

3DEXPERIENCE Connector for NX

User Guide

3DEXPERIENCE R2022x



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Overview

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

3DEXPERIENCE Connector for NX in a Nutshell

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

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

From within 3DEXPERIENCE Connector for NX, 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 NX Install and Administration Guide* and *3DEXPERIENCE Connector for NX 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 NX.

The [Using Templates](#) section describes the procedure to create new designs using predefined templates.

The [Saving Designs in 3DEXPERIENCE](#) section describes the procedure to save a design to the 3DEXPERIENCE database from NX.

The [Derived Output Enhancements](#) section describes enhancements done to the derived output generation process during checkin.

The [Interpart Expression Support](#) section describes support to the interpart expression CAD feature.

The [Using 3DEXPERIENCE Open or Insert Option](#) section describes the procedure to search and open or insert designs that in 3DEXPERIENCE database in NX.

The [Lock/Unlock](#) section describes the procedure to lock/unlock NX design in 3DEXPERIENCE.

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 [Open Examine](#) section describes the feature to examine the structure of selected iteration of a design based on the configuration chosen during checkout process.

The [Open Partial](#) section describes the feature to process partial structures without affecting the structure integrity.

The [Update PLM Parameters During Save](#) section describes PLM attributes getting updated in the CAD properties during Save so that these attributes can be configured to reflect in the title block of the drawing

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 NX.

The [Global Refresh](#) section describes the procedure to view the properties of an active structure in 3DEXPERIENCE from NX 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 [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 NX.

The [Disconnecting from 3DEXPERIENCE](#) section describes the procedure to disconnect from 3DEXPERIENCE.

The [3DEXPERIENCE Interface Description](#) section describes the menu commands that are available in NX and also within other 3DEXPERIENCE dialog boxes that are specific to 3DEXPERIENCE Connector for NX.

Getting Started

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

Connecting to 3DEXPERIENCE from NX

This task shows you how to connect from NX to 3DEXPERIENCE.

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

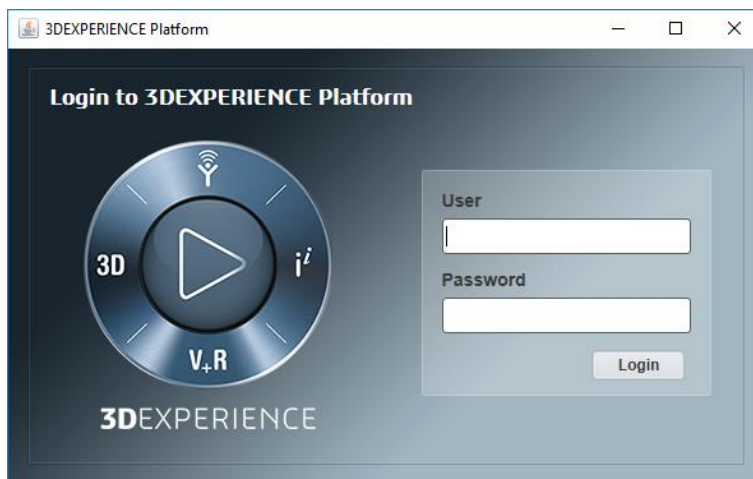
1. Click **3DEXPERIENCE > Login** in the NX tool bar.



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

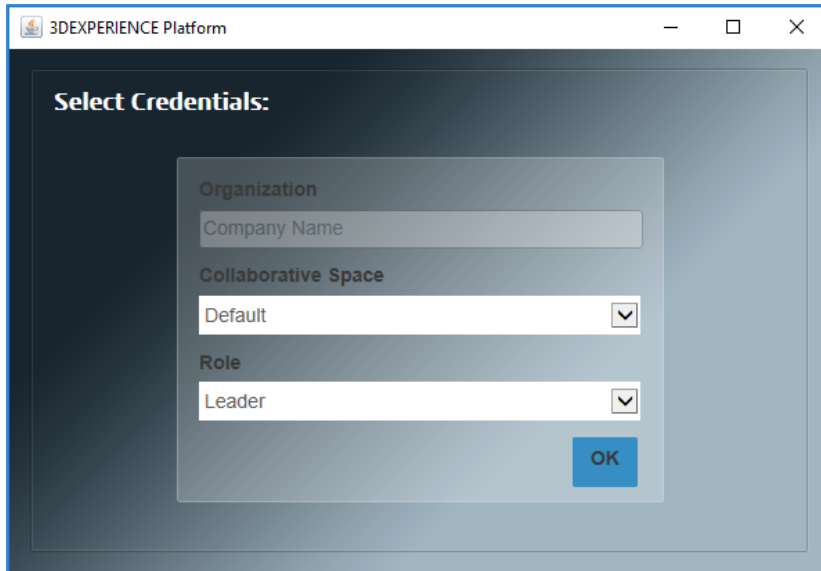
3DEXPERIENCE Login dialog box opens.

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 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 now logged in to 3DEXPERIENCE. To check whether your log in is successful, click 3DEXPERIENCE from NX toolbar. All 3DEXPERIENCE commands except Login are enabled



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.

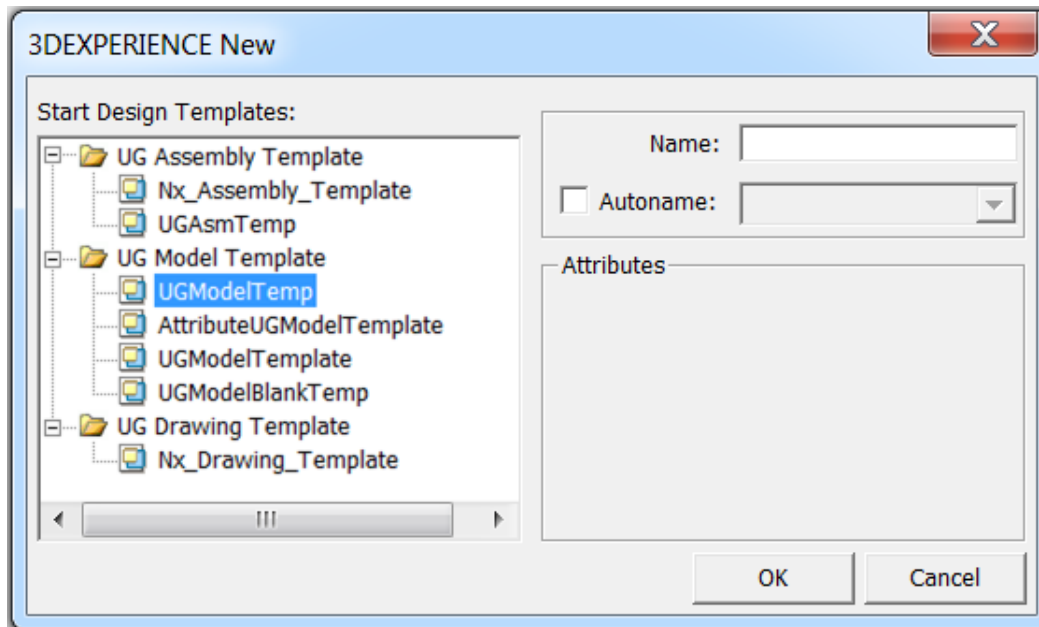
Creating Designs Using Templates

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 NX toolbar.

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 NX 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 NX 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 NX.

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 describes some concepts of the Save function and steps to save new designs, modified designs, configurations, and also save designs without using the user interface.

About Saving in 3DEXPERIENCE

The design information consists of the physical files created in NX plus metadata or descriptive characteristics. You must save the designs often to keep the collaborative group informed of any changes. When saving NX objects to 3DEXPERIENCE, 3DEXPERIENCE Connector for NX 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.

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 ENVOIA during save.

The integration will now add the default values of mapped attributes in the Mx-To-CAD GCO option to the NX 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 Connector for NX 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 NX 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 V2. When user A tries to save iteration "1", then the design is displayed as obsolete in **3DEXPERIENCE Save** dialog box.

Saving Designs to 3DEXPERIENCE

This feature is used to save new and modified designs to 3DEXPERIENCE store. 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.

Saving New and Modified Designs to 3DEXPERIENCE

This task shows you how to save new and modified designs to 3DEXPERIENCE database that are open in NX. There are two ways to save to 3DEXPERIENCE in NX. Save Active which saves only the current active model in session and Save All which saves all of the models open in session.

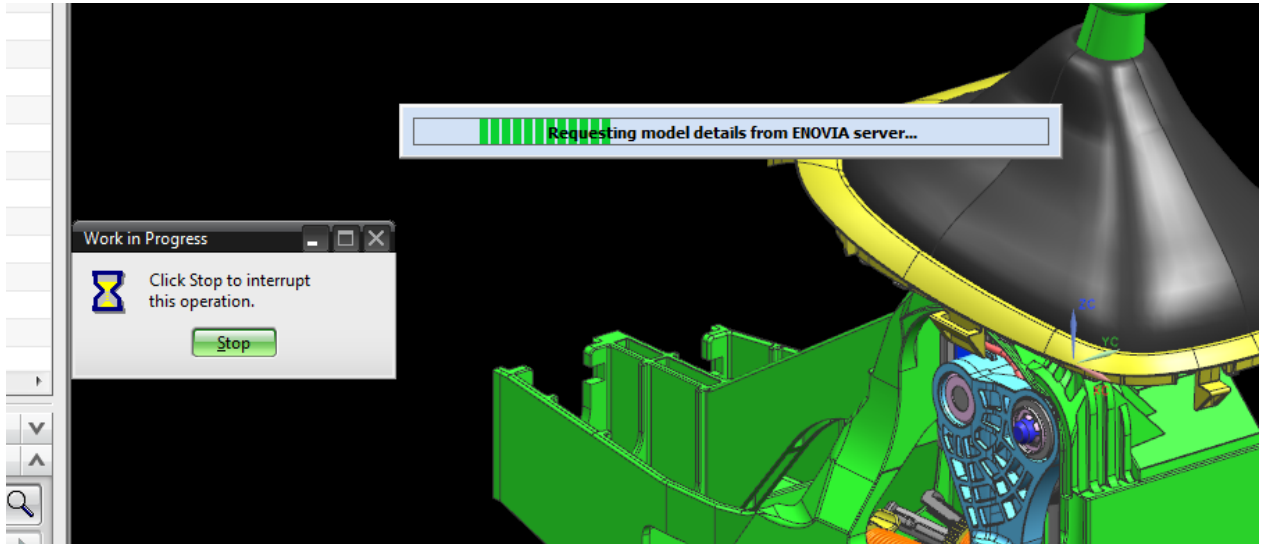
Ensure that the designs you want to save to 3DEXPERIENCE is open in NX.

1. Click **3DEXPERIENCE > Save > Active/All** from NX toolbar. 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 of the hidden windows, to the 3DEXPERIENCE.

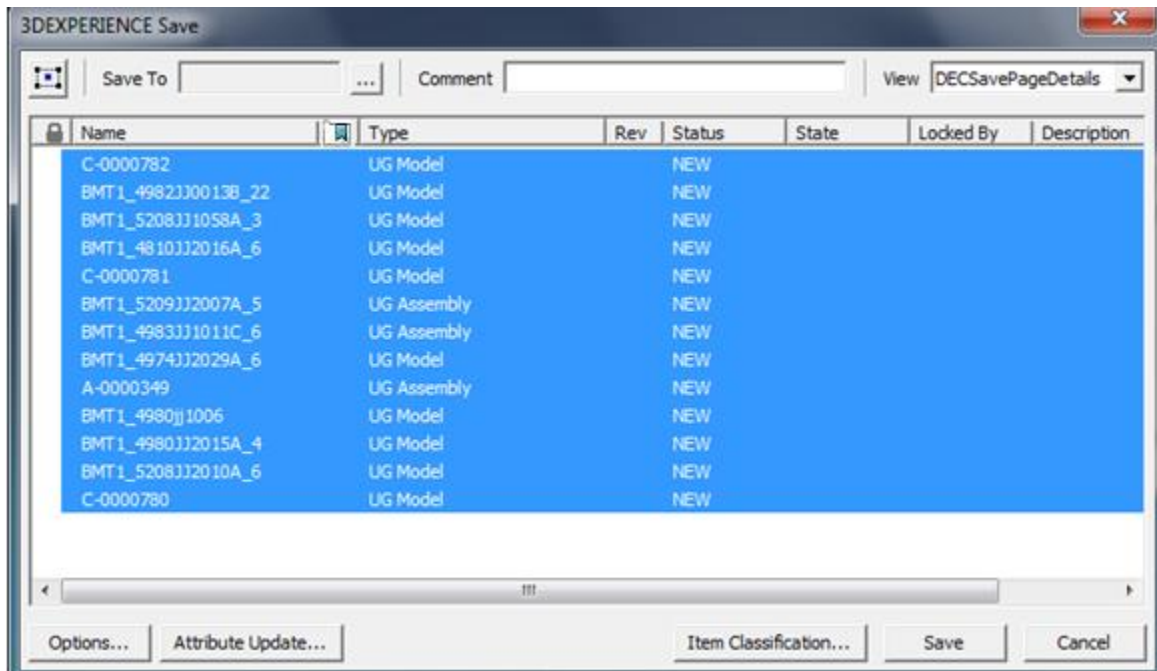


Note: An error message is displayed and **3DEXPERIENCE Save** dialog box does not open if there are no changes to be saved in the designs open in NX.

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.



Note: An error message is displayed and **3DEXPERIENCE Save** dialog box does not open if there are no changes to be saved in the designs open in NX.

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

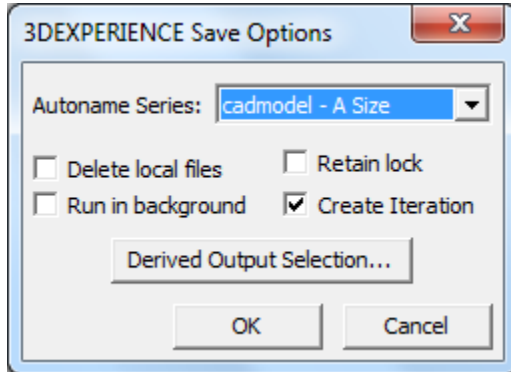
- . Displays the lock status of the design in 3DEXPERIENCE.  icon is displayed if the design is locked by you or any other user.  icon is displayed if the design is not locked in 3DEXPERIENCE. For a new design, no icon is displayed.
- **Name.** Name of the design.
-  This icon is displayed for designs that are saved to 3DEXPERIENCE 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. NX types are: UG Model, UG Assembly, and UG Drawing
- **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.** Displayed if the design does not exist in 3DEXPERIENCE.
 - **Modified.** Displayed if the design is modified in NX since the design was last saved or opened.
 - **Obsolete.** Displayed if newer iteration of the design exists in 3DEXPERIENCE database.
- **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 NX 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 in Save dialog.
- **Description.** Description entered for the design by the user.
- If a column name of the Save dialog matches the attribute name of a NX model present in the Save dialog, the integration will show the CAD value (File->Properties->Attributes of the NX 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](#) for more details.
- **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:



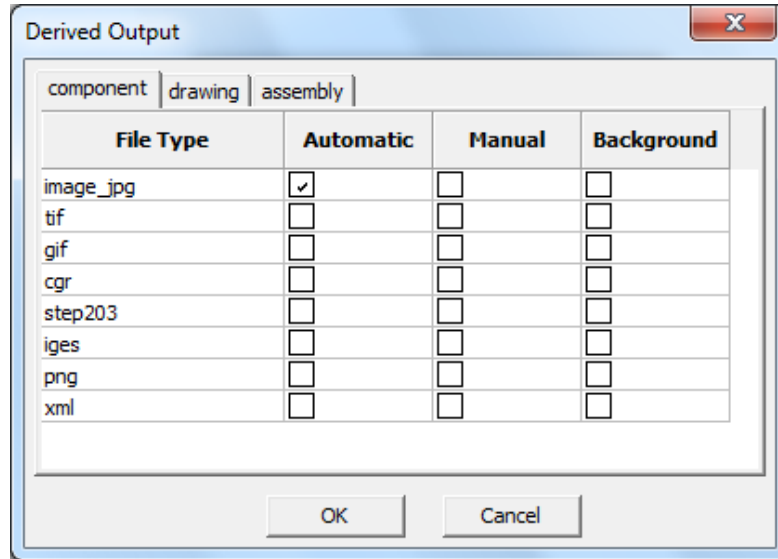
Note: To support your business processes, your Integration Administrator may have defined some options, so they cannot be changed. If the options are modified during a save operation, these parameters will return to their default setting during the next save operation. To permanently change these settings, 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](#) for more details.
- **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 NX operations. Copies of the files to be saved to ENVOIA are made and control is returned back to NX, so you can continue to use NX 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.
- **Derived Output Selection** button. Select to display Derived Output dialog box.



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

- component
- drawing
- assembly

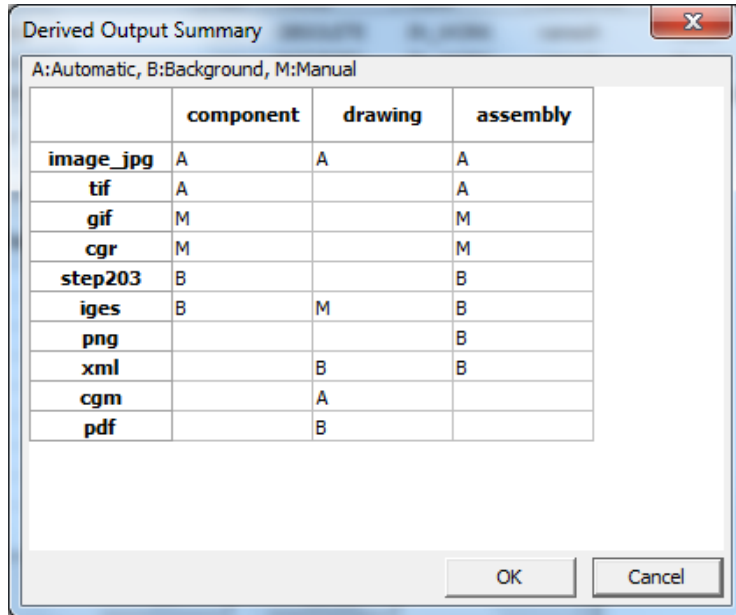
Select or de-select check boxes to generate the desired derived output files during the Save operation.

The columns indicate how the derived outputs will be generated.

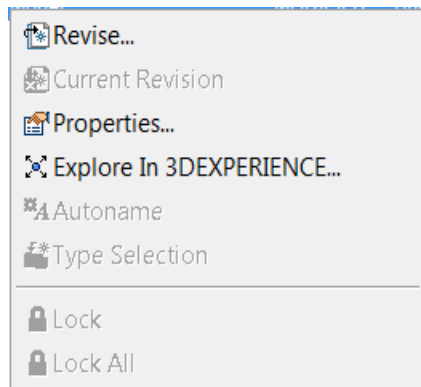
- **Automatic** indicates that the 3DEXPERIENCE Connector for NX will generate the specified derived output file during the Save operation
- **Manual** indicates that the 3DEXPERIENCE Connector for NX will look for an existing file in the same directory with the same base name as the file being saved, but with the appropriate extension (e.g., '.jpg', '.tif', '.gif', '.cgm', '.cgr', '.igs', 'ap203.stp', '.pdf', '.png')
- **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.



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



- **Revise...** Click to revise design. See [Revise Design](#) for more details.
- **Current Revision.** Click to set back revision to current revision. See [Revise Design](#) for more details.
- **Properties...** Click to see properties for a selected design. See [Viewing Properties](#) for more details
- **Explore In 3DEXPERIENCE...** Opens CAD Portal view of the design in the browser. See [Viewing Design Details in X-CAD Design](#) for more details.
- **Autaname.** Click to assign a name automatically to the selected designs. See [Using Autaname](#) for more details.
- **Lock.** Click Lock to lock the selected design. See [Locking and Unlocking Designs](#) for more details.

- **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. See [Locking and Unlocking Designs](#) for more details.
- **Lock All.** Lock All can be used to lock all of the unlocked designs in Save dialog at once. See [Locking and Unlocking Designs](#) for more details.

5. Click **OK** when done.

The new 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.

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

This functionality is implemented to block the User from continuing with save operation if dependent objects are not selected in Save dialog.

NX Integration will enable this functionality based on a configuration that needs to be enabled in the settings.ini file. The OOTB behavior of the NX Integration will continue to allow save operation even if all dependent objects are 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 Save operation if user is trying to continue with the operation by not selecting some of the dependent objects 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:

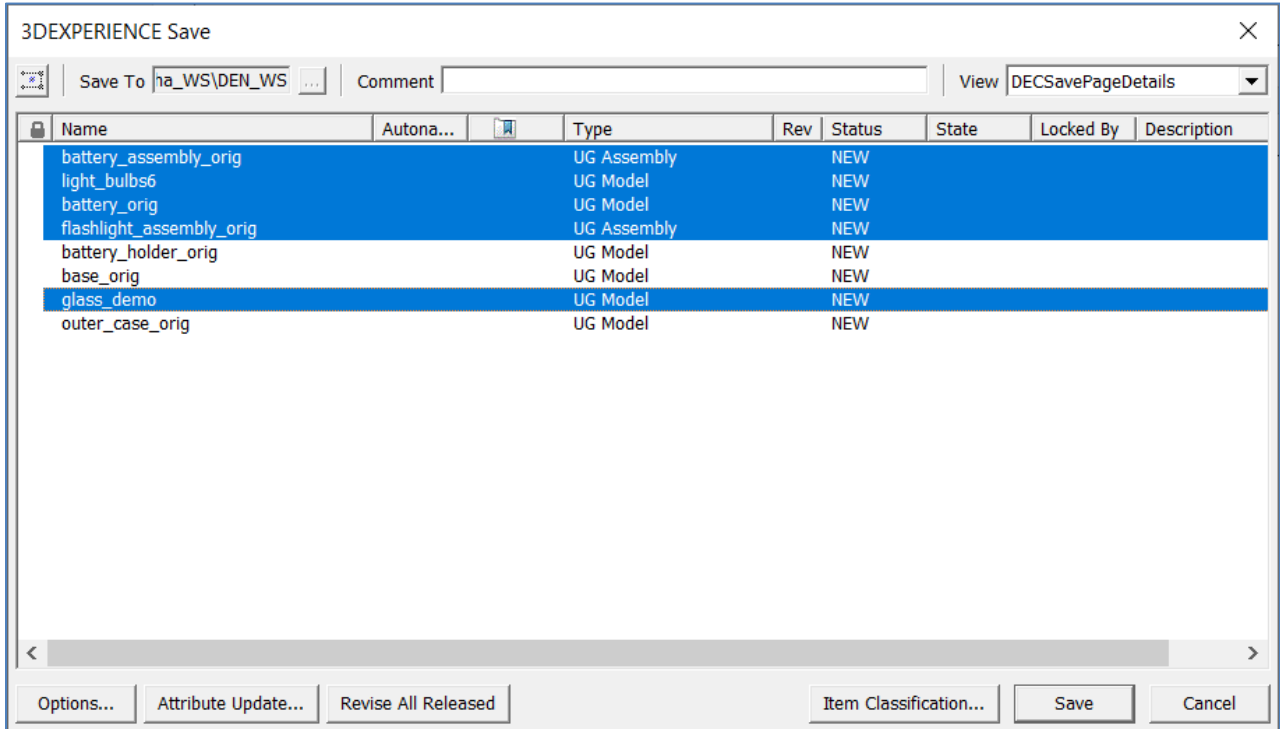
This functionality is implemented only for the Save operation invoked by using '3DEXPERIENCE>Save>Active' command from the integration ribbon.

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

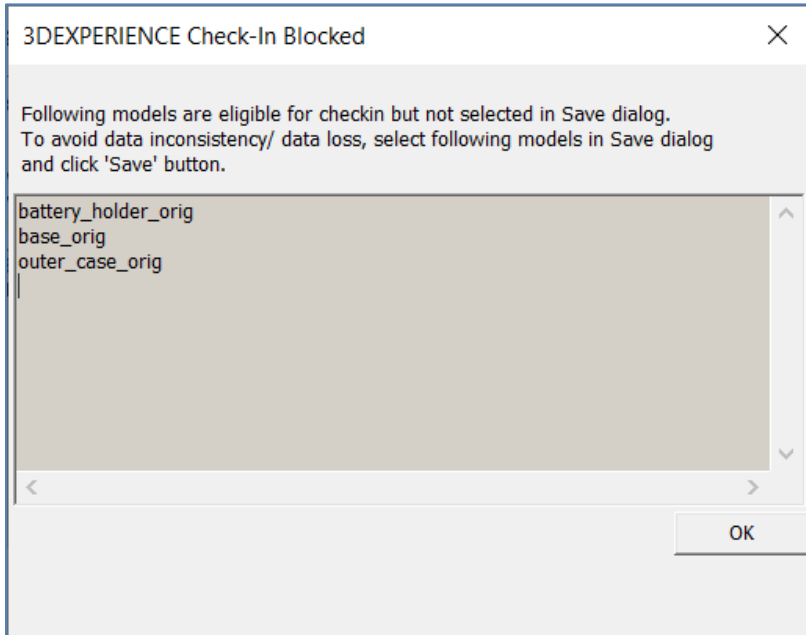
`NXForceSelectableUnloadedChildRelationships = 1 ;`

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.



NX1847 Support

NX 1847 release marks a major milestone in industry as it supports a continuous release process. This will make it easier for user to stay current with the latest release of NX, giving you convenient access to new functionality, as well as performance and quality improvements. NX will automatically check for updates, and you can control if and when you deploy those changes. In addition, this major release brings significant new and enhanced functionality in all areas of the product. It also delivers enhancements that enable you to increase your productivity in product development and manufacturing while working in a collaborative managed environment.

For every NX continuous release, new 4-digit release tracking number will be used. For e.g. NX 1847 is next major release after NX 12. 1847 is 4 digit tacking number represents siemens foundation year. Siemens will be providing monthly releases as per the enhancements and customer need and IR/SR processed for that release.

Below are the monthly releases available for NX 1847 series:


- NX 1859
- NX 1855
- NX 1851

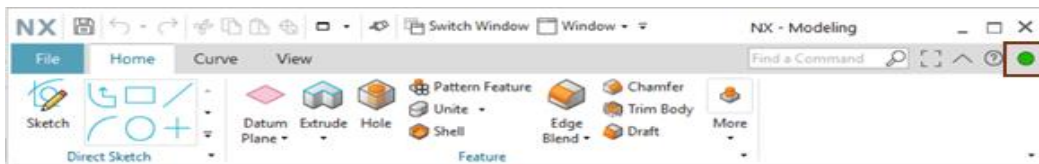
If you install any of the above patch, then it will get installed in 1847 installation folder itself. No separate folder will be created. These releases are treated as patches on top of NX 1847 installation. Next major release after NX 1847 will be NX 1872 and all other are intermediate release for maintenance or monthly enhancements.

Automatic installs and updates

All new features, enhancements, and quality improvements in NX are now delivered in a continuous fashion.

The following features are introduced to support the new process:

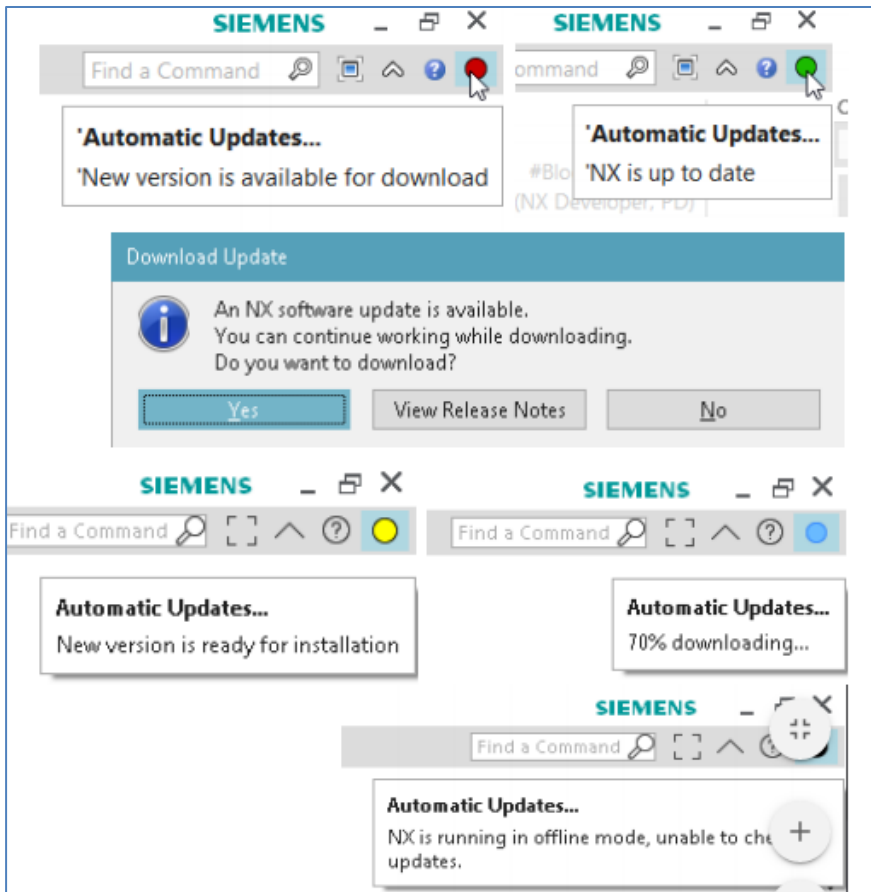
The update notification icon  is available on the ribbon of the NX interface. The color of the icon indicates the status of the NX installation and the availability of software updates.



This color	Indicates
Green	The NX software that is installed on the workstation is up to date.
Red	One of the following: A software update is available and not yet downloaded on the workstation. An important software update will be available soon.

This color	Indicates
Blue	An update is downloading.
Yellow	A software update is downloaded but not installed on the workstation.
Grey	One of the following: The software update notifications are disabled. You can use the Software Update customer defaults to enable the notifications. The NX software is not installed using a .msi file. The NX software is not part of the main channel.
Black	The NX software cannot connect to the update server.

You can point the cursor to the notification icon to know more about actions required from the user.



- You can download an update by clicking the notification icon.
- NX checks the user authentication before downloading the updates.
- By default, the automatic notifications and downloads are disabled.

UGII_ROOT_DIR obsolete

UGII_ROOT_DIR has historically been used to find the following in the UGII directory:

- Libraries and executables
- Configuration files

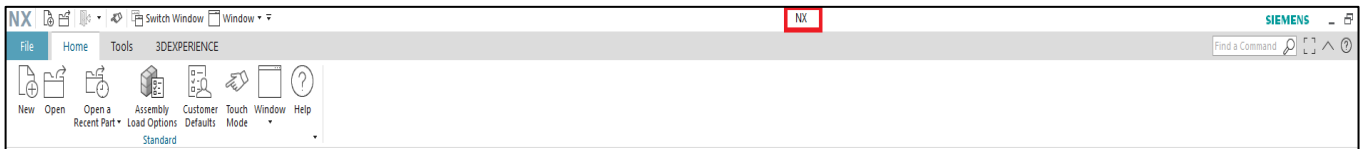
With the creation of the central runtime directory, there are two different locations for these files (nxbin and ugii), so the concept of UGII_ROOT_DIR is obsolete.

Use the UGII_BASE_DIR variable instead when you write custom application code and scripts.

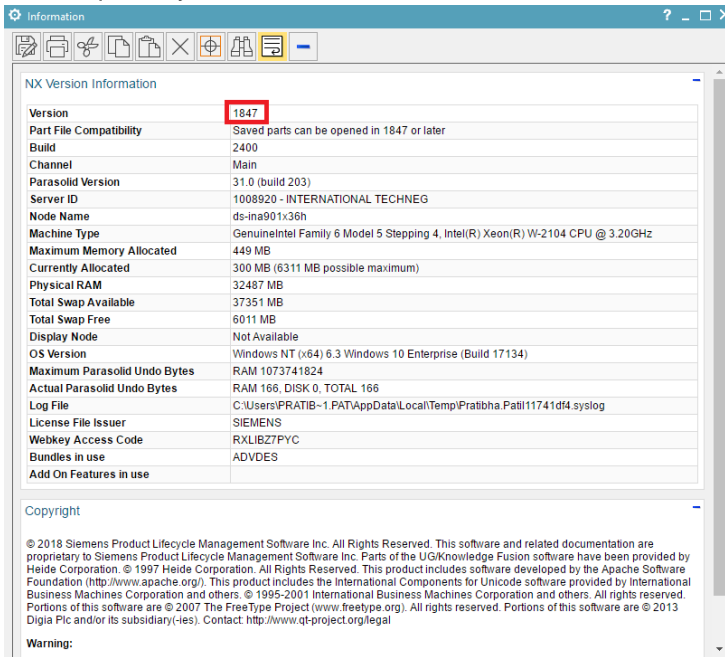
While the NX install no longer sets UGII_ROOT_DIR, the definition on the system will remain to support earlier releases. If your code or scripts rely on UGII_ROOT_DIR, modify them to ensure they work in NX 1847.

NX release version visibility

You can no longer see the NX release version number on the title bar of the NX window.

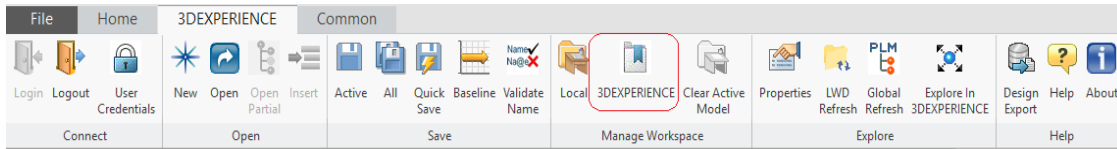


As before, if required, you can view it in locations such as the About NX dialog box and the Log File.





New Bookmark UI and Syntax

The new Menus and icons will look as shown below.



