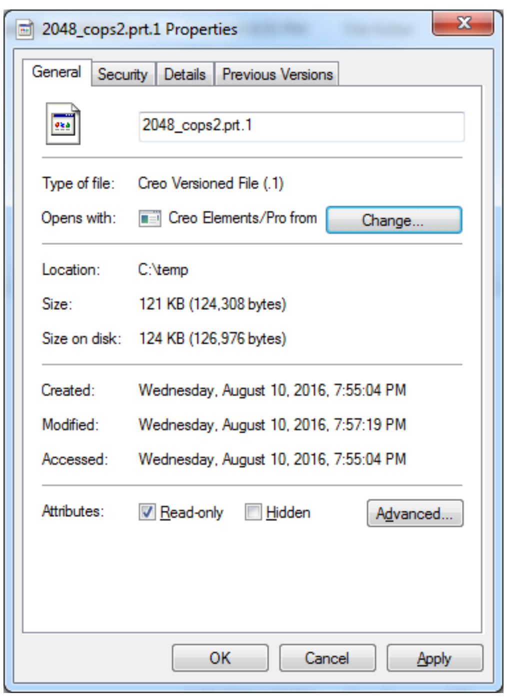
The options to open a file from 3DEXPERIENCE in Creo Parametric are,

- **3DEXPERIENCE** > **Open**. Use this option to open designs from 3DEXPERIENCE in Creo Parametric.
- **3DEXPERIENCE** > **Insert**. Use this option to insert the design from 3DEXPERIENCE in the currently active design in Creo Parametric.
- **3DEXPERIENCE** > **Open Partial**. Use this option to open preconfigured partial structures from 3DEXPERIENCE in Creo Parametric.

Read Only

When using **3DEXPERIENCE > Open** to download files without lock, the part will be marked as 'Read Only' on the disk.

The 'Read Only' designation on a file can be removed by either locking the 'Read Only' Creo Parametric model in 3DEXPERIENCE or modifying the part on disk to remove the 'Read Only' designation.



Attribute Synchronization

Attribute synchronization ensures that the attributes in the design file on your computer match the attributes of the associated 3DEXPERIENCE object.

During the checkout process, there can be discrepancy in the attributes or properties of the design. When you open the design, you want those attributes and properties to become part or updated in the Creo Parametric design object.

Your Integration Administrator specifies which attributes and properties that can be synchronized. Attributes will be synchronized from the 3DEXPERIENCE metadata to CAD data during Open.

As the integration added default values of mapped attributes in the Mx-To-CAD GCO option to the Creo objects on the checkin. See Attribute Synchronization during Save. The integration will update those attributes to the objects on the checkout.

Note: See *X-CAD Design Install and Administration Guide* for details on attributes that are mapped and mapping attributes.

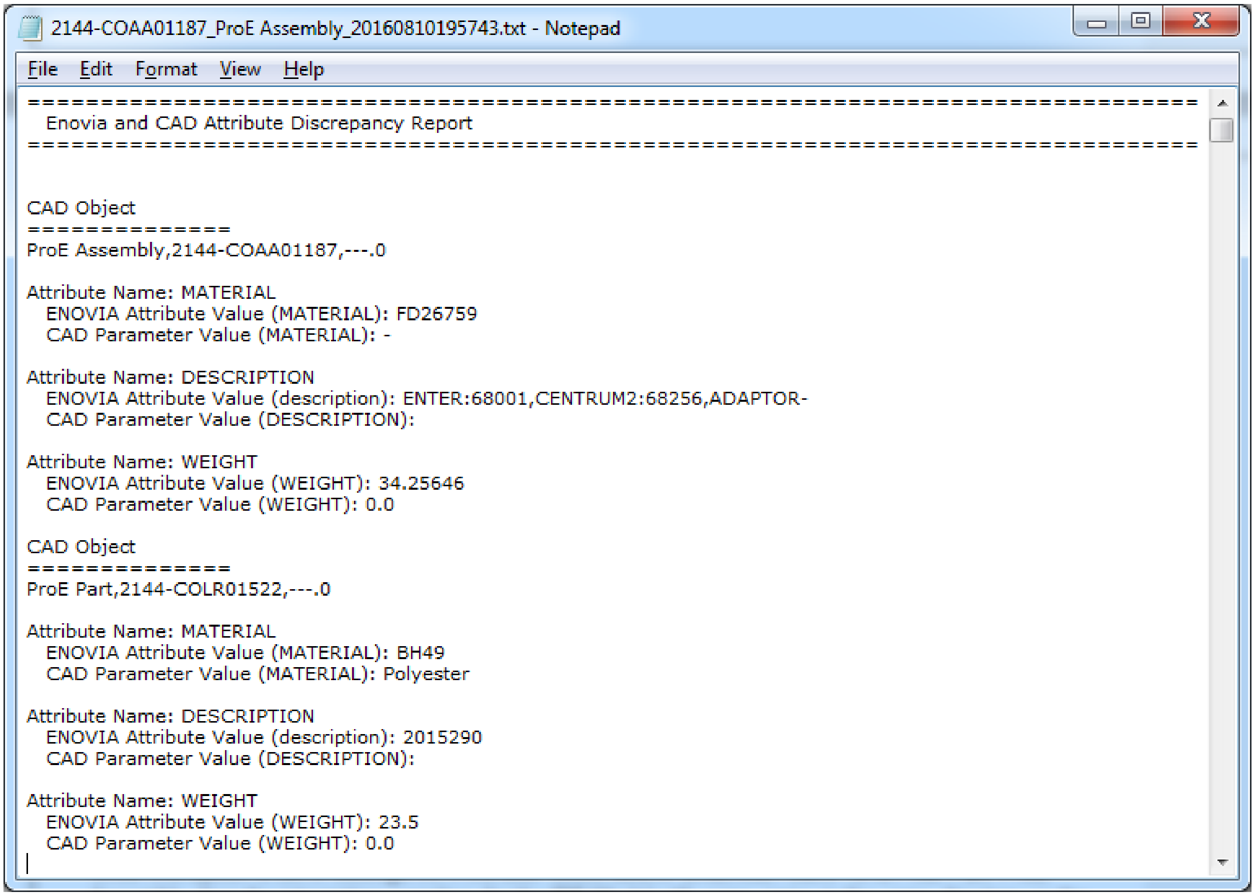
Attribute Discrepancy Report Generation

The attribute discrepancy report is a text file report that is generated and saved on disk upon check-out when there is a discrepancy detected between the attributes of 3DEXPERIENCE metadata and CAD data. This functionality is enabled through the client side's settings.ini file by defining the parameter 'attribDiscrepancyReport = 1 ;'.See Configure setting.ini

If this functionality is enabled, then attribute synchronization of attributes of 3DEXPERIENCE metadata to corresponding CAD attributes in the design file will be ignored.

If this functionality is disabled, then the integration will do attribute synchronization of attributes of the associated 3DEXPERIENCE object to corresponding CAD attributes in the design file during check-out.

On check-out, the integration will check all attributes listed in the GCO attribute "MCADInteg-MxToCADAttribMapping" mapping for finding discrepancies between the attributes value of 3DEXPERIENCE metadata and the value of CAD attribute stored in the native Creo Parametric design.




```
==============================================================================
  Enovia and CAD Attribute Discrepancy Report
==============================================================================


CAD Object
==============
ProE Assembly,2144-COAA01187,---.0

Attribute Name: MATERIAL
  ENOVIA Attribute Value (MATERIAL): FD26759
  CAD Parameter Value (MATERIAL): -

Attribute Name: DESCRIPTION
  ENOVIA Attribute Value (description): ENTER:68001,CENTRUM2:68256,ADAPTOR-
  CAD Parameter Value (DESCRIPTION):

Attribute Name: WEIGHT
  ENOVIA Attribute Value (WEIGHT): 34.25646
  CAD Parameter Value (WEIGHT): 0.0

CAD Object
==============
ProE Part,2144-COLR01522,---.0

Attribute Name: MATERIAL
  ENOVIA Attribute Value (MATERIAL): BH49
  CAD Parameter Value (MATERIAL): Polyester

Attribute Name: DESCRIPTION
  ENOVIA Attribute Value (description): 2015290
  CAD Parameter Value (DESCRIPTION):

Attribute Name: WEIGHT
  ENOVIA Attribute Value (WEIGHT): 23.5
  CAD Parameter Value (WEIGHT): 0.0
```

The report will be saved to local disk at the path specified by the settings.ini parameter 'attribDiscrepancyReportDir'.The report file name consists of the name of selected design its 3DEXPERIENCE type name and the time stamp of checkout.

Note: Separate Report file is generated for each object selected for checkout in Open dialog.

Opening Multiple Designs

Using **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box you can search, select, and open or insert multiple designs from 3DEXPERIENCE in Creo Parametric. The Global Preferences set by the Integration Administrator or the options selected in 3DEXPERIENCE Options dialog box are applied for all the selected designs.

Note: Preferences that are enforced by the Integration Administrator cannot be modified.

Selected designs are opened to the working directory and are loaded to the active session.

It is possible when selecting multiple designs to open simultaneously that a conflict may occur if your business process is not enforcing the "Latest Iteration" rule. Multiple iterations of the same design may not be opened at the same time.

An example of iteration collision is: Iteration 1 of component X which is not a part of any assembly and iteration 3 of the same component which is a part of the assembly are selected for opening in the 3DEXPERIENCE Open dialog box. When you try to open the selected designs an error message is displayed to indicate the iteration collision.

Opening Designs from 3DEXPERIENCE in Creo Parametric

This task shows you how to open designs from 3DEXPERIENCE in Creo Parametric.

The designs must already be saved in 3DEXPERIENCE. See *Saving Designs to 3DEXPERIENCE*

1. Click **3DEXPERIENCE** > **Open** from Creo Parametric ribbon/toolbar.

   The **3DEXPERIENCE Open** dialog box opens.

2. Search for the designs using **Search** option in **3DEXPERIENCE Open** dialog box. See *Searching Designs*

   The results of the search are listed.

   Select a design from the list to view the preview image of the design which is displayed in the left corner of **3DEXPERIENCE Open** dialog box.

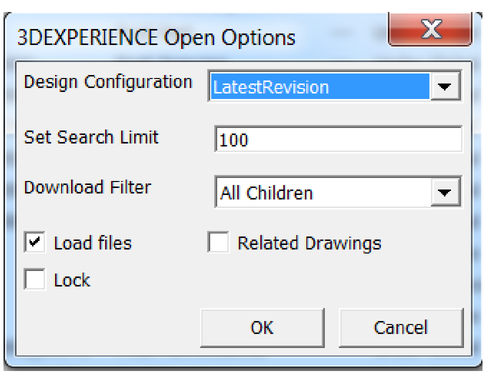
3. Enter the following options in **3DEXPERIENCE Open** dialog box:

   - **Copy file To**. The designs opened from 3DEXPERIENCE are downloaded to the directory specified. By default, the integration will set the checkout directory to the working directory of the ProE session. Click **[...]** to browse and select another directory as the checkout directory.

   - **View**. Click and select a table defined by the Integration Administrator from the drop down list. The selected table is applied to the **3DEXPERIENCE Open** dialog box.

4. Click **Options** in **3DEXPERIENCE Open** dialog box.

   The **3DEXPERIENCE Open Options** dialog box opens



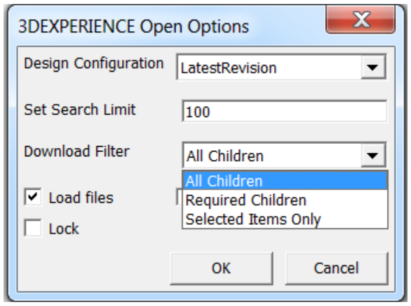
.

   Choose options to be executed when you open a design from 3DEXPERIENCE:

   - **Related Drawings**. Downloads all associated designs to the selected model. The files are copied to the user's designated local working directory.

- **Set Search Limit**. Using this option, an integration user can set the search limit.
- **Design Configuration**. Select an option from the drop down list to display specific iterations or revisions of designs in **3DEXPERIENCE Open** dialog page.
- **Load files.** Loads files to PROE session.

  - **Download Filter**: There are three options available in this dropdown.



    - **All Children.** Select this checkbox to checkout the child objects of selected objects. The intial value is populated based on the user's MCADInteg-SelectChildItems Local Configuration setting (also available via 'Select Children if Parent is Selected' option in Designer/Preferences). Leaving this checkbox blank will improve performance if, for example, the user desires to only checkout the top level of an assembly or the user desires to checkout a family table template without its family table members.
    - **Required Children.** This option allows user to only download direct children, models connected by assemblyComponent and drawing relationships, on checkout. On checkout with this option, the user will not get models connected by external-copy geometry relationship
    - **Selected items only.** This option allows user to only download the items selected in the search results pane.

- **Lock.** Locks assembly and its components on checkout

Click **OK**.

5. Click **Open**.

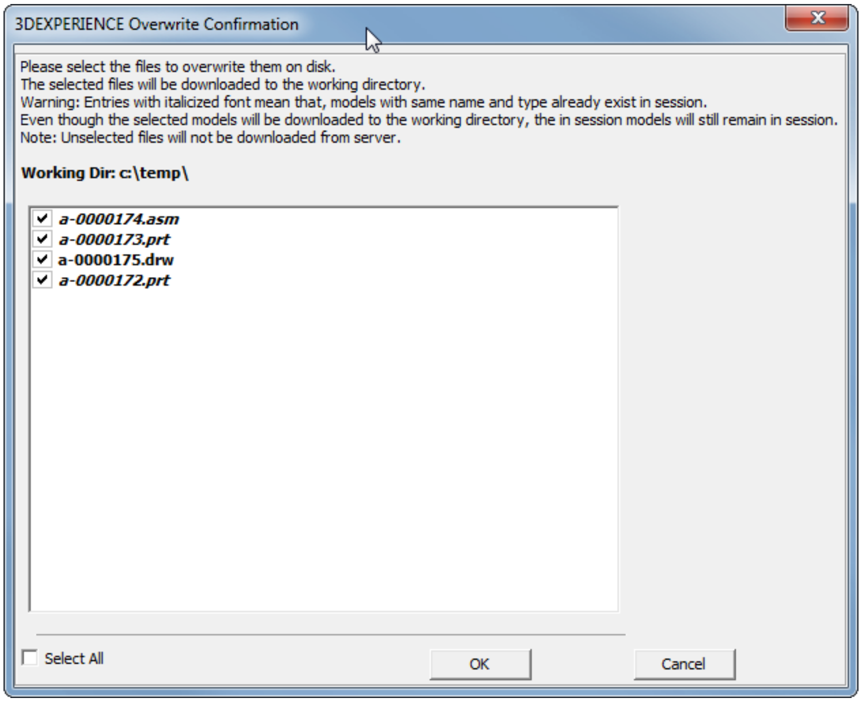
During open, if RFA setup is enabled, then the designs that are locked by user are downloaded to local disk. Hard links are also created for all the designs that are not locked by user.

Open Enhancements to complete checkout when models are in session

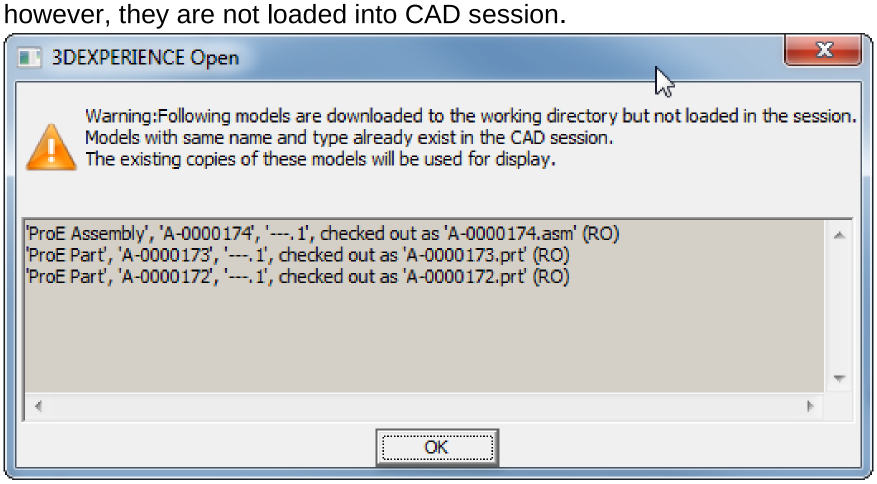
The selected designs are opened in Creo Parametric and also downloaded to the location specified in '**Copy file To'** in **3DEXPERIENCE Open** dialog box.

The 3DEXPERIENCE->Open behavior is enhanced to improve the usability and make it convenient for the CAD users to continue the checkout without having to close the models they are working on in current CAD Session.

User is working on few of the CAD models in the current session. If 3DEXPERIENCE | Open command is invoked and user has selected the models to open, which are already in the session, integration will prompt the user with below overwrite confirmation dialog. If the models exist on disk but not loaded in session, they will be displayed in normal font. However, if the models exist on disk and are already loaded into the session, they will be displayed in *Italic* font. User can select the files they want to overwrite on disk. If user responded to this dialog with "OK", the selected files will be downloaded to the working directory.



User will be prompted with the below warning that files were downloaded to the working directory, however, they are not loaded into CAD session.



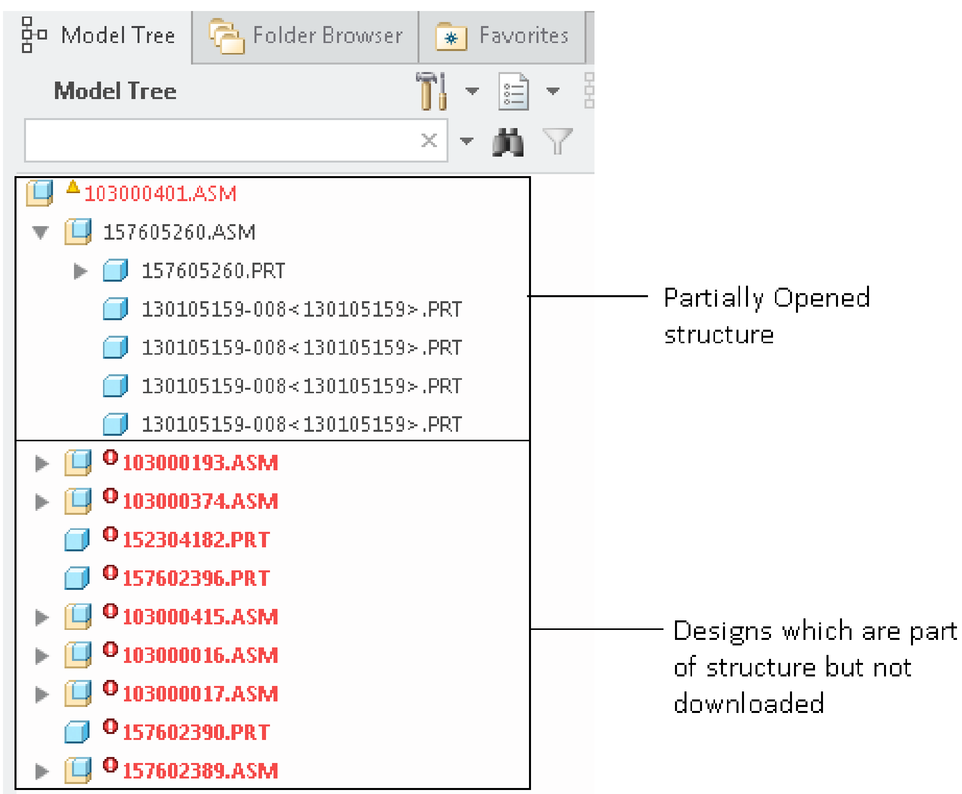
Opening partial structures from 3DEXPERIENCE in Creo Parametric

This feature is helpful when user has to work on a small portion of a big assembly.

The feature allows user to open partial structure, from a filtered BOM generated in 3DEXPERIENCE. Such partial structure after opening in CAD, if modified and saved to 3DEXPERIENCE, maintains structure integrity.

This feature requires server side and client side configuration. Partial open functionality will work when server side components are configured. For setting up Open Partial client side configuration please refer 'CreoOpenPartialFromWebInstallationGuide.pdf' available in 3DEXPERIENCE Connector For Creo Parametric installation.

1. Open Partial operation is initiated from 3DEXPERIENCE thin client by selecting filtered BOM.
2. Once user initiates the operation from 3DEXPERIENCE thin client, user has to click on 3DEXPERIENCE >Open Partial, menu from 3DEXPERIENCE ribbon to complete the operation, this will initiate download of the partial structure and open it in Creo Parametric.
3. By default, the 'Open Partial' menu is disabled. Integration Administrator can enable this menu from Global Preferences on 3DEXPERIENCE server.
4. Following is a Model Tree of partially opened structure in Creo Parametric, where only a portion of structure is downloaded.



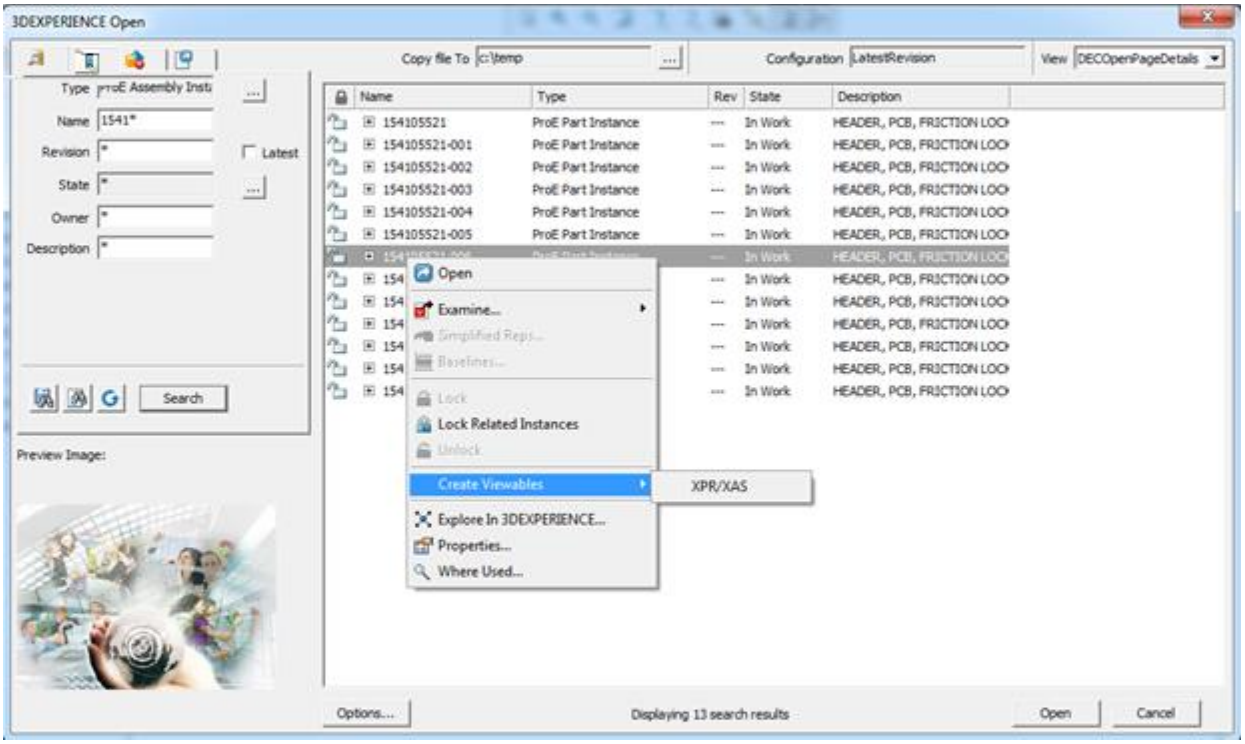
5. In case above Partially Opened structure if the designs are modified and checked in by using 3DEXPERIENCE>Save functionality, the preexisting structure members which are not partially Opened will still remain in structure in 3DEXPERIENCE, and the changes made in CAD session will be merged with the structure in 3DEXPERIENCE.

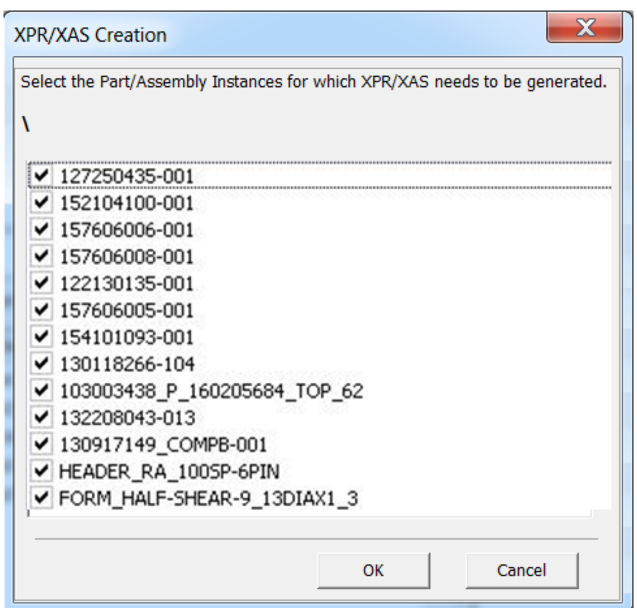
Note: If a structure is partially checked out from 3DEXPERIENCE>Open dialog and then it is checked in after modification, then the designs not checked out may get disconnected from the parent structure and the structure integrity will be compromised.

XPR/XAS Derived Output

This feature allows users to create and upload XPR/XAS files of Part and Assembly Families.

The below dialogs show the steps to create XPR/XAS files.





Inserting Designs from 3DEXPERIENCE in an Active Design

This task shows you how to insert designs from 3DEXPERIENCE to a design that is the active open design in Creo Parametric.

Using **3DEXPERIENCE > Insert** function, you can insert,

- Part in Assembly
- Assembly in Assembly

You cannot insert,

- Assembly in Part
- Drawing in Part
- Drawing in Assembly
- Part in Drawing
- Assembly in Drawing

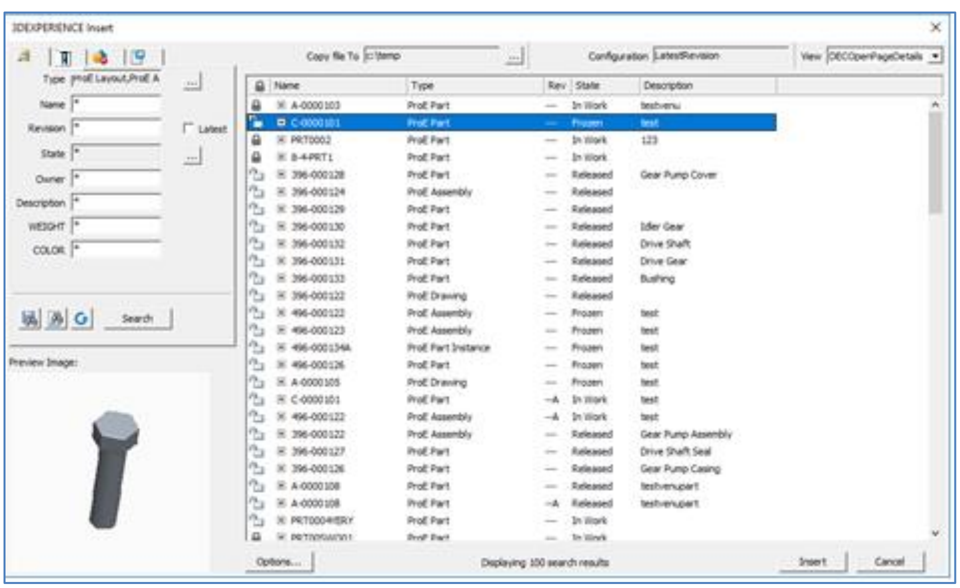
The design in which you want to insert another design must be open and active in Creo Parametric.

1. Click **3DEXPERIENCE** > **Insert** from Creo Parametric ribbon/toolbar.

   The **3DEXPERIENCE Insert** dialog box opens.

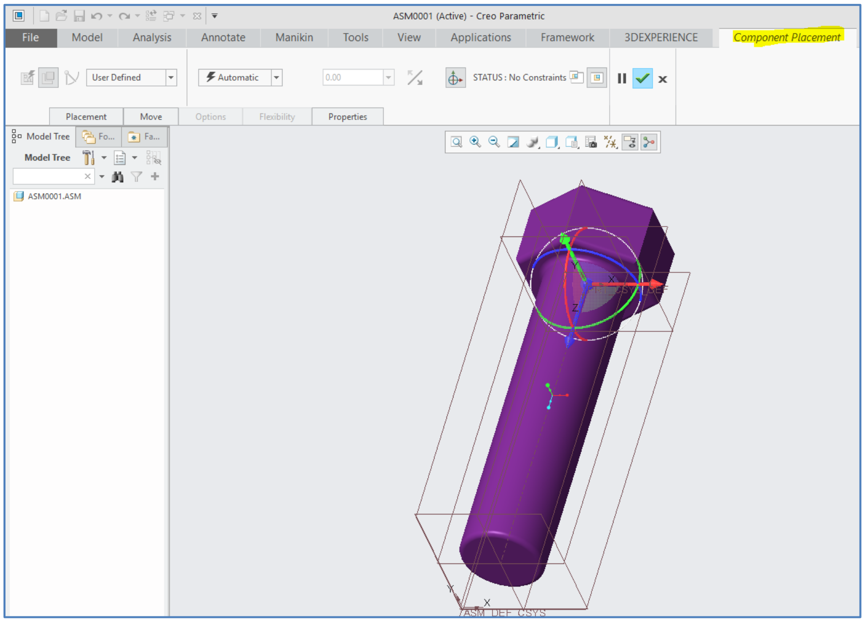
2. Search for the design using **Search** option in **3DEXPERIENCE Open** dialog box. See *Searching Designs*.

