User Guide

CATIA V5i to NX

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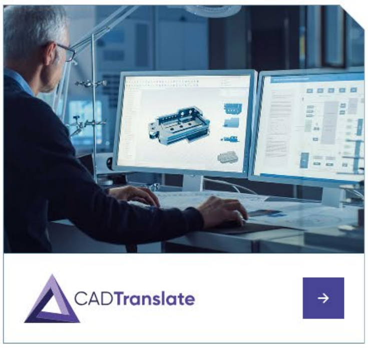


Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of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

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.

The CATIA V5i Bi-directional NX CADverter

The CATIA V5i to NX CADverter is a direct database converter between CATIA V5 and NX. It enables the user to convert all forms of mechanical design geometry, as well as assembly and attribute information, between these two systems without requiring access to a CATIA V5 license.

CADverter can be purchased as a uni-directional, CATIA V5 to NX, or NX to CATIA V5 product, or as a bi-directional product.

The translator can be invoked in 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

- 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
- If you wish to visualise and interrogate the CATIA V5 or NX data this can be done by using the integrated User Interface, which is included with CADverter
- In addition CADverter will work with other Theorem products including Data Exchange Navigator
- There is no dependency on a CATIA V5 installation or application license
- The conversion process can be in Batch Mode or using the Unified Interface
- Command line interface allows process integration

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 visual 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 V5i to 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

