



Multi-CAD for CATIA V5 – NX Product Range MC5NX

Product Release Version 25.3



USER GUIDE

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Overview of Multi-CAD

About Theorem

Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of 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.

What is Multi-CAD?

Theorem have 3 main Product brands. These are:



TRANSLATE

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

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



PUBLISH

The creation of documents enriched with 3D content

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



Theorem-XR

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

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

The CATIA V5 Multi-CAD to NX translator is part of the Translate Brand. It may be installed on a number of machines each accessing a central network-floating license.

The CATIA V5 Multi-CAD to NX Translator is a bi-directional direct database converter between the Dassault Systemes CATIA V5 Modelling Application and the 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, 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.

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 the conversion process may be operated by using the 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 installation.
- The conversion process can be run Interactively, Batch Mode or using the new Unified Interface
- Command line interface allows process integration into any workflow or automated process.
- Uses the Dassault Systems CATIA V5 Multi-CAD API and Siemens NX API to read and write the respective data formats.
- When writing CATIA V5 data the user is able to configure the derived geometry to be created in either VISU (CGR), VISU + Snap (CGR + Canonical data to aid positioning) or CATPart format 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 Multi-CAD for CATIA V5 – NX product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

Getting Started

Documentation

A copy of the latest documentation for all Theorem Solutions products can be found on our web site at:

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

For each of the Multi-CAD related products there is a specific link that provides user documentation in the form of PDF and Tutorials. For this product navigate to the Multi-CAD CATIA V5 – NX section.

Installation Media

The latest copy of each release is specified in the Product Release Document associated with each specific release. A copy of the current product release notes can be found on the Theorem Solutions web site at:

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

Simply download the Product Release Document and select the hyperlink within the document to download the installation media.

For Multi-CAD related products the installation requires 3 CD's to be installed

1. The Theorem Solution TXX Multi-CAD Platform CD
2. The Theorem Solutions Unified Interface CD
3. The Theorem Solutions Multi-CAD CATIA V5 –NX CD

Alternatively, you can request a copies of the software to be shipped on a physical CD media. Please contact your sales representative sales@theorem.com to arrange the shipment of the physical CD media.

Software and License Installation

The software installation and License configuration process is fully defined in a separate document. Please refer to the document titled “Multi-CAD_CATIAV5_Installation_Guide” for complete installation instructions

Running the Product

Once configured and licensed, the product will be ready to be used.

Prior to initially using the translator CATIA V5 must have been launched at least once to allow the selection of the CATIA V5 licenses to be recorded in the users CATSettings folder. Once the CATSettings are generated the translator will be able to be executed either internally or externally to CATIA V5 until the CATSettings are removed. Therefore if you operate a procedure that programmatically generates the users CATSettings this

requirement to maintain a reference to existing CATSettings must be considered when using the product in batch external to the CATIA V5 application.

All specific CATIA V5 environment configuration details are documented in [Appendix A](#) of this document.

Methods of Invoking the CADverter

The CATIA V5 <> NX Multi-CAD CADverters can be invoked with a variety of methods, these include;

- Interactively from within CATIA V5 using standard CATIA V5 menu options such as “Insert Existing Component”.
- From the Theorem Graphical User Interface (GUI)
- From a command prompt window using the command line. This mechanism can be used to automate the translation process as part of any automated workflow.

Interactive usage within CATIA V5

Starting CATIA V5

In order to use the Theorem CADverter products interactively within the CATIA application, you must start CATIA V5 with the correct CATEnv environment settings. A desktop icon and a start menu item are both able to be created during the installation process to achieve this.

The desktop icon is labelled “Theorem Multi-CAD NX CATIAV5Rnn” (where nn is the release version).

The Start menu item is located under the “Start + All programs + CATIA” folder and is also named “Theorem Multi-CAD NX CATIAV5Rnn”.

Using “Insert Existing Component”

The CATIA V5 Multi-CAD architecture allows you to invoke the Theorem NX to V5 Multi-CAD translator to import components or assemblies into an active CATIA V5 CATProduct by using the “Insert > Existing Component...” menu item. Also note that the CATIA associativity (Update Status Checker) mechanism can invoke the translator execution.

In order for the Theorem translator to be invoked from within CATIA V5, the setting of the “**Preferred Conversion Technology**” must be set to “**Indirect**” on the CATIA V5 **Tools > Options > Compatibility > “External Formats”** option page.

