User Guide

CATIAV5<> NX

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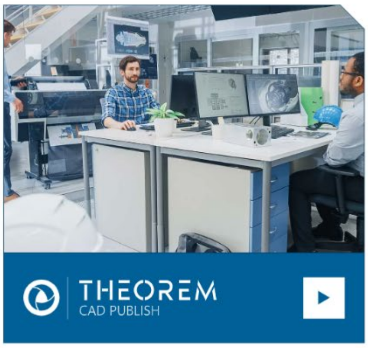


Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of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

We are proud that the vast majority of the world's leading Automotive, Aerospace, 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:

CAD**Translate**

CAD Data Exchange: Seamless and robust data translation between CAD and Visualization formats.

See our website for more detail.

CAD**Publish**

Interactive Documentation: 3D PDF Publisher and Composer. Making CAD data accessible to non-CAD users.

See our website for more detail.

Theorem**XR**

Visualization for Augmented (AR), Mixed (MR) and Virtual (VR) Reality applications

Extended Reality – XR: Augmented, Mixed and Virtual Reality for the Engineering Metaverse.

See our website for more detail.

The CATIA V5 Bi-directional NX Translator

The CATIA V5 to NX translator may be installed on a number of machines each accessing a central network-floating license.

The CATIA V5-NX Translator is a bi-directional direct database converter between the Dassault Systèmes CATIA V5 Modelling Application and the NX file format, used by the Siemens NX Products.

It enables the user to convert all forms of 3D Mechanical Design Geometry and Assembly data, together with system defined attribute information, colour information, between these two systems. This product is designed for companies using CATIA V5 who have selected NX to be their main method of collaboration and communication between OEMs and their customers or suppliers.

It is also a major method of visualization and therefore companies using NX based solutions need to translate their CATIA V5 data into the NX format.

The translator can be invoked in either an interactive or batch mode with the command line interface allowing the conversion process to be integrated into any process oriented operation. Alternatively from Theorem R18 the conversion process may be operated by using the new Theorem Unified Interface.

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 CATIA V5 and NX installations.
- The conversion process can be run Interactively, Batch Mode or using the new Unified Interface
- Command line interface allows process integration.
- Data can be filtered by layer and entity type during processing. Geometry can be filtered and selectively processed.
- Uses the CATIA V5 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 the Visualize 3D for CATIA V5 – NX 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. 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 .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation and select UI from the product selection list.

License Configuration

To run any product a valid license file is required. The Flex License Manager is run from the .msi file download provided. 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

Using the Product

Default Translations

Default Translation – via the Unified Interface

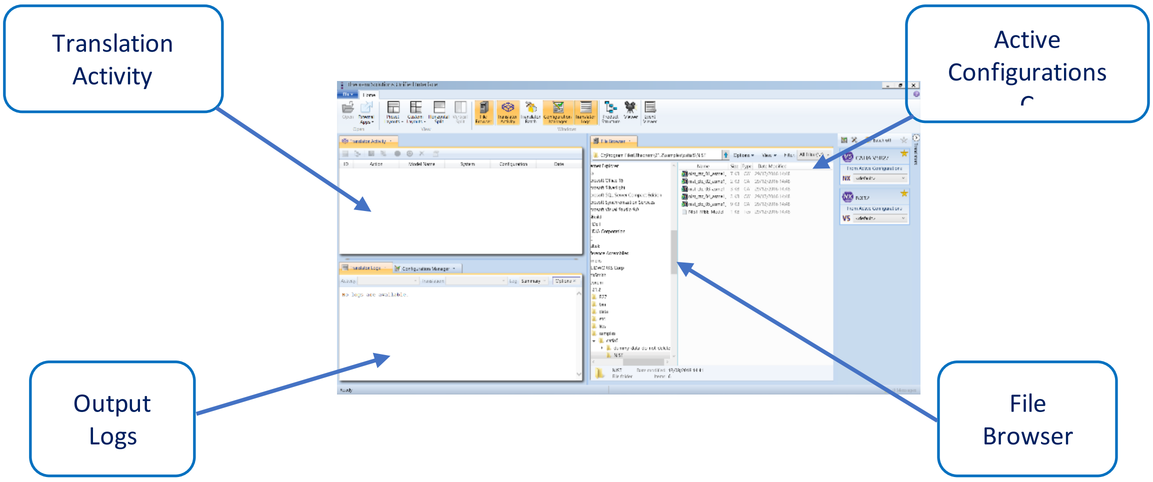
The Unified Interface can be started via the Start Menu – if a shortcut was added during installation.

Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

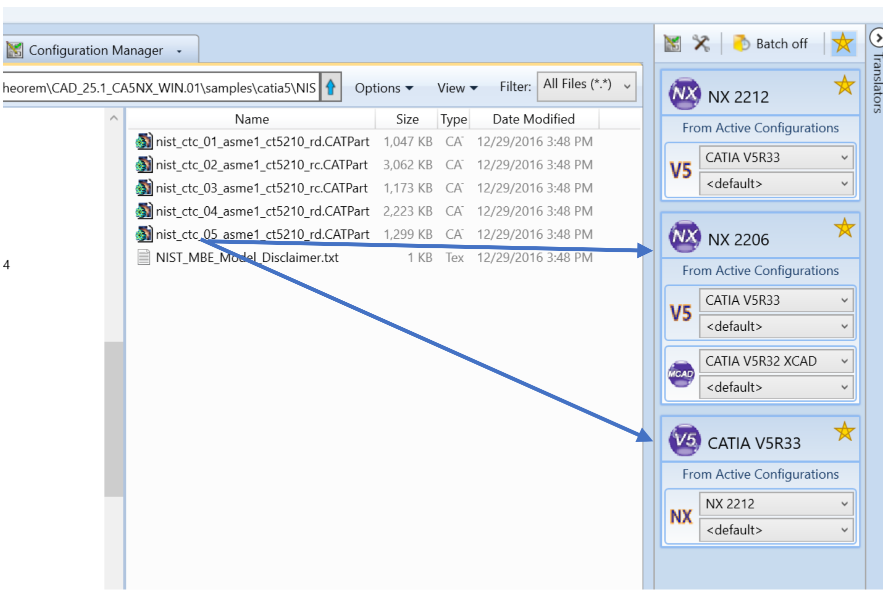
<UI_installation_directory>\bin\Unified_Interface.cmd

The following interface will be launched:

The default layout is split into 4 primary areas, which can be altered to the users prefer:

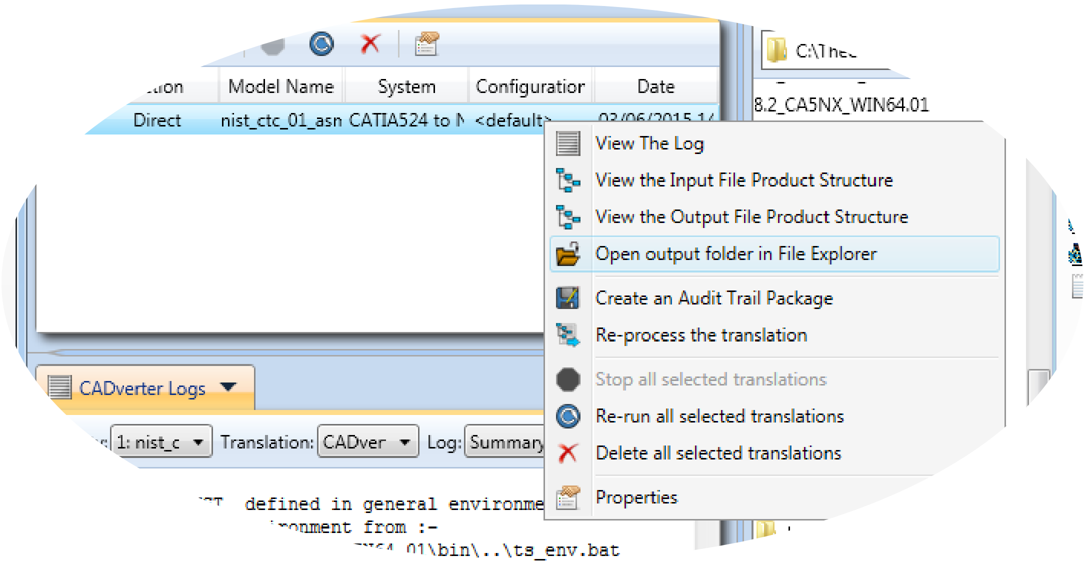


The simplest way to translate from CATIA V5 or NX is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



On completion, the Unified Interface will display the activity information and details from the log file created during the translation, if requested, in the Translation Activity and Output Log panes, respectively.

The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:



Default Translation – via the Command Line

Running a translation via the command line can be carried out via the ***cad_run.cmd*** file located in the ***<installation_directory>\bin*** directory. The format of the command is as follows when translating from CATIA V5 to NX:

<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_NX[XX] –I <input_file> -o <output_file>

The format of the command is as follows when translating from NX to CATIA V5:

<Translator_installation_directory>\bin\cad_run.cmd NX[XX]_CATIA5[XX] –I <input_file> -o <output_file>

(Note! Replace the [XX] seen in the example with the version of CATIA V5 or NX you are using. E.g. for CATIA V5 R33, change to CATIA533 and for NX 2212 change to NX2212):

```
C:\Users\bpittman>"C:\Program Files\Theorem\CAD_26.0_CA5NX_WIN.01\bin\cad_run.cmd" CATIA533_NX2212 -i "C:\Program Files\Theorem\CAD_26.0_CA5NX_WIN.01\samples\catia5\NIST\nist_ctc_05_asme1_ct5210_rd.CATPart" -o "C:\Temp\demo_output\nist_ctc_05_asme1_ct5210_rd.prt"
```

The example above will translate a CATIA V5 sample file provided within the installation and produce the following screen output

```
C:\Program Files\Theorem\CAD_26.0_CA5NX_WIN.01\bin\..\ts_env.bat
Setting run time environment please wait ...
Using TS_INST = C:\Program Files\Theorem\CAD_26.0_CA5NX_WIN.01\
using CATIAV5_INST = C:\Program Files\Dassault Systemes\B33\
Using Platform win_b64
Launching translation please wait ...
DIRENV C:\Users\bpittman\AppData\Roaming\DassaultSystemes\CATEnv
ENV TheoremCatia5R33

 ******************************************************
 * Copyright Theorem Solutions Limited                *
 * CATIA - Siemens NX 2212 CADverter Version 26.0.002 *
 ******************************************************

 Tue Apr 25 11:07:51 2023

 Input
   CATIA5    CATPart : C:\Program Files\Theorem\CAD_26.0_CA5NX_WIN.01\samples\catia5\NIST\nist_ctc_05_asme1_ct5210_rd.CATPart
   NX           Part : C:\Temp\demo_output\nist_ctc_05_asme1_ct5210_rd.prt
   Progress File     : C:\Temp\demo_output\nist_ctc_05_asme1_ct5210_rd.progy5.log

 V4          option license available
 PDF         option license available
 SCAN        option license NOT available

   CATIA5 Read Leg : 26.0.002

   Build Release   : CATIAV5.B33.SP0 64
 CATIA V5 Cache Mode   : Disabled
 CATIA V5 Saved Version: V5R21 SP0 BD 04-14-2011.20.00


 List of gco entities :-
 -----------------------
 Type         Total    Standalone  Subordinate
 -----------  -------  ----------  -----------
 Arcs          203                  203
 Lines         262      1           261
 Curves        39                   39
 Surfaces      4                    4
 Cones         26                   26
 Cylinders     100                  100
 Spheres       4                    4
 Torus         8                    8
 Planes        63                   63
 Faces         205                  205
 Edges         504                  504
 Vertices      328                  328
 Bsolids       1        1
 Open solids   1        1
 Axis systems  1        1
 -----------  -------  ----------  -----------


 ***************************************************
 * Copyright Theorem Solutions Limited             *
 * GCO - Parasolid 34.1 CADverter Version 26.0.001 *
 ***************************************************

 Tue Apr 25 11:08:01 2023

 Input
   GCO File       : c:\users\bpittman\appdata\local\temp\tscug2_31900.gco
   Parasolid File : c:\users\bpittman\appdata\local\temp\tscug2_31900
   Progress File  : C:\Temp\demo_output\nist_ctc_05_asme1_ct5210_rd.progy5.log.a6


 List of gco entities :-
 -----------------------
 Type         Total    Standalone  Subordinate
 -----------  -------  ----------  -----------
 Arcs          203                  203
 Lines         257                  257
 Curves        39                   39
 Surfaces      4                    4
 Cones         26                   26
 Cylinders     100                  100
 Spheres       4                    4
 Torus         8                    8
 Planes        62                   62
 Faces         204                  204
 Edges         499                  499
 Vertices      322                  322
 Bsolids       1        1
 -----------  -------  ----------  -----------

 *****************************************************
 * Parasolid 35.0 file successfully created          *
 * c:\users\bpittman\appdata\local\temp\tscug2_31900 *
 *****************************************************
 *******************************************************
 * Siemens NX(2212) file successfully created          *
 * C:\Temp\demo_output\nist_ctc_05_asme1_ct5210_rd.prt *
 *******************************************************
```

The file will be output to the target location. In this case:

C:\Temp\nist_ctc_05_asme1_ct5210_rd.prt

Translator Customization

The Theorem translator 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.

