



USER GUIDE

NX <> 3DEXPERIENCE CATIA V6

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Overview of Multi-CAD

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 staff, but also from the vigorous industrial use of our technology & services by world leading customers.

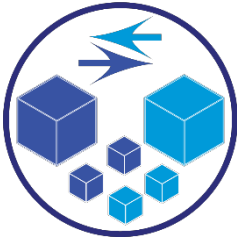


We are proud that the vast majority of the world's leading Automotive, Aerospace, Defense, Power Generation and Transportation companies and their Supply chains use our products and services daily. Working closely with our customers, to both fully understand their requirements and feed their input into 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 CATIA V6 Bi-directional NX Translator

The 3DEXPERIENCE CATIA V6 bi-directional NX translator may be installed on a number of machines each accessing a central network-floating license.

The 3DEXPERIENCE CATIA V6 bi-directional NX Translator is a bi-directional direct database converter between the Dassault Systemes 3DEXPERIENCE CATIA V6 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 as a method of collaboration and communication between OEMs and their customers or suppliers.

The translator can be invoked in either an interactive or batch mode.

Primary Product Features

- Converts all types of geometry, wire frame, 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 CATIA V6 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 CATIA V6 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 processes.
- With over 20 years of industrial use Theorem translation products robustness and quality is well proven, reducing your business risk.

This document will focus specifically on guidance for the use of Multi-CAD for the 3DEXPERIENCE CATIA V6 bi-directional NX product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

Getting Started

Documentation

The latest copy of this documentation can be found on our web site at:

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

Installation Media

The latest copy of Theorem software can be found via our web site at:

<http://www.theorem.com/Product-Release-Notes>

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

Alternatively, you can request a copy of the software to be shipped on a physical CD.

Installation

The installation is run from the CD or ZIP file download provided.



To install the translator, select the **Setup.exe** file and follow the installation process. For a full guide to the process, please see our 'Translator Installation Process' demonstration video located [here](#).

License Configuration



In order for the translation to run successfully, the Theorem license file provided to you needs to be configured using FlexLM. For a full guide to this process, please see our 'FlexLM License Set Up and Configuration' demonstration video located [here](#).

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 CATIA V6
 - The Interactive Interface provides a direct method of importing and exporting to and 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
- Translating from the UI (User Interface)

Using the Product

Translating Interactively from within 3DEXPERIENCE

The 3DEXPERIENCE CATIA V6 to NX translator allows an opened 3DEXPERIENCE CATIA V6 part or Assembly to be exported directly to NX, and for a NX part or assembly to be imported directly into the 3DEXPERIENCE CATIA V6 application.

In order to translate from within 3DEXPERIENCE CATIA V6, the 3DEXPERIENCE CATIA V6 application must be started using a Theorem environment, so that the appropriate Theorem partner plug-ins are available. (See [3DEXPERIENCE CATIA V6 Environment files](#))

3DEXPERIENCE CATIA V6 can be started from a desktop shortcut created at installation time.



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_NXxx_CATIAV6<version>.cmd

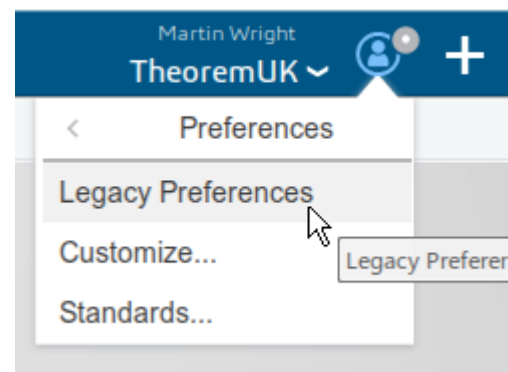
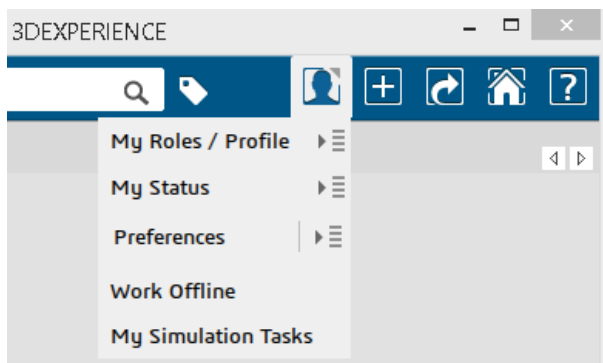
(where xx should be substituted with the version of NX – e.g. 10, 11, 12 and 1847)

(where <version> should be substituted for the version of 3DEXPERIENCE CATIA V6 that you have installed – e.g. 2019x, 2020x, 2021x):