(See section - Setting Conversion Options in CATIA V5)

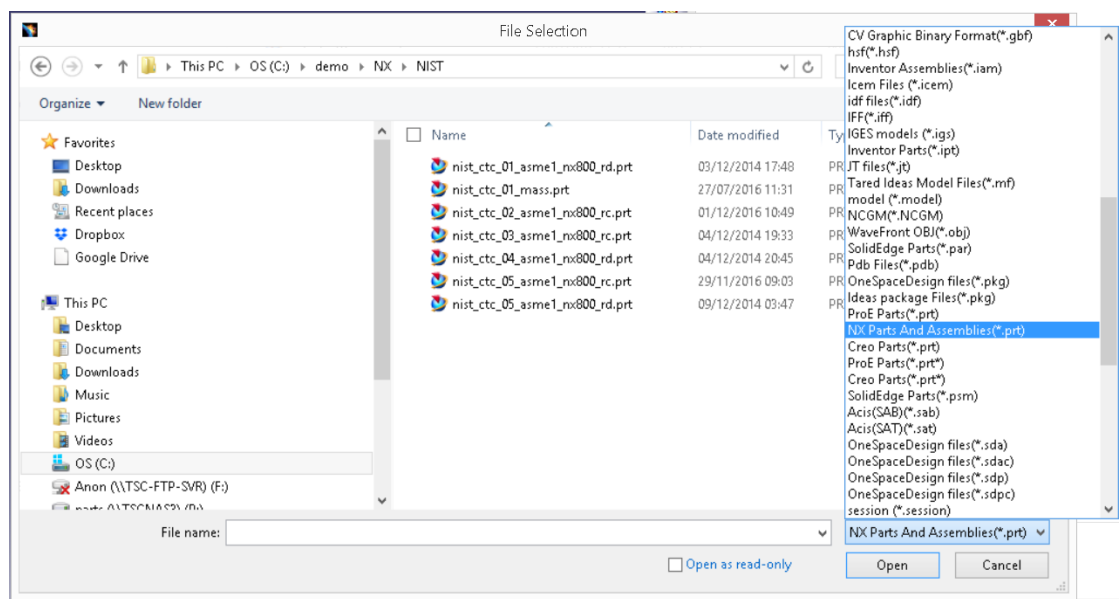
Note that this option page is only available if the TXX-THEOREM-MULTICAx GATEWAY (Theorem Partner) license is available which is provided by the Theorem license manager. Please see the installation document named “**CATIA V5 MultiCAD_Installation_Guide.pdf**” for details.

There are some extra conversion options available to the Theorem User Interface that are not presented on the CATIA V5 External Formats options page.

One method to allow you to specify additional options to the NX to V5 Multi-CAD translator is provided by a configuration file that the translator reads during the conversion process.

The configuration file is named **%TS_INST%\data\ug\ug_xcad_opts.txt** and contains information on the various options that can be specified using this method.

On invoking the **“Insert Existing Component...”** command in CATIA V5, and selecting a product node for the ‘insert’ operation, you will see the following file selector. The **“NX Files”** file type should be selected and a NX **“.prt”** file chosen.



The translation will now proceed and the CATIA V5 representation of the NX assembly/components will be incorporated in the active CATProduct.

It is worth noting here that CATIA launches and manages a conversion sub-process that feeds the option page settings into the Theorem Multi-CAD translator via a temporary file named **%CATTemp%\UGToNavConfigs.txt**. This is referenced using the **‘-c <config_file>’** command line option.

The Theorem User Interface allows you to select a CATIA generated configuration file as described above, but if none is selected, a default configuration file installed with the Theorem software under **%TS_INST%\data\ug\UGToNavConfigs.txt** will be used. For further information on the CATIA V5 Multi-CAD integration methods and commands see the CATIA Infrastructure Documentation.

The Theorem NX to V5 Multi-CAD translator is also invoked when the CATIA **“Update Status Checker”** command determines that a NX part file that was previously inserted using **“Insert Existing Component”** has been subsequently modified outside of the CATIA environment.

For further information on the usage of the CATIA commands “Insert Existing Component” and “Update Status Checker”, consult the CATIA Documentation under headings:

- Mechanical Design → Assembly Design
- Here you will find topics
 - “Insert an Existing Component”
 - “User Tasks” → “Updating an Assembly”

Visualizing Inserted Annotations

It should be noted that when you perform the “Insert Existing Component” command with a NX file containing PMI data, you must perform the following operations to visualise it if the Link Mode used is ‘Visu’ or Visu Snap’

- Activate a workbench supporting FTA operations such as with menu item “Start > Mechanical Design > Product Functional Tolerancing & Annotation”.
- Use menu “Insert > Visualization > List Annotation Set Switch On/Switch Off”
- Enable the listed Annotation Sets and apply the change.

The CATIA V5 XCAD to NX processing messages, errors and warnings are recorded in a ‘.log’ file located in the CATIA CATReport directory. The file is named after the active selected CATIA part/product name. E.g. if the part/product **My part** were selected, the log file name would be **%CATReport%\My part.log** .

A summary file is also produced which contains key information and the process completion status information. It is named after then log file with the additional suffix .summary. (e.g. **%CATReport%\My part.log.summary**).