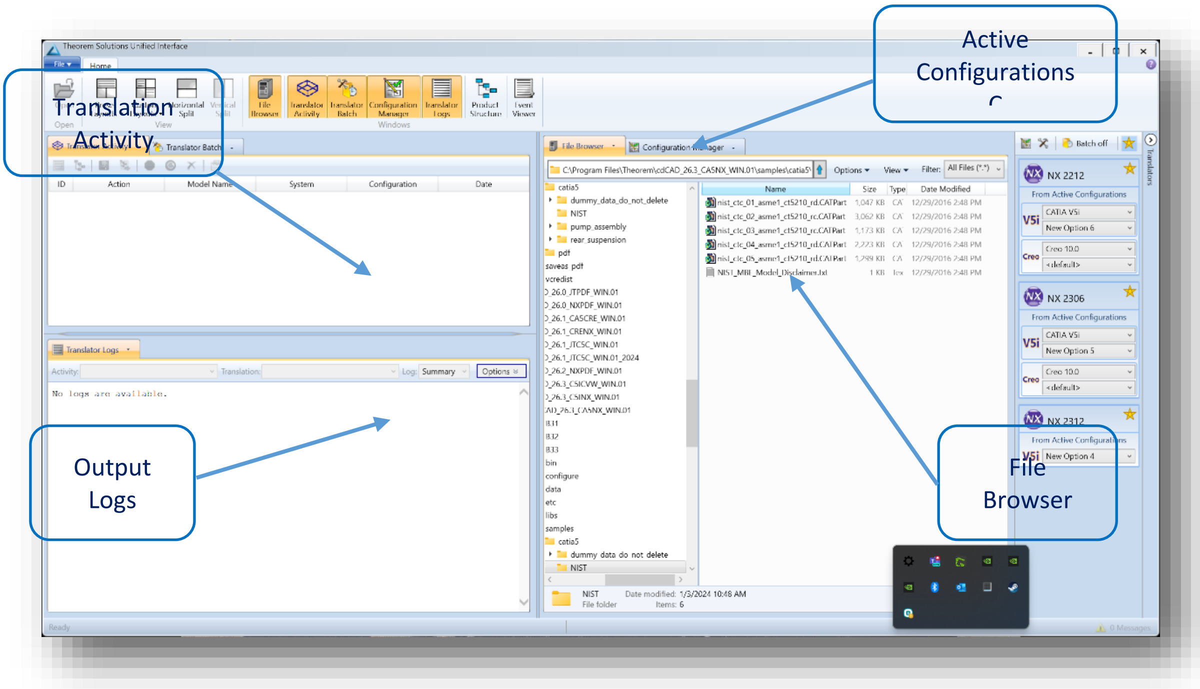
Default Translations

Default Translation – via the Unified Interface

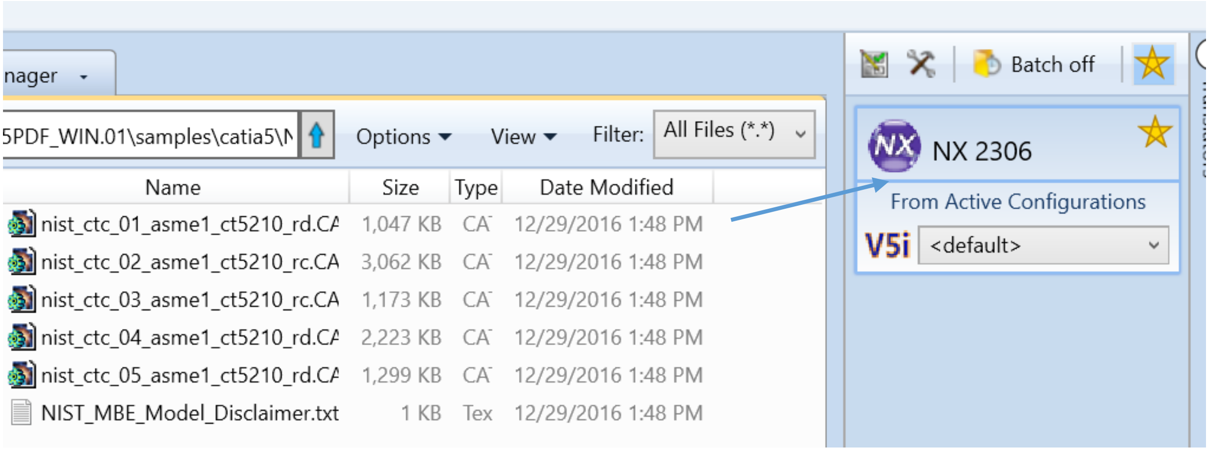
The Unified Interface can be started via the Start Menu – if a shortcut was added during installation.
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

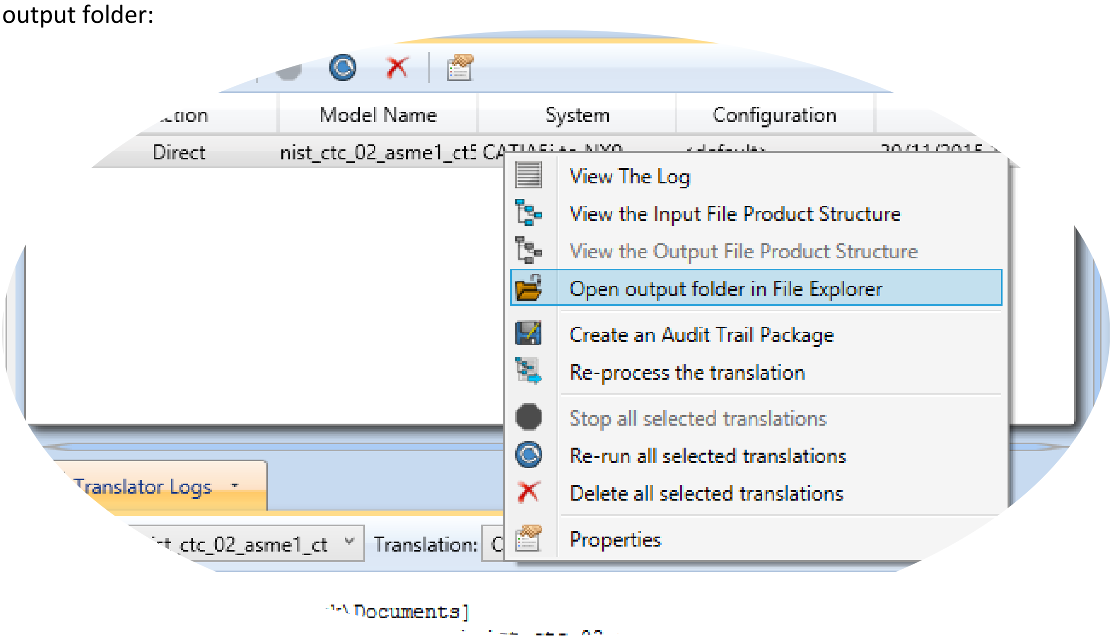
The following interface will be launched:



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 to 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

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 Catia5i_NX[XX] –i <input_file> -o <output_file>

(Note! Replace the [XX] seen in the example with the version of NX you are using. E.g. for NX 1953 change to NX1953):

```
Command Prompt
Microsoft Windows [Version 10.0.19042.867]
(c) 2020 Microsoft Corporation. All rights reserved.

C:\WINDOWS\system32>D:\Theorem_Software\Translators\24.0\cad_run.cmd Catia5i_NX1953 -i C:\temp\sample.CATPart -o C:\temp
\sample_output.prt
```

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

```
 ***********************************************
 * Copyright Theorem Solutions Limited         *
 * CATIA5i - NX11.0 CADverter Version 20.3.001 *
 ***********************************************

 Mon Jan  8 11:11:29 2018

 Input
   CATIA5i Document :
 C:\Program Files\Theorem\20.3\samples\catia5\NIST\nist_ctc_02_asme1_ct5210_rc.CATPart
   NX          File : C:\Temp\nist_ctc_02_asme1_ct5210_rc.prt
   Progress    File : C:\Users\rdugmore\AppData\Local\Temp\tscprogressb5.log

 Mon Jan  8 11:11:29 2018


 List of gco entities :-
 -----------------------
 Type         Total    Standalone  Subordinate
 -----------  -------  ----------  -----------
 Points        8        8
 Arcs          452                  452
 Conics        4                    4
 Lines         689      1           688
 Curves        368                  368
 Surfaces      154                  154
 Cones         158                  158
 Cylinders     228                  228
 Planes        129                  129
 Faces         669                  669
 Edges         1513                 1513
 Vertices      942                  942
 Bsolids       1        1
 -----------  -------  ----------  -----------

Using TS_INST = C:\Program Files\Theorem\20.3\

 ***************************************************
 * Copyright Theorem Solutions Limited             *
 * GCO - Parasolid 29.0 CADverter Version 20.3.001 *
 ***************************************************

 Mon Jan 08 11:12:05 2018

 Input
   GCO File       : c:\users\rdugmore\appdata\local\temp\tscug2_8428.gco
   Parasolid File : c:\users\rdugmore\appdata\local\temp\tscug2_8428
   Progress File : C:\Users\rdugmore\AppData\Local\Temp\13384_ts_c5i_ug_write_leg.la6


 List of gco entities :-
 -----------------------
 Type         Total    Standalone  Subordinate
 -----------  -------  ----------  -----------
 Arcs          452                  452
 Conics        4                    4
 Lines         792                  792
 Curves        264                  264
 Surfaces      154                  154
 Cones         158                  158
 Cylinders     228                  228
 Planes        129                  129
 Faces         669                  669
 Edges         1512                 1512
 Vertices      940                  940
 Bsolids       1        1
 -----------  -------  ----------  -----------

 ****************************************************
 * Parasolid 29.0 file successfully created         *
 * c:\users\rdugmore\appdata\local\temp\tscug2_8428 *
 ****************************************************

 *****************************************************
 *                                                   *
 * CATIA5i to NX conversion ...                      *
 *                           ... Completed Successfully *
 * Output NX file created                            *
 *                                                   *
```

The file will be output to the target location. ***C:\temp\sample_output.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.