   The search results are listed in **3DEXPERIENCE Insert** dialog box.

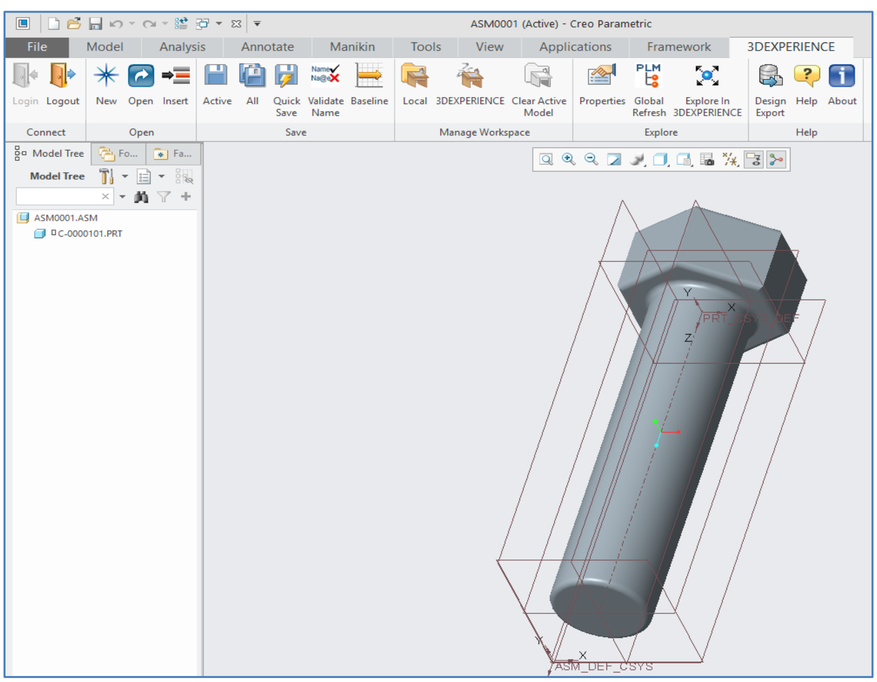


3. Click **Insert** button or right-click the design and click **Open**.

4. With Assembly in session, the design will be downloaded to the disk and Creo Native 'Component placement' UI will be shown to user, as shown below. In this UI, user can select the constraints and alter the component placement.



5. Once user confirms the constraint and placements, the component will be placed in the assembly as shown below



Derived Output Support

The 3DEXPERIENCE Connector for Creo Parametric supports the checkin of derived output files. These derived output files can be dynamically generated (native) or may already exist on disk (user-defined).

Native derived outputs

The 3DEXPERIENCE Connector for Creo Parametric uses the native CAD tool API's to programmatically generate these files during checking based on the users' preferences and selections. The files are generated in the same directory as the ProE files and are deleted upon completion of the checkin depending on the value of delete_on_checkin in the ProEDerivedOutputOptions. The 3DEXPERIENCE Connector for Creo Parametric supports the following native derived outputs with the necessary additions to the global configuration object.

Parts and assemblies:

JPEG

TIF

IGES

STEP_AP203

CGR

PNG

Drawings:

POSTSCRIPT

DXF

IGES

CGM

PDF

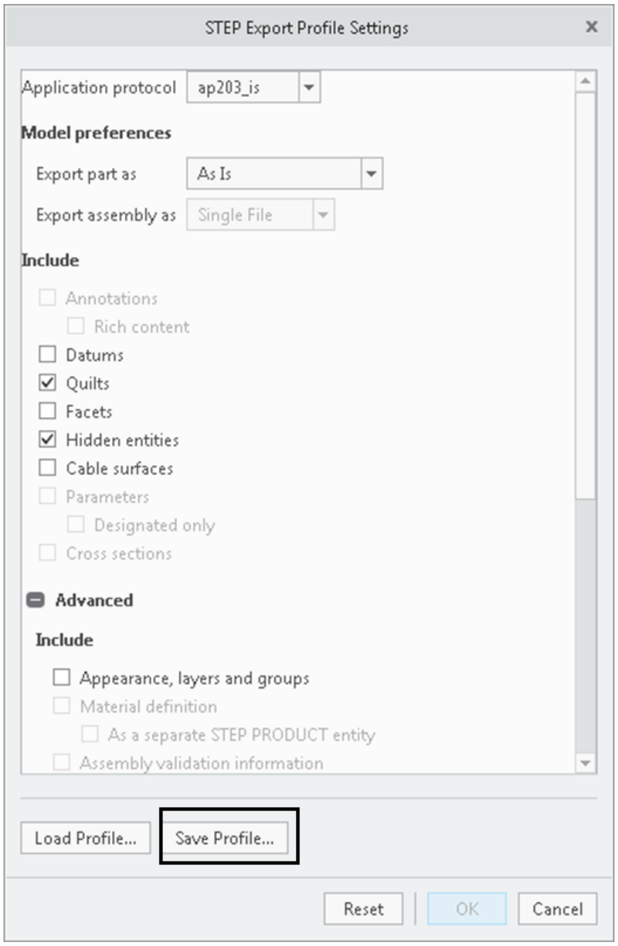
For ProE Part Instances, the integration supports the derived supports XPR derived output in addition to the derived outputs mentioned under that "Parts and Assemblies." Similar in addition to the derived outputs mentioned under the Parts and Assemblies, the integration supports XAS derived output for ProE Assembly Instances.

In the GCO, the users can modify the parameters, such as black\white PDF or color PDF output, used to generate derived outputs.

ProE allows some level of control over the generation of JPEG, TIF, and POSTSCRIPT files. Refer to the ProEDerivedOutputOptions section for more information.

STEP derived output generation with STEP translator config options.

Automatic STEP derived output generation supports following STEP export options, available in Creo 4.



For this enhancement to work a STEP Export Profile must be generated using the 'Save Profile…' button from Creo Parametric STEP Export Profile Settings dialog shown above.

The generated STEP export profile path must be specified in integration settings.ini option **'step_export_profile'** under 'CAD' section.

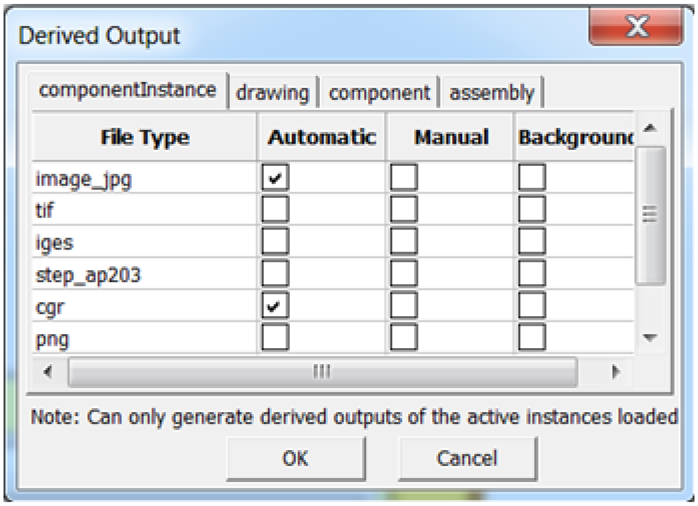
Once the settings.ini is set and Integration is restarted, during checkin of a design, if STEP derived output is selected under automatic section then the STEP derived output generated will be as per the options specified in the STEP export profile.

Note: This enhancement is only supported Creo4 M010 onwards.

Creation of CGR Derived Output on Save

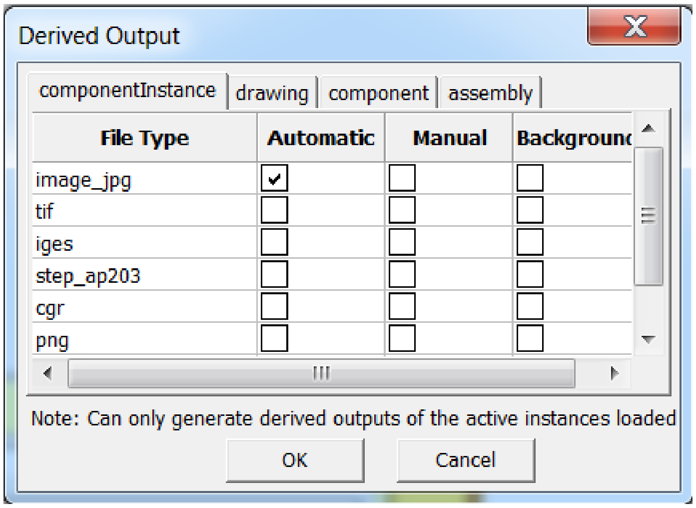
During the Save operation is where the user can verify that CGR is a selected Type for generation during Save. Save is the only process currently available for CGR generation. Once Save is selected for execution the user may specify Derived Output options. For each 3DEXPERIENCE Type being saved the user may select specific file formats to be generated. For CGR generation validate that the “Automatic” option is selected for CGR format for each Type required.

The CGR generated and stored in 3DEXPERIENCE is in MM units regardless of Creo Parametric session units.



Store Viewable image in Viewable Image Type

Previously, JPEG derived outputs are stored into the type of object “Derived Output”. In this release, JPEG derived outputs are now stored into the type of object “Viewable”.



IDF Model Support

This task shows you how to manage IDF files of PROE Models in 3DEXPERIENCE.

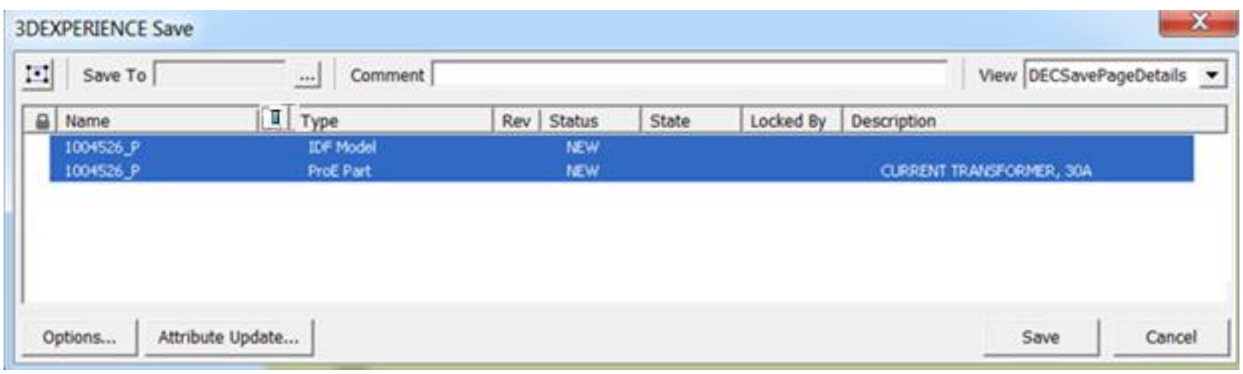
Checkin of IDF Models

Note: IDF Models should be manually (Files->SaveAs) created by the user, before saving them to 3DEXPERIENCE. The feature supports 'IDF Model' checkin for 'ProE Part' and 'ProE Assembly' only. Due to ProE limitation, the integration does not support checkin\checkout of the IDF models of Part Instances and Assembly Instances. Currently the supported IDF Model file formats are .emn and .emp. The IDF files should be present in the same directory as the respective ProE cad file with same name.

To enable checkin of the IDF Model, the integration user needs to modify the value of variable 'enableIDFfeature' equal to 1, by default it is set to 0 in <3DEXPERIENCE PROE Integration Install>\bin\settings.ini file. The modified setting should look as follows.

enableIDFfeature = 1 ; Specify to checkin the IDF Model (0: disable 1: enable)

Once this option is set in settings.ini, if the user clicks 3DEXPERIENCE>Save for a ProE Part/ ProE Assembly, then the integration searches for the IDF Model with same name in the same directory as the ProE Part/Assembly, If the IDF Model (.emn /.emp files) of the in-session Pro/E Model is new or modified, then the integration would show it in the Save dialog. User should click the save button to checkin the 'IDF Model'.



Checkout of IDF Models:

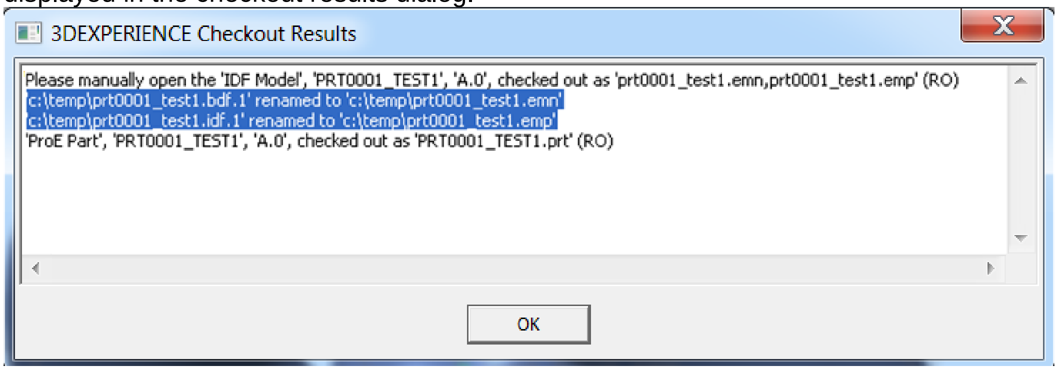
User can search for existing (already checked in) IDF Models in 3DEXPERIENCE Open dialog.

On checkout of an IDF object, the integration will download the IDF file and its child 'ProE Part' or 'ProE Assembly'; at the end of checkout process, the integration will ask the user to manually import the IDF file. The integration will not automatically import IDF files in session, after the checkout operation; the integration user needs to do this step manually.

Note: A Creo Parametric user can't import the IDF file with deafult (same as PROE Model) name if its PROE model is already present in the working directory.

GCO option 'ProEIDFFileTypeChangeMapping' is provided to facilitate ECAD-MCAD IDF files interchange. i.e. ProEIDFFileTypeChangeMapping is set with default values of ECAD IDF and corresponding MCAD IDF file extensions
'bdf|emn
idf|emp"

During checkout, if IDF Model has ECAD IDF files (.bdf, .idf) attached then, as per the GCO mapping the files are renamed after checkout as MCAD IDF files(.emn, .emp) and the rename information will be displayed in the checkout results dialog.



Opening Particular Iteration of a Design

This task shows you how to open an earlier or later iteration of a design from 3DEXPERIENCE in Creo Parametric. This feature is used to retrieve an earlier design or a later iteration of a design if it exists to restart development.

Multiple iterations of the design must exist in 3DEXPERIENCE.

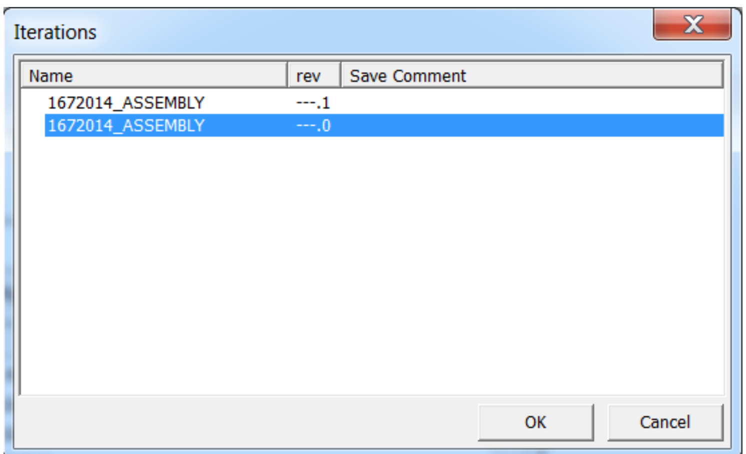
1. Click **3DEXPERIENCE** > **Open** from Creo Parametric tool bar.

   The **3DEXPERIENCE Open** dialog box opens.

2. Search for the design using **Search** option in **3DEXPERIENCE Open** dialog box. See *Searching Designs*.

   The search results are listed in **3DEXPERIENCE Open** dialog box.

3. Right-click a design and click **Iterations** to open the **Iterations** dialog box.



Note: The above dialog is table driven. The displayed dialog can be customized using the table specified in the IEF-DefaultConfigTables GCO variable.

4. Select a single iteration to be opened.

   The selected iteration of the design is opened in Creo Parametric.

Examine Particular Iteration of a Design

This task shows you how to select an earlier or current iteration of selected revision of a design from 3DEXPERIENCE in Creo Parametric. This feature is used to view the expanded structure of an earlier design or a later iteration of a design.

1. Click **3DEXPERIENCE** > **Open** from Creo Parametric tool bar.

   The **3DEXPERIENCE Open** dialog box opens.

2. Search for the design using **Search** option in **3DEXPERIENCE Open** dialog box. See *Searching Designs*.

   The search results are listed in **3DEXPERIENCE Open** dialog box.

3. Right-click a design and select **Examine** to view available iterations as extended sub context menu.

4. Select a single iteration to be examined.

   The selected iteration of the design is opened in Open Examine dialog with the Design Configurations and settings set in Open Options .See *Open Examine*

Opening a Simplified Rep of a Design

This task shows you how to open a simplified rep of a design from 3DEXPERIENCE in Creo Parametric.

The design must correspond to a Creo Parametric assembly.

1. Click **3DEXPERIENCE** > **Open** from Creo Parametric ribbon/tool bar.

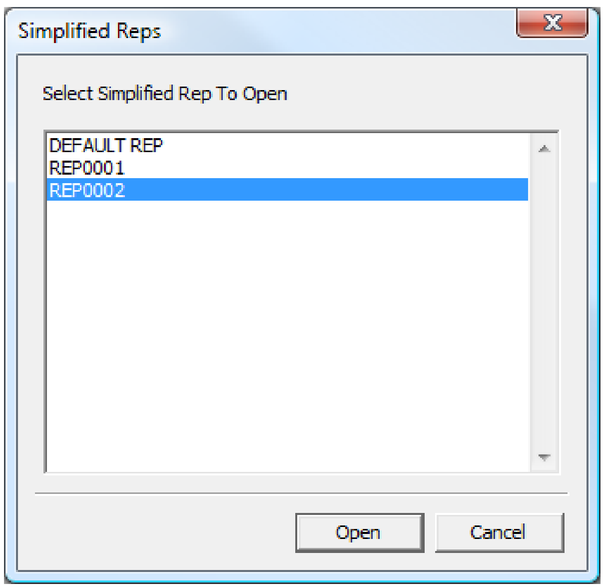
   The **3DEXPERIENCE Open** dialog box opens.

2. Search for the design using **Search** option in **3DEXPERIENCE Open** dialog box. See *Searching Designs*.

   The search results are listed in **3DEXPERIENCE Open** dialog box.

3. Right-click a design and click **Simplified Reps** to open the **Simplified Reps** dialog box.

   Note: This option is only displayed if the design is a ProE assembly.



4. Select a single simplified rep to be opened.

   The selected simplified rep of the assembly design is opened in Creo Parametric. 3DEXPERIENCE Connector for Creo Parametric downloads components necessary to open simplified representation of assembly on checkout, and the excluded components of representation will not be downloaded on checkout. The checkin of Simplified Representation design is now based on ProEAutoLoadMasterRep GCO. The default value of ProEAutoLoadMasterRep GCO option is ‘First Time’, which means that first checkin of Creo Parametric Assembly will loads MASTER representation and checkin complete structure to 3DEXPERIENCE. The subsequent checkin of ProE Assembly representation will checkin components loaded in Creo Parametric session.

Lock

Locking a design in 3DEXPERIENCE reserves the right to modify the design for the user. The following sections describe the concept and working of this feature.

About Locking and Unlocking

Locking a design in 3DEXPERIENCE reserves the right to modify the design for the user. The Lock feature is used to lock a design in 3DEXPERIENCE to ensure that no other user can overwrite the design with their changes. For example, if Part A is locked by User A, then another user, User B cannot modify Part A.

It is recommended to retain the lock for your designs while saving the designs to 3DEXPERIENCE. Leaving a design unlocked in 3DEXPERIENCE allows other users to modify the design and save their changes to 3DEXPERIENCE.

Locking and Unlocking Designs

This task shows you how to lock or unlock designs in 3DEXPERIENCE from Creo Parametric. This is used to lock the designs after you have made changes and to prevent other users from modifying the design.

a. Ensure that you are in the **3DEXPERIENCE Open** dialog box.

- Right-click one or more selected designs in **3DEXPERIENCE Open** dialog box.
- Click **Lock** or **Unlock** depending on the lock status of the selected designs.

  Note: You cannot unlock a design locked by another user. When you try to unlock a design locked by another user, an error message is displayed indicating that you cannot unlock the design and also displays the user name of the person who has locked the design.

  The following icons in the Lock Status ( ) column in **3DEXPERIENCE Open** dialog box show the current lock status of the designs:

  - Indicates that the design is locked.
  - Indicates that the design is unlocked.

The selected designs are locked or unlocked depending on your choice.

b. You can also lock design at the time of checkin to 3DEXPERIENCE using Right Mouse Button -> Lock. Select ProENGINEER design in **3DEXPERIENCE Save** dialog and use Lock contextual menu to lock design. Multiple unlocked designs can be locked at once by using ‘Right Mouse Button->Lock All’.

c. User can lock or unlock design from 3DEXPERIENCE webpage. Use **3DEXPERIENCE Explore in 3DEXPERIENCE** menu to launch 3DEXPERIENCE properties page of active Creo Parametric design, and lock Creo Parametric design from 3DEXPERIENCE properties webpage.

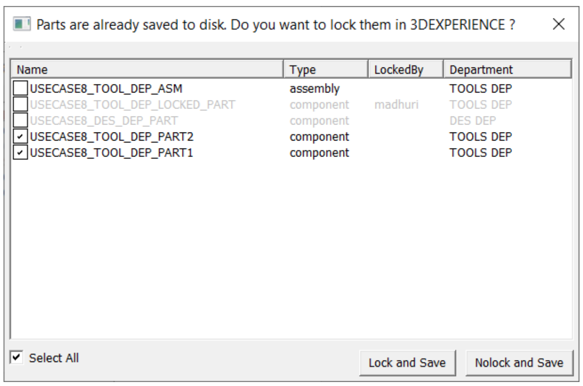
d. Open a design in Creo Parametric session, and click **File>Save** menu of Creo Parametric to lock active design.

Consolidated lock feature

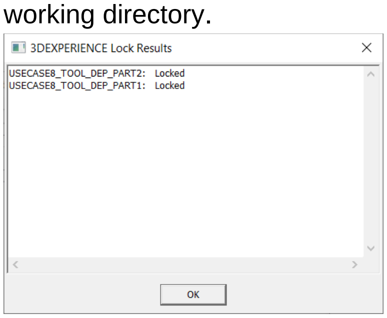
Consolidated lock feature is added to **File>Save** menu of Creo Parametric. Once user clicks on **File->Save** menu all modified object will be listed in a one single dialog. Using consolidated Lock functionality user can save multiple designs with lock or without lock. No lock save option is available to proceed save without locking the designs. Consolidated Lock dialog displays the information of object like Name, Type, LockedBy and Department. Department column will display the collaborative Space of the designs. Based on the Collaborative Space access user will be allowed to lock the designs. Designs locked by another user or from different collaborative Space (user do not have access to design's collaborative Space) are shown in gray color and disabled state. Such objects are not allowed for lock selection.

Checkout an assembly, which belongs to TOOL DEP. It has one part, which belongs to DES DEP.It has one more part which belongs same TOOL DEP but locked by other user.

1. Make some modifications in all parts. Regenerate the assembly and try to do CREO save.



2. One dialog pops up displaying list of the files that user wants to save locally.
3. Parts, which belongs to other department appear grey and are not selectable.
4. Parts, which belongs to same department but locked by other user also appear grey in the dialog and are not selectable.
5. User can sort the parts based on Name, Type, Department and Locked By.
6. User can select other parts to lock.
7. User can make use of "Select All" to select all remaining parts.
8. After making selection among remaining parts user can "Lock and Save" or "Nolock and Save".
9. User clicks on LOCK and SAVE, all the selected parts get locked and made writable in local working directory.



10. If user selects Nolock and Save, then all the selected parts will be made writable but not locked in the server.

Validate Name

Validate Name feature is used to check the naming convention of the newly created Creo Parametric object name.

About Validate Name

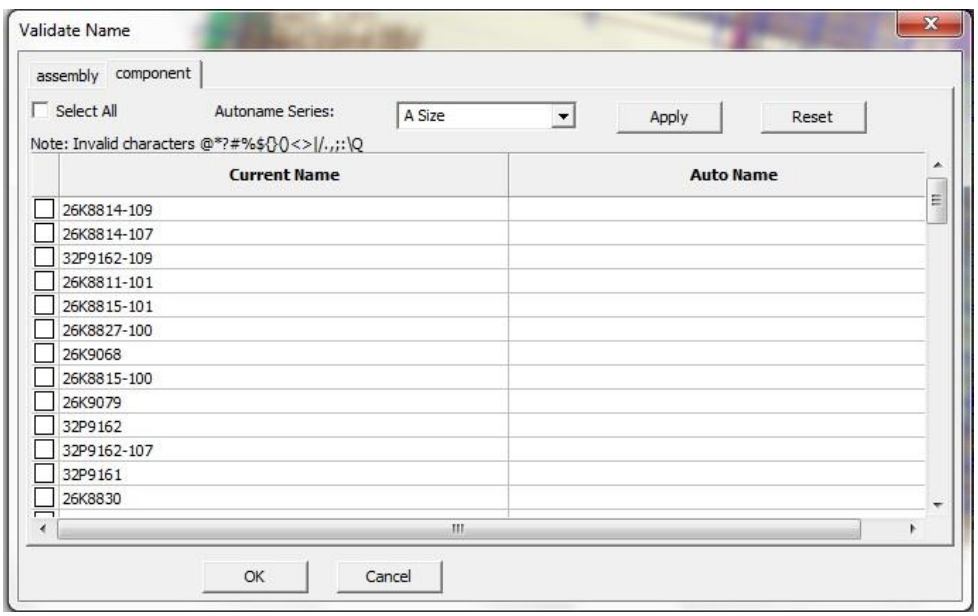
The Validate Name feature will check for name of all the newly created CAD object in session, in case user has used any invalid or unsupported characters as specified in GCO setting ‘MCADInteg-NonSupportedCharacters’ while naming the CAD object, it will be displayed in Red color and Italic font. The feature allows user to rename the CAD object either by using the Autoname feature with the given Autoname series or user can edit object name.

Validate Name Designs

This task shows you how to validate the name of the CAD files in session

- The CAD files should be newly created

1. Create a NEW CAD model and save it on disk
2. Click on “Validate Name” command on the ribbon, once user clicks on command below shown dialog will be displayed to the user:



Validate Name dialog consists of following controls:

It consists of dedicated **tabs** for CreoParametric Object type supported by Integration,

Clicking **Select All** will enable the checkbox in the row of every object. This is useful for applying the autoname from Autoname series to all objects via the Apply button.