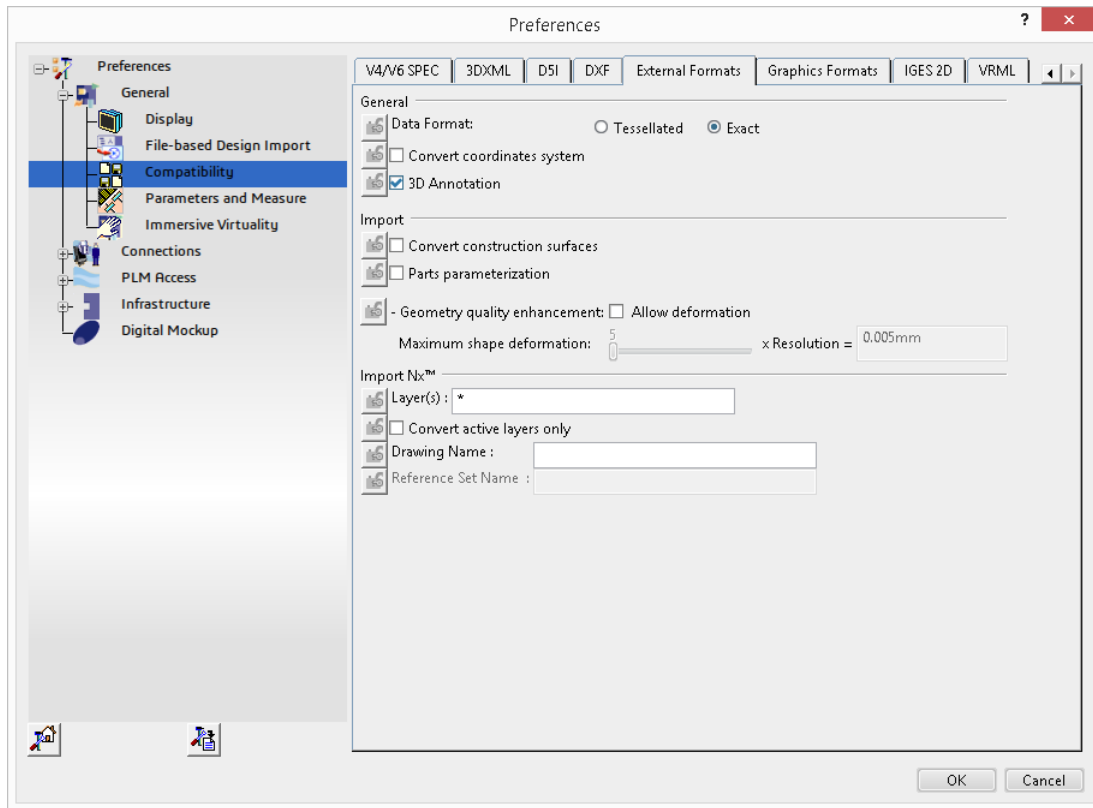
Theorem Interactive Conversion Settings

The 3DEXPERIENCE interface offer a number of setting to enable configuration of the Translation Process.

These are accessed through the User Preferences options (Preferences/Legacy Preferences for 2019x onwards) :-