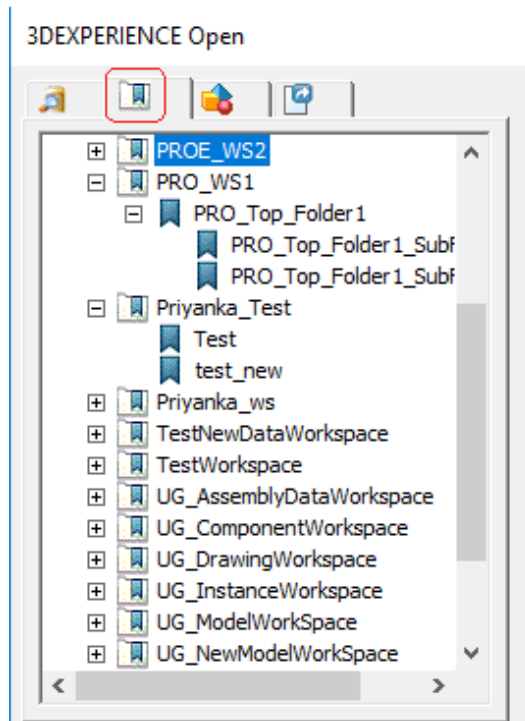
3DEXPERIENCE Ribbon


Note: 3DEXPERIENCE menu icon is replaced by new Bookmark icon 

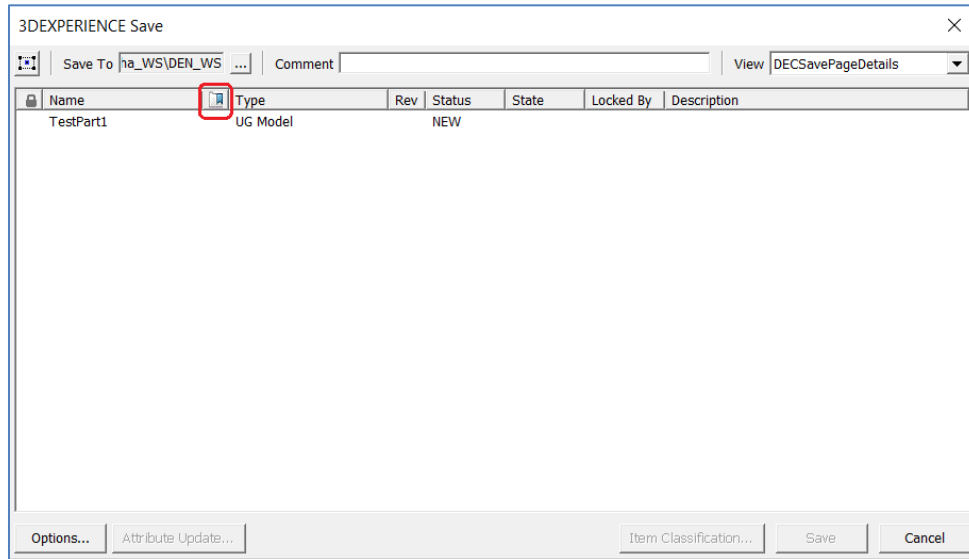
- Workspace is renamed to Bookmark Workspace and Workspace Folder is renamed to Bookmark Folder.
- Workspace Icon is replaced by new Bookmark Workspace icon 
- Open/Closed Folder icon is replaced by new icon 


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

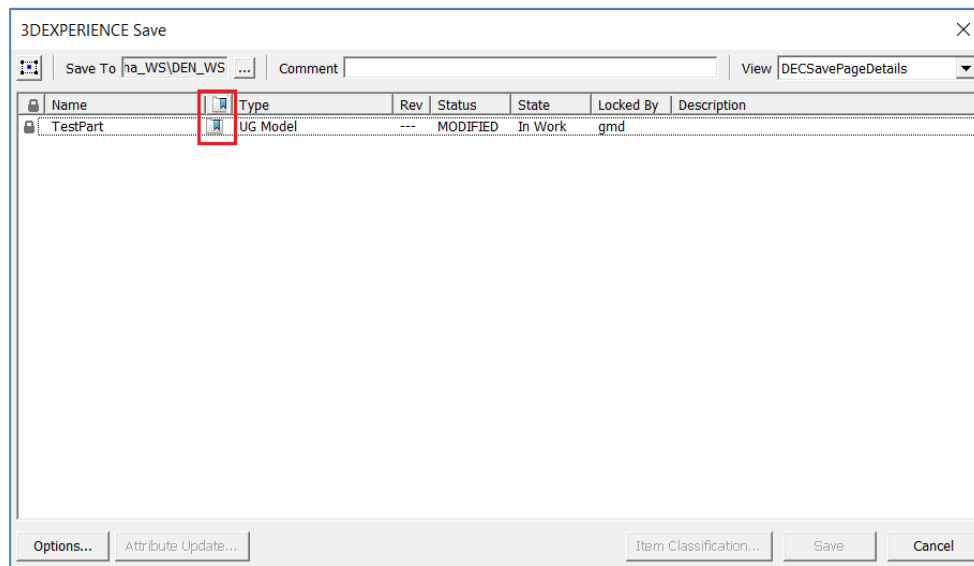
- Open Dialog Bookmark Workspace selection for browsing content:



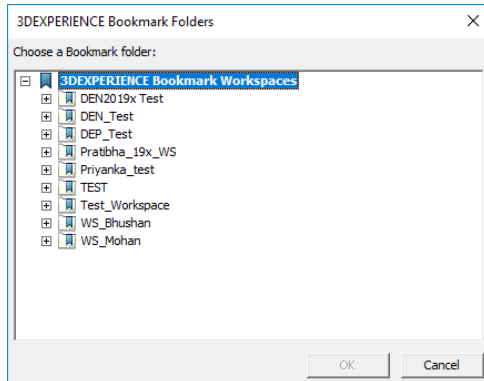
- When user performed save on newly created part then save dialog will be displayed with new Bookmark Workspace icon  as below.



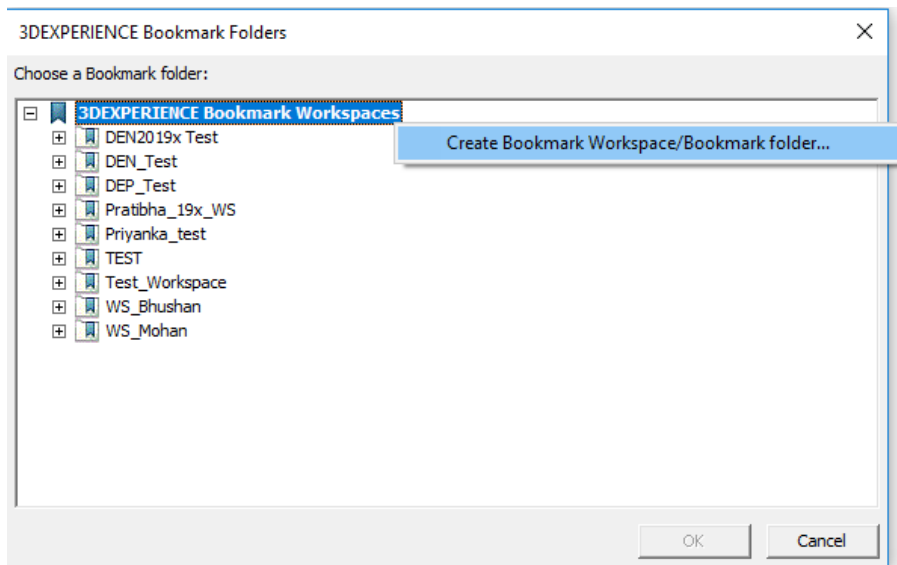
- When user performed save on existing part then save dialog will be displayed with new Bookmark Workspace icon  as below.



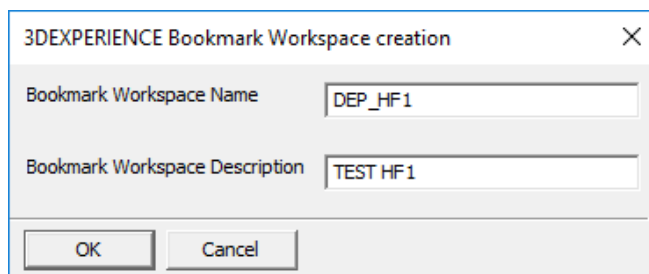
- New Bookmark Workspace UI

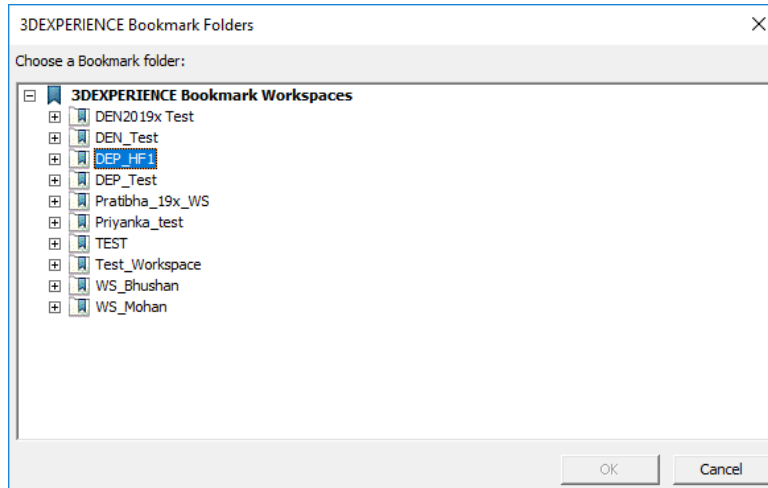


- The RMB command 'Create Bookmark Workspace/Bookmark folder...' can be used for creating a new Bookmark Workspace in 3DEXPERIENCE.

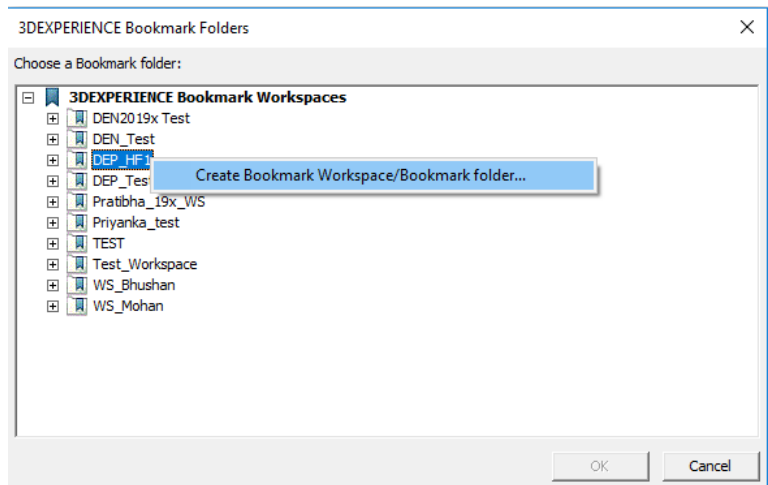


- Upon selecting the RMB command, 'Create Bookmark Workspace/Bookmark folder...', The new '3DEXPERIENCE Bookmark Workspace creation' will look like as below

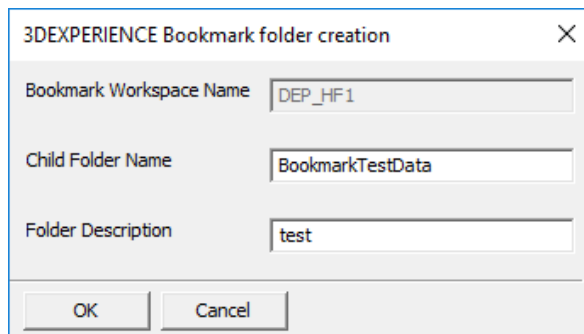


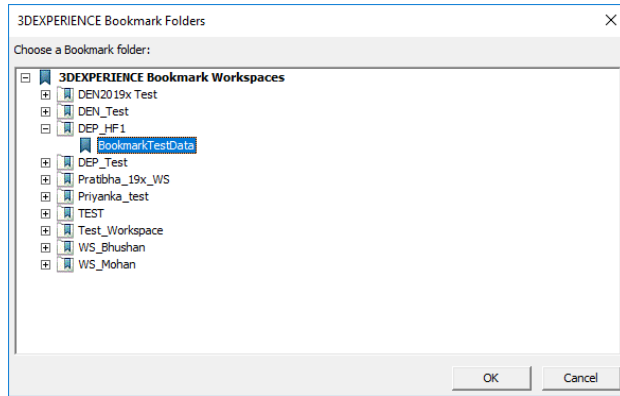


- On any existing Bookmark Workspace, the User can use the RMB command 'Create Bookmark Workspace/Bookmark folder...' to create a new Bookmark Folder.



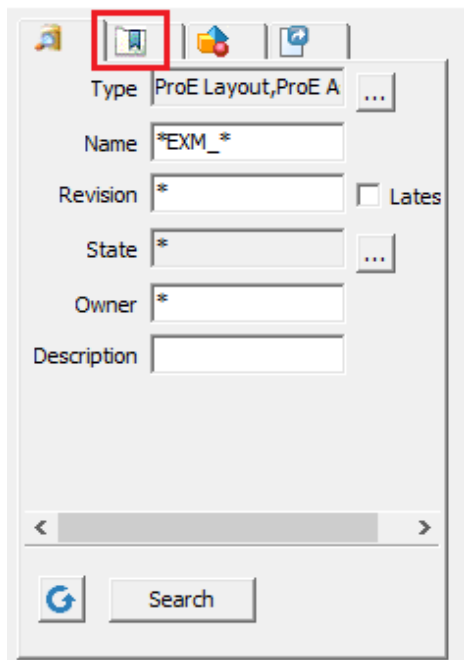
- Upon selecting the RMB command to create the Bookmark folder, following Bookmark folder creation dialog will be displayed.

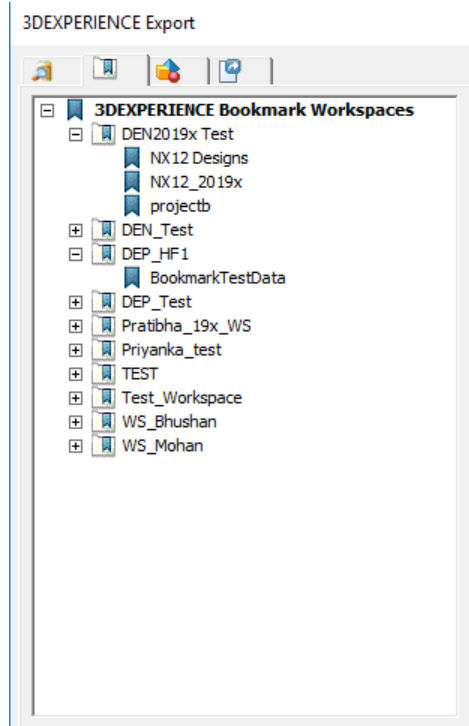




- Design Export Bookmark Workspace selection

3DEXPERIENCE Export





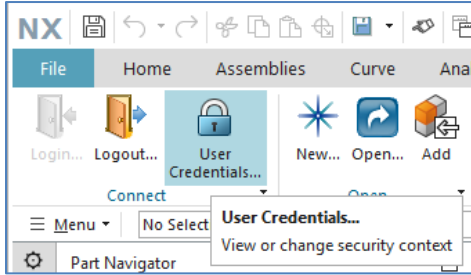
User Credentials

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

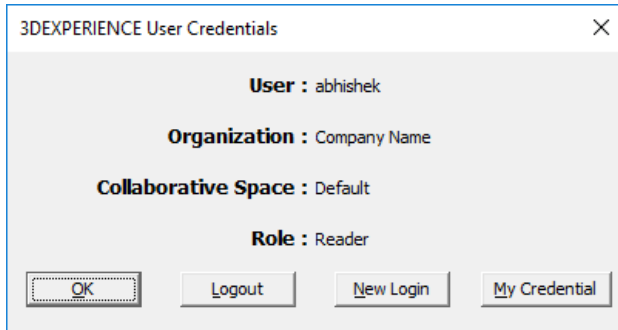
This functionality supports below options:

- 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:



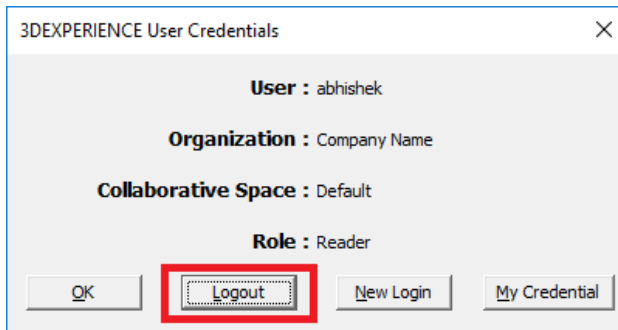
GUI displays existing User, organization, collaborative space and role assigned to user

Connect ->User Credentials ->OK

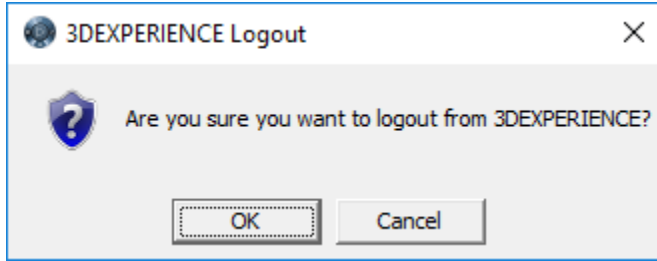
On OK button click user credentials UI will be closed.

Connect->User Credentials ->Logout

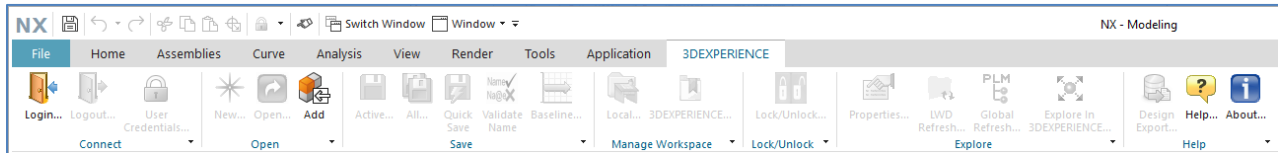
User can logout from the User Credentials popup dialog by selecting **Connect->User Credentials ->Logout** option



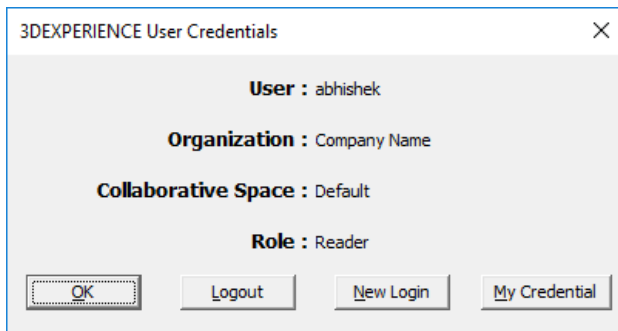
If user clicks on Logout button, then below confirmation window will be shown.



If user proceeds 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.



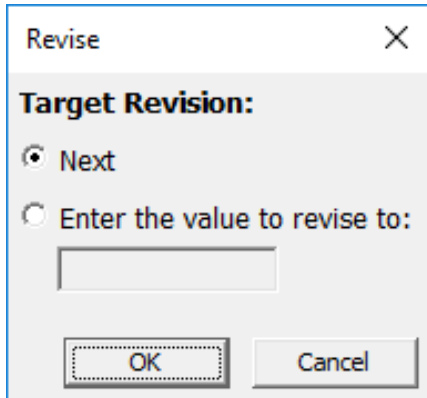
Connect-> User Credentials ->New Login

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

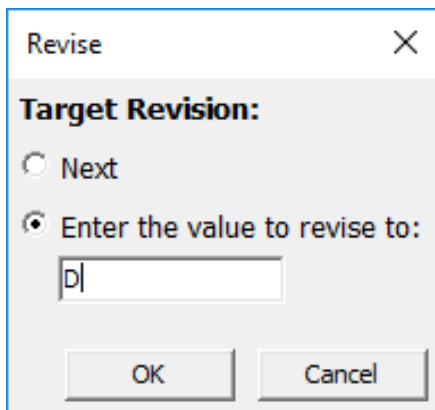
Revise Design

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

1. Right-click the design and click Revise to revise design to next revision. Select ‘next’ to revise NX 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.

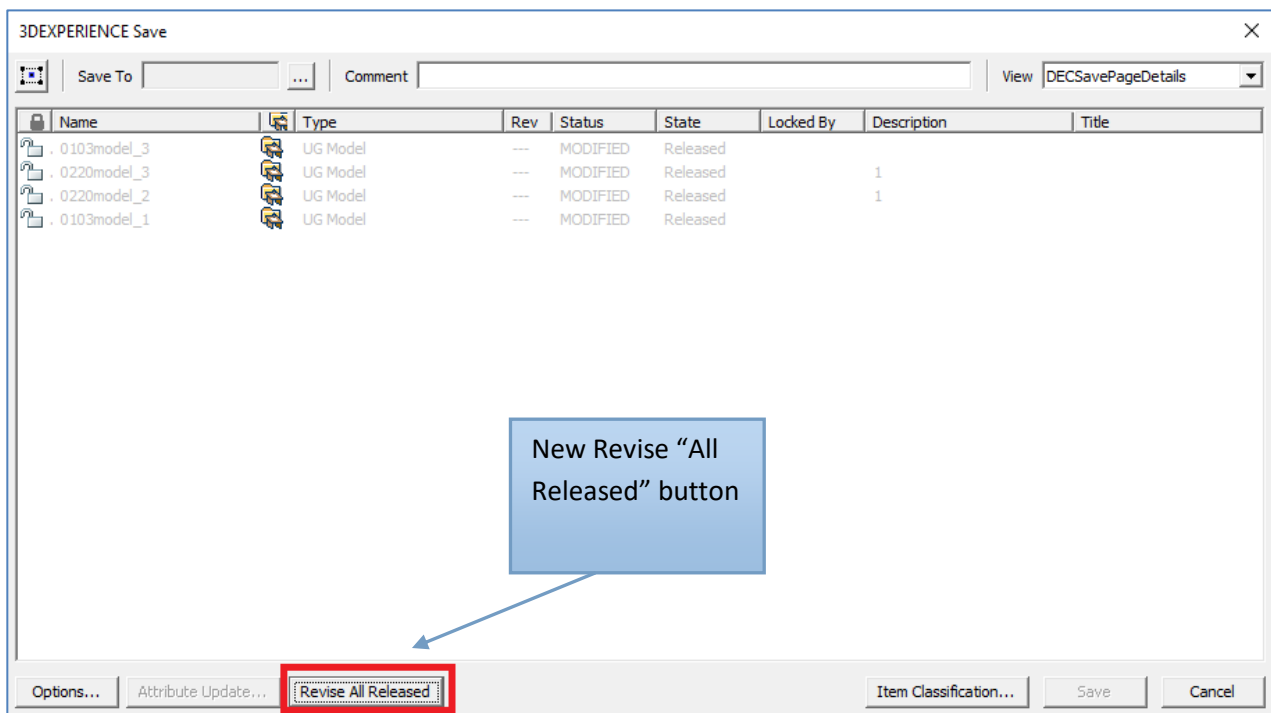
Revise All Released

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.

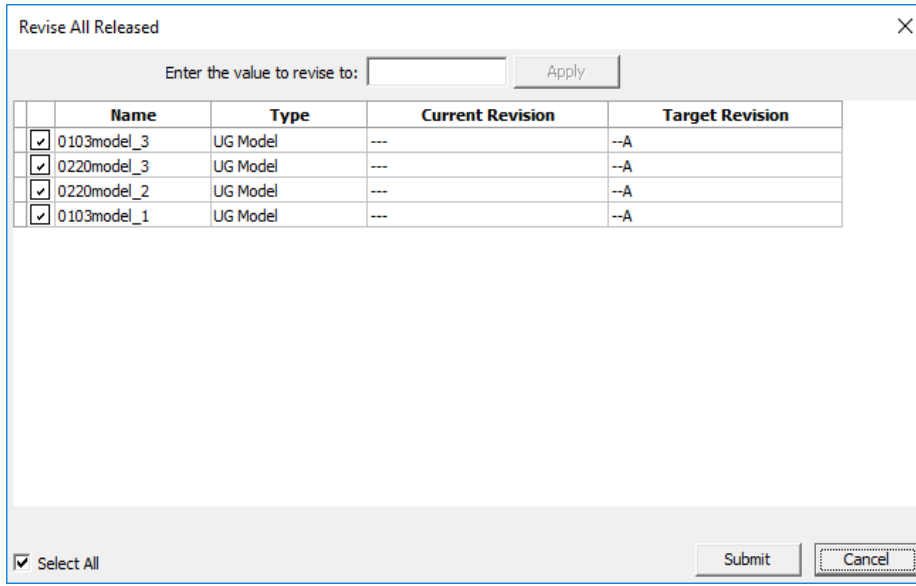
When a released assembly is checked out and modified and attempted to check in using 3DEXperience Save functionality, modified objects will appear in save dialog as a grayed out, '**Status**' being displayed as '**MODIFIED**' and '**State**' displayed as '**Released**' .

➤ Save dialog changes :

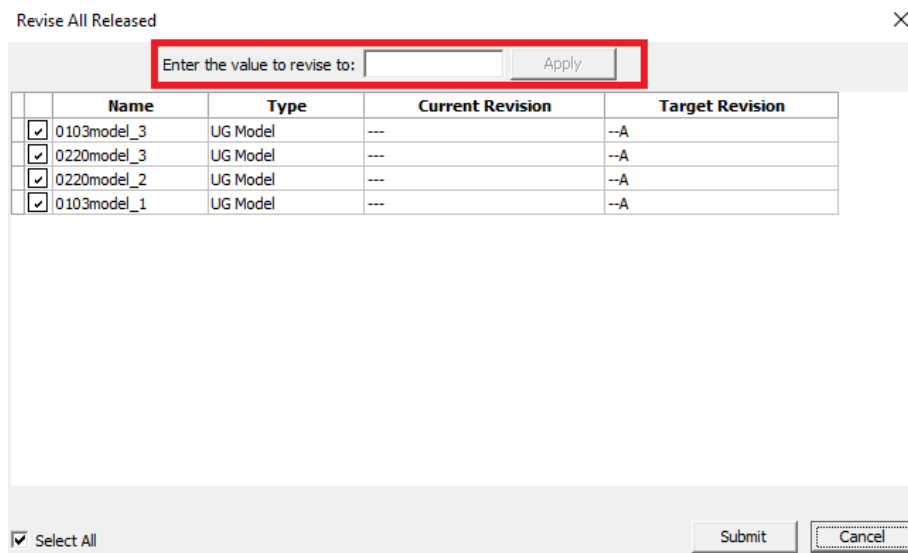
- New '**Revise All Released**' button available in save dialog for mass revise .



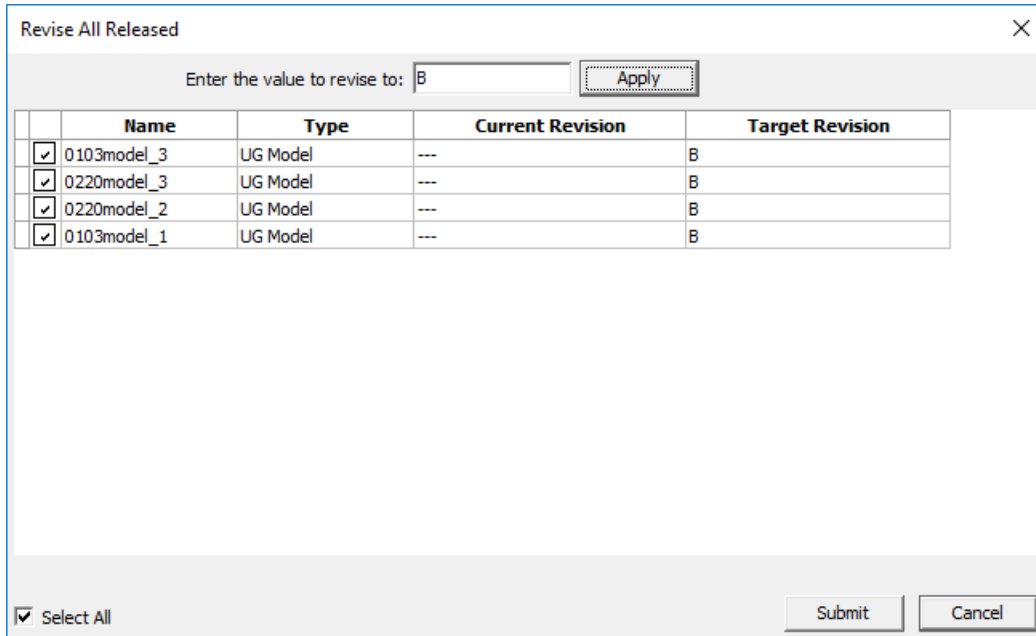
- Even though the objects in save dialog are grayed out, user can click on '**Revise All Released**' button, and user will see the '**Revise All Released**' dialog launched.
- New '**Revise All Released**' dialog will look like as:



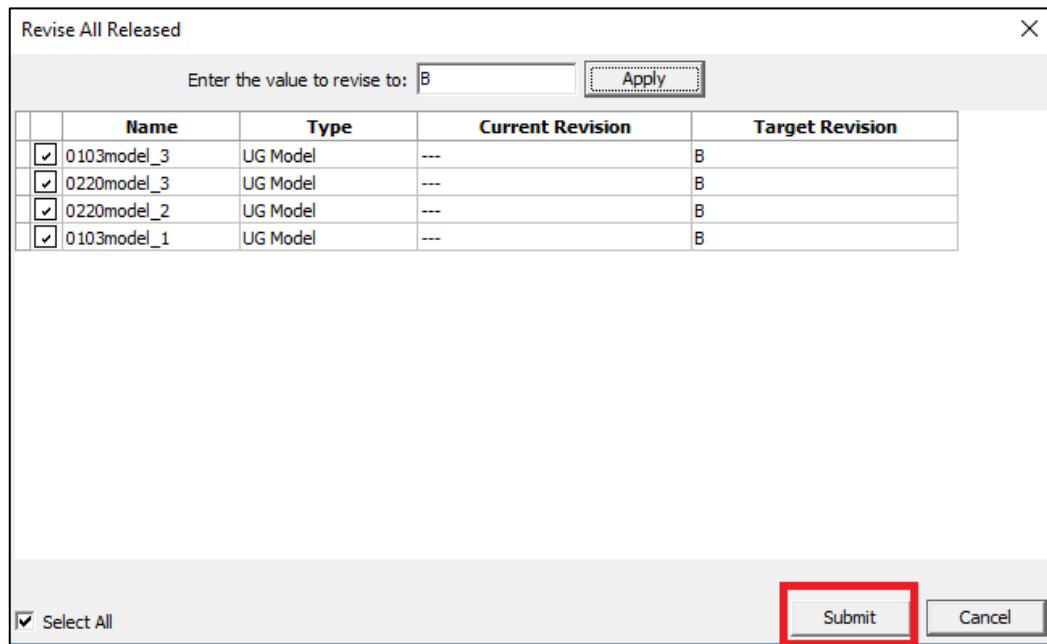
- When '**Revise All Released**' dialog is launched, all the assemblies/parts from **Save** dialog which are in '**Released**' '**State**' will appear in '**Revise All Released**' dialog and all will be selected by default, and the revision of those assemblies /parts will be automatically displayed to 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.



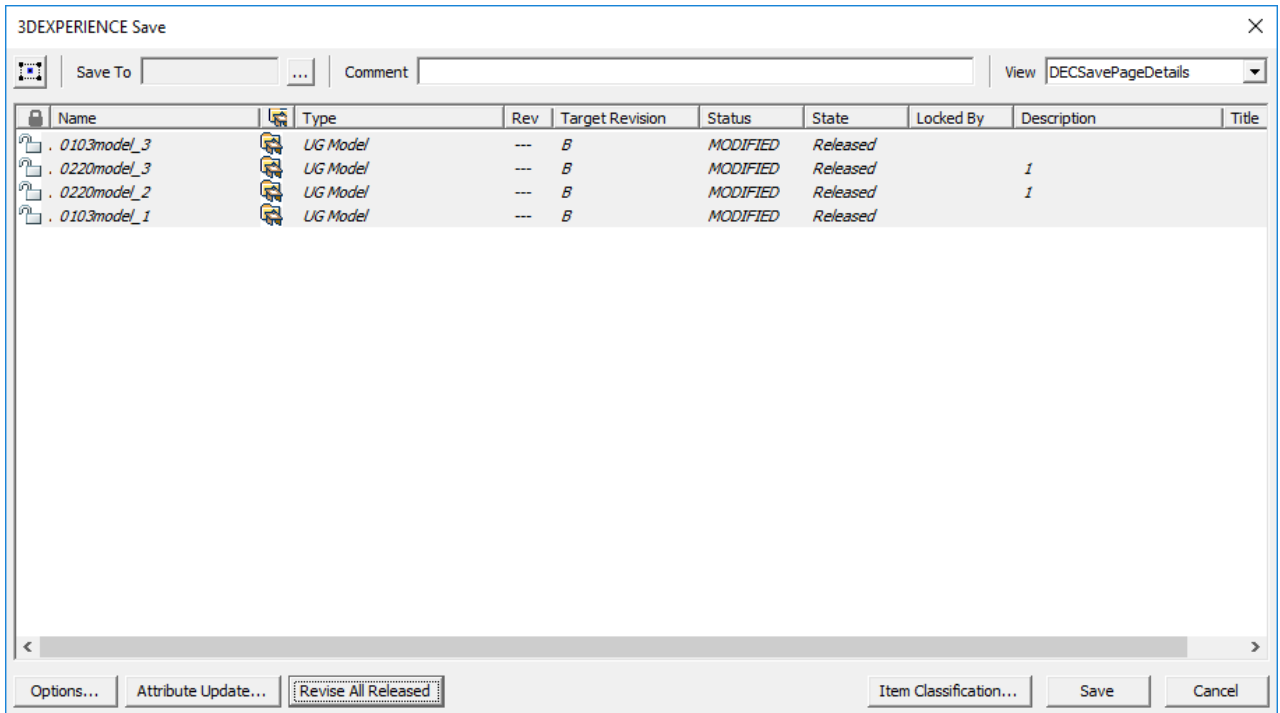
- The '**Apply**' button is disabled by default, when user enters revision value in edit box, the '**Apply**' button will get enabled.
- When user clicks on '**Apply**' button, the '**Target Revision**' column gets updated with entered value, this change will happen only on selected assemblies/parts.



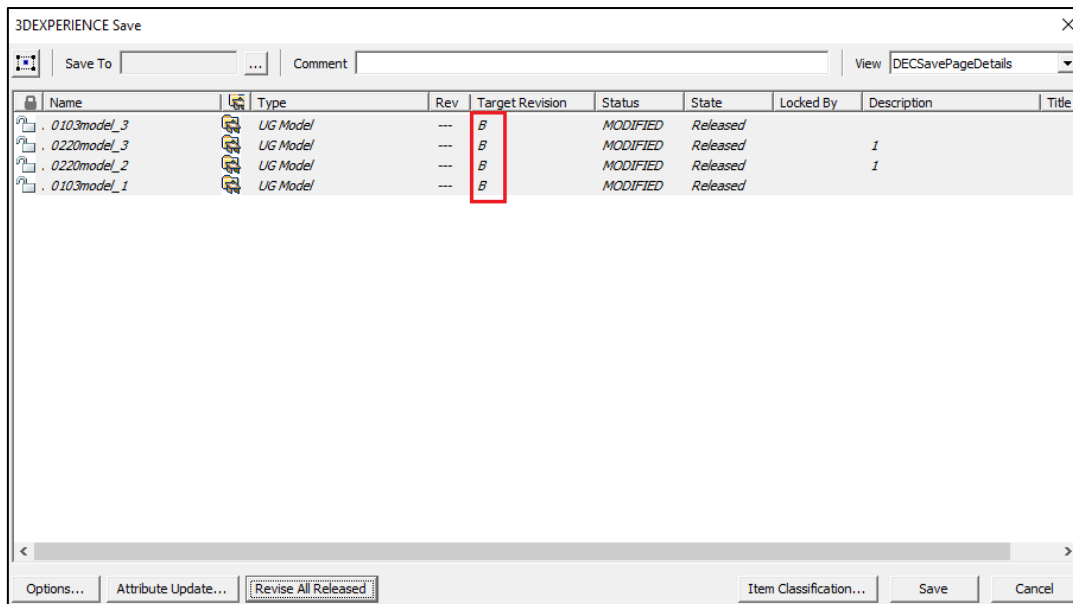
- To save all these changes done in '**Revise All Released**' dialog, user must have to click on '**Submit**' button.



- 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.



