# User Guide
# CATIAV5<>NX

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# Contents

# 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, 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:
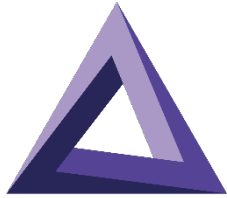
### CAD**Translate**

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

See our website for more detail.

### CAD**Publish**
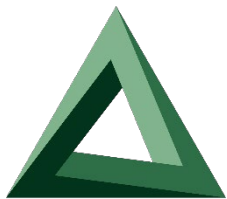
The creation of documents enriched with 3D content

See our website for more detail.

### Theorem**XR**

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

See our website for more detail.

## Primary Product Features

- CADverter converts all geometry

- If assembly data (product structure) is in the file, the assembly structure will be mapped between the two systems as well as colour information

- The user can filter data to optimize the process

- Command line interface allows process integration

## The CATIA V5 Bi-directional NX Translator

Theorem's CADverter product for CATIA V5 to NX is a direct database converter between Dassault Systemes CATIA V5 and Siemens NX. It enables the user to convert all forms of mechanical design geometry, as well as assembly and attribute information, between these two systems.

The CATIA V5-NX CADverter product can be purchased as a uni-directional, CATIA V5 to Siemens NX, or Siemens NX to CATIA V5 product, or as a bi-directional product. It can be used interactively or in a batch mode, from a standard GUI Interface, offering combined viewing, data filtering and translation capabilities.

The CADverter directly accesses native CATIA V5 parts, assemblies and drawing files using the Dassault Systemes supported programming interface. Assembly structure details and geometry colour information is retained during translation.

The Adapter may be installed on a number of machines each accessing a central network-floating license.

## Primary Product Features

- Converts all types of geometry, wire frame, surfaces, trimmed surfaces (faces) and solid models

- Converts assembly structure between the systems

- Converts attribute data including colour and layer information

- The conversion process can be run Interactively or in Batch mode

- Data can be filtered by layer and entity type

- Geometry can be filtered and selectively processed

## Primary Product benefits?

- Direct conversion between CATIA V5 and NX reduces processing time, simplifies integration and retains accuracy of the model

- The integrated viewing capability enables visually verification, pre and post translation

- The integrated data filtering options allows selected data ONLY to be processed, enabling optimisation of translations and time savings

- By converting all forms of geometry no data is lost, eliminating the time required to recreate missing data

- 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 CADverter 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

## Running the Product

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

Prior to running the product CATIA V5 must have been started at least once to allow CATIA V5 licenses to be accessed.  All specific CATIA V5 environment configuration details are documented in **Appendix A** of this document.

There are 4 distinct ways of running the translator:

- Via the Theorem Unified Interface

    o The Unified Interface offers a Desktop Environment that allows CAD and Visualization data to be viewed pre and post translation

- Via the Command Line

    o The Command Line Interface provides a direct method of invoking the translator. It can be used via a DOS shell or called via a third party application as part of a wider process requirement.

- Interactively from within CATIA V5

    o The Interactive Interface provides a direct method of Translating CATIA V5 data to and from NX from within CATIA V5 itself.

- Interactively from within NX

    o The Interactive Interface provides a direct method of Translating NX data to and from CATIA V5 from within NX itself.