From the command line

The Theorem User Interface provides the ability to generate conversion command lines that are appended to a nominated batch file for subsequent execution.

```
%TS_INST%\bin\cad_run.cmd <translator_name>  
-i <input_file_name>  
-o <output_file_name>  
[other options]
```

This is wrapped in a cmd.exe 'call' command to redirect output as follows:-
call <cmd> > <screen_and_error_output_file> 2>&1

E.g. here is a command line example from a batch file:-

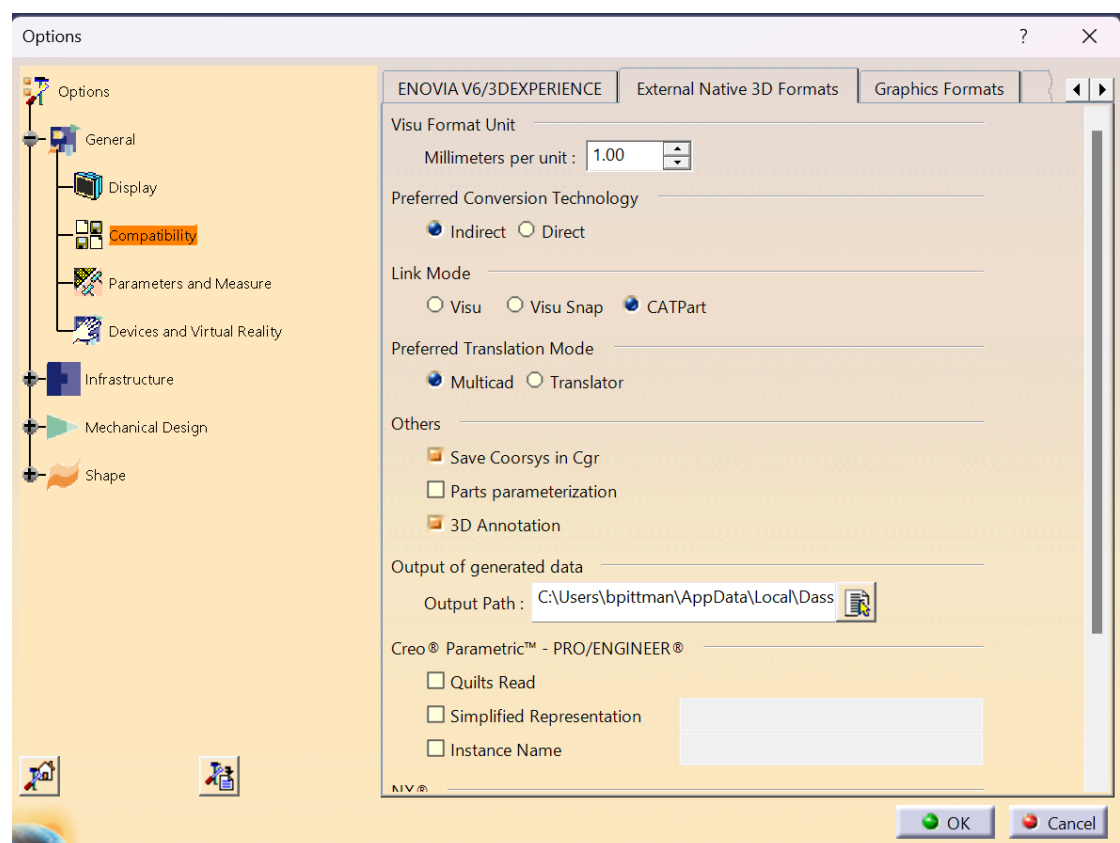
```
call "C:\Theorem\CAD_25.3_MC5NX_WIN.01\bin\cad_run.cmd"  
NX12_CATIA5R29XCAD -i C:\theorem 25.3  
testing\CAD_25.3_MC5NX_WIN.01\samples\nx\landing_gear_linkage\  
Landing_gear_linkage_assy.prt -o c:\temp\Landing_gear_linkage_assy.CATProduct  
mode_catpart progress_file c:\temp\Landing_gear_linkage_assy.log > c:\temp\  
Landing_gear_linkage_assy.log 2>&1
```

NX to CATIA V5 MultiCAD Usage

Setting Conversion Options in CATIA

If you have the TXX-THEOREM-MULTICAx license, the following categories of options will be visible from the CATIA menu - Tools > Options on the Compatibility “External Formats” page

- Visu Format Unit
- Preferred Conversion Technology
- Link Mode
- Preferred Translation Mode
- Others
- Output of generated data
- Ideas®
- ProEngineer®



Some of these settings are accessible to the Theorem CATIA V5 to NX Multi-CAD NX translator when it is run in both interactive and command line (batch) mode.

Visu Format Unit



Visu Format Unit

Millimeters per unit : 1.00

◆ Millimeters per unit

The conversion of units from NX parts is incorporated into the Theorem CADverter, so this value should always be set to 1.00

Preferred Conversion Technology



Preferred Conversion Technology

☒ Indirect ☐ Direct

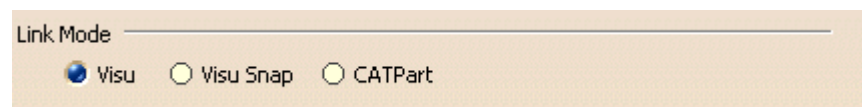
Indirect

This parameter determines that the 3rd party Theorem converters will be used in conversion operations.