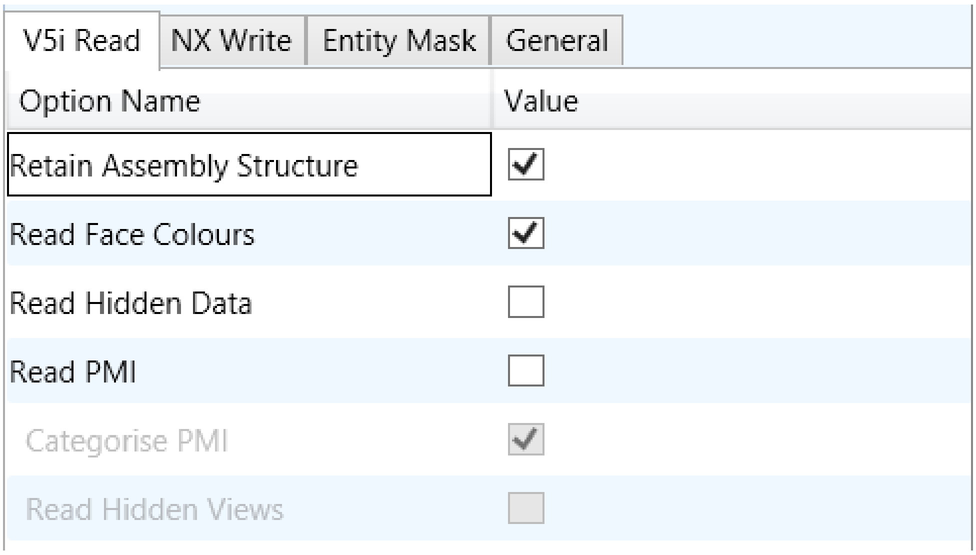
Common Options for CATIA V5i to NX

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

- CATIA V5i 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
- 3DPDF Write – Those arguments that affect how the data is written to 3DPDF
- Masking - Additional Read/Write options to limit the types of data translated
  e.g. Solids Only
- General – Those arguments that are common to ALL Publishing activities regardless of source data

CATIA V5i Read Arguments

The image below shows the CATIA V5I Read arguments that are available, with their default settings:

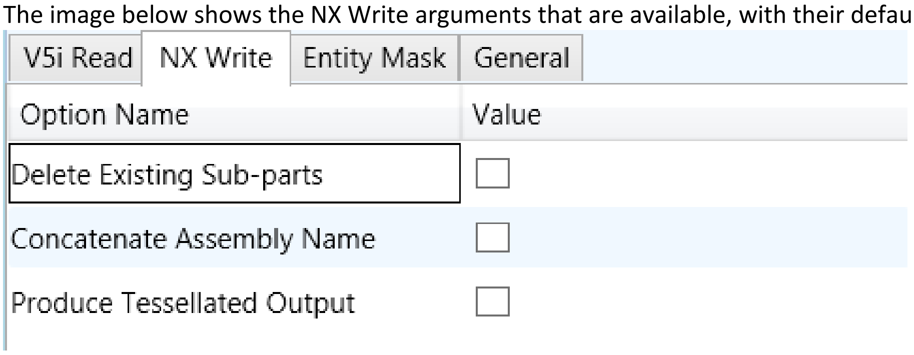


Each of these options is described below:

Option	Description
Retain Assembly Structure	Retain the assembly structure. Default is ON. ○ Command Line Syntax *offditto* (to disable) – reduces an assembly to a single Part
Read Face Colours	Process face colours in preference to body colours. Default is ON ○ Command Line Syntax ▪ *disable_face_colours*
Read Hidden Data	Allow selective data types to be read regardless of hide/show state. Default is OFF. ○ Command Line Syntax ▪ *read_hidden_geometry*
Read PMI	Reads any PMI data available in the V5 file(s)
Categorise PMI	Categorise the PMI by its type (Notes, Datum, GDT etc.
Read Hidden Views	Reads any Views that are hidden