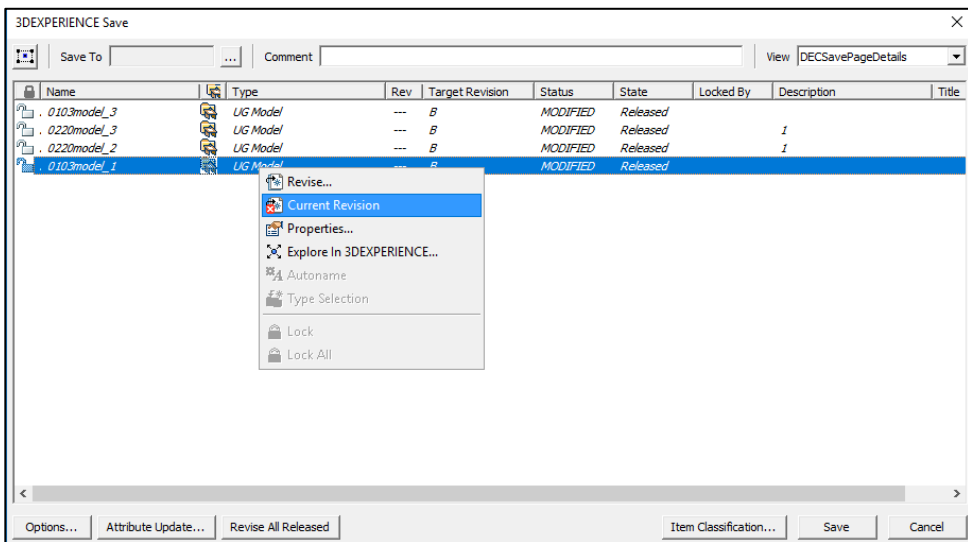
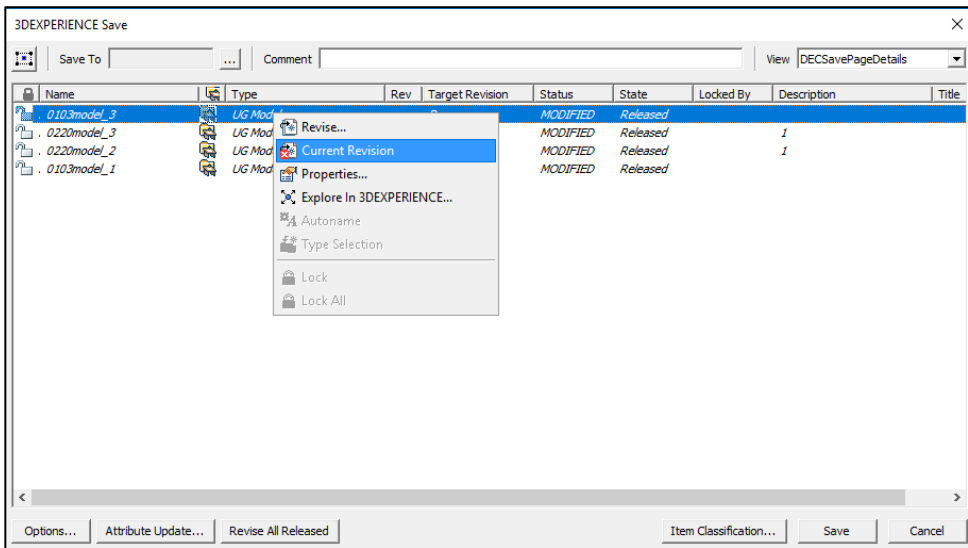
- Update to Save dialog after click of **'Submit'** button of **'Revise All Released'** dialog.
 - In **'Save'** dialog, all the assemblies/parts which are in **Released** State are now selectable (not grayed out).
 - The **'Target Revision'** get updated to new revision as per changed in **'Revise All 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

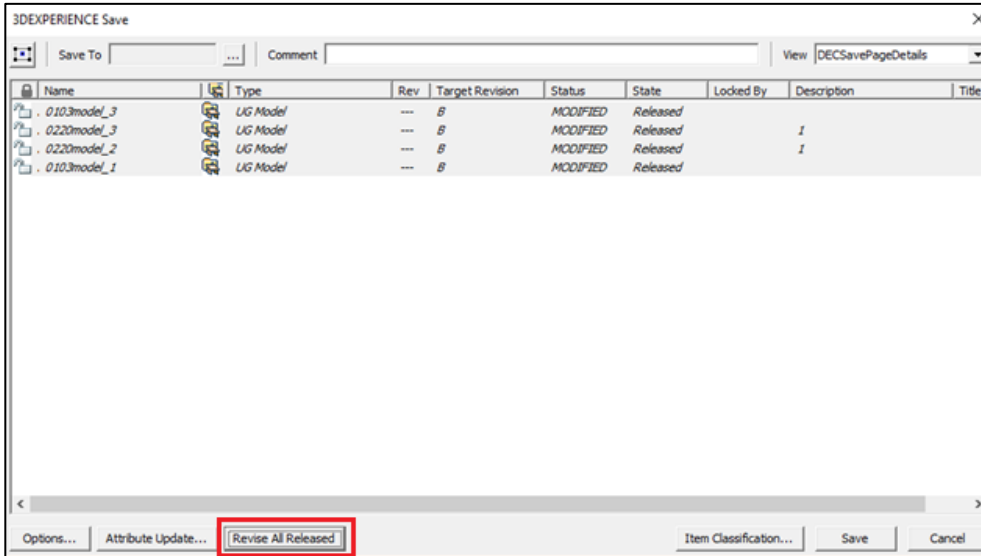
After the user performs the operation in the “**Revise All Released**” dialog and for some reason the user intends to undo revise for one of the Released CAD designs, there will be 2 ways to perform Undo Revise All Released

- 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.

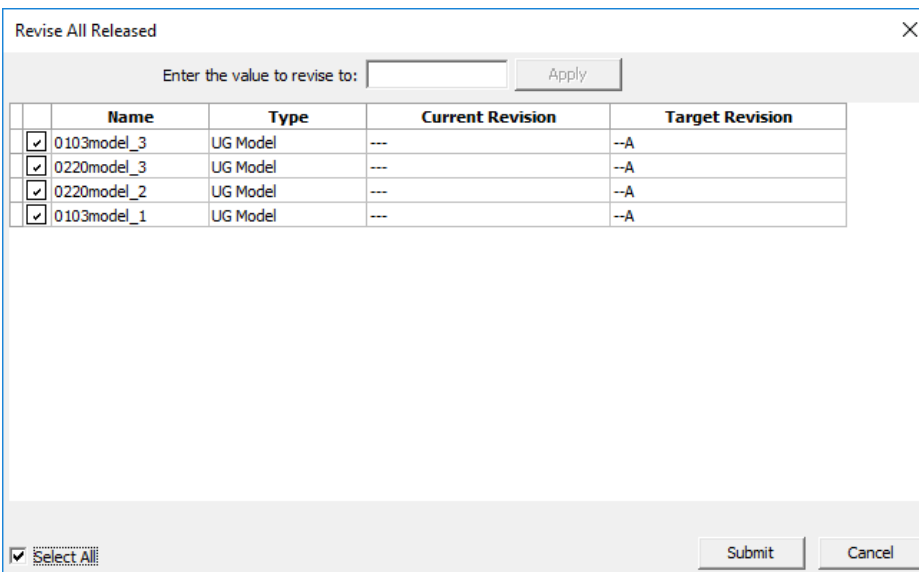


- This is also done one by one.

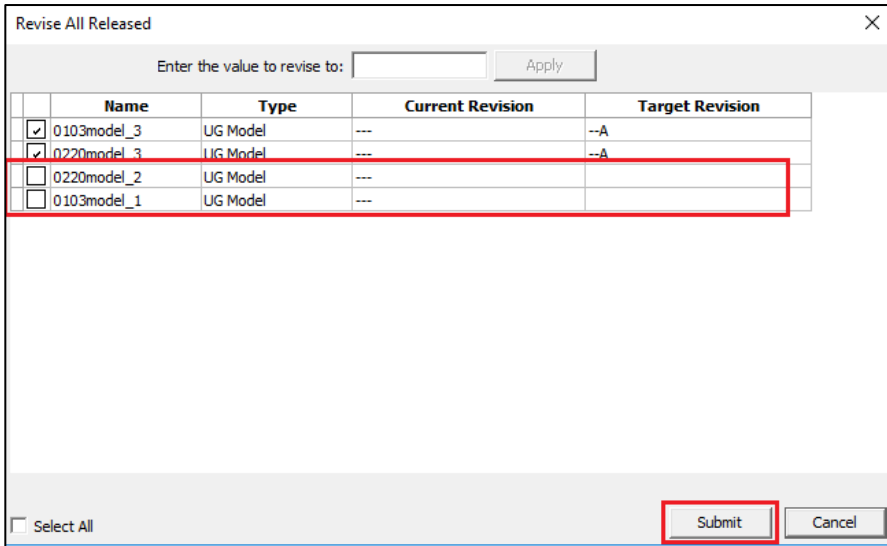
- Or either user can go back to **Revise All Released** dialog and deselect all the objects , it will undo the revise operation, and click on **Submit**.
 - First user will revise all the objects in **Revise All Released** dialog, and click on **Submit** button, it will come back to **Save** dialog and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



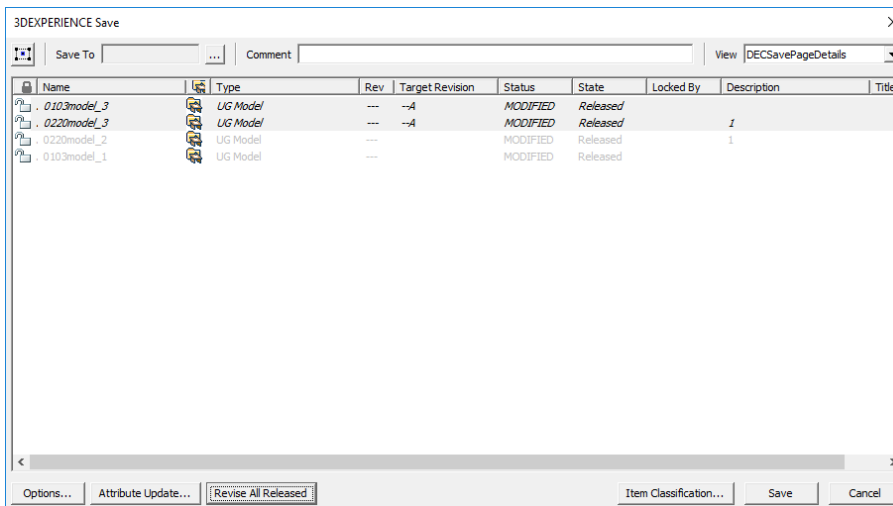
- Again, user wants to roll back the **Revise All Released** operation , then user will again click on **'Revise All Released'** button, the **Revise All Released** dialog will launch.



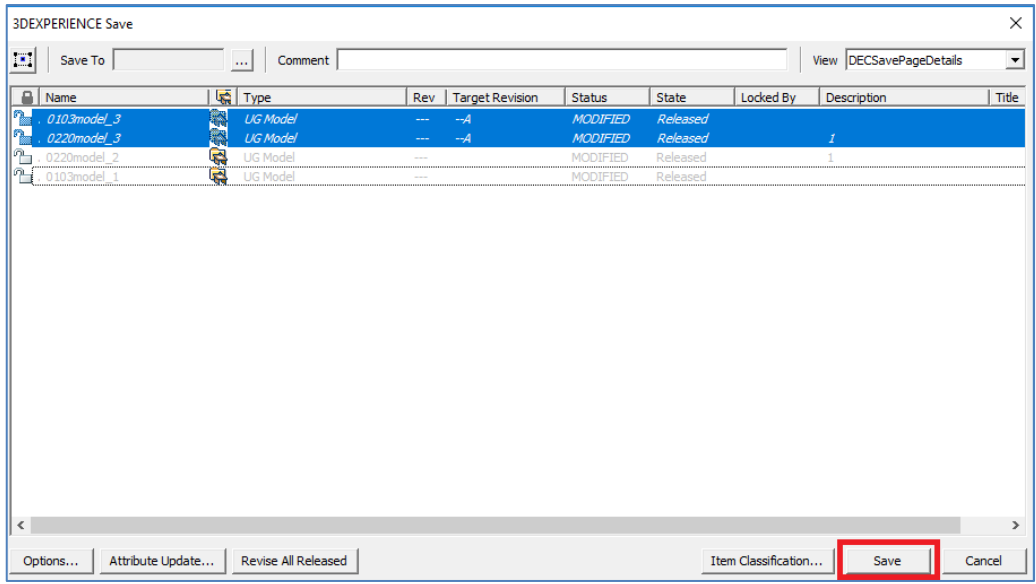
- User will deselect the checkbox of required assembly/part whose revision not to be changed and then click on **Submit** button.



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



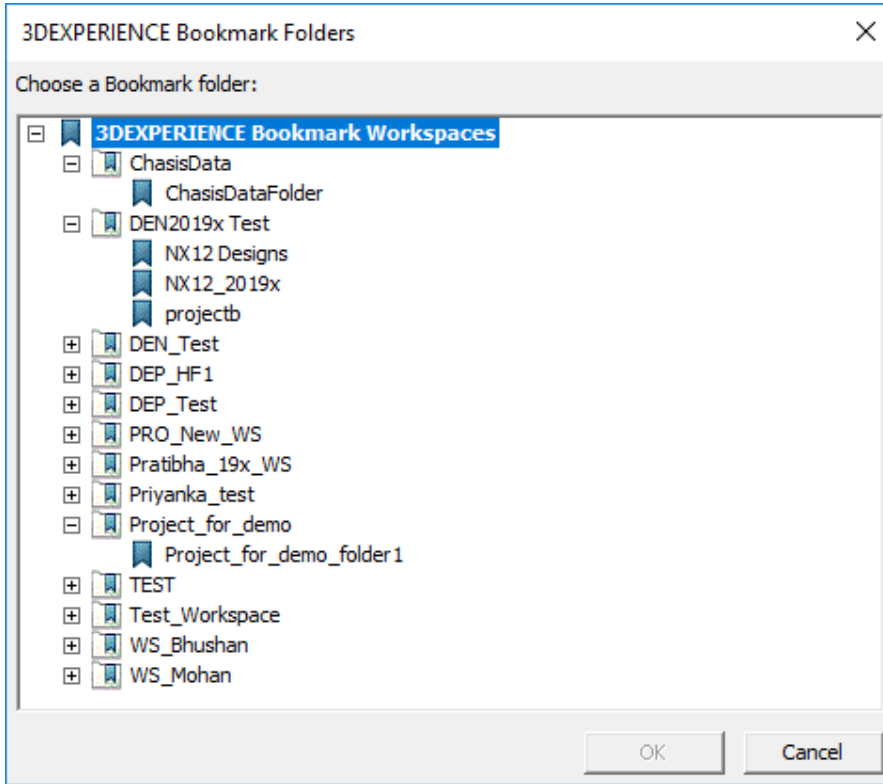
- After these changes, select all revised items and click on **Save** button to perform check in operation.



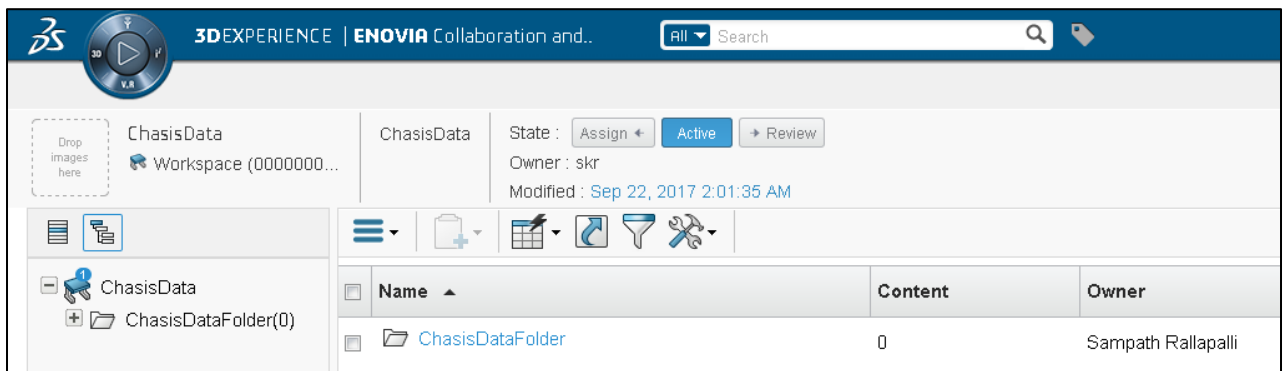
Selecting a Workspace to Save Designs

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

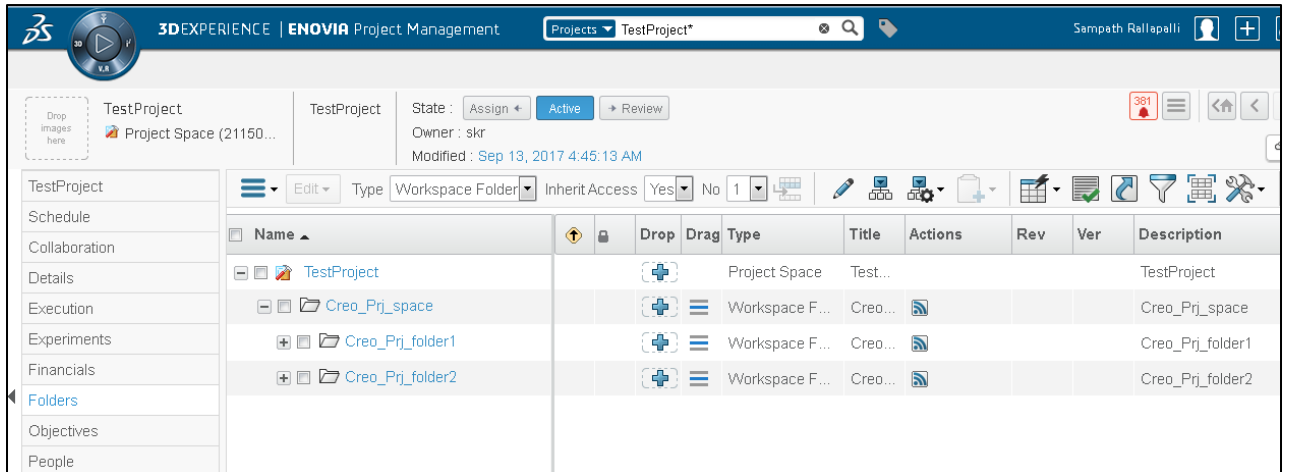
The **3DEXPERIENCE Folders** dialog box opens.



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 workspaces and click **OK**.
- If the 3DEXPERIENCE Admin sets the GCO option “MCADInteg-ForceWorkspaceOnSave” to TRUE, then the integration users need to select a 3DEXPERIENCE workspace folder on check-in.
- The integration allows user to check-in designs of an 3DEXPERIENCE Bookmark folder to another Bookmark folder.

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

- Workspaces and Workspace folders can also be created in the above dialog, please refer [create workspace/workspace folders during Save operation](#) section for more details.

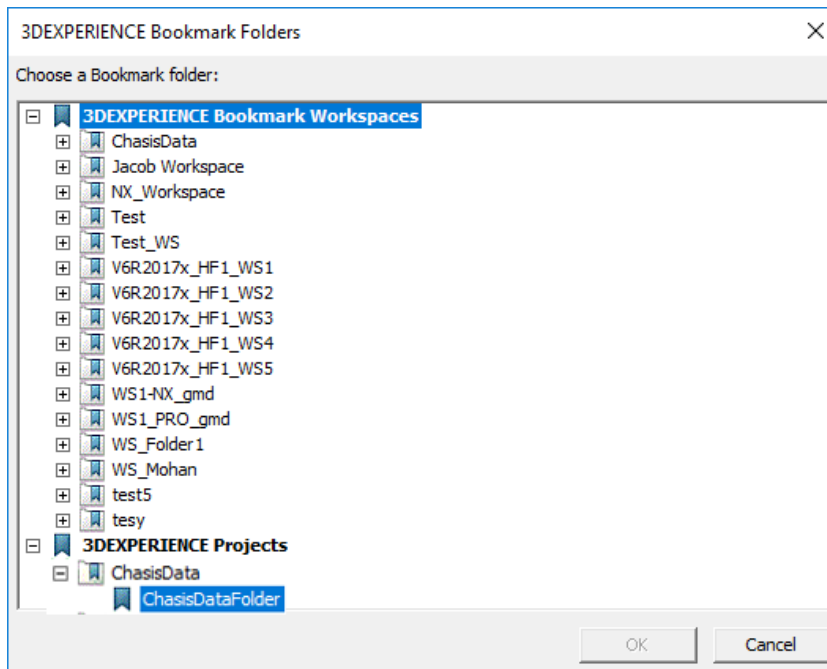
Selecting a Projectspace folder to Save Designs



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.



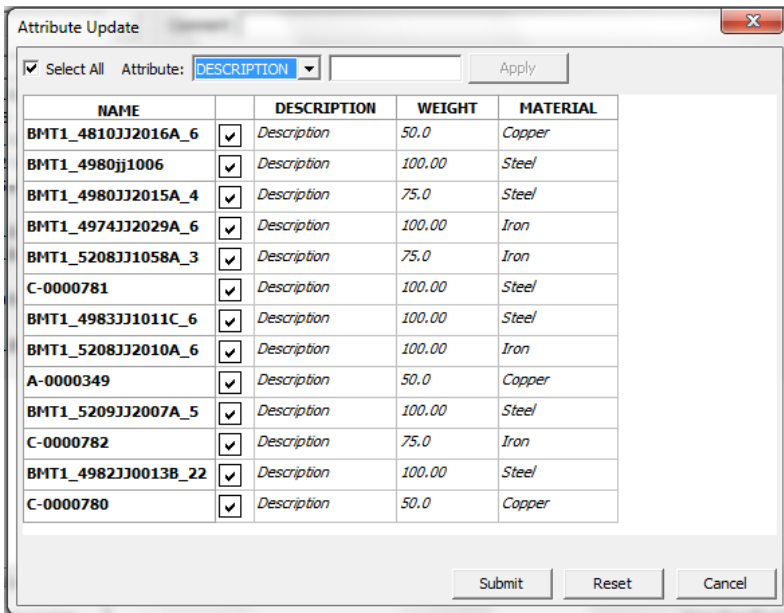
2. Select a Projectspace from the list of 3DEXPERIENCE Projects and click **OK**.
3. If the 3DEXPERIENCE Admin sets the GCO option “MCADInteg-ForceWorkspaceOnSave” to TRUE, then the integration users need to select a 3DEXPERIENCE workspace folder or a 3DEXPERIENCE Projectspace folder on checkin.
4. The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

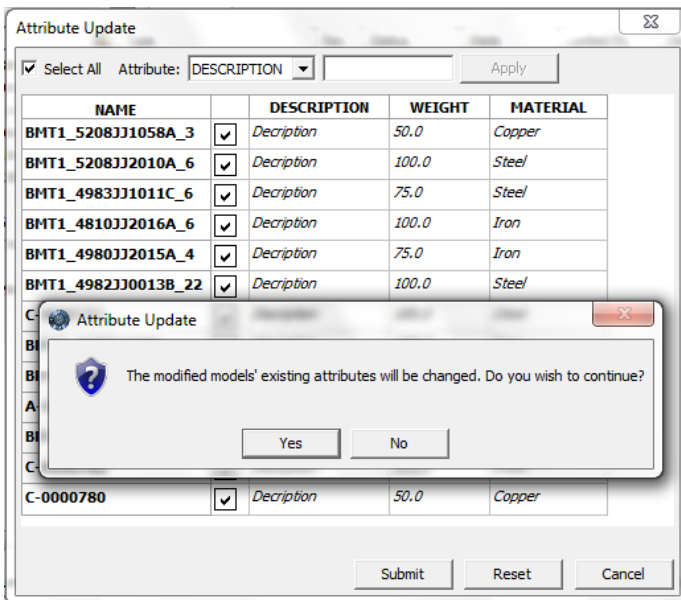
Using Update Attribute

The Update Attribute option allows user to view and update the specified attributes of all selected models at once during the save process.

In the **3DEXPERIENCE Save** dialog box, select models and click **Update Attribute**. The **Update Attribute** dialog will appear showing all models, assemblies, and instances on the left that were selected in the **3DEXPERIENCE Save** dialog. The **Update Attribute** dialog will also list all attributes along the top that were specified in the MCADInteg-UpdateAttribMapping GCO option.



Once user clicks on "Submit" a warning message will be displayed to the user:



Once user clicks on “Yes” the attributes will be updated.

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

Clicking **Attribute** will display the drop-down combo box to select from the list of attributes if there are multiple attributes to be updated

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

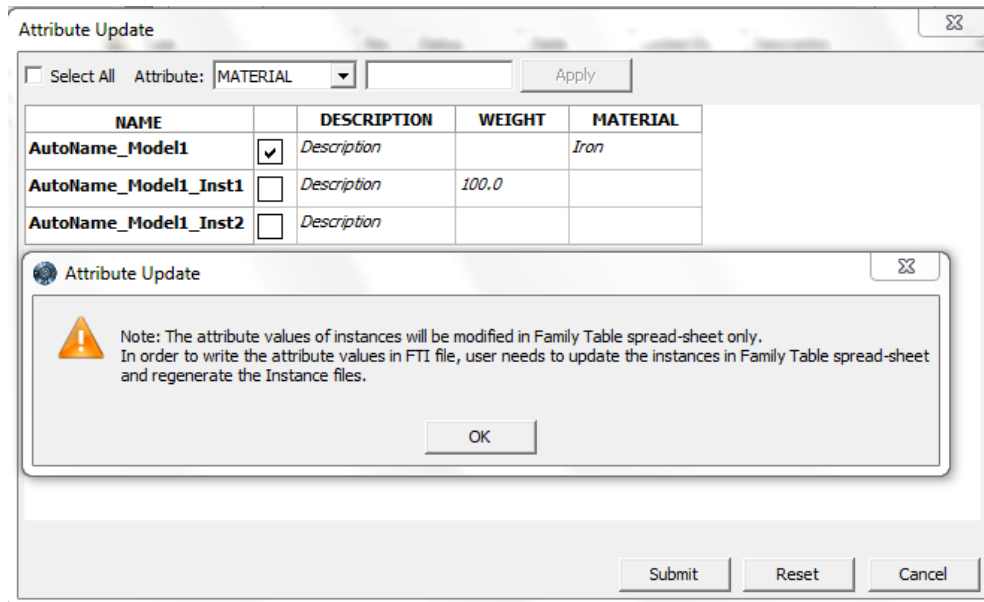
Clicking **Submit** will apply all the attribute changes made in the dialog to the CAD Models and then return the user to the **3DEXPERIENCE Save** dialog.

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

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

Attribute Update for Part Family Instances

Attribute Update feature has been enhanced to support attribute update for Part Family Instances. When user tries to update the attributes for Part Family Instances, apart from the above shown messages an extra warning message will be displayed to the user:



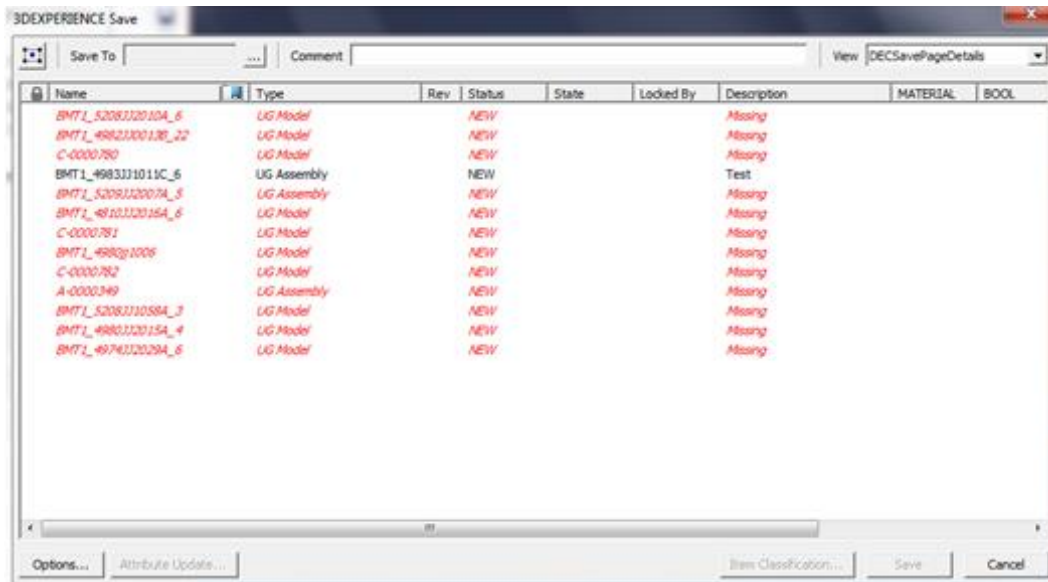
Once user clicks on “OK” the attributes are updated for the Part Family Instances

Mandatory Attribute

The mandatory attribute feature restricts users from checkin of a CAD model if mandatory attributes for the CAD model are not up-to-date. This information of mandatory attributes is retrieved from the GCO settings.

Before using this feature, update the GCO setting. Please refer the Admin Installer document for GCO setting for mandatory attribute.

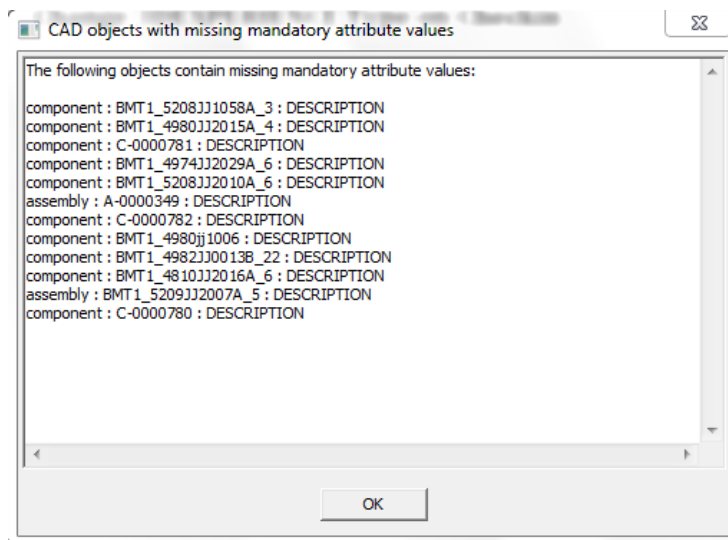
As per GCO setting whenever user tries to Save design, the feature will check for the mandatory attributes and if no value is found corresponding to these mandatory attributes then those CAD models are shown in Red color and the attribute column will contain “missing” to signify that these mandatory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



The screenshot shows the '3DEXPERIENCE Save' dialog box. It features a table with columns: Name, Type, Rev, Status, State, Locked By, Description, MATERIAL, and BOOL. The 'Name' and 'Description' columns are highlighted in red, indicating missing mandatory attributes. The 'Description' column contains the word 'Missing' for all listed items.

Name	Type	Rev	Status	State	Locked By	Description	MATERIAL	BOOL
BMT1_5208JJ2010A_6	UG Model		NEW			Missing		
BMT1_4982JJ0013B_22	UG Model		NEW			Missing		
C-0000780	UG Model		NEW			Missing		
BMT1_4983JJ1011C_5	UG Assembly		NEW			Test		
BMT1_5209JJ2007A_5	UG Assembly		NEW			Missing		
BMT1_4810JJ2016A_6	UG Model		NEW			Missing		
C-0000781	UG Model		NEW			Missing		
BMT1_4980jj1006	UG Model		NEW			Missing		
C-0000782	UG Model		NEW			Missing		
A-0000349	UG Assembly		NEW			Missing		
BMT1_5208JJ1058A_3	UG Model		NEW			Missing		
BMT1_4980JJ2015A_4	UG Model		NEW			Missing		
BMT1_4974JJ2029A_6	UG Model		NEW			Missing		

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



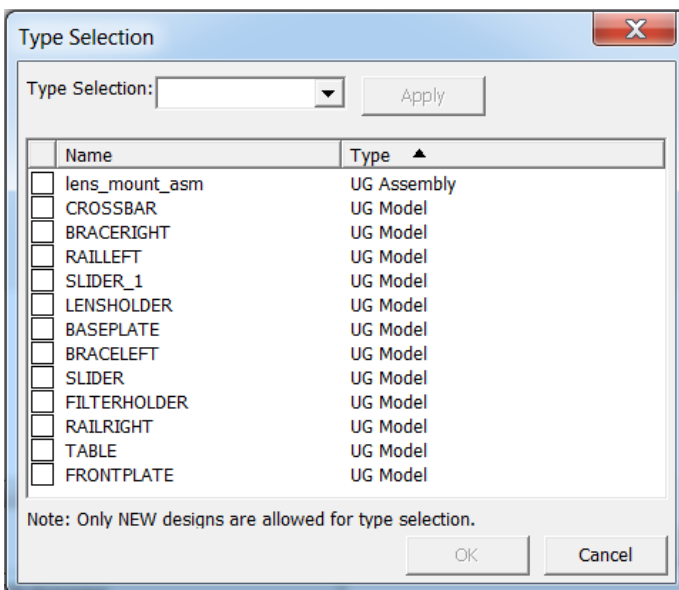
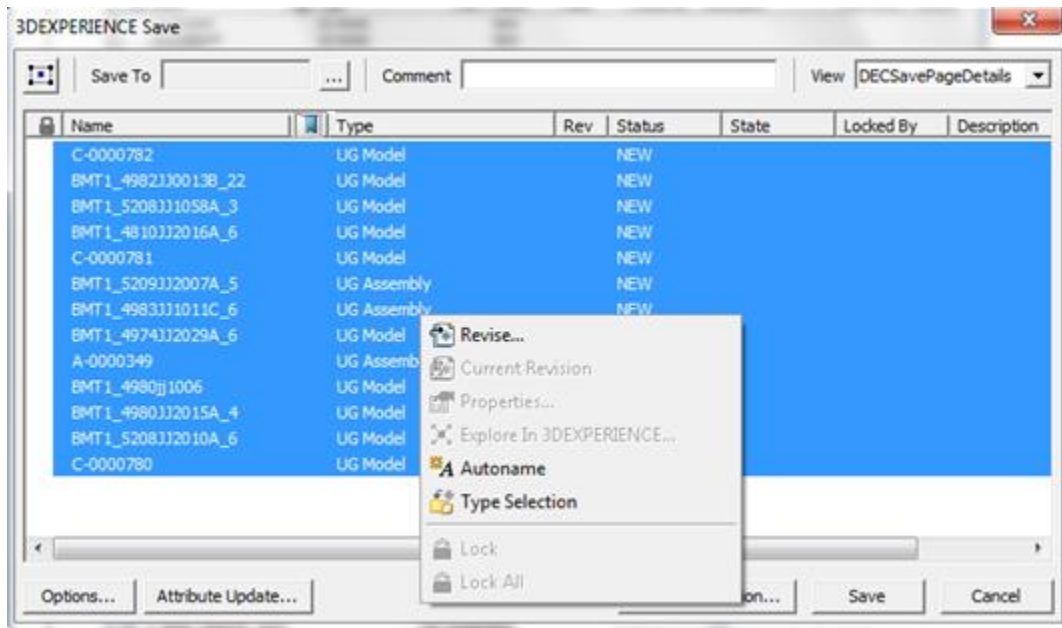
Change 3DEXPERIENCE Type on Checkin

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.

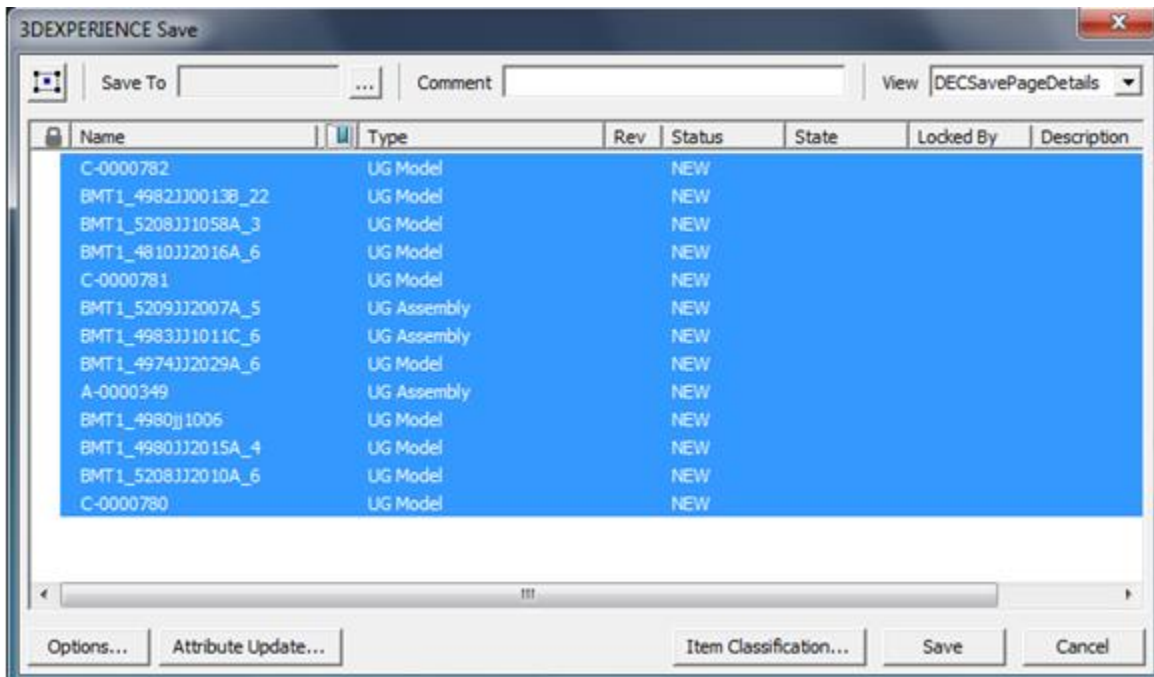
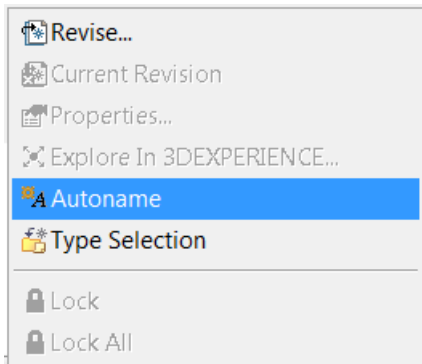
Before using this feature, the 3DEXPERIENCE Admin needs to create new 3DEXPERIENCE types and map them in the Global Configuration.