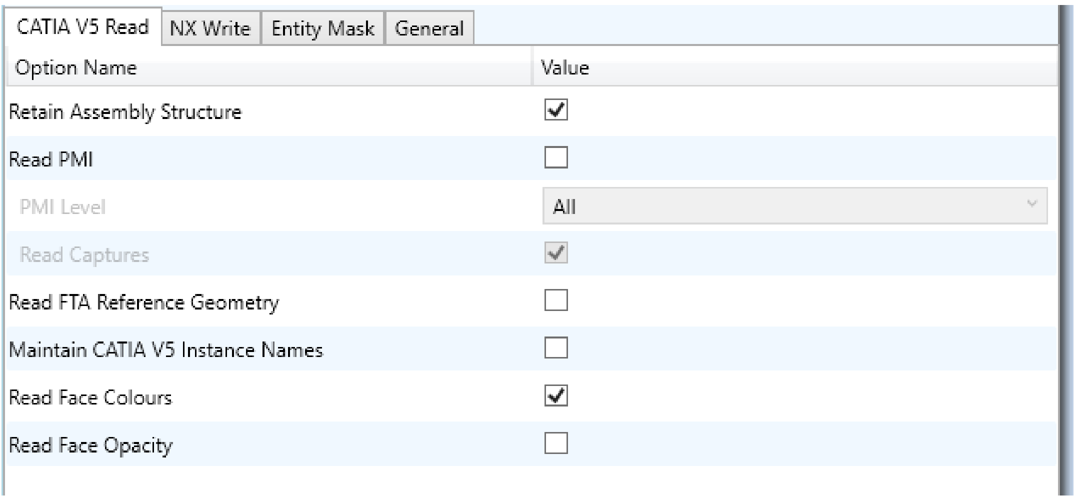
Common Options for CATIA V5 to NX

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CATIA V5 data into NX are grouped into 4 areas:

- CATIA V5 Read – Those arguments that affect how data is read from CATIA V5
- NX Write – Those arguments that affect how the data is written to NX
- Entity Mask – Those arguments that allow specific read entities to be masked
- General – Those arguments that are common to ALL Publishing activities regardless of source data

CATIA V5 Read Arguments

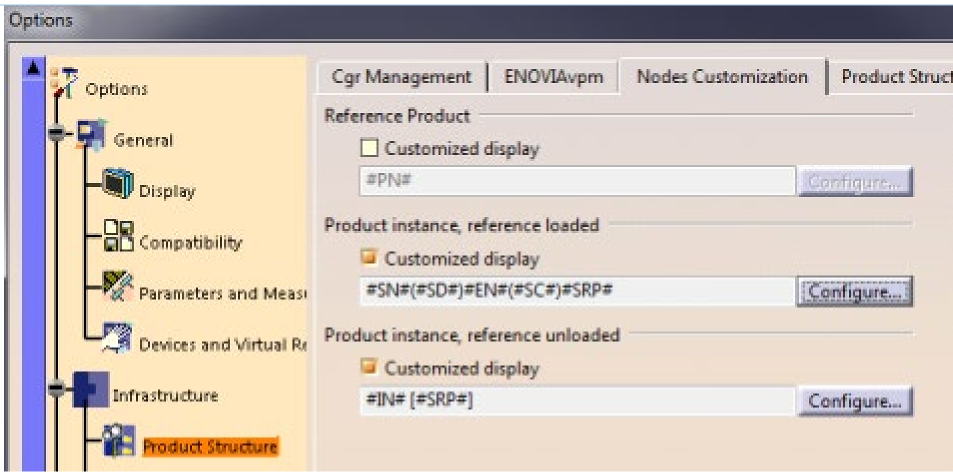
The image below shows the CATIA V5 Read arguments that are available, with their default settings:



Each of these options is described below:

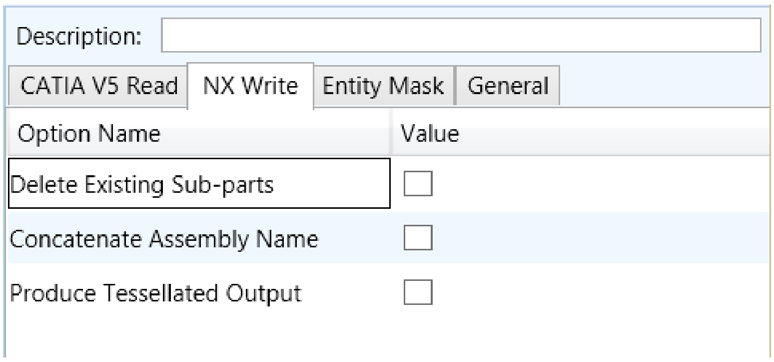
Option	Description
Retain Assembly Structure	Enables Assembly Structure to be retain *(Default is On)* Disabling this option will remove all assembly structure and collapse ALL geometry into a single selectable object ◦ Command Line Syntax:

	▪ *off_ditto – to turn off*
Read PMI	Enables PMI data read from the V5 file. *(Default is ON).* ○ Command Line Syntax: ▪ *dont_read_pmi – to turn off* ***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'*
PMI Level	A secondary argument to 'Read PMI' and allows control of the level of PMI to be read. Default is ALL when 'Read PMI' is marked as ON. • Options Available (command line syntax in italics and square brackets next to the option) ○ All - *[read_pmi]* ○ Part Level - *[read_part_pmi]* ○ Assembly Level - *[read_assy_pmi]* ○ Assembly Set (From CATPart) - *[read_part_assy_pmi]* ○ Assembly Set (All) - *[read_all_assy_pmi]*
Read Captures	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. ○ Command Line Syntax: ▪ *read_captures* ▪ *dont_read_captures – to turn off*
Read FTA Reference Geometry	Enables reading of FTA Reference Geometry *(Default is Off)* ○ Command Line Syntax: ▪ *read_geometry – to turn on*
Maintain CATIA V5 Instance Names	Honours CATIA V5 *Tools->Options->Infrastructure->Product Structure->Nodes Customization* panel settings *(Default is Off)*

	 ○ Command Line Syntax: ▪ *ditto_naming V5 – to turn on*
Read Face Colours	Processing geometry colours by face: ○ Command Line Syntax: ▪ *face_colour* ▪ *solid_colour – to complete the opposite*
Read Face Opacity	Processing face opacity: ○ Command Line Syntax: ▪ *Face_opacity – to turn on*

NX Write Arguments

The image below shows the Write NX arguments that are available, with their default settings:

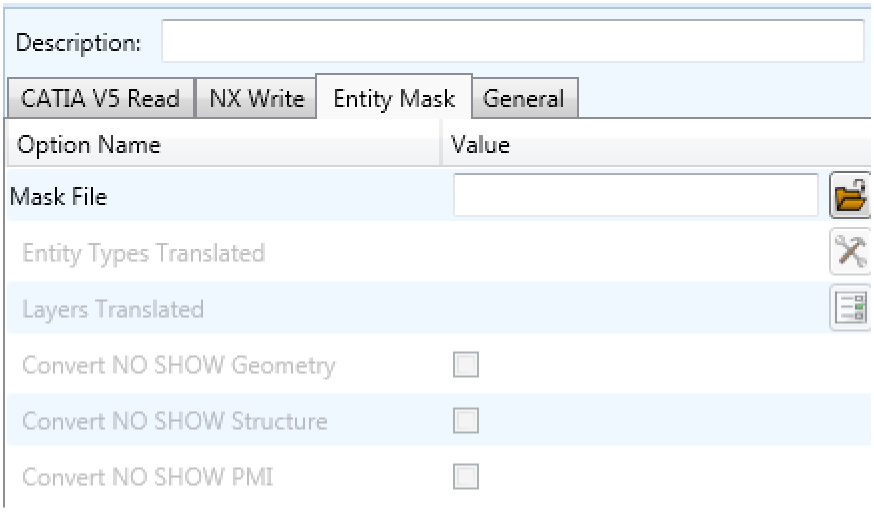


Each of these options is described below:

Option	Description
Delete Existing Sub-parts	Delete existing sub-parts. Default is OFF. ○ Command Line Syntax ▪ *delete_parts*
Concatenate Assembly Name	Concatenates assembly name. Default is OFF. ○ Command Line Syntax ▪ *concat_assy*
Produce Tessellated Output	Produce Tessellated NX file. Default is OFF. ○ Command Line Syntax ▪ *tess_output – to turn on*

CATIA V5 to NX Entity Masking Arguments

The image below shows the Masking arguments that are available, with their default settings:



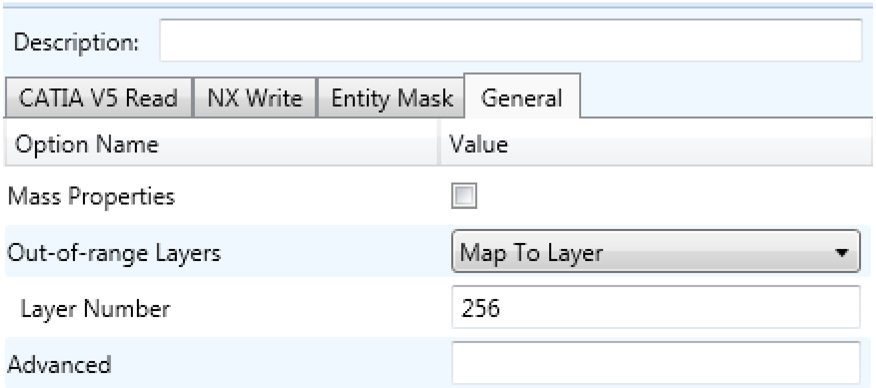
Each of these options is described below:

Option	Description
Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: ○ Command Line Syntax: ▪ *Mask <filename>*
Entity Types Translated	Specifies a selection list from which to select which entity types are to be processed. The following types are available: "POI","CUR",”SKI”,"SOL",”ISO”,"TEX","AXI" ○ Command Line Syntax: ▪ *Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g.:* ***ON POI*** *to ensure they are considered in the translation*
Layers Translated	Specifies a selection list from which to select which layers are to be processed. ○ Command Line Syntax:

	▪ *A single entry of* ***ON ALL LAY*** *Must precede any Layer Mask command.* ▪ *Add a list or range of numbers representing layer to be processed to the specified mask file to ensure they are NOT considered in the translation* *e.g.:* ***OFF LAY 114,149,166,167,168***
Convert NO SHOW Geometry	Enables Hidden geometry to be processed *(Default = Off)* ○ Command Line Syntax: ▪ *Add the following entry to the Mask file* ***ON NOSHOW***
Convert NO SHOW Structure	Enables Hidden Assembly Structure to be processed *(Default = Off)* ○ Command Line Syntax: ▪ *Add the following entry to the Mask file* ***ON NOSHOW STR***
Convert NO SHOW PMI	Enables Hidden PMI to be processed *(Default = Off)* ○ Command Line Syntax: ▪ *Add the following entry to the Mask file* ***ON NOSHOW PMI***

CATIA V5 to NX General Arguments

The image below shows the General arguments that are available, with their default settings:



Each of these options is described below:

Option	Description
Mass Properties	Allows Mass Property information to be read from the source data and written as attributes to the PDF document. Default is OFF. ○ Command Line Syntax: ▪ *mprops*
Out-of-range Layers	How to handle layers from the input system that are out-of-range in the output system. Default is 'Map To Layer' ○ Command Line Syntax: ▪ *Map To Layer: Default* ▪ *Layer Modulus (Cycle): cycle_layer*
Layer Number	A secondary option used with Out-of-range Layers when 'Map To Layer' is selected. Allows the layer number to be specified. Default is 256. ○ Command Line Syntax: ▪ *base_layer 256*
Advanced	Allows any of the Command Line Advanced arguments documented below to be passed to the Unified Interface invocation

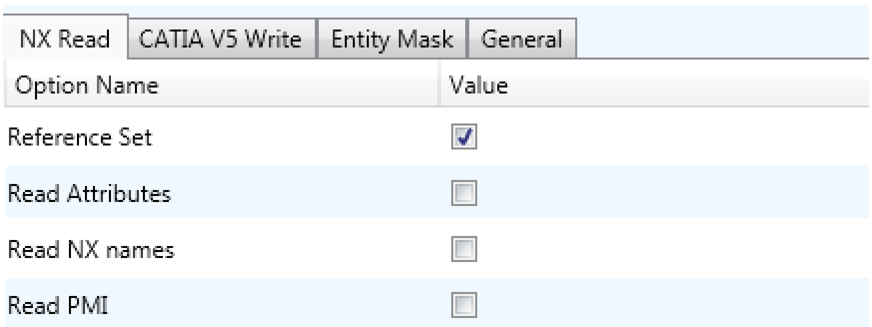
Common Options for NX to CATIA V5

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CATIA V5 data into NX are grouped into 4 areas:

- NX Read – Those arguments that affect how data is read from CATIA V5
- CATIA V5 Write – Those arguments that affect how the data is written to NX
- Entity Mask – Those arguments that allow specific read entities to be masked
- General – Those arguments that are common to ALL Publishing activities regardless of source data

NX Read Arguments

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



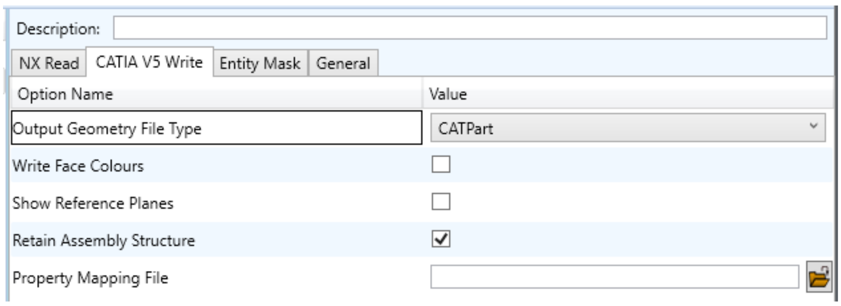
Each of these options is described below.

Option	Description
Reference Set	Enabled reference set processing. Default is OFF ○ Command Line Syntax: ▪ *No_ref_set – to turn off*
Read NX Attributes	Read NX detail user attributes. Default is OFF. ○ Command Line Syntax: *read_attrs*

Read NX names	Read NX entity names, if they exist. Default is OFF. ○ Command Line Syntax: ▪ *no_read_name – default* ▪ *read_name – to turn on*
Read PMI	Read 3D PMI. Default is OFF. ○ Command Line Syntax: ▪ *read_pmi write_stroked_pmi*

CATIA V5 Write Arguments

The image below shows the CATIA V5 Write arguments that are available, with their default settings:



Each of these options is described below:

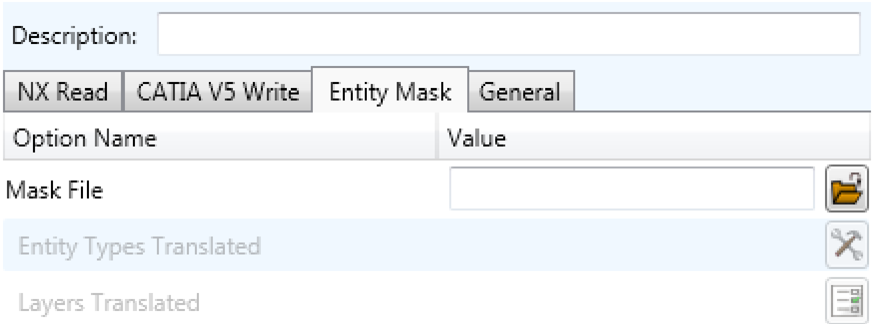
Option	Description
Output Geometry File Type	Output Geometry file type. Default is CATPart. ○ Command Line Syntax ▪ *CATPart: output_type CATPart* ▪ *Model: output_type model* ▪ *Cgr: output_type cgr* ▪ *Igs: output_type igs* ▪ *CATShape: output_type CATShape*

	▪ *Tessellated: create_cgr*
Write Face Colours	Writes face colours. Default is ON. ○ Command Line Syntax ▪ *FACE_COLOUR* ▪ *SOLID_COLOUR – to turn off*
Show Reference Planes	Creates reference planes. Default is OFF. ○ Command Line Syntax ▪ *Show: dont_blank_planes* ▪ *No Show: Default*
Retain Assembly Structure	Maintains the structure from the source file ○ Command Line Syntax ▪ *off_ditto*
Property Mapping File	This mapping file is used to control which properties are exported from V5 to NX ○ Command Line Syntax ▪ *CAD_PROP_MAP_FILE <path_to_file>* Detail on map file structure below: Line Format:- SourceName, TargetName, Control, Dummy, Dummy, Dummy Control:- 0 - Do not convert 1 - Use the source values as given By 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 in the output file. This is performed by switching the name between the input name = field 1 and the output name = field 2 Example settings shown with "#" comment for each line _ActivateBOM,NULL,0,,, _LastModifier,NULL,0,,,

	_Maturity,NULL,0,,, _PrdVersion,NULL,0,,,

NX to CATIA V5 Entity Masking Arguments

The image below shows the Masking arguments that are available, with their default settings:



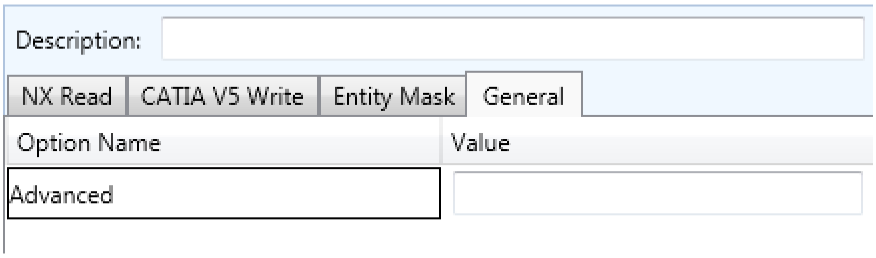
Each of these options is described below:

Option	Description
Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: ○ Command Line Syntax: ▪ *Mask <filename>*
Entity Types Translated	Specifies a selection list from which to select which entity types are to be processed. The following types are available: "POI","LIN","ARC","CON","CUR","SUR","FAC","SOL" ○ Command Line Syntax: ▪ *Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g.:* ***ON POI*** *to ensure they are considered in the translation*
Layers Translated	Specifies a selection list from which to select which layers are to be processed. ○ Command Line Syntax:

	▪ *A single entry of **ON ALL LAY** Must precede any Layer Mask command.* ▪ *Add a list or range of numbers representing layer to be processed to the specified mask file to ensure they are NOT considered in the translation* *e.g.:* ***OFF LAY 114,149,166,167,168***

NX to CATIA V5 General Arguments

The image below shows the General arguments that are available, with their default settings:



The option is described below:

Option	Description
Advanced	Allows any of the Command Line Advanced arguments to be passed to the Unified Interface invocation

Command Line Advanced Arguments

Any of the Advanced arguments can be added to the Command Line Invocation or to the General->Advanced field when run from within the User Interface.

NX Advanced Arguments

Argument	Description
facet_solid_read ***create_facets***	Options to support the reading of scanned data

CATIA V5 Advanced Arguments

Argument	Description
convert_curves	Converts curves to NURBS form
convert_surfaces	Converts surfaces to NURBS form
dont_fill_pmi_arrows	Disables the read of filled arrow information (default for *'read_pmi'*)
dont_fill_pmi_text	Disables the read of filled text information (default for *'read_pmi'*)
dont_read_captures	Disables the read of PMI Capture information (default for *'read_pmi'*)
face_opacity	Read face opacity
no_face_colour	Sets the default to SOLID colours
noshow **noshow_geom** **noshow_struct** **noshow_pmi**	Reads hidden geometry / structure / pmi
output_mbd	Allows sub-part specification tree information to be read and presented to 3D PDF as product structure information. This option also enables the *'part_level_views'* and *'part_level_pmi'* 3D PDF options
read_geometry_edges	CATIA V5 has a display mode that allows the display of shaded surfaces and edges. This option allows the translator to mimic this for FTA construction geometry by promoting the edge curves to standalone wireframe.

SEPARATE_FSOLS_ON	Support to process NX data that contains both solid and tessellated bodies (convergent model) to CATIA V5. The solid data is created as a CATPart and the tessellated data in a CGR file then an assembly file is created to hold them together.

CATIA V5 – NX PDF Add On Products

As an optional feature, the creation of 3D PDF documents can be added to the functionality of the CATIA V5 – NX license.

This requires an additional software download and is documented within that download. Please contact sales@theorem.com for more information.

Translating Interactively from within CATIA V5

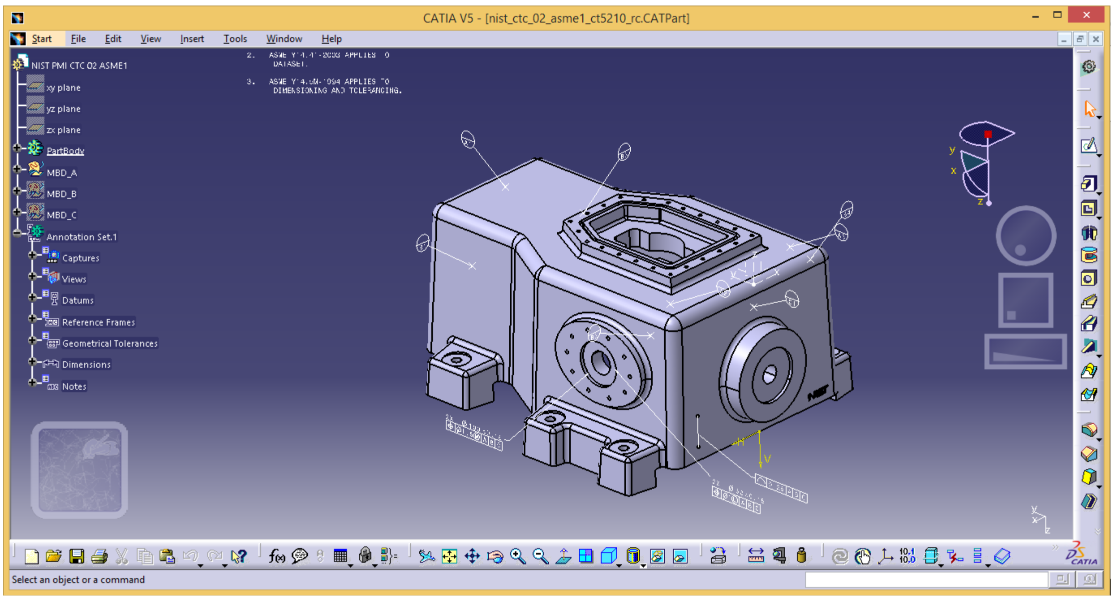
The CATIA V5 to NX translator allows an active CATIA V5 Part or Assembly to be translated directly into NX or an NX Part or assembly to be imported, directly from the CATIA V5 application.