# Using the Product

Default Translations

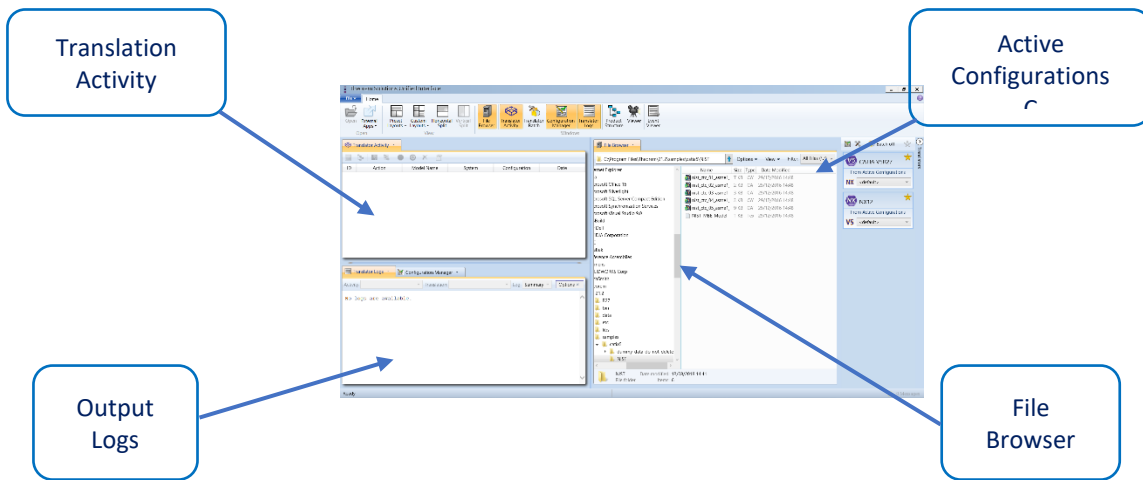*Default Translation – via the Unified Interface*

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**
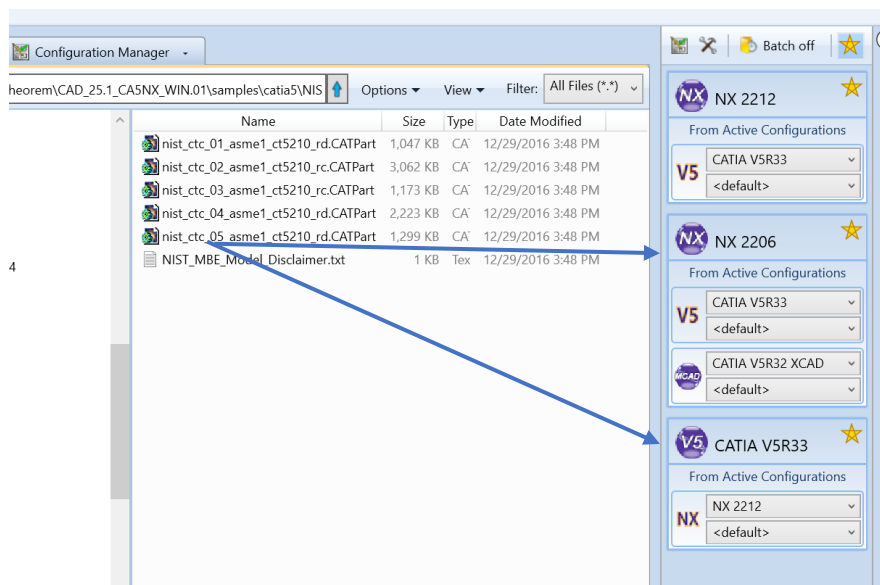
The following interface will be launched:

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



Translation Activity

Active Configurations

Output Logs

File Browser

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:
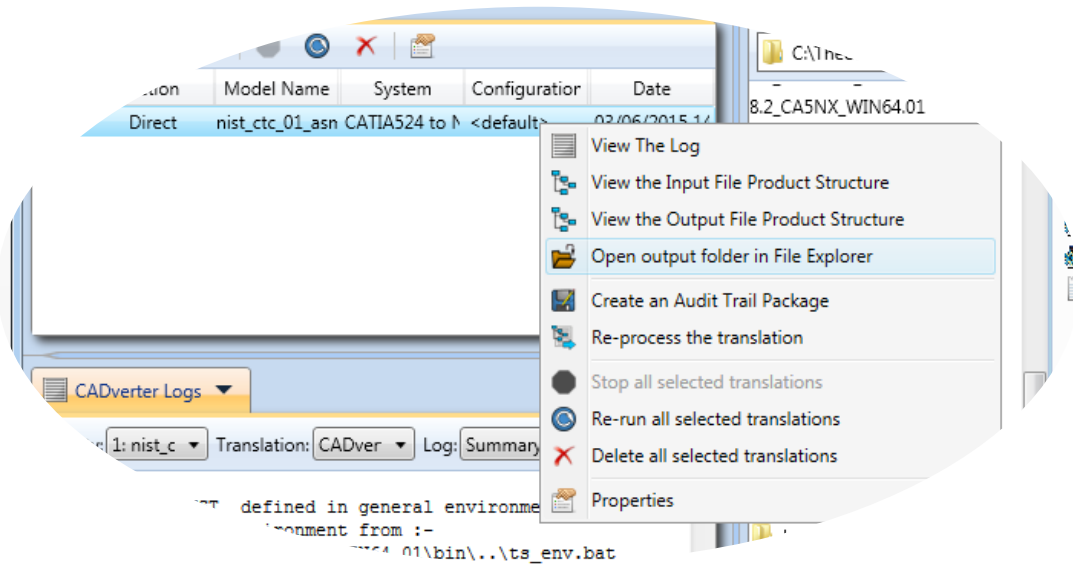
*Default Translation – via the Command Line*

