

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 <u>website</u> for more detail.



PUBLISH

The creation of documents enriched with 3D content

See our <u>website</u> for more detail.



VISUALIZE

Visualization for <u>Augmented (AR)</u>, <u>Mixed (MR)</u> and <u>Virtual (VR)</u> Reality applications

See our <u>website</u> 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 <u>sales@theorem.com</u>





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:

CATIA 3DEXPERIENCE <> NX	
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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 <u>www.theorem.com/documentation</u>

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 <u>www.theorem.com/documentation</u>

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 <u>www.theorem.com/documentation</u>

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



Steve Bee TheoremUK ~	٩	2850.5.000
My Roles / Profile		
My Status	>	< Preferences
Preferences	5	Legacy Preferences
Work Offline		Customize
My Offline Content	- 1	Standards
My Favorite Content	5	

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.

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OK Cancel





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.

Preferences ?	×	
Preferences General Display File-based Design Import Compatibility Parameters and Measure Immersive Virtuality Connections Content Access Natural Shape Infrastructure Product Structure Design Poduct Finder System Finder Solplay Simulation Finder Shape Shape	• 1	▶
	uncer	IJ

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.

Preferences	? ×
Preferences General Display File-based Design Import Compatibility Parameters and Measure Immersive Virtuality Connections Content Access Natural Shape Infrastructure Product Structure Design Product Structure Design Product Structure Design Product Structure Design Product Structure Sustem Finder Manufacturing Finder Sustem Finder Material Definition Finder Infrastructure So Shape Infrastructure DELMIA Infrastructure Stape Shape	? X
	OK Cancel





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

Theorem Solutions (Configuration Manager	- 🗆 ×
Exit Home	nslator: CE R2022x -> NX 2206 Y Help Selected Translator	elete Clear Copy
Configuration	Description:	
<default></default>	3DExperience Read NX Write Genera	1
	Option Name	Value
	Preferred Read Data Type	Exact
	Read PMI	
	Read Captures	×
	Read FTA Reference Geometry	
	Read NOSHOW Entities	
	Disable Points	
	Disable Axis Systems	
	Disable Wireframe	
	Disable Surfaces	
	Disable Solids	
	Read As Torus	

Theor	em Solutions Configu	iration Manager		_	×
Exit Home	CADverter Translator 3DEXPERIENCE R20 Select	22x -> NX 2206 Y Help New Delete Configu	Clear Copy		
Configura <default></default>	tion	Description: 3DExperience Read NX Write General			
New Optio	n 6	Option Name	Value		
		Preferred Read Data Type	Exact		× ^
		Read PMI			
		Read Captures	\checkmark		
		Read FTA Reference Geometry			
		Read NOSHOW Entities			
		Disable Points			
		Disable Axis Systems			
		Disable Wireframe			
		Disable Surfaces			
		Disable Solids			
		Read As Torus			\sim





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

Preferences		?	×
Preferences Preferences General Display File-based Design Import Compatibility Parameters and Measure Immersive Virtuality Connections Content Access Natural Shape Infrastructure Product Finder System Finder System Finder System Finder System Finder System Finder System Finder System Finder System Finder System Finder Simulation Finder Simulation Finder DELMIA Infrastructure DELMIA Infrastructure DELMIA Infrastructure So Shape Shape	Export NX Import NX Configuration Description Active Configuration Default New Option 6 RefreshConfigurationList LaunchConfigurationManager	?	×
		ОК	Cancel





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

Theorem Solutions Configuration Manager – 🗆 🗙		
Exit CADverter Translato	r: D22x -> NX 2206 Y Help New Delete	Clear Copy
Home Selec	ted Translator Configur	rations
Configuration	Description:	
<default></default>	3DExperience Read NX Write General	
New Option 6	Option Name	Value
	Preferred Read Data Type	Exact ~
	Read PMI	
	Read Captures	\checkmark
	Read FTA Reference Geometry	
	Read NOSHOW Entities	
	Disable Points	
	Disable Axis Systems	
	Disable Wireframe	
	Disable Surfaces	
	Disable Solids	
	Read As Torus	
	Read As Cylinder	
	Read As Cone	
	Read As Conics	
	Export Body Names	





