



TRANSLATE for 3DExperience <> NX



USER GUIDE

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Overview of Translate

About Theorem

Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of our technology & services by world leading customers.



We are proud that the vast majority of the world's leading Automotive, Aerospace, Defence, Power Generation and Transportation companies and their Supply chains use our products and services daily. Working closely with our customers, to both fully understand their requirements and feed their input into our development processes has significantly contributed to our technology and industry knowledge.

Theorem Solutions is an independent UK headquartered company incorporated in 1990, with sales and support offices in the UK and USA. Theorem has strong relationships with the major CAD and PLM vendors, including Autodesk, Dassault Systemes, ICEM Technologies (a Dassault company), PTC, SolidWorks, Spatial Technology and Siemens PLM Software. These relationships enable us to deliver best in class services and solutions to engineering companies worldwide.

Theorem's Product Suite

Theorem have 3 main Product brands. These are:



TRANSLATE

Direct translation of 3D data to or from an alternate CAD, Visualization or Standards Based format.

See our [website](#) for more detail.



PUBLISH

The creation of documents enriched with 3D content

See our [website](#) for more detail.



VISUALIZE

Visualization for [Augmented \(AR\)](#), [Mixed \(MR\)](#) and [Virtual \(VR\)](#) Reality applications

See our [website](#) for more detail.

The 3DExperience to NX Bi-directional Translator

The 3DExperience to NX translator may be installed on a number of machines each accessing a central network-floating license.

The 3DExperience to NX Translator is a bi-directional direct database converter between the Dassault Systemes 3DExperience Modelling Application and Siemens NX file format.

It enables the user to convert all forms of 3D Mechanical Design Geometry and Assembly data together with system defined attribute information and colour information, between these two systems. This product is designed for companies using 3DExperience who have selected NX to be their main method of collaboration and communication between OEMs and their customers or suppliers.

The translator can be invoked either interactively or in batch mode.

Primary Product Features

- Converts all types of geometry, wireframe, surfaces, trimmed surfaces (faces) and solid models.
- Converts assembly structure between both systems
- Converts attribute data including colour and layer information.
- Integrated with the 3DExperience installation
- The conversion process can be run interactively or in Batch mode
- Data can be filtered by layer and entity type during processing. Geometry can be filtered and selectively processed.
- Uses the 3DExperience API and Siemens NX API to read and write data.

Primary Product benefits?

- Being a direct database converter, all pre and post processing is eliminated, saving time.
- Reduce costs due to processing time and increase overall conversion success levels by filtering input data and focusing the conversion to only those elements required.
- Reduce costs and risks associated to accessing the wrong version of data by integrating the conversion process into a related business process.
- With over 20 years industrial use Theorem's product robustness and quality is well proven, reducing your business risk

This document will focus specifically on guidance for the use of the translator for the 3DExperience to NX bi-directional product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

Getting Started

Documentation & Installation Media

The latest copy of the User Guide documentation can be found on our web site at:

<http://www.theorem.com/Documentation>

Each product has a specific link that provides user documentation in the form of PDF and Tutorials.

The latest copy of Theorem software can be found via the link above and by searching for the specific product. See image below:



Latest Release: Version V24.1

- [Product Release Notes](#)
- [User Guide](#)

Setup Tutorials

- [License Server Configuration](#)

Product Tutorials

- [3DEXPERIENCE to NX Interactive Translation](#)
- [NX to 3DEXPERIENCE Interactive Translation](#)
- [NX to 3DEXPERIENCE Batch Translation](#)

Each product has a specific link to the Product Release Notes, which contains a link to the download location of the installation CD.

Installation

The installation is run from the .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation

License Configuration

To run any product a valid license file is required. The Flex License Manager is run from the .msi file download provided. This can be accessed from the Product Release Notes. For full details of the installation process, visit www.theorem.com/documentation

Using the Product

To use the product, follow the documented steps found in this document or follow the online video tutorials which can be found from www.theorem.com/documentation

Running the Product

Once configured and licensed, the product is ready to be run.

There are 3 distinct ways of running the translator:

- Interactively from within 3DExperience



- The Interactive Interface provides a direct method of importing to and exporting from 3DExperience.

- In Batch via CATUtil - DataExchangePLMBatch



- The 3DExperience DataExchangePLMBatch Interface provides a direct method of invoking the translator. It can be used on an On Demand basis to translate single or multiple files.

- On the command Line



- A command line method of invoking the translator is possible.
(3DExperience to NX only)

Translating Interactively from within 3DExperience

Launching 3DExperience with Theorem Plug-ins

The 3DExperience to NX bi-directional translator allows an opened 3DExperience part or assembly to be exported directly to NX, and for an NX part or assembly to be imported directly into the 3DExperience application.

In order to translate from within 3DExperience, the application must be started using a Theorem environment, so that the appropriate Theorem partner plug-ins are available. (See 3DExperience Environment files)

3DExperience can be started from a desktop shortcut created during installation.

Alternatively, it can be started via the script provided in the Translator installation located in:

<installation_directory>\bin

The script name is:

start_3DEXPERIENCE_Theorem_Multi-CAD_NX<xxxx>_CATIAV6R<version>.cmd

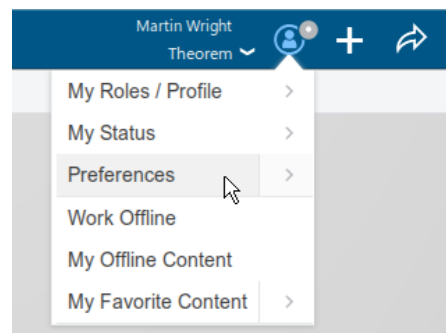
(where <xxxx> should be substituted with the version of NX that you have installed – e.g. 1926, 1953, 1980, 2007, 2206)

(where <version> should be substituted for the version of 3DExperience that you have installed – e.g. 2020x, 2021x, 2022x)

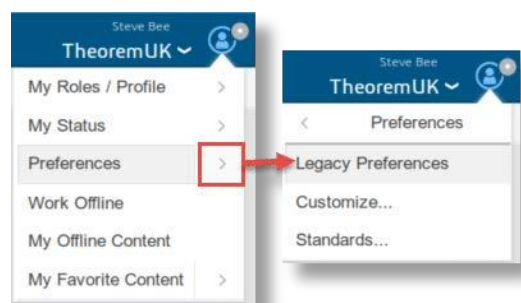
Theorem Interactive Conversion Settings

The 3DExperience interface does not currently require the user to apply any specific settings for the translation. There are some general settings that should be checked if required (e.g. for PMI conversion.) These are accessed through Preferences>Legacy Preferences>General>Compatibility>External Native 3D Formats. There are also some options that can be applied within the Theorem setting. These are accessed through Preferences>Legacy Preferences> Infrastructure>Theorem:

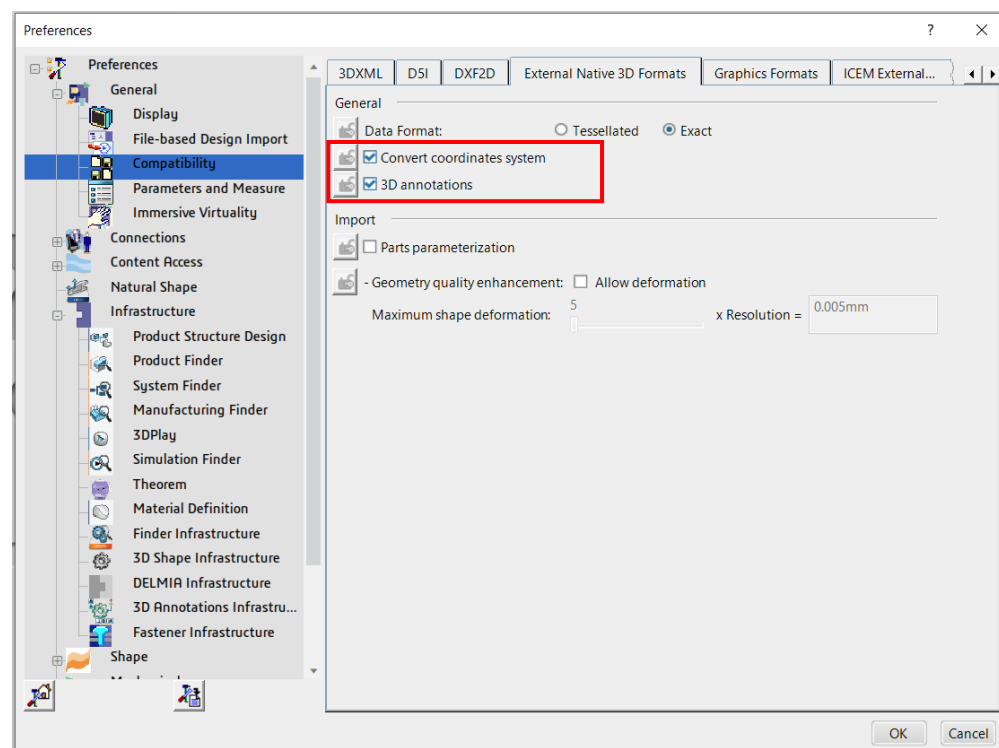
3DExperience 2018x



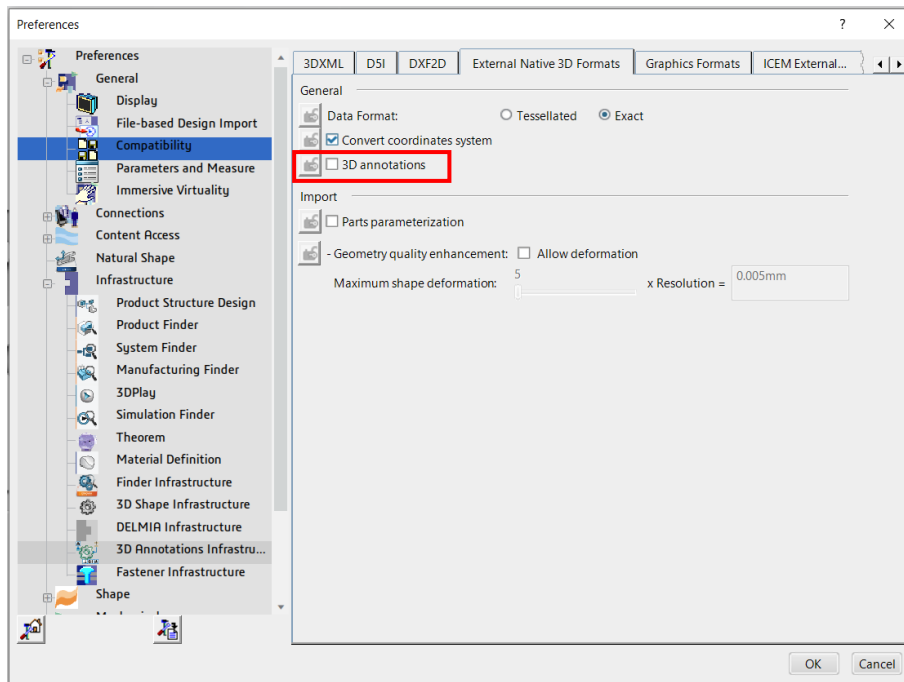
3DExperience 2019x onwards



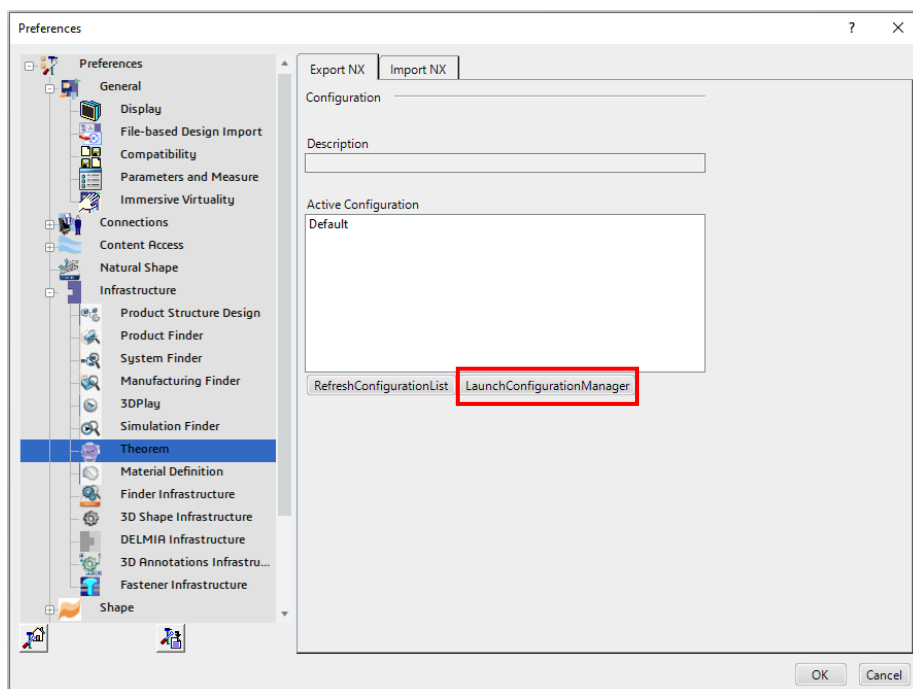
This page is a standard Dassault Page that sets the preferred mode of conversion (in this case Exact), and also the general options “Convert coordinate systems” and “3D Annotation”. Both of which are toggled on.