Click **Autoname Series** to assign a name from a predefined series (these series need to be created by Integration Administrator)

Click **Apply** to apply the selected autoname series name

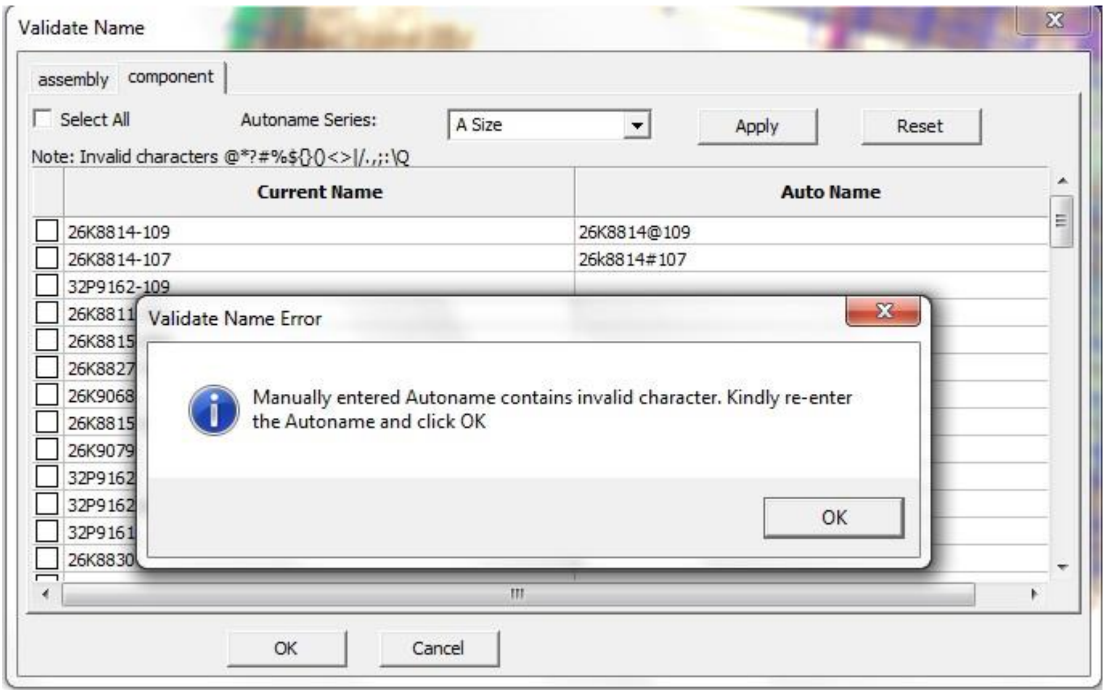
Click **Reset** remove all the changes done and remove the entries from **Auto Name** column

Click **OK** to confirm the changes done and rename the files on the disk.

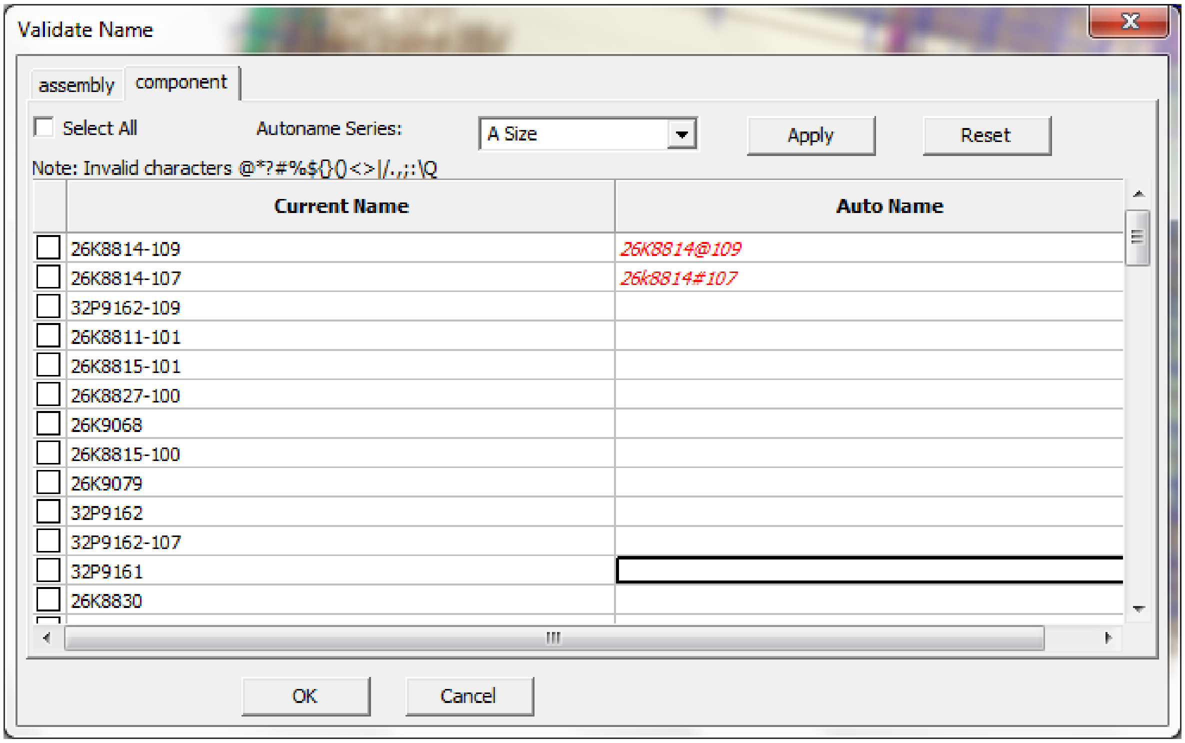
Click **Cancel** to return to Creo Parametric tool without making any changes

3. If the current name consists of any invalid or unsupported characters, then the CAD object name will be displayed in Italic font and Red color.

4. If you enter an invalid name for Creo design in the Validate Name Dialog then an error is displayed.

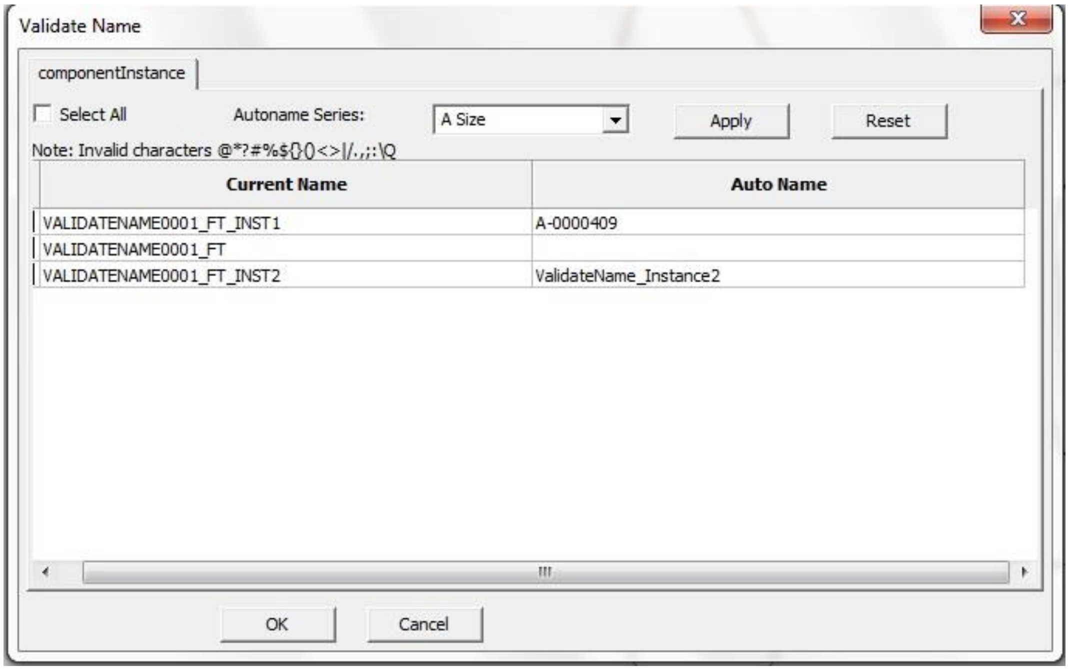


5. The new validated entries under the Autoname Column are shown as red italic font.



Validate Name for Family Table instances

The Validate Name supports the validation and renames the newly created Family Table instances. The family table instances new names get updated in Family Table.



Baselines

Baselines are used to capture a design or design structure which the designer feels can be used as a benchmark before proceeding to make more changes to the design. The following sections describe the concept of baselines, steps for creating baselines, and opening a design based on the baseline.

About Baselines

Baselines are used to capture a design or design structure which the designer feels can be used as a benchmark before proceeding to make more changes to the design. Using 3DEXPERIENCE Connector for Creo Parametric, baselines can be created to capture the structure of a design active in Creo Parametric that exists in the 3DEXPERIENCE database. The baselined structures can be opened at any stage of the design in Creo Parametric from **3DEXPERIENCE Open** dialog box.

When a design that is opened from 3DEXPERIENCE is baselined, the baseline is created for the as stored structure of the design in 3DEXPERIENCE database. For example, if you create a baseline for a design that was opened from 3DEXPERIENCE and modified, then the baseline does not capture the modifications because they do not exist in the 3DEXPERIENCE database. The baseline is created for the stored design in 3DEXPERIENCE.

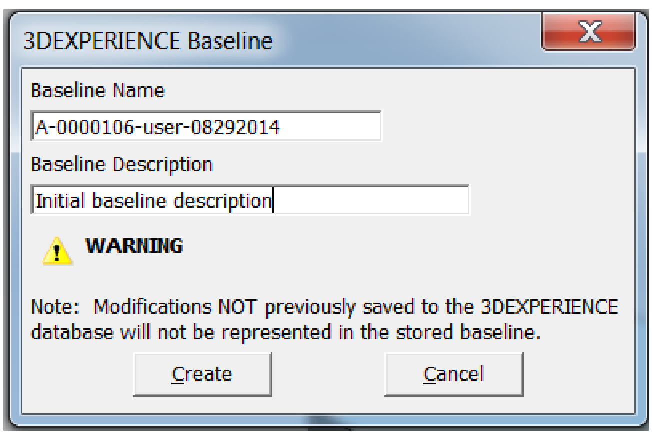
Baselining Designs

This task shows you how to create a baseline for designs that exist in 3DEXPERIENCE.

- The design must exist in 3DEXPERIENCE.
- The design must be the active design in Creo Parametric.

1. Click **3DEXPERIENCE** > **Baseline** from Creo Parametric ribbon/toolbar.

   The **3DEXPERIENCE Baseline** dialog box opens.



   Enter the following parameters for the baseline:

   • **Baseline Name**. Enter a name for the baseline. By default, the name field specifies the design name with date stamp in <design name>-<mmddyy> format.

   • **Baseline Description**. Enter a description for the baseline.

2. Click **OK**.

A warning message is displayed if a baseline exists in 3DEXPERIENCE with the same name. Click **Yes** to overwrite the existing baseline or click **No** to enter new parameters for the baseline.

Note: Baselines are applied only to the design node active in Creo Parametric and the structure beneath the node does not inherit the baseline.

A success message is displayed once the baseline is created. A baseline is created for the design active in Creo Parametric with the specified parameters.

Opening Baselined Designs

This task shows you how to open a design from 3DEXPERIENCE in Creo Parametric using an existing baseline.

Baseline must exist for the design.

1. Click **3DEXPERIENCE** > **Open** in Creo Parametric ribbon/toolbar.

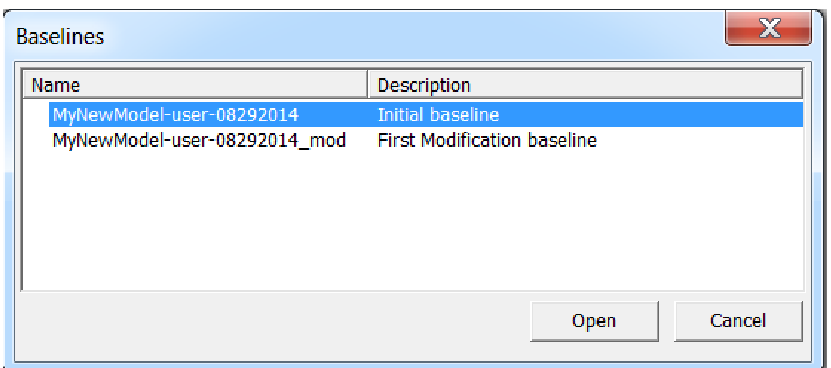
   The **3DEXPERIENCE Open** dialog box opens.

2. Search for the baselined design in the **3DEXPERIENCE Open** dialog box. See *Searching Designs*.

   The results are listed in the **3DEXPERIENCE Open** dialog box.

3. Right-click the name of baselined design and click **Baselines** ><Name of the Baseline> .

   If multiple baselines exist for the design, then all the baseline names are listed.



Note: The above dialog is table driven. The displayed dialog can be customized using the table specified in the IEF-DefaultConfigTables GCO variable.

Note: Baselines are applied only to the node selected during creation and the structure beneath the node does not inherit the baseline.

The design opens in Creo Parametric retaining the design's characteristics at the time the baseline was created.

Open Examine

About Open Examine

3DEXPERIENCE Open Examine is an extension to 3DEXPERIENCE Open dialog .It is used to examine the expanded structure of selected iteration for the design selected in Open dialog.

This dialog offers an option to provide iteration information of the dependent objects in structure of selected Design. The initial structure displayed provides the details that are governed by various selections applied from the 'Open Options' in Open dialog. It allows you to alter these options and view the effect of configuration change on structure of the selected design.

This feature provides the facility to observe the comparison between the selected 'Design Configuration' iteration with the 'Latest Iteration' available of latest revision for all the objects in expanded structure.

Examining Designs

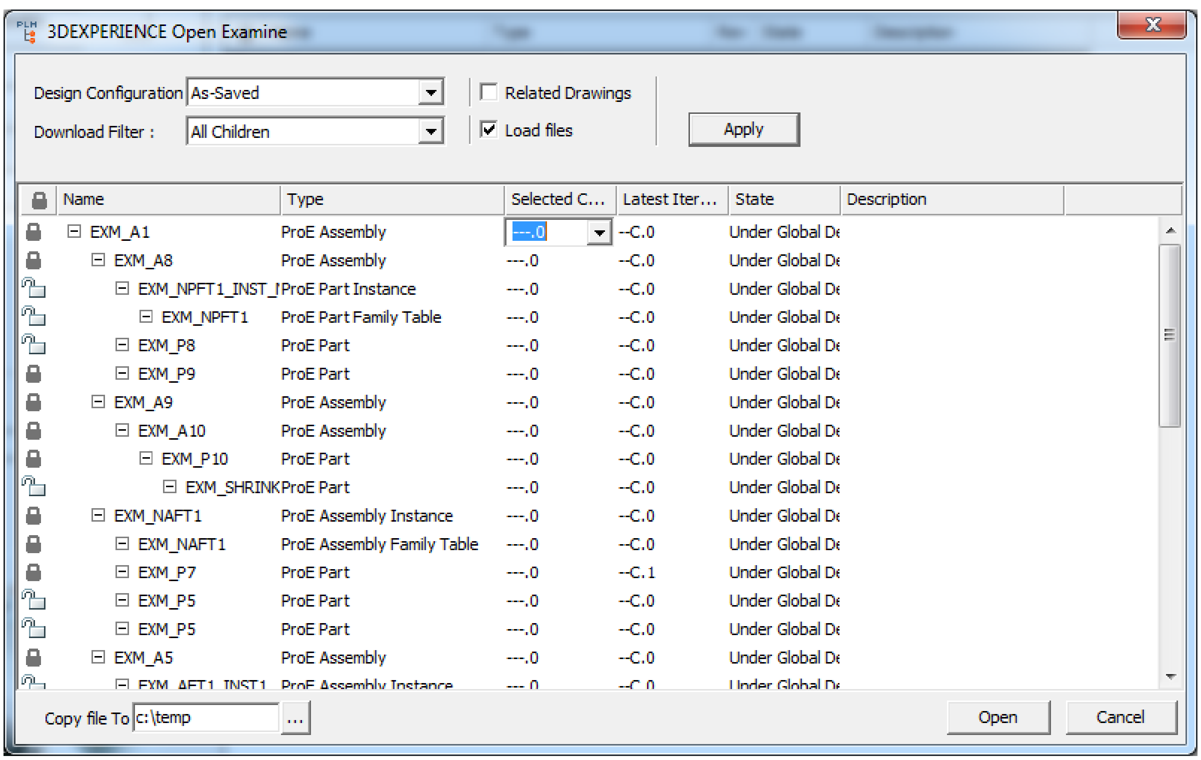
1. Click **3DEXPERIENCE** > **Open** in Creo Parametric ribbon/toolbar.

   The **3DEXPERIENCE Open** dialog opens.

2. Search for a design in the **3DEXPERIENCE Open** dialog box. See *Searching Designs*.

   The results are listed in the **3DEXPERIENCE Open** dialog box.

3. Right-click the name of a design and click **Examine**. All the iterations are listed as extended context sub menu. The **3DEXPERIENCE Open Examine** dialog opens.



4. Choose options to be executed to view or open a design from 3DEXPERIENCE:

   - **Related Drawings**. Downloads all associated designs to the selected model. The files are copied to the user's designated local working directory.

   - **Design Configuration**. Select an option from the drop down list to display specific iterations or revisions of designs in **3DEXPERIENCE Open** dialog page.

   - **Load files.** Loads files to Creo session.

   - **Download Filter**: There are three options available in this dropdown.

     - **All Children.** Select this checkbox to checkout the child objects of selected objects. The initial value is populated based on the user's MCADInteg-SelectChildItems Local Configuration setting (also available via 'Select Children if Parent is Selected' option in Designer/Preferences). Leaving this checkbox blank will improve performance if, for example, the user desires to only checkout the top level of an assembly or the user desires to checkout a family table template without its family table members.

     - **Required Children.** This option allows user to only download direct children, object connected by assemblyComponent and drawing relationships, on checkout. On checkout with this option, the user will not get models connected by external-copy geometry relationship

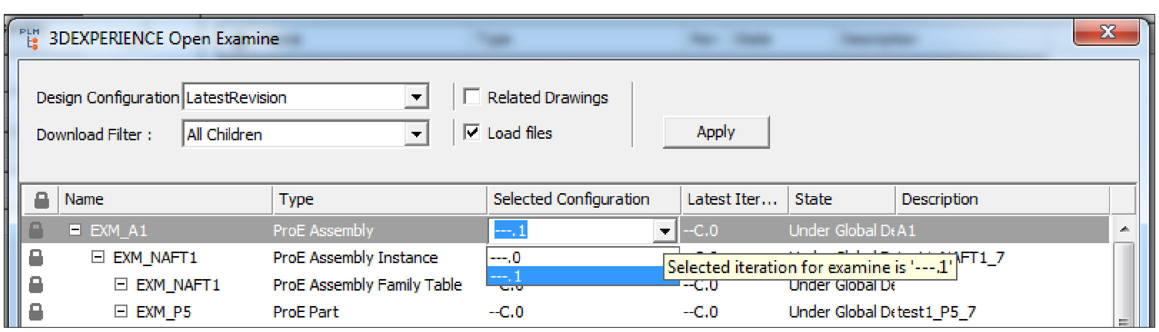
     - **Selected items only.** This option allows user to only download the items selected in the search results pane.

   Note: Configuration settings made in the Examine Dialog are not remembered in the Open dialog.

5. Clicking on **Apply** button will confirm the changes made to above options update the stucture in the Open Examine dialog.

6. Choose Iterations to view or open a respective design change from 3DEXPERIENCE:

   You can select other iteration of designated revision for a selected design object.



   e.g. if you have selected --- revision of the design to examine from 3DEXPERIENCE Open dialog then you can change the view for iterations associated with revision '---', like ---.0,---.1,---.2,so on.

7. The ‘Copy **file to’** option in Open Examine dialog. The designs opened from 3DEXPERIENCE are downloaded to the directory specified. By default, it will download to directory set in 3DEXPERIENCE Open dialog. Click to browse and select another directory as the checkout directory.

8. Clicking on **Cancel** button, the control will return back to 3DEXPERIENCE Open dialog and all the configurations or option changes from the 3DEXPERIENCE Open Examine dialog wll not be remembered in 3DEXPERIENCE Open dialog. .

9. Clicking on **Open** button, to open the design in Creo Parametric.

Note: The above dialog is table driven. The displayed dialog can be customized using the table specified in the IEF-DefaultConfigTables GCO variable.

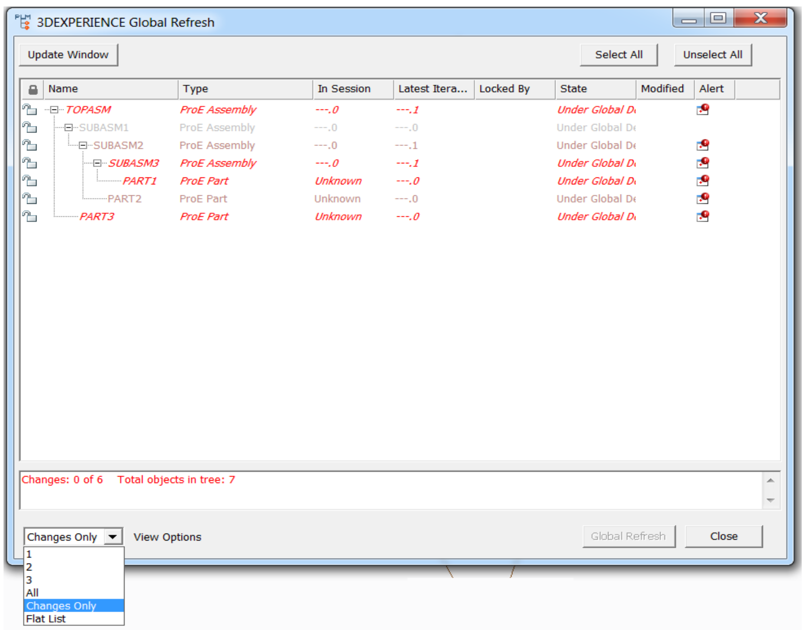
Global Refresh

Global Refresh allows you to see a merged view of what resides in the CAD Structure with what resides in the 3DEXPERIENCE structure.

Before performing Global Refresh the directory in which CAD data to be refreshed is present should be set as current working directory.

To Start Global Refresh: Click **3DEXPERIENCE > Global Refresh** from Creo Parametric menu (Ribbon).

- The **Global Refresh** dialog box will appear displaying the merged tree structure of the Current CAD Session and the structure of the data in 3DEXPERIENCE. The information listed includes the lock status, name, type, current iteration, locked by, state, revision, and latest iteration.
- Clicking **"Update Window"** will return updated information regarding the current active design.
- Clicking **"Unselect All"** will unselect any components that are selected.
- Clicking **"Expand All"** will expand the complete tree to display all the components.
- View Filters: The user is provided several different options on how they wish to view their structure in the global refresh dialog. View filters are available in lower left corner.



Various view filter options are

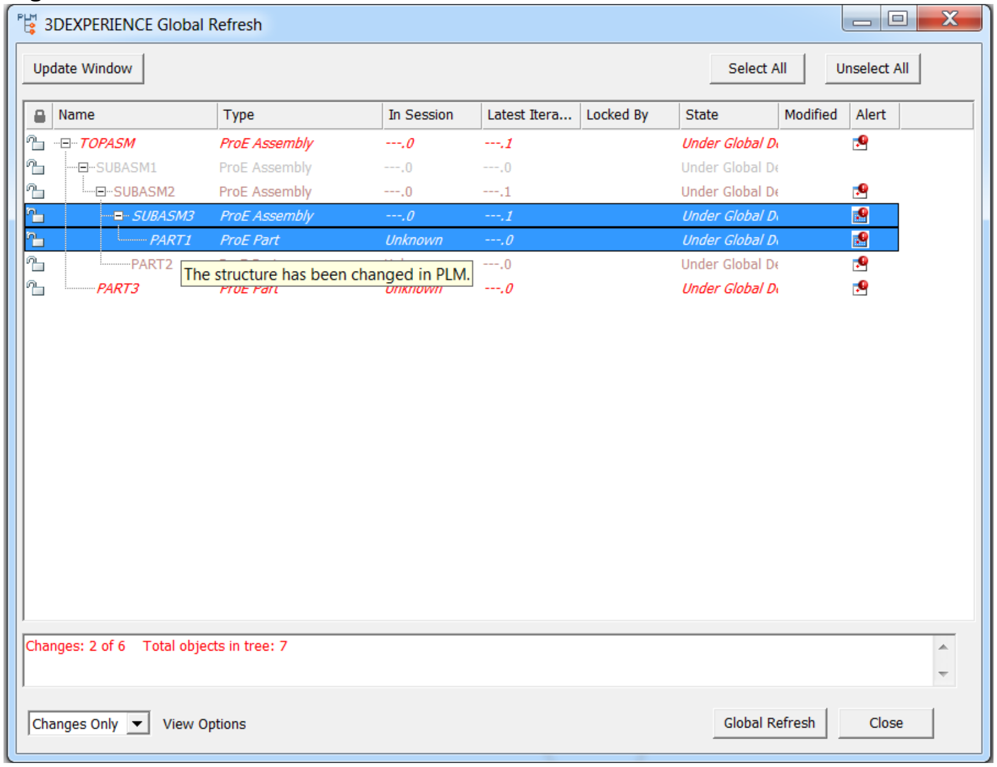
- 1, 2, …, All
  - This is the number of levels to expand in the CAD/PLM tree. The pull down combo box will allow the user to select from 1 level to display to have all levels of the tree displayed.
- Changes Only
  - The display will grey out those items that are not eligible to be updated and make those eligible to be updated Bold/Italicized red. This allows the user to easily see which items need to be updated.

- Flat List
  - This will display in a single list only those items that are eligible to be updated

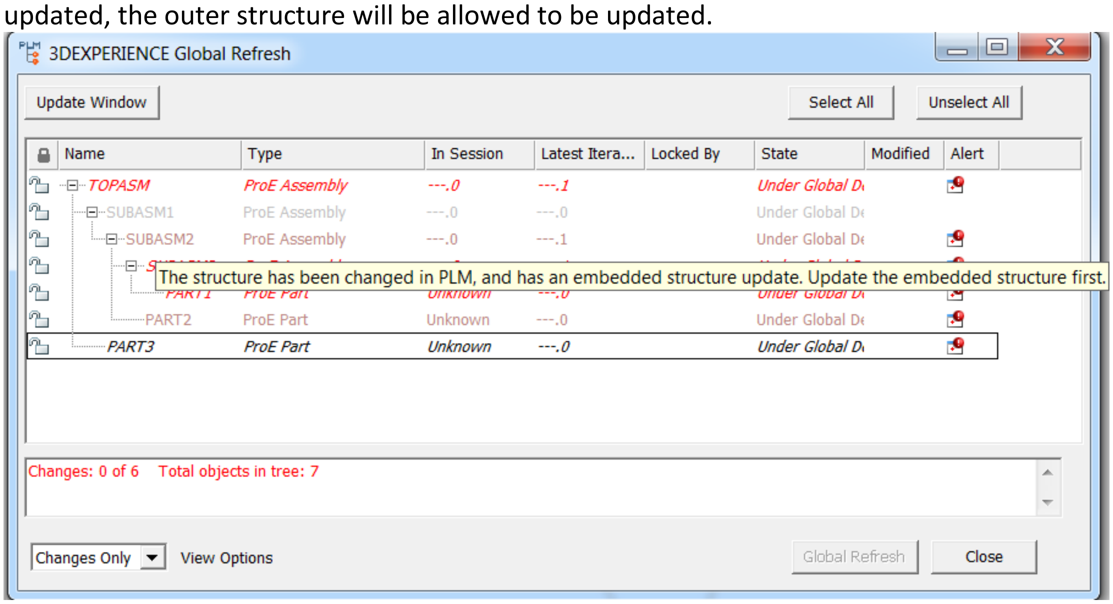
- Clicking **"Global Refresh"**
  - The selected design(s) will be overwritten in session with the latest iteration from 3DEXPERIENCE.
  - If the **"Global Refresh"** button is not selectable it means one of the following:
    - There are no items selected.
    - The dialog has lost focus and regained focus. This means that the **"Update Window"** button must be pressed to refresh the contents being displayed.
- Clicking **OK** will close the **Global Refresh** dialog box.
- The user can work in Creo Parametric or 3DEXPERIENCE without closing this dialog; he\she can put this dialog on the side and work on his/her tasks.
- In the dialog the different display (color/italics..) mean: (*also available on tool tip of row*)
  a. Black, non-bold, state the component is up to date.
  b. Red italics state that the component is able to be refreshed.
     - A later iteration, or revision, of a component exists in 3DEXPERIENCE.
     - The component has been renamed in 3DEXPERIENCE
  c. Blue bold state the component exists just in the CAD structure and not in the 3DEXPERIENCE structure.

- Structure Updates: With the help of the new "Global Refresh" Structure Updates, the user will be able to easily traverse the differences between the CAD/PLM structures. The behavior for updating PLM Structural Changes will be the same as for PLM Revision updates except for 2 points:

1. The Parent Assembly and the child Component will only be allowed to be selected together.

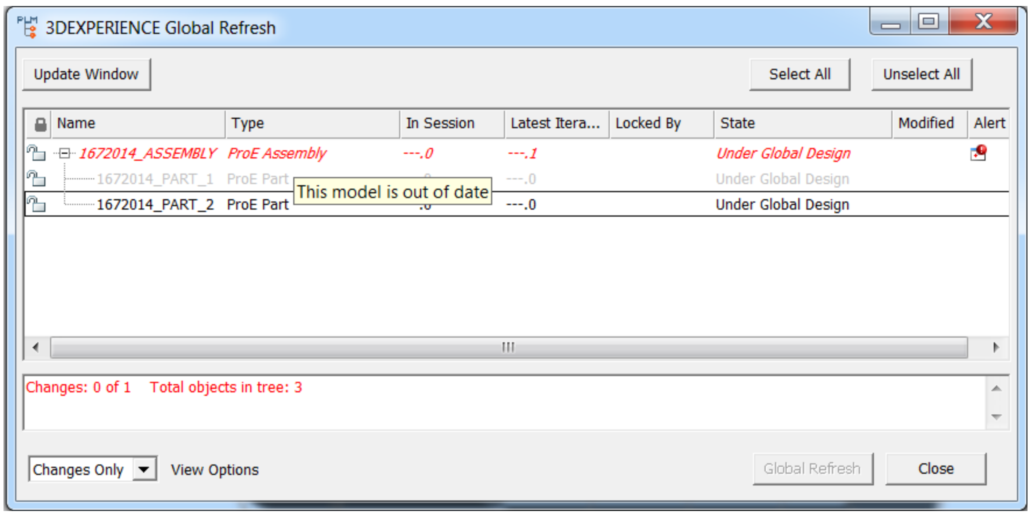


2. If there is PLM Structural update that exists inside of another PLM Structural Update only the inner Structure will be allowed to be updated. Once the inner structure has been updated, the outer structure will be allowed to be updated.