Direct

This parameter determines that the Dassault Systemes converters will be used in conversion operations.

Link Mode



Link Mode

☒ Visu ☐ Visu Snap ☐ CATPart

Visu

Tessellated data. By default, this option is activated.

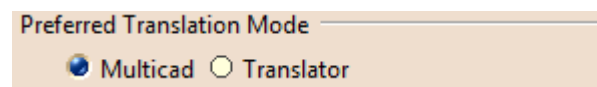
Visu Snap

Selecting this option forces the processing of NX precise BREP geometry to generate a CATIA V5 CGR output. In addition canonical shape properties are also generated in the CGR file. Consequently the CGR data resulting from importing NX will have additional geometry snapping capabilities. After selecting this option, this message appears: "Please, restart session to take modifications into account".

CATPart

This option causes the import of exact/brep geometry from the NX file to be created in CATPart form.

Preferred Translation Mode



Selecting this option controls if links will be retained between the source NX files which are imported, and the resulting CATIA files generated. In Multi-CAD mode, links are retained and modifications are locked, in Translator mode, links are not retained and modifications are allowed.

Others

Save Co-ordinate system in Cgr

This option saves the NX PMI co-ordinate system data into the derived CGR representation when Visu output is selected. Note this feature would also require the 3D Annotation setting to be selected to create the required output.

3D Annotation

This parameter determines whether 3D annotation data, referred to as “PMI” data in NX, will be imported.

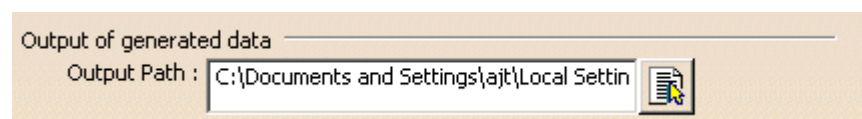
(Note, requires one of the following licenses):

CATIA: FT1.prd or FTA.prd

ENOVIA: DT1.prd

DELMIA: MTR.prd or MFT.prd

Output of generated data



Output Path

Setting the Output Path location enables you to customize the folder location that is used when writing the derived CATIA V5 generated data. It specifies the location where CGR, CATPart and CATProducts will be generated.

The NX to CATIA V5 Multi-CAD options file

There are some extra conversion options available in the Theorem CADverter GUI that are not presented to the user on the CATIA External Formats options page. A mechanism to allow you to specify additional options to the NX to V5 Multi-CAD translator is provided via a text file that the translator reads during the conversion process.

The default location of this options file is defined in the %TS_INST%\bin\TheoremProps.txt file by the setting:

Theorem.NxServerCfile=%TS_INST%\\data\\ug\\ug_xcad_opts.txt

This text file can be edited to contain any of the following command line options on a separate line:-

info

Outputs extra processing information to log files

debug

Outputs extra debugging information to log files help diagnose problems.

noprep

This option affects the reading of Parasolid data within the NX file. By default the prep data is prepared for writing to another system by steps such as removing small edges, splitting complex faces, and converting surfaces to NURBS. This option disables this preparation and uses the original parasolid definition.

rd_native_edge

By default some analytic edge definitions are converted to NURBS curves. This option forces the use of native analytic edge definitions.

show_axis_system

By default axis systems are created in CATIA hidden visibility space. This option will create them in the shown visibility space.

read_blank

By default entities which are blanked are not read. This option allows the reading of blanked entities.

ignore_std_views

By default the translator will read standard views (top, right, front etc). This option forces these views to be ignored.

The NX to V5 Progress and Log File Outputs

The NX to CATIA V5 XCAD process log and error messages are recorded in a '.log' file located in the CATIA CATReport directory. A process summary file is also produced here which contains the completion status of the conversion. These files are named after the selected input file name. E.g. if the file Landing_gear_linkage_assy.prt were selected, the log and summary file names would be %CATReport%\Landing_gear_linkage_assy.log and %CATReport%\tea.log.summary.

If the NX to CATIA V5 XCAD process is run using the CADVerter GUI or using the command line option 'progress_file' <name>, the log file output will honour this name, and the summary file will be named similarly with the suffix '.summary'.

The V5 to NX Progress and Log File Outputs

The CATIA V5 XCAD to NX process log and error messages are recorded in a '.log' file located in the CATIA CATReport directory. The file is named after the selected output file name. E.g. if the file Mypart were selected, the log file names would be %CATReport%\Mypart.log and %CATReport%\Mypart.log.summary.