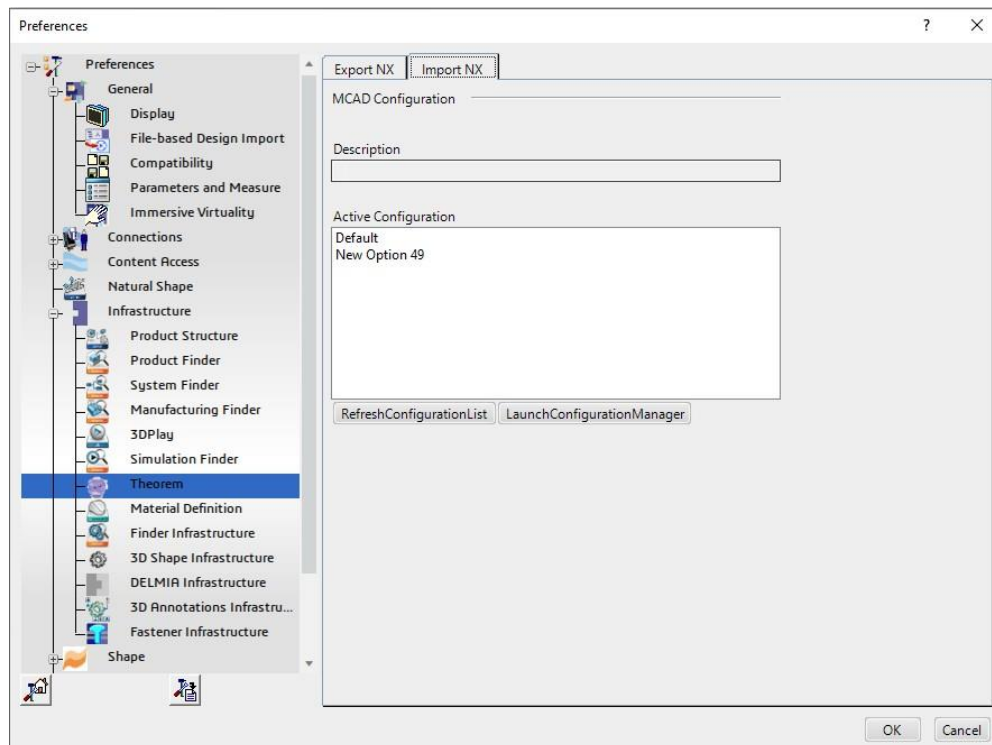


From Preferences select **<Compabability><External Formats>**



This page sets the preferred mode of conversion (in this case Exact), and also the general options “Convert coordinate systems” and “3D Annotation”.

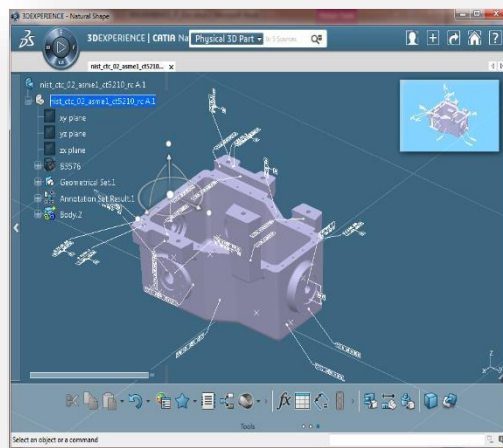
A dedicated page of options for ‘Infrastructure -> Theorem’ Export NX and Import NX’ operations is available under the ‘Infrastructure’ settings and is presented as follows:-



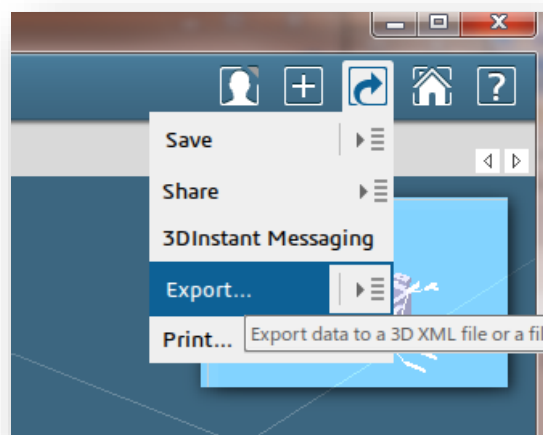
From this Panel, the user can select a predefined configuration, these are set from within the UI (see section UI) or Configuration Manager. You can also launch and refresh the Configuration Manager.

Theorem Export

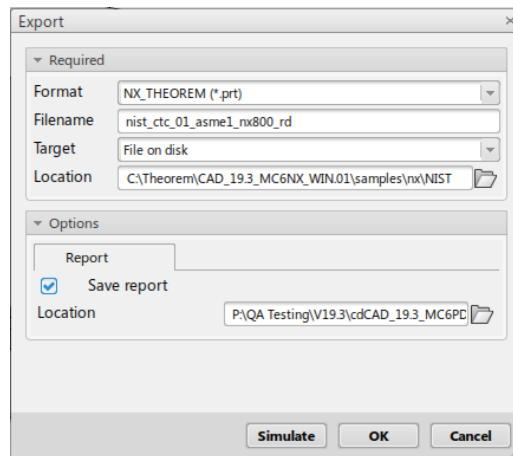
Once 3DEXPERIENCE CATIA V6 has been started it is possible for an opened product to be exported to NX.



The user selects the menu item as shown here:-



This menu action will then present the user with a Save As dialog box similar to this:-



The “Save as types:” or “Format” selection should be set to NX_THEOREM (*.prt). The **Save** or **OK** button can now be pressed to initiate the conversion to the selected output file name and location.

NX Template file usage

If, when a representation is read from 3DEXPERIENCE CATIA V6, no geometry is found, or a major write error occurs preventing an NX file being produced, an empty NX file (named by default 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 an environment variable set in the Theorem CATEnv file e.g. %TS_INST%\B214\win_b64\CATEnv\Theorem_Multi-CAD_NX_CATIAV6R<ver>.txt file, typically as follows:

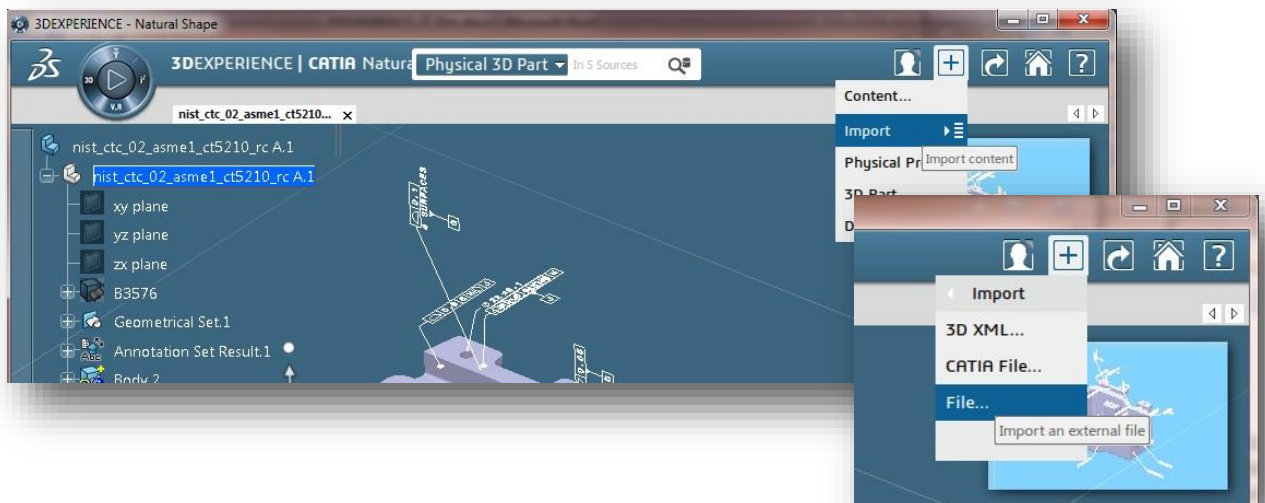
TS_NX_TEMPLATE_FILE=C:\Program_Files\Theorem\CAD_22.1_MC6NX12_WIN.01\data\ug\template.prt

The user can change the content of this JT file or the location of this file and its name as required.

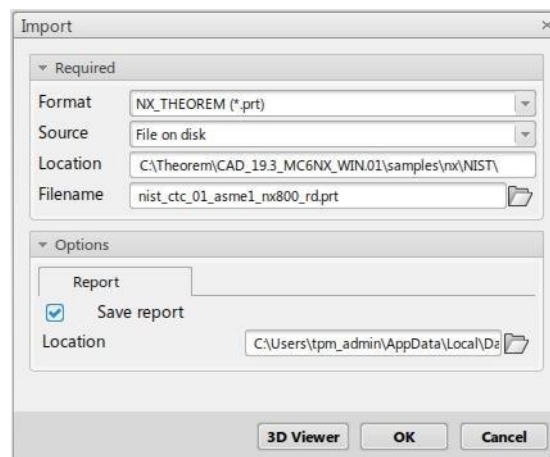
Theorem Import

Once 3DEXPERIENCE CATIA V6 has been started it is possible for a model to be imported from NX.

In order to import, the user selects the following Menu Option:



This launches the Theorem Import Menu:



On selecting **Open** or **OK** the on the File selection Panel the NX data will be translated and imported into the 3DEXPERIENCE CATIA V6 database. It will also be opened in the user's session.

Translating in Batch using CATUtil – PLMBatchDataExchange

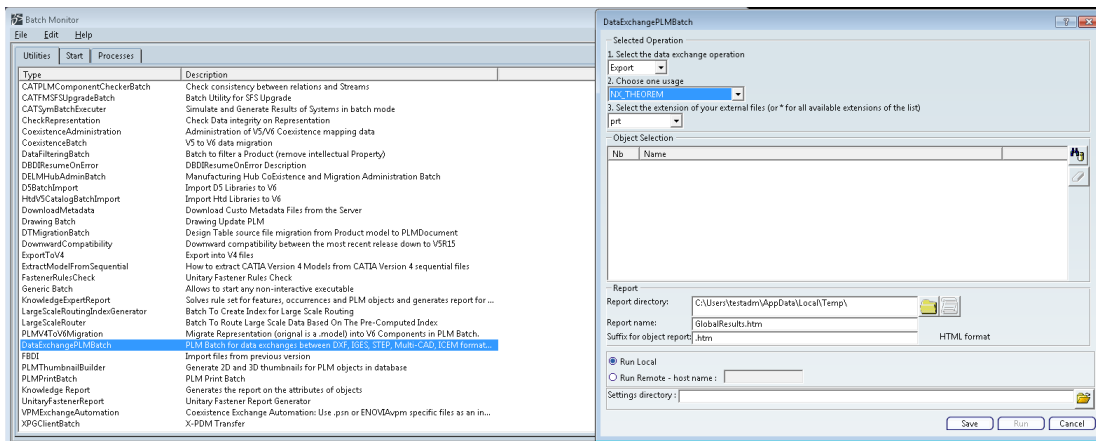
The Dassault Systemes CATUTIL batch management tool can be launched with the correct Theorem enabled environment using the scripts provided in the Translator installation at:

<installation_directory>\bin\start_CATUTIL_Theorem_Multi-CAD_NX12_CATIAV6R<revision>.cmd

(where [revision] should be substituted for the version of 3DEXPERIENCE that you have installed – e.g. 2017x, 2018x or 2019x)

Batch Export to NX

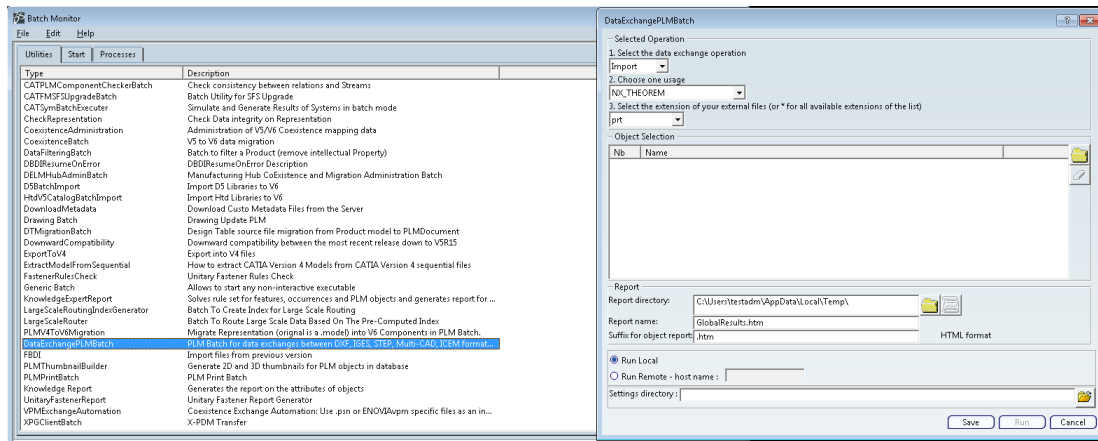
Here is an example showing the appropriate settings in preparation for a database selection and an export “save” or “run”:-



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

Batch Import from NX

Here is an example showing the appropriate settings in preparation for an input NX file selection and an export “save” or “run”:-



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

Translating from the UI (User Interface)

For CATIA V6 there are no physical files on disk. So to run a User Interface process a job file is used instead of CAD files. This job file contains all the necessary information to locate, open and translate the V6 data. Running a translation with a job file is similar to running CADverter with file based CAD models e.g. for CATIA V5. More about the job file and its contents is described in Appendix A

CADverter allows the information that is read from the source system and written to the target system to be tailored via a set of user specified arguments. Commonly used arguments are supported via the Unified Interface, with Advanced Arguments being described within this document for use in the Unified Interface or via the Command Line invocation.