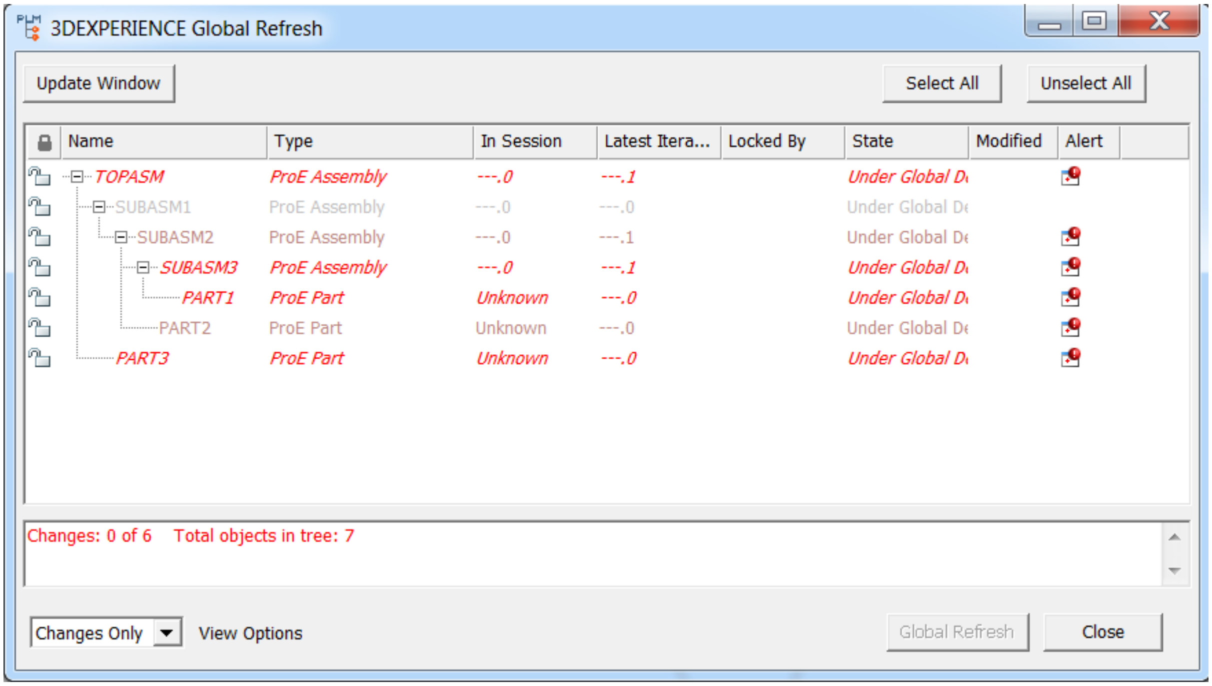


Note:

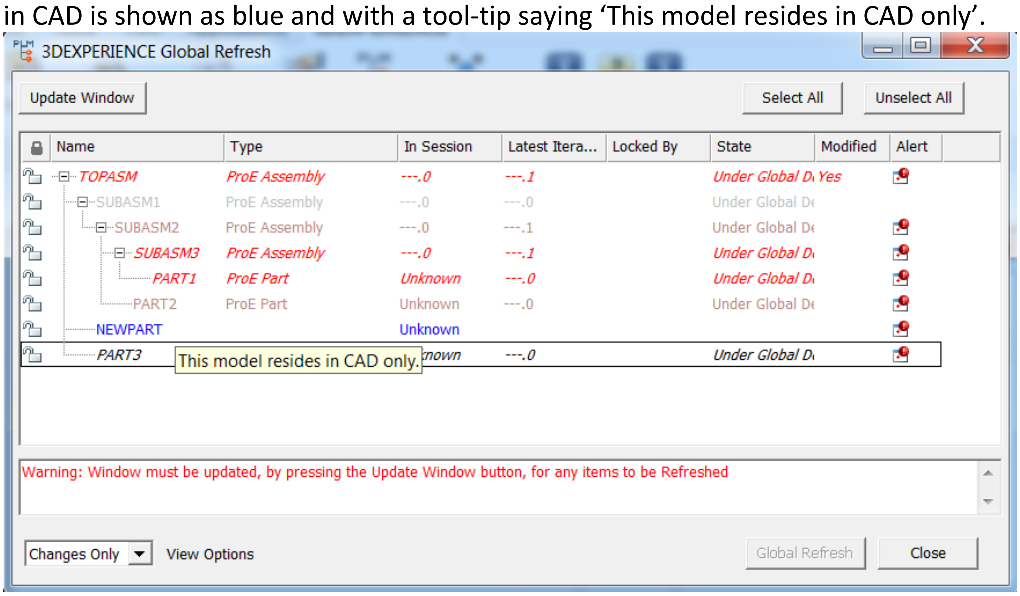
- This feature will not merge the design modifications made by the user with the latest iteration of the design downloaded from 3DEXPERIENCE; it can cause potential work loss. If the model displayed in the dialog was modified in the Creo Parametric session, the integration will warn the user by putting a “Yes” in the Modified column.
- The integration will display all the warning messages at the bottom of the dialog.
- Only items that are allowed to be refreshed will be selectable.
- An Assembly with an Out Of Date component



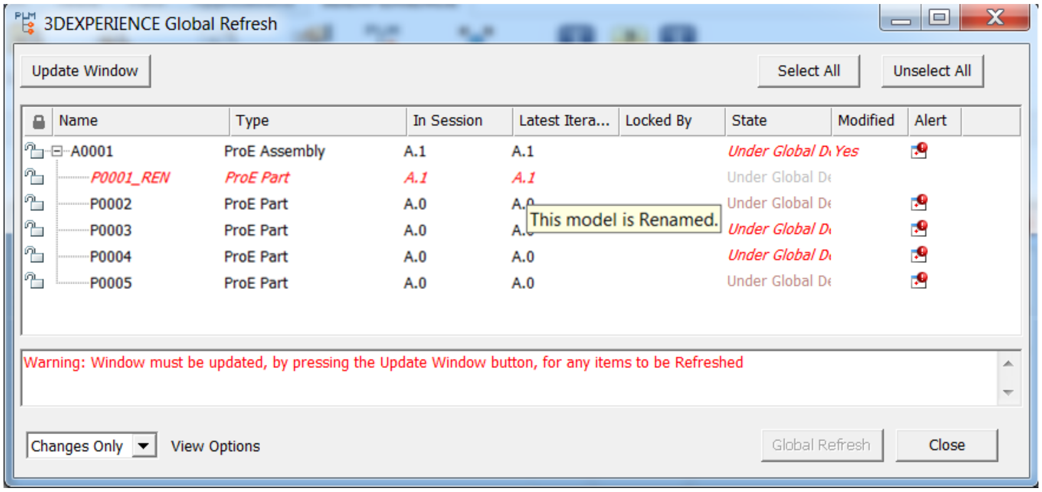
- An Assembly with a component added in 3DEXPERIENCE (structure change). In below screenshot one can see that Part1, Part2 and Part3 are added to the various assemblies in 3DEXPERIENCE. The Unknown status here means that these parts are available in 3DEXPERIENCE but not in the assembly in session.



- An Assembly with a component (NewPart) added to just the CAD structure. The newly added part in CAD is shown as blue and with a tool-tip saying 'This model resides in CAD only'.



- An Assembly where a component (P0001) has been renamed in 3DEXPERIENCE to P0001_REN


3DEXPERIENCE Global Refresh
Update Window
Select All
Unselect All
Name
Type
In Session
Latest Itera...
Locked By
State
Modified
Alert
A0001
ProE Assembly
A.1
A.1
Yes
P0001_REN
ProE Part
P0002
P0003
P0004
P0005
A.0
This model is Renamed.
Warning: Window must be updated, by pressing the Update Window button, for any items to be Refreshed
Changes Only
View Options
Global Refresh
Close

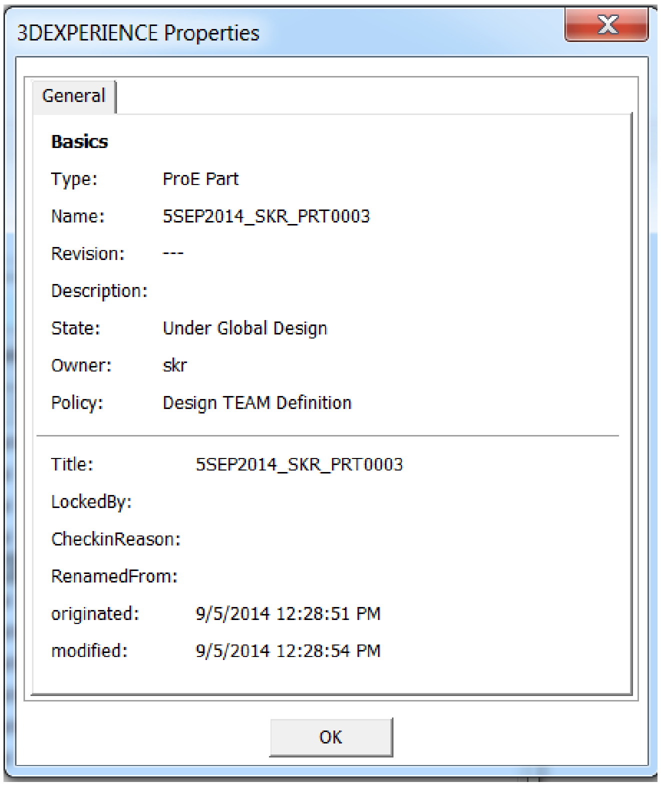
Viewing Properties

This task shows you how to view properties of a design that exists in 3DEXPERIENCE which is active in Creo Parametric.

The design must be active in the ProE session and exist in 3DEXPERIENCE database.

1. Click **3DEXPERIENCE** > **Properties…** from Creo Parametric ribbon/toolbar.

   The **3DEXPERIENCE Properties** dialog box opens.



   For the design, the **3DEXPERIENCE Properties** dialog box lists the type, name, revision, description, state, owner, policy, and vault. It also lists,

   • **Title**. The title entered for the design.

   • **LockedBy**. User name of the person who has locked the design.

   • **CheckinReason**. Comments entered when the design is saved to 3DEXPERIENCE.

   • **RenamedFrom**. Last used name of the design if the design has been renamed.

2. Click **OK**.

   The **3DEXPERIENCE Properties** dialog box closes.

The **3DEXPERIENCE Properties** dialog box lists details of the design in 3DEXPERIENCE.

Using Workspaces

This feature is used to manage designs using Integration Exchange Framework Client. The following sections describe the concept and methods of working with Integration Exchange Framework Client.

About Workspaces

A workspace is a collection of folders that contain objects, documents and other information that you use while working with 3DEXPERIENCE Connector for Creo Parametric.

The **Workspace Manager** feature of 3DEXPERIENCE Connector for Creo Parametric is used to view and manage files stored in the 3DEXPERIENCE database using 3DEXPERIENCE Integration Exchange Framework client. 3DEXPERIENCE Integration Exchange Framework Client must be installed for this feature to function.

Managing Designs Using Integration Exchange Framework Client

This task shows you how to manage designs using Integration Exchange Framework client.

1. Click **3DEXPERIENCE** > **Manage Workspaces** from the Creo Parametric ribbon/toolbar.

   Three options for which workspace to be managed are given:
   Local… Lists file location of the active model on the local disk.

   3DEXPERIENCE... Lists file location of the active model in the 3DEXPERIENCE database. If the user has no active models, it will display a list of all 3DEXPERIENCE workspaces accessible to the user.

   Clear Active Model… It removes active model and its children from PROE session.

2. You can use local and 3DEXPERIENCE workspace folders for managing the designs.

   For more details on working with 3DEXPERIENCE Integration Exchange Framework client, see Working with 3DEXPERIENCE Integration Exchange Framework Client section the *X-CAD Design User Guide*.

Viewing Design Details in X-CAD Design

This task shows you how to view the CAD Portal view of a design in X-CAD Design from Creo Parametric.

The **3DEXPERIENCE** > **Explore in 3DEXPERIENCE** menu is available from Creo Parametric or upon right-click in **3DEXPERIENCE Open, 3DEXPERIENCE Insert**, or **3DEXPERIENCE Save** dialog boxes.

The design must exist in 3DEXPERIENCE database.

1. Click **3DEXPERIENCE** > **Explore in 3DEXPERIENCE…** from Creo Parametric ribbon/toolbar if the design is active in Creo Parametric.

2. Search for the design in **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box.

    - For details on searching for designs in 3DEXPERIENCE from Creo Parametric, see *Searching Designs*.

    Note: Only one design must be selected in **3DEXPERIENCE Open** or **3DEXPERIENCE Save** dialog box.

    - Right-click the design and select **Explore in 3DEXPERIENCE**

3. Access to design details is also available from the 3DEXPERIENCE Save dialog box.

    - Right-click the design and select **Explore in 3DEXPERIENCE**

The CAD Portal View page of the design opens in the browser.

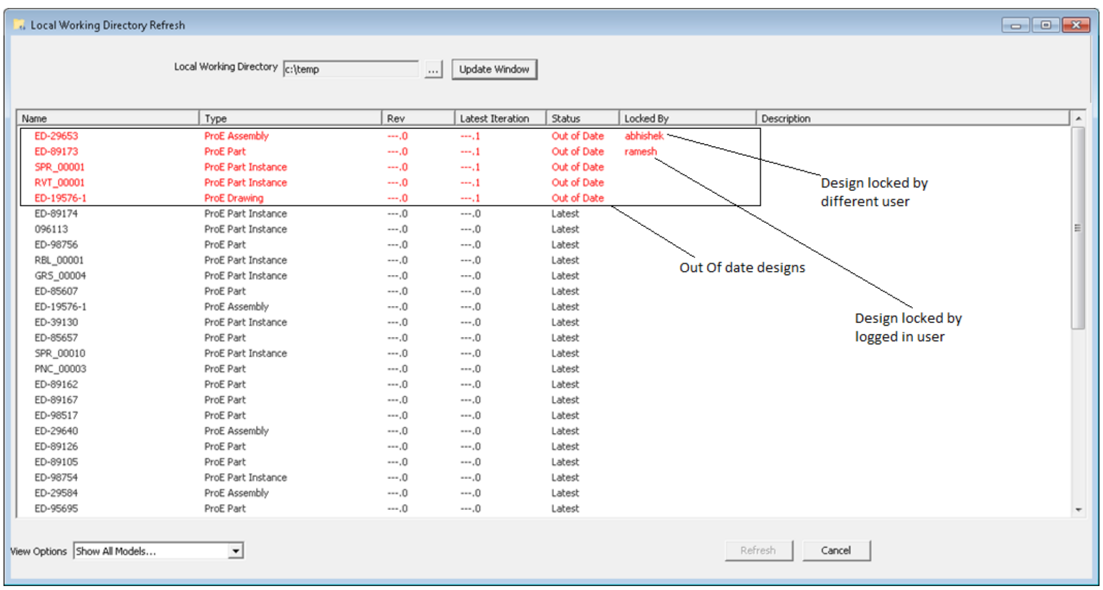
Refreshing Designs from Local Working Directory

The Local Working Directory of a Creo Designer contains multiple assemblies/drawings with the structures containing several hundreds of objects. Some of these assemblies are independent of each other. This feature enables user to know the status of the designs in the 'Local Working Directory' and refresh the models without bringing them in to Creo session.

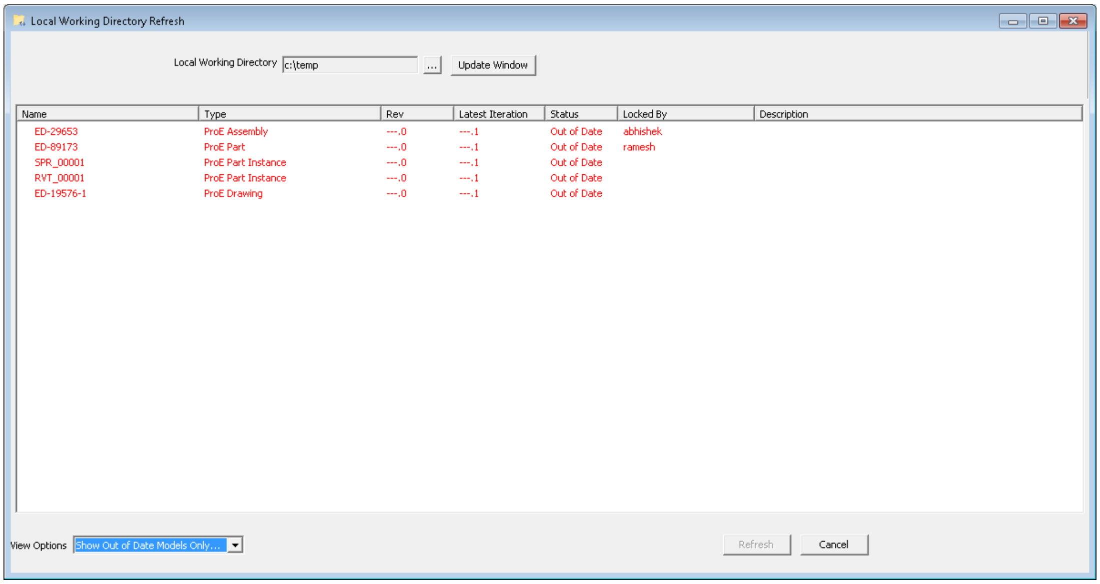
1. To use this feature user has to click on the **3DEXPERIENCE>Local Working Directory Refresh'** menu. The local working directory refresh dialog is as shown below

2. It shows 'Local Working Directory' path, user can change the working directory by clicking on the ellipsis and the designs in that directory will be displayed in the dialog.

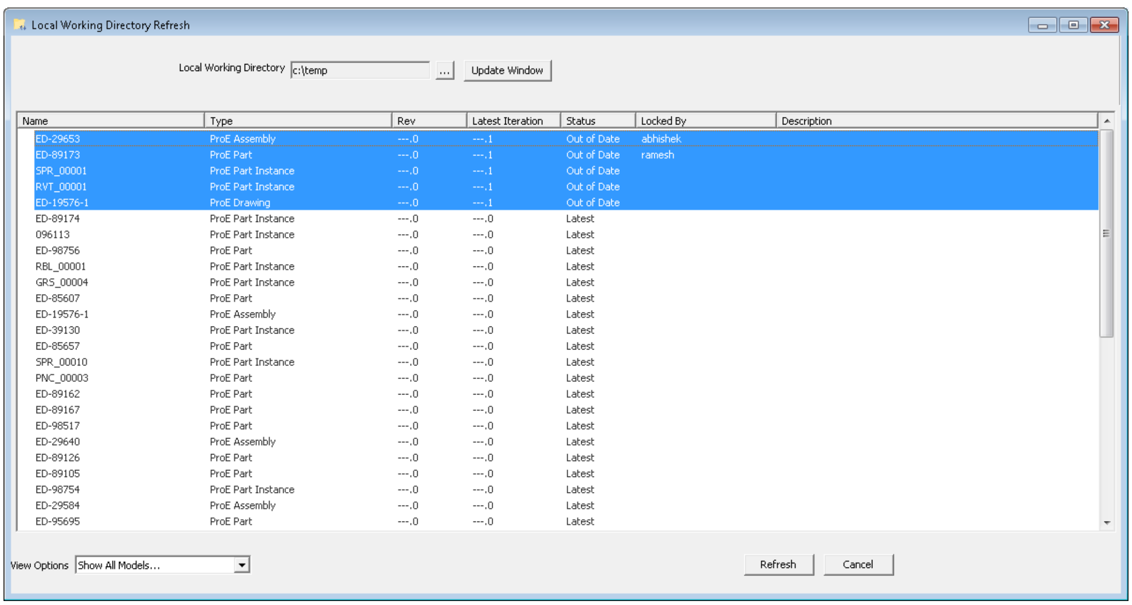
3. 'Update Window' button is used to refresh the dialog content. Dialog displays 'Name', 'Type', 'Rev', 'Latest Iteration', 'Locked By' and 'Description' attribute value.



4. 'View Options' filter is available at the bottom, 'Show All Models…' view option displays all designs as shown above, whereas View Option when set as 'Show Out Of Date Models Only…' only Out Of Date designs will be displayed as shown below.

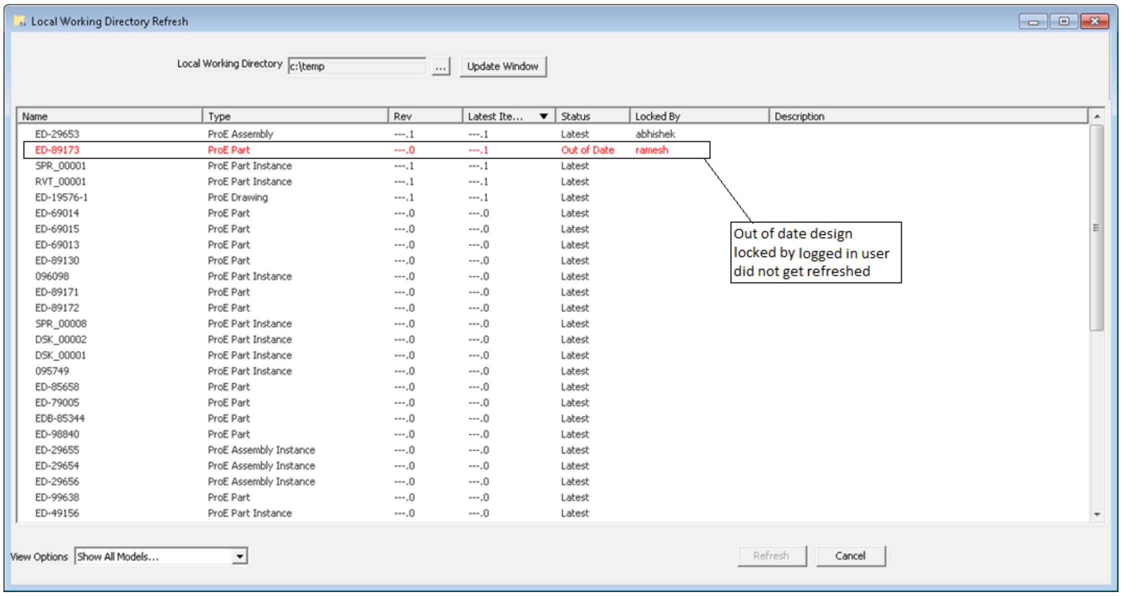


5. To refresh the designs, select the out of date designs and click the Refresh button.



6. Once the Refresh button is clicked, all the selected designs are refreshed but the designs locked by current logged in user are not refreshed assuming user has locked them in order to work on them

7. The result of Refresh operation can be seen below



8. The designs for which the Type Name Revision information is not available on disk are recognized as 'Unknown' and can't be selected for Refresh.

Note: Local Working Directory Refresh feature processes the CAD files present in the working directory selected, since the instances share a common file, the dialog only displays generic instance.

Disconnect From 3DEXPERIENCE

This task shows you how to disconnect from 3DEXPERIENCE from Creo Parametric.

Click **3DEXPERIENCE** > **Logout…**

The connection to 3DEXPERIENCE is terminated.

Design Export

This task helps users to package and export designs from their 3DEXPERIENCE.

Design Export is a new capability provided with R2016x release. Users can make use of this functionality to package and export the CAD data to suppliers etc. Essentially if a designer has to send some CAD models outside of 3DEXPERIENCE then this functionality will be highly useful.

Once user clicks on the 3DEXPERIENCE > Design Export menu in Creo Parametric, a design export dialog is shown to the user. User need to search for the models from 3DEXPERIENCE which needs to be exported. User can select multiple models for export. The selected models will be exported to the unique sub-directory within default checkout directory. This unique directory which has naming convention as *<model name>_<model type>_<timestamp (yy-mm-dd-hr-min-sec) >* i.e. for *example <DE_Test1_ProE>_< ProE Part>_< 20150406143911>.* All the contents of this unique directory are zipped in a zip archive with same name as that of the directory. After zip is created, integration will delete the folder created in default directory.

The following sections such as, searching for designs, exporting objects, describe the function available in 3DEXPERIENCE > design export dialog box.

Searching Designs

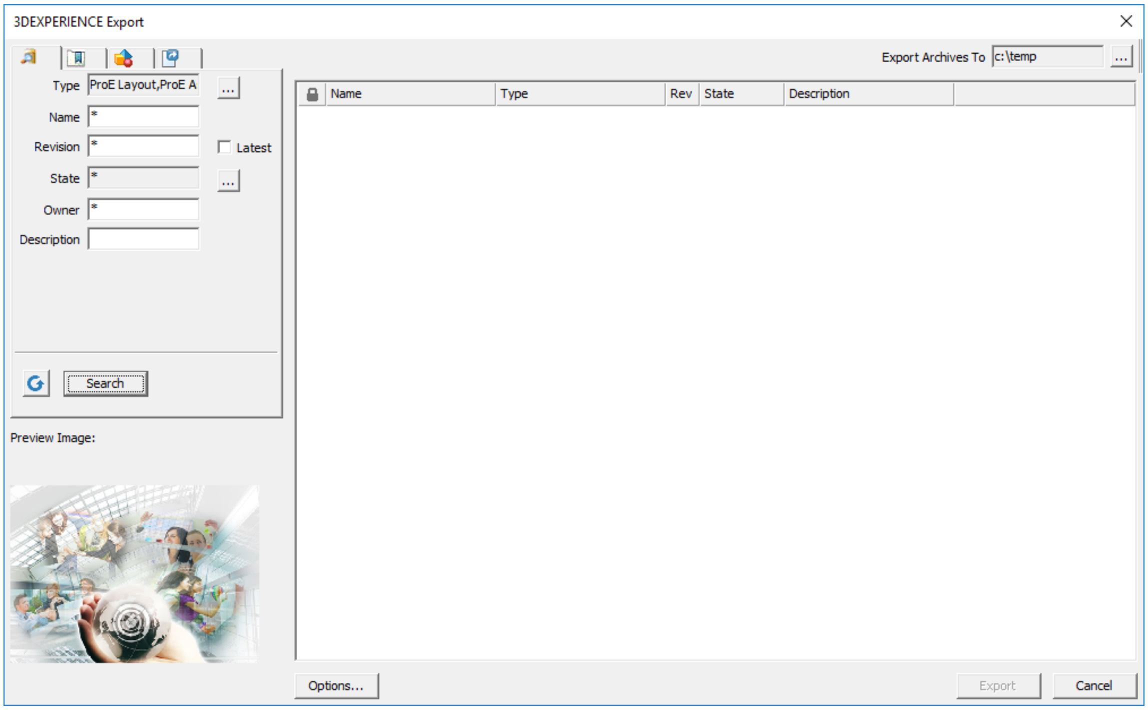
You can search for designs that exist in 3DEXPERIENCE vault from **3DEXPERIENCE > Design Export** dialog box. The following sections describe the steps used to search for designs

Searching Designs

This task shows you how to search for designs in 3DEXPERIENCE database from **3DEXPERIENCE Design export** dialog box.

1. Click **3DEXPERIENCE** > **Design Export** from Creo Parametric toolbar.

   The Design export Dialog box opens.



There are 4 different methods of searching for designs in the 3DEXPERIENCE database:

- Searching via webform driven search parameters.
- Searching user's Workspaces.
- Searching user's Collections.
- Searching user's recently accessed designs.

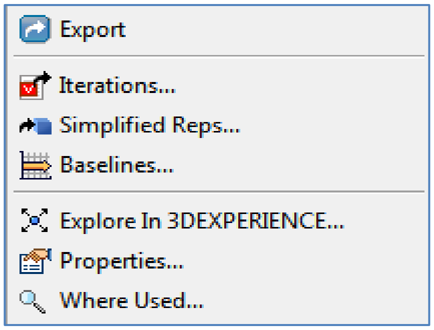
For Searching Designs Using Webform Driven Search Parameters, please refer "Searching Designs".