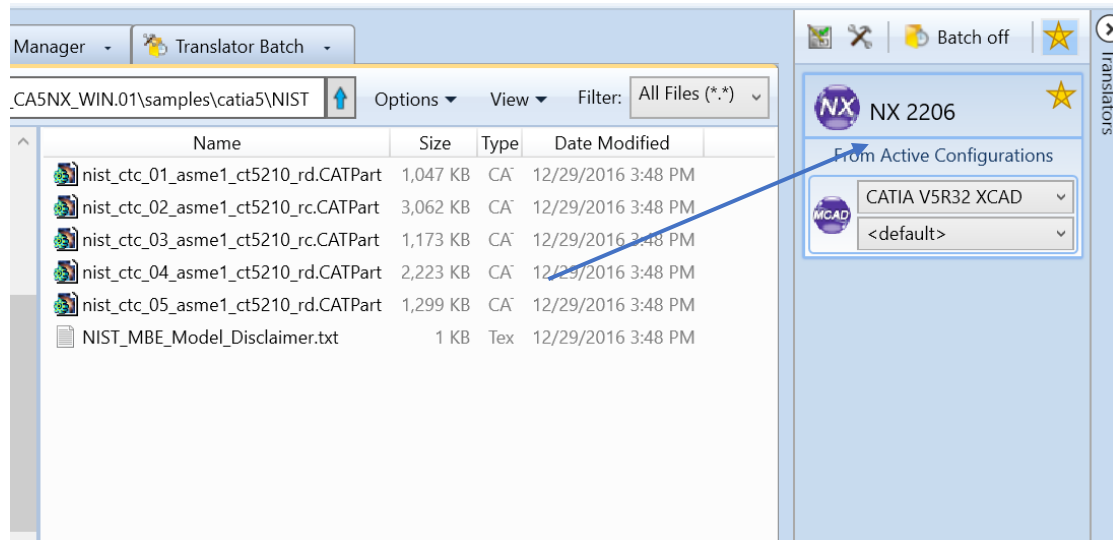
If the CATIA V5 XCAD to NX process is run using the CADVerter GUI or using the command line option 'progress_file' <name>, the process log and error messages will be output to the specified name, and the summary file will also be named after this name with the suffix '.

Default Translations

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

The following interface will be launched:

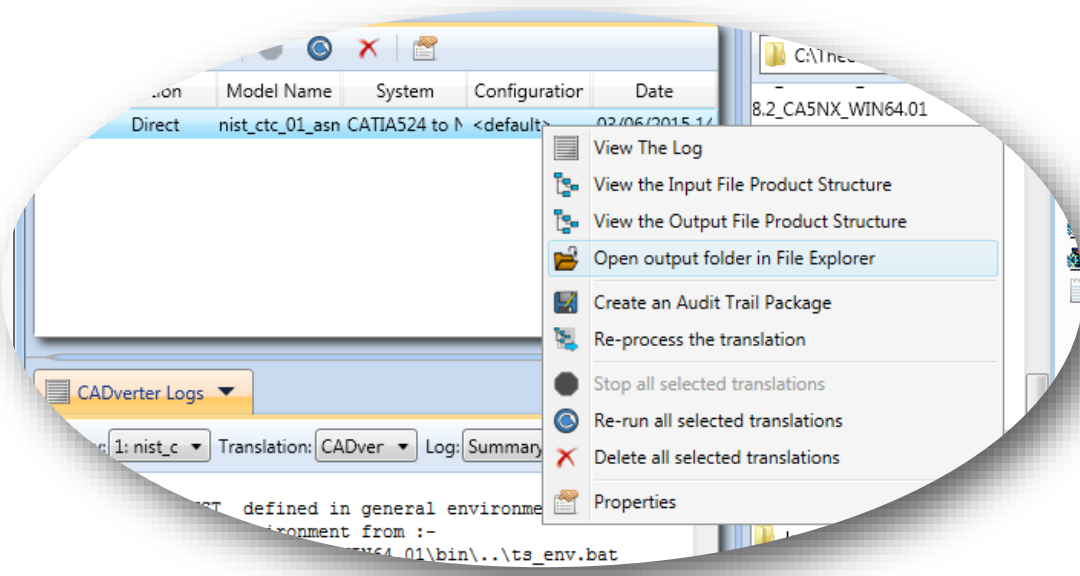




The simplest way to translate from CATIA V5 or NX is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.