Translate for 3DExperience <> NX

Option	Description
Preferred Read Data type	The setting options are EXACT (default) or TESSELATEDCommand Line Syntax:<i>read_tess</i>
Read PMI	 Enables PMI data read (Default is OFF). Command Line Syntax: dont_read_pmi – default read_pmi – to turn on Note! When 'read_pmi' is enabled it also enables the 'fill_pmi_arrows', 'fill_pmi_text' and 'pmi_filled_text' options. These can be overridden by setting the Advanced arguments: 'dont_fill_pmi_arrows' and/or 'dont_fill_pmi_text'
Read Captures	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. Command Line Syntax: • read_captures – default • don't_read_captures – to turn off
Read FTA Reference Geometry	 Enables reading of FTA Reference Geometry (Default is Off). FTA Reference Geometry can ONLY be read when '<i>Read PMI</i>' is ON. Command Line Syntax: read_geometry – to turn on
Read NOSHOW Entities	 Read any entities that are in NOSHOW. Default is to not read NOSHOW entities Command Line Syntax: noshow
Disable Points	Switches off Point processing Command Line Syntax: • disable_points
Disable Axis Systems	Switches off Axis System processing Command Line Syntax: • disable_axes





Translate for 3DExperience <> NX

Disable Wireframe	Switches off Wireframe processingCommand Line Syntax:<i>disable_wireframe</i>
Disable Surfaces	Switches off Surface processing Command Line Syntax: • disable_surfaces
Disable Solids	Switches off Solids processing Command Line Syntax: • disable_solids
Read As Torus	 Read Toroidal surfaces in analytical form (default is NURBS) Command Line Syntax: read_torus
Read As Cylinder	 Read Cylindrical surfaces in analytical form (default is NURBS) Command Line Syntax: read_cylinder
Read As Cone	Read Cone surfaces in analytical form (default is NURBS)Command Line Syntax:read_cone
Read As Conics	 Read surfaces generated from a Conic in analytical form (default is NURBS) Command Line Syntax: read_conics
Export Body Names	Maintains body names for parts that consist of multiple bodies.Command Line Syntax:<i>body_names</i>





NX Write

Option	Description
Delete Existing Sub-parts	 Delete existing assembly subcomponents. (Default is Off) Command Line Syntax: <i>delete_parts</i>
Concatenate Assembly Name	Concatenate top level assy name to all subcomponents. (Default is Off) Command Line Syntax: • concat_assy
Produce Tessellated Output	Produce Tessellated NX file. (Default is Off)Command line syntax:tess_output







General

Theor	Theorem Solutions Configuration Manager – 🗆 🗙					
Exit Home	CADverter Translator 3DEXPERIENCE R20 Select	22x -> NX 2206 Y Help New Delete Clear Copy Configurations				
Configura <default></default>	tion	Description: 3DExperience Read NX Write General				
New Optio	no	Option Name Value				
		Mass Properties				
		Advanced				

Option	Description
Mass Properties	Allows mass property information to be readCommand line syntax:<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

T heo	rem Solutions Configu	uration Manager	- 🗆 X
Exit Home	CADverter Translator NX 2206 -> 3DEXPE Select	ERIENCE R2022x Y Help New Delete configura	Clear Copy ations
Configura	ation	Description: NX Read 3DEXPERIENCE Write General	
New Option 7		Option Name	Value
		Reference Set	\checkmark
		Read Attributes	
		Read NX names	
		Read PMI	

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

Theorem Solutions Configuration Manager				×
Exit Home CADverter Translator NX 2206 -> 3DEXP	ERIENCE R2022x Y Help New Delete Clear Copy ted Translator Configurations			
Configuration	Description:			
<default></default>	NX Read 3DEXPERIENCE Write General			
New Option /	Option Name Value			
	PLM Connection File			=
	Axis Systems - Show			
	Property Mapping File			6





Translate for 3DExperience <> NX

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	A mapping file is used to control which properties are converted from NX to 3DExperience by setting a control value. 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 The File Line Format is as follows: - SourceName, TargetName, Control, Dummy, Dummy, Dummy • SourceName – is the input attribute name. • TargetName – is the output attribute name (NULL means use SourceName) • Control – is flag to control conversion: • 0 – Do no convert • 1 – Do convert • Dummy – unused fields Examples: MPARTNAME,,O,,,, attribute not exported MPARTNAME,NULL,1,,,, attribute exported using the SourceName MPARTNAME,Part Name,1,,,, attribute exported using the TargetName specified