Search Results

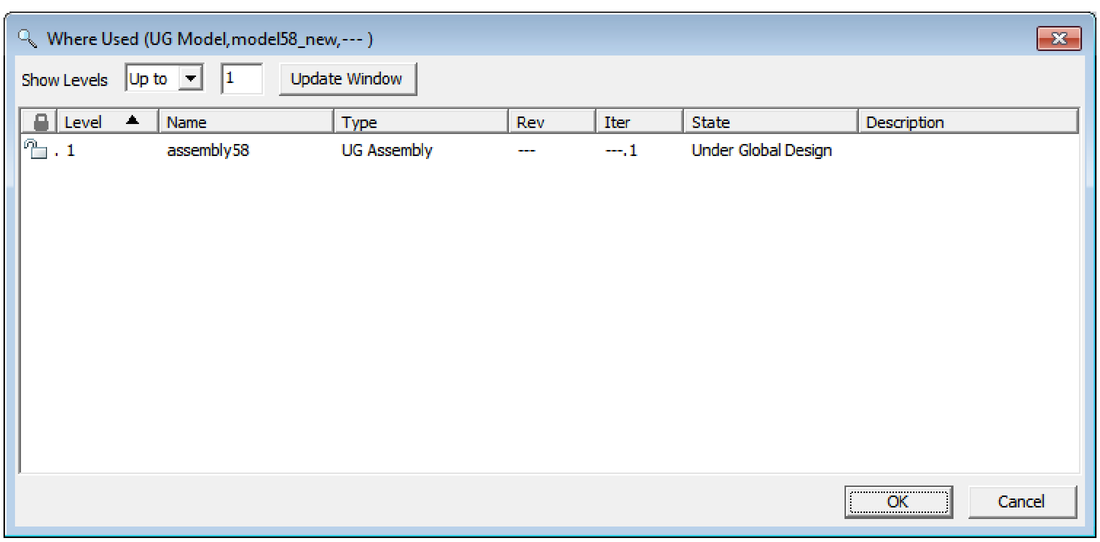
Regardless of the search method used, the search results are displayed in **3DEXPERIENCE Design Export** dialog box. For each design, the **3DEXPERIENCE Design Export** dialog box lists,

-  **Lock Status.** Whether or not the design is locked.
- **Name**. Name of the design.
- **Type**. Type of the design.
- **Rev**. Revision of the design
- **State**. Current state of the design in lifecycle.
- **Description**. Description of the design.

Right-click the selected design to display the following popup menu:



- **Export**. This option exports selected designs from 3DEXPERIENCE in to the zip archives created under the default directory mentioned in 'Export Archive to' path in design export dialog. See *Exporting Designs From 3DEXPERIENCE* for more details.
- **Iterations**. Exports the selected iteration of the design to the zip archives created under the default directory mentioned in Export Archive to path in design export dialog . See *Exporting Particular Iteration of Design* for more details.
- **Simplified Reps.** For Creo Parametric assemblies, this option will allow the user to export a specified simplified representation See *Exporting a simplified Rep of a Design*
- **Baselines**. This option is available only if baselines exist for the selected design. See *Baselining Designs.*
- **Explore in 3DEXPERIENCE**. Opens CAD Portal view of the design in the browser. See *Viewing Design Details in X-CAD Design*
- **Properties**. Displays properties of a design that already exists in 3DEXPERIENCE. See *Viewing Properties*
- **Where Used**. Displays names of designs that are using the selected model.



Exporting Designs from 3DEXPERIENCE in Creo Parametric

This task shows you how to Export designs from 3DEXPERIENCE.

The designs must already be saved in 3DEXPERIENCE. See *Saving Designs to 3DEXPERIENCE*.

Click **3DEXPERIENCE > Design Export** from Creo Parametric ribbon/toolbar.

The **3DEXPERIENCE Design Export** dialog box opens.

1. Search for the designs using **Search** option in **3DEXPERIENCE Open** dialog box. See *Searching Designs*

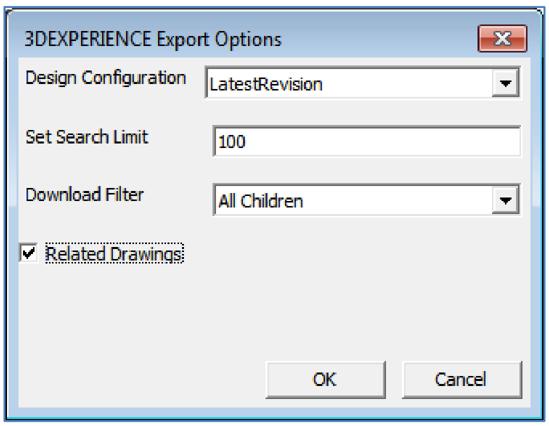
   The results of the search are listed.

   Select a design from the list to view the preview image of the design which is displayed in the left corner of **3DEXPERIENCE Design Export** dialog box.

2. Enter the following options in **3DEXPERIENCE Design Export** dialog box:

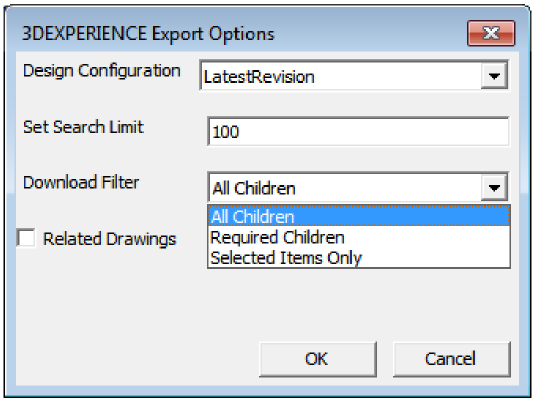
   - **Export Archives To**. The designs exported from 3DEXPERIENCE are exported to the new zip archive created under directory specified. The directory specified in the Checkout preferences by the Integration Administrator is shown by default. Click **[...]** to browse and select another directory as the checkout directory.

3. Click **Options** in **3DEXPERIENCE Design Export** dialog box.



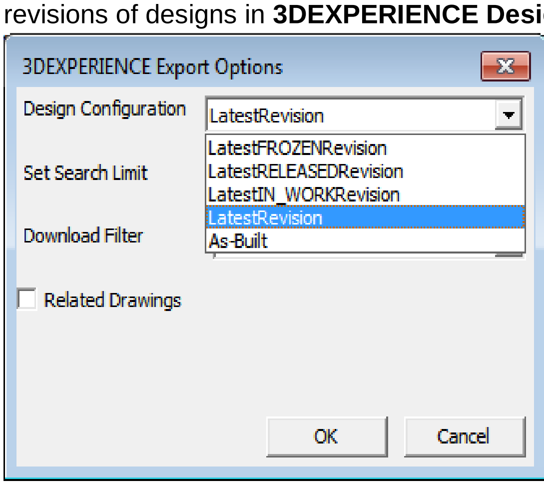
   The **3DEXPERIENCE Design Export Options** dialog box opens. Choose options to be executed when you export a design from 3DEXPERIENCE:

   - **Related Drawings**. Downloads all associated drawing to the selected model. When export is submitted new zip archives will be created under the default checkout folder for each of the top model having names like *<model name>_<model type>_<timestamp>*
   - **Set Search Limit**. Using this option, an integration user can set the search limit.
   - **Download Filter**: There are three options available in this dropdown.



- **All Children.** Select this checkbox to export the child objects of selected objects. The intial value is populated based on the user's MCADInteg-SelectChildItems Local Configuration setting (also available via 'Select Children if Parent is Selected' option in Designer/Preferences). Leaving this checkbox blank will improve performance if, for example, the user desires to only checkout the top level of an assembly or the user desires to checkout a family table template without its family table members.
- **Required Children.** This option allows user to only download direct children, components connected by assemblyComponent and drawing relationships, on export. On export with this option, the user will not get objects connected by external-copy geometry relationship
- **Selected items only.** This option allows user to only export the items selected in the search results pane.

- **Design Configuration**. Select an option from the drop down list to display specific Iterations or revisions of designs in **3DEXPERIENCE Design Export** dialog page:



Click **OK**.

4. Click **Export**.

During export, irrespective of enabling RFA setup and Lock, Unlock status of design, the designs are downloaded to the new zip archives that are created under the default directory i.e. specified in

‘Export Archive to’ path. The format of zip archive name is *<model name>_<model type>_<timestamp>.*

Exporting Particular Iteration of Designs

This task shows you how to export an earlier or later iteration of a design from 3DEXPERIENCE. This feature is used to retrieve an earlier design or a later iteration of a design if it exists to restart development.

Multiple Iterations of the design must exist in 3DEXPERIENCE.

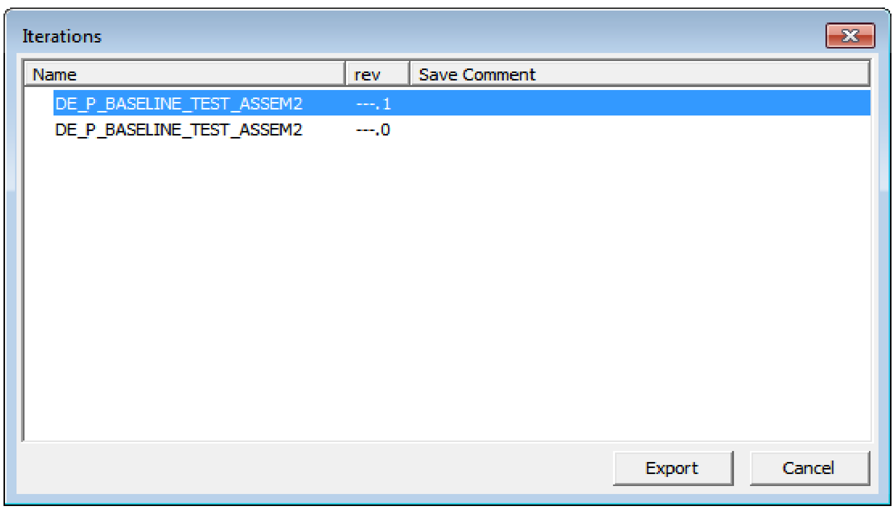
1. Click **3DEXPERIENCE > Design Export** from Creo Parametric ribbon/toolbar.

The **3DEXPERIENCE Design Export** dialog box opens.

2. Search for the design using **Search** option in **3DEXPERIENCE Design Export** dialog box. See *Searching Designs*

The search results are listed in **3DEXPERIENCE Design Export** dialog box.

3. Right-click a design and click **Iterations** to open the **Iterations** dialog box.



Note: The above dialog is table driven. The displayed dialog can be customized using the table specified in the IEF-DefaultConfigTables GCO variable.

4. Select a single iteration to be Exported.

Exporting a simplified Rep of a Design

This task shows you how to export a simplified rep of a design from 3DEXPERIENCE in Creo Parametric.

The design must correspond to a Creo Parametric assembly.

1. Click **3DEXPERIENCE** > **Design Export** from Creo Parametric ribbon/tool bar.

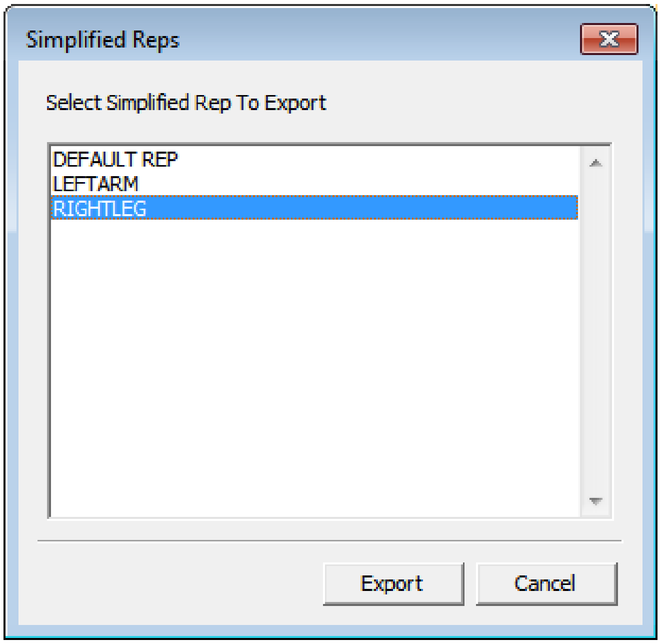
The **3DEXPERIENCE Design Export** dialog box opens.

2. Search for the design using **Search** option in **3DEXPERIENCE Design Export** dialog box. See *Searching Designs*

3. The search results are listed in **3DEXPERIENCE Design Export** dialog box.

4. Right-click a design and click **Simplified Reps** to export the **Simplified Reps** dialog box.

Note: This option is only displayed if the design is a ProE assembly.



5. Select a single simplified rep to be export.

   - The selected simplified rep of the assembly design is exported to to the zip archives created under the default directory mentioned in 'Export Archive to' path in design export dialog. 3DEXPERIENCE Connector for Creo Parametric downloads components necessary to export simplified representation of assembly, and the excluded components of representation will not be downloaded to the zip archive created under default directory. This zip archive is with name <model name>_<model type>_<timestamp>.

Baselines

Baselines are used to capture a design or design structure which the designer feels can be used as a benchmark before proceeding to make more changes to the design. The following section explains how to export design on baseline

Exporting Baselines Designs

This task shows you how to export a design from 3DEXPERIENCE using an existing baseline.

Baseline must exist for the design.

1. Click **3DEXPERIENCE > Design Export** in Creo Parametric ribbon/toolbar.

The **3DEXPERIENCE Design Export** dialog box opens.

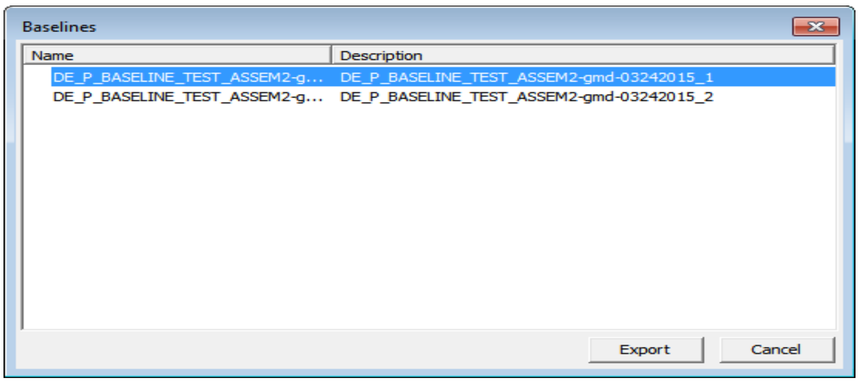
2. Search for the baselined design in the **3DEXPERIENCE Design Export** dialog box. See [Searching Designs]

   The results are listed in the **3DEXPERIENCE Design Export** dialog box.

3. Right-click the name of baselined design and click **Baselines** ><Name of the Baseline> .

   If multiple baselines exist for the design, then all the baseline names are listed.

   Note: Baselines are applied only to the node selected during creation and the structure beneath the node does not inherit the baseline.



   Note: The above dialog is table driven. The displayed dialog can be customized using the table specified in the IEF-DefaultConfigTables GCO variable.

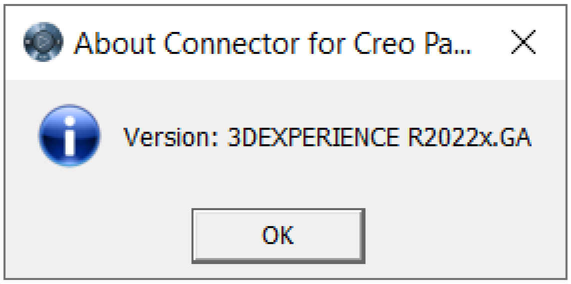
The design, retaining the design's characteristics at the time the baseline was created is exported to the zip archive created under the default directory mentioned in “Export Archive to” path in design export dialog

About

This command helps users to see the version of their ‘3DEXPERIENCE Connector for Creo Parametric’.

Click **3DEXPERIENCE** > **About…** from Creo Parametric ribbon/ toolbar, menu or Ribbon

The version of the integration will be displayed to the user.



Family Tables are an advanced feature of Creo Parametric. The 3DEXPERIENCE Connector for Creo Parametric supports family tables and allows you to save or open family tables and instances.

When you check in a family table for the first time the Save page displays all the instances of the family table. On successful checkin of the family table, the integration will create a family table object and upload the prt or asm file to it. The 3DEXPERIENCE Connector for Creo Parametric allows you to checkout one family table instance or the entire family. If you want to modify only one instance, you can open the instance object, make the change, and check it back in. In the case of assembly family tables, if you checkout only one family table and not the entire family, the 3DEXPERIENCE Connector for Creo Parametric brings only the part (component) files required for the assembly family table being checked out. All the operations such as baseline (labeling), iteration, Lock/Unlock works the same way as with any other CAD-object. Click Instances in the category list of the objects Portal View page to access all the Instances or family tables of the parameterized model.

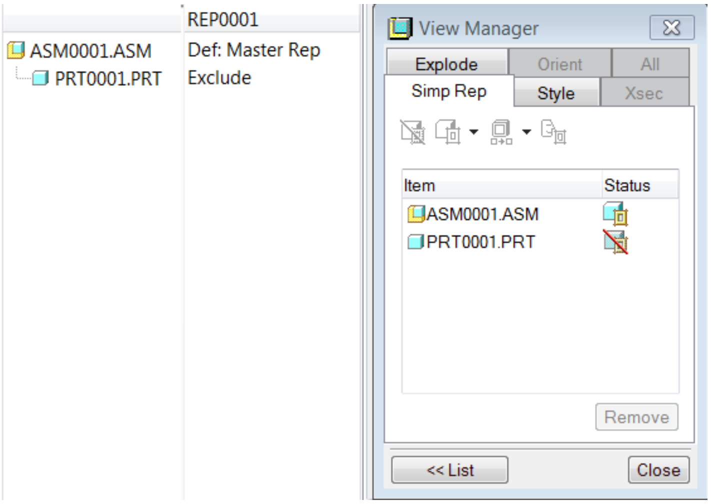
Lock of a Family Table Instance

On lock of an instance, the integration will automatically lock the family table object. If the family table object is locked by another user, the integration may not be able its instance.

Simplified Representations

Simplified Representations are an advanced feature of Creo Parametric. The 3DEXPERIENCE Connector for Creo Parametric has a limited support for simplified representations of Assemblies and allows you to save or open simplified representations. To know how to open the saved simplified representations of an assembly please refer Opening Particular Simplified Rep of a Design .

Simplified representations in an Assembly are supported only if their default simp rep rule is set to 'Master Rep'. (Def: Master Rep) However, you can 'Exclude' components to define the simplified representation.



- During creation of Simplified Representations, ensure that the default simp rep rule is always 'Master Rep'.
- Till Wildfire 4.0, the default simp rep rule is 'Master Rep'. So, user should not change the default simp rep rule.
- In Wildfire 5.0, the config.pro option 'simprep_default_model_status' should be set to 'master'.

The following features of Simplified Representations are not supported in this release.

- Include and Substitute features of Simplified Representations.
- Simplified representations in Parts.
- Drawings associated with Simplified representations.
- The simp rep rule of 'Exclude Comp'.

The checkin of Simplified Representation design is now based on ProEAutoLoadMasterRep GCO. The default value of ProEAutoLoadMasterRep GCO option is 'First Time', which means that first checkin of Creo Parametric Assembly will loads MASTER representation and checkin complete structure to 3DEXPERIENCE. The subsequent checkin of ProE Assembly representation will checkin components loaded in Creo Parametric session.

ProE Types

Below ProE Types are supported in 3DEXPERIENCE Connector for Creo Parametric:

- Part
- Sheet Metal
- Composite
- Bulk Item
- Part Family Table
- Assembly
- Interchange Assembly
- Assembly Family Table
- Drawing
- Format
- Layout
- Diagram
- Manufacture
- Note
- IDF

Language settings

This section shows you how to change language settings of 3DEXPERIENCE Connector for Creo Parametric.

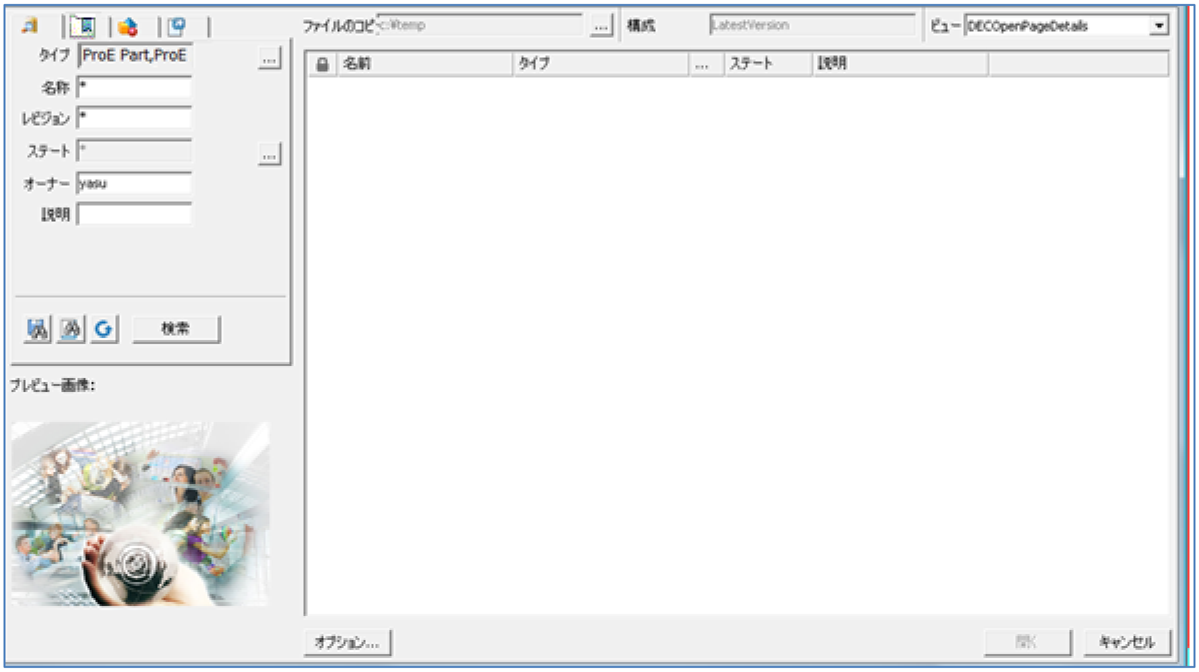
Use below steps to work in Japanese environment. If you need to change other supported language, change 'Japanese' to those supported, 'French' or 'German', or 'Italian' or 'Chinese' and use associated files.

- Set IEFClient and 3DEXPERIENCE Connector for Creo Parametric resource files to Japanese language
    - Start -> Control Panel
    - If you are using Category View
        - Click
          (XP) Date, Time Language, and Regional Options
          (Vista) Clock, Language, ad Region
        - Click Regional and Language Options
    - If you are using Classic view
        - Double Click Regional and Language Options
    - In "Regional and Language Options" dialog
      (XP) Change "Select an item to match its preferences, or click Customize to choose your own format" drop down menu to "Japanese (Japan)"

  For XP, assuming the East Asian Language files are already installed to use with 'Japanese'. If it is not done, In "Regional and Language Options" dialog, Select "Language" tab and check "Install files for East Asian languages" and hit "Apply". This requires reboot.

- Change Creo Parametric language
    - Add system variable LANG=Japanese to your local machine

Below is snapshot of Open dialog in Japanese language



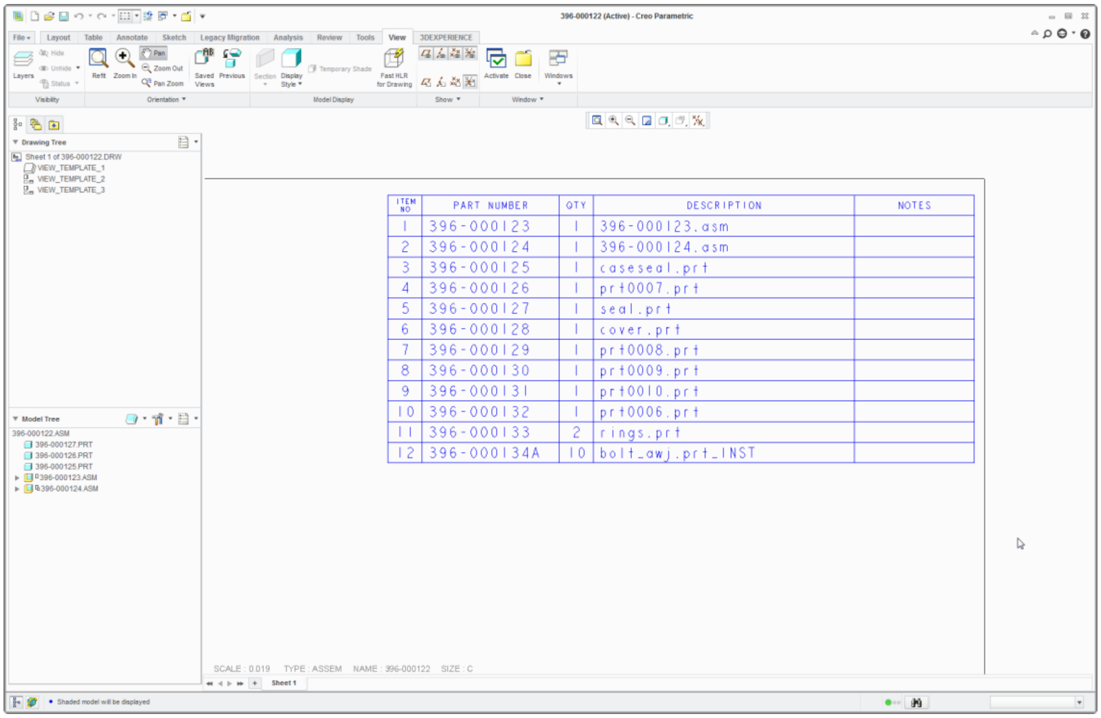
Balloon Transfer from Drawing

This section shows how to transfer balloon numbers of Creo Parametric Models from Creo Parametric drawing to 3DEXPERIENCE.

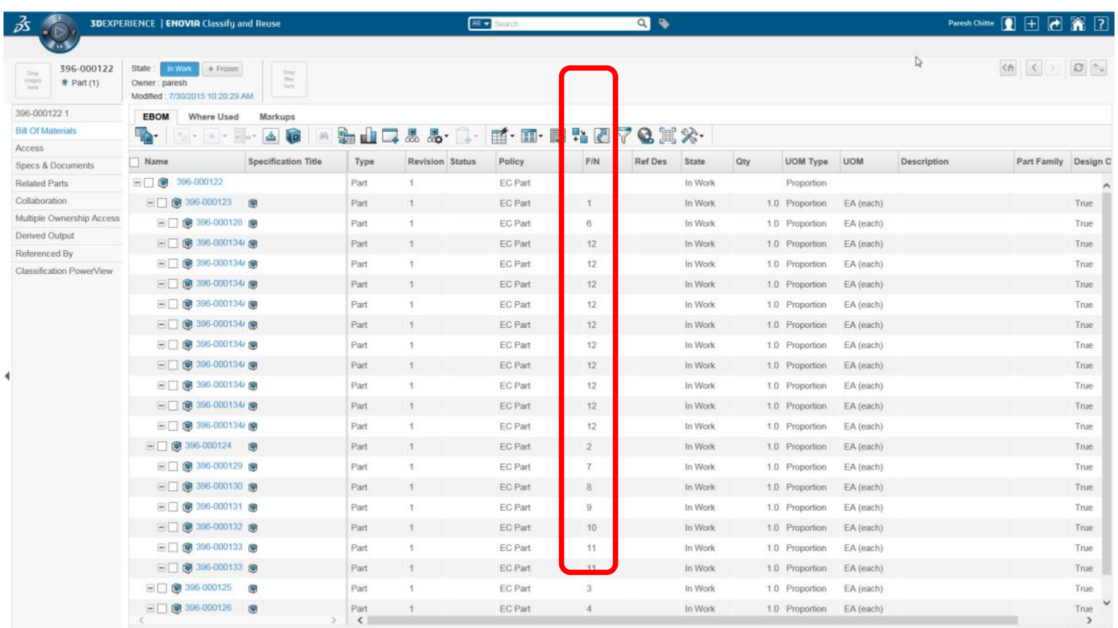
- Add *drawing, balloonnumbers***|***Associated Drawing,Balloon Numbers* to GCO MCADInteg-CADToMxRelAttribMapping.
- Set “Copy Relationship Attributes” on checkin to TRUE in Local Configuration or Preferences page
- Create Drawing of assembly, add balloons to drawing and checkin drawing to 3DEXPERIENCE

Multi-Level Drawing Balloon Transfer to EBOM Find Number

With this new enhancement, CAD users will no longer need to associate a drawing (containing BOM table) with each sub-assembly in order to get the Find Number values of all the components of the BOM structure. Just a single multi-level BOM table associated with top drawing would be enough to get the Find Number after EBOM Sync.



Below is the screenshot representing automatically populated Find Numbers in 3DEXPERIENCE based on CAD Drawing Balloon numbers.