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:



NX to V5 translations using the Command Line

To run the translator from the command line you must ensure that the CATIA V5 environment is set correctly. The easiest method to achieve this is to use the standard CATIA start-up process. This uses the following syntax;

```
%V5_INSTALL_PATH%\win_b64\code\bin\CATSTART.exe
-run "%TS_INST%\B%V5_VER%\win_b64\code\bin\UGToNav.exe
%COMMAND_LINE_ARGUMENTS%"
-env Theorem_Multi-CAD_NX90_CATIAV5R%V5_VER%
-direnv "%TS_INST%\B%V5_VER%\win_b64\CATEnv"
```

Where;

%V5_INSTALL_PATH% = the path to the CATIA V5 installation e.g C:\Program Files\Dassault Systems\v5r29\B29

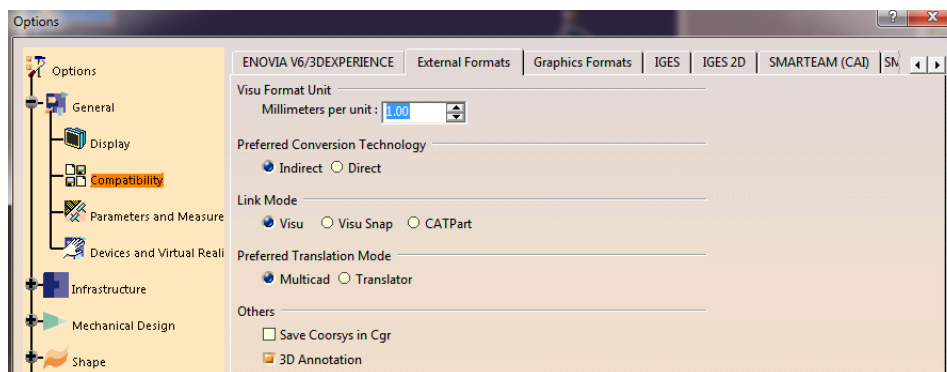
%TS_INST% = the path to the Theorem Solutions installation e.g.

C:\Theorem\CAD_25.3_MC5NX_WIN.01

%V5_VER% = the version of CATIA V5 that is being used e.g. 27, 28 or 29

%COMMAND_LINE_ARGUMENTS% = these will depend upon the type of Link Mode data type that you want to create in the output e.g. VISU, VISU+SNAP or CATPart

Link Mode = Visu



Command Line Arguments

-c %TS_INST%\data\ug\UGToNavConfigs.txt

-i <input_NX_filename>

-oproduct <output_CATProduct_filename>

-d <path_to_output_folder>

read_pmi - optional argument to process NX PMI data. Note this requires access to a V5 FTA license

e.g `-c %TS_INST%\data\ug\UGToNavConfigs_VISU.txt -i %TS_INST%\samples\ug\NIST\nist_ctc_01_asme1_ct5210_rd.prt -oproduct C:\temp\NIST_01.CATProduct -d C:\temp read_pmi`

Default Translation – via the Alternate Command Line Interface

Running a translation via the alternate 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 CATIA5R[XX]XCAD_NX[YY] -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[YY]_CATIA5R[XX]XCAD -i <input_file> -o <output_file>

(Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for CATIA V5 R29, change to 29

Replace the [YY] seen in the example with the version of NX you are using. E.g. for Siemens NX 2007, change to 2007

```
U:\>"C:\Program Files\Theorem\cdCAD_25.1_MC5NX_WIN.01\bin\cad_run.cmd" CATIA5R32XCAD_NX2007 -i "C:\Program Files\Theorem\cdCAD_25.1_MC5NX_WIN.01\samples\catia5\NIST\nist_ctc_01_asme1_ct5210_rd.CATPart" -o C:\Temp\nist_ctc_01_asme1_ct5210_rd.prt
```