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**

# CADverter Customization

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 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 |
| --- | --- |

| Option | Description |
| --- | --- |
| **Retain Assembly Structure** | Enables Assembly Structure to be retain *(Default is On)*<br>Disabling this option will remove all assembly structure and collapse ALL geometry into a single selectable object<br>     o  Command Line Syntax:<br><br>         ▪  *off_ditto – to turn off* |

| | |
|---|---|
| **Read PMI** | Enables PMI data read from the V5 file. *(Default is ON)*.<br>    o   Command Line Syntax:<br><br>         ▪  *dont_read_pmi – to turn off*<br><br><br>**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.<br>    •  Options Available (command line syntax in italics and square brackets next to the option)<br><br>        o  All - *[read_pmi]*<br><br>        o  Part Level - *[read_part_pmi]*<br><br>        o  Assembly Level - *[read_assy_pmi]*<br><br>        o  Assembly Set (From CATPart) - *[read_part_assy_pmi]*<br><br>        o  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.<br>    o   Command Line Syntax:<br><br>         ▪  *read_captures*<br><br>         ▪  *dont_read_captures – to turn off* |
| **Read FTA Reference Geometry** | Enables reading of FTA Reference Geometry *(Default is Off)*<br>    o   Command Line Syntax:<br><br>         ▪  *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)* |
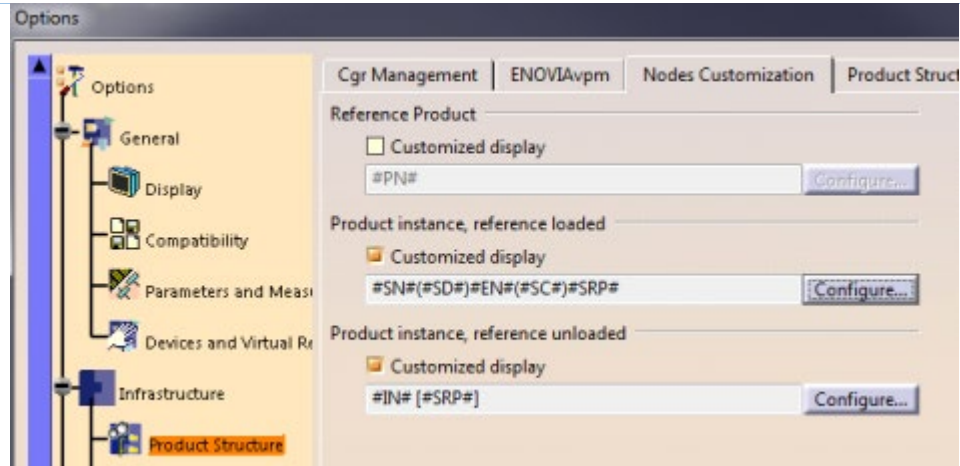
- o Command Line Syntax:

    - *ditto_naming V5 – to turn on*

| | |
|---|---|
| **Read Face Colours** | Processing geometry colours by face: <br><br> o Command Line Syntax: <br><br>     ▪ *face_colour* <br><br>     ▪ *solid_colour – to complete the opposite* |
| **Read Face Opacity** | Processing face opacity: <br><br> o Command Line Syntax: <br><br>     ▪ *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.<br><br>    o   Command Line Syntax<br><br>        ▪   *delete_parts* |
| **Concatenate Assembly Name** | Concatenates assembly name. Default is OFF.<br><br>    o   Command Line Syntax<br><br>        ▪   *concat_assy* |
| **Produce Tessellated Output** | Produce Tessellated NX file. Default is OFF.<br><br>    o   Command Line Syntax<br><br>        ▪   *tess_output – to turn on* |