Using Autaname

The Autaname option is available depending on the Preferences set by the Integration Administrator. Only designs with status "New" can be named using Autaname. As a prerequisite, the Autaname 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 **Autaname**. Names are generated automatically according to the selected Autaname series in 3DEXPERIENCE Save options dialog box and are displayed under the **Autaname** column in **3DEXPERIENCE Save** dialog box.



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

Maintaining Structural Integrity of Data during 3DEXPERIENCE Save

NX allows partial loading of model structure. Therefore, care must be taken such that relationships to unloaded models are not lost during 3DEXPERIENCE Save when partial structures are loaded in session.

Related Configuration

Loading of additional objects into the NX session during 3DEXPERIENCE Save is controlled by the Global Configuration Object (GCO) attribute **UGObjectLoadingInSession**. If an object and its children are not loaded in session, it is not possible for the 3DEXPERIENCE Connector for NX to gather the required information for the object to be selectable during the command. The default value for this option is **'TemporaryLoadLevel1ObjectsInSession'** which indicates that all level 1 components of the active (displayed) model will be loaded into session during 3DEXPERIENCE Save. Other values for the **UGObjectLoadingInSession** attribute are **'LoadObjectsInSession'** and **'DoNotLoadObjectsInSession'**. Please refer to the *Install and Administration Guide for 3DEXPERIENCE Connector for NX* for further details.

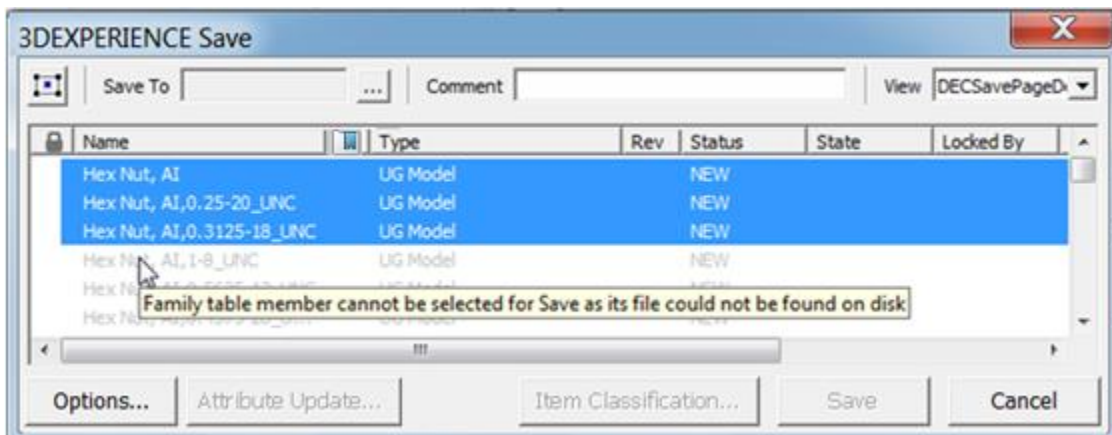
An option, `checkForNewNonSelectableChild`, is present in the `settings.ini` file located in the `bin` directory where 3DEXPERIENCE Connector for NX is installed. This option controls whether or not to mark an object not selectable for Save if the object has any NEW children that are not selectable, even if those children are in session. By default, `checkForNewNonSelectableChild = 0`, selectability of a parent object is only based on its direct children. If `checkForNewNonSelectableChild = 1` selectability of a parent object will recursively check if any child object in its structure is new and not selectable then the parent will not be selectable.

Partial Loading Scenarios

Below is a listing of partial loading scenarios and how they are handled by the NX integration.

1. Part Family Template with New Part Family Members that do not have Files on Disk

If a part family template is being Saved and it has Part Family members that have not been previously Saved to 3DEXPERIENCE, and the members do not have files on disk, those member objects will be grayed out in the Save dialog. The tooltip on the member objects will indicate that the part family member cannot be selected for Save as its file could not be found on disk.



The part family template can still be saved, and the new iteration created in 3DEXPERIENCE will have relationships to its part family members selected for Save as well as its part family members that already exist in 3DEXPERIENCE from previous Save operation.

If the user would like to Save the part family template with relationships to the new part family members then they must use NX part family functionality to create the new part family member files on disk and then perform another 3DEXPERIENCE Save operation.

2. Parent Object References Child Not Loaded

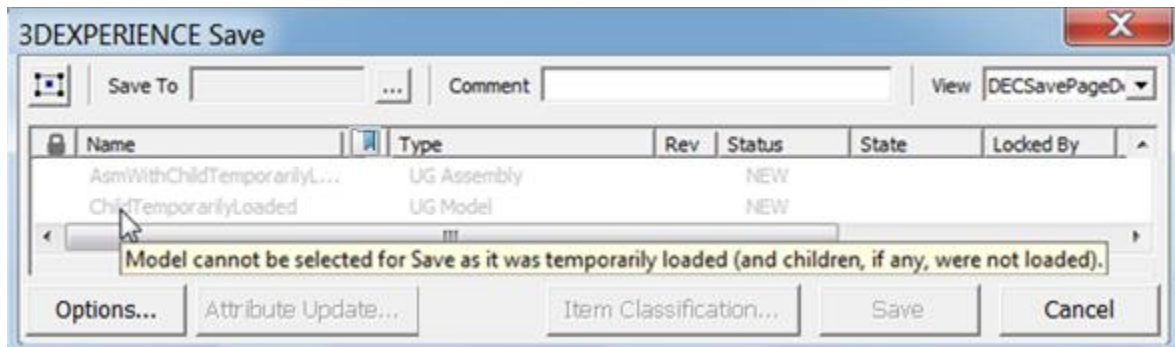
If a child is not loaded (due to **UGObjectLoadingInSession** setting or child object could not be found) the parent object will not be selectable. The parent and child object will be grayed out in the Save dialog. The tooltip on the child object will indicate that the object cannot be selected for Save as it is not loaded. The tooltip on any object that references the unloaded child will indicate that the parent cannot be selected as it has a child that is not loaded.



To perform 3DEXPERIENCE Save on such parent objects, the child objects must be loaded into the NX session (or available for loading if GCO **UGObjectLoadingInSession** is set to load the child objects) and another 3DEXPERIENCE Save operation should be initiated.

3. Object Temporarily Loaded

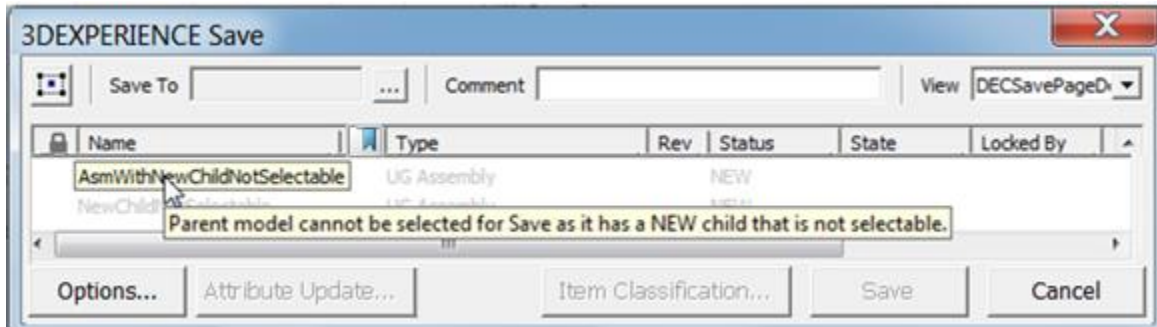
If a child is temporarily loaded into session due to GCO **UGObjectLoadingInSession** attribute set to '**TemporaryLoadLevel1ObjectsInSession**' then that child object will be grayed out in the Save dialog. Its tooltip will indicate that the object cannot be selected as it was temporarily loaded.



To perform 3DEXPERIENCE Save on such objects, the objects should be brought into the NX session (and their children, if any, should be available during Save) and another 3DEXPERIENCE Save operation should be initiated.

4. Parent Object References New Child that is not Selectable

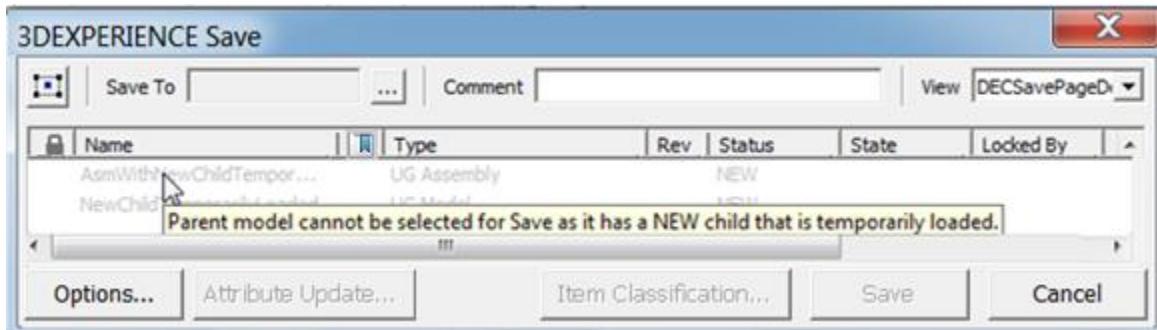
If a parent object references a child that is New to 3DEXPERIENCE and that child is not selectable, then the parent object will also be grayed out and not selectable in the 3DEXPERIENCE Save dialog. The tooltip for the parent object will indicate that the parent object cannot be selected as it has a new child that is not selectable.



To perform 3DEXPERIENCE Save on such parent objects, the user must take appropriate action to ensure that the NEW child objects are selectable and then another 3DEXPERIENCE Save operation should be initiated.

5. Parent Object References New child that is Temporarily Loaded

If a parent object references a child that is New to 3DEXPERIENCE and that child is temporarily loaded, then the parent object will also be grayed out and not selectable in the 3DEXPERIENCE Save dialog. The tooltip for the parent object will indicate that the parent object cannot be selected as it has a new child that is temporarily loaded.



To perform 3DEXPERIENCE Save on such parent objects, the user must load the NEW child objects into the NX session and then another 3DEXPERIENCE Save operation should be initiated.

6. Partially Loaded Assembly that References Models Existing in 3DEXPERIENCE

If a parent object references a child that exists in 3DEXPERIENCE, then that parent will be selectable in the 3DEXPERIENCE Save dialog even if the child was loaded or temporarily loaded into session during the 3DEXPERIENCE Save process.

Update PLM Parameters During Save

The PLM parameters such as STATE, REVISION, DESIGNED_BY, MODIFIED_BY is updated in the NX CAD model's properties so that these attributes can be configured to reflect in to the title block of the NX Drawing.

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

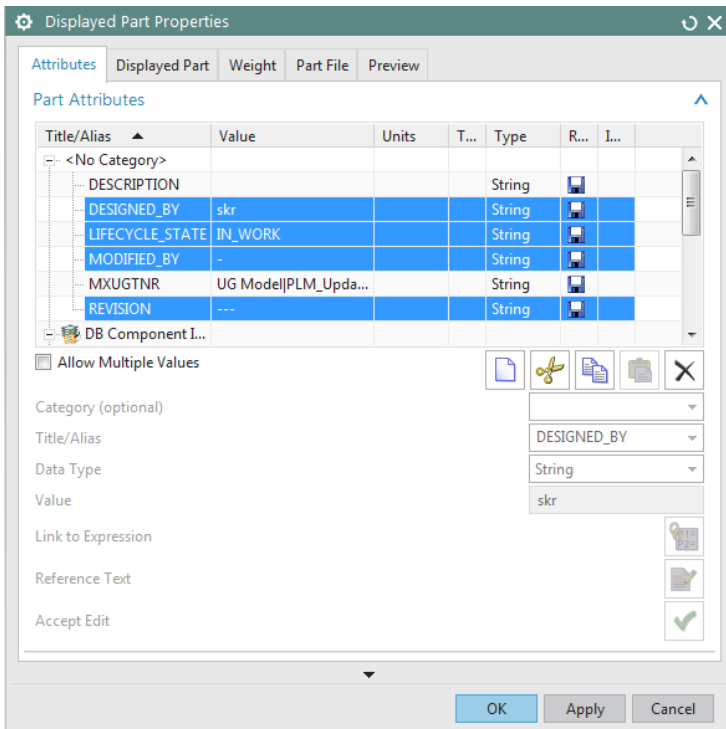
MODIFIED_BY and DESIGNED_BY attribute information will be updated with the User credentials after checkin through integration.

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

Updating PLM parameters

Parameters updated during Checkin

During first Checkin of NX CAD model Design to 3DEXPERIENCE, the attributes DESIGNED_BY, LIFECYCLE_STATE, REVISION are populated with appropriate values. MODIFIED_BY will be updated as "--"



During checkin of modified NX CAD model Design to 3DEXPERIENCE, the attributes DESIGNED_BY, LIFECYCLE_STATE, REVISION should be populated with appropriate values. MODIFIED_BY will be updated as user credentials

Using Quick Save

This section shows you how to save active design to 3DEXPERIENCE without opening the **3DEXPERIENCE Save** dialog box. The Global preferences set by the Integration Administrator are applied to the designs on save operation. If Save dialog is not launched during Quick Save operation, then lock of the object checked in is retained by the user.

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

Click **3DEXPERIENCE > Quick Save** from NX toolbar

3DEXPERIENCE Save dialog box opens if any obsolete nodes are present in the designs being saved or the design is locked by another user.



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 Preference**.

- **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 NX operations. Copies of the files to be saved to ENVOIA are made and control is returned back to NX, so you can continue to use NX 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.

Using 3DEXPERIENCE Open or Insert Option

The Open option in 3DEXPERIENCE menu in NX 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

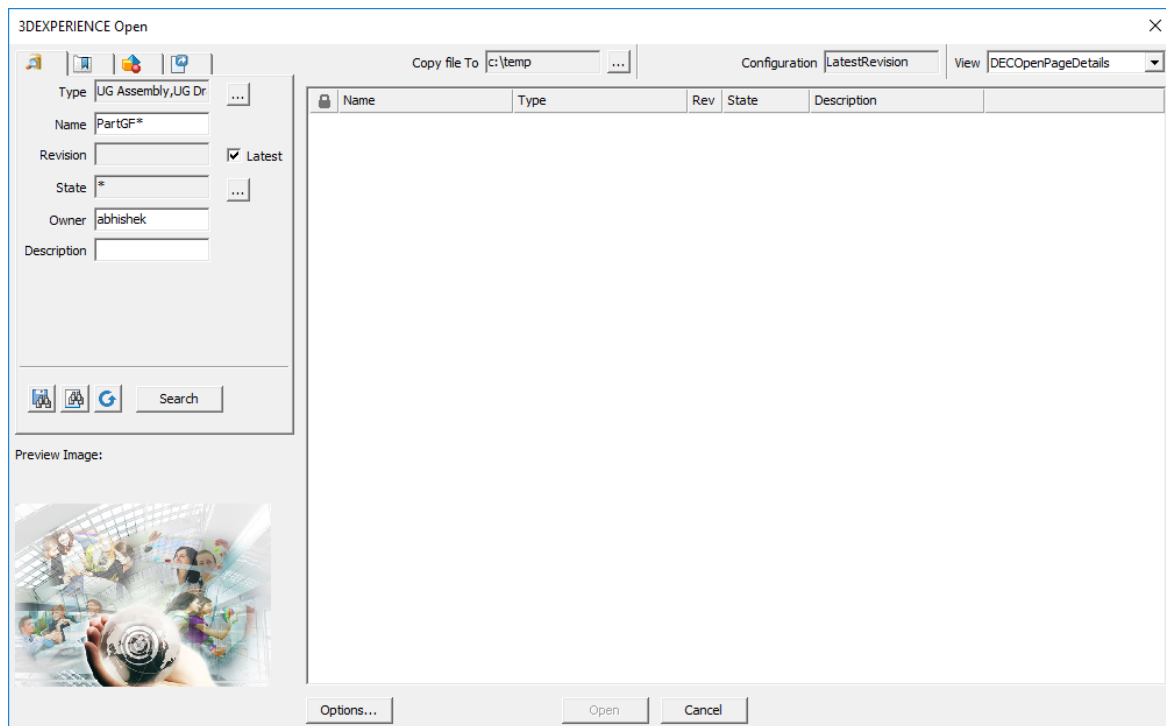
You can search for designs that exist in 3DEXPERIENCE vault from **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box. The following sections describe the steps used to search for designs, saving your query, using the saved query, and removing saved queries.

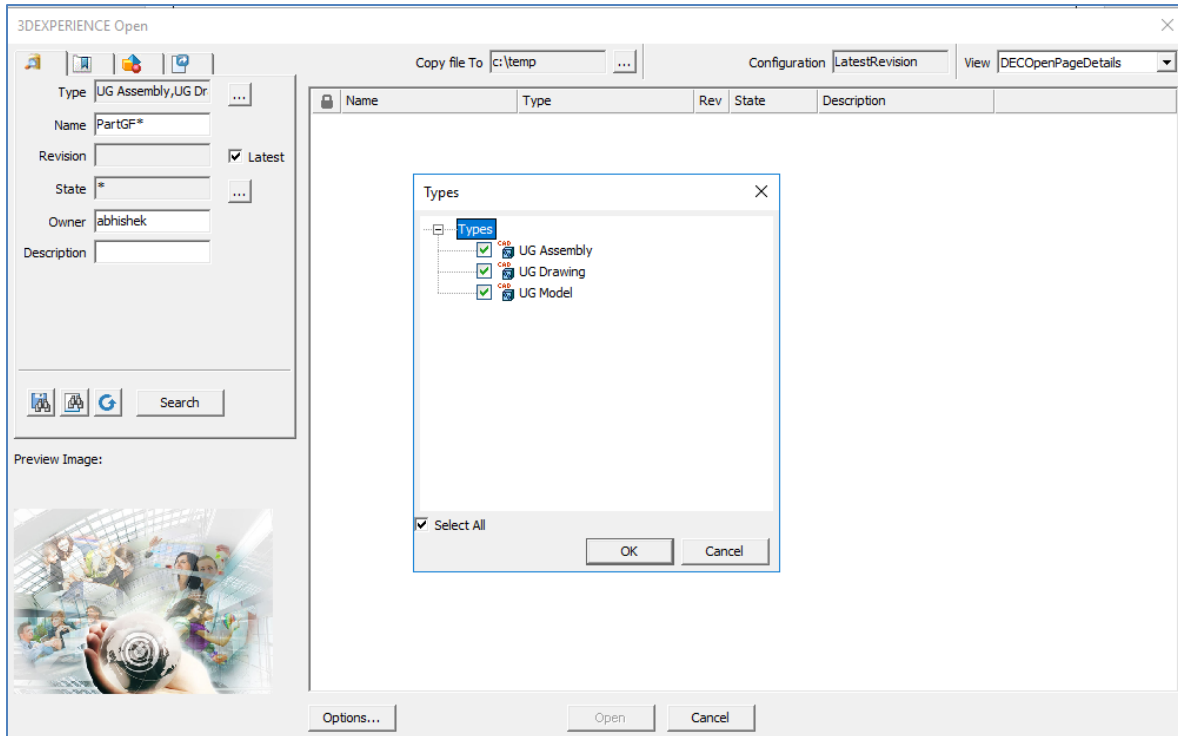
Searching Designs

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

1. Click **3DEXPERIENCE > Open** from 3DEXPERIENCE ribbon.

The **3DEXPERIENCE Open** 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.



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. The very first dialog invocation in a new session will show the logged in user name by default set in this field.

- **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.

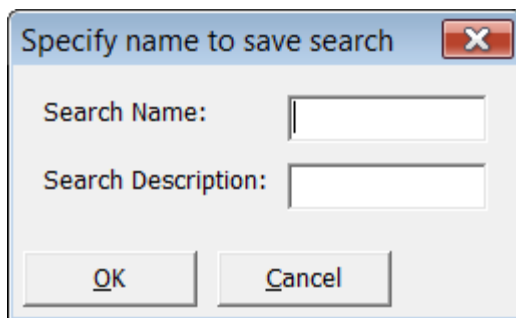
Saving Your Webform Driven Search Parameters

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.

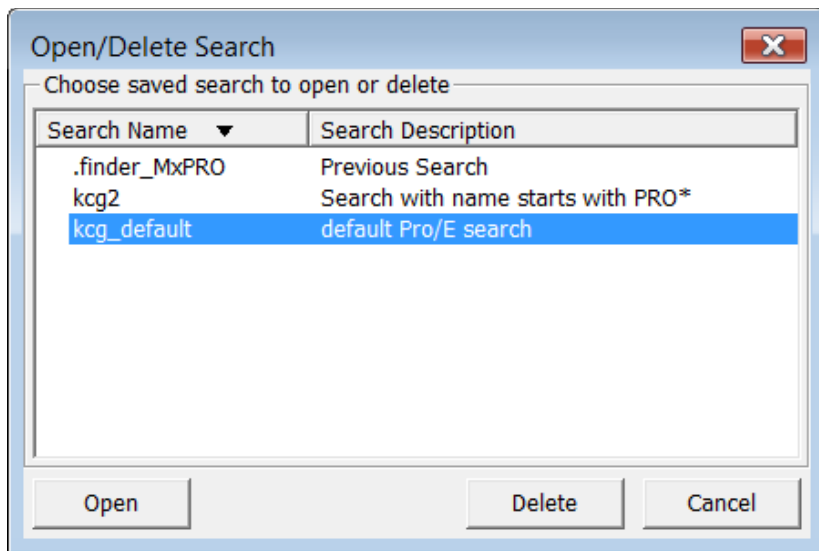
Using Saved Queries

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.



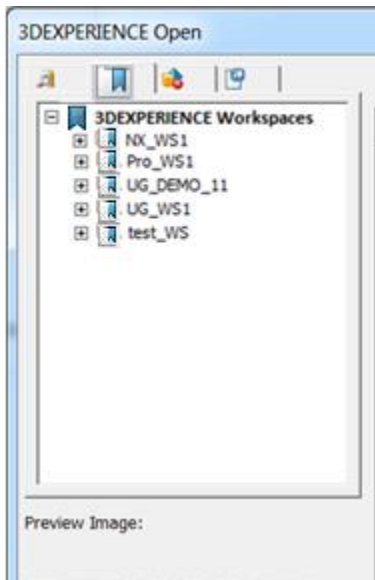
2. Select a query and click Open to populate the search fields.
3. Once all search parameters are set to desired values click **Search**

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:

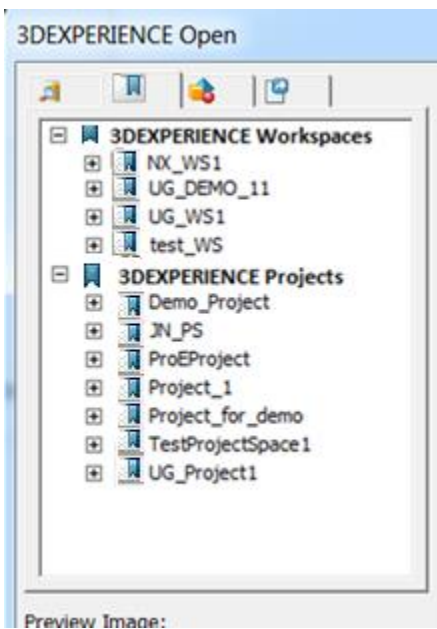


When a workspace is selected, the corresponding search results are displayed in the Open dialog.

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.

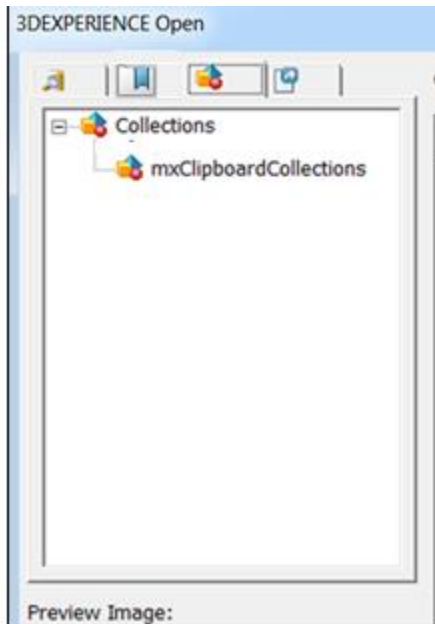
To search designs in user's Projectspace, click on the  tab. The user's Projectspace will be displayed:



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


Searching Designs in User's Collections

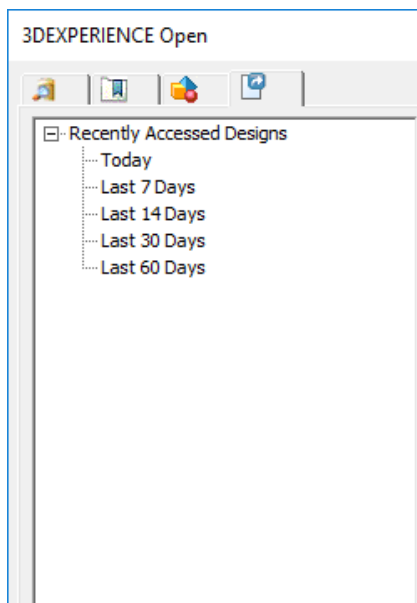
To search designs in user's Collections, click on the  tab. The user's collections will be displayed:



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:



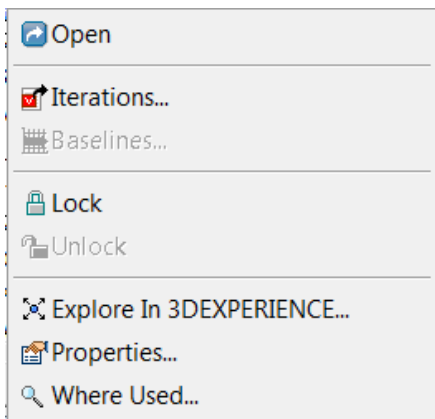
When a recently accessed classification is selected, the corresponding search results are displayed in the Open dialog.

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,

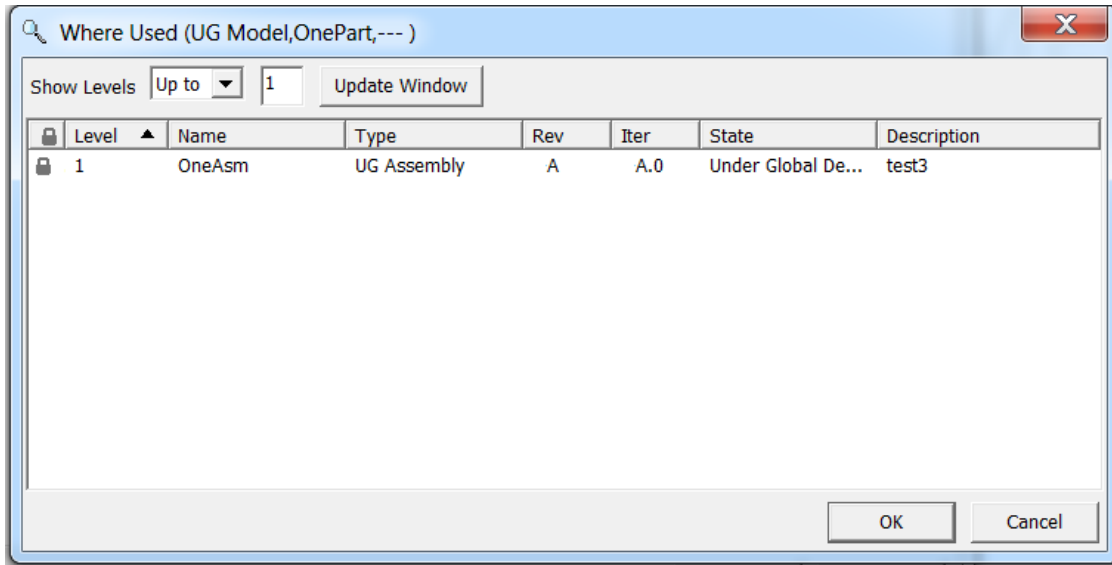
-  **Lock Status.** Whether or not the design is locked.
- **Name.** Name of the design.
- **Type.** Type of the design.
- **Rev.** Revision of the design
- **State.** Current state of the design in lifecycle.
- **Description.** Description of the design.

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



- **Open.** This option is available only if you clicked **3DEXPERIENCE > Open** in step 1. Opens selected designs from 3DEXPERIENCE in NX. See [Opening Designs From 3DEXPERIENCE in NX](#) for more details.
- **Iterations.** Opens the selected iteration of the design in NX. See [Opening Particular Iteration of Design](#) for more details.
- **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](#)
- **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 NX 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 NX, 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 NX 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 NX are,

- **3DEXPERIENCE > Open.** Use this option to open designs from 3DEXPERIENCE in NX.
 - Only designs locked by user will have write permissions after checkout. The user must lock the design if he intends to make changes to design.
- **3DEXPERIENCE > Insert.** Use this option to insert the design from 3DEXPERIENCE or from local disk in the currently active design in NX.

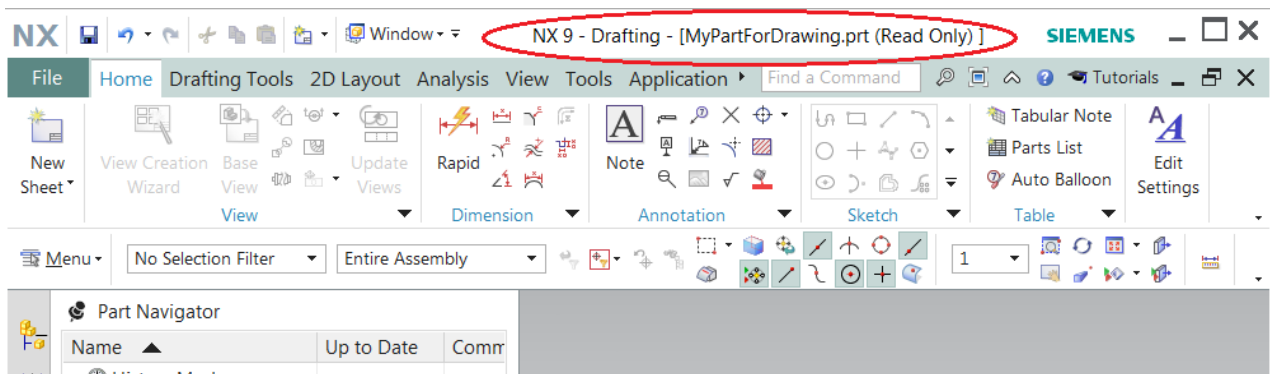


Note: Rename of Family Table members in 3DEXPERIENCE is not supported during **3DEXPERIENCE/Open** (or **Insert**). Per NX documentation regarding **Creating Family Members**:

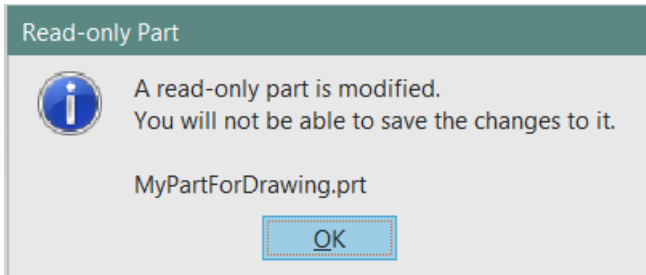
Family member parts are created read-only. Any editing must be done either by editing the template part or the spreadsheet. If you want to create a copy of a family member part, use File→ Save As to save it under a different name. This also breaks the associativity to the spreadsheet.

Read Only

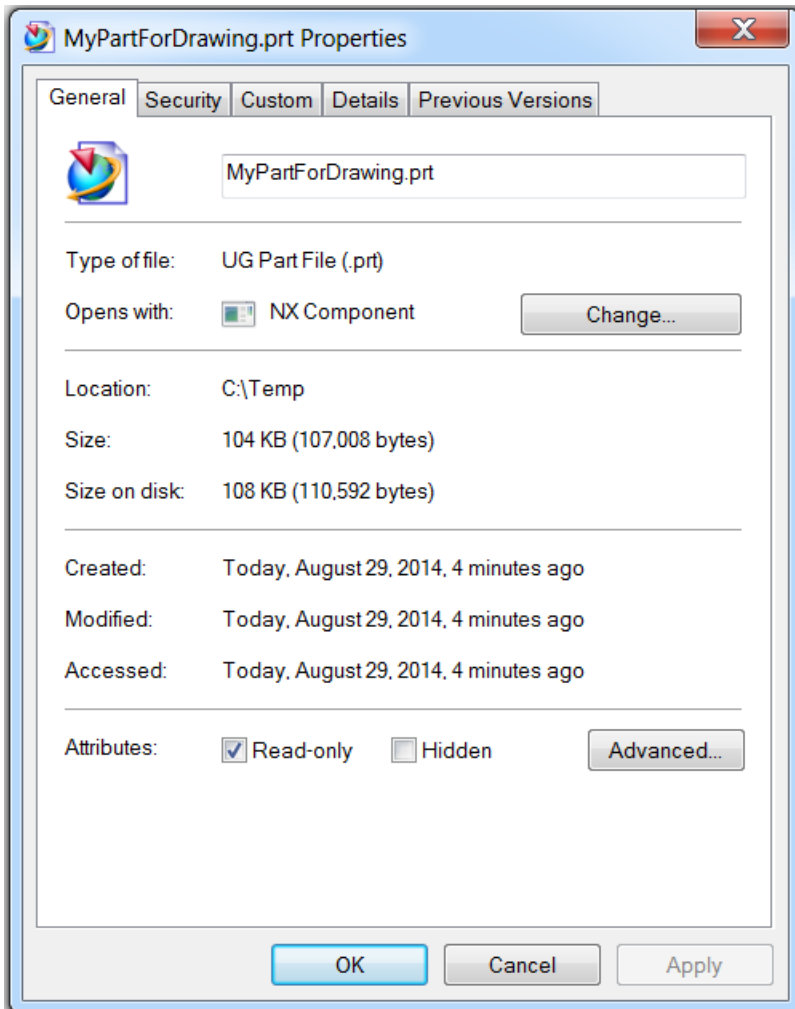
When using **3DEXPERIENCE > Open** to download files without lock, the part will be marked as 'Read Only' on the disk. This 'Read Only' status is indicated on the title bar after the part name.



Modifications to a 'Read Only' file *cannot* be saved to disk. A warning message will be displayed stating this if a part marked 'Read Only' is modified.



The 'Read Only' designation on a file can be removed by either locking the 'Read Only' part 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.

Attribute Synchronization is done each time you open a design in NX. If you are not connected to 3DEXPERIENCE while opening a design, the **3DEXPERIENCE Attribute Synchronization** message is displayed. Click **Login to 3DEXPERIENCE** to launch the **3DEXPERIENCE Login** dialog box or click **Continue Load** to load the design without synchronizing attributes. For example, during the design process User A modified the Author and Comments attributes of a design Part A using Properties dialog box in NX. This modified metadata is not copied to the Part in 3DEXPERIENCE store. When you search, and open Part A with **Copy Attributes** selected from the **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box, then the attributes are copied to 3DEXPERIENCE store for Part A.

Your Integration Administrator specifies which attributes and properties can be synchronized. You can synchronize attributes during open using the **Copy Attributes** option.



Note: See *X-CAD Design Install and Administration Guide* for details on mapping attributes, and enforcing attribute synchronization.



Note: You can also synchronize attributes when you save a design to 3DEXPERIENCE.

Opening Multiple Designs

Using **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box you can search, select, and open or insert multiple designs from 3DEXPERIENCE in NX. 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.