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

```

C:\Program Files\Theorem\cdCAD_25.1\MCSNX_WIN_01\bin\cad_run.cmd" CATIASR31KCAD_NX2007 -i "C:\Program Files\Theorem\cdCAD_25.1\MCSNX_WIN_01\samples\catia5\NIST\nist_ctc_01_asm1_ct5210_rd.CATPart" -o C:\Temp\nist_ctc_01_asm1
Warning: TS_INGT defined in general environment
Sourcing Theorem Environment from:
C:\Program Files\Theorem\cdCAD_25.1\MCSNX_WIN_01\bin\..ts_env.bat
Using CATIA environment file C:\Program Files\Theorem\cdCAD_25.1\MCSNX_WIN_01\1831\win_b64\CATEnv\Theorem\Multi-CAD_NX2007_CATIAVSR31.txt
Launching Translation please wait...
[+]
*****
[+] * Copyright Theorem Solutions Limited *
[+] * CATIAV5 - NX CADverter Version 25.1.001 *
[+] *****
[+] CATIA Runtime Version is 831
[+] Start Time Tue Oct 11 09:21:06 2022
[+]
[+] Log File will be created in CATReport directory:
[+] C:\Users\bpittman\AppData\Local\ DassaultSystemes\CATReport
*****
[+] * Copyright Theorem Solutions Limited *
[+] * CATIA Multi-CAD - NX CADverter Version 25.1.001 *
[+] *****
Tue Oct 11 09:21:35 2022
Input
CATIA File : C:\Program Files\Theorem\cdCAD_25.1\MCSNX_WIN_01\samples\catia5\NIST\nist_ctc_01_asm1_ct5210_rd.CATPart
NX File : C:\Temp\nist_ctc_01_asm1_ct5210_rd.prt
Progress File : C:\Users\bpittman\AppData\Local\Temp\ntc_xcad_export.log
*****
[+] * GCD file successfully created *
[+] * C:\Temp\nist_ctc_01_asm1_ct5210_rd.prt.vwr *
[+] *****
List of gco entities :-
-----
Type      Total      Standalone  Subordinate
-----
Lines      284          284
Curves    150          150
Surfaces   4            4
Cylinders  67           67
Planes     88           88
Faces      151          151
Edges      414          414
Vertices   270          270
Solids      1            1
Axis systems 1            1
-----
[+] Please wait, process is active
[+] Save of Output Document C:\Temp\nist_ctc_01_asm1_ct5210_rd.prt completed
[+] End Time Tue Oct 11 09:21:43 2022
*****
[+] * Successful Completion *
[+] * C:\Temp\nist_ctc_01_asm1_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 3 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
- **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:

CATIA V5 XCAD Read NX Write General	
Option Name	Value
Preferred Read Data Type	Exact
SAG Value	0.2
Read 3D Annotations	<input type="checkbox"/>
Disable Points	<input type="checkbox"/>
Disable Axis Systems	<input type="checkbox"/>
Disable Wireframe	<input type="checkbox"/>
Disable Surfaces	<input type="checkbox"/>
Disable Solids	<input type="checkbox"/>
Export Body Names	<input type="checkbox"/>

Each of these options is described below:

Option	Description
Preferred Read Data Type	Allows the user to specify Exact or Tessellated read <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> Exact (Default) Tessellated <ul style="list-style-type: none"> read_tess
SAG Value	Edit the SAG value being used (Default 0.2) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> sag [value]
Read 3D Annotations	Read PMI (Default is OFF) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> read_pmi
Disable Points	Disable the processing of standalone Points (Default is OFF) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> disable_points

Disable Axis Systems	Disable the processing of Axis Systems (Default is OFF) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> disable_axes
Disable Wireframe	Disable the processing of standalone Wireframe (Default is OFF) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> disable_wireframe
Disable Surfaces	Disable the processing of standalone Surfaces (Default is OFF) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> disable_surfaces
Disable Solids	Disable the processing of solids (Default is OFF) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> disable_solids
Export Body Names	Create Body Named Containers (Default is OFF) <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> body_names

NX Write Arguments

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

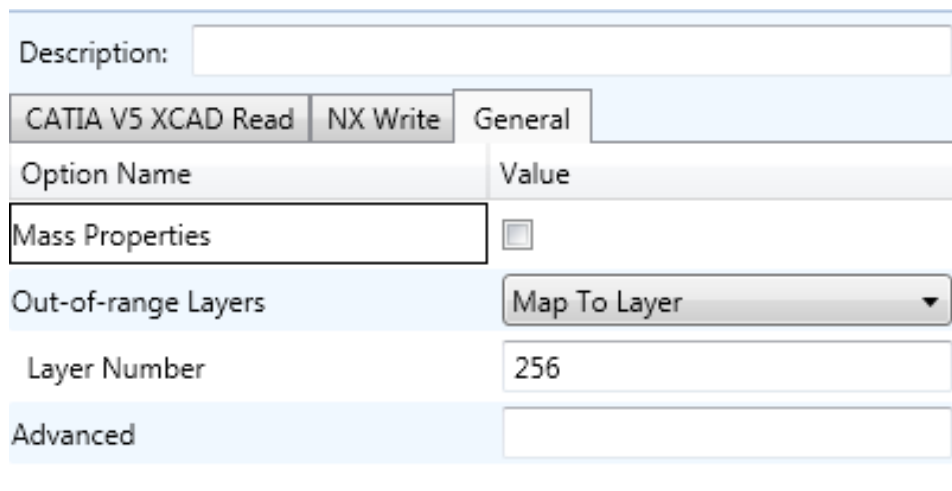
CATIA V5 XCAD Read NX Write General	
Option Name	Value
Delete Existing Sub-parts	<input type="checkbox"/>
Concatenate Assembly Name	<input type="checkbox"/>
Produce Tessellated Output	<input type="checkbox"/>

These options are described below:

Option	Description
Delete Existing Sub-parts	Delete existing sub-parts. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> delete_parts
Concatenate Assembly Name	Concatenates assembly name. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> concat_assy
Produce Tessellated Output	Produce Tessellated NX file. Default is OFF <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> tess_output – to turn on