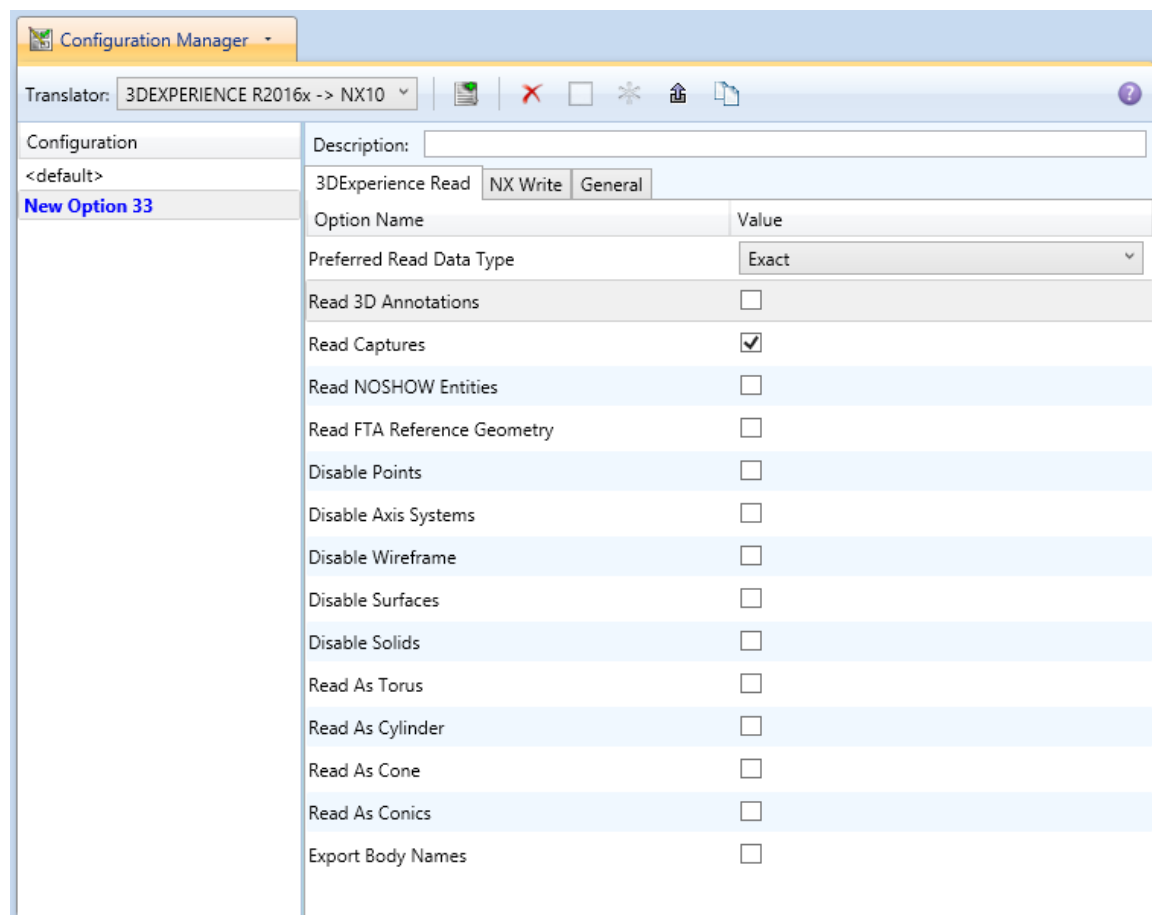
Common Options for CATIA V6 to NX

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CATIA V6 data into NX are grouped into 3 areas:

- 📁 CATIA V6 Read – Those arguments that affect how data is read from CATIA V6
- 📁 NX Write – Those arguments that affect how the data is written to NX
- 📁 General – Those arguments that are common to ALL Publishing activities regardless of source data CADverter v21.2 for CATIA V5 - NX

3DExperience Read Arguments

The image below shows the 3DExperience Read arguments that are available, with their default settings:

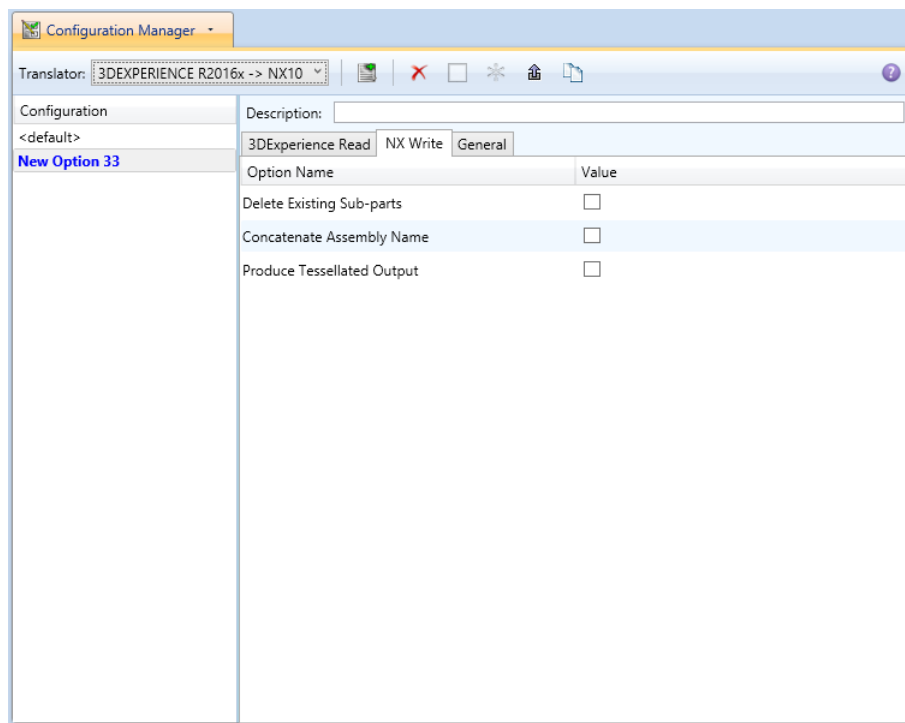


Each of these options is described below:

Option	Description
Preferred Read Data Type	<p>The setting options are EXACT (default) or TESSELATED</p> <ul style="list-style-type: none"> Command Line Syntax: <i>read_tess</i>