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CATIA V5 to NX Entity Masking Arguments

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



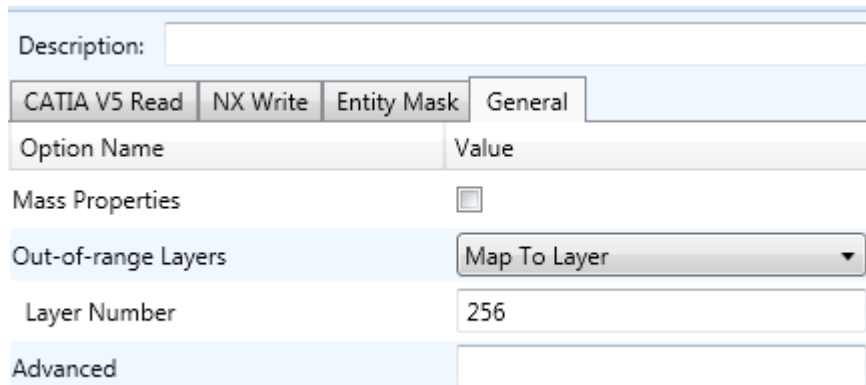Each of these options is described below:

| 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:<br>   o   Command Line Syntax:<br><br>     ▪   *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"<br>   o   Command Line Syntax:<br><br>     ▪   *Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g.:*<br><br>       ***ON POI***<br>       *to ensure they are considered in the translation* |
| **Layers Translated** | Specifies a selection list from which to select which layers are to be processed.<br>   o   Command Line Syntax:<br><br>     ▪   *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*<br><br>*e.g.:*<br>**OFF LAY 114,149,166,167,168** |
| **Convert NO SHOW Geometry** | Enables Hidden geometry to be processed *(Default = Off)* | |
| | o   Command Line Syntax: | |
| | ▪ *Add the following entry to the Mask file*<br><br>**ON NOSHOW** | |
| **Convert NO SHOW Structure** | Enables Hidden Assembly Structure to be processed *(Default = Off)* | |
| | o   Command Line Syntax: | |
| | ▪ *Add the following entry to the Mask file*<br><br>**ON NOSHOW STR** | |
| **Convert NO SHOW PMI** | Enables Hidden PMI to be processed *(Default = Off)* | |
| | o   Command Line Syntax: | |
| | ▪ *Add the following entry to the Mask file*<br><br>**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.<br>    o   Command Line Syntax:<br><br>        ▪   *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'<br>    o   Command Line Syntax:<br><br>        ▪   *Map To Layer: Default*<br><br>        ▪   *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.<br>    o   Command Line Syntax:<br><br>        ▪   *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<br>  o Command Line Syntax:<br><br>    ▪ *No_ref_set – to turn off* |
| **Read NX Attributes** | Read NX detail user attributes. Default is OFF.<br>  o Command Line Syntax:<br><br>  *read_attrs* |
| **Read NX names** | Read NX entity names, if they exist. Default is OFF.<br>  o Command Line Syntax:<br><br>    ▪ *no_read_name – default*<br><br>    ▪ *read_name – to turn on* |

| | |
|---|---|
| **Read PMI** | Read 3D PMI. Default is OFF. |
| |      o    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.<br><br>     o    Command Line Syntax<br><br>         ▪    *CATPart: output_type CATPart*<br><br>         ▪    *Model: output_type model*<br><br>         ▪    *Cgr: output_type cgr*<br><br>         ▪    *Igs: output_type igs*<br><br>         ▪    *CATShape: output_type CATShape*<br><br>         ▪    *Tessellated: create_cgr* |
| **Write Face Colours** | Writes face colours. Default is ON.<br><br>     o    Command Line Syntax<br><br>         ▪    *FACE_COLOUR*<br><br>         ▪    *SOLID_COLOUR – to turn off* |