Transformations

3DEXPERIENCE Connector for Creo Parametric checks for attribute mapping on the relationship "CADSubComponent". If the "relativeXform" attribute is mapped on the relationship, then the Transformation functionality starts working.

While saving a design, the relative Transformation information is extracted and set on the corresponding relationship in the design structure. Similarly, while opening a design from 3DEXPERIENCE, this information is received and set on the corresponding parent-child (assembly-component) relationship and the model is rebuilt.

The Creo Parametric transformation matrix is stored as a homogeneous matrix of 16 elements, stored in the order:

$$x_i, x_j, x_k, 0, y_i, y_j, y_k, 0, z_i, z_j, z_k, 0, T_x, T_y, T_z, 1$$

Values x_i, x_j, and x_k are the x-axis components of rotation. Values y_i, y_j, and y_k are the y-axis components of rotation. Values z_i, z_j, and z_k are the z-axis components of rotation. T_x, T_y, and T_z are the x, y, and z components of the translation.

Help

This section shows how to open the quick startup help guide of the 3DEXPERIENCE Connector for Creo Parametric.

Click **3DEXPERIENCE** > **Help…** from Creo Parametric ribbon/toolbar

The integration will take you to the **3D**EXPERIENCE documentation webpage: http://www.3ds.com/support/documentation/v6-users-guide/.

You can change the location of the help guide by modifying the helpURL setting of the file <3DEXPERIENCE PROE Integration Install>\bin\settings.ini.

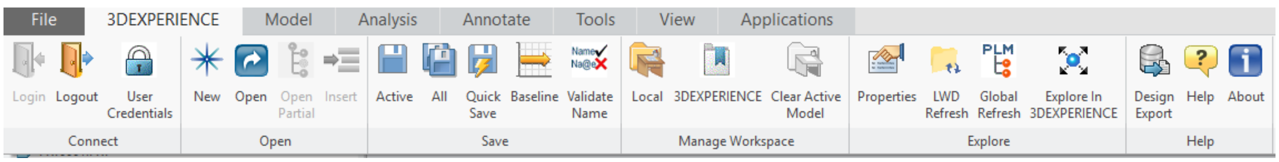
3DEXPERIENCE Interface Description

The various menus and menu commands that are specific to 3DEXPERIENCE Connector for Creo Parametric are described below.

- *3DEXPERIENCE Menu*
- *Context in 3DEXPERIENCE Open Dialog box*
- *Context in 3DEXPERIENCE Save Dialog box*

Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*.

3DEXPERIENCE Ribbon interface for CREO Parametric



Login

See *Connecting to 3DEXPERIENCE From Creo Parametric*

New

See *Creating Designs Using Templates*

Open

See *Opening Designs From 3DEXPERIENCE in Creo Parametric*

Open Partial

See *Opening Designs Partially From 3DEXPERIENCE in Creo Parametric*

Insert

See *Inserting Designs From 3DEXPERIENCE in an Active Design*

Save

See *About Saving in 3DEXPERIENCE*

Quick Save

See *Using Quick Save*

Validate Name

See *Validate Name*

Baseline

See *Baselining Designs*

Properties

See *Viewing Properties*

Manage Workspaces

See *Managing Designs Using Integration Exchange Framework Client*

Explore in 3DEXPERIENCE

See *Viewing Design Details in X-CAD Design*

Global Refresh

See *Global Refresh*

Design Export

See *Design Export*

Help

See *Help*

About

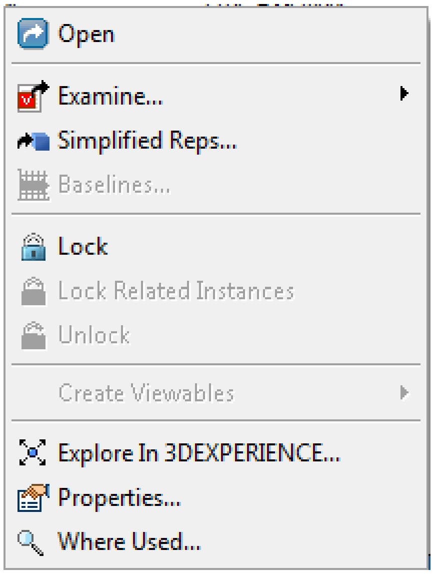
See *About*

Logout

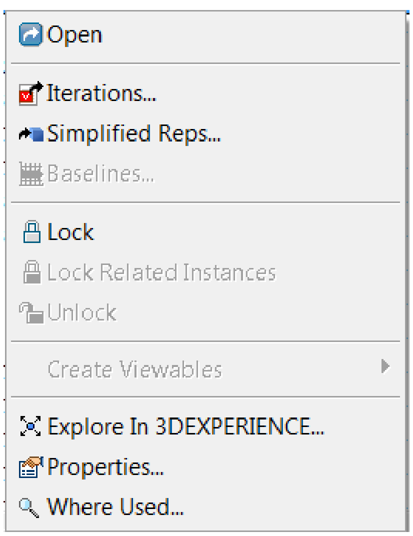
See *Disconnect From 3DEXPERIENCE*

Context Menu in 3DEXPERIENCE Open/ Insert Dialog box

In 3DEXPERIENCE Open dialog, Right-click the selected design to display the following context menu:



In 3DEXPERIENCE Insert dialog, Right-click the selected design to display the following context menu:



Open

See *About Opening Designs From 3DEXPERIENCE*

Iterations (Available in Insert Dialog)

See *Opening Particular Iteration of a Design*

Examine (Available only in Open Dialog)

See *Examine Particular Iteration of a Design*

Simplified Reps

See *Opening Particular Simplified Rep of a Design*

Baselines

See *Opening Baselined Designs*

Lock

See *Locking and Unlocking Designs*

Unlock

See *Locking and Unlocking Designs*

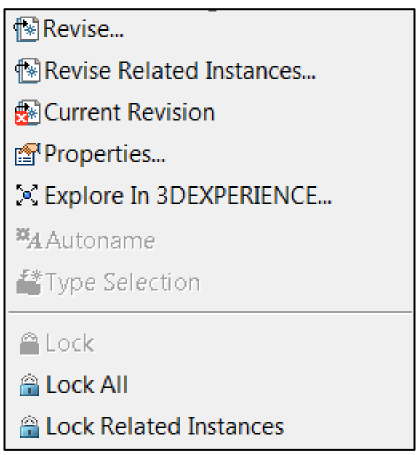
Explore in 3DEXPERIENCE

See *Viewing Design Details in X-CAD Design*

Properties

See *Viewing Properties*

Context Menu in 3DEXPERIENCE Save Dialog box



Revise

See *Revise Design*

Current Revision

See *Revise Design*

Properties

See *Viewing Properties*

Explore In 3DEXPERIENCE

See *Viewing Design Details in X-CAD Design*

Autoname

See *Using Autoname*

Type Selection

Lock

See *Locking and Unlocking Designs*

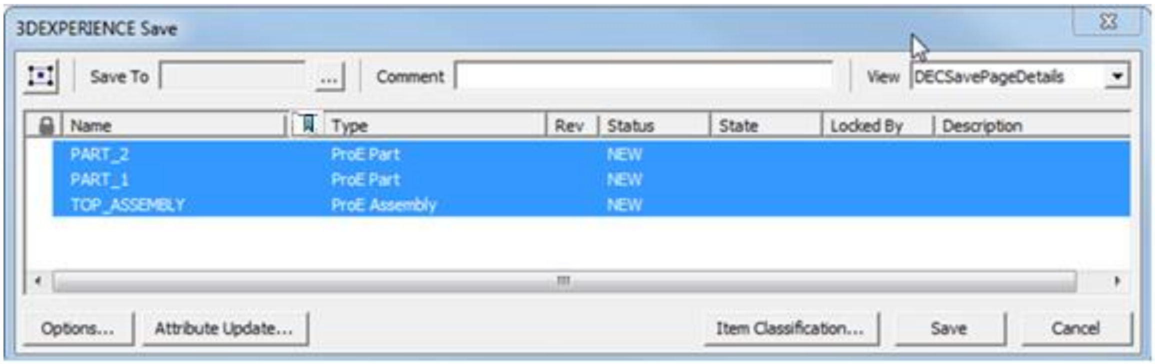
Lock All

See *Locking and Unlocking Designs*

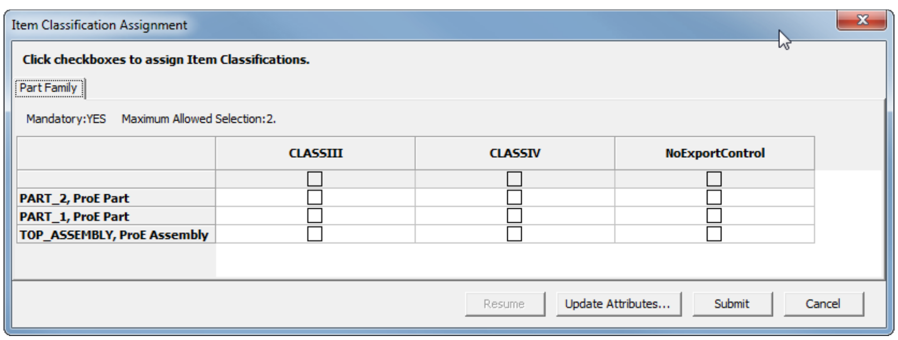
Design Classification During Save

A new functionality is available which allows Creo users to classify the objects right inside the CAD session using integration's 3DEXPERIENCE Save UI. The Save dialog is enhanced to provide all the support needed to classify the objects. Following are the steps to guide the users.

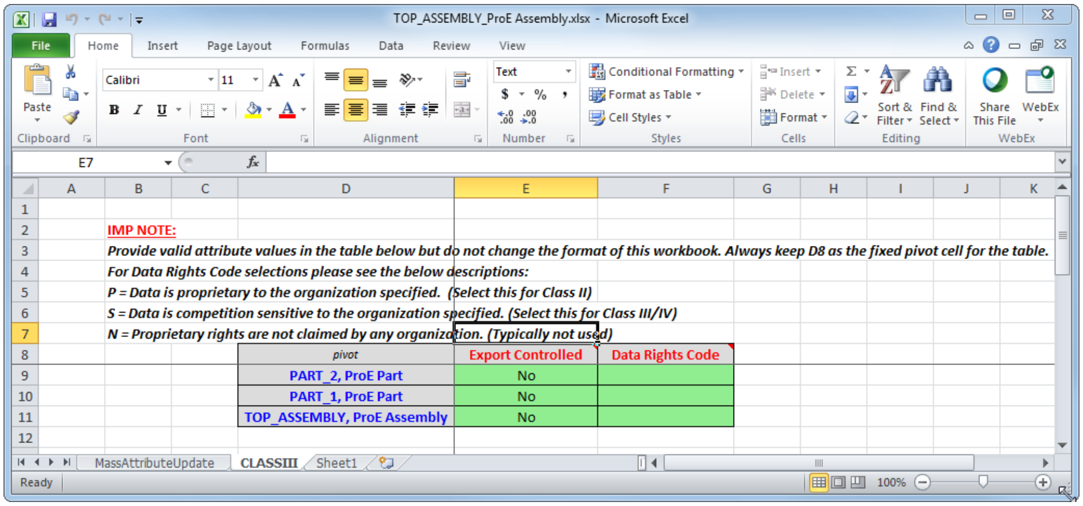
1. Designer completes the CAD design work.
2. Designer selects 3DEXPERIENCE->Save->Active or Save->All
3. Save dialog comes up with all the NEW and MODIFIED objects
4. Save dialog will have an additional button called "Item Classification" (Only **NEW** objects can be classified)



5. On clicking 'Item Classification' button designer will be presented with following dialog



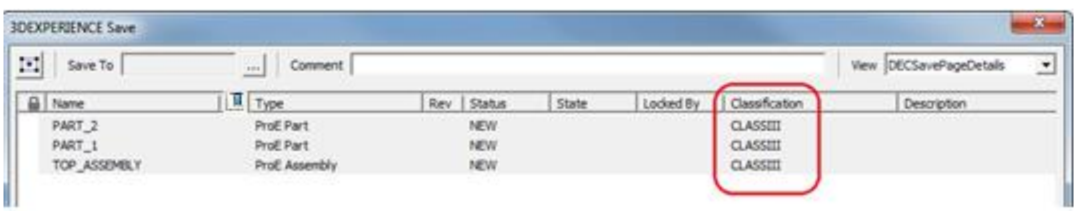
6. The Designer can make the selection for desired classification objects in the checkboxes against all **NEW** objects (parts, assemblies, drawings) and can then assign values to the attributes of those classification objects by clicking on the "Update Attributes" button on above dialog.
7. On clicking the "Update Attributes" button a spread sheet will be launched for each top assembly in the classification assignment dialog. Following is the screenshot for sample spreadsheet



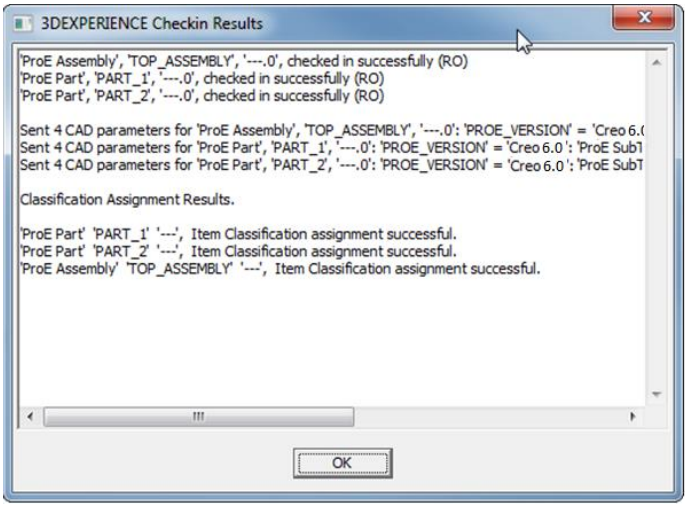
Note: Spreadsheet contains following worksheets

- Mass Attribute Update to update the CAD Parameters
- Classification worksheets for updating attributes

8. Once Designer is done with assigning attribute values to all the classification objects for all objects he can save and close the spreadsheet(s), come back to the classification dialog and hit “Resume” and then “Submit” buttons. This will add a new column on Save dialog and list all the assigned classifications for each NEW object.



9. After this step when Designer submits the save dialog, all the Objects will be first Saved to 3DEXPERIENCE and then a server process will be invoked to assign classifications in 3DEXPERIENCE for all NEW objects as per the user selection.
10. Once the Save and Classification assignment is successful the designers see a success message.



Capability Summary

The designers will be allowed to assign the classifications to the **NEW** MCAD designs only. The users will not have the ability to change the classification assignments to the existing CAD objects

The integration allows only those CAD designers who have appropriate access to assign the appropriate Classifications to the CAD objects (The Classification Assignment control for users is described in detail in later sections).

The integration allows designers to modify the non-classification attributes such as description, owner and other CAD attributes inside of the Excel spreadsheet to help the users enter the data quicker. This is essentially providing “Mass Attribute Update” capability along with the classification attribute updates.

Appendix

Configure settings.ini

This section shows you how to configure the settings.ini of 3DEXPERIENCE Connector for Creo Parametric.

The file is located in <3DEXPERIENCE PROE Integration Install>\bin\settings.ini.

General Section

- attrSyncOnFileOpen = 0
    - Synchronize attributes on File/Open command (0: disabled, 1: enabled)
- cachingInCheckin= 1
    - Cache the object details during checkin. (0: disabled, 1: enabled)
- copyAttribOnCheckin = 1
    - Copy attributes on checkin (0: disabled, 1: enabled)
- copyAttribOnCheckout = 1
    - Copy attributes on checkout (0: disabled, 1: enabled)
- deleteModelsFromDisk = 1
    - Delete files from disk while doing 3DEXPERIENCE->Clear Active Model
- enableIDFfeature = 0
    - Checkin the IDF Model (0: disabled, 1: enabled)
- ftKeepTogetherMode = 0
    - Handles Family Table together mode [1: enables together mode 0: disabled to have the OOTB behavior]
- lockReadOnlySave = 1
    - Lock on read only file for Save (0: disabled, 1: enabled)
- mandatory_image_jpg = 1
    - Makes image_jpg derived output selected by default if enabled (ignores LCO setting)
- openSearchLimit = 100
    - Limit of searched item for open dlg
- parallelProcessingInCheckin = 0
    - Do parallel processing on both client and server side during checkin. (0: disabled, 1: enabled)
- progressdialogs = 1
    - Show progress dialogs (0: disabled, 1: enabled)
- readOnlyWarnings = 0
    - Show read-only warnings for unlocked models. If enabled, the parameters specified in the CAD Section below will then become enabled. (0: disabled, 1: enabled)
- saveConfirmationDlg = 1
    - Displays confirmation dialog if files in Save dialog are not selected or grayed out and Ok button is hit. (0: disabled, 1: enabled)
- showMessages = 1
    - Messages to show user. (0: None, 1: Warnings and Errors, 2: All)
- showFTObjInSaveDlg = 0
    - Show Family Table Object in Save Dialog (0: disabled, 1: enabled)

- SSOEnabled = 0
    - SSO is enabled. (0: disabled, 1: enabled)
- enforceItemClassification = 1
    - Set it to 1 if Library central item Classification setup is available
- enableUpdateClassificationAttributes = 1
    - 0: default, not update attributes. 1: update attributes
- attribDiscrepancyReport = 0
    - For enabling the generation of attribute discrepancy report on checkout
- attribDiscrepancyReportDir =''
    - Specify the directory where the attribute discrepancy report will be saved
- attribDiscrepancyStateName ='All'
    - States of the object which will be considered for generating the attribute discrepancy report. (Migrated or Preliminary,Migrated or All)
- grayNonLockInSave = 1
    - specify to disable non-lock in save dialog
- unifiedlogin = 1
    - unifiedlogin login is enabled (0: disabled, 1: enabled)
- automaticDerivedOutputs [No default setting]
    - Update the list of Dictionaries for Automatic options
- manualDerivedOutputs [No default setting]
    - Update the list of Dictionaries for Manual options
- backgroundDerivedOutputs[No default setting]
    - Update the list of Dictionaries for Background options
- queryWorkspacesAssignedToUserOnly = <>
    - Query workspaces directly assigned to logged in user. (0: disabled, 1: enabled)
- enableWorkspaceExpansionToAllLevels = 0
    - Enables workspace folders to be expanded one level each time, In Save dialog and in Open dialog. (0: expand one level, 1: expand all levels)
- showTitleforControlledFolder = 1
    - For Control folders title will be shown in Open and Save dialog. (0: show name, 1: show title)
- hideFoldersWithState = <lifecycle state of the folders to be hidden >
    - Hide control folders with specified lifecycle state and do not show them in Open and Save dialog.
- enableWorkspaceExpansionToAllLevels = 1
    - for expanding workspaces at all levels.(1 :Expand workspaces at all level, 0: Don't Expand workspace at all level)
- processAllInstances = 0
    - 1: Process all family table instances on checkout to rename all instances (not just ones used in structure) after save as operation.
    - 0 : Disable Process all Instances
- CreoForceSelectableUnloadedChildRelationships = 0
    - 1 : In case of Save Active if eligible children from assembly are not selected then inform user about the objects and block save
    - 0 : Disable Block save functionality
- doNotBlockModifiedUnlockedDesigns =0
    - 1: modified objects will be ignored for blocking save,

- 0: Save will blocked for modified objects

Server Section

- appletLogLevel = 4
  - log level for the harness applet
- connectToServerTime = 15
  - Number of seconds to keep trying to login to the JSP before timeout.
- waitToConnectSocketTime = 2
  - Delay time between connecting (socket) and login.

Debug Section

- extendedDebugPrints = 0
  - For debugging purposes only. Prints extensive information to the log file for the investigation of issues. (0: disabled, 1: enabled)
- debugPrintTime = 0
  - Add Time statements to all the logging outputs. (0: disabled, 1: enabled)

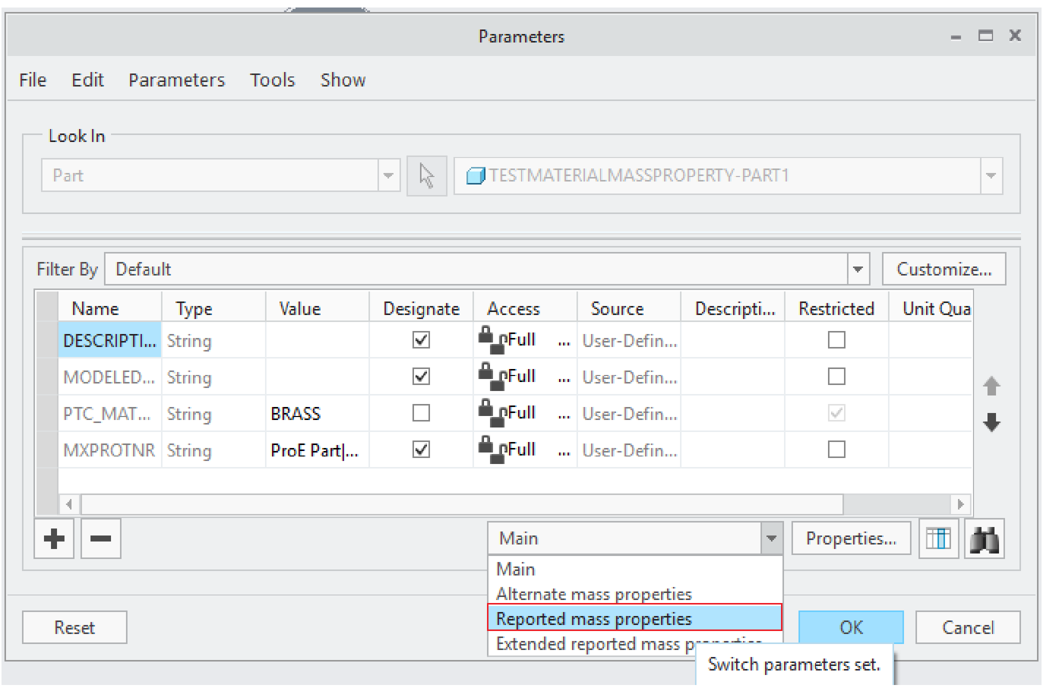
CAD Section

- rename_pdm_objects = 1
  - This will set the let_proe_rename_pdm_objects in config.pro
- set_remember_replaced_components = 0
  - Set to (1 or 0) will preserve/remove dependency to replaced component
- show_readonly_warn_feature = 1
  - To warn user while modifying features of the readonly (unlocked) model.
- show_readonly_warn_param = 1
  - To warn user while modifying parameters of the readonly (unlocked) model.
- show_readonly_warn_ft = 1
  - To warn user while modifying Family Table of a readonly (unlocked) part/assembly.
- show_readonly_warn_asm_insert = 1
  - To warn user while modifying readonly (unlocked) assembly through Insert commands.
- show_readonly_warn_dwg_insert = 1
  - To warn user while modifying readonly (unlocked) drawing through Insert commands.
- show_readonly_warn_dwg_sketch = 1
  - To warn user while modifying readonly (unlocked) drawing through Sketch commands.
- show_readonly_warn_dwg_table = 1
  - To warn user while modifying readonly (unlocked) drawing through Table commands.
- show_readonly_warn_dwg_edit = 1
  - To warn user while modifying readonly (unlocked) drawing through Edit commands.
- set_save_instance_accelerator = 1
  - This will set the save_instance_accelerator in config.pro. (0: none, 1: always,2: saved_objects, 3: explicit)
- step_export_profile = <path of step export profile file>;
  - This will set the export_profiles_step option from config.pro to specified step export profile. Applicable Creo4 M010 onwards.

Transfer Material Property to 3DEXPERIENCE

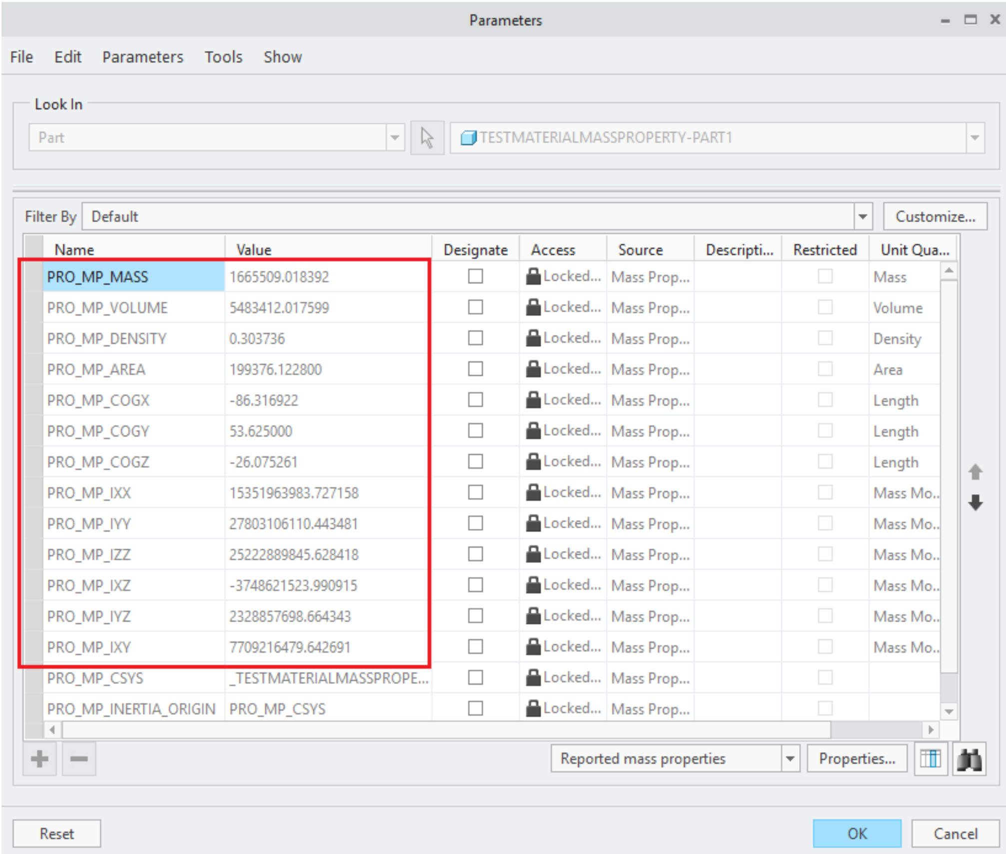
CREO cad data has MATERIAL properties such Density, Volume, Area. Mass, center of gravity etc. These values can be seen in parameters under section “reported mass properties “as follows. These values can be transferred and recorded in 3DEXPERIENCE Attributes on the object that gets created after checkin.

Prerequisite: CREO cad object (Assembly/Part/Assembly Instance/Part instance) should be loaded into session.



Supported material mass properties for transfer to 3DEXPERIENCE:

- ‘PRO_MP_DENSITY',
- 'PRO_MP_MASS',
- 'PRO_MP_VOLUME',
- 'PRO_MP_AREA',
- 'PRO_MP_COGX',
- 'PRO_MP_COGY',
- 'PRO_MP_COGZ',
- 'PRO_MP_IXX',
- 'PRO_MP_IYY',
- 'PRO_MP_IZZ',
- 'PRO_MP_IXZ',
- 'PRO_MP_IYZ',
- 'PRO_MP_IXY’

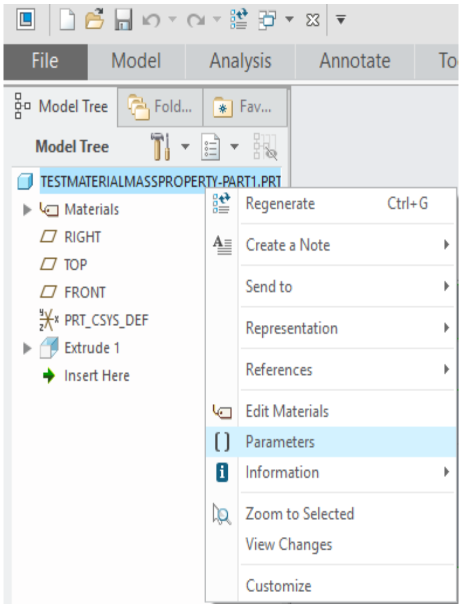


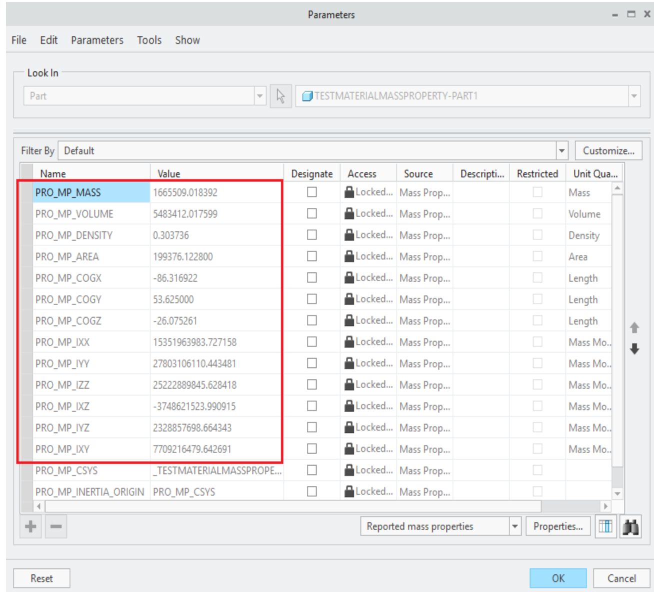
On Server side:

Please refer to the *Install and Administration Guide for 3DEXPERIENCE Connector for CREO* for further details.