CATIA V5 to NX General Arguments

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



Option Name	Value
Mass Properties	<input checked="" type="checkbox"/>
Out-of-range Layers	Map To Layer
Layer Number	256
Advanced	

This option 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. <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> 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' <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> 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. <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> 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 NX data into CATIA V5 are grouped into 3 areas:

- **NX Read** – Those arguments that affect how data is read from NX
- **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:

Option Name	Value
Reference Set	<input checked="" type="checkbox"/>
Read Attributes	<input type="checkbox"/>
Read NX names	<input type="checkbox"/>
Read PMI	<input type="checkbox"/>

Each of these options is described below.

Option	Description
Reference Set	Enabled reference set processing. Default is OFF <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> ref_set – to turn on
Read NX Attributes	Read NX detail user attributes. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> read_attrs
Read NX Names	Read NX entity names, if they exist. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> no_read_name – default read_name – to turn on
Read PMI	Read 3D PMI. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> 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:

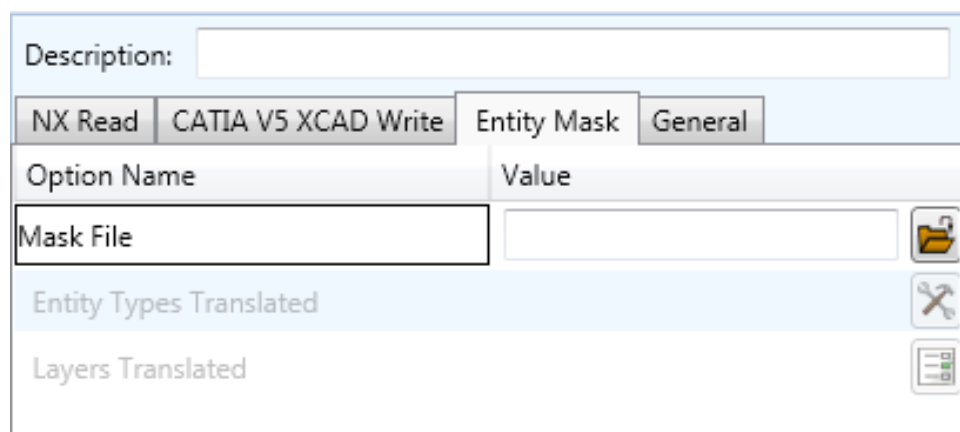
Option Name	Value
Link Mode Selection	CATPart
Preferred Translation Mode	Multicad
Axis Systems - Show	<input type="checkbox"/>
Property Mapping File	<input type="text"/>

Each of these options is described below:

Option	Description
Link Mode Selection	<p>Write Visu, Visu Snap or CATPart data.</p> <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> Multicad (Default) <ul style="list-style-type: none"> mode_catpart Visu <ul style="list-style-type: none"> mode_visu Visu Snap <ul style="list-style-type: none"> mode_visu_snap
Preferred Translation Mode	<p>Write non-editable Multicad or editable Translator data</p> <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> CATPart (Default) <ul style="list-style-type: none"> translator_mode_off Translator <ul style="list-style-type: none"> translator_mode_on
Axis Systems - Show	<p>Axis Systems set to Show state (Default is Off)</p> <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> show_axis_system
Property Mapping File	<p>Specify a file which allows filtering of Detail user attributes. Default is OFF.</p> <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> cad_prop_map_file [file]

NX to CATIA V5 Entity Masking Arguments

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



Option	Description
Mask File	<p>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:</p> <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> Mask <filename>
Entity Types Translated	<p>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"</p> <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> 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	<p>Specifies a selection list from which to select which layers are to be processed.</p> <ul style="list-style-type: none"> Command Line Syntax: <ul style="list-style-type: none"> 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:

Description:

NX Read CATIA V5 XCAD Write Entity Mask General

Option Name	Value
Advanced	<input type="text"/>

The option is described below:

Option	Description
Advanced	Allows any of the Command Line Advanced arguments provided by Theorem Solutions to be passed to the translator via the Unified Interface invocation

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. 28, 29, 30 or 31.

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

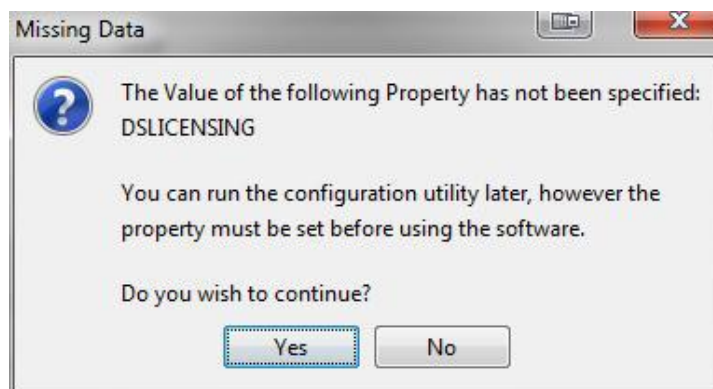
The Theorem translator installation directory is set at installation time in the translator **ts_env.bat** file. This directory shall be noted as <%TS_INST%> in this Appendix.

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.

If no entry is included for DSLICENSING a warning dialog will be displayed which warns of the empty field. Selecting **Yes** to continue will allow the installation to continue.



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 Translator 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 attempt 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.V5R29.B29`

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.V5R29.B29

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.