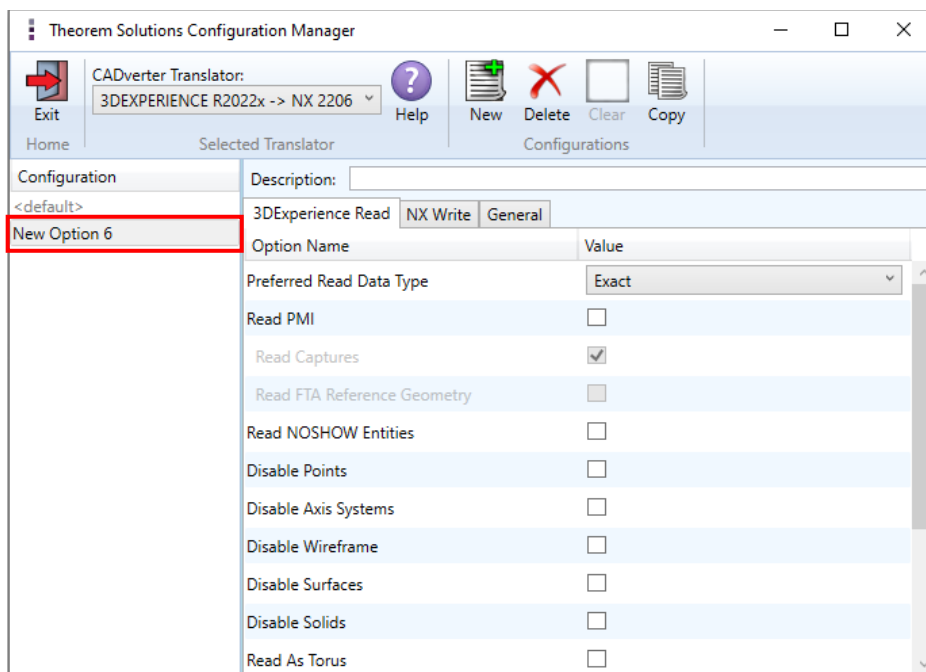
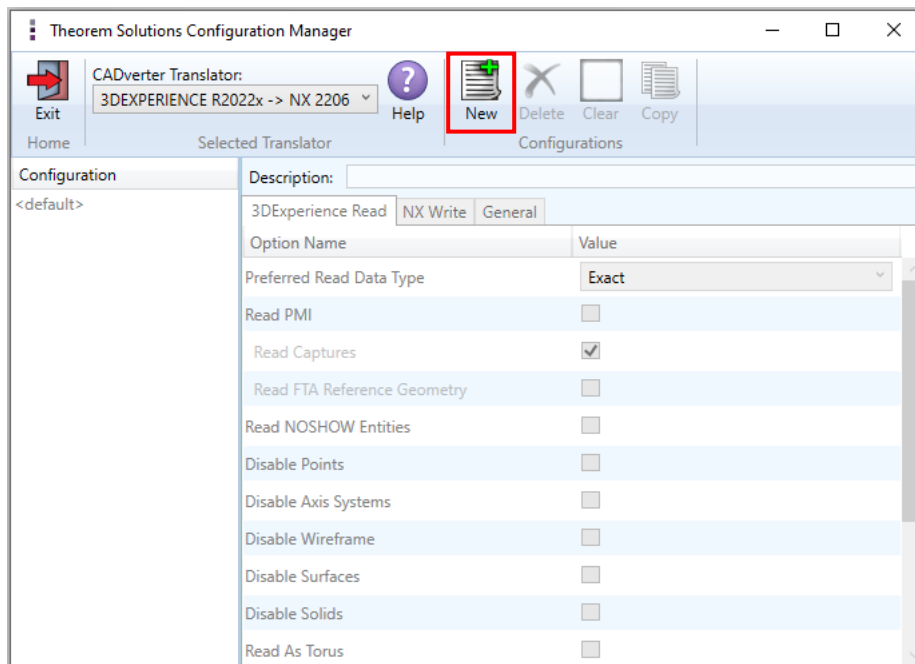
Note – Toggle off 3D Annotations to ensure that the Read PMI option is respected when using the Theorem import, i.e. PMI is displayed on the imported part when it is toggled on and is not displayed when it is toggled off.



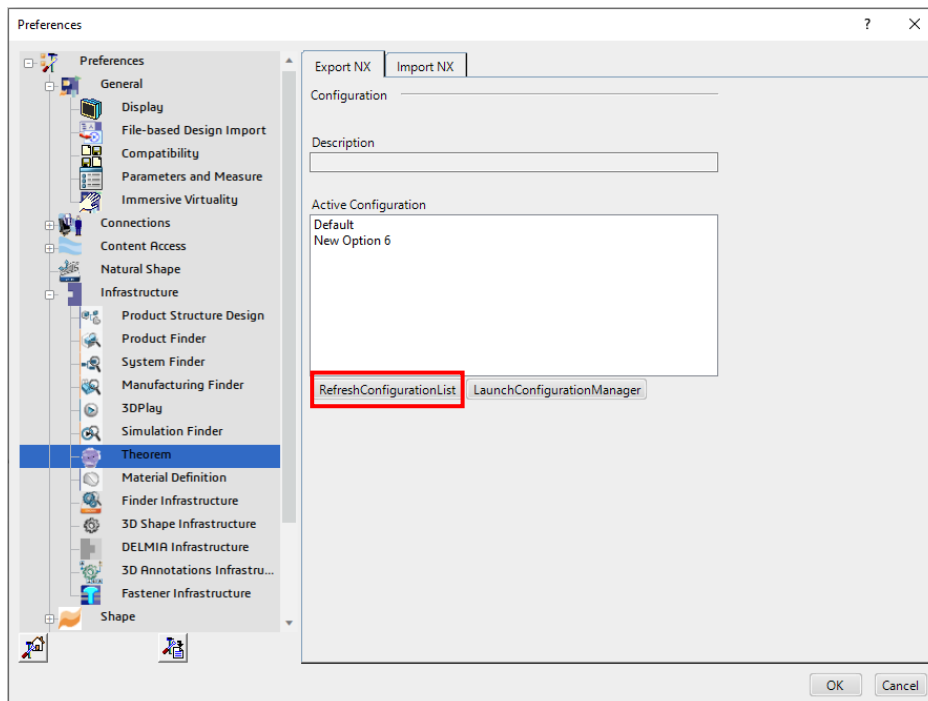
Two dedicated tabs under '**Infrastructure>Theorem**' allow the user access to Theorem Configurations for import and export. From this Panel, the user can select a predefined configuration or create a new configuration. To create a new configuration, select the '**LaunchConfigurationManager**' command.



In the configuration manager window, select New. Rename as required.



Any new configurations created will be displayed in the Active Configurations list once it has been refreshed. To do this select **'RefreshConfigurationList'**



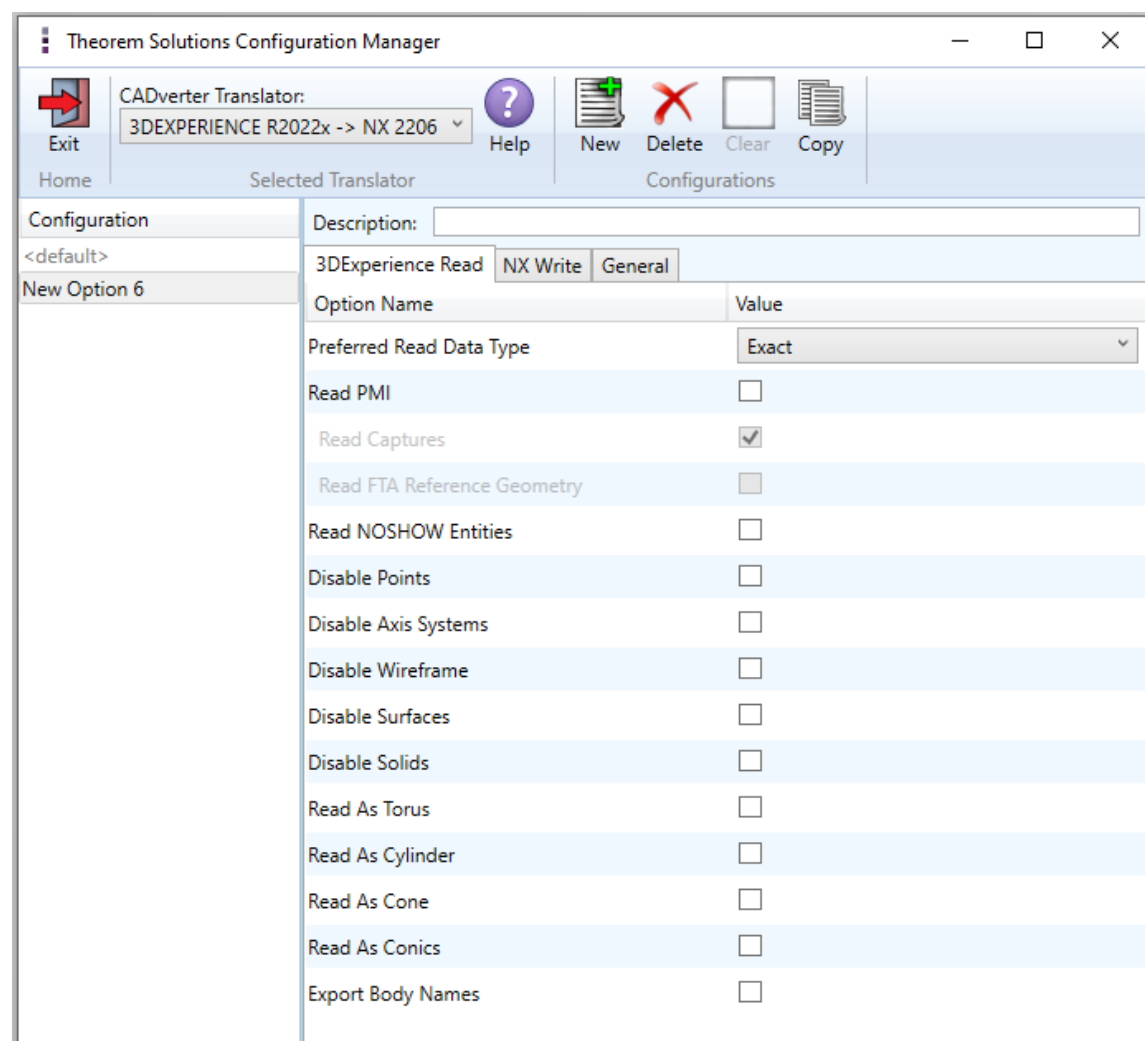
Configuration Manager

3DExperience to NX

Within the Configuration Manager, the options that are available to use are grouped into the following 3 areas when translating data from 3DExperience to NX.

- 3DExperience Read – Those options that affect how data is read from 3DExperience.
- NX Write – Those options that affect how data is written to NX.
- General – Those options that are common to ALL translation activities regardless of source data. Advanced arguments are added in this tab.

3DExperience Read



The screenshot shows the 'Theorem Solutions Configuration Manager' window. The 'CADverter Translator' is set to '3DEXPERIENCE R2022x -> NX 2206'. The '3DExperience Read' tab is selected, showing a list of options with checkboxes for their values.