Read 3D Annotations	<p>Enables PMI data read from the V5 file. (Default is OFF).</p> <ul style="list-style-type: none"> ○ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>dont_read_pmi</i> – to turn off ▪ <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</p>
Read Captures	<p>A secondary argument to 'Read PMI' and allows the control over whether captures are read as part of the process. Default is ON when 'Read PMI' is marked as ON.</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>read_captures</i> - default <i>dont_read_captures</i> – to turn off
Read NOSHOWN entities	<p>Read any entities that are in NOSHOWN. Default is to not read NOSHOWN entities</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>noshown</i>
Read FTA Reference Geometry	<p>Enables reading of FTA Reference Geometry (Default is Off)</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>read_geometry</i> – to turn on
Disable Points	<p>Switches off Point processing</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>disable_points</i>
Disable Axis Systems	<p>Switches off Axis System processing</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>disable_axes</i>
Disable wireframe	<p>Switches off Wireframe processing</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>disable_wireframe</i>
Disable Surfaces	<p>Switches off Surface processing</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>disable_surfaces</i>
Disable Solids	<p>Switches off Solids processing</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>disable_solids</i>
Read as Torus	<p>Read Toroidal surfaces in analytical form (default is NURBS)</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> <i>read_torus</i>
Read as Cylinder	<p>Read Cylindrical surfaces in analytical form (default is NURBS)</p> <ul style="list-style-type: none"> • 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) <ul style="list-style-type: none">• Command Line Syntax: <i>read_cone</i>
Read as Conic	Read surfaces generated from a Conic in analytical form (default is NURBS) <ul style="list-style-type: none">• Command Line Syntax: <i>read_conics</i>
Export Body Names	Maintains body names for parts that consist of multiple bodies. <ul style="list-style-type: none">• Command Line Syntax: <i>body_names</i>



Delete Existing Sub-parts	Delete existing sub-parts. Default is OFF. <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>delete_parts</i>
Concatenate Assembly Name	Concatenates assembly name. Default is OFF. <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>concat_assy</i>
Produce Tessellated Output	Produce Tessellated NX file. Default is OFF. <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>to turn o</i>

Appendix A – Job Files

Introduction to Command Line Processing

For CATIA V6 there are no physical files on disk. So to run a command line process a job file is used instead of CAD files.

This job file contains all the necessary information to locate, open and translate the V6 data. Running a translation with a job file is similar to running CADverter with file based CAD models e.g. for CATIA V5. More about the job file and its contents is described in Appendix A

The basic command line syntax is

Option	Description
"<TS_INST>\bin\catia6r2020x_nx120.cmd"	Command to run
C:\parts\2020xjobs\AS1_2020x.xml	Input job file
AS1_2020xTest	NX .prt name

Where:-

<TS_INST>
is the Theorem Solutions software installation directory.

<input_file>

is an xml file defining the access to a specific object in a specified 3DEXPERIENCE PLM database.

This file provides a user Login Ticket (2019x, 2020x, 2021x), to a specified Enovia repository plus a set of **3 attribute names and values** which will uniquely identify the input PLM object of the conversion.