| Show Reference Planes | Creates reference planes. Default is OFF. |
|---|---|
| | o Command Line Syntax |
| | ▪ *Show: dont_blank_planes* |
| | ▪ *No Show: Default* |
| **Retain Assembly Structure** | Maintains the structure from the source file |
| | o Command Line Syntax |
| | ▪ *off_ditto* |
| **Property Mapping File** | This mapping file is used to control which properties are exported from V5 to NX |
| | o Command Line Syntax |
| | ▪ *CAD_PROP_MAP_FILE <path_to_file>* |
| | Detail on map file structure below: |
| | Line Format:- |
| | SourceName, TargetName, Control, Dummy, Dummy, Dummy <br> Control:- <br> 0 - Do not convert <br> 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,,, <br> _LastModifier,NULL,0,,, <br> _Maturity,NULL,0,,, <br> _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:<br>    o   Command Line Syntax:<br><br>        ▪   *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"<br>    o   Command Line Syntax:<br><br>        ▪   *Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g.:*<br><br>          ***ON POI***<br>          *to ensure they are considered in the translation* |
| **Layers Translated** | Specifies a selection list from which to select which layers are to be processed.<br>    o   Command Line Syntax:<br><br>        ▪   *A single entry of **ON ALL LAY** Must precede any Layer Mask command.*<br><br>        ▪   *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*<br><br>          *e.g.:*<br>          ***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):
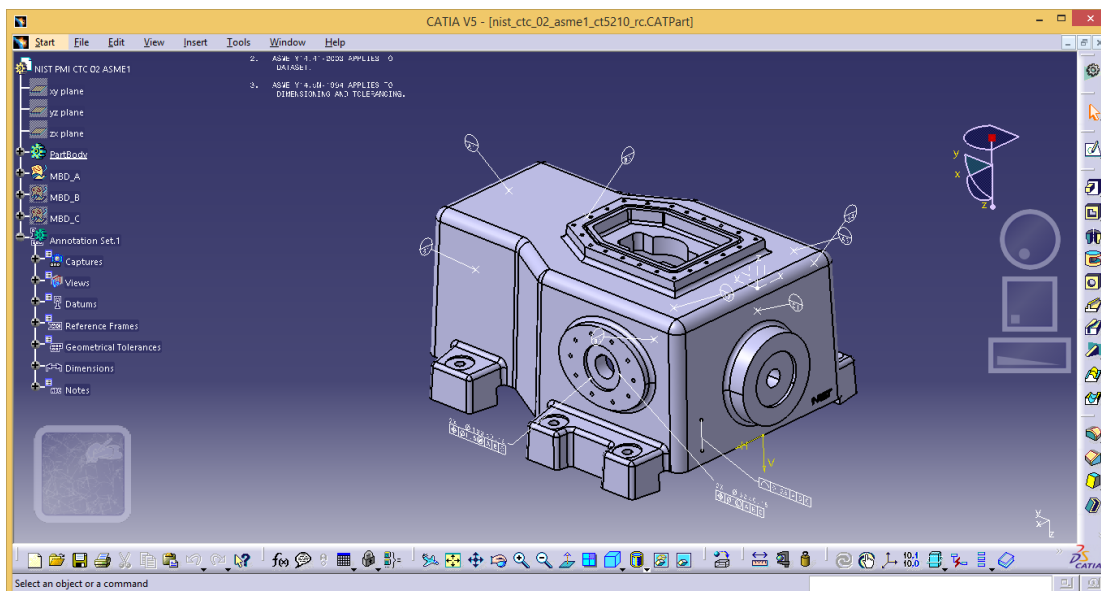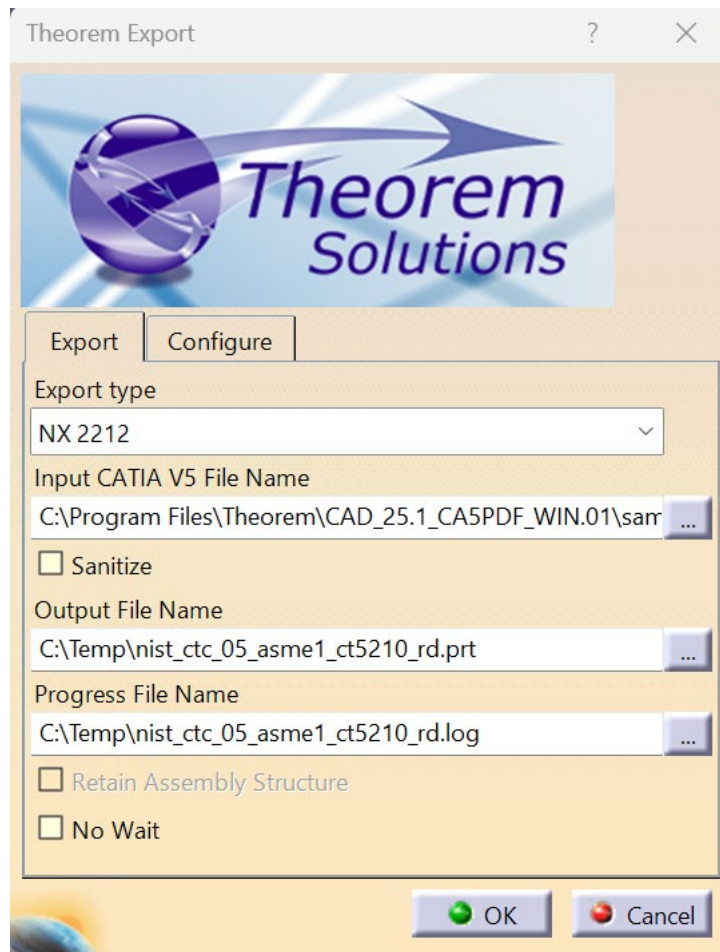