In order to translate from within CATIA V5, the CATIA V5 application must be started from within a Theorem environment, so that the appropriate CATIA V5 menus are loaded.

CATIA V5 can be started from a shortcut, if requested at installation time. Alternatively, it can be started via the script provided in the CADverter installation at:

<installation_directory>\bin\catia5r[version]_start.cmd

(where [version] should be substituted for the version of CATIA V5 that you have installed – e.g29, 30, 31, 32 & 33):

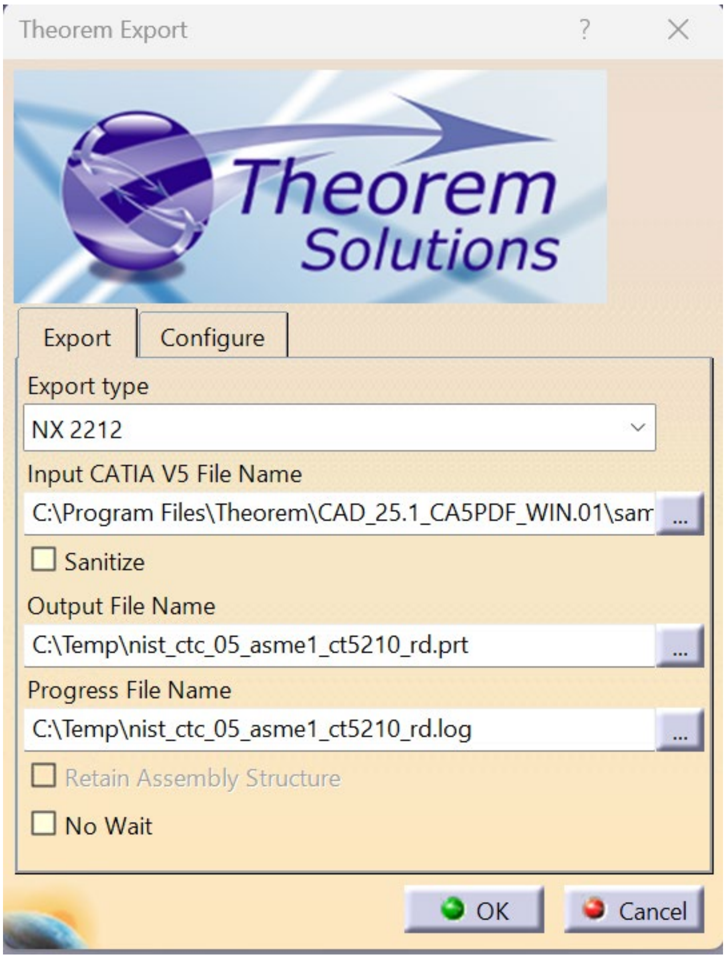


Theorem Export

Once CATIA V5 has been started and a model loaded, the active Part or Assembly can be exported to NX.

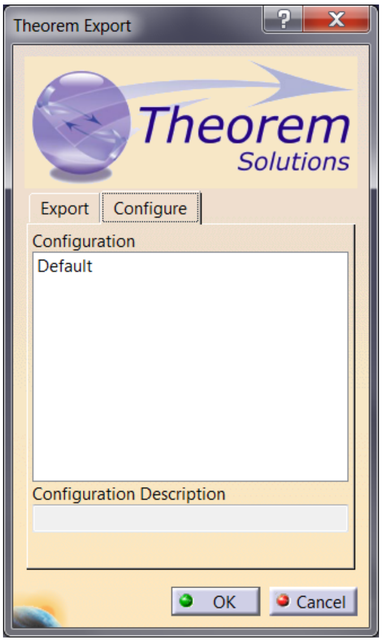
In order to export, the user selects the File -> Theorem Export Menu Option:

Which in turn launches the Theorem Export panel:



The ***Input CATIA V5 File Name***, ***Output File Name*** and ***Progress File Name*** fields will be prepopulated if a model is already loaded into the CATIA V5 session. However, these locations can all be modified prior to selecting the ***OK*** button.

The ***Configure*** tab allows a configuration file to be given to the translation containing any additional arguments specified by the user. The options seen from within this configure panel will mirror those configurations created within the Theorem Unified Interface. Please see Page 8 of this document for details on using the Unified Interface.



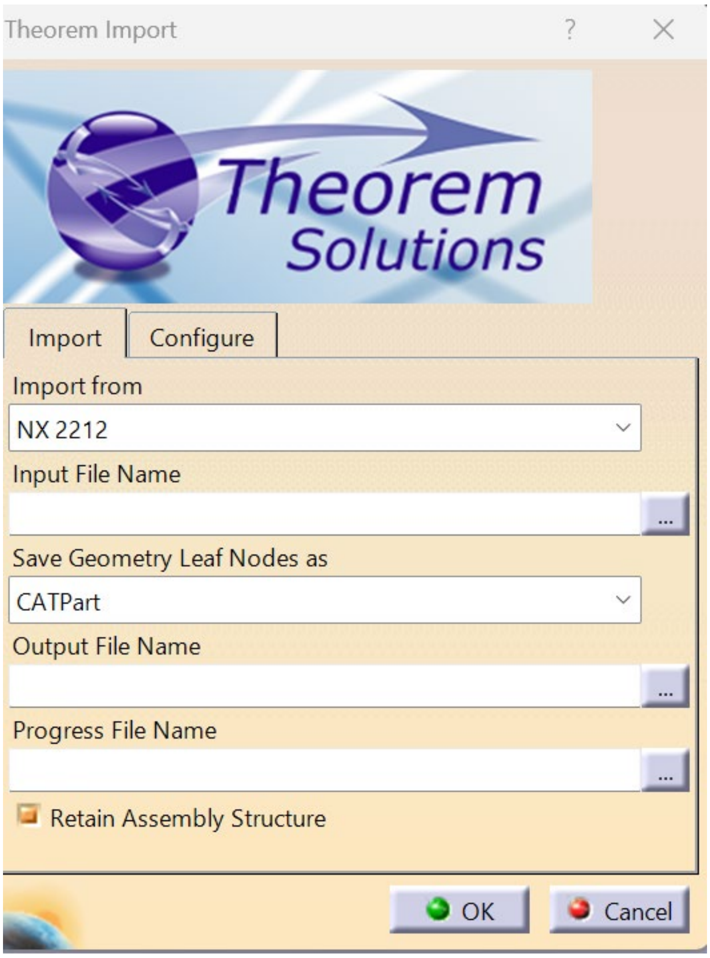
On selecting ***OK*** the on the Export Panel the active Part or Assembly will be written to NX using the into the selected output directory.