General

Theorem Solutions Config	uration Manager	_	×
Exit CADverter Translator NX 2206 -> 3DEXPE	ERIENCE R2022x Y Help New Delete Clear Copy Configurations		
Configuration <default></default>	Description: NX Read 3DEXPERIENCE Write General		
New Option 7	Option Name Value		
	Advanced		

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

	—	þ	×
Theorem U	к~ 🧐 +	À	?
	Save	>	×
	Share	>	
	3DMessaging		
	Export	>	
	Print		





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 intitiate the translation.

Export			×
▼ Required			
Format	NX_THEORE	M (*.prt)	
Target	File on disk	•	
Location	C:\temp		
Filename	BRAKE DISC		
▼ Options Report			
Save rep	port		
Location		C:\Users\stephen.clews\AppData\L	
Report name		ExportReport_(719)	
		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.

		—	þ	×
Theo	rem UK 🗸 🐨	+	Â	?
	New Content			×
	Import	>		
	Physical Product	t		
	3D Part			
	Drawing			

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 intitiate 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.

Import		×
▼ Required		
Format	NX_THEORE	EM (*.prt)
Source	File on disk	•
Location	C:\Program	Files\Theorem\25.3_3DExperience_NX\
Filename	nist_ctc_05_a	asme1_nx800_rd.prt
▼ Options		
Report	ort	
Location		C:\Users\stephen.clews\AppDat
Report name		ImportReport_(131)
		3D Viewer OK Cancel





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.

3DEXPERIENCE				
	TIA Assembly Design	Search	۹ 🛇	
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htt_ctc_01_asme1_m800_rd sist_ctc_01_asme1_m800_rd yplane yplane			Select or search an object or Press ESC to exit Multiselection 1 object selected	
yz piane zx piane Axis Systems How B1271				
⊕r⊼ <u>Geometries</u>				
	Nest			







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.

Batch Monitor		-		>
Litilities Start Processes				
Tare	Description			_
Type The Data Charl	Description			
FastenerRulesCheck	Unitary Fastener Rules Check			
Generic Batch	Allows to start any non-interactive executable			
mportific Custiviap	Batch IFC Import Create Custom Mapping File			
	Import a 3D XIVIL file			
Laws laduate Manufacturing Operations	Solves rule set for features, occurrences and PLW objects and general	es repo	IT TOF K	
Heavy industry Manufacturing Operations	Utility to convert Thread Standard files to YML format	docum	ients	
	Transition logacy CAD data to 2DEVDEPIENCE CAD origin			
DataExchangePI MPatch	Batch app for data exchanges between DYE IGES STEP, Multi-CAD //	CEM for	mate a	-
	Import files from previous version	CEIVITOI	illats a	-
PLMCobotIntegrationBatch	Launch Integration Batch			
	Generate 2D and 3D thumbhails for PLM objects in database			
	Launch Interference Simulation computation			
PI MPrintBatch	Print Batch			
DecBatchReport	Run Profiles and generates report for knowledge check			
Knowledge Report	Generates the report on the attributes of objects			
V6 CATIA Analysis Migration	Migrates CATIA Analysis Model to Physics App Model and the Elfini d	lomain	of mat	
Import Simulation Results	Imports Simulation Results in a Simulation Object	omann	or mac	
Create Sim Nav Data Back	Grantes sim nav data nackage for confiles			
NX_I HEOREM	s (or * for all available extensions of the list)			
prt ~				
Object Selection Nb Name			- 1	H _a
			(9
Report				
Report directory: C:\Users\STEPHE~	1.CLE\AppData\Local\Temp\ 🦳			
Report name: GlobalResults.htm				
Suffix for object report: .htm	HTML format			
Run Local				
O Run Remote - host name :				
Settings directory :				P
				_
	Save		Ca	nce

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.