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

<3dex_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 to be defined 51097330-
00023109"/>
  <attributekey="V_version" value="A"/>
  <attributekey="minorrevision" value="1"/>
</arguments>
</3dex_read>
```

<output_file>

is the required output NX file name.

Example :

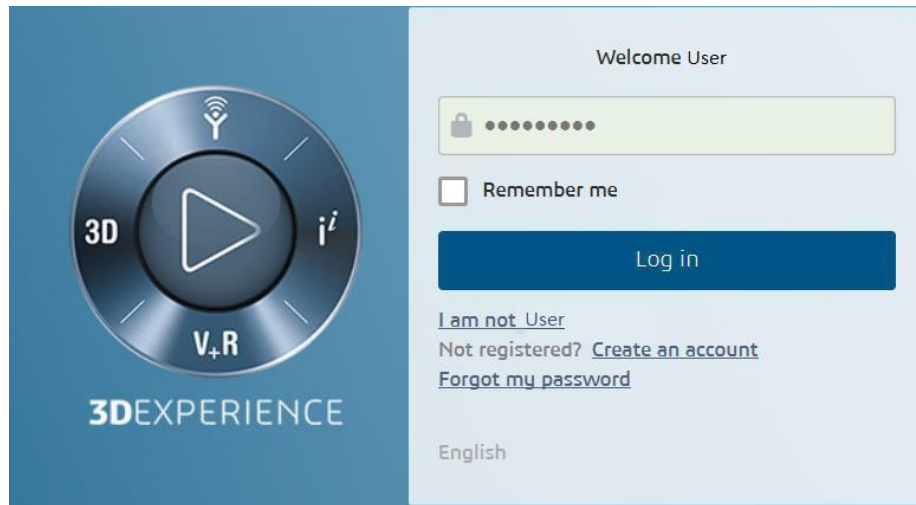
C:\Program Files\Theorem\24.1\MC6NX12\bin\catia6r2020x_nx120.cmd
C:\Temp\3dex_r2020x_inputs.xml C:\Temp\3dex_r2020x_inputs.prt

Appendix B – Creating a Login Ticket

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

<https://3dspace.2018x.theorem.com:447/3dspace/common/emxNavigator.jsp>

A page will appear in the browser

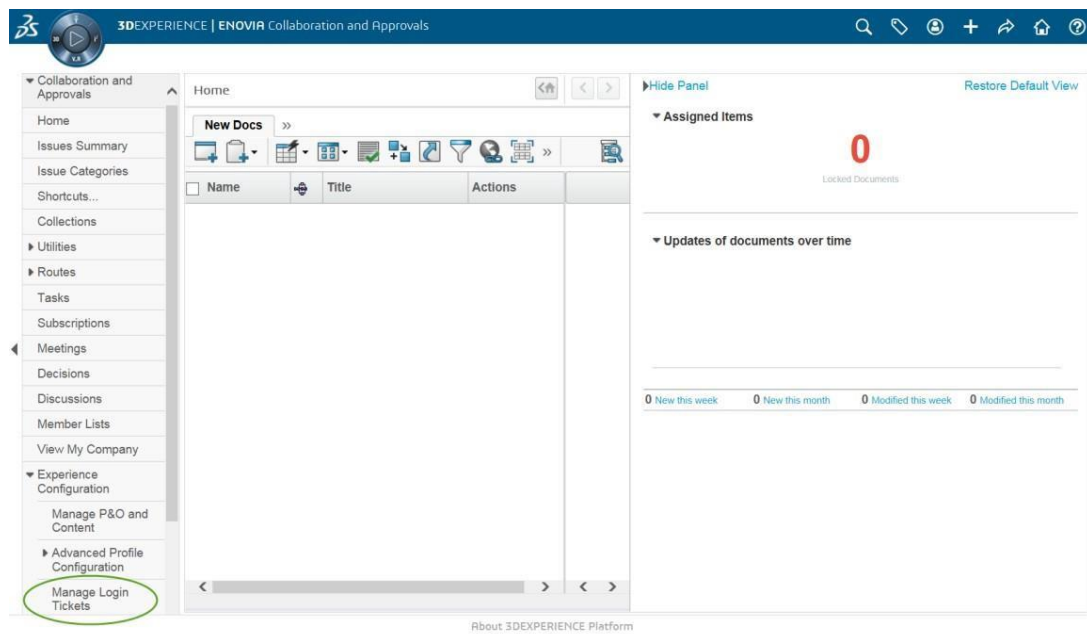



Login as normal, e.g.

User

Password

Select Collaboration and Approvals > Experience Configuration > Manage Login Tickets





The screenshot shows the 3DEXPERIENCE | ENOVIA Collaboration and Approvals interface. On the left is a navigation menu with options: Collaboration and Approvals, Home, Issues Summary, Issue Categories, Shortcuts..., Collections, and Utilities. The main content area is titled "Login Ticket Creation" and contains a form with the following fields:

User:	User
Security Context:	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 in the job xml file described in Appendix A

Appendix C – Log File Generation

Export process Log Files

In the process of exporting the selected 3DEXPERIENCE CATIA V6 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 3DEXPERIENCE CATIA V6 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 CATIA V6 'data read' processing into Theorem Intermediate data format

%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.

%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.

Import process Log Files

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

The process log and error messages are recorded in a '.err' file located in the 3DEXPERIENCE CATIA V6 CATReport directory. The file is named after the selected input file name. e.g. if the file Mypart.jt were selected, the log file names would be %CATReport%\Mypart.err and %CATReport%\Mypart.rpt.

%TS_TEMP_DIR%\<input_file_name>_viewer.log

%TS_TEMP_DIR%\<input_file_name>_viewer_screen_output.log

%CATReport%\<input_file_name>.err

Where <Input_file_name> - is the base file name of the imported file.

3DEXPERIENCE CATIA V6 Environment Files

As part of the Theorem installation process, a set of 3DEXPERIENCE CATIA V6 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 import and export.

A 'CATEnv' file is created for each installed version of 3DEXPERIENCE CATIA V6.

These environment files are located in

<installation_directory>\B420\win_b64\CATenv\Theorem_Multi-CAD_NXxx_CATIAV6R2018x.txt

<installation_directory>\B421\win_b64\CATenv\Theorem_Multi-CAD_NXxx_CATIAV6R2019x.txt

<installation_directory>\B422\win_b64\CATenv\Theorem_Multi-CAD_NXxx_CATIAV6R2020x.txt

<installation_directory>\B423\win_b64\CATenv\Theorem_Multi-CAD_NXxx_CATIAV6R2021x.txt

Where xx is the NX ve