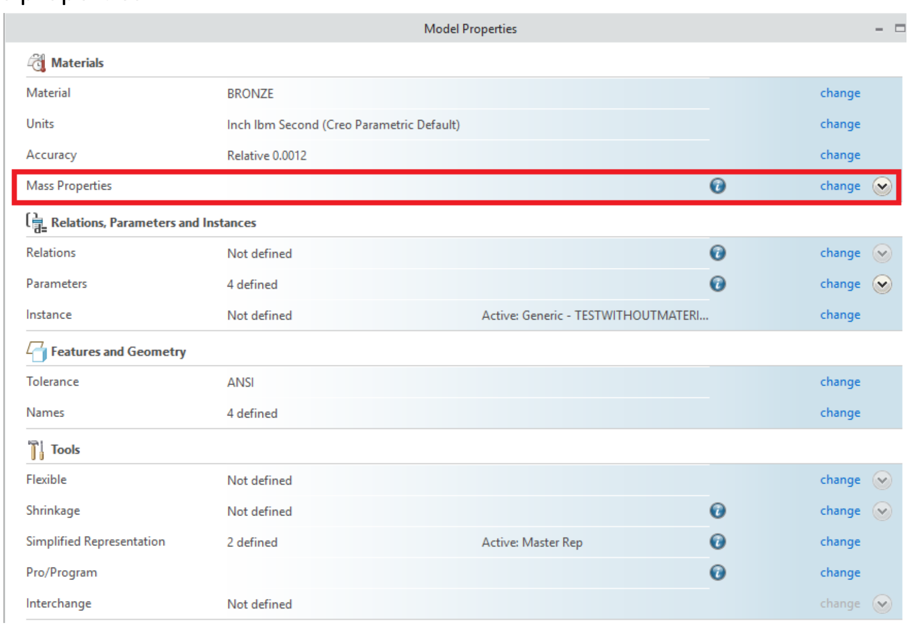
On Client side:

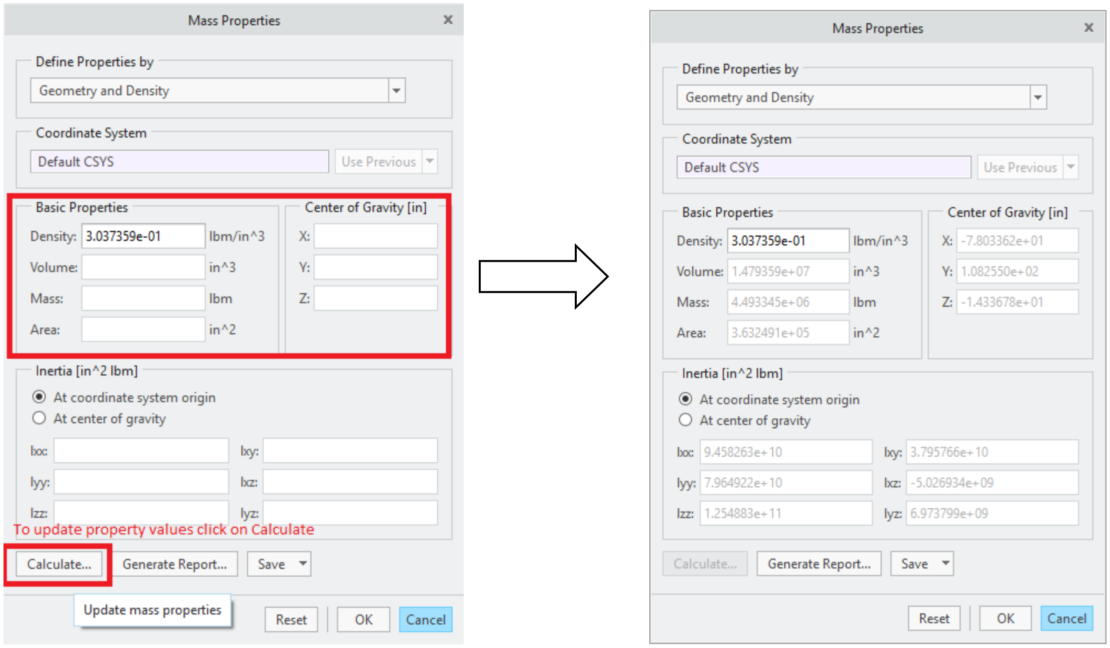
1. Assign material to the Solid body in CREO.
2. Once material assignment is done, click on Model Parameters and change parameter set to "**reported mass properties**" to check material attributes.





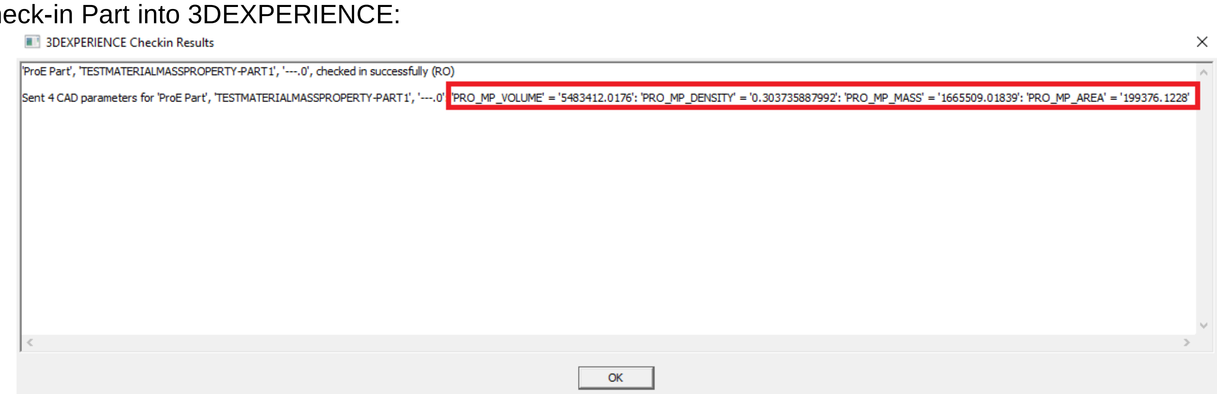
3. If user checks the material mass attributes, and sees material property attributes value information for Mass, Area, Volume, Weight are not updated then to update properties select Change option of Mass properties





4. After calculating, material property values will get reflected in Model parameters as well.

5. Check-in Part into 3DEXPERIENCE:



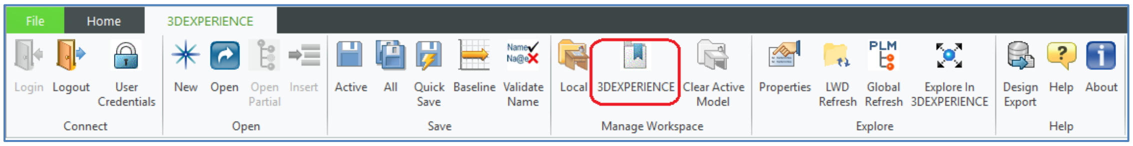
6. Check attributes of the 3DEXPERIENCE object created on server. The object should contain all the material properties.

7. If user assigns material to Part/Assembly and not updated the material properties then in that case default values (Volume = 0.0, Mass = 0.0, Area = 0.0) will get transferred to 3DEXPERIENCE. Material properties will not get updated automatically, user have to calculate the material properties manually. Only density value gets update automatically when material is assigned to the Part.

   Note: Make sure that material properties will have updated values before checkin the Part.

New Bookmark UI and Syntax

The new Menus and icons will look as shown below.



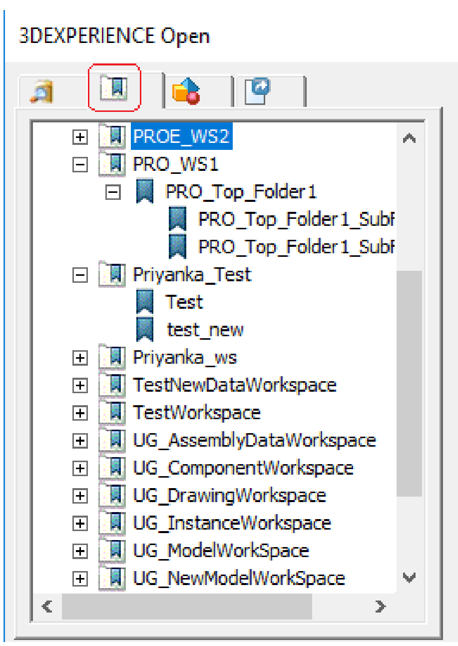
3DEXPERIENCE Ribbon

Note: 3DEXPERIENCE menu icon is replaced by new icon 

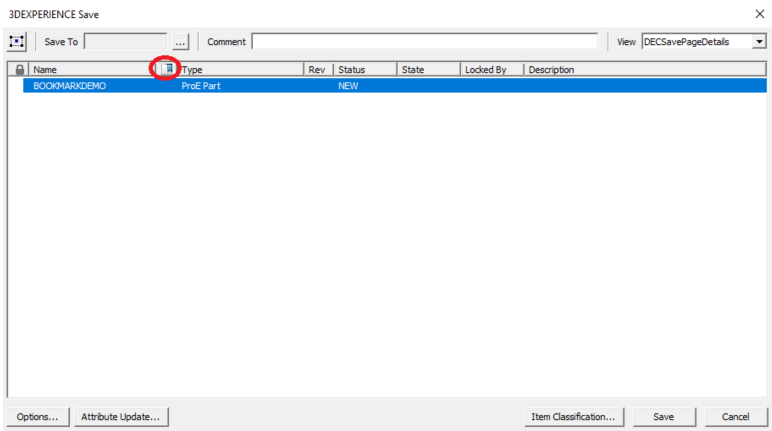
- Workspace is renamed to Bookmark Workspace and Workspace Folder is renamed to Bookmark Folder.
- Workspace Icon is replaced by new icon 
- Open/Closed Folder icon is replaced by new icon 

Above changes will reflect in Open Dialog, Save Dialog and Design Export Dialog as shown below.

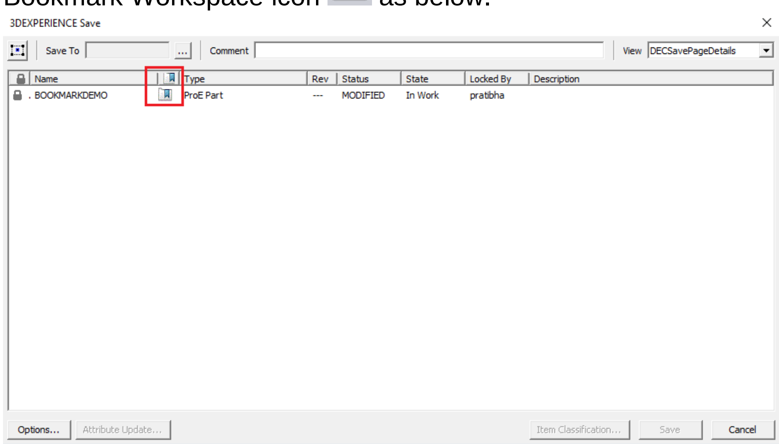
- Open Dialog Bookmark Workspace selection for browsing content:



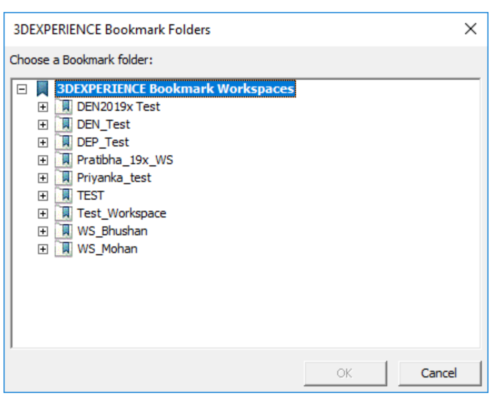
- When user performed save on newly created part then save dialog will be displayed with new Bookmark Workspace icon  as below.



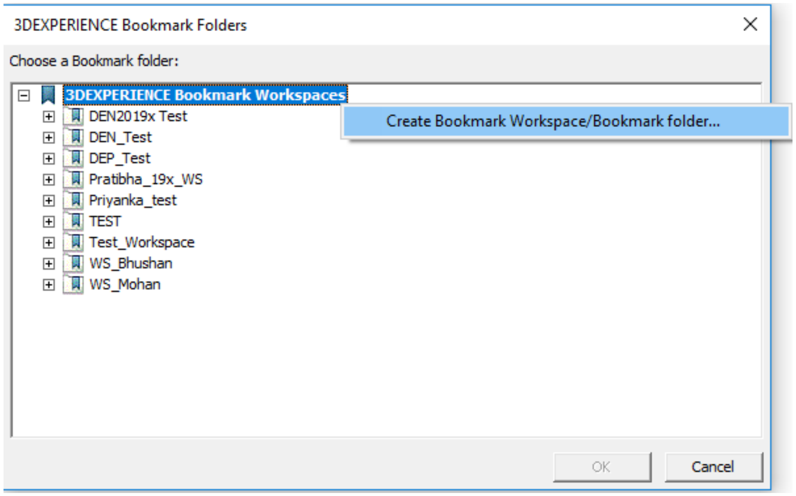
- When user performed save on existing part then save dialog will be displayed with new Bookmark Workspace icon  as below.



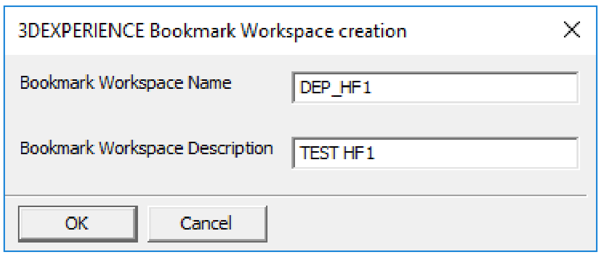
- New Bookmark Workspace UI

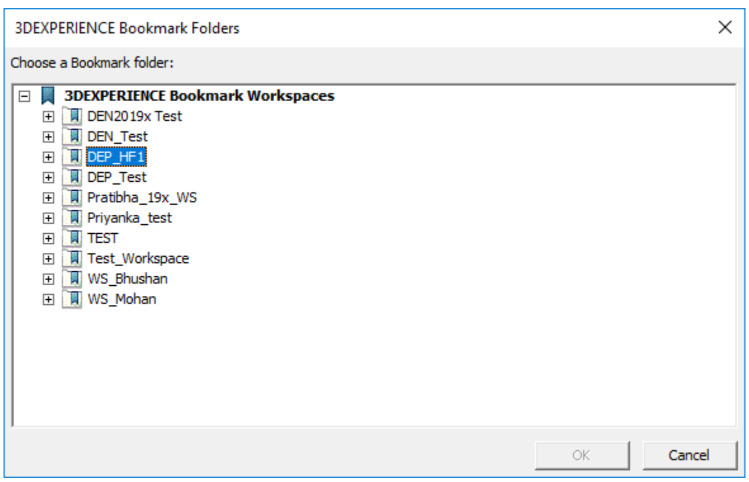


- The RMB command ‘Create Bookmark Workspace/Bookmark folder…’ can be used for creating a new Bookmark Workspace in 3DEXPERIENCE.

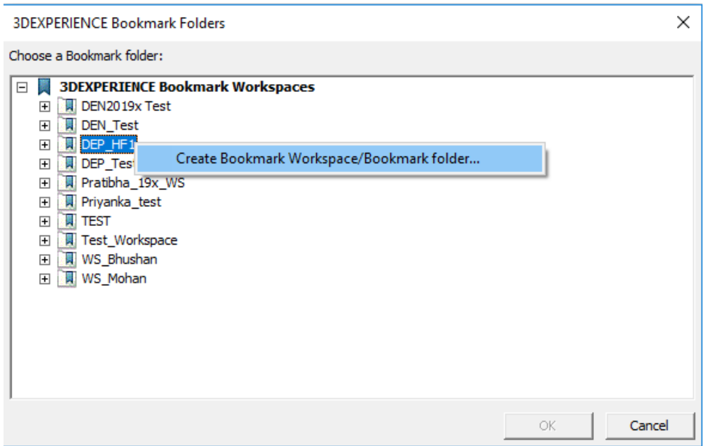


- Upon selecting the RMB command, ‘Create Bookmark Workspace/Bookmark folder…’, The new ‘3DEXPERIENCE Bookmark Workspace creation’ will look like as below

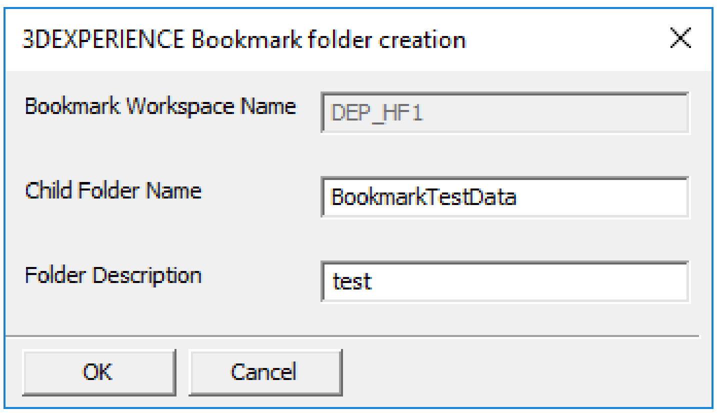


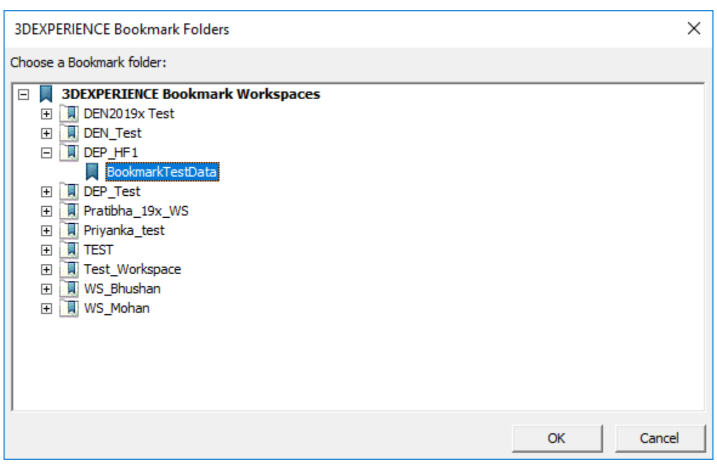


- On any existing Bookmark Workspace, the User can use the RMB command ‘Create Bookmark Workspace/Bookmark folder…’ to create a new Bookmark Folder.

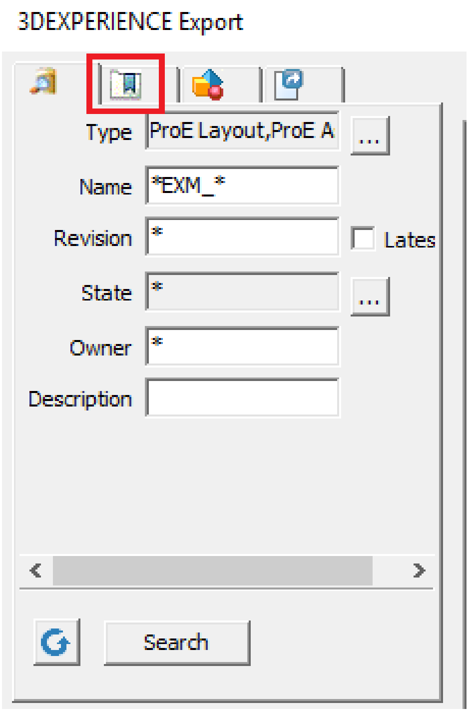


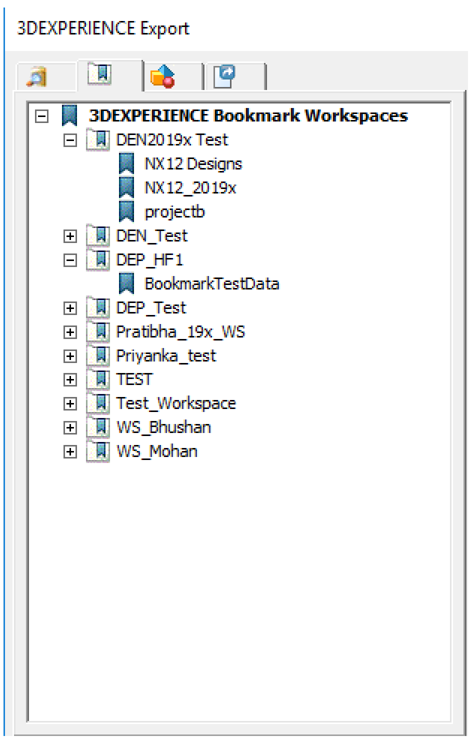
- Upon selecting the RMB command to create the Bookmark folder, following Bookmark folder creation dialog will be displayed.





- Design Export Bookmark Workspace selection





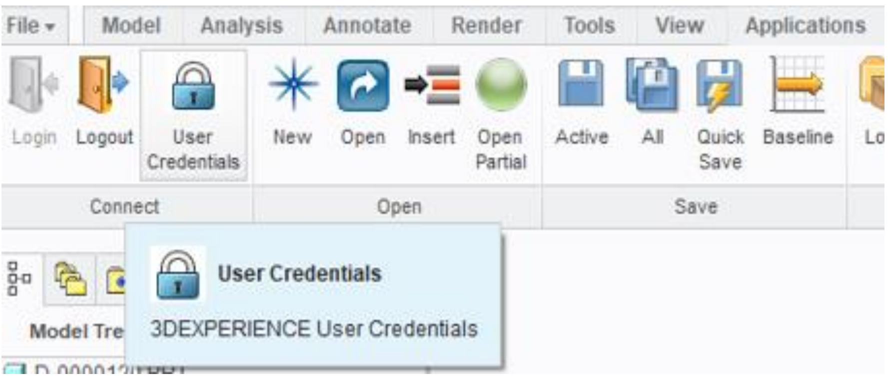
User Credentials

User Credentials menu will provide the options to change the credentials of the user whenever required by the user.

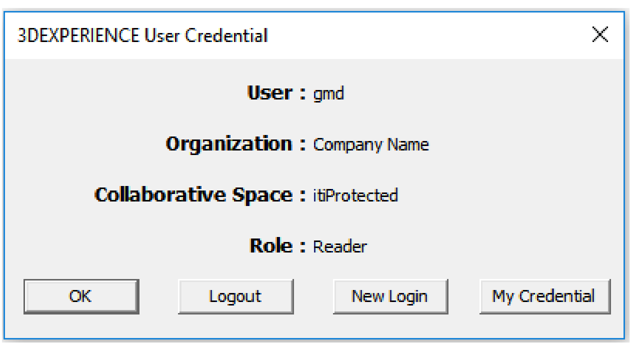
This functionality supports below options:

- Show User Credentials
- Provide Logout option in the User Credentials popup dialog
- Provide "New Login" option in the User Credentials popup dialog
- Change User Credentials

Connect-> User Credentials



When user clicks User Credentials button (**Connect-> User Credentials**), user will be able to view the current user's credentials information as shown below:



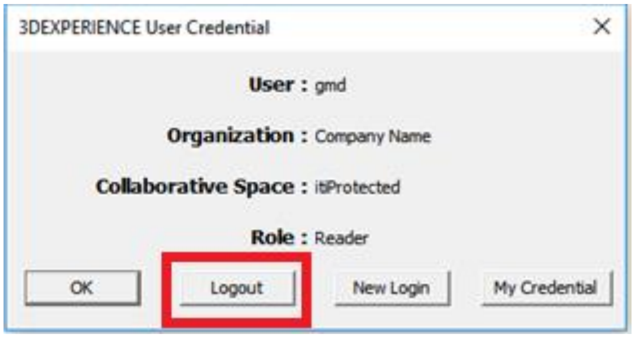
GUI displays existing User, organization, collaborative space and role assigned to user

Connect ->User Credentials ->OK

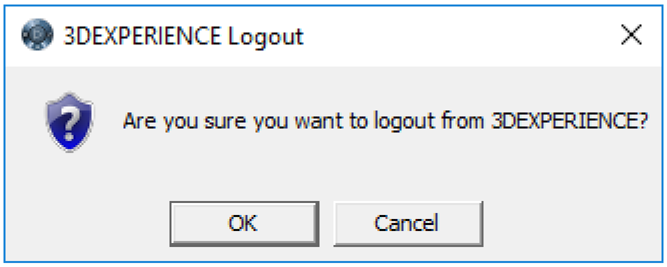
On OK button click user credentials UI will be closed.

Connect->User Credentials ->Logout

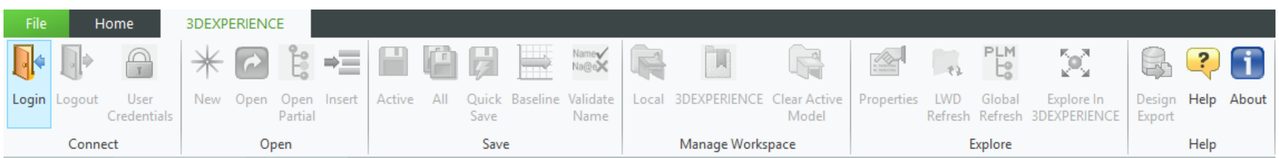
User can logout from the User Credentials popup dialog by selecting **Connect->User Credentials ->Logout** option



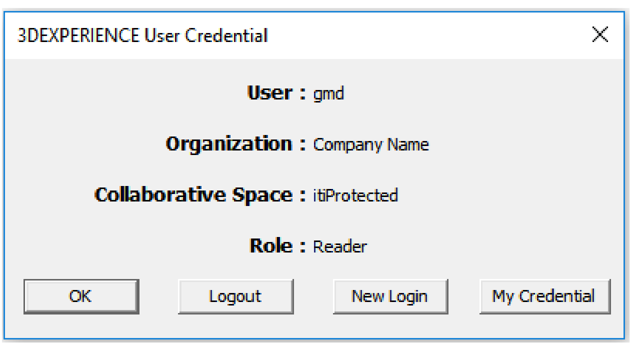
If user clicks on Logout button, then below confirmation window will be shown.



If user proceed with OK button, then the CAD user will be logged out from 3DEXPERIENCE and the UI will be closed. All the menus except login, help and about should get disabled.



If user click on Cancel button, then User credentials UI will remain open in the session.



Connect-> User Credentials ->New Login

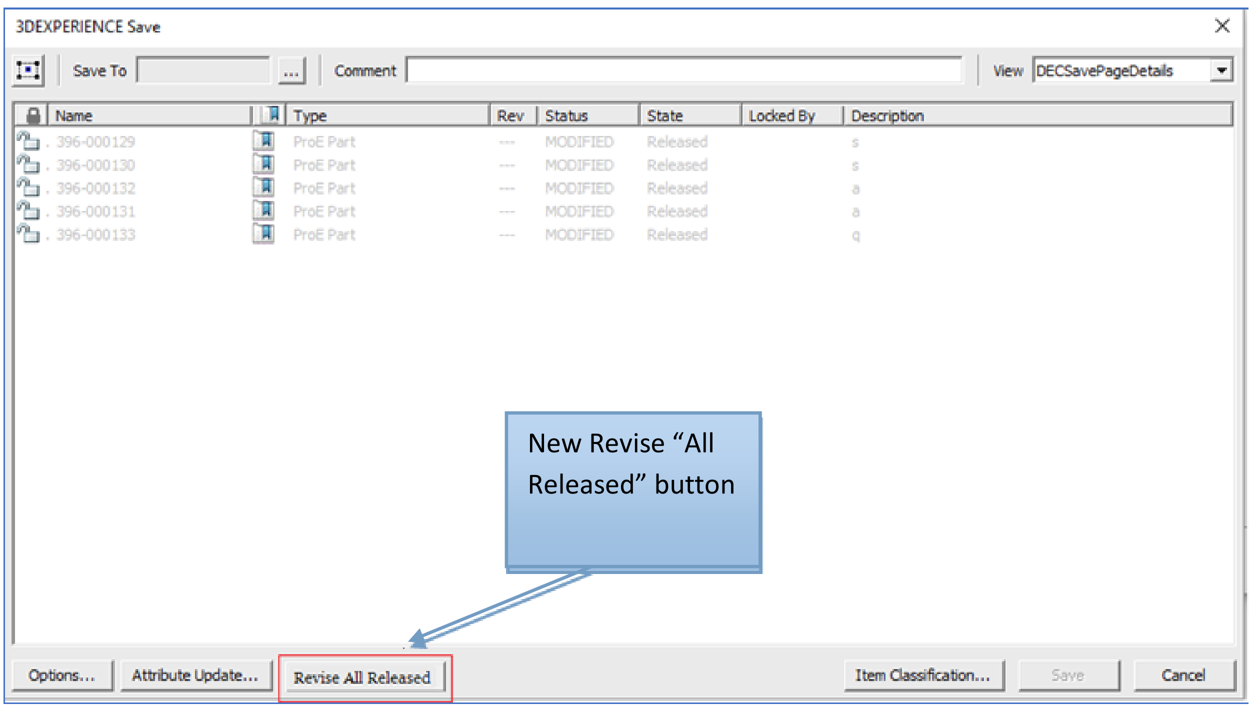
On “New Login” button click user will be logged out from 3DEXPERIENCE and then login dialog will be shown to connect the CAD user to 3DEXPERIENCE.