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	Add	Vodify Remove Clear				
Attribute	Operator	Value	End value			
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itle	Like	ENGINE BRACKET & BOLT				
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elected Operation — Select the data excha xport v Choose one usage X_THEOREM Select the extension rt v biost Selection	nge operation	n al files (or * for all available	extensions of t	he list)		
elected Operation Select the data excha kport Choose one usage X_THEOREM Select the extension rt T T T T T T T T T T T T T T T T T T	nge operation	n al files (or * for all available	extensions of t	he list)	Partition Co.	Descript #
elected Operation — Select the data excha kport v Choose one usage X_THEOREM Select the extension of the vector of the set of the se	nge operation	al files (or * for all available	extensions of t Revision	he list) Usage	Revision Co	Descriptic
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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 of Export ~ 2. Choose one usage NX_THEOREM 3. Select the extension of you prt ~	peration	ilable extensions of t	he list)			
Object Selection						
Nb Title 1 BRAKE DISC & BOLTS 2 ENGINE BRACKET & . 3 HOLLOW BLOCK	Name prd-TSL-00017166 . prd-TSL-00004533 prd-TSL-00004529	Revision A A.1 A.1	Usage 3DPart	Revision Co	Description For part Parts with With PM	** 3
Remark						
Report directory: C:\ter Report name: Glob Suffix for object report: .htm	np\Batch\ alResults.htm		нтм	L format		
Run Local Run Remote - host name Settings directory :	:					
				Save	Run	Cancel

Click Run to initiate the Batch Export to NX.

Name	Date modified	Туре	Size
📕 Data	13/01/2023 15:16	File folder	
🎐 Bolt.prt	13/01/2023 15:16	ImageView Docu	70 KB
💿 BRAKE DISC & BOLTSA.htm	13/01/2023 15:16	Chrome HTML Do	1 KB
夑 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
夑 ENGINE BRACKET BOLT.prt	13/01/2023 15:16	ImageView Docu	73 KB
夑 ENGINE BRACKET.prt	13/01/2023 15:16	ImageView Docu	226 KB
🧿 GlobalResults.htm	13/01/2023 15:16	Chrome HTML Do	2 KB
O HOLLOW BLOCKA.1.htm	13/01/2023 15:16	Chrome HTML Do	1 KB
夑 prd-TSL-00004529.prt	13/01/2023 15:16	ImageView Docu	85 KB
夑 prd-TSL-00004533.prt	13/01/2023 15:16	ImageView Docu	76 KB
夑 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.

Batch Monitor	- 0	
e <u>E</u> dit <u>H</u> elp		
Utilities Start Processes		
Туре	Description	
FastenerRulesCheck	Unitary Fastener Rules Check	
Generic Batch	Allows to start any non-interactive executable	
mportlfcCustMap	Batch IFC Import Create Custom Mapping File	
Import 3D XML file	Import a 3D XML file	
KnowledgeExpertReport	Solves rule set for features, occurrences and PLM objects and generates report for k	
Heavy Industry Manufacturing Operations	Generate Manufacturing features or Extract Manufacturing workshop documents	
MigrateThreadStandardToXML	Utility to convert Thread Standard files to XML format	
3DEXPERIENCE CAD Transition	Transition legacy CAD data to 3DEXPERIENCE CAD origin	
DataExchangePLMBatch	Batch app for data exchanges between DXF, IGES, STEP, Multi-CAD, ICEM formats a	
FBDI	Import files from previous version	
PLMCobotIntegrationBatch	Launch Integration Batch	
PLMThumbnailBuilder	Generate 2D and 3D thumbnails for PLM objects in database	
PLMItfCheck	Launch Interference Simulation computation.	
PLMPrintBatch	Print Batch	
QccBatchReport	Run Profiles and generates report for knowledge check	
Knowledge Report	Generates the report on the attributes of objects	
V6 CATIA Analysis Migration	Migrates CATIA Analysis Model to Physics App Model and the Elfini domain of mat	
Import Simulation Results	Imports Simulation Results in a Simulation Object	
Croate Sim Nav Data Dack	Craster sim nav data nackano for carfilor	
Croate Sim Nav Data Dack	Croster cim nau data nackana far car filer	
DataExchangePLMBatch	?)
Selected Operation		
1. Select the data exchange operation		
Import ~		
2. Choose one usage		
NX_THEOREM ~		
3. Select the extension of your external file	es (or * for all available extensions of the list)	
prt ~		
Object Selection		
,		-

2. choose one usag		
NX_THEOREM	~	
3. Select the extens	sion of your external files (or * for all available extensions o	f the list)
prt	~	
Object Selection —		
Nb Name		
		e
Report		
Report directory:	C:\Users\STEPHE~1.CLE\AppData\Local\Temp\	
Report name:	GlobalResults.htm	
Suffix for object rep	port: .htm	HTML format
Run Local		
Kun Local		
O Run Remote - h	nost name :	
Settings directory :		
		Save Run Canc

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.