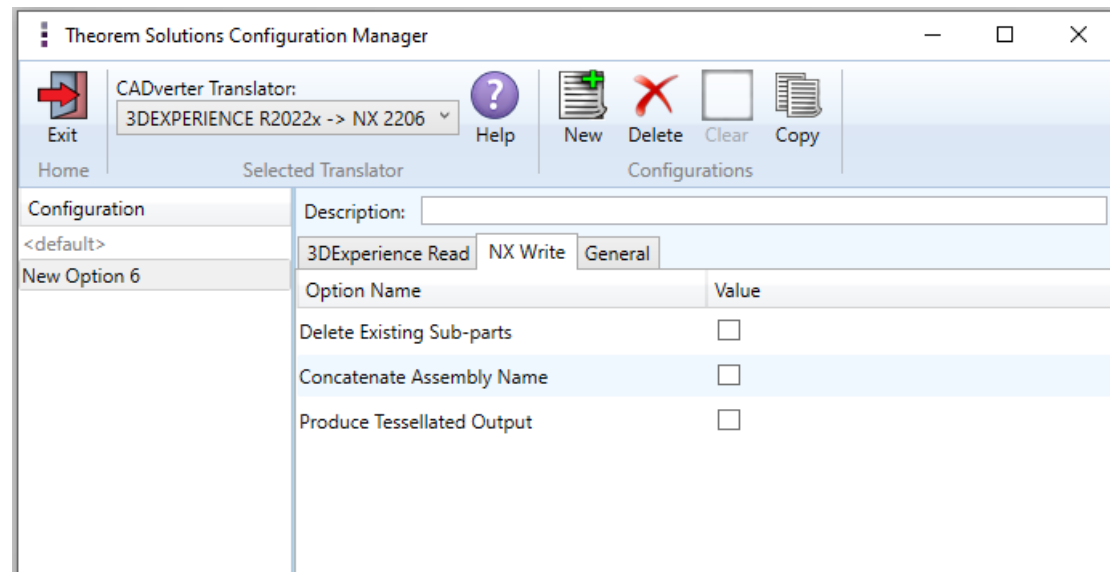
Option Name	Value
Preferred Read Data Type	Exact
Read PMI	<input type="checkbox"/>
Read Captures	<input checked="" type="checkbox"/>
Read FTA Reference Geometry	<input type="checkbox"/>
Read NOSHOW Entities	<input type="checkbox"/>
Disable Points	<input type="checkbox"/>
Disable Axis Systems	<input type="checkbox"/>
Disable Wireframe	<input type="checkbox"/>
Disable Surfaces	<input type="checkbox"/>
Disable Solids	<input type="checkbox"/>
Read As Torus	<input type="checkbox"/>
Read As Cylinder	<input type="checkbox"/>
Read As Cone	<input type="checkbox"/>
Read As Conics	<input type="checkbox"/>
Export Body Names	<input type="checkbox"/>

Each of the options displayed in the image above are described below:

Option	Description
Preferred Read Data type	<p>The setting options are EXACT (default) or TESSELATED</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> <i>read_tess</i>
Read PMI	<p>Enables PMI data read (Default is OFF).</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> <i>dont_read_pmi</i> – default <i>read_pmi</i> – to turn on <p>Note! When '<i>read_pmi</i>' is enabled it also enables the '<i>fill_pmi_arrows</i>', '<i>fill_pmi_text</i>' and '<i>pmi_filled_text</i>' options. These can be overridden by setting the Advanced arguments: '<i>dont_fill_pmi_arrows</i>' and/or '<i>dont_fill_pmi_text</i>'</p>
Read Captures	<p>A secondary argument to 'Read PMI' and allows control over whether captures are read as part of the process. Default is ON when 'Read PMI' is marked as ON. Captures can ONLY be read when 'Read PMI' is ON.</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> <i>read_captures</i> – default <i>don't_read_captures</i> – to turn off
Read FTA Reference Geometry	<p>Enables reading of FTA Reference Geometry (Default is Off). FTA Reference Geometry can ONLY be read when 'Read PMI' is ON.</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> <i>read_geometry</i> – to turn on
Read NOSHOW Entities	<p>Read any entities that are in NOSHOW. Default is to not read NOSHOW entities</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> <i>noshow</i>
Disable Points	<p>Switches off Point processing</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> <i>disable_points</i>
Disable Axis Systems	<p>Switches off Axis System processing</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> <i>disable_axes</i>

Disable Wireframe	Switches off Wireframe processing Command Line Syntax: <ul style="list-style-type: none"> <i>disable_wireframe</i>
Disable Surfaces	Switches off Surface processing Command Line Syntax: <ul style="list-style-type: none"> <i>disable_surfaces</i>
Disable Solids	Switches off Solids processing Command Line Syntax: <ul style="list-style-type: none"> <i>disable_solids</i>
Read As Torus	Read Toroidal surfaces in analytical form (default is NURBS) Command Line Syntax: <ul style="list-style-type: none"> <i>read_torus</i>
Read As Cylinder	Read Cylindrical surfaces in analytical form (default is NURBS) Command Line Syntax: <ul style="list-style-type: none"> <i>read_cylinder</i>
Read As Cone	Read Cone surfaces in analytical form (default is NURBS) Command Line Syntax: <ul style="list-style-type: none"> <i>read_cone</i>
Read As Conics	Read surfaces generated from a Conic in analytical form (default is NURBS) Command Line Syntax: <ul style="list-style-type: none"> <i>read_conics</i>
Export Body Names	Maintains body names for parts that consist of multiple bodies. Command Line Syntax: <ul style="list-style-type: none"> <i>body_names</i>

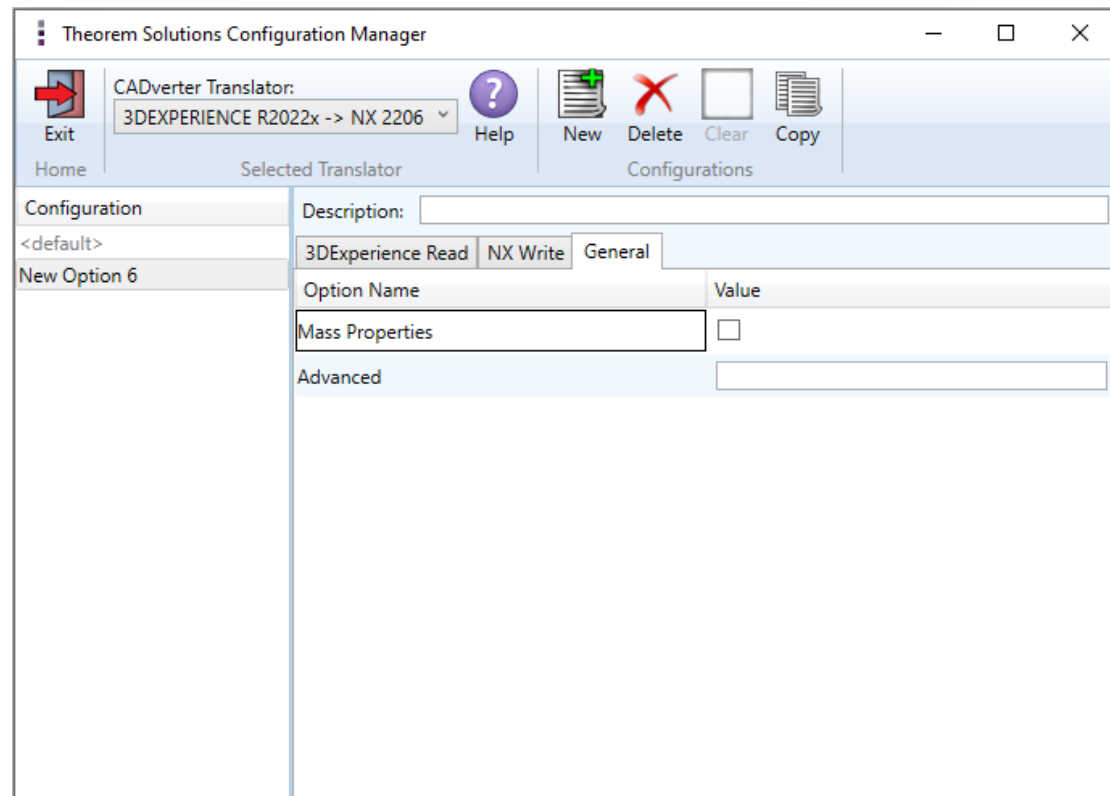
NX Write



Each of the options displayed in the image above are described below:

Option	Description
Delete Existing Sub-parts	Delete existing assembly subcomponents. (Default is Off) Command Line Syntax: <ul style="list-style-type: none"> <code>delete_parts</code>
Concatenate Assembly Name	Concatenate top level assy name to all subcomponents. (Default is Off) Command Line Syntax: <ul style="list-style-type: none"> <code>concat_assy</code>
Produce Tessellated Output	Produce Tessellated NX file. (Default is Off) Command line syntax: <ul style="list-style-type: none"> <code>tess_output</code>

General



Each of the options displayed in the image above are described below:

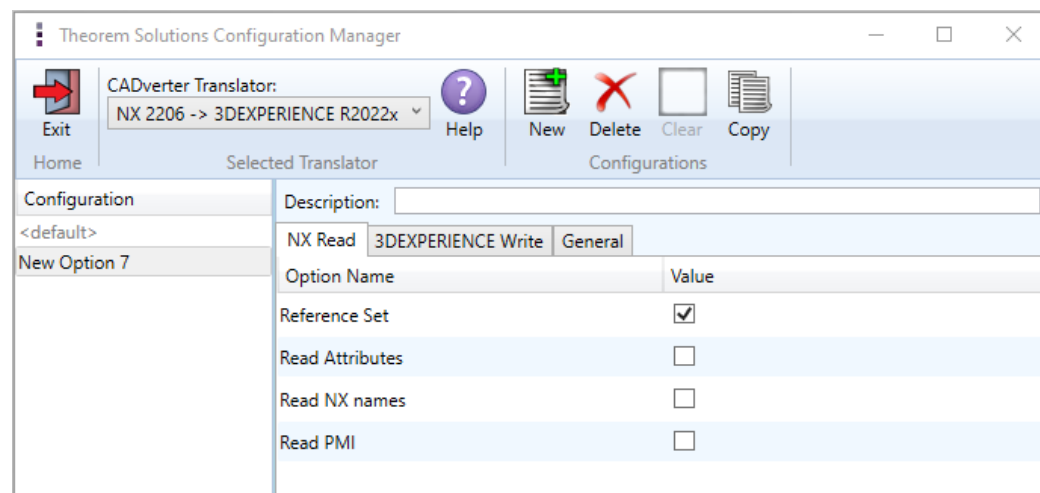
Option	Description
Mass Properties	Allows mass property information to be read Command line syntax: <ul style="list-style-type: none"> <i>mprops</i>
Advanced	Allows any advanced arguments to be added to the configuration manager and applied during the translation

NX to 3DExperience

Within the Configuration Manager, the options that are available to use are grouped into the following 3 areas when translating data from NX to 3DExperience. Some options are selectable, some require a file input.

- NX Read – Those options that affect how data is read from NX.
- 3DExperience Write – Those options that affect how data is written to 3DExperience.
- General – Those options that are common to ALL translation activities regardless of source data. Advanced arguments are added in this tab.

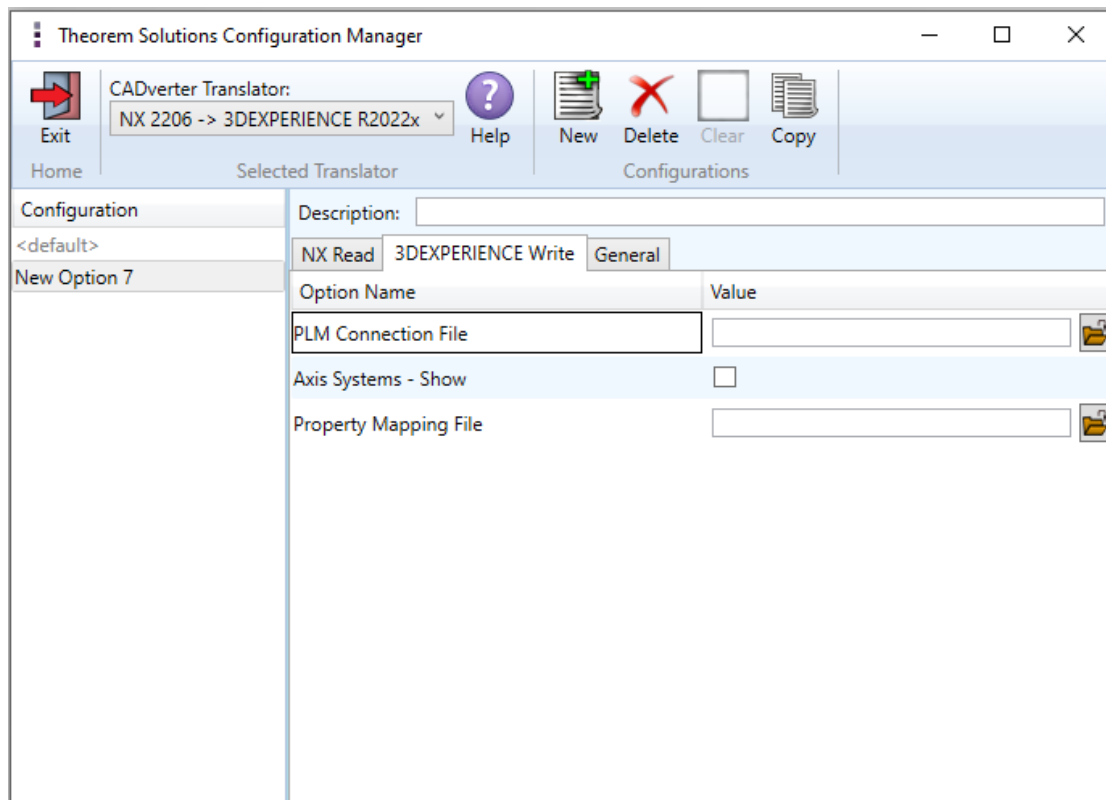
NX Read



Each of the options displayed in the image above are described below:

Option	Description
Reference Set	Enables reference set processing. (Default is On)
Read Attributes	Read NX attributes on entity/part/instance. (Default is Off)
Read NX Names	Read NX entity names (if they exist). (Default is Off)
Read PMI	Enables PMI data read. (Default is Off)

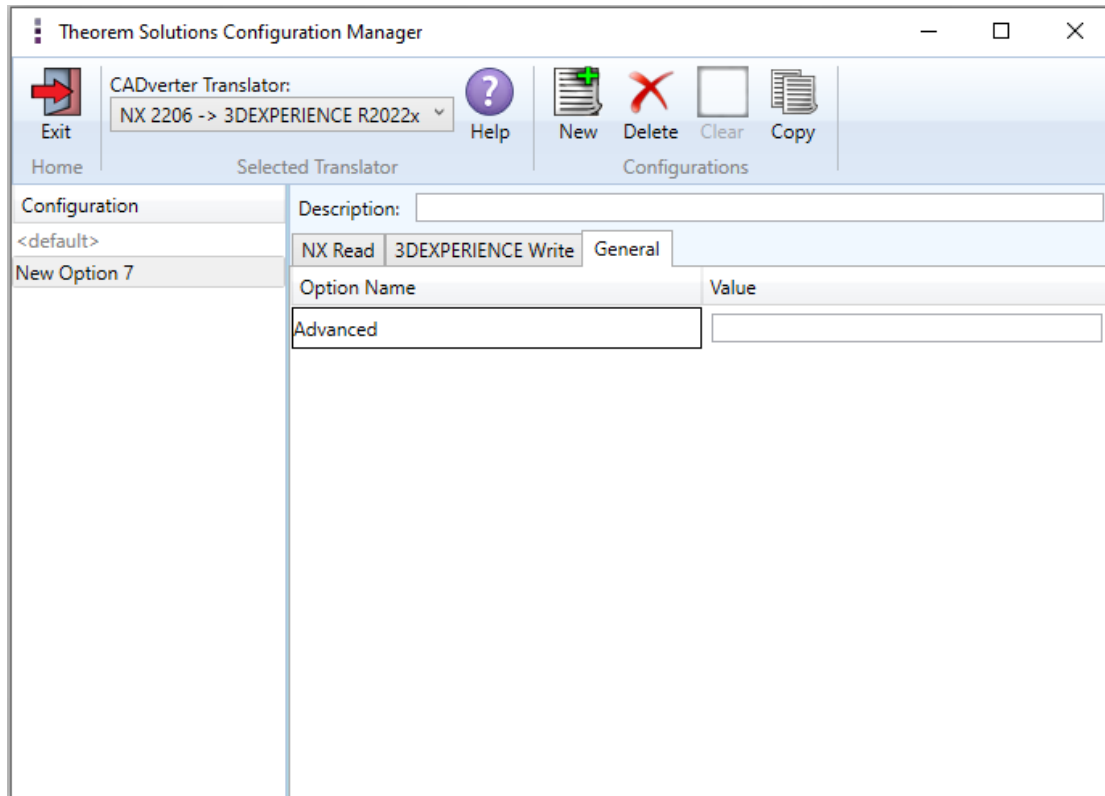
3DExperience Write



Each of the options displayed in the image above are described below:

Option	Description
PLM Connection File	The PLM Connection file is one of the xml files defining the connection parameters to the PLM database. It is required for creating a connection and logging into a PLM session programmatically.
Axis System – Show	Display all axis systems on the imported model
Property Mapping File	<p>A mapping file is used to control which properties are converted from NX to 3DExperience by setting a control value.</p> <p>Setting the control value to 0 will stop a specific property from being exported. Alternatively you can switch the property name that is found in the input data to a different name specified by the user in the output file. This is performed by switching the name between the input name = field 1 and the output name = field 2</p> <p>The File Line Format is as follows: - SourceName, TargetName, Control, Dummy, Dummy, Dummy</p> <ul style="list-style-type: none"> • SourceName – is the input attribute name. • TargetName – is the output attribute name (NULL means use SourceName) • Control – is flag to control conversion: <ul style="list-style-type: none"> ▪ 0 – Do no convert ▪ 1 – Do convert • Dummy – unused fields <p>Examples: MPARTNAME,,0,,, attribute not exported MPARTNAME,NULL,1,,, attribute exported using the SourceName MPARTNAME,Part Name,1,,, attribute exported using the TargetName specified</p>

General

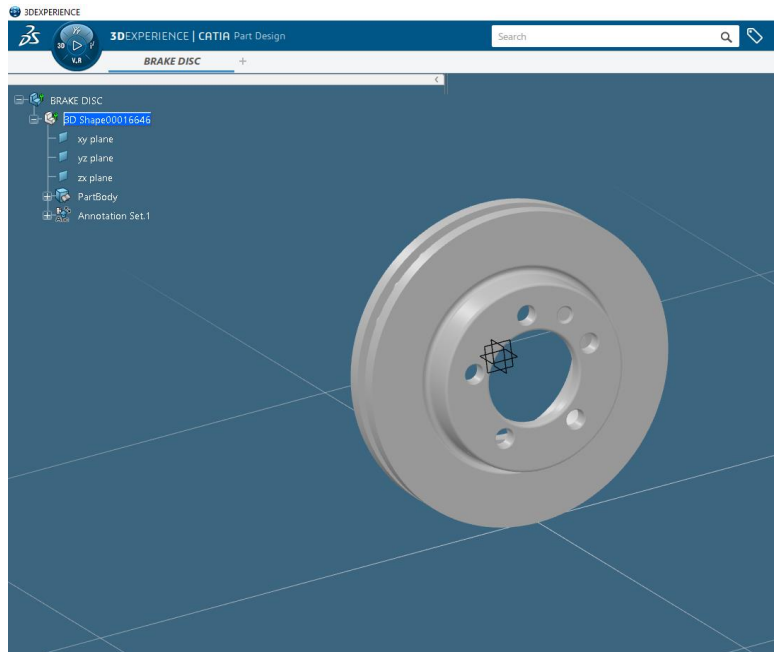


The option displayed in the image above is described below:

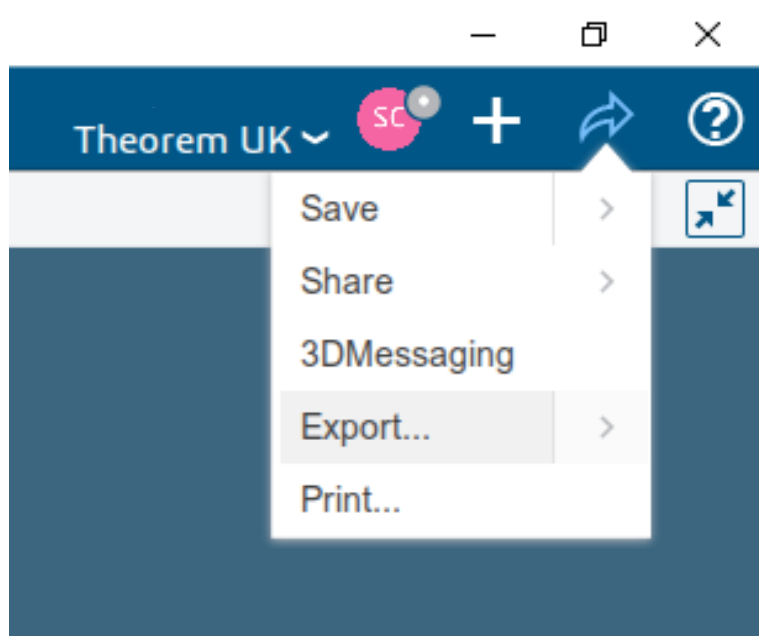
Option	Description
Advanced	Allows any advanced arguments to be added to the configuration manager and applied during the translation

Interactive Export to NX

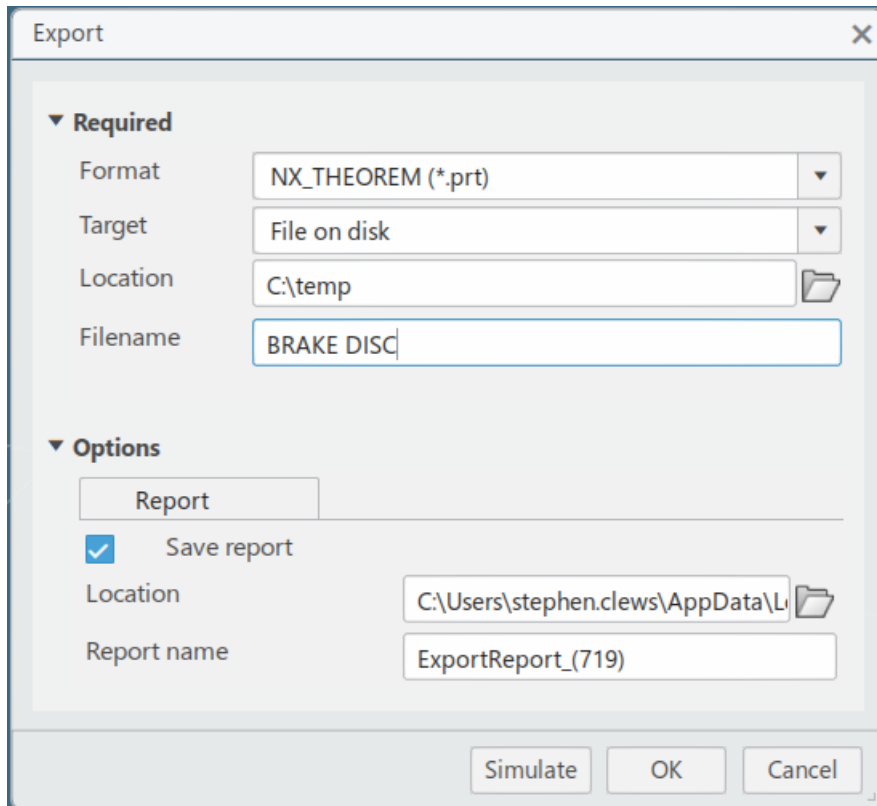
Once the 3DEXPERIENCE application has been launched, open the product or representation that is going to be exported to NX.



Select the arrow in the top right corner, then from the list displayed select Export



In the Export dialog box displayed, ensure the “Format” is set to NX_THEOREM (*.prt). Select the required location for the file and ensure the Filename displayed is correct. Click OK to initiate the translation.



The screenshot shows the 'Export' dialog box with the following settings:

- Required**
 - Format: NX_THEOREM (*.prt)
 - Target: File on disk
 - Location: C:\temp
 - Filename: BRAKE DISC
- Options**
 - Report: (button)
 - ☒ Save report
 - Location: C:\Users\stephen.clews\AppData\Local
 - Report name: ExportReport_(719)

Buttons at the bottom: Simulate, OK, Cancel.

NX Template Files

If, when a representation is read from 3DExperience, no geometry is found in the representation, all the geometry is hidden or a major write error occurs preventing a NX file being produced, then an empty NX file (named by default as template.prt), will be copied to the expected output file name.

This enables the conversion process to complete successfully and maintains the expected file outputs.