## Theorem Export

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

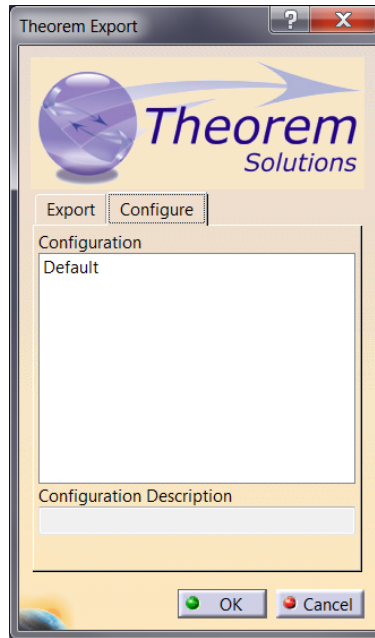In order to export, the user selects the File -> Theorem Export Menu Option:



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.

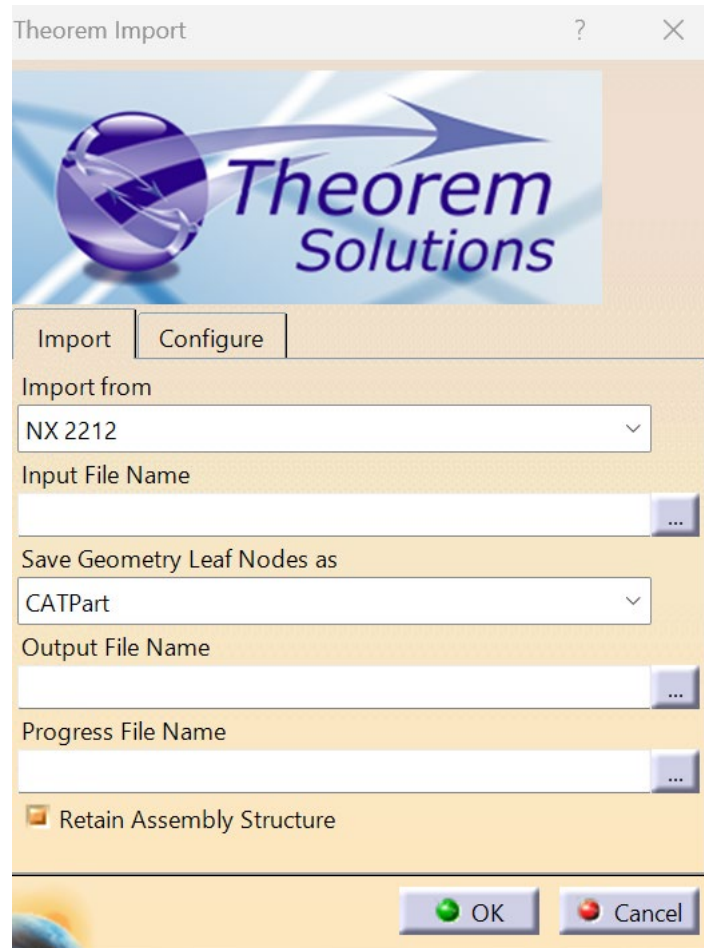## Theorem Import

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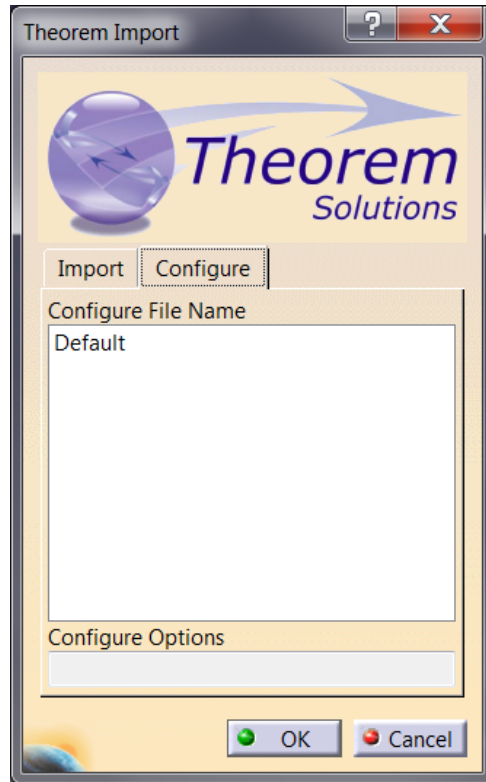