3DVIA - Live File Connector -> 3DEXPE	RIENCE Platform Object Management	_	X
Input Options Select Member All St Extend selection to sub-directorie Select by modification date Sinc Add documents O Replace doc	ring	B	<u> </u>
Output Options		ОКС	ancel Help

DataExchangePLMB	atch	?	×
Selected Operation 1. Select the data e Import 2. Choose one usa NX_THEOREM 3. Select the extense prt	a cchange operation je ion of your external files (or * for all available extensions of the list) v		
- Object Selection -			
Nb Name 1 C:\Progran 2 C:\Progran 3 C:\Progran	Files\Theorem\25.3_3DExperience_NX\samples\nx\NIST\nist_ctc_02_asme1_nx800_rc Files\Theorem\25.3_3DExperience_NX\samples\nx\NIST\nist_ctc_03_asme1_nx800_rc Files\Theorem\25.3_3DExperience_NX\samples\nx\NIST\nist_ctc_04_asme1_nx800_rd		
Report			
Report directory:	C:\Users\STEPHE~1.CLE\AppData\Local\Temp\		
Report name:	GlobalResults.htm		
Suffix for object re	bort: htm HTML format		
 Run Local Run Remote - H 	ost name :		
Settings directory			2
	Save	n	Cancel

Click Run to initiate the Batch Import from NX.

3 R	tesults Q		
	Title	Туре	Description
1	 ✓ Image: Second state ✓	Physical Product	
2	 ✓ Image: Second state of the second	Physical Product	
3	✓ Inist_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="REEyNzM3M0Q1REM2NDgxQzIFNzk1QzIwNjZGRDYzODN8Um9iaW58Um9ia
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_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	:-	
Туре	Total	Standalone	Subordinate
Lines Curves	237		237 468
Surfaces Planes	189 81		189 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.





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

3DEXPER	IENCE ENOVIA	Collaborat	ion and Approval	S					Q	0	٢	÷	Ŵ	ŵ	0
Collaboration and Approvals	Home				<n< td=""><td>< ></td><td>▶Hide Panel</td><td></td><td></td><td></td><td></td><td>Res</td><td>tore De</td><td>afault \</td><td>/iew</td></n<>	< >	▶Hide Panel					Res	tore De	afault \	/iew
Home	New Docs) »>					 Assigned Iter 	ms							
Issues Summary		mf - 1	()- III - I	75	· 🛛 🗐 »	1			0						
Issue Categories					Co Frag	leav			kant Deces						
Shortcuts	Name	-	Title		Actions										
Collections															
▶ Utilities							 Updates of d 	ocuments over the	ne						
▶ Routes															
Tasks															
Subscriptions															
▲ Meetings															
Decisions															
Discussions							0 New this week	0 New this month	0 1	locified t	his week	0 м	oddied ti	his mon	th
Member Lists															
View My Company															
 Experience Configuration 															
Manage P&O and Content															
 Advanced Profile Configuration 															
Manage Login Tickets	<				>	< >									
					About 3 DEXPER	ENCE Platform	n ()								





	ERIENCE ENOVIA Colla	boration and Approvals				
 Collaboration and Approvals 		akat Craatian				
Home	Login II	cket Creation				
Issues Summary	User:	User				
Issue Categories	Security Context:	VPLMProjectLeader.Company Name.Acme	~			
Shortcuts	Ticket type:	● Infinite				
Collections		Create				
▶ Utilities						

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:-

REEyNzM3STE1MER2NDgxQzlFNzk1QzlwNjZGNATzDAN8Um9iaW58Um9iaW58fHwwfA==

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