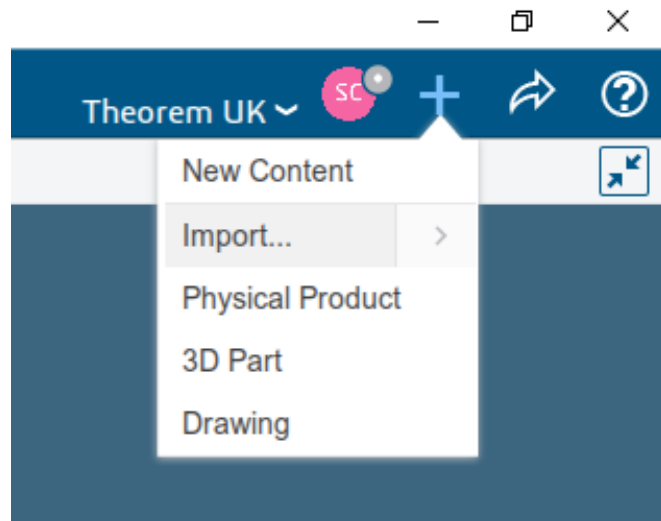
This behaviour is implemented via environment variables set in the Theorem CATEnv file e.g. %TS_INST%\B422\win_b64\CATEnv\Theorem_Multi-CAD_NX<xxxx>_CATIAV6R<version>.txt file, typically as follows:

TS_NX_TEMPLATE_FILE=<installation_directory>\data\ug\template.prt

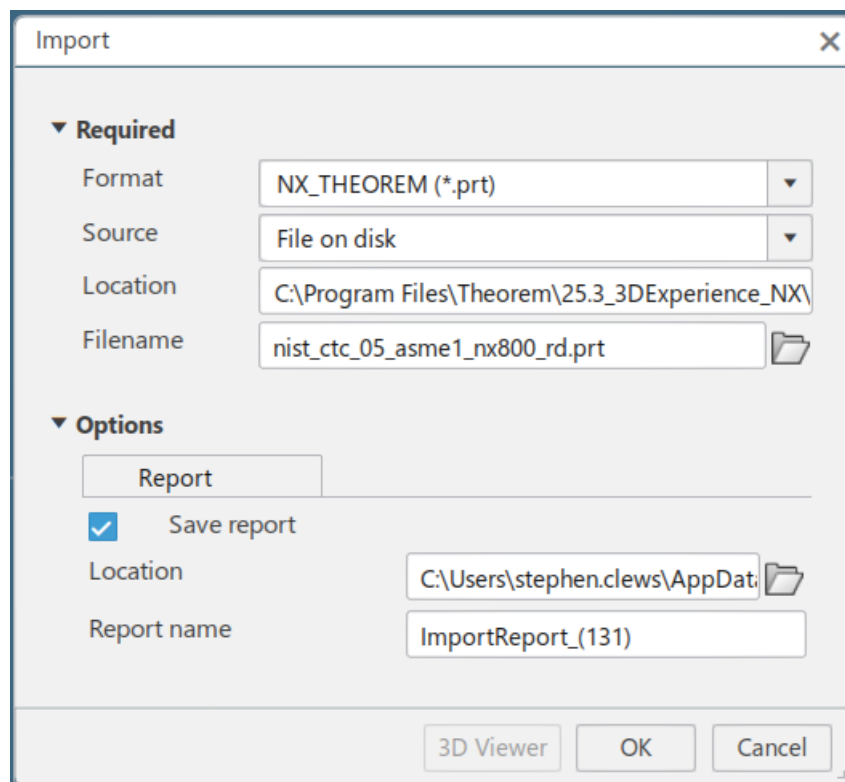
The user can change the content of these NX files, or their location and names as required.

Interactive Import from NX

Once the 3DExperience application has been launched it is possible for a model to be imported from NX. To do this select the '+' icon in the top right corner, then from the list displayed select import.



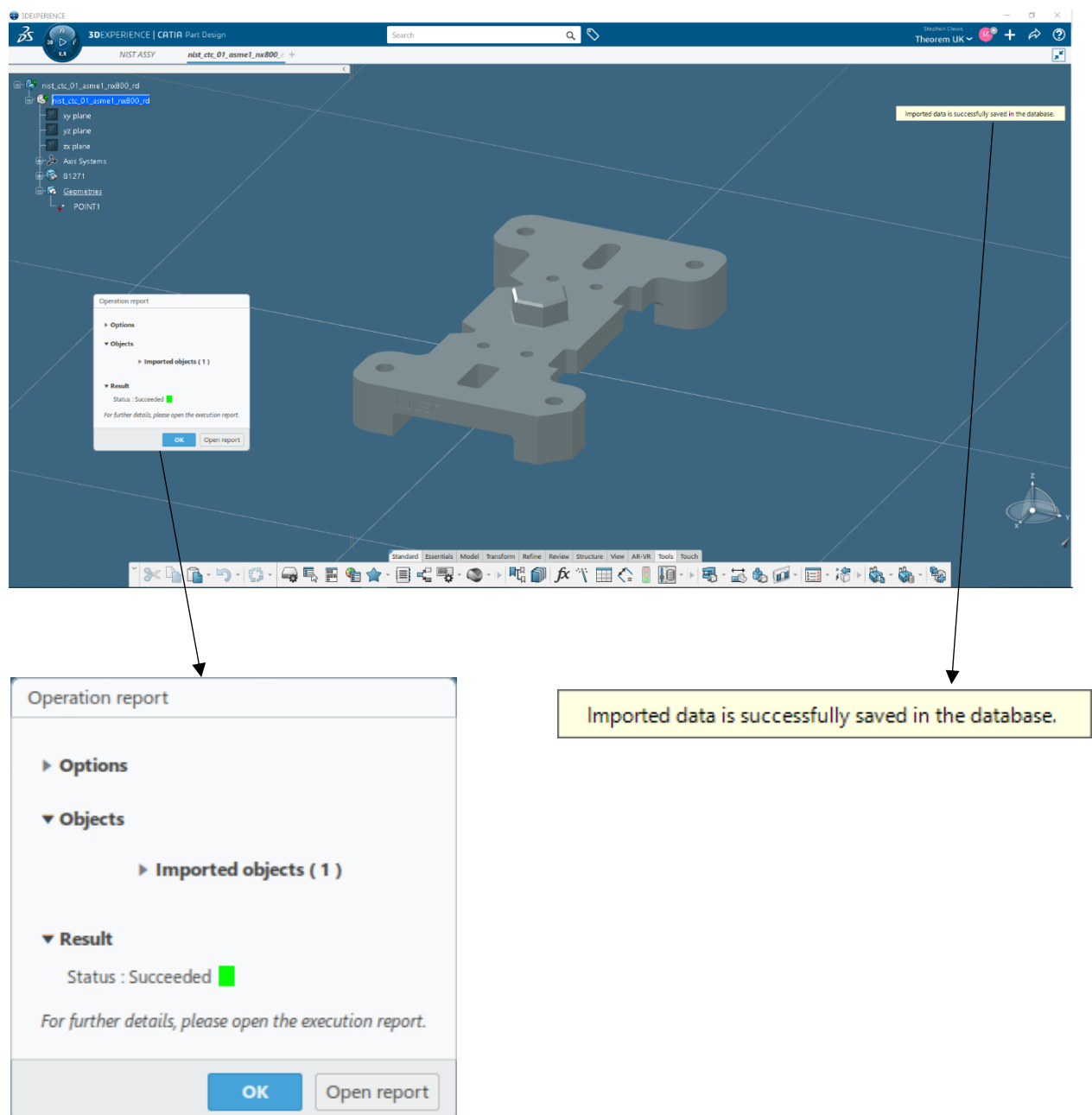
Import dialog box is displayed. Ensure the "Format" is set to NX_THEOREM (*.prt). Click on the folder icon next to the Filename field to choose the required file for import, then click OK to initiate the translation. NX data will be imported into the 3DExperience database and when complete will be opened into a new tab in the user's session.



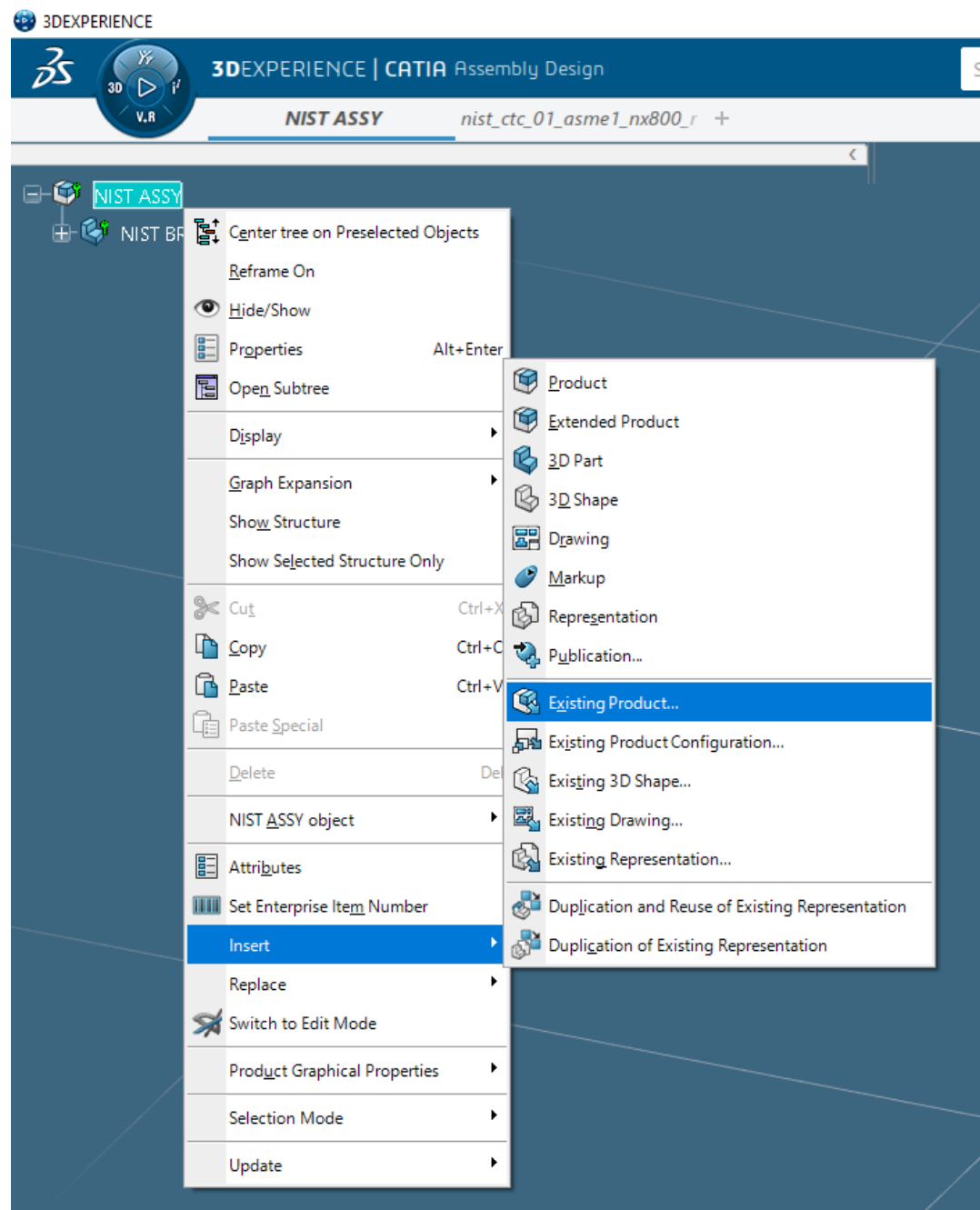
3DExperience Multi-CAD

This is an interactive solution that allows users to work with NX parts or assemblies in an active 3DExperience session. This process enables the user to import the NX data and use it for design in context activities. This eliminates the need for a separate translation process. In order to use the NX data in an active session, firstly the data will need to be imported into 3DExperience. (See Interactive Import from NX.)