Revise All Released

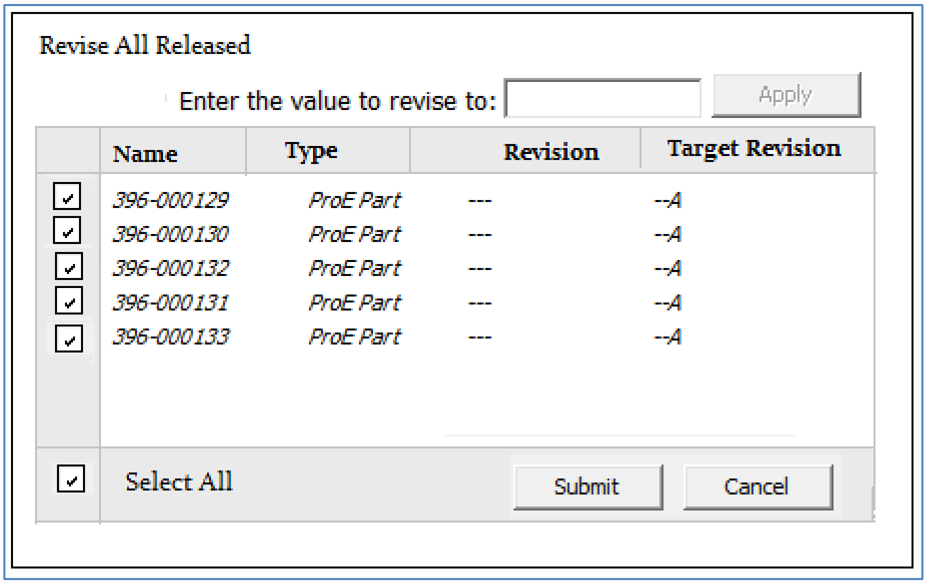
After modification in NX integration session, if release data is to be checked in to PLM , it must be revised in Save dialog. The current revise process in Save dialog requires user to perform right mouse button on each released object and click revise menu, if number of released objects are too many then it will involve too many right mouse button clicks, this could become time consuming and repetitive activity, The purpose of the feature is to provide solution for this problem, so that user can revise all released object at once.

When a released assembly is checked out and modified and attempted to check in using 3DExperience Save functionality, modified objects will appear in save dialog as a grayed out, '***Status'*** being displayed as '***MODIFIED'*** and '***State'*** displayed as '***Released'*** .

- Save dialog changes :
  - New '***Revise All Released'*** button available in save dialog for mass revise .

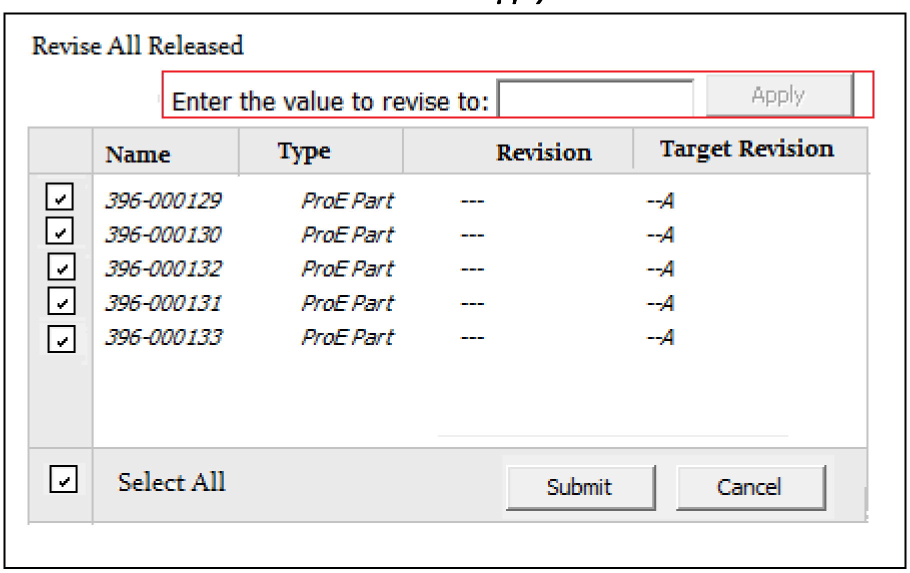


  - Even though the objects in save dialog are grayed out, user can click on '***Revise All Released***' button, and user will see the '***Revise All Released'*** dialog launched.
- New '***Revise All Released'*** dialog will look like as:

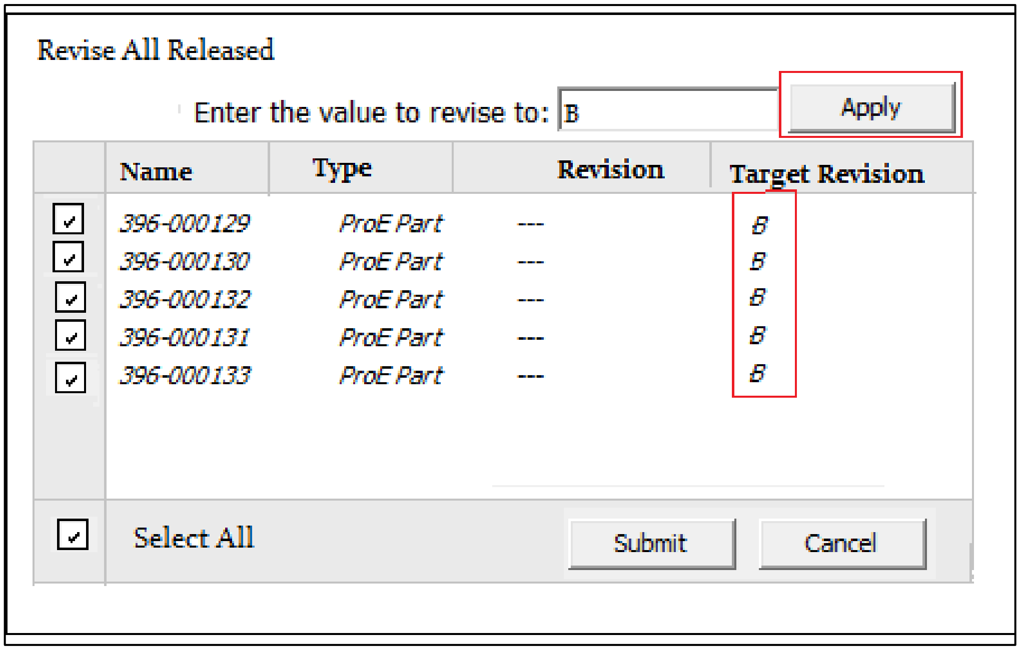


- When ‘***Revise All Released***’ dialog is launched, all the assemblies/parts from **Save** dialog which are in ‘**Released’** ‘**State’** will appear in ‘***Revise All Released’*** dialog and all will be selected by default, and the revision of those assemblies /parts will be automatically displayed to NEXT revision in ‘Target Revision’ column.
- The ‘***Revise All Released’*** dialog has ***Name, Type ,Revision*** and ***Target Revision*** columns in it.

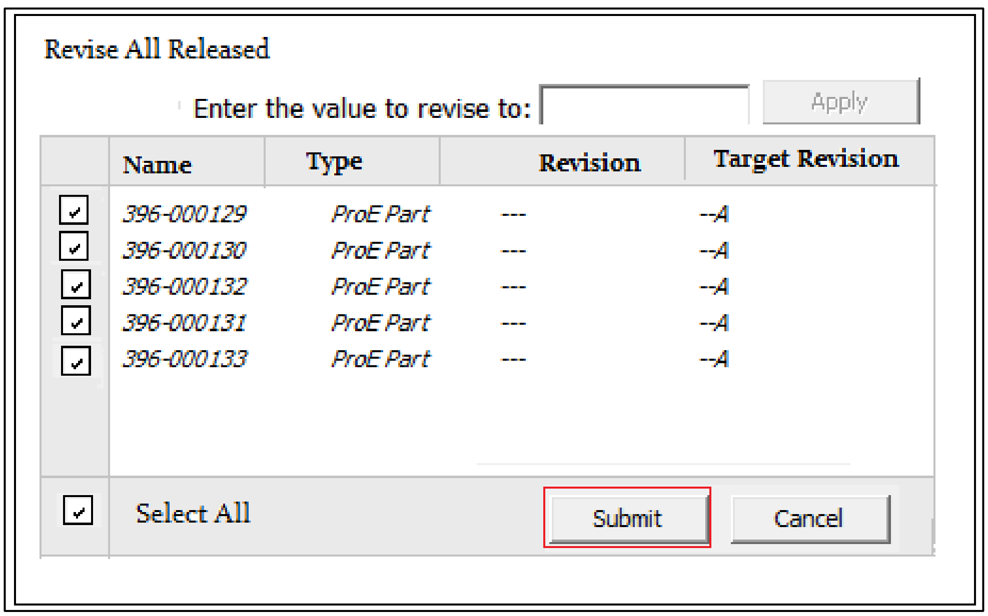
- If user does not want to go with NEXT revision and wants to change the revision by entering revision value manually ,then there will be an option “***Enter the value to revise to***: “ to type the desired revision in edit box and an ‘***Apply’*** button.



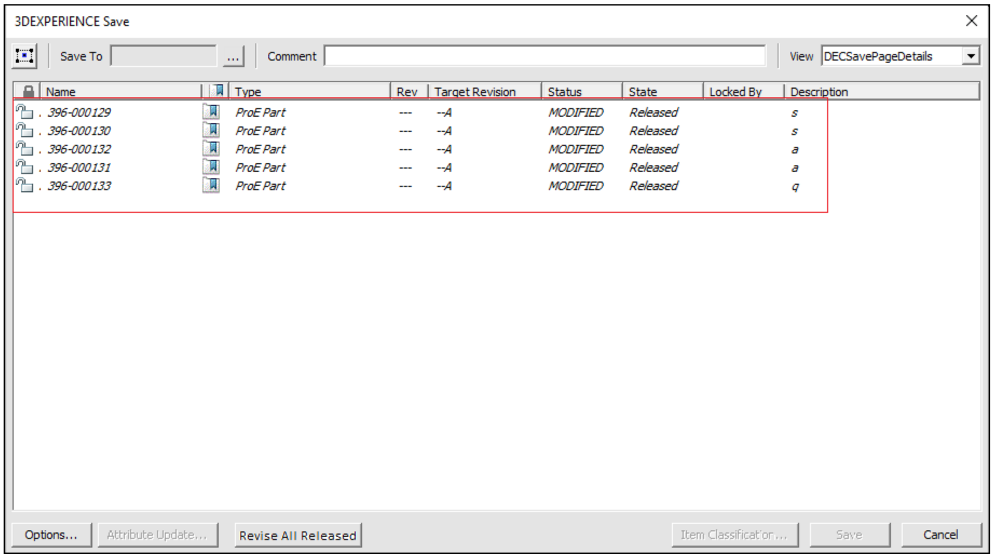
- The ‘***Apply’*** button is disabled by default, when user enters revision value in edit box, the ‘Apply’ button will get enabled.
- When user clicks on ‘***Apply’*** button, the ‘***Target Revision’*** column gets updated with entered value, this change will happen only on selected assemblies/parts.



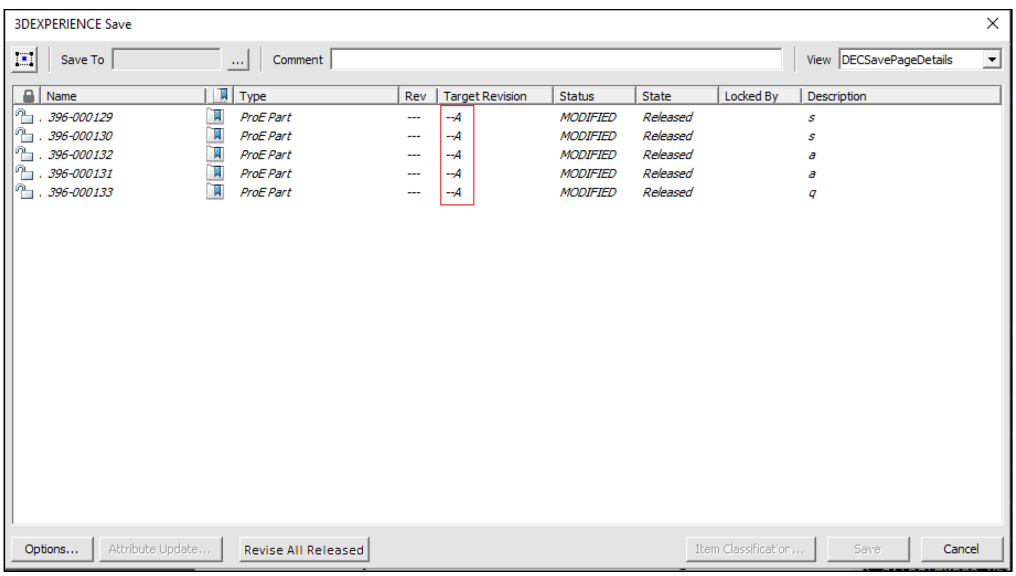
- To save all these changes done in '***Revise All Released'*** dialog, user must have to click on '***Submit'*** button.



- If user doesn't want to save the changes then he can click on '***Cancel'*** button. This will ensure there is no new Revisions applied to the "Released" objects and the control is sent back to the SAVE dialog
- After he clicks on '***Submit'*** button, '***Save'*** dialog gets updated with changes done in '***Revise All Released'*** dialog.



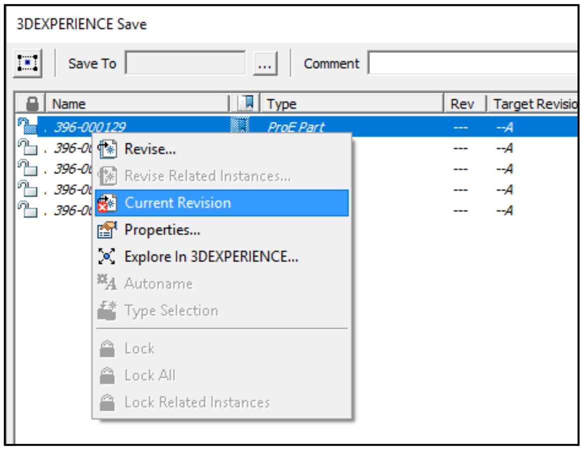
- Update to Save dialog after click of '***Submit'*** button of '***Revise All Released'*** dialog.
    - In '***Save'*** dialog, all the assemblies/parts which are in ***Released*** State are now selectable (not grayed out).
    - The '***Target Revision'*** get updated to new revision as per changed in ***'Revise All Released'***
    - Now User can select all the Assemblies/Parts and the ***Save*** button get enabled, click on **Save** button from **Save** dialog to perform check in operation.

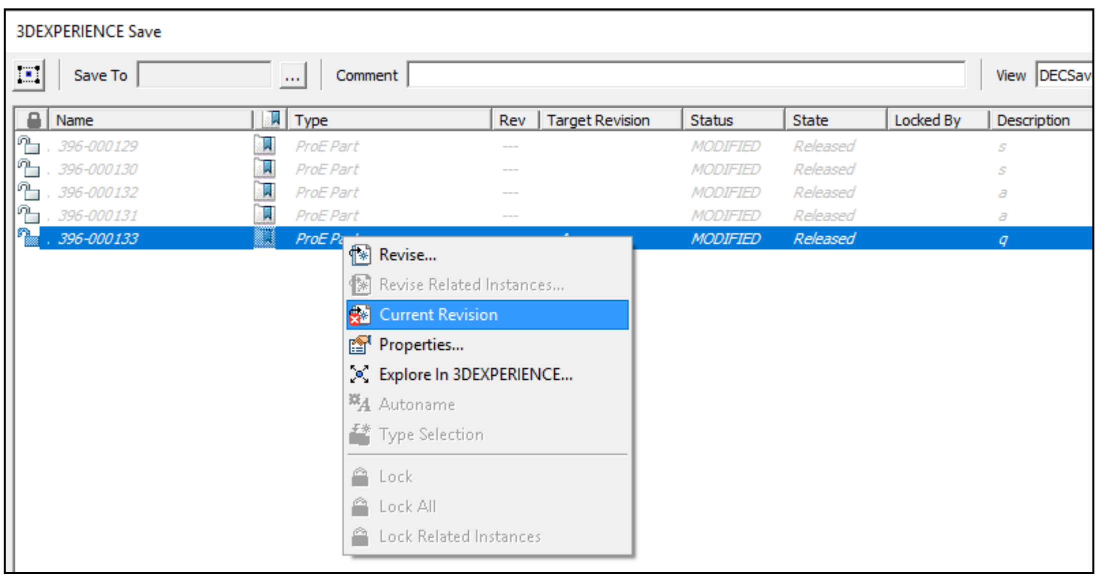


Undo Revise All Released

After the user performs the operation in the "**Revise All Released**" dialog and for some reason the user intends to undo revise for one of the Released CAD designs, there will be 2 ways to perform Undo Revise All Released

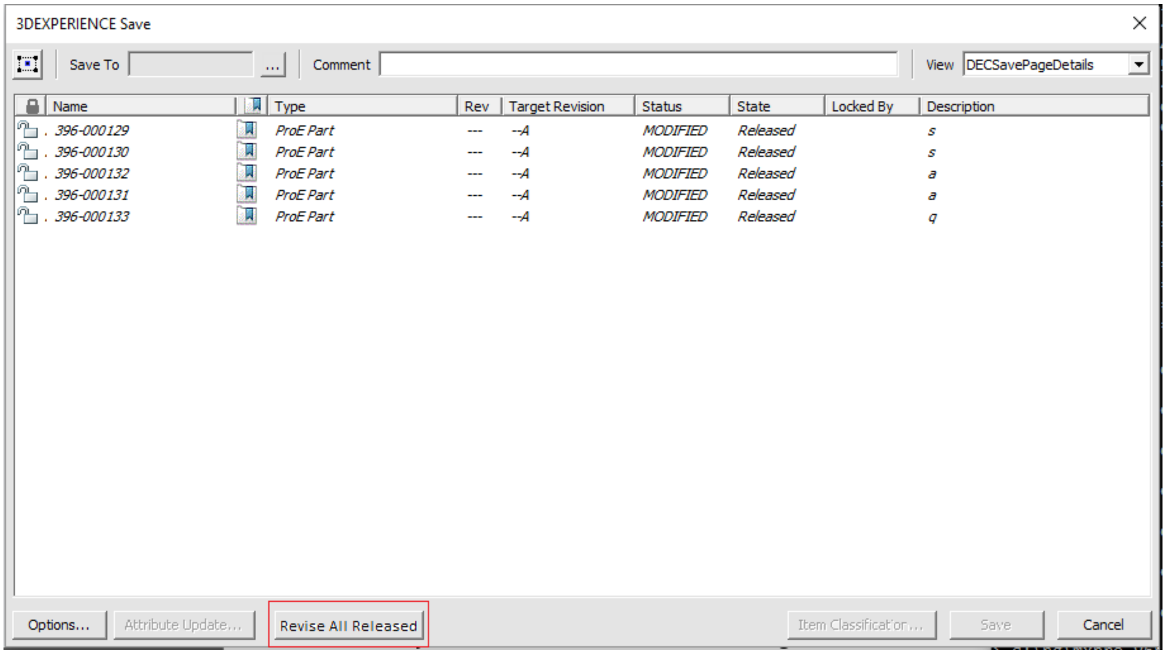
- Integration is having **Current Revision** option in Save dialog, on click of right mouse button of object it does the undo revise operation.
  - Once in ***Save*** dialog, all the released assemblies/parts are revised properly, and if user wants to undo the ***Revise All Released*** operation, user can click on each object and can do ***RMB→Current revision***, it will set its previous revision and that object is again grayed out.



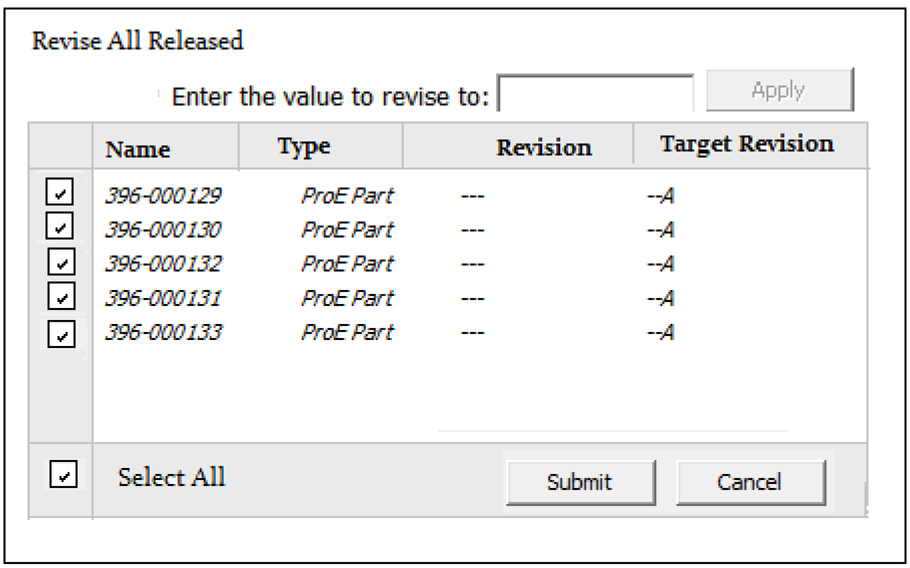


  - This is also done one by one.

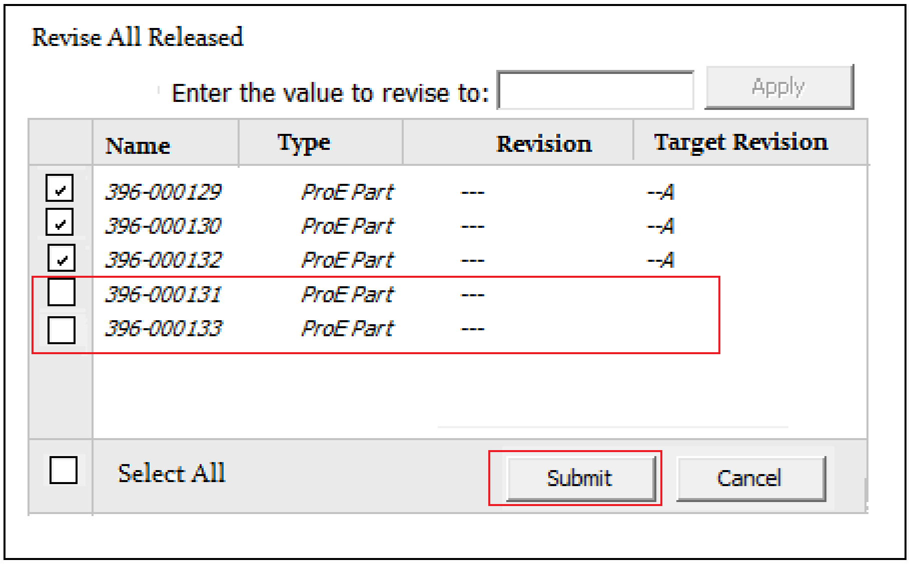
- Or either user can go back to ***Revise All Released*** dialog and deselect all the objects , it will undo the revise operation, and click on ***Submit***.
  - First user will revise all the objects in ***Revise All Released*** dialog, and click on ***Submit*** button, it will come back to **Save** dialog and dialog will look like:



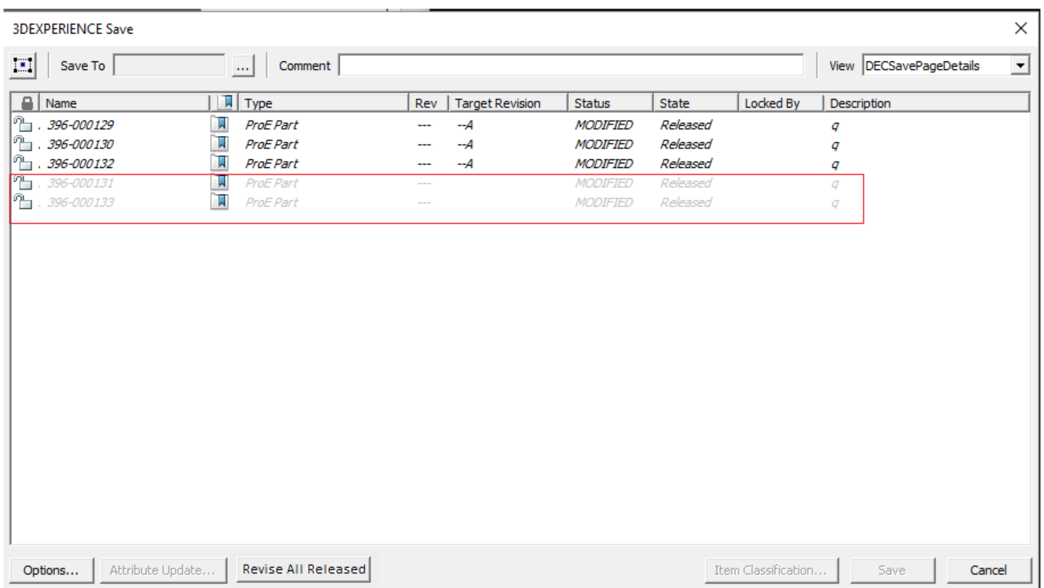
- Again, user wants to roll back the ***Revise All Released*** operation , then user will again click on '***Revise All Released'*** button, the ***Revise All Released*** dialog will launch.



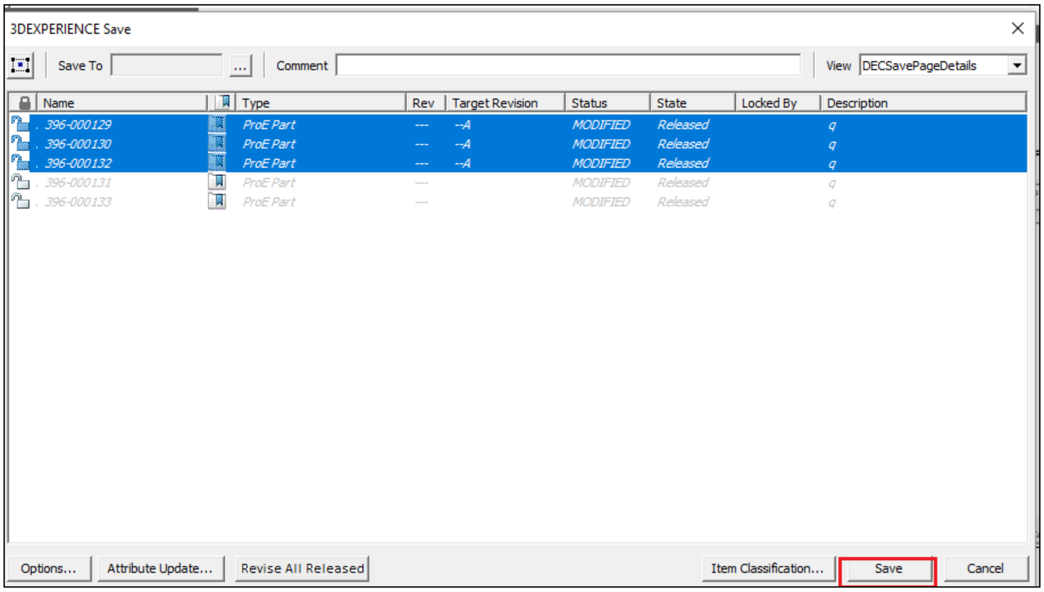
- User will deselect the checkbox of required assembly/part whose revision not to be changed and then click on ***Submit*** button.



- Then in ***Save*** dialog , those objects appear as grayed out/ non-selectable and rest of the(selected in ***Revise All Released***) objects get revised.



- After these changes, select all revised items and click on ***Save*** button to perform check in operation.


3DEXPERIENCE Save
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396-000130
ProE Part

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MODIFIED
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396-000133
ProE Part

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Options...
Attribute Update...
Revise All Released
Item Classification...
Save
Cancel