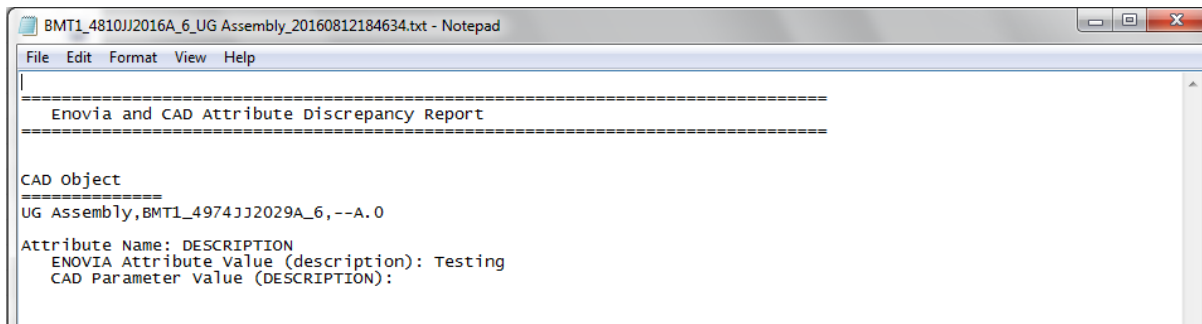
Attribute Discrepancy Report Generation

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 ;' in the section.

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-MxToCADAttrbMapping" mapping for finding discrepancies between the attributes value of 3DEXPERIENCE metadata and the value of CAD attribute stored in the native NX design.



```
BMT1_4810JJ2016A_6_UG Assembly_20160812184634.txt - Notepad
File Edit Format View Help
-----
Enovia and CAD Attribute Discrepancy Report
-----
CAD Object
-----
UG Assembly, BMT1_4974JJ2029A_6, --A.0
Attribute Name: DESCRIPTION
ENOVIA Attribute Value (description): Testing
CAD Parameter Value (DESCRIPTION):
```

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 Designs from 3DEXPERIENCE in NX

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

The designs must already be saved in 3DEXPERIENCE. See [Saving Designs to 3DEXPERIENCE](#).

1. Click **3DEXPERIENCE > Open** from NX toolbar.

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

2. Search for the designs using **Search** option in **3DEXPERIENCE Open** dialog box. See [Searching Designs](#)

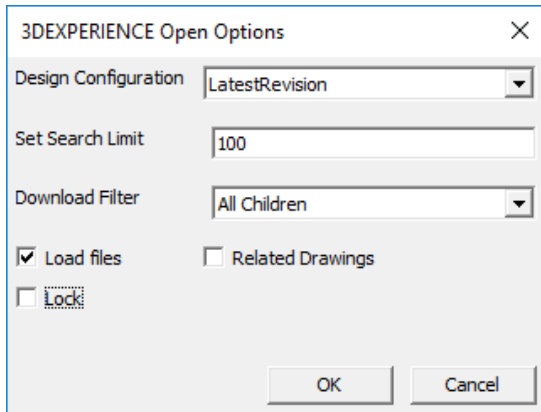
The results of the search are listed.

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:

- **Checkout To.** The designs opened from 3DEXPERIENCE are downloaded to the 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.
- **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.

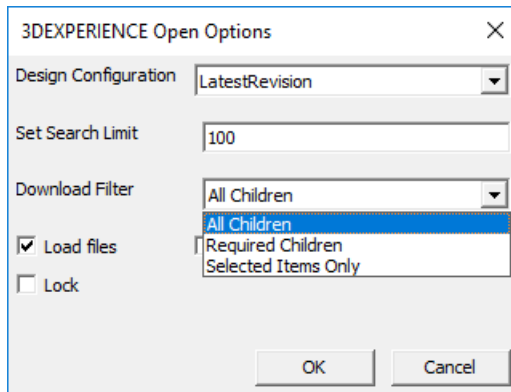
4. Click **Options** in **3DEXPERIENCE Open** dialog box.



5.

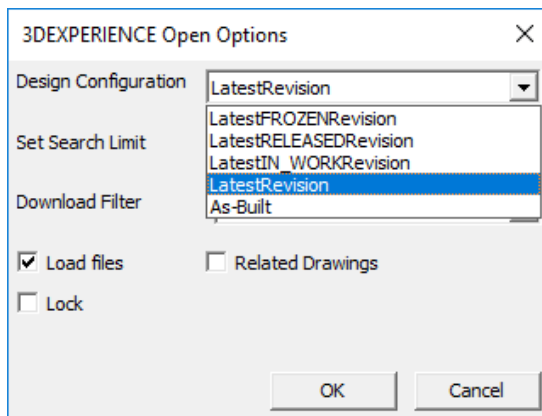
The **3DEXPERIENCE Open Options** dialog box opens. Choose options to be executed when you open a design from 3DEXPERIENCE:

- **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.
- **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, 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.** Select this checkbox to lock checked out files
- **Load files.** Select this checkbox to load checked out files in NX session
- **Design Configuration.** Select an option from the drop-down list to display specific iterations or revisions of designs in **3DEXPERIENCE Open** dialog page:



Click **OK**.

6. Click **Open**.

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

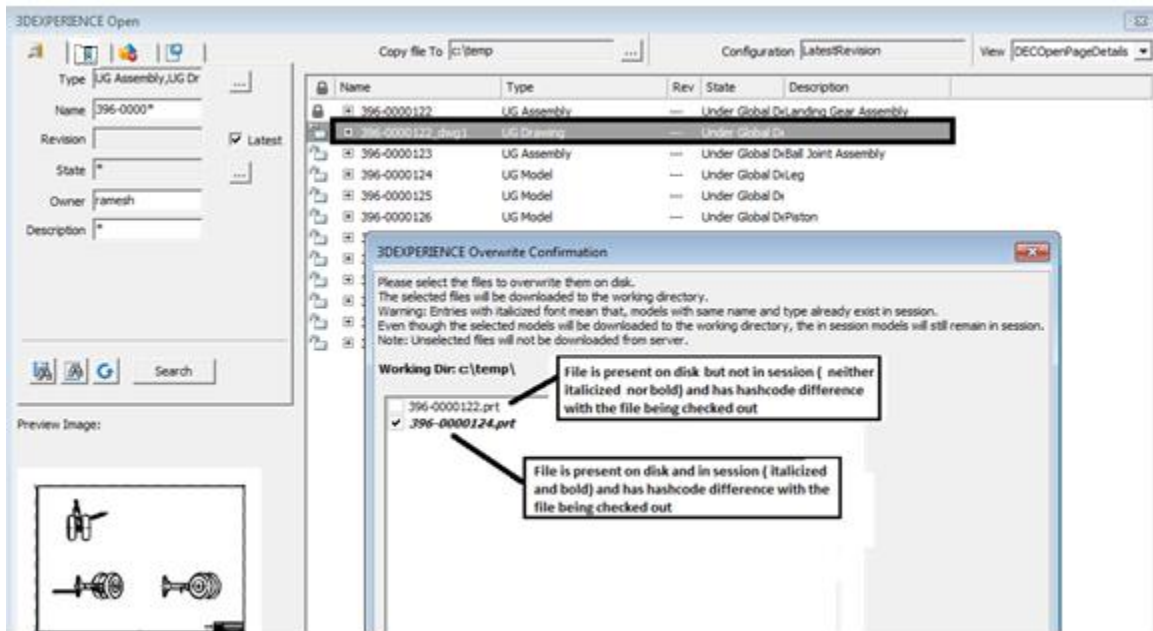
Open Enhancements to complete checkout when models are in session

The selected designs are opened in NX and also downloaded to the location specified in **Copy File To** in **3DEXPERIENCE Open** dialog box.

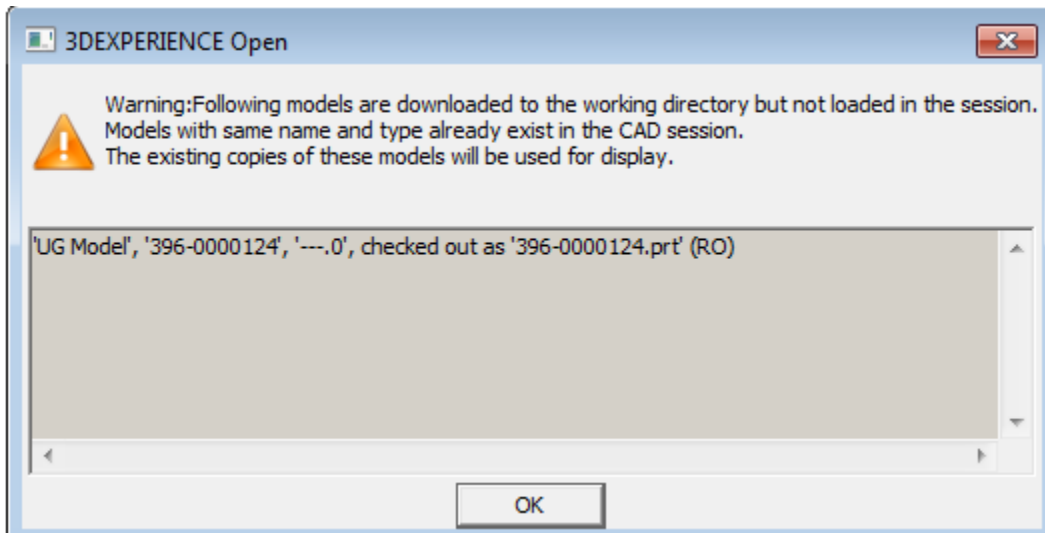
The 3DEXPERIENCE->Open behavior is enhanced to improve the usability and make it convenient for the CAD users to continue the checkout without having to close the models they are working on in current CAD Session.

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 or a drawing to checkout. The models which are already in the session or are associated to the drawing which is getting checkout, 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.



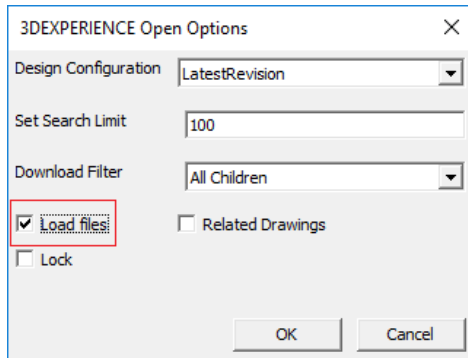
User will be prompted with the below warning that files were downloaded to the working directory, however, they are not loaded into CAD session.



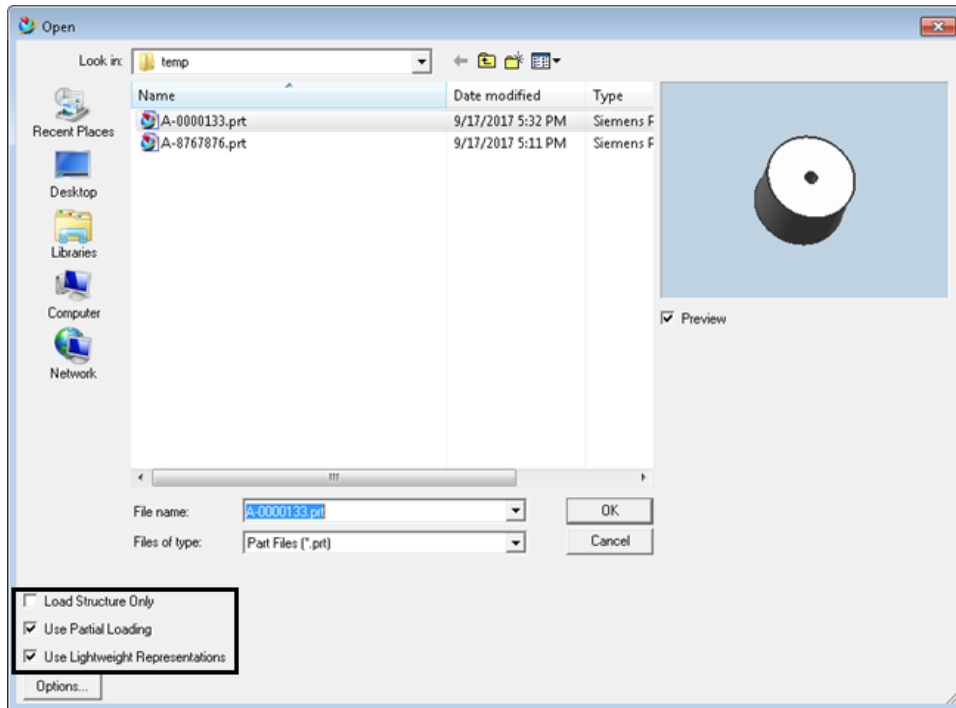
Note: This enhancement also works for the associated drawing. When user checkout any drawing and if the associated models are already loaded in session or saved on disk the above shown dialog and warning message will be displayed.

Enhanced 3DEXPERIENCE>Open functionality to provide NX File>Open dialog

This functionality enables user to choose load options for designs downloaded from 3DEXPERIENCE, by displaying NX native File>Open dialog. To use this functionality user should turn off 'Load files' option in '3DEXPERIENCE Open Options' dialog.



When checkout is done with 'Load Files' option turned off, at the end of checkout, designs will be downloaded in checkout directory but will not be loaded in session, instead NX Native File>Open dialog is displayed to user, where downloaded design will be preselected. In File>Open dialog user has choice to decide the open options and load the downloaded design as per the load options in NX.



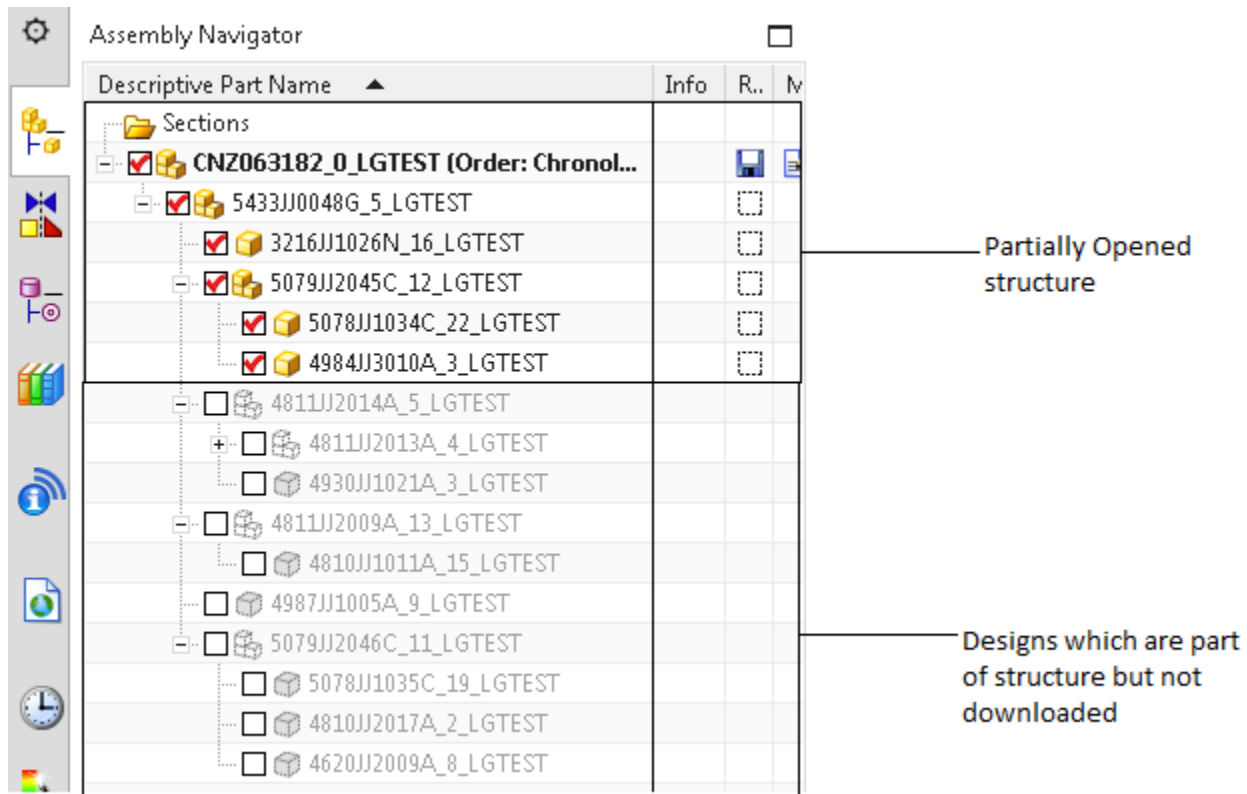
Open Partial from 3DEXPERIENCE

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 'NXOpenPartialFromWebInstallationGuide.pdf' available in 3DEXPERIENCE Connector For NX installation.

Open Partial operation is initiated from 3DEXPERIENCE thin client. Once user initiates the operation from 3DEXPERIENCE thin client, the partial structure is downloaded to disk and opened in NX session.



Partially Opened structure

Designs which are part of structure but not downloaded

In case above Partially Opened structure if the designs are modified and checked in by 3DEXPERIENCE>Save functionality, the pre-existing structure members which are not partially Opened will still remain in structure of the top Assembly in 3DEXPERIENCE, and the changes made in CAD session will merge with the assembly structure in 3DEXPERIENCE.



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

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 NX.

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

- Part in Assembly
- Assembly in Assembly

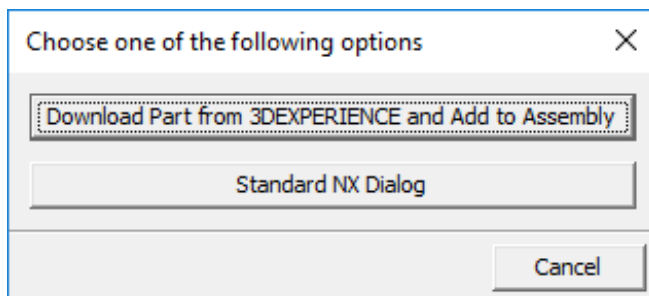
You cannot insert,

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

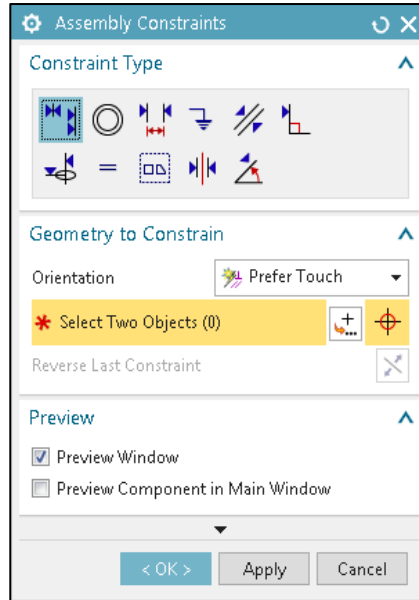
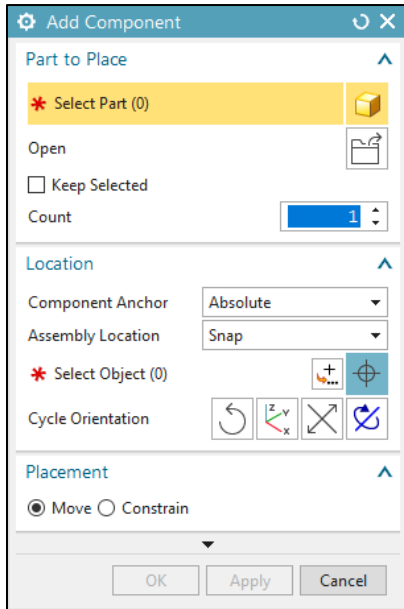
The design in which you want to insert another design must be open and active in NX.

1. Click **3DEXPERIENCE > Add** from 3DEXPERIENCE toolbar.

Following dialog box opens.



If 'Standard NX Dialog' button is clicked then , standard NX 'Add Component' functionality is initiated, which is followed by 'Assembly Constraints' dialog. Using this functionality designs from disk can be added to Assembly with constraints and positioning.



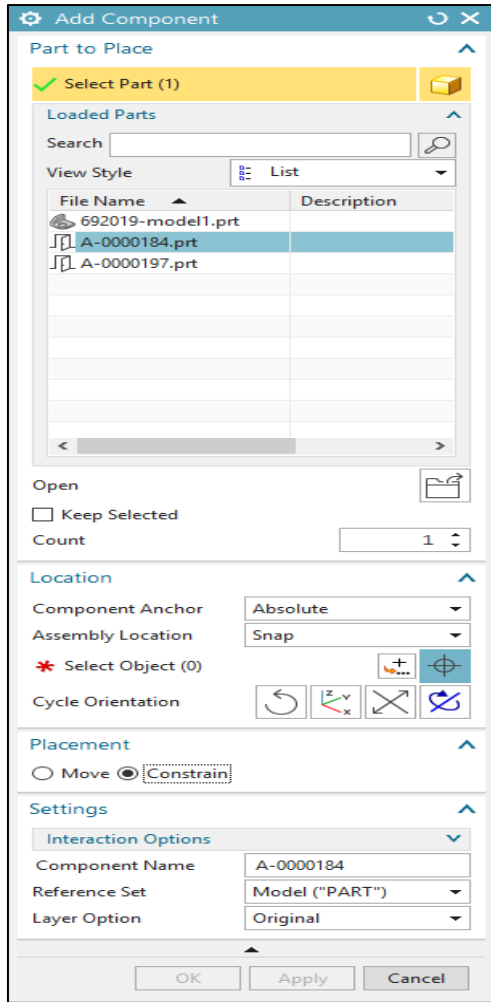
If 'Download Part from 3DEXPERIENCE and Add to Assembly' button is clicked, then

2. 3DEXPERIENCE Insert dialog Opens, same as 3DEXPERIENCE Open dialog.
3. Search for the design using **Search** option in **3DEXPERIENCE Open** dialog box. See [Searching Designs](#) for more details.

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

4. Click **Insert** or right-click the design and click **Insert**.

The selected design is downloaded to checkout directory followed by 'Add Component' dialog, where downloaded component is preselected. Using this functionality designs downloaded from 3DEXPERIENCE to disk can be added to assembly with constraints and positioning.



Opening Particular Iteration of Design

This task shows you how to open an earlier or later iteration of a design from 3DEXPERIENCE in NX. 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.

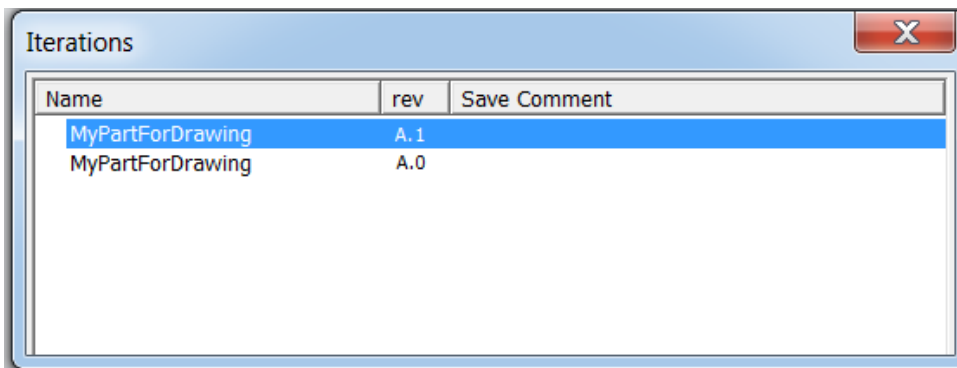
- a) Click **3DEXPERIENCE > Open** from NX tool bar.

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

- b) 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.

- c) 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.

- d) Select a single iteration to be opened.

The selected iteration of the design is opened in NX.

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 NX. This feature is used to view the expanded structure of an earlier design or a later iteration of a design.

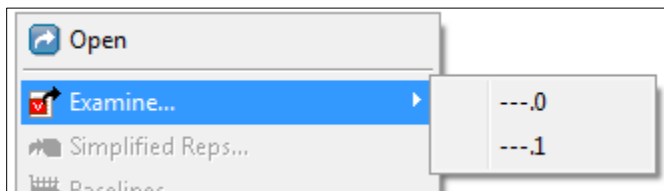
1. Click **3DEXPERIENCE > Open** from NX tool bar.

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](#)

Derived Output Support

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

Parts and assemblies:

JPEG

TIF

GIF

CGR

STEP_AP203

IGES

PNG

XML

Drawings:

JPEG

POSTSCRIPT

DXF

IGES

CGM

PDF

TIF

Unigraphics allows some level of control over the generation of JPEG, TIF, GIF, PNG and CGM files. Refer to the `UGDerivedOutputOptions` section for more information.

Creation of Single PDF from multiple drawing sheets

The integration supports the creation of pdf derived outputs from drawings in two ways, controlled by the configuration of the GCO attribute `UGDerivedOutputOptions` :

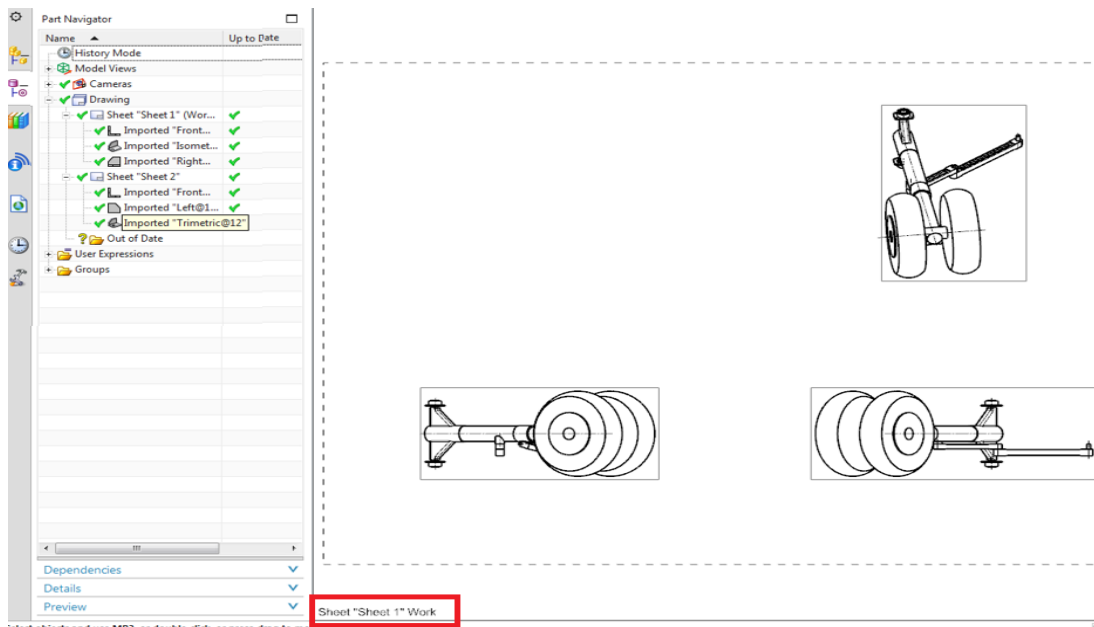
- The creation of a pdf for each sheet present in a drawing
- The creation of a single pdf that contains all sheets in a drawing **(NEW)**

If the latter option is configured, if there are multiple sheets found on the NX Drawing, it will consolidate into one PDF and check into the Derived Output Object within 3DEXPERIENCE.

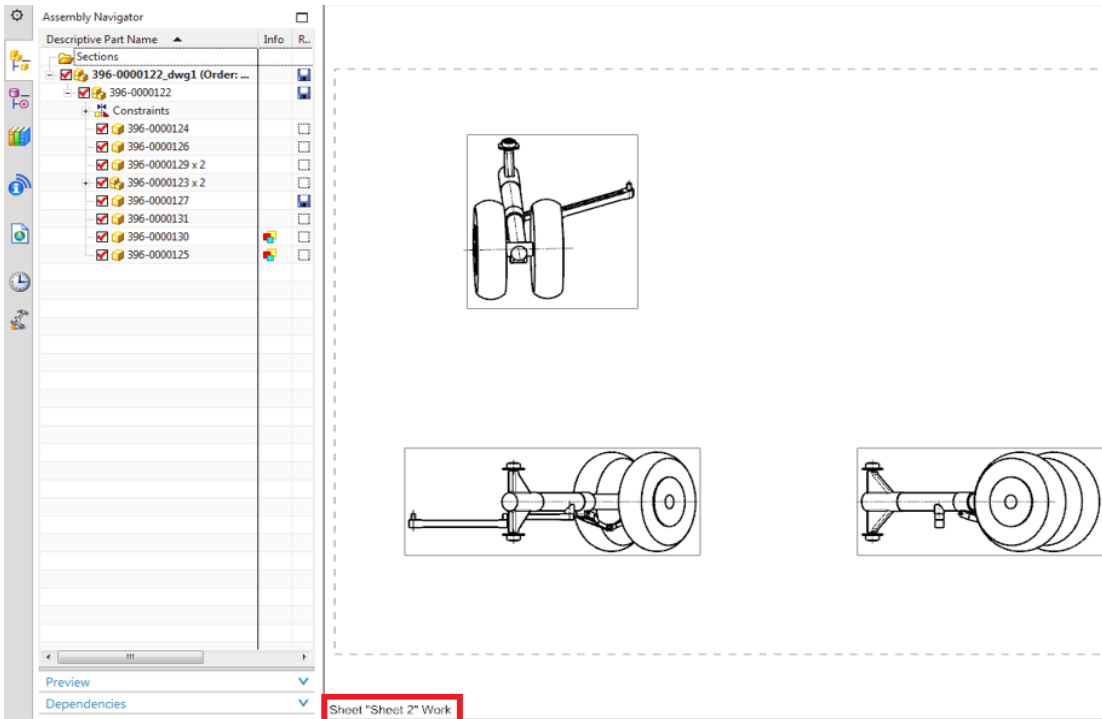
Example:

We have a drawing with two different sheets as shown below.

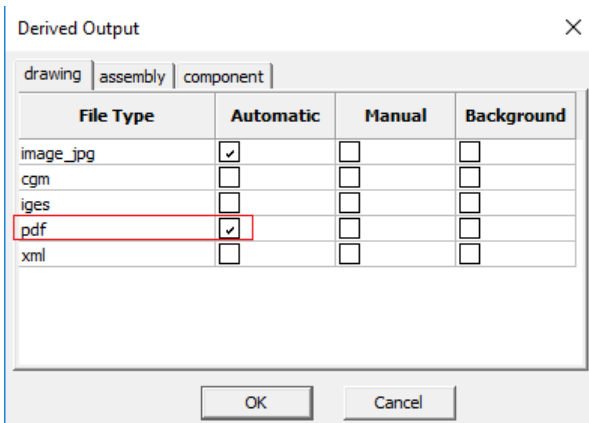
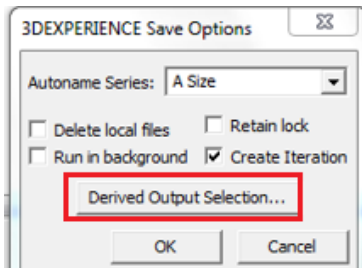
Sheet 1:



Sheet 2:



Now user saves the drawing, by selecting derived output option as pdf.



After save operation is successful, click on 'Explorer in 3DEXPERIENCE' to view the drawing.

396-0000122_dwg1 State : Under Global Design → Design Frozen

396-0000122_dwg1.prt Owner : ramesh

UG Drawing (---) Modified : 9/7/2015 8:42:57 AM

396-0000122_dwg1

Summary Lifecycle Navigate Where Used

Actions Edit Details...

Thumbnail

Actions

Active Iteration UG Drawing 396-0000122_dwg1 ---.0

Type UG Drawing

Name 396-0000122_dwg1

Revision ---

Description

Title 396-0000122_dwg1.prt

State Under Global Design

Owner ramesh

Locked By

Save Comment

Policy Design TEAM Definition

Related Models Derived Output Related Markups

Download Attach...

Name	Rev	FileName	Action
396-0000122_dwg1_drawing	---.0	396-0000122_dwg1.pdf	
396-0000122_dwg1_drawing	---.0	396-0000122_dwg1.jpg	

From the 'Derived Output' option, click on the pdf download icon.

Below is the screenshot for the single PDF with 2 pages, each page for one drawing sheet.

File Edit View Window Help

Open

1 / 2

18.6%

Tools Fill & Sign Comment

Sign In

Export PDF

Adobe ExportPDF

Convert PDF files to Word or Excel online.

Select PDF File:

396-0000122_dwg1.pdf 1 file / 373 KB

Convert To:

Microsoft Word (*.docx)

Recognize Text in English(U.S.)

Change

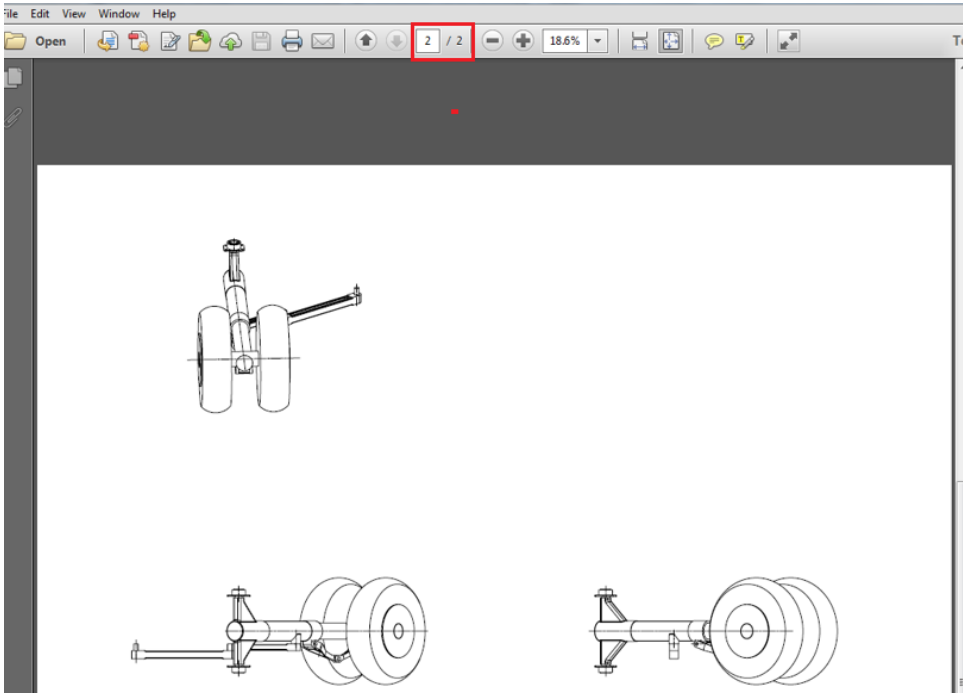
Convert

Create PDF

Edit PDF

Send Files

Store Files



Derived Output Enhancements

JPEG derived output support for specified modelling view

JPEG derived out can now be generated for a specific modeling view regardless of the model orientation in session. The view for JPEG derived output is to be generation needs to be specified in settings.ini under GENERAL section. By default the view for image generation is considered to be Trimetric. Following is an example of setting isometric view for JPEG generation.

```
view_for_image_generation = Isometric ;
```

PDF derived output support for custom widths

PDF derived output can now be generated with custom widths. Custom widths should be specified in wdf file. This wdf file must be specified in settings.ini GENERAL section as shown below.

```
wdf_file = C:\temp\Test_wdf.wdf ;
```

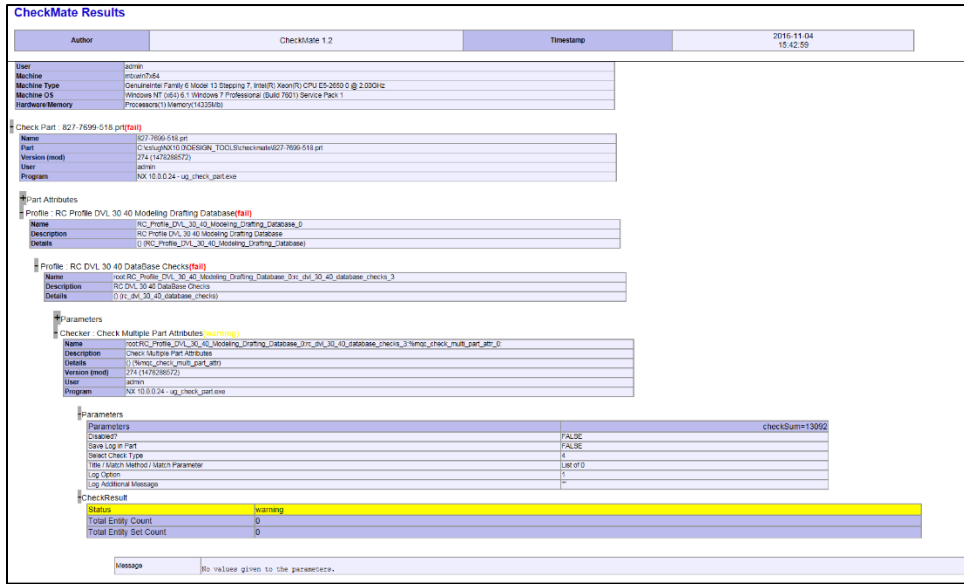
in GCO preferences on 3DEXPERIENCE server, Administrator should change attribute 'UGDerivedOutputOptions' to specify pdf width option as 'custom_palette' as shown below

```
pdf|color=black_on_white,width=custom_palette,tolerance=0.0001,bannerFlag=1,bannerMsg=,delete_on_checkin=yes,all_sheets=yes
```

Model Validation with Checkmate Automation

NX Checkmate provides an automated, customizable tool for knowledge-driven validation. The resulting validation file is an xml. Integration will handle it as additional derived output. Additional flexibility is offered to user that enables him to provide custom profiles for validation.