Theorem Import

Once CATIA V5 has been started, NX data can be loaded into CATIA V5.

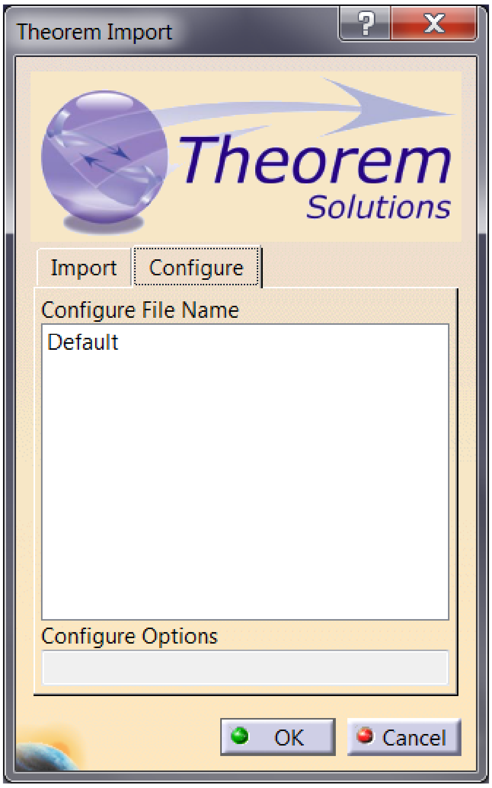
In order to import, the user selects the File -> Theorem Import Menu Option:

Which in turn launches the Theorem Import panel:



The ***Import From*** option allows the user to select the version of NX to import from.
The ***Input File Name*** option allows the selection of a NX .prt file. The ***Output File Name*** and ***Progress File Name*** fields will be populated when a model is selected and will, by default, save the associated files in the same directory as the NX part. ***Import Geometry Format*** allows the choice of CATPart or CGR for the translation output.

The ***Configure*** tab allows a configuration file to be given to the translation containing any additional arguments specified by the user. The options seen from within this configure panel will mirror those configurations created within the Theorem Unified Interface. Please see Page 8 of this document for details on using the Unified Interface.



On selecting ***OK*** the on the Import Panel the NX data will be imported into CATIA V5 and the data saved using the input into the selected output directory.

Translating Interactively from within NX

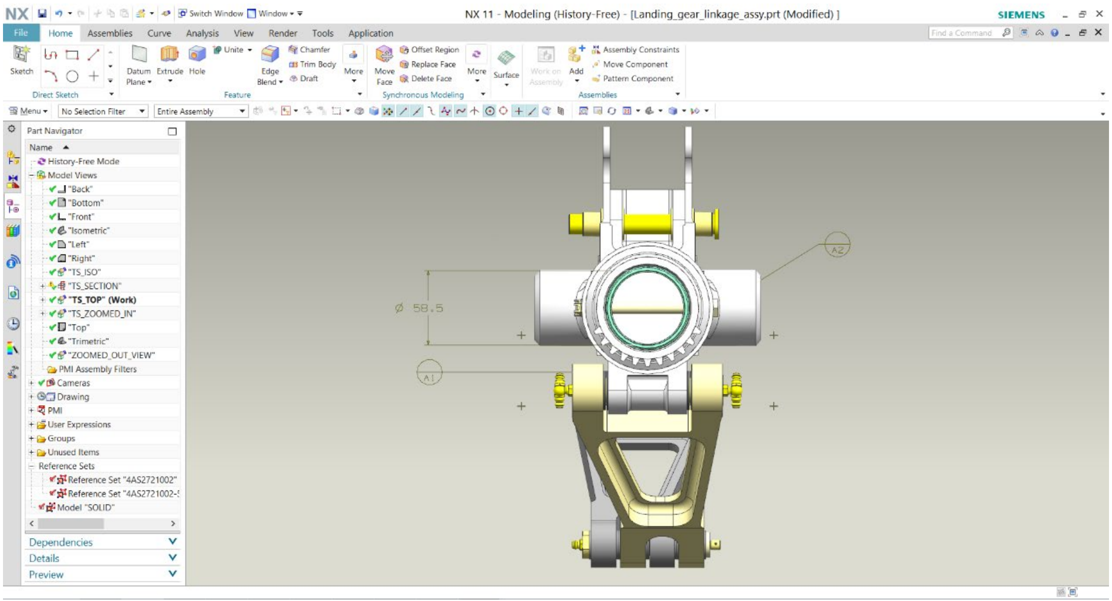
The NX to CATIA V5 translator allows an active NX Part or Assembly to be translated directly into CATIA V5 or a CATIA V5 Part or assembly to be imported, directly from the NX application.

In order to translate from within NX, the NX application must be started from within a Theorem environment, so that the appropriate NX menus are loaded.

NX can be started from a desktop shortcut, if requested at installation time. Alternatively, it can be started via the script provided in the CADverter installation at:

<installation_directory>\bin\ RunNX[version].cmd

(where [version] should be substituted for the version of CATIA V5 that you have installed – e.g. 2007, 2206 & 2212):

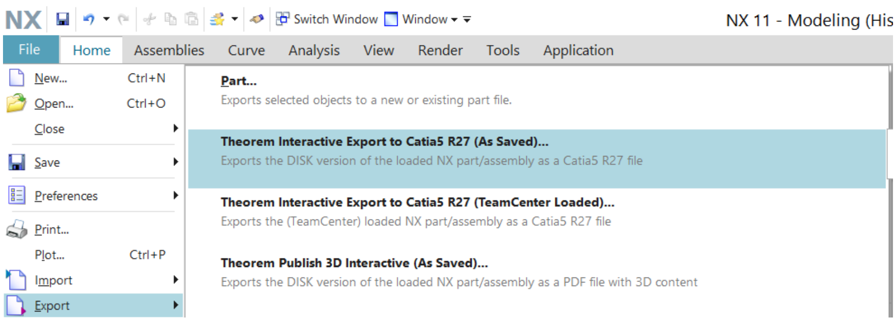


Theorem Export

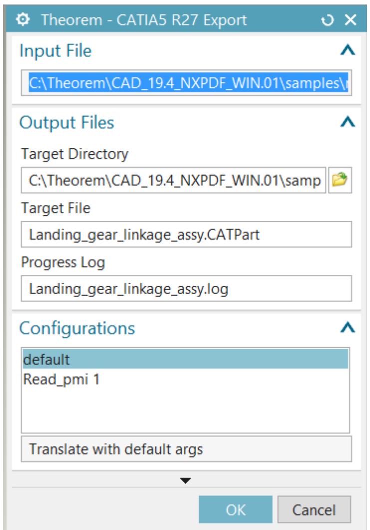
Once NX has been started and a model loaded, the active Part or Assembly can be exported to CATIA V5.

In order to export, the user selects the File -> Export Menu. There are then 2 Export options:

- Theorem Interactive Export to CATIA RXX (As Saved)
    - This is for use with data saved on a file system
- Theorem Interactive Export to CATIA RXX (TeamCenter Loaded)
    - This is for use with data loaded from TeamCenter



Once an option is selected the Theorem Export panel opens:



The ***Input File***, ***Output File > Target Directory*** and ***Progress Log*** fields will be prepopulated if a model is already loaded into the NX session. However, these locations can all be modified prior to selecting the ***OK*** button.