On selecting **OK** the on the Import Panel the NX data will be imported into CATIA V5 and the data saved using the input into the selected output directory.

# 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:

<p align="center"><i><b><installation_directory>\bin\ RunNX[version].cmd</b></i></p>

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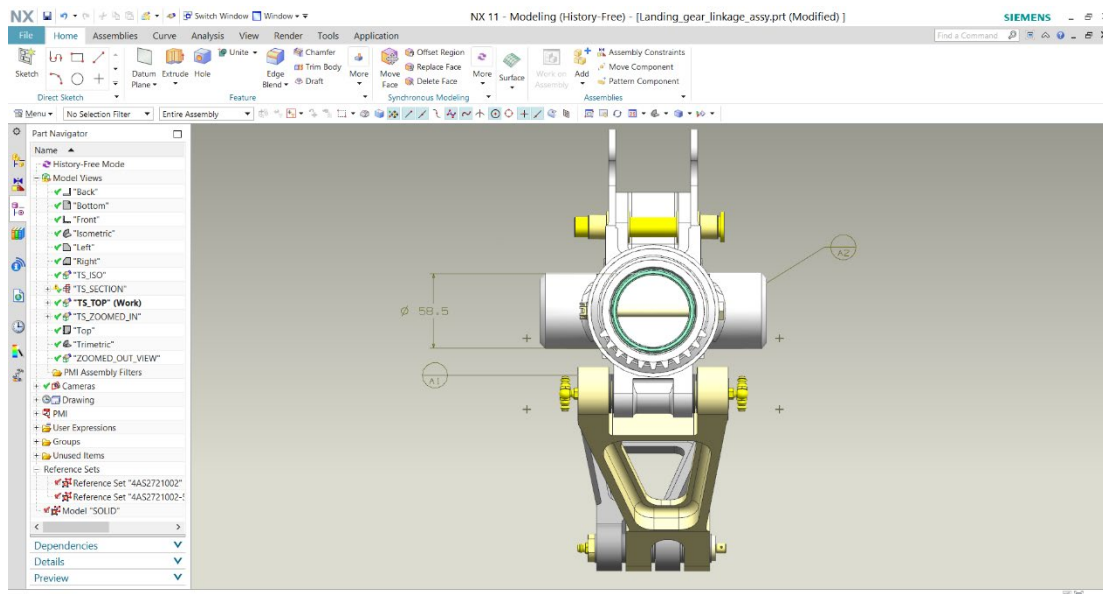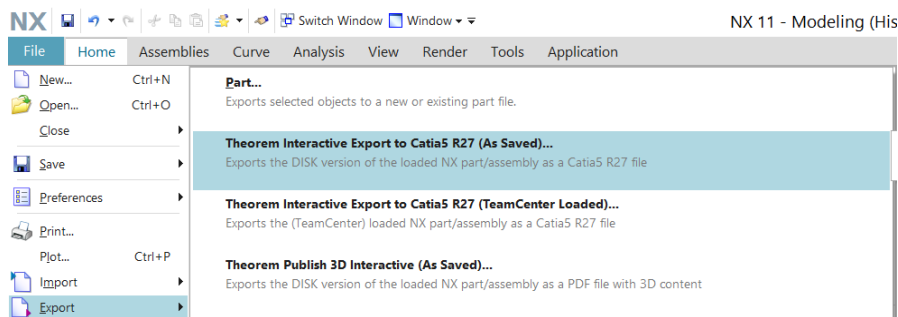
## Theorem Export