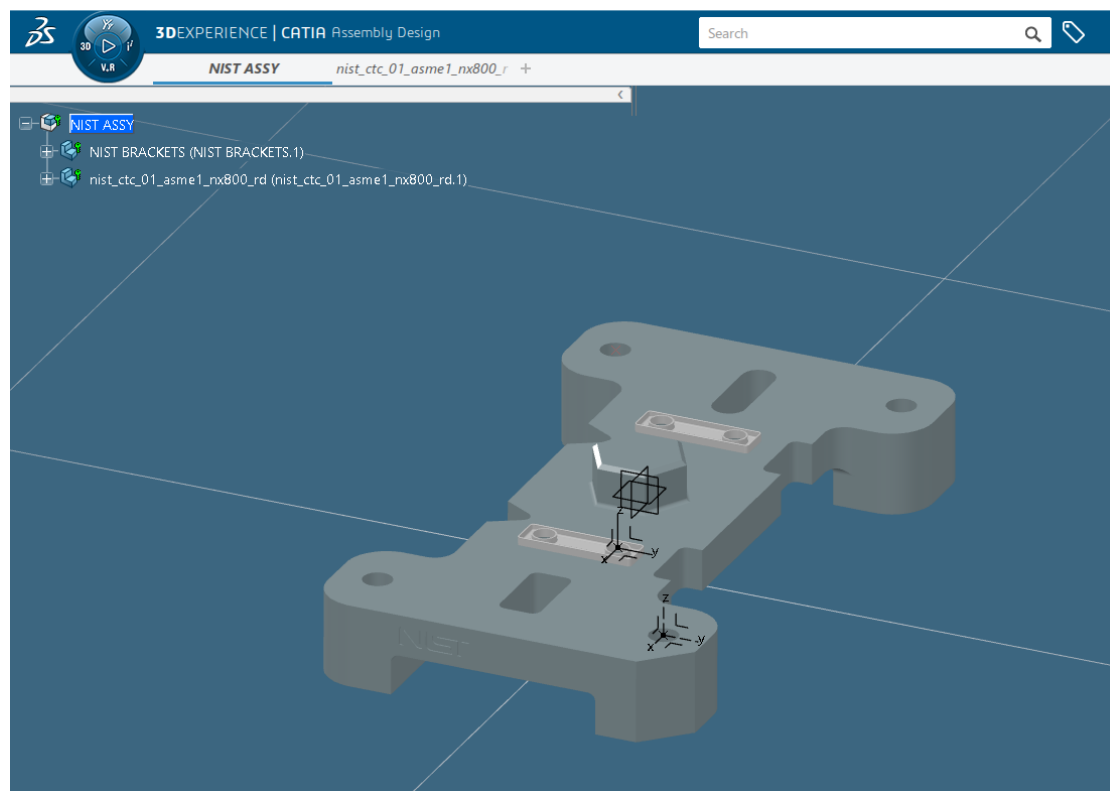
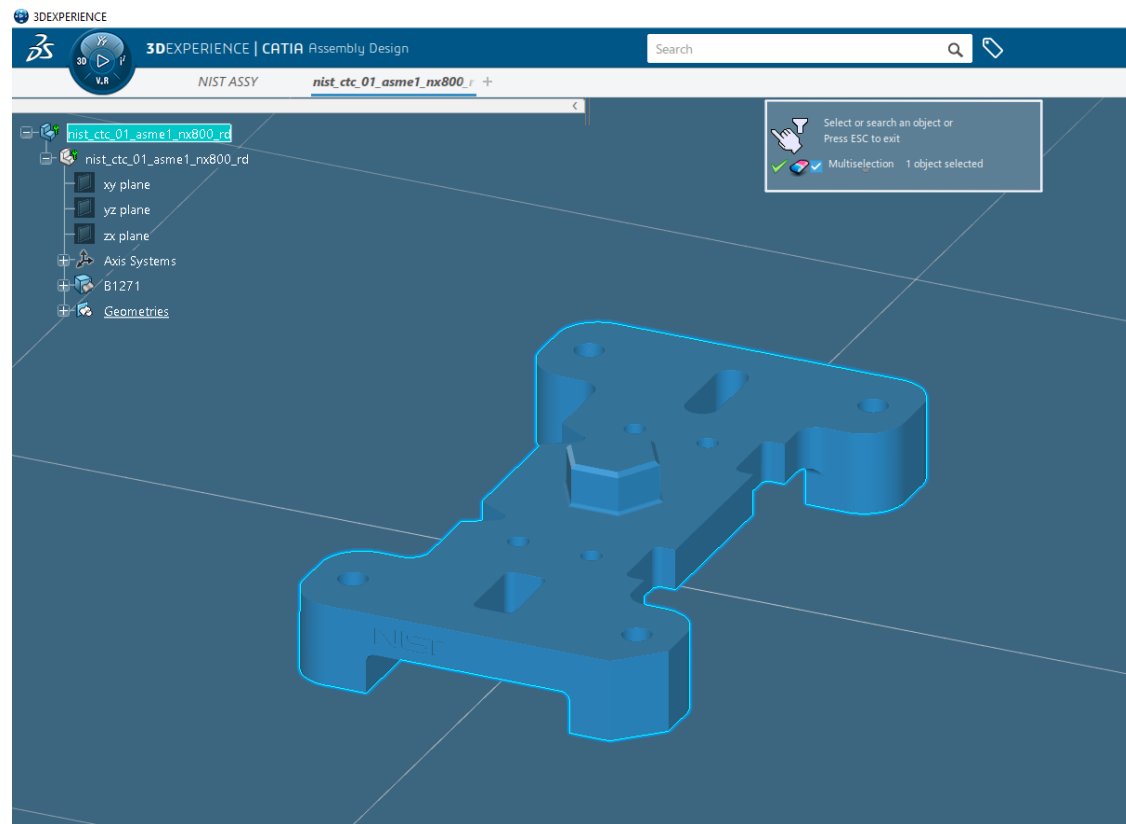
When complete an import successful message will be displayed along with an operation report, with the data opened into a new tab in the user's session.



To incorporate the imported data into an existing assembly, select the tab where the assembly is displayed, right click on the required node (typically the top node) and select Insert>Existing Product.



Select the tab where the imported data is displayed, then select the top node to add it to the assembly. If the Multiselection option is toggled on, click on the green tick.



Translating in Batch using CATUtil – DataExchangePLMBatch

Launching DataExchangePLMBatch

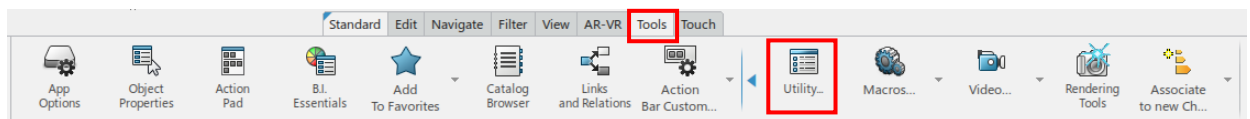
The Dassault Systemes CATUTIL Batch Monitor tool can be launched with the correct Theorem enabled environment using the scripts provided in the Translator installation at:

```
<installation_directory>\bin\start_CATUTIL_Theorem_Multi-  
CAD_NX<xxxx>_CATIAV6R<revision>.cmd
```

(where <xxxx> should be substituted with the version of NX that you have installed – e.g. 1926, 1953, 1980, 2007, 2206)

(where <revision> should be substituted for the version of 3DExperience that you have installed – e.g. 2020x, 2021x, 2022x)

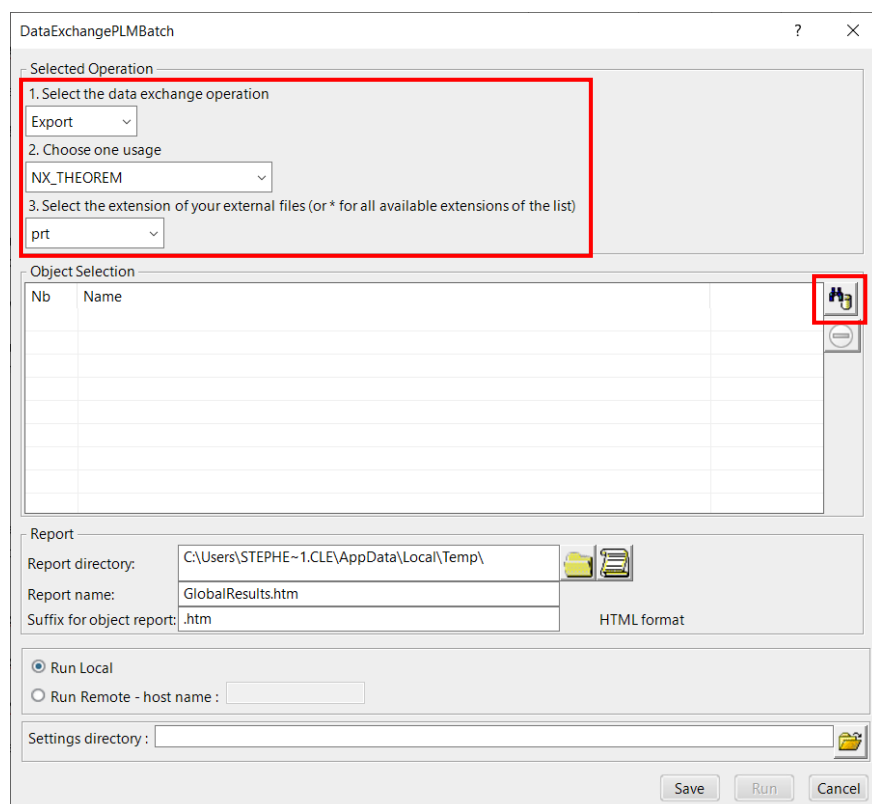
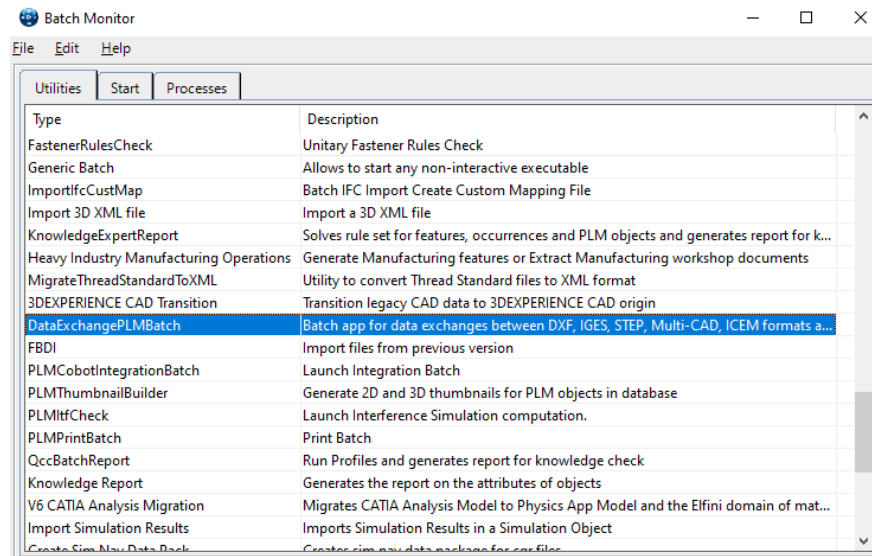
It can also be started from the Tools>Utility workbench within an Interactive 3DExperience session.




Batch Export to NX

Here is an example showing what is required in preparation for a 3DExperience database selection and export.

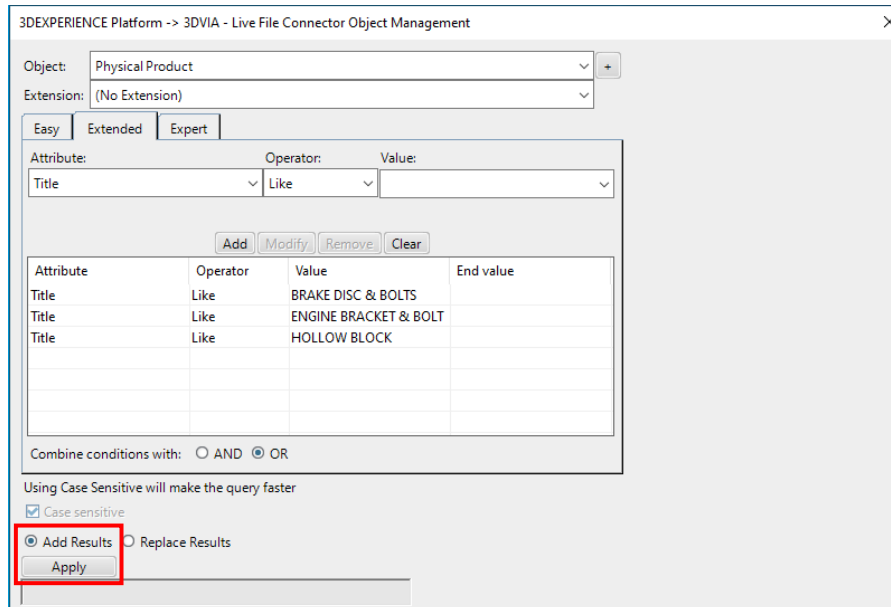
Double click on **DataExchangePLMBatch** to launch the command.



Note selection of “Export” operation, usage “NX_THEOREM” and extension “prt”.

To add the required products to the batch for export, select the 3DEXPERIENCE Platform Object Search icon. 

In the subsequent window displayed, add the products using the correct attribute, operator and value ensuring that the correct object type has been selected. (E.g. Physical Product.) For multiple products ensure that the OR condition has been selected. Select Add results and click Apply to add the list of products to the Object Selection field.