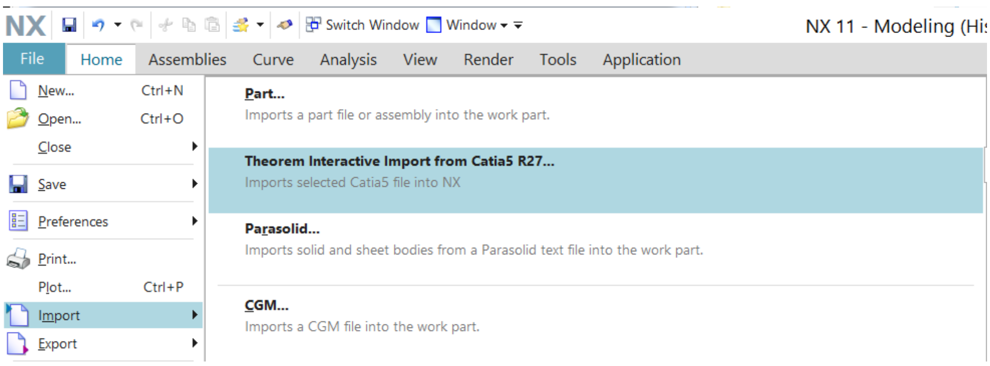
The ***Configurations*** tab allows a configuration file to be given to the translation containing any additional arguments specified by the user. The options seen from within this configure panel will mirror those configurations created within the Theorem Unified Interface. Please see Page 8 of this document for details on using the Unified Interface.

On selecting ***OK*** the on the Export Panel the active Part or Assembly will be written to CATIA V5 using the into the selected output directory.

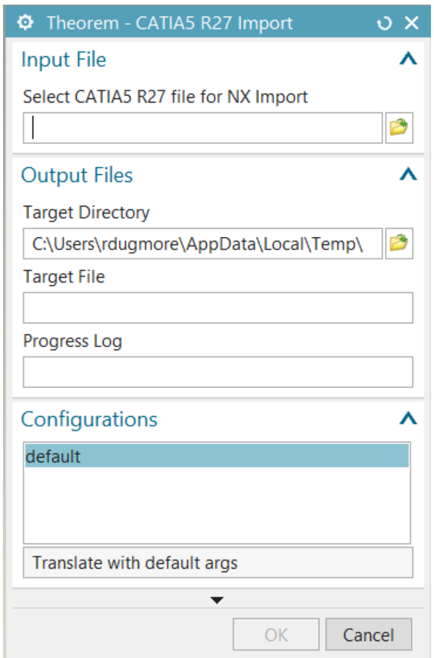
Theorem Import

Once NX has been started and a new model opened, CATIA V5 data can be loaded into NX.

In order to import, the user selects the File -> Import -> Theorem Interactive Import from CATIA5 RXX Menu Option:



Which in turn launches the Theorem Import panel:



The ***Select CATIA5 RXX file for NX Import*** option allows the selection of a CATIA V5 part. The ***Target File*** and ***Progress Log*** fields will be populated when a model is selected and will, by default, save the associated files in the same directory as the V5 part.

The ***Configurations*** tab allows a configuration file to be given to the translation containing any additional arguments specified by the user. The options seen from within this configure panel will mirror those configurations created within the Theorem Unified Interface. Please see Page 8 of this document for details on using the Unified Interface.

On selecting ***OK*** the on the Import Panel the NX data will be imported into CATIA V5 and the data saved using the input into the selected output directory.

Appendix A – CATIA V5 Configuration

Introduction

This Appendix details how to define and configure the CATIA V5 and Theorem environment to work together.

Conventions

Release of CATIA V5

To indicate a release of CATIA V5 the notation <XX> shall be used. This needs to be replaced with the specific release to be used i.e. 29, 30, 31, 32 & 33

Platform specific directory

Within the installation directory of CATIA V5 there is a platform specific directory i.e. win_b64. This directory shall be referred to as ***<OSDS>*** in this Appendix.

Theorem Installation directory

The Theorem translator installation directory is set at installation time in the translator ***ts_env.bat*** file. This directory shall be noted as ***<%TS_INST%>*** in this Appendix.

CATIA V5 Installation Directory

Upon installation of a CATIA V5 product the user will be asked to specify the installation directory. This is the directory which contains the platform specific <OSDS> directory.

Having selected the CATIA V5 installation directory via the browse button, the installation process will record the location of the CATIA V5 installation directory in the ts_env.bat file. This file is located in the Theorem translator installation directory. If the location of CATIA V5 subsequently changes, the translator can be guided to the changed location by modifying this file using a text editor to modify the ***ts_env.bat*** that is located in the translator installation directory.

Running CATIA V5 Translators

Before running the translator the user must run CATIA V5 interactively at least once to configure the CATIA V5 environment and license settings. This can be achieved by running the catia5r<XX>_start script as follows:

%TS_INST%\bin\catia5r<XX>_start.cmd

Once CATIA has been run the Translator can run as described in the relevant product User Guide.

CATIA V5 Environment DIRENV & ENV

The default location for CATIA V5 to store its global environment files is in the global directory:

Windows XP:

C:\Documents and Settings\All Users\Application Data\DassaultSystemes\CATEnv

Windows 7 & 8:

C:\ProgramData\DassaultSystemes\CATEnv

Or

%APPDATA%\CATEnv

You can find this location by running:

%CATIAV5_INST%\<OSDS>\code\bin\setcatenv -h

The environment files are named in the form ***CATIA.V5RN.B<XX>.txt***

If when installing CATIA V5 the default environment file location was replaced with another location then this location needs to be indicated to the CADverter by defining in the ***ts_env.bat*** the environment variable CATIAV5_DIRENV:

set CATIAV5_DIRENV=/some/directory

If the Theorem installation is needed to support multiple releases of CATIA. Then the user can define release specific locations using:

set CATIAV5R<XX>_DIRENV=/some/directory

The Theorem translator will attempts to create its own environment file called ***TheoremCatia5R<XX>.txt*****.** The user must therefore have write permission to the CATEnv directory. If this is not possible an existing environment file can be specified using the variable **CATIAV5_ENV**. e.g.

set CATIAV5_ENV=CATIA.V5R27.B27

Note. the extension ***.txt*** is not required. The user can specify a release specific name using ***CATIAV5R<XX>_ENV*** e.g.

set CATIAV5R19_ENV=CATIA.V5R27.B27

Checking the CATIA V5 Environment

A script is provided to check that the CATIA V5 environment is set up correctly. In a command window run the command script:

%TS_INST%\bin\checkcatia5r<XX>env.cmd

Checking the Theorem Shared Library

A script is provided to ensure that the CATIA V5 environment is compatible with the Theorem shared library. In a command window run the command script:

%TS_INST%\bin\checkcatia5r<XX>cadverter.cmd

A successful output is an indication that the location for CATIA V5 has been specified to the Theorem translator correctly and that the correct version of the Theorem CATIA V5 translator products have been installed.

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