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)

  o This is for use with data saved on a file system

- Theorem Interactive Export to CATIA RXX (TeamCenter Loaded)

  o This is for use with data loaded from TeamCenter

Once an option is selected the Theorem Export panel opens:
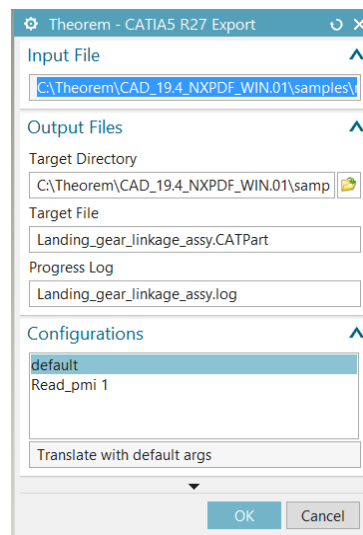
The **Input File**, **Output File > Target Directory** and **Progress Log** fields will be prepopulated if a model is already loaded into the NX session.  However, these locations can all be modified prior to selecting the **OK** button.
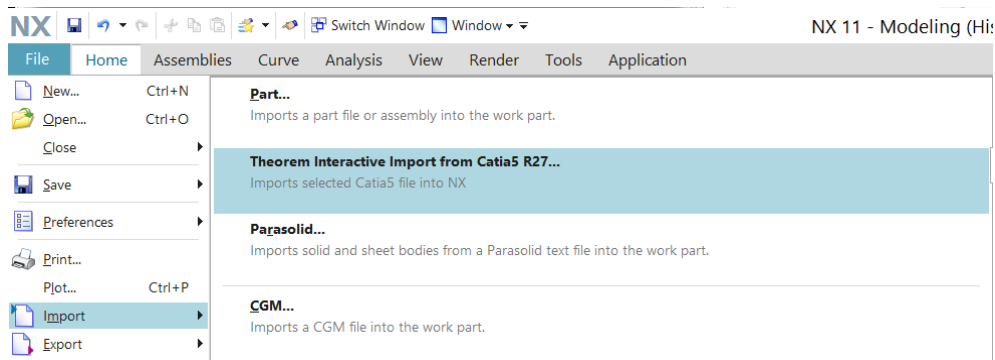
The **Configurations** tab allows a configuration file to be given to the translation containing any additional arguments specified by the user. The options seen from within this configure panel will mirror those configurations created within the Theorem Unified Interface. Please see Page 8 of this document 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.
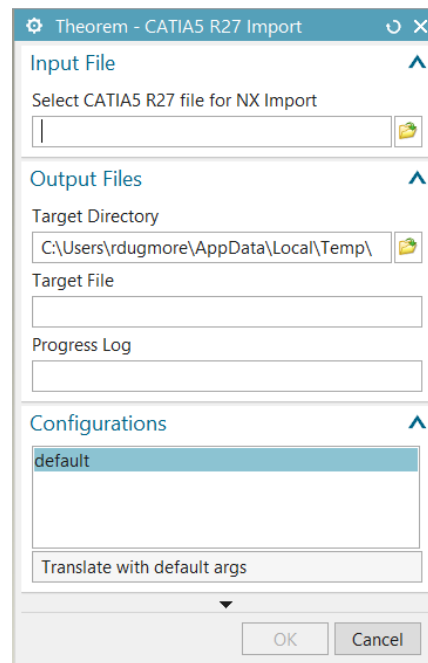
## Theorem Import

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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