3DEXPERIENCE Platform -> 3DVIA - Live File Connector Object Management

Object: Physical Product
Extension: (No Extension)

Easy Extended Expert

Attribute: Title Operator: Like Value:

Attribute	Operator	Value	End value
Title	Like	BRAKE DISC & BOLTS	
Title	Like	ENGINE BRACKET & BOLT	
Title	Like	HOLLOW BLOCK	

Add Modify Remove Clear

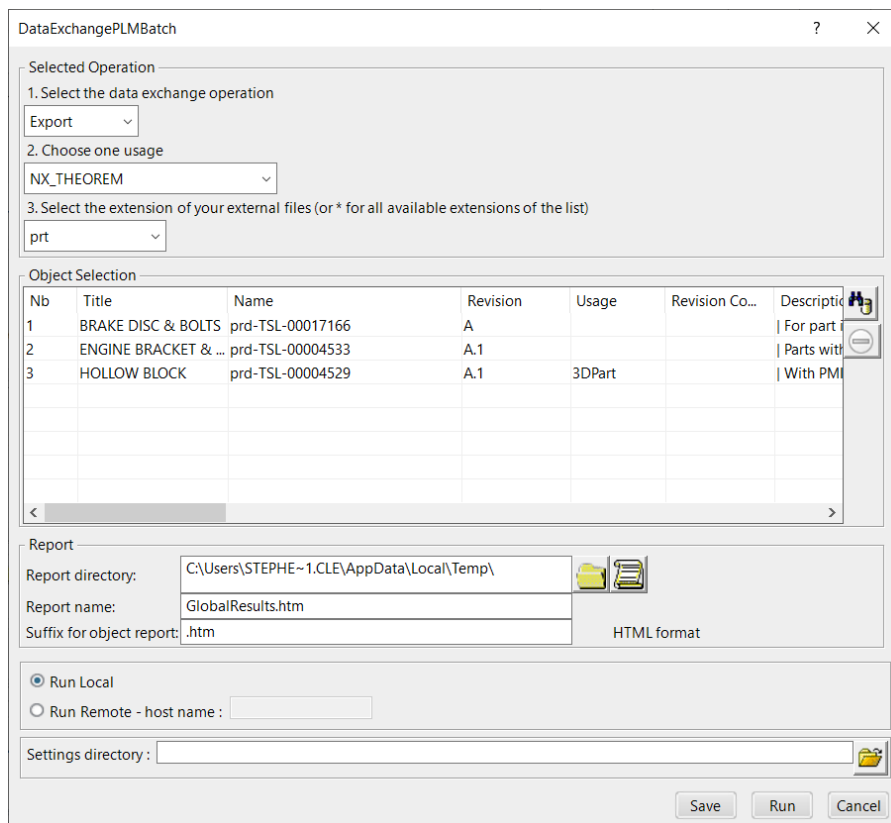
Combine conditions with: ☐ AND ☒ OR

Using Case Sensitive will make the query faster

☒ Case sensitive

☒ Add Results ☐ Replace Results

Apply



DataExchangePLMBatch

Selected Operation

1. Select the data exchange operation
Export

2. Choose one usage
NX_THEOREM

3. Select the extension of your external files (or * for all available extensions of the list)
prt

Object Selection

Nb	Title	Name	Revision	Usage	Revision Co...	Description
1	BRAKE DISC & BOLTS	prd-TSL-00017166	A			For part
2	ENGINE BRACKET & ...	prd-TSL-00004533	A.1			Parts with
3	HOLLOW BLOCK	prd-TSL-00004529	A.1	3DPart		With PM

Report

Report directory: C:\Users\STEPHE~1.CLE\AppData\Local\Temp\

Report name: GlobalResults.htm

Suffix for object report: .htm HTML format

☒ Run Local
☐ Run Remote - host name:

Settings directory:

Save Run Cancel

To change the location where both the report and NX files will be saved, select the icon next to the Report directory field, then select the required directory.



DataExchangePLMBatch

Selected Operation

1. Select the data exchange operation

Export

2. Choose one usage

NX_THEOREM

3. Select the extension of your external files (or * for all available extensions of the list)

prt

Object Selection

Nb	Title	Name	Revision	Usage	Revision Co...	Descriptio
1	BRAKE DISC & BOLTS	prd-TSL-00017166	A			For part i
2	ENGINE BRACKET & ...	prd-TSL-00004533	A.1			Parts with
3	HOLLOW BLOCK	prd-TSL-00004529	A.1	3DPart		With PM

Report

Report directory: C:\temp\Batch\

Report name: GlobalResults.htm

Suffix for object report: .htm

HTML format

☒ Run Local

☐ Run Remote - host name :

Settings directory :

Save Run Cancel

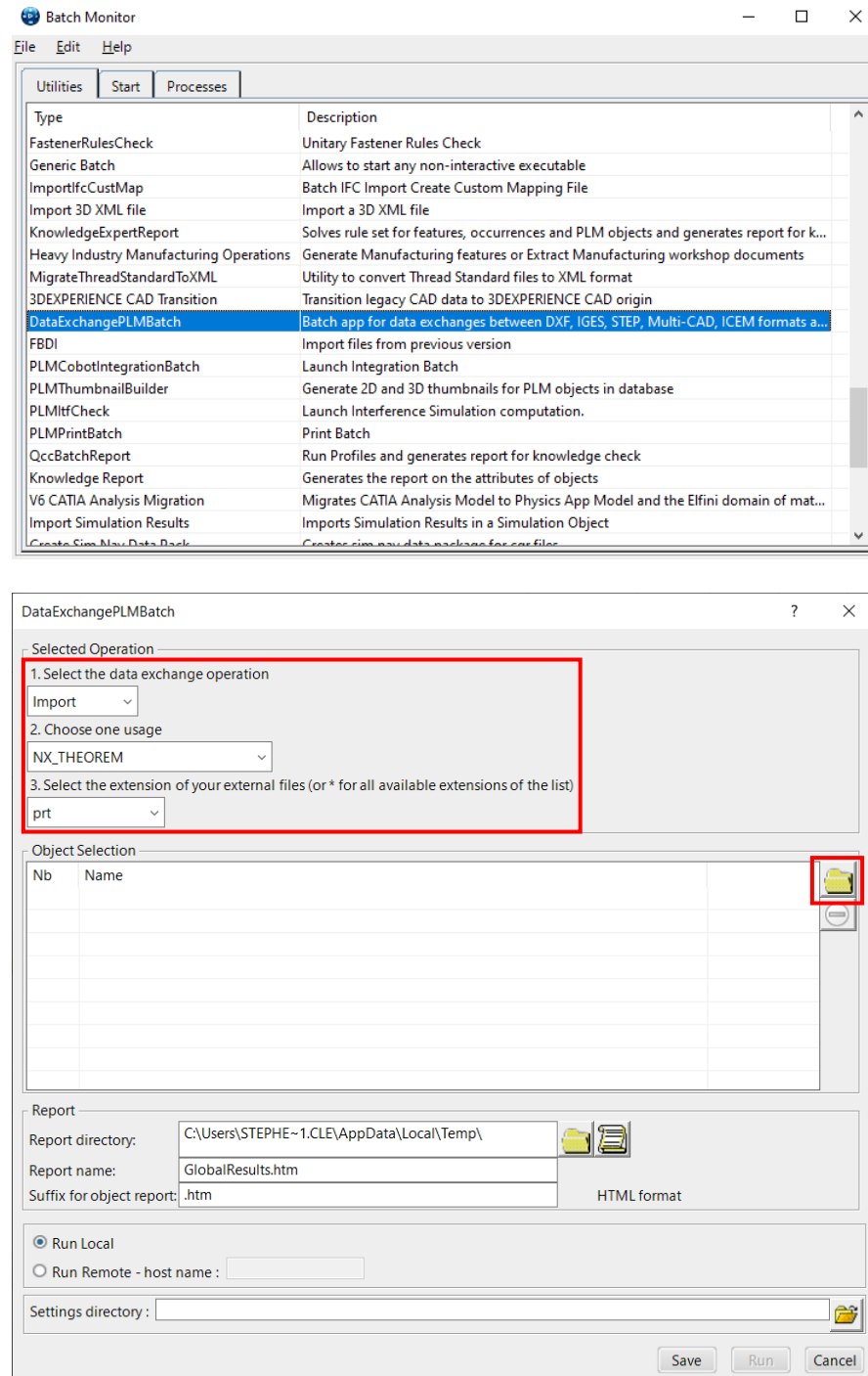
Click Run to initiate the Batch Export to NX.

Name	Date modified	Type	Size
Data	13/01/2023 15:16	File folder	
Bolt.prt	13/01/2023 15:16	ImageView Docu...	70 KB
BRAKE DISC & BOLTA.htm	13/01/2023 15:16	Chrome HTML Do...	1 KB
BRAKE_DISC.prt	13/01/2023 15:16	ImageView Docu...	466 KB
ENGINE BRACKET & BOLTA.1.htm	13/01/2023 15:16	Chrome HTML Do...	1 KB
ENGINE BRACKET BOLT.prt	13/01/2023 15:16	ImageView Docu...	73 KB
ENGINE BRACKET.prt	13/01/2023 15:16	ImageView Docu...	226 KB
GlobalResults.htm	13/01/2023 15:16	Chrome HTML Do...	2 KB
HOLLOW BLOCKA.1.htm	13/01/2023 15:16	Chrome HTML Do...	1 KB
prd-TSL-00004529.prt	13/01/2023 15:16	ImageView Docu...	85 KB
prd-TSL-00004533.prt	13/01/2023 15:16	ImageView Docu...	76 KB
prd-TSL-00017166.prt	13/01/2023 15:16	ImageView Docu...	77 KB

Batch Import from NX

Here is an example showing what is required in preparation for an NX file selection and import.

Double click on **DataExchangePLMBatch** to launch the command.

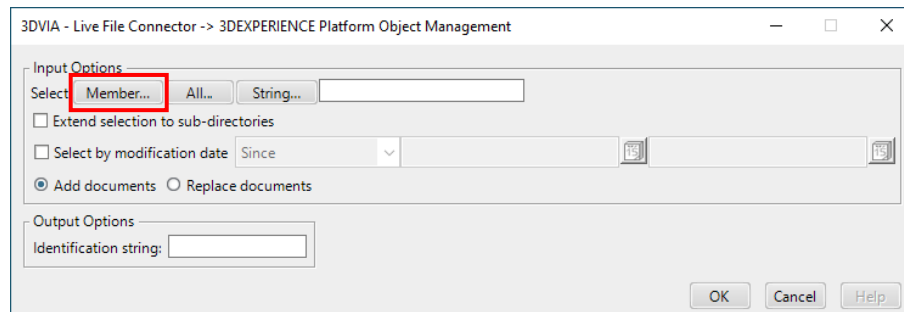


Note selection of "Import" operation, usage "NX_THEOREM" and extension "prt".

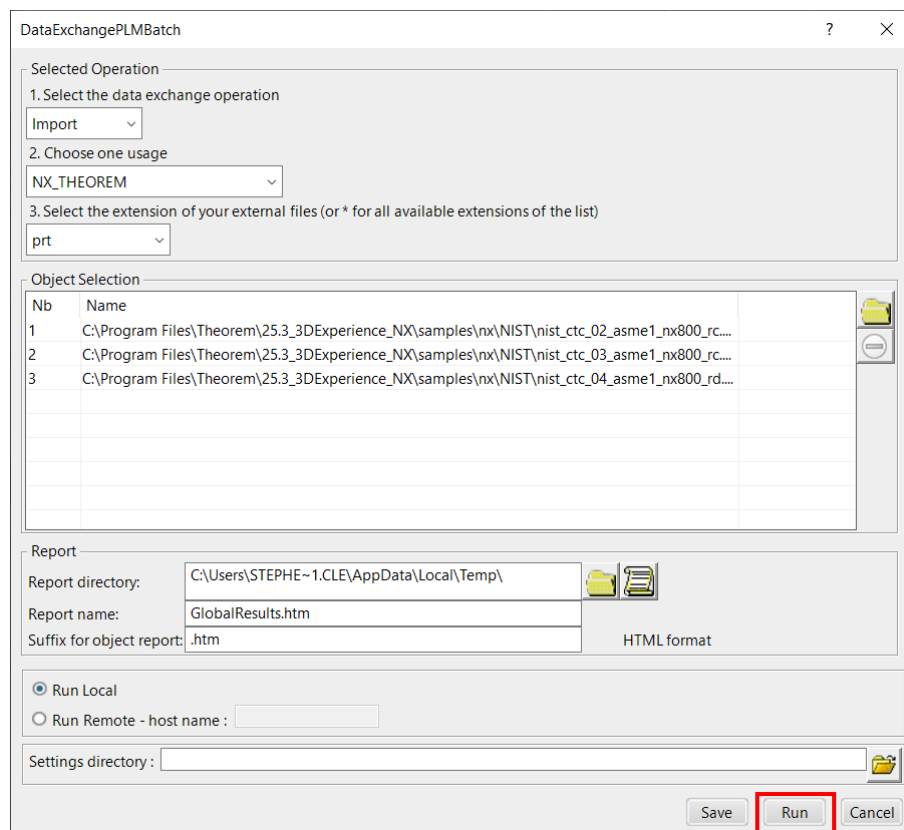
To add the required products to the batch for import, select the 3DVIA – Live File Connector Object Search icon, next to the Object Selection field.



In the subsequent window displayed, add the products by clicking on Member and selecting the required NX files. These will then be added to the Object Selection field.



The dialog box shows the 'Input Options' section with the 'Member...' button highlighted in red. Other options include 'All...', 'String...', 'Extend selection to sub-directories', 'Select by modification date', and 'Add documents' (selected) vs 'Replace documents'. The 'Output Options' section has an 'Identification string' field.



The dialog box shows the 'Selected Operation' section with 'Import' selected. The 'Object Selection' table lists three items. The 'Report' section shows the report directory, name, and suffix. The 'Run' button is highlighted in red.