Checkmate is a native NX function that provides users with additional analysis of the associated design structure. There will be an option in the Derived Output dialogue that allows users to select 'xml' and in doing so the 3DEXPERIENCE Connector will automatically run the native function and store the resulting output file in 3XEPERIENCE as a Derived Output. Checkmate output will be in XML/XSL format. With this format the files are viewable with any web browser. A sample validation result is as shown below.



Settings.ini configurations

- **default_checkmate_profile**

If design to be analyzed has attribute 'CHECKMATE_PROFILE' with value defined as the checkmate profile to be used for validation, then integration uses it for checkmate analysis. If this attribute is not defined on the design, then the default checkmate profile defined in settings.ini will be used for checkmate analysis. Following is an example of default checkmate profile defined in settings.ini.

default_checkmate_profile = mqc_check_assembly_clearance_analysis ;

- **checkmate_stylesheet**

Checkmate stylesheet should be specified as shown below

checkmate_stylesheet = <path of checkmate_log.xml> ;

This referral style sheet is required to open the xml file in readable form.

Environment variable configurations

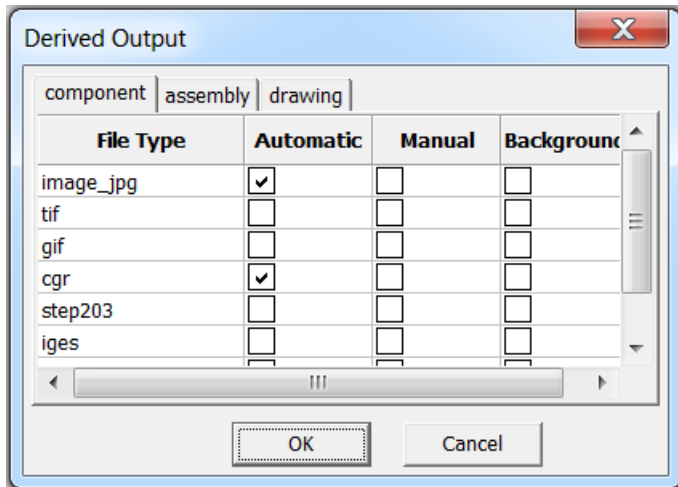
Following environment variables need to be set up, please refer NX documentation for additional information.

- **UGCHECKMATE_DIR**
- **UGCHECKMATE_USER_DIR**
- **UGCHECKMATE_ALLOW_CHECKER**

Once the above settings are done, during the save operation, 'xml' derived output should be selected for integration to perform Checkmate analysis, the analysis report in the form of xml will be generated and checked in.

Creation of CGR Derived Output during Save operation

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.



Lock/Unlock

Locking a design in 3DEXPERIENCE reserves the right to modify the design for the user whereas unlocking any design revokes any rights 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.

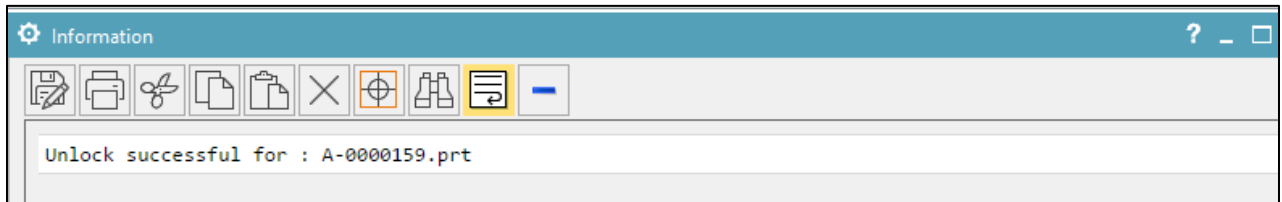
It is recommended to retain the lock for your designs while saving the designs to 3DEXPERIENCE. Leaving a design unlocked in 3DEXPERIENCE allows other users to modify the design and save their changes to 3DEXPERIENCE.

Unlocking a design in 3DEXPERIENCE revokes the right to modify the design by the current user. This gives open access to other user to lock the design if need for further CAD modeling. Also if user has some extra privileges then a particular design can be unlocked even though its locked by some other user.

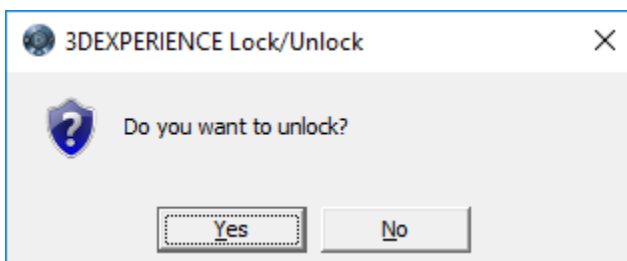
Locking and Unlocking Designs

There are multiple ways to lock a design in 3DEXPERIENCE Connector for NX. This section explains various ways to lock designs in 3DEXPERIENCE.

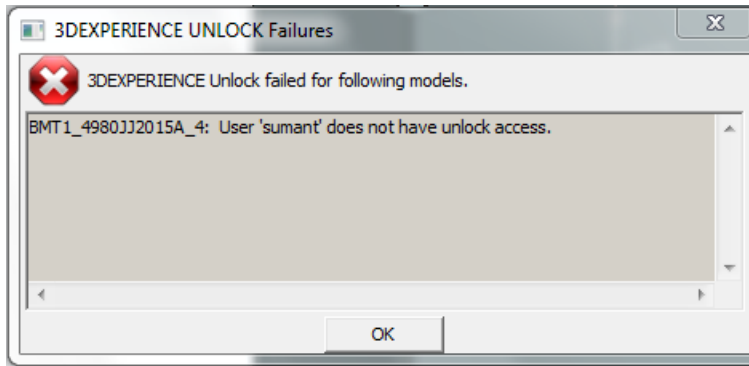
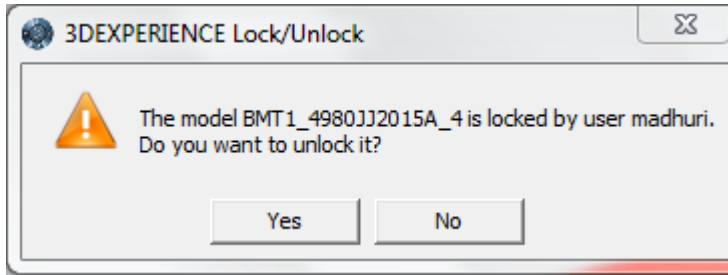
- a) If the checkout design is not locked by any user, then the feature will go ahead and the lock the design with the current user and successful message will be displayed to the user:



- b) If the checkout design is locked by same user, then a message will be displayed asking for confirmation if feature can unlock the design as shown below:



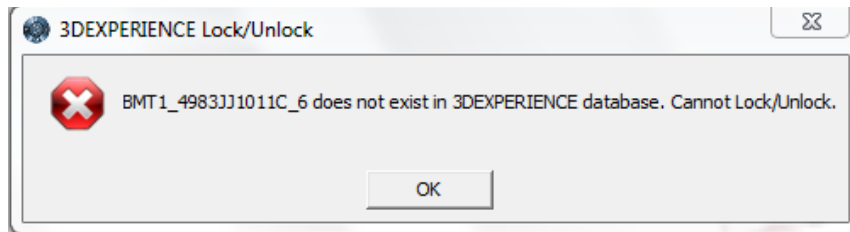
- c) If the checkout design is locked by some other user, then a confirmation message will be displayed. If user clicks on "Yes" depending upon the privileges current user has either an error message will be displayed as shown below or the design will be locked by current user:






- d) Use **3DEXPERIENCE Open** menu to bring Open dialog box, search and select NX design that you want to lock and use **Lock** or **Unlock** contextual (right mouse button) menu to lock or unlock selected design.



Note: If the design is newly created and user clicks on Lock/Unlock feature an error message will be displayed, as shown below:



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.

- a. You can also lock design at the time of checkin to 3DEXPERIENCE using Right Mouse Button -> Lock. Select NX 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'.

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

Validate Name

Validate Name feature is used to check the naming convention of the newly created file. The following sections describe the concept of validate name, steps for validating and renaming CAD model names.

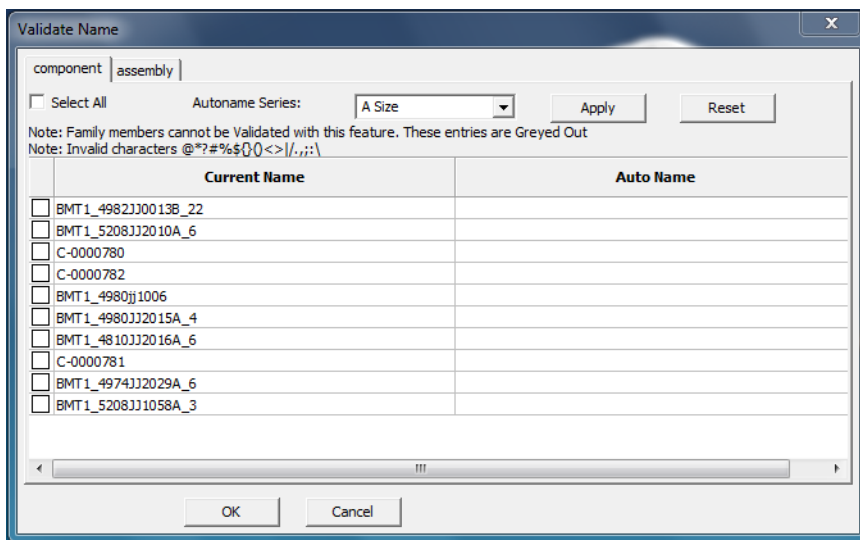
About Validate Name

The Validate Name feature will check for naming of all the newly created CAD models in session, in case user has used any invalid or unsupported characters while naming the file it will be displayed in Red color and *Italic* font. The feature allows user to rename the CAD models either by using the Autaname feature with the given Autaname series or user can manually enter the 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 consist of dedicated **tabs** for NX model type supported by Integration,

Click on **Select All** to select all the checkboxes to select the corresponding CAD object names for applying the autaname from Autaname series

Click **Autaname Series:** to assign a name from a predefined series (there series need to be created by your Administrator)

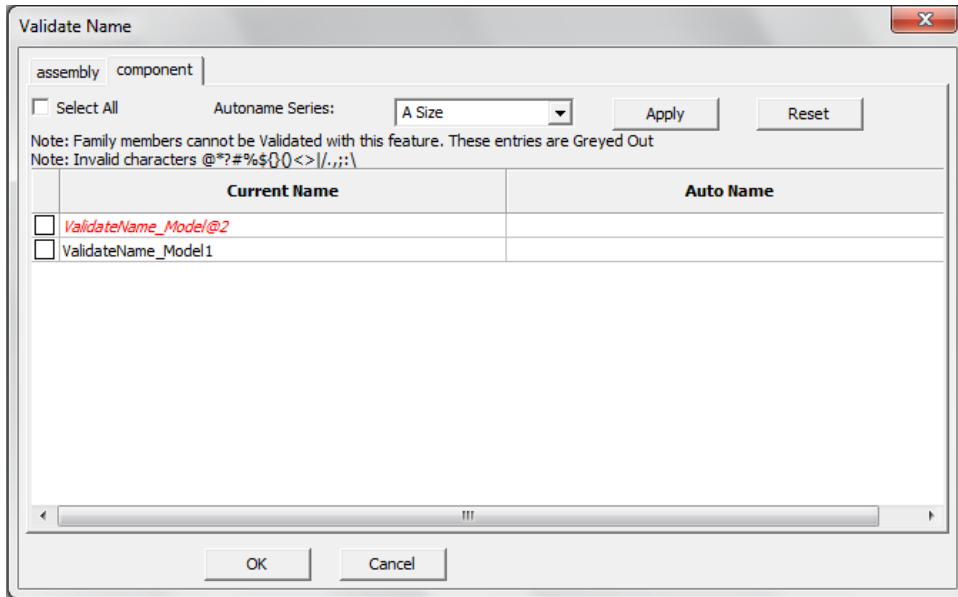
Click **Apply** to apply the autaname series name

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

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

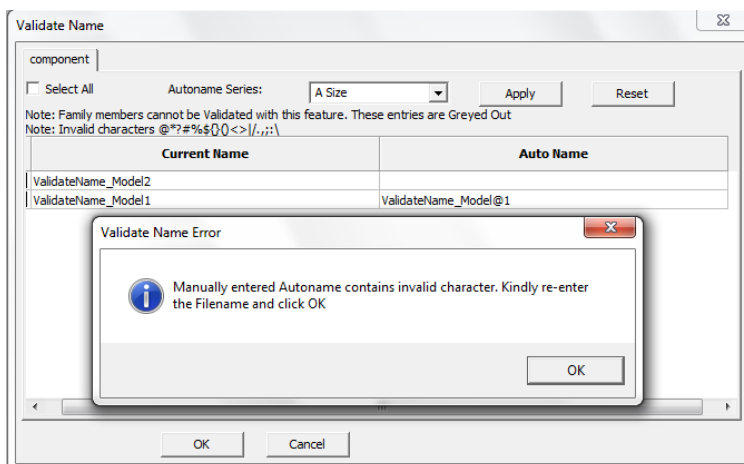
Click **Cancel** to return to NX model tool without making any changes

3. If the current name consists of any invalid or unsupported characters then the CAD model name will be displayed in *Italic* font and Red color, as shown below:

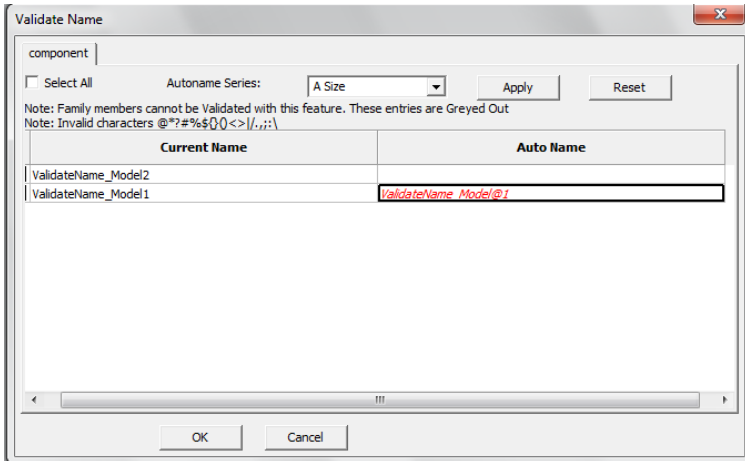


Validate Name feature allows user to either select Autoname series to generate a new Autoname or user can manually entry any value.

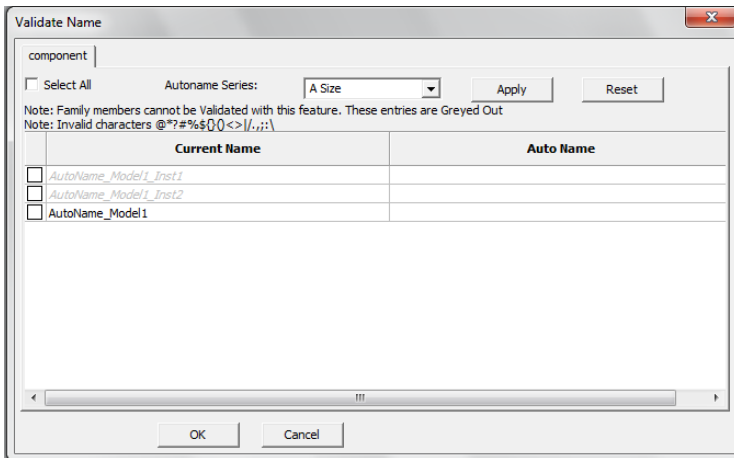
As per note above, if user is manually entering the name then invalid characters displayed at the top note should not be used. If user still uses any invalid character, then an information message is displayed to the user as shown below:



Once user click on “OK” the control goes back to the dialog and the entry in the “Auto Name” column with invalid character will be displayed in *Italic* font and Red color, as shown below:



Note: Validate Name feature does not support name validation of Family members. These entries of Family members will be displayed as not editable rows and will be greyed out, as shown below:



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 information of the dependent object structure in form of an expanded tree structure. 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 modify these options and view the effect 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 stream for all the objects in expanded structure.

Examining Designs

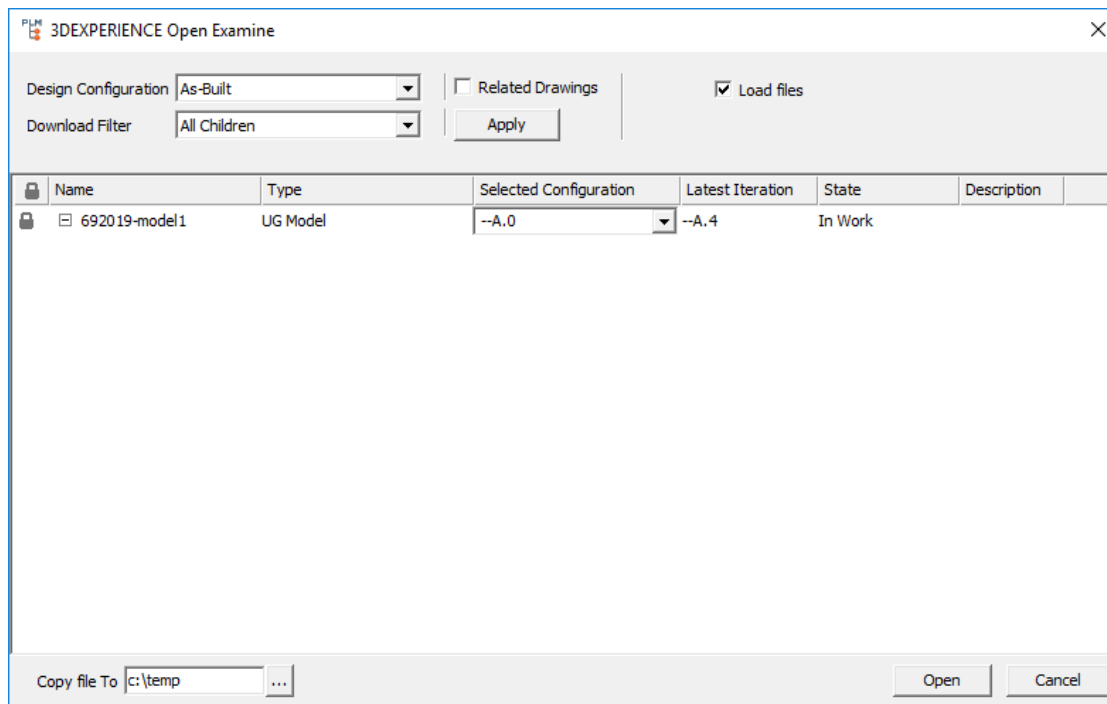
1. Click **3DEXPERIENCE > Open** in NX ribbon/toolbar.

The **3DEXPERIENCE Open** dialog opens.

2. Search for the 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 NX 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, 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.

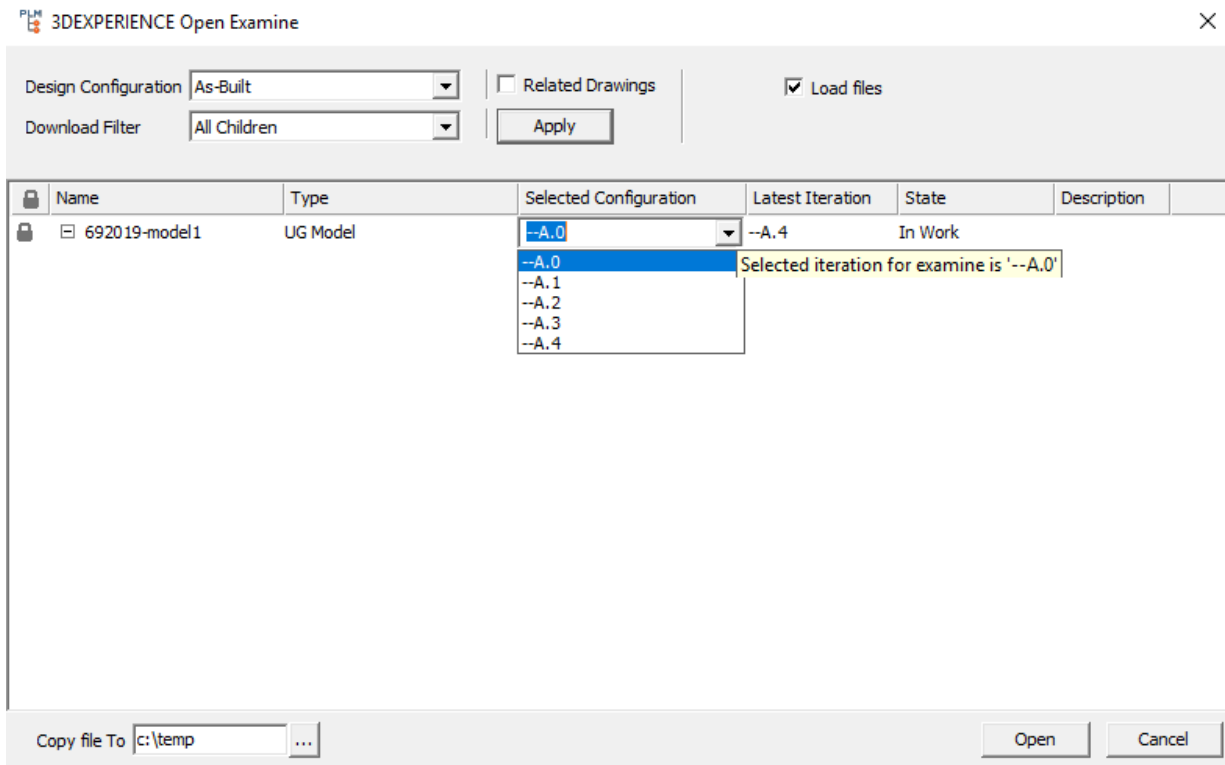


Note: Configuration settings made in the Examine Dialog are not remembered in the Open dialog.

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


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

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



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

7. Other option in **3DEXPERIENCE Open Examine** dialog box:

Copy file to. 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 will not be remembered in 3DEXPERIENCE Open dialog.

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



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

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 NX, baselines can be created to capture the structure of a design active in NX that exists in the 3DEXPERIENCE database. The baselined structures can be opened at any stage of the design in NX from **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** 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.

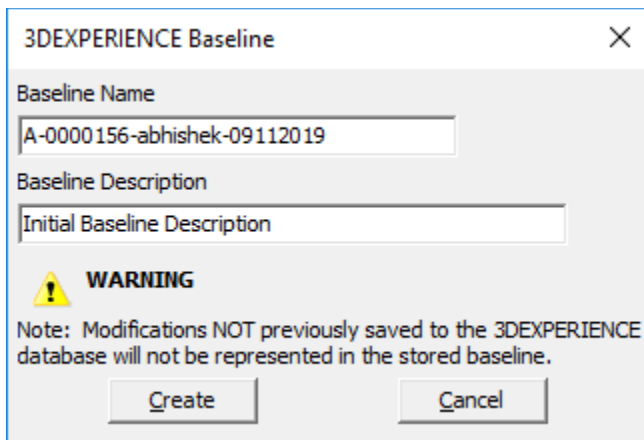
Baseline Designs

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 NX.

1. Click **3DEXPERIENCE > Baseline** from NX 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.

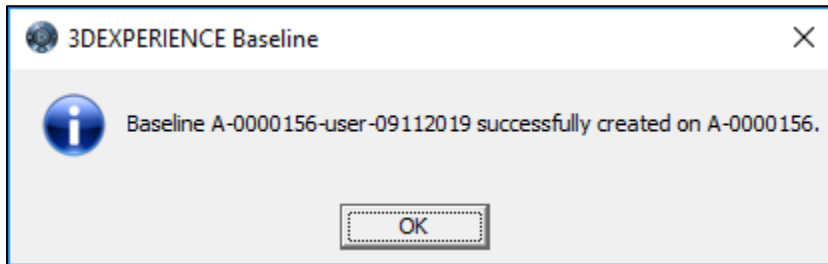
2. Click **OK**.

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 NX 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 NX with the specified parameters.

Opening Baseline Designs

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

Baseline must exist for the design.

1. Click **3DEXPERIENCE > Open** in NX toolbar.

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

2. Search for the baselined design in the **3DEXPERIENCE Open** dialog box. See [Searching Designs](#).

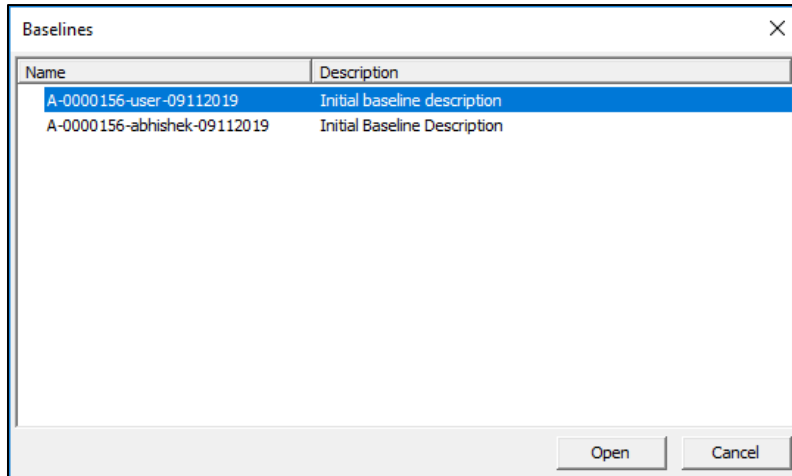
The results are listed in the **3DEXPERIENCE Open** dialog box.

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: 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 opens in NX retaining the design's characteristics at the time the baseline was created.

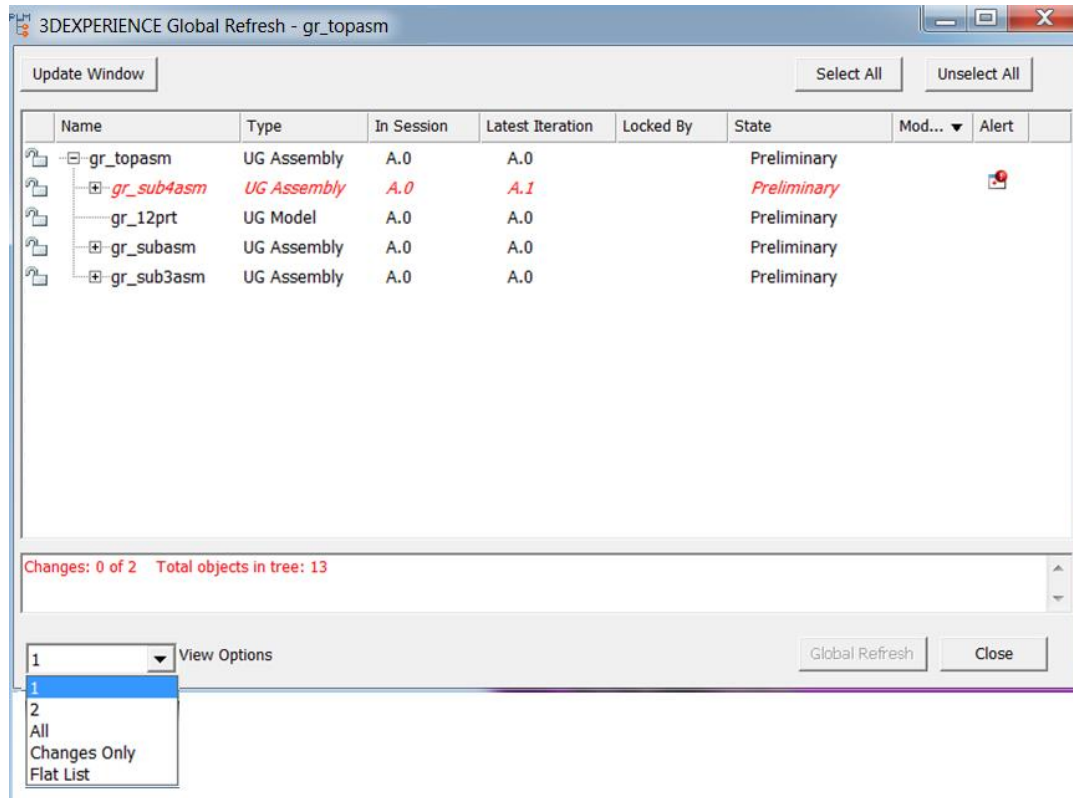
Global Refresh

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

To Start Global Refresh: Click **3DEXPERIENCE > Global Refresh** from NX toolbar.

- 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.
- 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:
 - i. There are no items selected.
 - ii. 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 NX or 3DEXPERIENCE without closing this dialog; he/she can put this dialog on the side and work on his/her tasks.

- View Filters: The user is now given several different options on how they wish to view their structure.

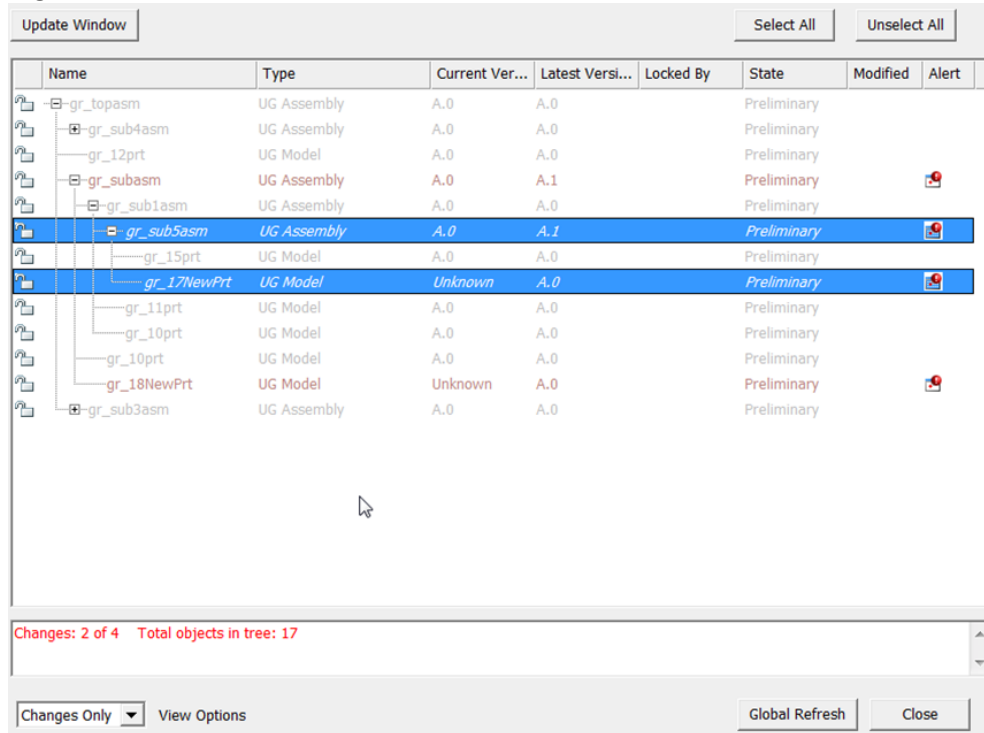


Various options are

- 1,2,3..., 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.
- In the dialog the different display (color/italics..) mean: (*info also available on tool tip of the row*)
 - Black, non-bold, state the component is up to date.
 - 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
 - Blue bold state the component exists just in the CAD structure and not in the 3DEXPERIENCE structure.

- Structure Update: 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.

Update Window								Select All	Unselect All
Name	Type	Current Ver...	Latest Versi...	Locked By	State	Modified	Alert		
gr_topasm	UG Assembly	A.0	A.0		Preliminary	Yes			
gr_sub4asm	UG Assembly	A.0	A.0		Preliminary				
gr_14prt	UG Model	A.0	A.0		Preliminary				
gr_12prt	UG Model	A.0	A.0		Preliminary				
gr_subasm	UG Assembly	A.0	A.1		Preliminary	Yes			
gr_sub1asm	UG Assembly	A.0	A.0		Preliminary	Yes			
gr_sub5asm	UG Assembly	A.1	A.1		Preliminary				
gr_17New	UG Model	A.0	A.0		Preliminary				
gr_15prt	UG Model	A.0	A.0		Preliminary				
gr_11prt	UG Model	A.0	A.0		Preliminary				
gr_10prt	UG Model	A.0	A.0		Preliminary				
gr_10prt	UG Model	A.0	A.0		Preliminary				
gr_18NewPrt	UG Model	Unknown	A.0		Preliminary				
gr_sub3asm	UG Assembly	A.0	A.0		Preliminary				
gr_FamInst2	UG Model	A.0	A.0		Preliminary				
gr_13prt	UG Model	A.0	A.0		Preliminary				
gr_10prt	UG Model	A.0	A.0		Preliminary				

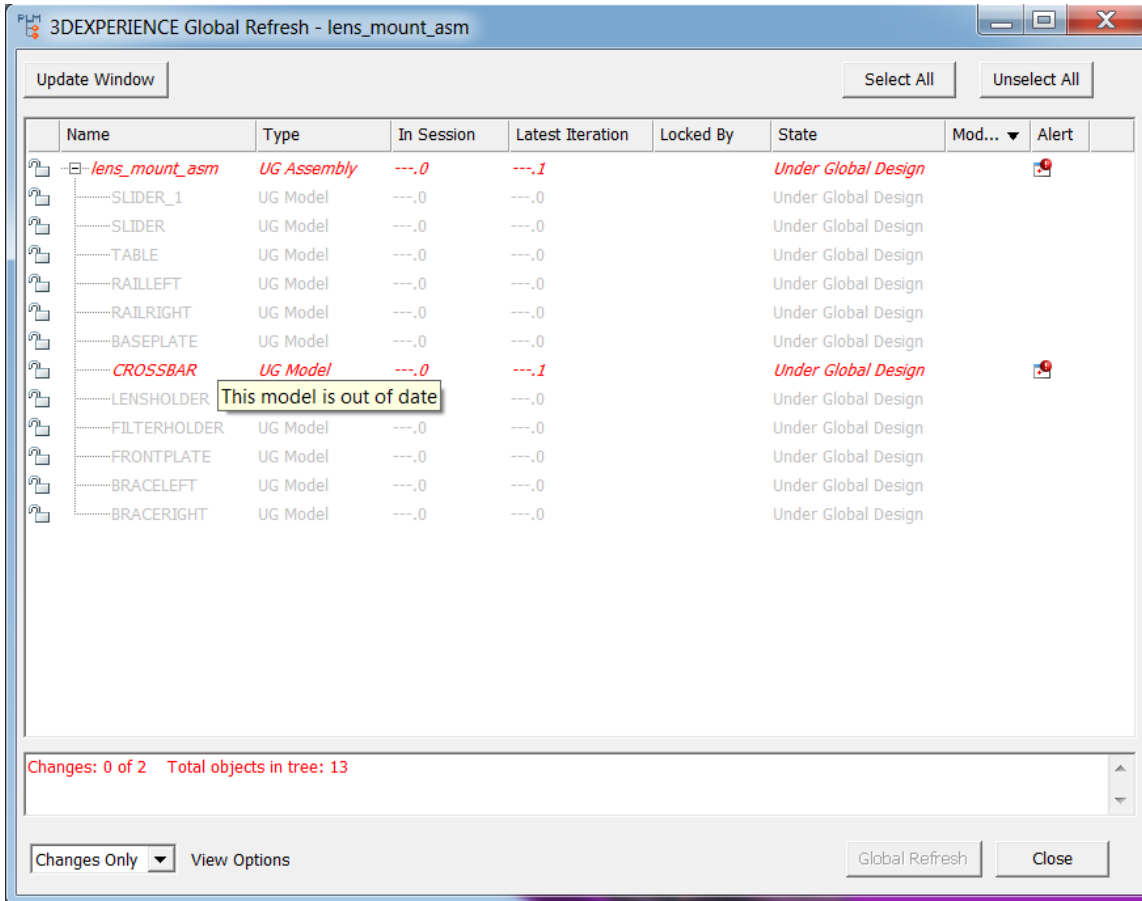
Changes: 0 of 2 Modified In CAD: 0 of 3 Total objects in tree: 17

All View Options Global Refresh Close

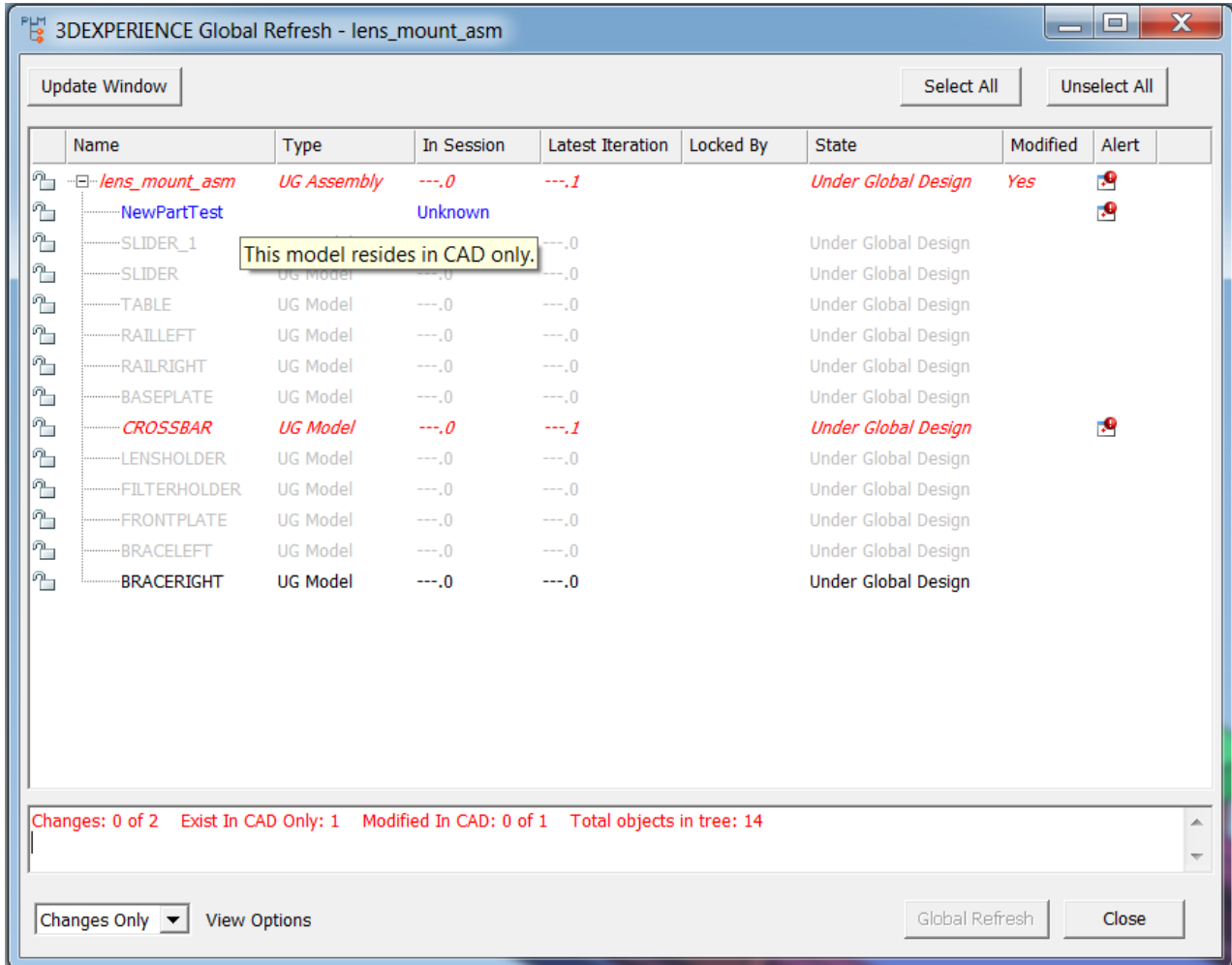


Note:

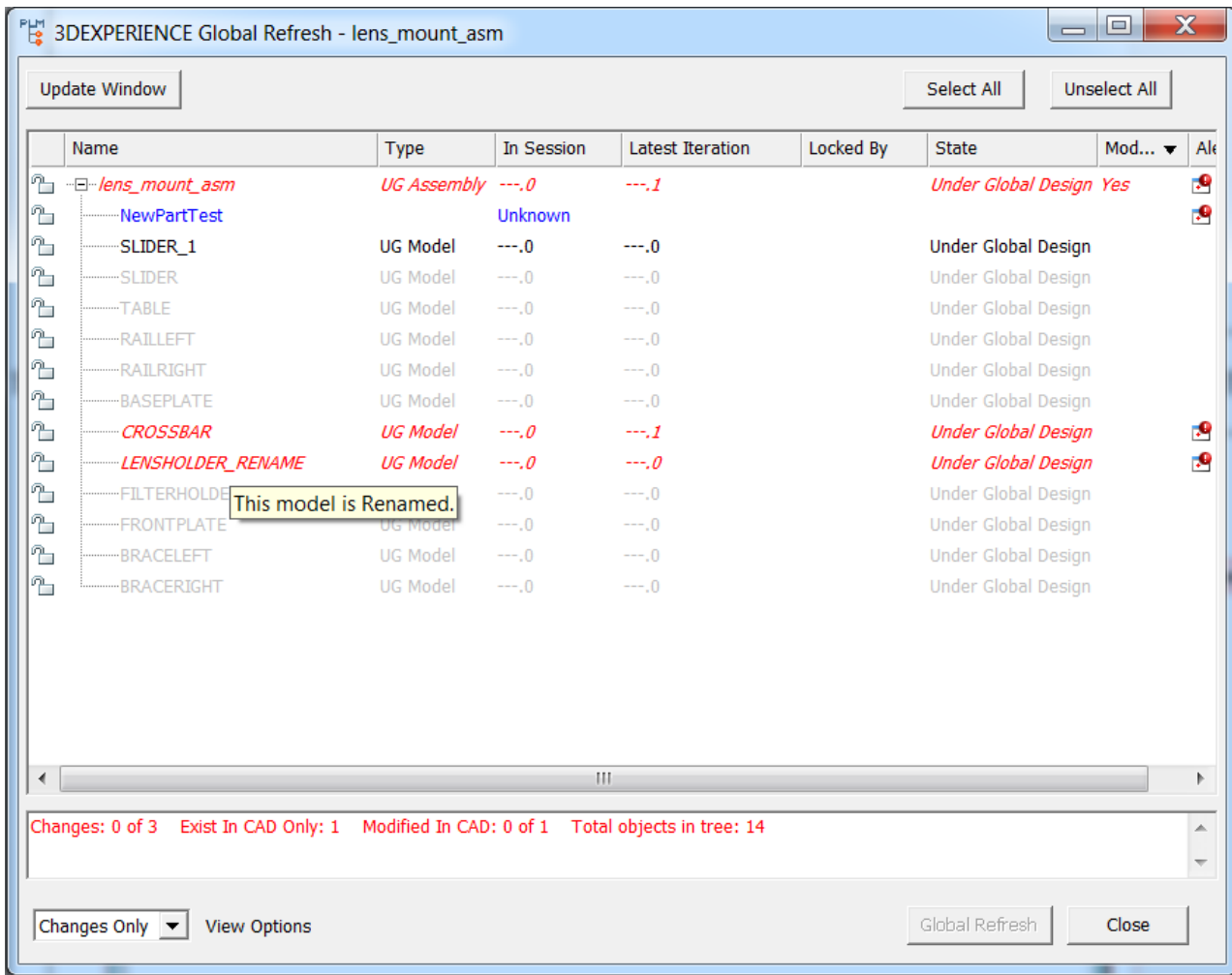
- This feature will not merge the design modifications made by the user with the latest iteration of the design downloaded from 3DEXPERIENCE; it can cause potential work loss. If the model displayed in the dialog was modified in the NX session, the integration will warn the user by putting a “Yes” in 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 (NewPartTest) added to just the CAD structure.



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



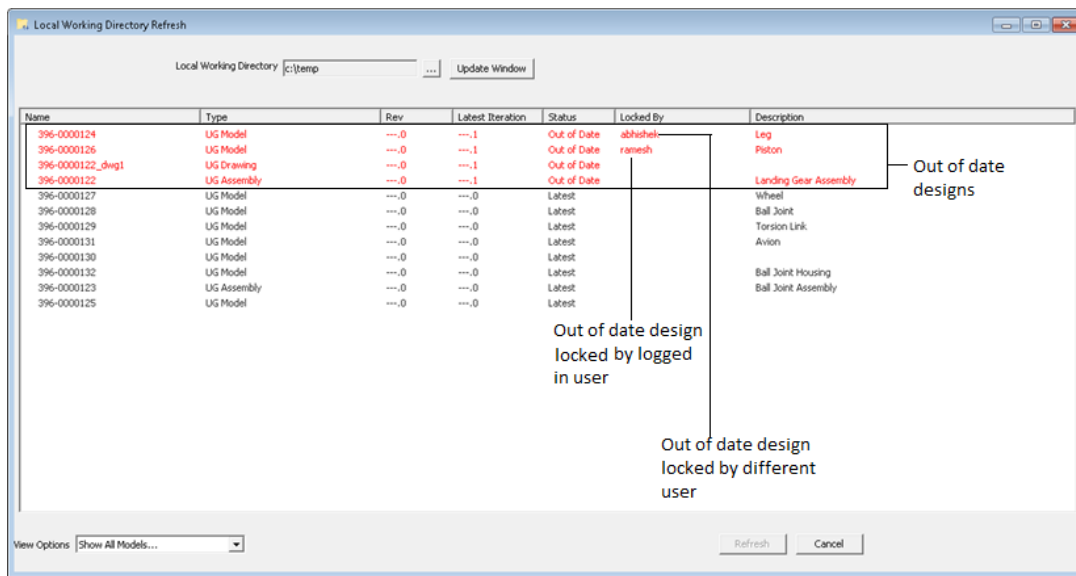
Refreshing Designs from Local Working Directory

The Local Working Directory of a NX 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 be able to know the status of the designs in the 'Local Working Directory' and refresh the models without bringing them in to NX session.

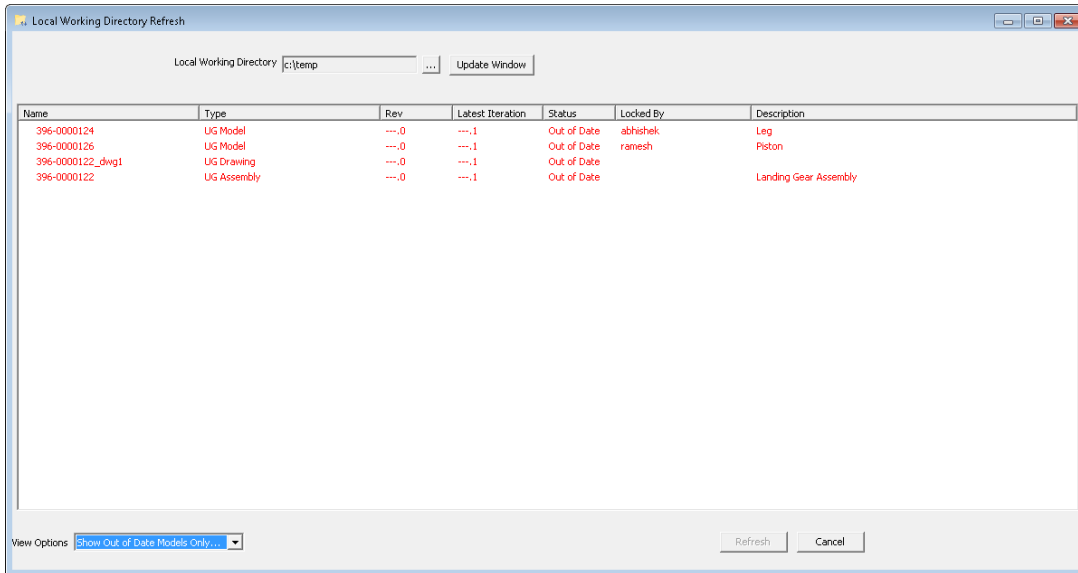
To use this feature user has to click on the 3DEXPERIENCE>Local Working Directory Refresh' menu. The local working directory refresh dialog is as shown below

It shows 'Local Working Directory' path which can be changed by clicking on the ellipsis, 'Update Window' button is used to refresh the dialog content. Dialog displays 'Name', 'Type', 'Rev', 'Latest Iteration', 'Locked By' and 'Description' attribute value.

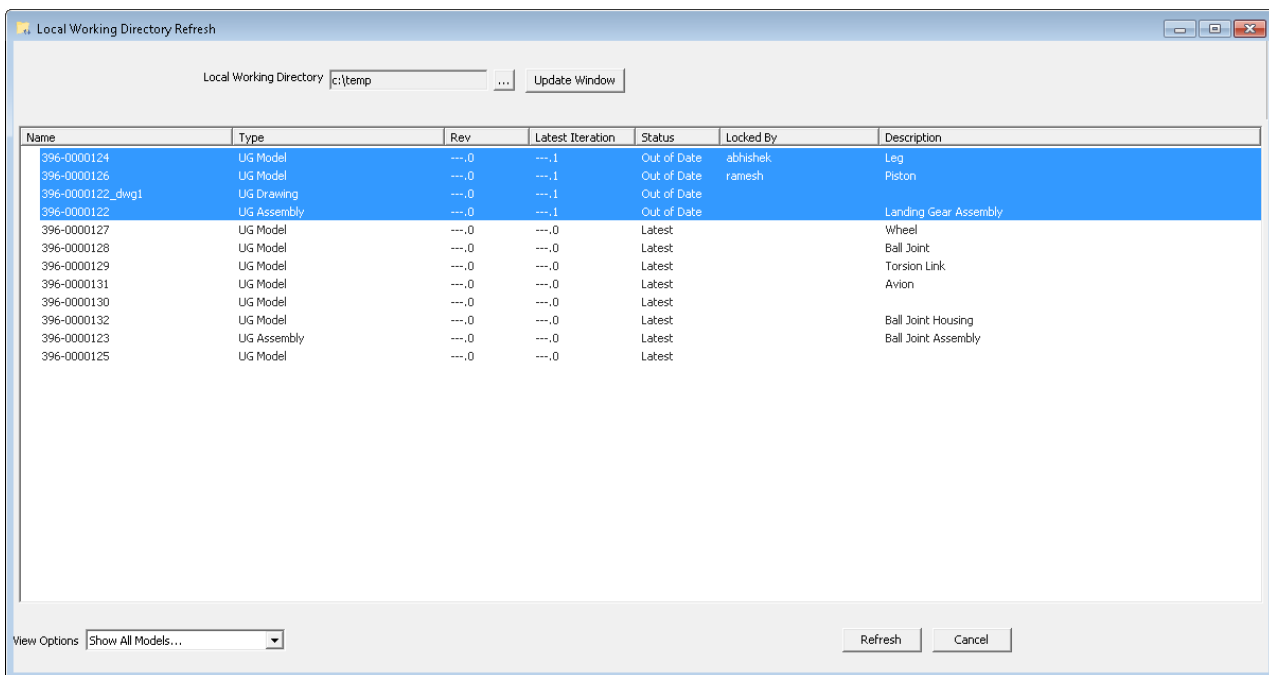
User can change the working directory by clicking on the ellipsis and the designs in that directory will be displayed in the dialog.



'View Options' filter is available at the bottom, 'Show All Models...' 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.

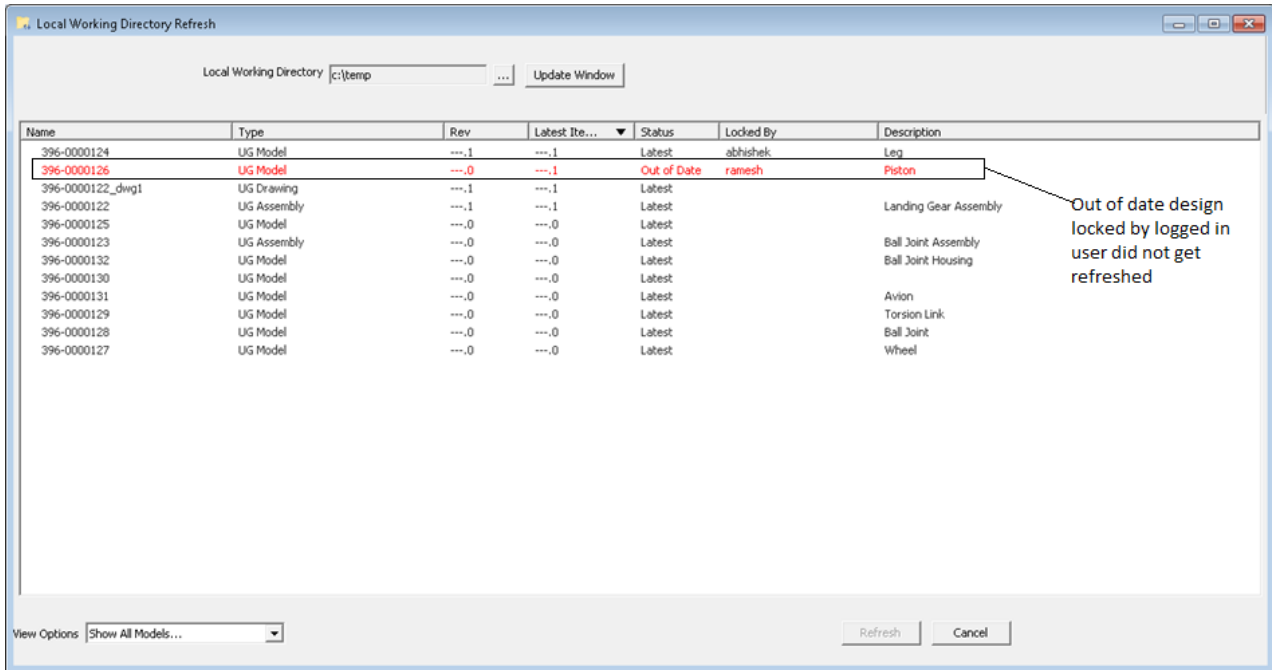


To refresh the designs, select the out of date designs and click the Refresh button.



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

The result can be seen below



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.

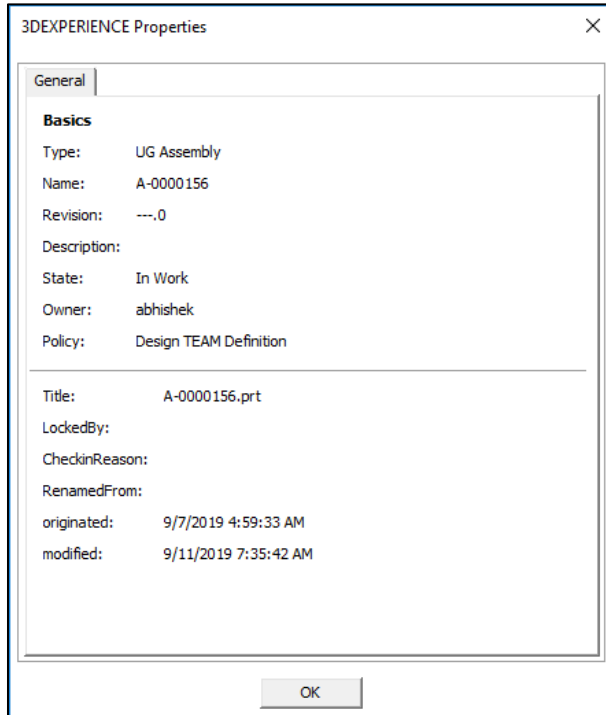
Viewing Properties

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

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

1. Click **3DEXPERIENCE > Properties...** from NX 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 NX.

The **Manage Workspaces** feature of 3DEXPERIENCE Connector for NX is used to view and manage a user's workspaces stored in both the 3DEXPERIENCE database, as well as the user's local drive.

Managing Designs Using Integration Exchange Framework Client

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



NOTE: The Integration Exchange Framework client must be installed for this feature to function.

1. Click **3DEXPERIENCE > Manage Workspaces** from the NX toolbar.

Two options for which workspace to be managed are given:

Local... Lists file location of the active model on the local disk. If the user has no active models, it will display the checkout directory defined in preferences.

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 workspaces accessible to the user.

2. You can use these folders for managing 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*.

Derived Outputs

This task shows you how to customize various derived outputs in NX.

- JPG, TIF, GIF, CGM and PDF – Use the GCO options to control the derived output

`image_jpg|width=500,height=500`

`tif|width=500,height=500`

`gif|width=500,height=500`

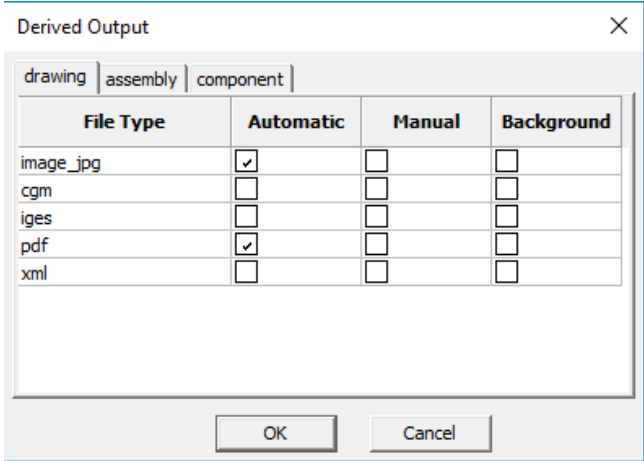
`cgm|color=as_displayed,width=standard,bannerMsg=,bannerFlag=1,all_sheets=no`

`pdf|color=black_on_white,width=standard,tolerance=0.1,bannerFlag=1,bannerMsg=,all_sheets=no`

- CGR – Click the 'Preferences->JT' menu, click 'modify configuration', go to the tessellation tab and change the 'Chordal Value' to 'Absolute' from the default 'Relative'. This option will improve the tessellation of NX models, and that would result in a better CGR output.
- Irrespective of the model dimensions, the CGR derived output file is always in mm.

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".



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 NX.

The **3DEXPERIENCE > Explore in 3DEXPERIENCE** menu is available from NX 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 NX toolbar if the design is active in NX
2. Search for the design in **3DEXPERIENCE Open** or **3DEXPERIENCE Insert** dialog box.
 - For details on searching for designs in 3DEXPERIENCE from NX, 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**

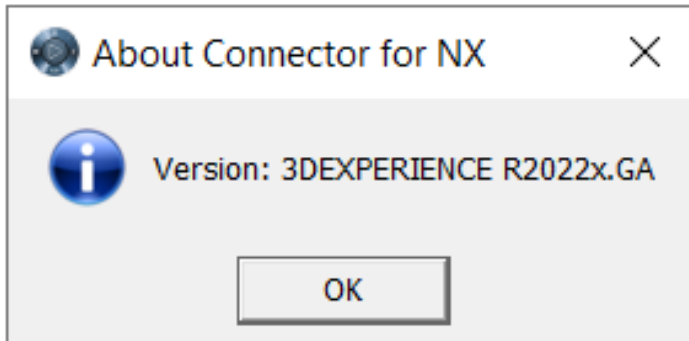
The CAD Portal View page of the design opens in the browser.

About

This task shows you how to find the iteration of the 3DEXPERIENCE Connector for NX.

Click **3DEXPERIENCE** > **About...** from NX toolbar

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



Design Export

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

Design Export is a new capability provided which enables users 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 NX, 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_UG>_<UG Model>_<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 functions available in 3DEXPERIENCE > design export dialog box.

Searching Designs

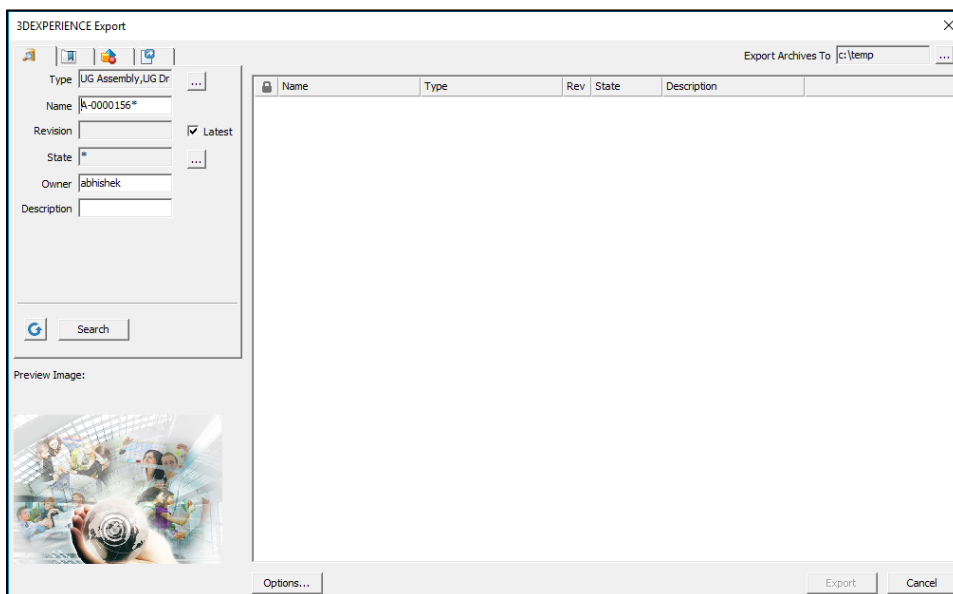
You can search for designs that exist in 3DEXPERIENCE vault from **3DEXPERIENCE > Design Export** dialog box. The following sections describe the steps used to search for designs.

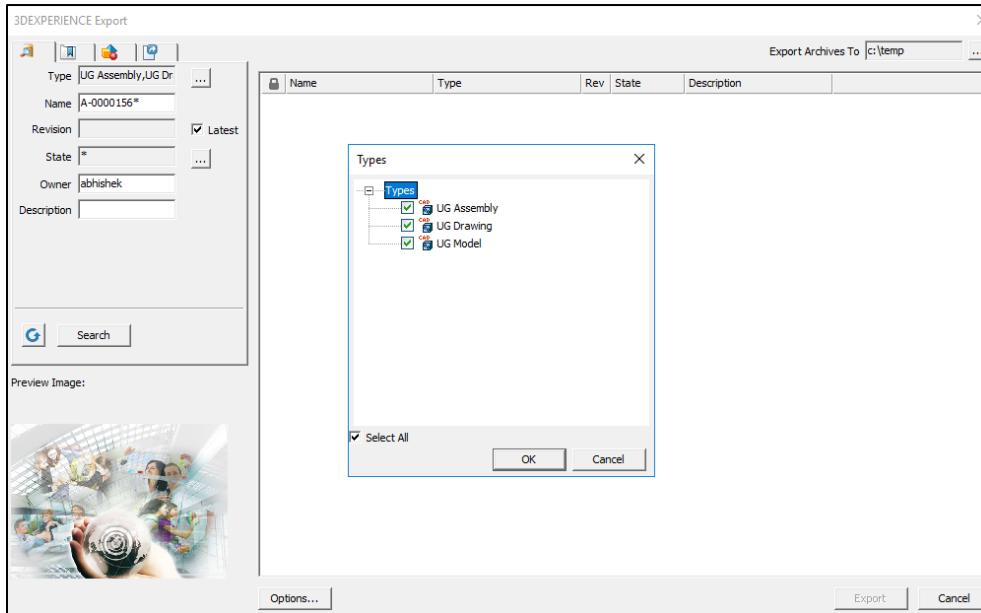
Searching Designs

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

1. Click **3DEXPERIENCE > Design Export** from NX 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.

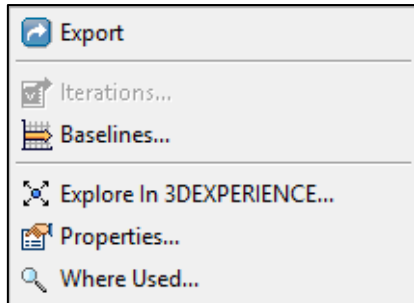
For Searching Designs Using Webform Driven Search Parameters, please refer "[Searching Designs](#)".

Search Results

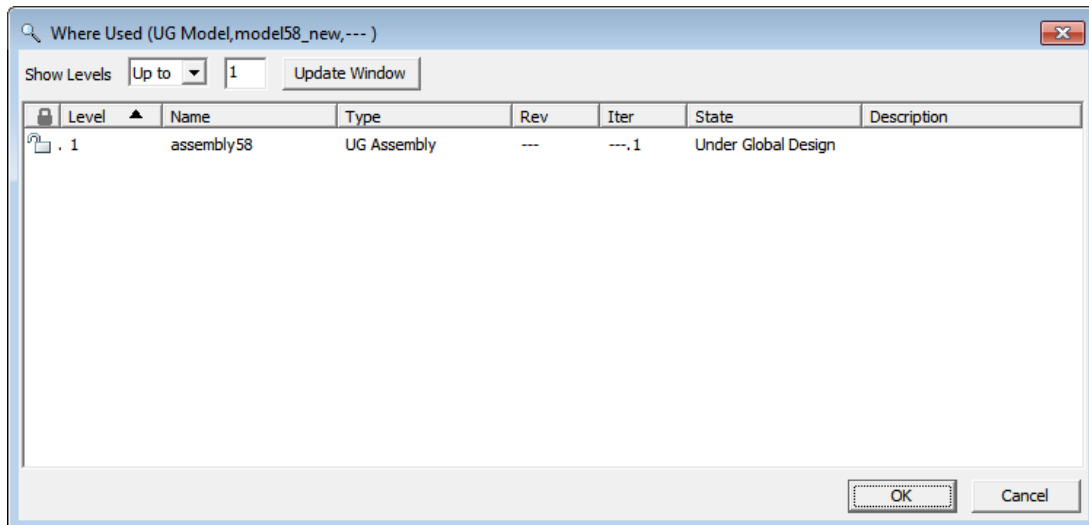
Regardless of the search method used, the search results are displayed in **3DEXPERIENCE Design Export** dialog box. For each design, the **3DEXPERIENCE Design Export** dialog box lists,

-  **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.
- **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 NX

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

The designs must already be saved in 3DEXPERIENCE. See [Saving Designs to 3DEXPERIENCE](#).

1. Click **3DEXPERIENCE > Design Export** from NX toolbar.

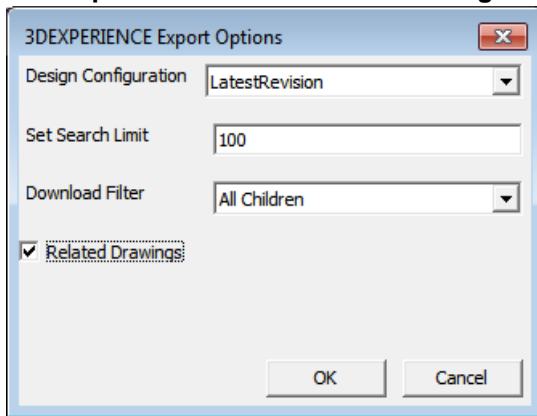
The **3DEXPERIENCE Design Export** dialog box opens.

2. Search for the designs using **Search** option in **3DEXPERIENCE Export** dialog box. See [Searching Designs](#)

The results of the search are listed.

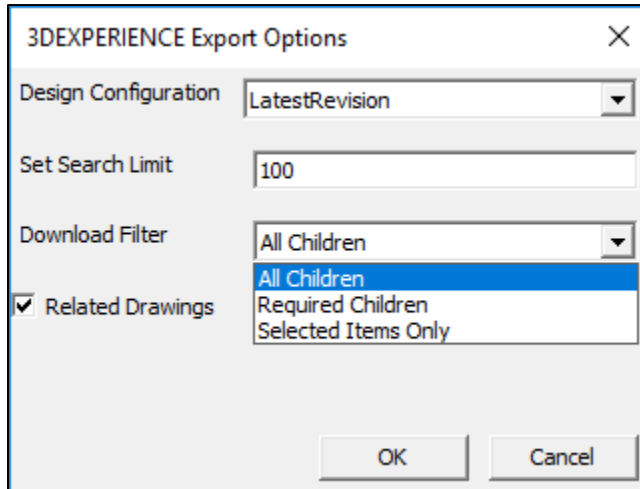
Select a design from the list to view the preview image of the design which is displayed in the left corner of **3DEXPERIENCE Design Export** dialog box.

3. Enter the following options in **3DEXPERIENCE Design Export** dialog box:
4. **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.
5. Click **Options** in **3DEXPERIENCE Design Export** dialog box.

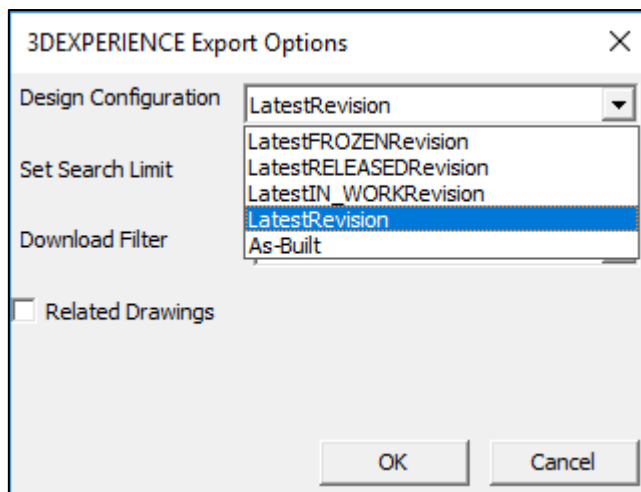


The **3DEXPERIENCE Design Export Options** dialog box opens. Choose options to be executed when you export a design from 3DEXPERIENCE:

- **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 selected 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 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, models connected by assemblyComponent and drawing relationships, on export. On export with this option, the user will not get models 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**.

6. Click **Export**.

During export, irrespective of RFA setup and Lock, unlock status of design, the designs are downloaded to a unique directory which later is archived in a zip file with same name 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.

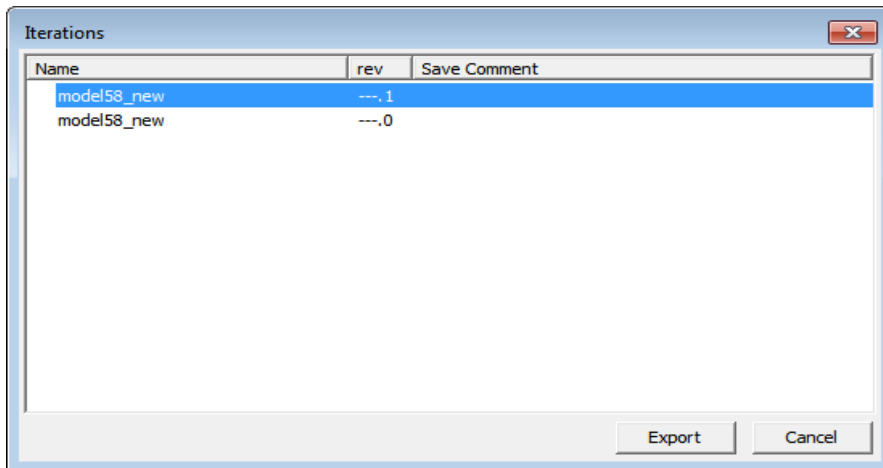
1. Click **3DEXPERIENCE > Design Export** from NX tool bar.

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

2. Search for the design using **Search** option in **3DEXPERIENCE Design Export** dialog box. See [Searching Designs](#)

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.

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 NX toolbar.