Nb	Name
1	C:\Program Files\Theorem\25.3_3DExperience_NX\samples\nx\NIST\nist_ctc_02_asme1_nx800_rc...
2	C:\Program Files\Theorem\25.3_3DExperience_NX\samples\nx\NIST\nist_ctc_03_asme1_nx800_rc...
3	C:\Program Files\Theorem\25.3_3DExperience_NX\samples\nx\NIST\nist_ctc_04_asme1_nx800_rd...

Click Run to initiate the Batch Import from NX.

3 Results 🔍		
<input type="checkbox"/> Title	Type	Description
1 <input checked="" type="checkbox"/> nist_ctc_04_asme1_nx800_rd	Physical Product	
2 <input checked="" type="checkbox"/> nist_ctc_03_asme1_nx800_rc	Physical Product	
3 <input checked="" type="checkbox"/> nist_ctc_02_asme1_nx800_rc	Physical Product	

Translating on the Command Line

It is possible to run a 3DExperience to NX translation on the command line, however as all CAD data is saved in the 3DExperience database an XML file will need to be used as the input file instead of a CAD file. This is explained in more detail below. In order to run a translation on the command line the following inputs are required: -

<TS_INST>\bin\catia6r<revision>_nx<xxxx>.cmd <input_file> <output_file> <options>

Where: - <TS_INST> is the Theorem Solutions software installation directory.

Where: - <revision> is the version of 3DExperience that you have installed – e.g. 2020x, 2021x, 2022x.

Where: - <xxxx> is the version of NX that you have installed – e.g. 1926, 1953, 1980, 2007, 2206

<input_file>

Is an XML file defining access to a specific object in the 3DExperience database.

This file provides user login details (V6R2013x) or a Login Ticket (2015x onwards) (see appendix A), to a specified Enovia repository plus a set of 3 attribute names and values which will uniquely identify the input object required for the conversion.

These are highlighted in **green** in the example below and will need changing to the object and revision being used for the conversion. The lines highlighted in **yellow** should be modified with 'your' login details which are required to access Enovia.

Examples of the input file XML can be found in the

<TS_INST>\samples\3dexperience folder. These can be modified and saved on your local drive.

Here is an example of the xml input file for 3DExperience

```
<?xml version="1.0" encoding="utf-8"?>

<mc6_read>
  <!-- parameters must be in this ORDER -->
  <!-- parameters only the value passed to V6 -->
  <parameters>
    <attribute name="repository" value="PLM1"/>
    <attribute name="ServerName" value="3dspace.theorem.com"/>
    <attribute name="ServerPort" value="447"/>
    <attribute name="ServerRootURI" value="3dspace"/>
    <attribute name="LoginTicket"
value="REeYNzM3M0Q1REM2NDgxQzIFNzk1QzlwNjZGRDYzODN8Um9iaW58Um9ia
W58fHwwfA==" />
    <attribute name="PLMType" value="VPMReference"/>
    <attribute name="ReportDirectory" value="C:\TEMP\V6Export"/>
    <attribute name="LicenseData" value="LIC"/>
    <attribute name="BatchXMLFileName" value="Default"/>
  </parameters>
  <!-- arguments both key and value are passed to V6 -->
  <arguments>
    <attribute key="PLM_ExternalID" value="prd-Interfix"/>
    <attribute key="V_version" value="A"/>
    <attribute key="minorrevision" value="1"/>
  </arguments>
</mc6_read>
```

Note – In some cases, the “minorrevision” value is not required in the XML file, only the V_version value is required. This value could be A.1 for example.

<output_file>

Is the required output NX file name.

<options>

Options displayed in the configuration manager can be added at the end of the command using the correct syntax, e.g. read_pmi. (See Configuration Manager – 3DExperience to NX for list of command line syntaxes.)

Progress_file <file name>

The path and file name for the log file e.g. C:\TEMP\progress.log

```
"C:\Program Files\Theorem\25.3_3DExperience_NX\bin\catia6r2022x_nx2206.cmd"
C:\temp\3dex_2022x_input_Engine_Bracket.xml "C:\temp\ENGINE BRACKET.prt" read_pmi
```

Log File Generation

Export process Log Files

In the process of exporting the selected 3DExperience part or assembly, the following log files are generated by the NX Export plug-in.

The process log and error messages are recorded in a '.err' file located in the DassaultSystemes CATReport directory. The file is named after the active CATIA component.

e.g. %CATReport%\model1.err

Additional log files are created in the TSC_TEMP_DIR directory. This directory is defined in the %TS_INST%\ts_env.bat file.

%TSC_TEMP_DIR%\Read_to_viewer_<input_part_name>.log

Where: <input_part_name> is the name of the input part (or the active part name in interactive usage)

This contains information describing the 3DExperience 'data read' processing into Theorem Intermediate data format. Normally a list of entities.

List of gco entities :-			
Type	Total	Standalone	Subordinate
Lines	237		237
Curves	468		468
Surfaces	189		189
Planes	81		81

%TSC_TEMP_DIR%\viewer_<part-name>_screen_output.log

Where <part-name> is the selected output file name.

This contains the screen output of the process of writing the data to NX. The status of the translation can be found here.

%TSC_TEMP_DIR%\viewer_<part-name>.log

Where <part-name> is the selected output file name.

This contains detailed process information of the write of the data to NX and contains additional information such as modifiers and options used.

Import process Log Files

In the process of importing an NX file, the following log files are generated by the NX Import plug-in.

- nist_ctc_05_asme1_nx800_rd.err – provides the full processing list of errors, warnings and information
- nist_ctc_05_asme1_nx800_rd.log.rpt – provides a short list of the entities created and failed
- nist_ctc_05_asme1_nx800_rd.log.summary – provides start and finish times and the status message code (these can be customised)
- nist_ctc_05_asme1_nx800_rd.log – provides a single file with data from the other three logs

The process log and error messages are, by default, located in the DassaultSystemes CATReport directory. The files are named after the selected input file name. e.g. Mypart.prt would produce the log file names ***Mypart.err***, ***Mypart.rpt*** etc.

3DExperience Environment Files

As part of the Theorem installation process, a set of 3DExperience environment files are created which are subsequently used in the launch of 3DExperience and CATUtil sessions to support the Theorem partner plug-ins for NX file imports and exports.

A 'CATEnv' file is created for each installed version of 3DExperience.

These environment files are located in the 3DExperience revision specific folder within the Theorem install e.g.

<installation_directory>\B422\win_b64\CATEnv\Theorem_Multi-CAD_NX2206_CATIAV6R2020x.txt

<installation_directory>\B423\win_b64\CATEnv\Theorem_Multi-CAD_NX2206_CATIAV6R2021x.txt

<installation_directory>\B424\win_b64\CATEnv\Theorem_Multi-CAD_NX2206_CATIAV6R2022x.txt

These files consist of the current 3DExperience settings with the required Theorem settings appended at the bottom.

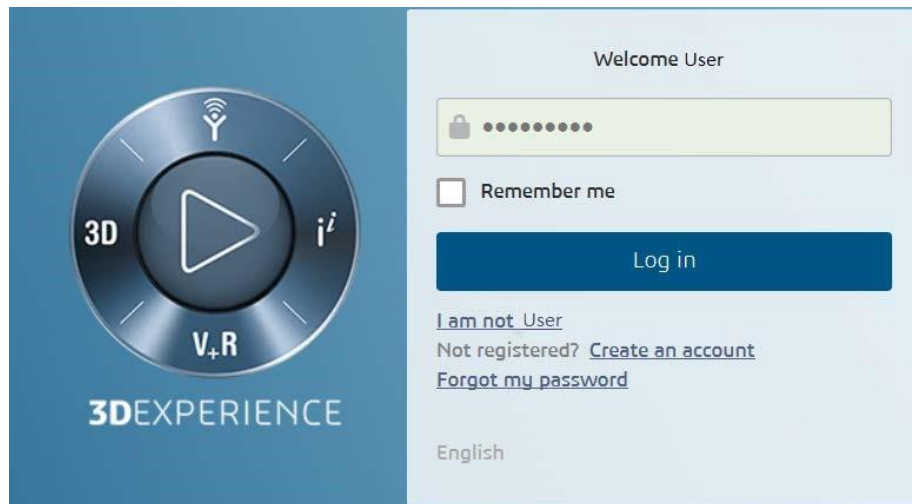
```
!-----
! Theorem Additional Multi-CAD Settings
!-----
PATH=C:\cad_apps\NX\NX 2206\;C:\cad_apps\NX\NX 2206\ugii\;C:\Program Files\Theorem\25.3_3DExperience_NX\bin;C:\Program Files
TS_INST=C:\Program Files\Theorem\25.3_3DExperience_NX\
THEOREM_LICENSE_FILE=7601@ts-tam-lic-svr
TSC_TEMP_DIR=%TEMP%
! Suppresses FTA/Geometry Links
!XCAD_FTA_NO_LINKS=1
OPTIONS_MULTICAD_PARTNER=1
UG_MULTICAD_PARTNER=1
TSC_UG_VER=2206
UGII_BASE_DIR=C:\cad_apps\NX\NX 2206\
UGII_ROOT_DIR=C:\cad_apps\NX\NX 2206\ugii\
MEPREF_LEGACYPANEL=1
```


Appendix A – Creating a Login Ticket

Use a browser link (similar to the one below) to your 3DExperience Server

<https://3dspace.2017x.theorem.com:447/3dspace/common/emxNavigator.jsp>

A page will appear in the browser

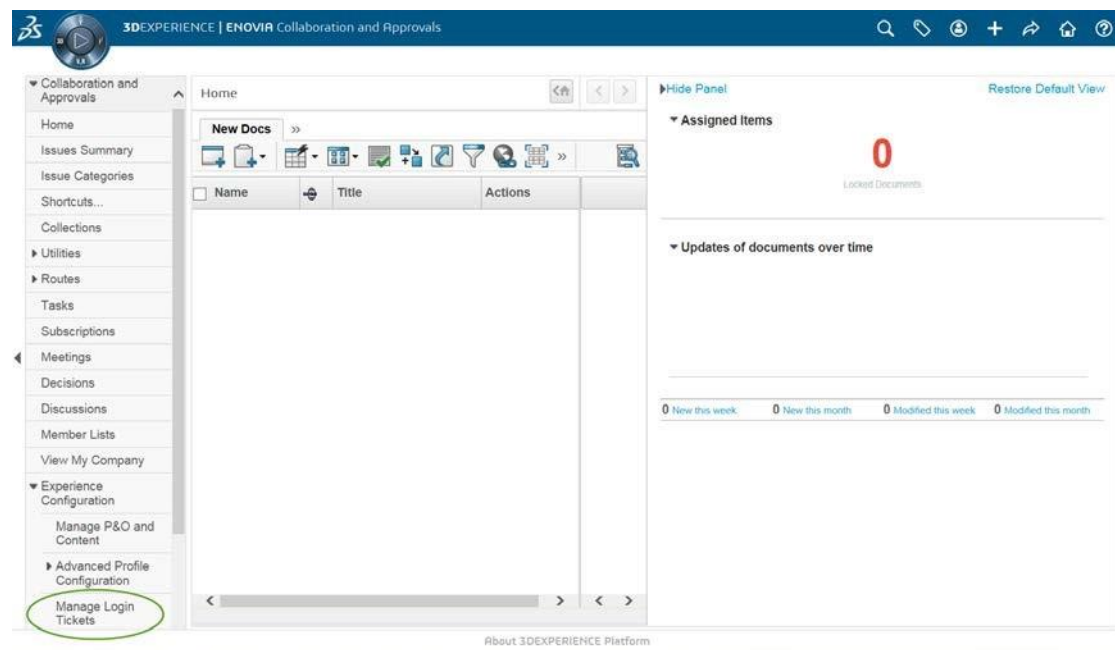



Login as normal, e.g.

User

Password

Select Collaboration and Approvals > Experience Configuration > Manage Login Tickets




3DEXPERIENCE | ENOVIA Collaboration and Approvals

Collaboration and Approvals

Home

Issues Summary

Issue Categories

Shortcuts...

Collections

Utilities

Login Ticket Creation

User:	<input type="text" value="User"/>
Security Context:	<input type="text" value="VPLMProjectLeader.Company Name.Acme"/> ▼
Ticket type:	<input checked="" type="radio"/> Infinite <input type="radio"/> Once
<input type="button" value="Create"/>	

Select the values required for your user:

User and Security Context should already be set, make sure that Ticket Type: Infinite is selected. Then click on 'Create'

"Create" will produce the ticket:-

REEyNzM3STE1MER2NDgxQzIFNzk1QzlwNjZGNATzDAN8Um9iaW58Um9iaW58fHwwfA==

This can then be used as the 'LoginTicket' value explained previously.