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

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

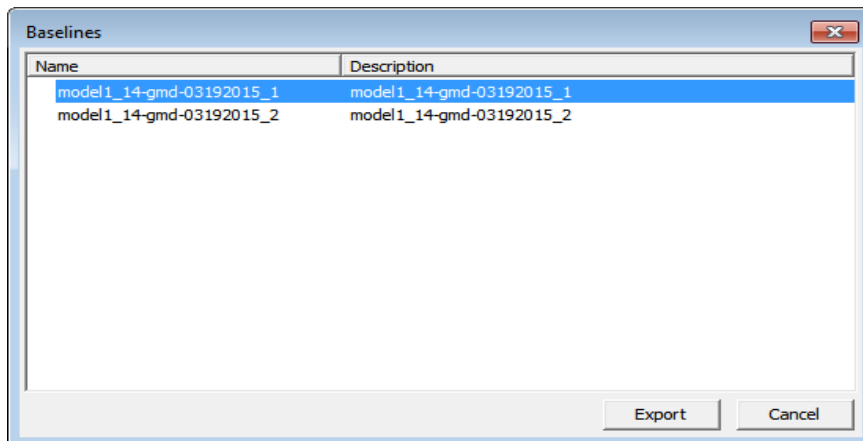
The results are listed in the **3DEXPERIENCE Design Export** dialog box.

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

Disconnecting From 3DEXPERIENCE

This task shows you how to disconnect from 3DEXPERIENCE from NX.

Click **3DEXPERIENCE > Logout...** from NX toolbar.

The connection to 3DEXPERIENCE is disconnected.

Help

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

Click **3DEXPERIENCE > Help...** from NX toolbar

The integration will take you to the **3DEXPERIENCE** 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 NX Integration Install>\bin\settings.ini.

Language settings

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

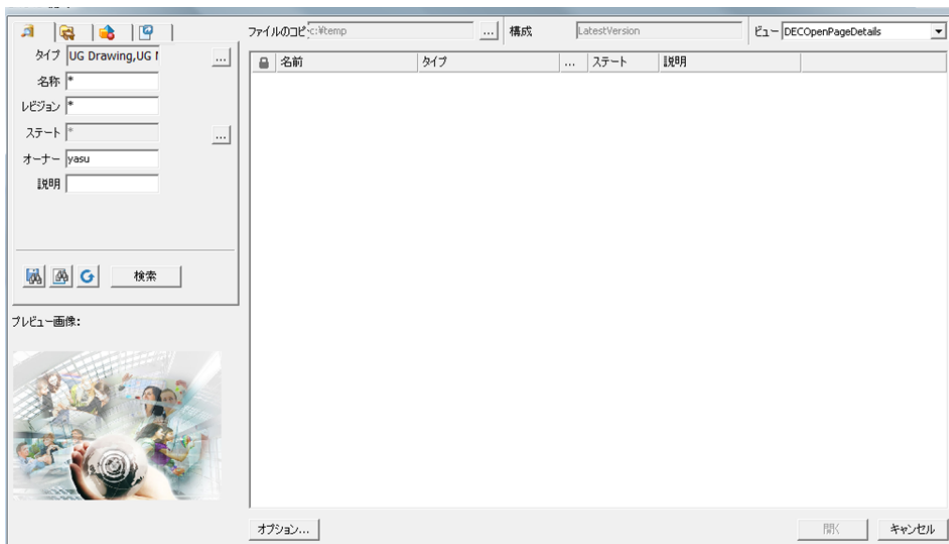
Use below steps to work in Japanese environment. If you need to change other supported language, change 'Japanese' to those supported, 'French' or 'German' or 'Italian' or 'Chinese' and use associated files.

- Set IEFClient and 3DEXPERIENCE Connector for NX 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, and 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 NX language
 - Change system variable UGII_LANG to "Japanese"

Below is snapshot of Open dialog in Japanese language



Balloon Transfer from Drawing

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

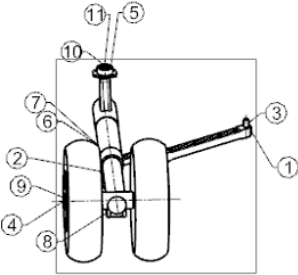
- Add *drawing, balloonnumbers|Associated Drawing,Balloon Numbers* to GCO MCADInteg-CADToMxRelAttribMapping.
- Set “Copy Relationship Attributes” on checkin to TRUE in Local Configuration or Preferences page
- Create Drawing of assembly, add balloons to drawing and checkin drawing to 3DEXPERIENCE

Multi-Level Drawing Balloon Transfer to EBOM Find Number

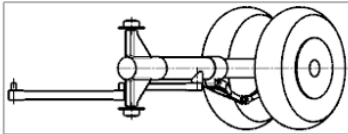
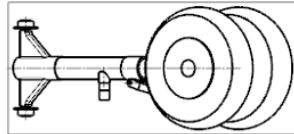
With this new enhancement, CAD users will no longer need to associate a drawing (containing BOM table) with each sub-assembly in order to get the Find Number values of all the components of the BOM structure. Just a single multi-level BOM table associated with top drawing would be enough to get the Find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

Assembly Navigator

Descriptive Part Name	Info	R..
Sections		
396-0000122.dwg1 (Order: Chrono)		
396-0000122		
Constraints		
396-0000124		
396-0000126		
396-0000129 x 2		
396-0000123 x 2		
396-0000127		
396-0000131		
396-0000130		
396-0000125		



11	396-0000122	1
10	396-0000124	1
9	396-0000126	1
8	396-0000129	2
7	396-0000123	2
6	396-0000132	2
5	396-0000128	2
4	396-0000127	1
3	396-0000131	1
2	396-0000130	1
1	396-0000125	1
PC NO	PART NAME	QTY

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

192.168.106.217:8080/ematrix/common/emxNavigatorDialog.jsp

396-0000122 rev 1: Engineering Bill of Materials

Name	Policy	F/N	Ref Des	Component Location	Description	State	Qty	UOM Type	UOI
396-0000125	EC ...	1		-1,0,0,0,0,-1,0,0,0,0,...		In W...	1.0	Proportion	EA
396-0000126	EC ...	9		0,0,-1,0,-0.59326434...		In W...	1.0	Proportion	EA
396-0000127	EC ...	4		-0.59326434715250...		In W...	1.0	Proportion	EA
396-0000129	EC ...	8		-0.75645980548548...		In W...	1.0	Proportion	EA
396-0000129	EC ...	8		0.766526086607377...		In W...	1.0	Proportion	EA
396-0000130	EC ...	2		-1,0,0,0,0,-1,0,0,0,0,...		In W...	1.0	Proportion	EA
396-0000131	EC ...	3		1,0,0,0,0,1,0,0,0,0,1,...		In W...	1.0	Proportion	EA

Part Family Support and Recommendations

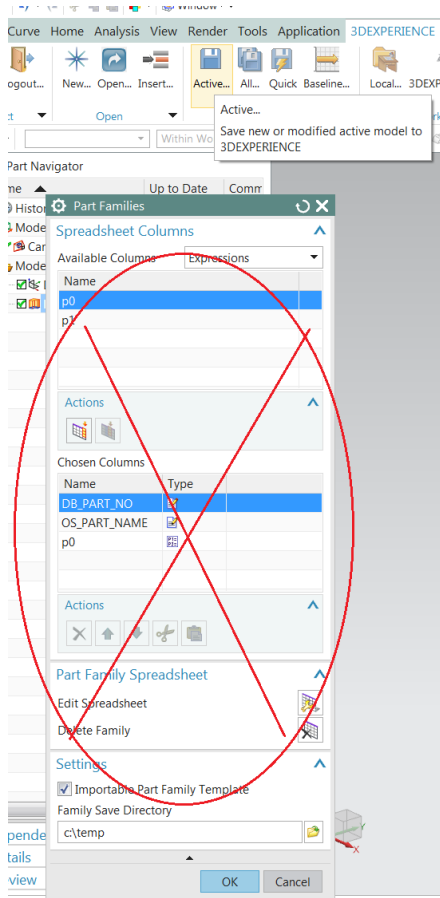
This section describes Part Family (PF) support in 3DEXPERIENCE Connector for NX and provides recommendations regarding the creation and modification of NX Part Family objects.

About NX Part Family Member File Create and Update

The 3DEXPERIENCE integration for NX will not generate missing part family member files or update PF member files that are not-up-to-date relative to the corresponding PF template. It is the NX user's responsibility to manage the creation of the PF member files on disk and update them when the PF template has been modified. Note that this is still the user responsibility even if 3DEXPERIENCE has been used to change the value of a mapped attribute in a PF template or PF member model. In that case the user must Open the template (and any modified members) with lock, update the member files on disk using NX functionalities, and then save the modified parts to 3DEXPERIENCE.



NOTE: The NX Part Families dialog should be terminated before 3DEXPERIENCE Save operation is initiated:



NOTE: For optimum performance NX Part Family templates themselves should not be instantiated as components in other assemblies



NOTE: NX part family templates and instances should be checked in separately before the instances are used as components in other assemblies.

Part Family Configuration Settings

The following configuration values determine how Part Family member files are handled during 3DEXPERIENCE Save and 3DEXPERIENCE Open. These values can be set in the settings.ini file located in the bin directory where 3DEXPERIENCE Connector for NX is installed.

1. ftiLightHandling

- 0 indicates to open all PF member models (also referred to as family table instances, or FTIs) into session when they are processed
- 1 indicates to use light handling for PF members that are not in session.

In previous releases, PF members being processed (e.g., during Save of PF templates or Open of PF templates) were always brought into session. For PF templates with many members this would require significant memory and could cause much slower performance. Enabling light PF handling will allow PF processing without bringing the PF member models into session. Information for these 'light' PF member models is obtained from the PF template's family table spreadsheet.



NOTE: Automatic generation of derived output files for PF members is not possible in light mode.

Recommended setting is: ftiLightHandling = 1

2. deleteFtiFilesOnOpen

- 0 indicates do not delete PF member files downloaded by 3DEXPERIENCE/Open
- 1 indicates delete files of family members downloaded by 3DEXPERIENCE/Open if they are only referenced from the template (and they were not already in Open directory)

Under default 3DEXPERIENCE/Open settings, when a PF template is explicitly selected for Open from 3DEXPERIENCE all of its PF member files are downloaded as well. The presence of potentially many PF member files on disk will require disk resources and will also cause 3DEXPERIENCE Save operations to have slower performance. When PF member files are on disk they must be compared to existing PF member files in the 3DEXPERIENCE db when Save is selected. If the files are not on disk it is assumed that the files in the 3DEXPERIENCE db are up-to-date.

This behavior can be avoided by setting the 3DEXPERIENCE Open Options dialog 'Select child objects' checkbox to blank when PF templates have been selected for Open. However, if it is desired to checkout an assembly PF template and get its assembly component children then the 'Select child objects' checkbox must be checked. In that case, while the user cannot prevent the download of the unnecessary PF member files, the deleteFtiFilesOnOpen option can be used to delete PF member files that were not already in the Open directory and are not participating in any

structure currently being Opened. If the user does desire to Open a PF template and all of its PF member children, then the deleteFtiFilesOnOpen should be set to 0.

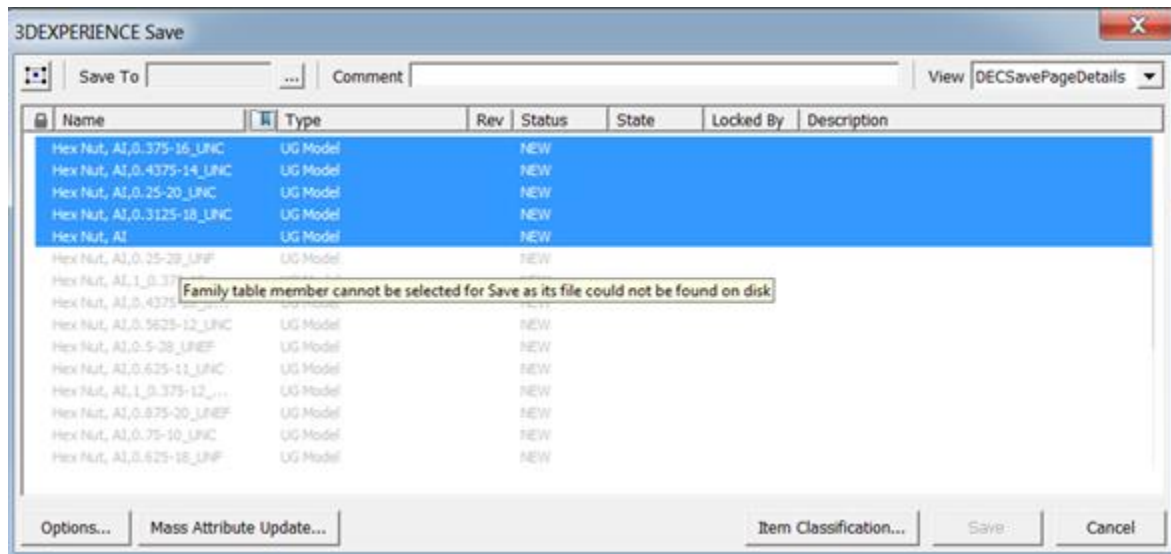
Recommended setting: deleteFtiFilesOnOpen = 1



NOTE: Adding Part Family templates to assemblies as components is not recommended. When such structures are checked out via 3DEXPERIENCE/Open, all of the templates child PF members are also downloaded under default operation.

New Part Family Creation

- Create desired part geometry in the PF template part.
- Create desired attributes. All attributes that are mapped from 3DEXPERIENCE to CAD (see GCO attribute MCADInteg-MxToCADAttribMapping) should exist in the template.
- Create Part Family itself.
 1. Use NX Part Family functions to create and update NX Part Families. The integration will not create missing Part Family member files, nor will it update out-of-date Part Family member files.
- After a Part Family and its desired member files are created on disk, use 3DEXPERIENCE Save to save the Part Family Template to 3DEXPERIENCE.
 1. Part Family template will appear as selectable and NEW in 3DEXPERIENCE Save dialog
 2. Part Family members with files on disk will appear as selectable and NEW in 3DEXPERIENCE Save dialog
 3. Part Family members without files on disk will appear as NEW in 3DEXPERIENCE Save dialog but will not be selectable. Flyover text for these objects will indicate that the family table member cannot be selected for Save (see image below).



Existing Part Family Modification

This section shows you how to make changes to Part Families that have been previously saved to 3DEXPERIENCE.

Modifying Part Family Template and Members that Exists in 3DEXPERIENCE

The steps below describe the recommended process to make edits to existing PF template and/or existing PF member objects. Examples of such changes would be modifications to template geometry and/or modifications to template attribute values or specific member attribute values.

1. Use 3DEXPERIENCE Open to lock and open the PF template and PF member objects to be modified.
2. Use NX PF functionality to modify the PF template and the PF members in the PF spreadsheet.
3. Use NX PF spreadsheet Update Parts functionality to update the desired PF member files on disk.
4. Use NX PF spreadsheet Save Family feature to save the modified PF.
5. Save the modified PF template file to disk.
6. Use 3DEXPERIENCE/Save to checkin modified objects. The modified PF template and any updated PF member objects will be pre-selected in the Save dialog. Click on Save button to commit the changes to the 3DEXPERIENCE db.

Adding a Part Family Member

The steps below describe the recommended process to add a new PF member or members to a PF template already existing in the 3DEXPERIENCE db.

1. Use 3DEXPERIENCE Open to lock and open the PF template.
2. Use NX PF functionality add the new PF members in the PF spreadsheet.
3. Use NX PF spreadsheet Create Parts feature to create the desired PF member files on disk.
4. Use NX PF spreadsheet Save Family feature to save the modified PF.
5. Save the modified PF template file to disk.
6. Use 3DEXPERIENCE/Save to checkin modified template and new PF member objects. The modified PF template and any new PF member objects will be pre-selected in the Save dialog. Click on Save button to commit the changes to the 3DEXPERIENCE db.

Removing a Part Family Member

The steps below describe the recommended process to remove an existing PF member or members from a PF template already existing in the 3DEXPERIENCE db.

1. Use 3DEXPERIENCE Open to lock and open the PF template.
2. Use NX PF functionality to remove the desired PF members from the PF spreadsheet.
3. Use NX PF spreadsheet Save Family feature to save the modified PF.
4. Save the modified PF template file to disk.
5. Use 3DEXPERIENCE/Save to checkin modified template. The modified PF template will be pre-selected in the Save dialog. Click on Save button to commit the changes to the 3DEXPERIENCE db.

Transformations

3DEXPERIENCE Connector for NX 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 NX 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.

3DEXPERIENCE Interface Description

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

[3DEXPERIENCE Menu](#)

[Right-click in 3DEXPERIENCE Open](#)

[Right-click in 3DEXPERIENCE Save Dialog box](#)

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

3DEXPERIENCE Commands

Login

See [Connecting to 3DEXPERIENCE From NX](#)

New

See [Creating Designs Using Templates](#)

Open

See [Opening Designs From 3DEXPERIENCE in NX](#)

Insert

See [Inserting Designs From 3DEXPERIENCE in an Active Design](#)

Save Active/All

See [About Saving in 3DEXPERIENCE](#)

Quick Save

See [Using Quick Save](#)

Lock

See [Locking and Unlocking Designs](#)

Baseline

See [Baselining Designs](#)

Global Refresh

See [Global Refresh](#)

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](#)

Help

See [Help](#)

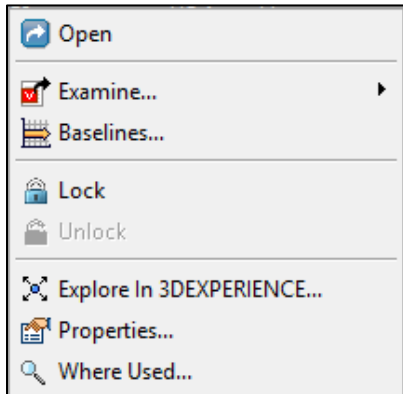
About

See [About](#)

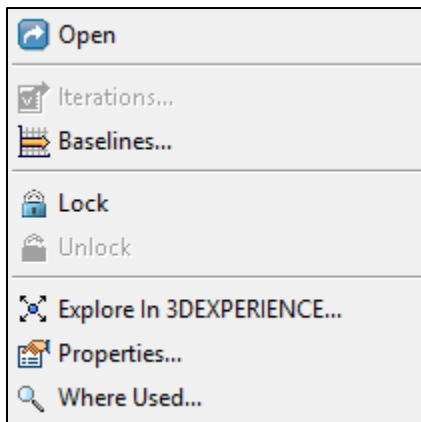
Logout

See [Logout](#)

Right-click in 3DEXPERIENCE Open Dialog box



Right-click in 3DExperience Insert Dialog



Open

See [About Opening Designs From 3DEXPERIENCE](#)

Iterations

See [Opening Particular Iteration of a Design](#)

Examine: Examine selected iteration of the model in NX

See [Examine Particular Iteration of a Design](#)

Baselines

See [Opening Baselined Designs](#)

Lock

See [Locking and Unlocking Designs](#)

Unlock

See [Locking and Unlocking Designs](#)

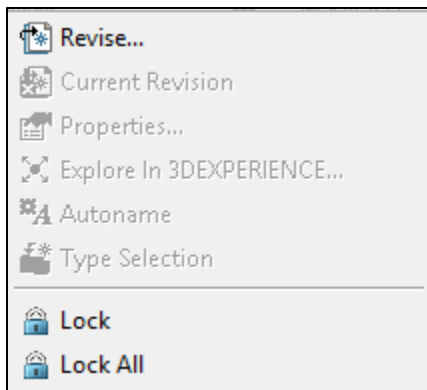
Explore in 3DEXPERIENCE

See [Viewing Design Details in X-CAD Design](#)

Properties

See [Viewing Properties](#)

Right-click 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](#)

Autaname

See [Using Autaname](#)

Type Selection

Lock

See [Locking and Unlocking Designs](#)

Lock All

See [Locking and Unlocking Designs](#)

Appendix

Configure settings.ini

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

The file is located in <3DEXPERIENCE NX 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)
- checkForNewNonSelectableChild = 0
 - Indicates whether or not to mark an object not selectable for Save if the object has any NEW children that are not selectable. Note: this behavior is always done for first level children. This setting indicates whether or not a recursive check is made for lower level children that are NEW and not selectable.
 - 0 indicates no such check should be performed (default).
 - 1 indicates parents should not be selectable in the Save dialog if it has children that are not selectable.
- copyAttribOnCheckin = 1
 - Copy attributes on checkin (0: disabled, 1: enabled)
- copyAttribOnCheckout = 1
 - Copy attributes on checkout (0: disabled, 1: enabled)
- deleteFtiFilesOnOpen = 0
 - 0: Do not delete family member files downloaded by 3DEXPERIENCE->Open
 - 1: Delete files of family members downloaded by 3DEXPERIENCE->Open if they are only referenced from the template (and they were not already in Open directory)
- ftiLightHandling = 1
 - 1 indicates to use light handling for Family Table Instances that are not in session
- fullModelLoadOnSave = 0
 - Ignore partial load option and fully load models into memory. (use 1 to fully load, and 0 to keep cad default)
- 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 dialog
- 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)
- showMessages = 1
 - Messages to show user. (0: None, 1: Warnings and Errors, 2: All)
- SSOEnabled = 0

- SSO is enabled. (0: disabled, 1: enabled)
- enforceltemClassification= 1 ; Set it to one if Library central item Classification setup is available
- 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)
- view_for_image_generation = <View orientation for JPEG generation> ;
 - Set the view for which should be used for JPEG generation
- wdf_file = <wdf file path> ;
 - wdf file to define custom widths for pdf derived output.
- default_checkmate_profile = <> ;
- checkmate_stylesheet = <path of checkmate_log.xml> ;
- queryWorkspacesAssignedToUserOnly = <>
 - Query workspaces directly assigned to logged in user. (0: disabled, 1: enabled)
- enableWorkspaceExpansionToAllLevels = 1
 - 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)
- readLargeCharacterAttributeValues = 1
 - Enables reading of CAD parameters with value more than 132 chars. (0: Read attribute values upto 132 char, 1:Read attribute values more than 132 chars)

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.

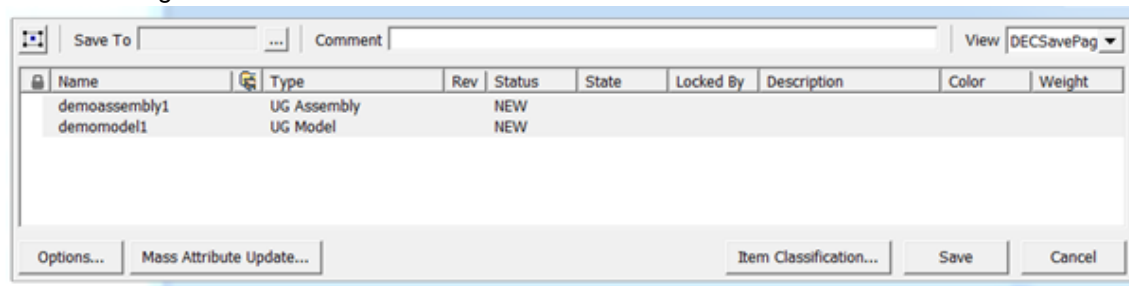
Debug Section

- debugPrints:
 - For debugging purposes only. Prints some basic logs.
- 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)

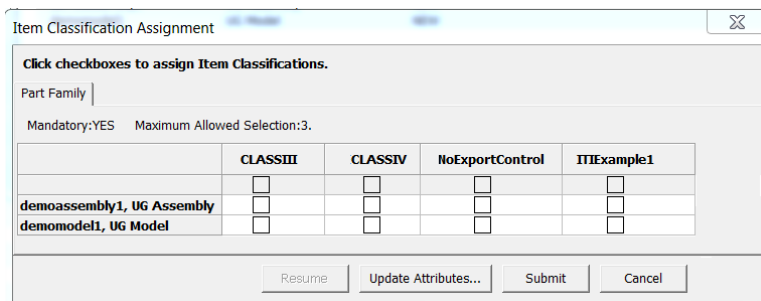
CAD Classification support

A new functionality is available starting R2022x which allows users of xCAD Design (who also have Library Central setup) 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.

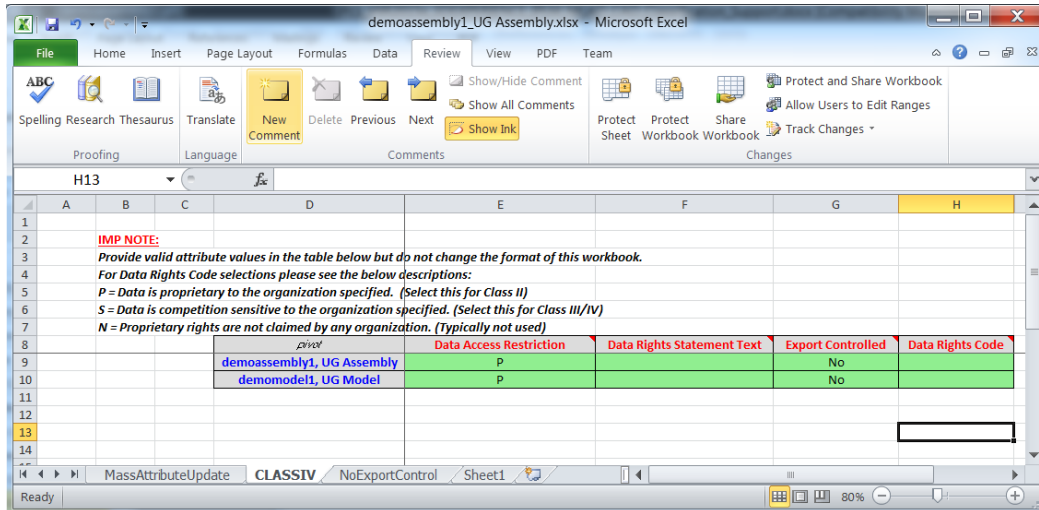
1. Designer completes the CAD design work.
2. Designer selects 3DEXPERIENCE->Save->Active or Save->All
3. Save dialog comes up with all the NEW and MODIFIED objects
4. Save dialog will have an additional button called "Item Classification"



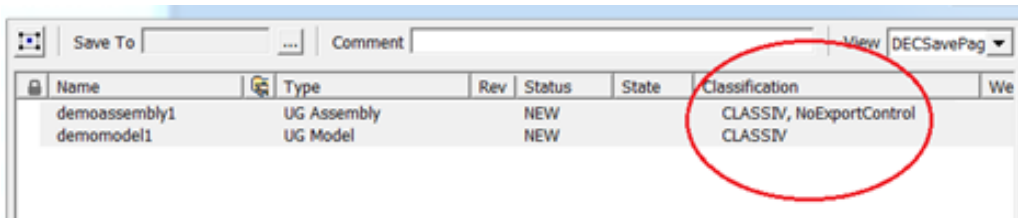
5. On clicking that 'Item Classification' button designer will be presented with following dialog



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

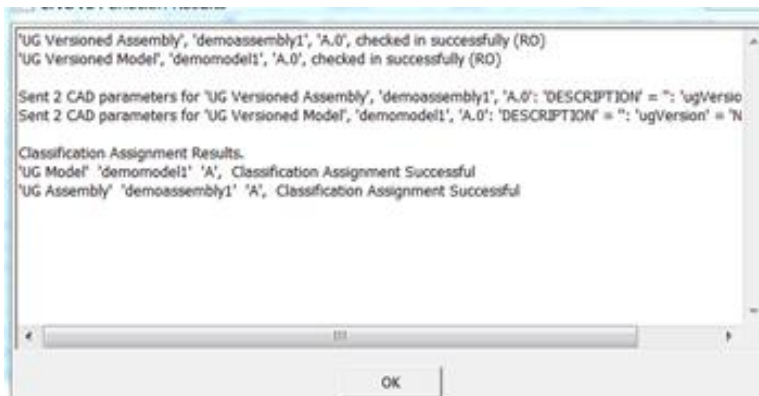


10. Once Designer is done with assigning attribute values to all the classification objects for all objects he can save and close the spreadsheet, come back to the classification dialog and hit “Resume” and “Submit” buttons. This will add a new column on Save dialog and list all the assigned classifications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



11. 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.

12. 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 MassAttributeUpdate capability along with the classification attribute updates.

Create Bookmark Workspace/Bookmark folder during 3DEXPERIENCE->Save

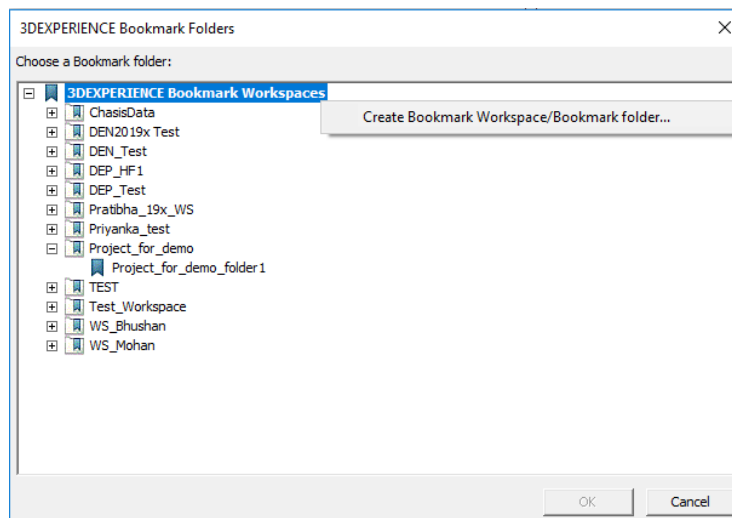
In the Save dialog, the User can access the '3DEXPERIENCE Folders' dialog for the purpose of selecting the Workspace folder in to which the designs will be saved in 3DEXPERIENCE.



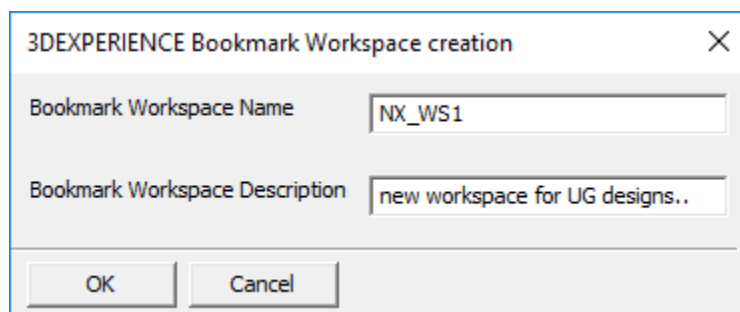
Note: The User will need to have appropriate privileges assigned for being able to create New Workspace and/or workspace folders.

When the User is performing the 3DEXPERIENCE->Save operation, the User will have an opportunity to choose the Workspace folder for saving the designs in 3DEXPERIENCE. The User might want to create a new Workspace and the following steps will provide the steps involved in creating a new Workspace.

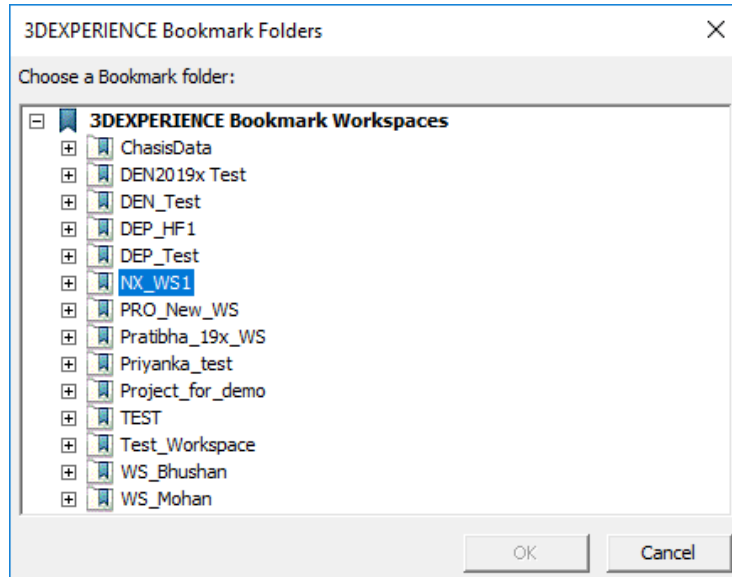
- The RMB command 'Create Bookmark Workspace/Bookmark folder...' can be used for creating a new Bookmark workspace in 3DEXPERIENCE.



- Upon selecting the RMB command, 'Create Bookmark Workspace/Bookmark folder...', The '3DEXPERIENCE Bookmark Workspace creation' dialog will be displayed.

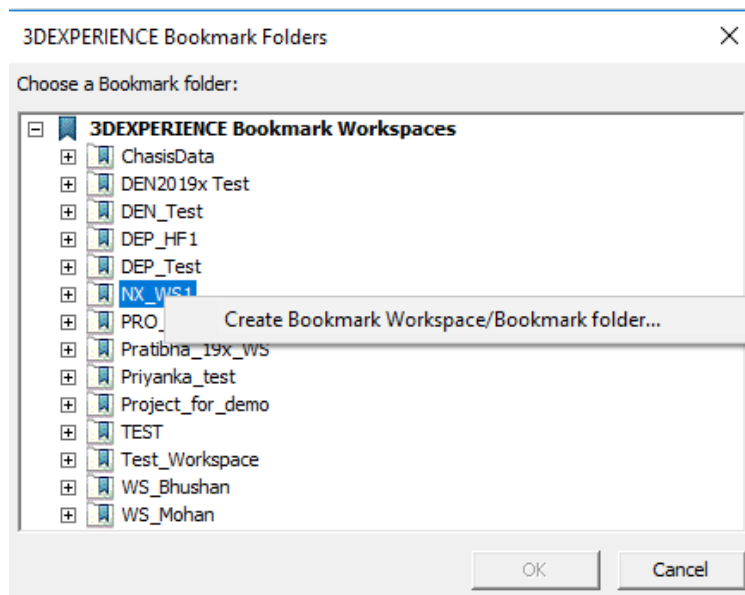


- User will have to enter the Workspace Name and Description and clicks OK to create the new workspace. The new workspace, NX_WS1 is created and selected by default.

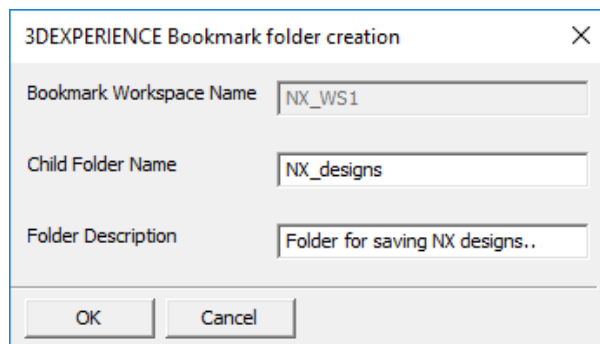


While performing the 3DEXPERIENCE->Save operation, The User will have the opportunity to create a new Bookmark Workspace Folder in any of the existing Workspaces. The following steps will provide information about creating a new Bookmark folder.

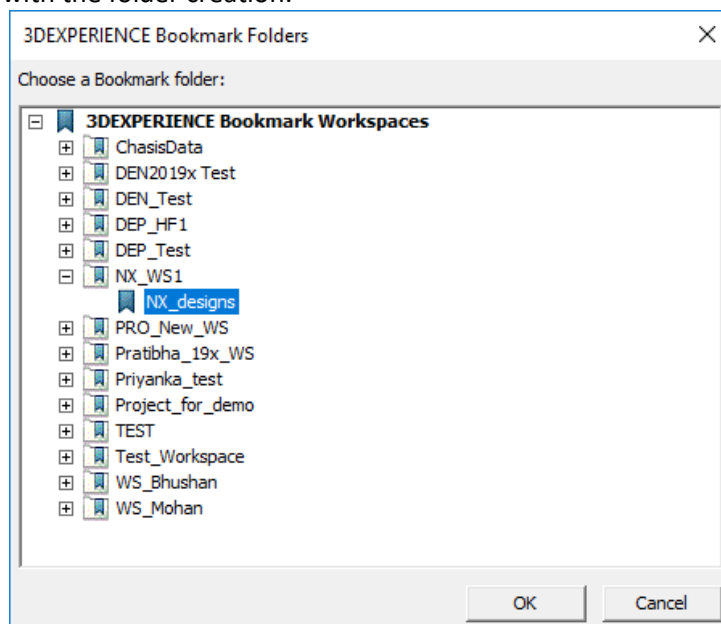
- On any existing Bookmark Workspace, the User can use the RMB command 'Create Bookmark Workspace/Bookmark folder...' to create a new Bookmark Workspace Folder.



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



User will have to enter the name of the New Bookmark Folder to be created and the description and click OK to continue with the folder creation.



- The folder ***NX_Designs*** is created successfully and is selected for use by default for the User to click OK so that this selected folder will be set as the Bookmark folder in the 'Save To' field in Save dialog.