NX Write Arguments

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

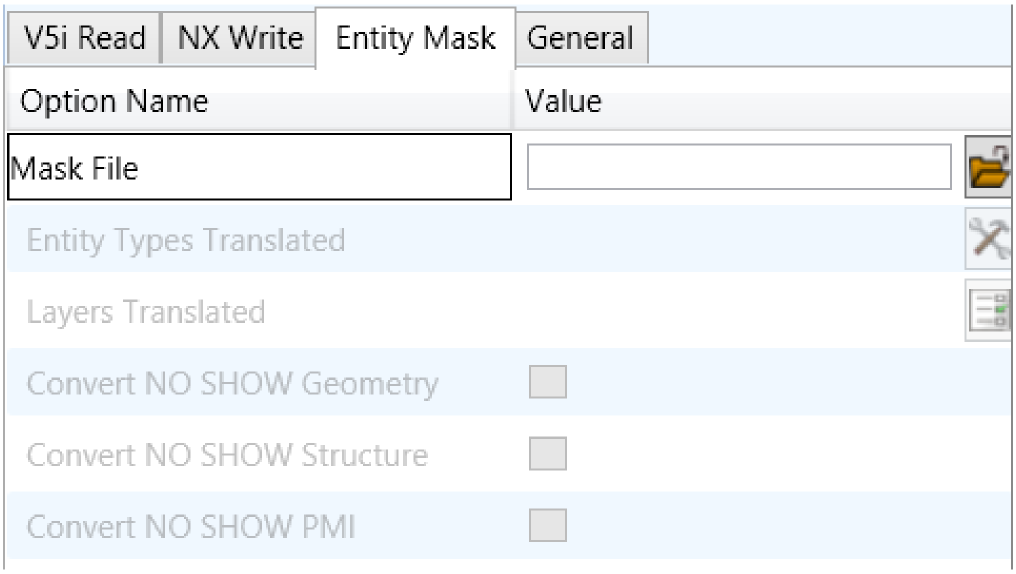


Each of these options is described below:

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	Create a tessellated JT file instead of an NX file. Default is OFF. ○ Command Line Syntax ▪ *Tess_output – to turn off*

CATIA V5i to NX Entity Mask Arguments

The image below shows the CATIA V5i to NX Entity Mask arguments that are available, with their default settings:



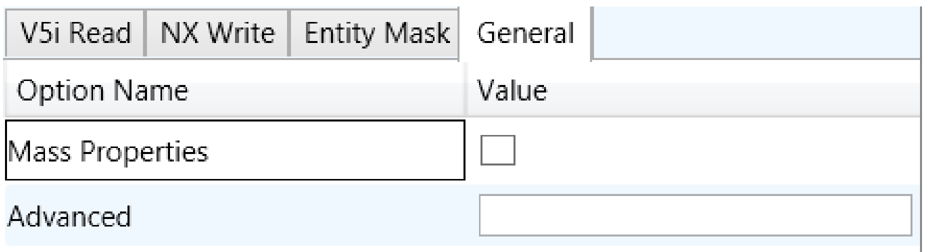
Each of these options is described below:

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: "SOL" - Masks any 3D entity "SKIN" - Masks any 2D entity "CUR" - Masks any 1D entity "POI" - Masks any 0D entity "AXIS" - Masks Axis Systems "ISOL" - Masks Isolated facetted solids "CCRV" - If on creates a CCRV curve for wire frame edges that have more than one supporting curve "TEXT" - Masks PMI Text ○ 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 GEO***
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 AXIS	Enables Hidden Axis Systems to be processed *(Default = Off)* ○ Command Line Syntax: ▪ *Add the following entry to the Mask file* ***ON NOSHOW AXI***

CATIA V5i 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	CATIA V5 mass properties (volume/area CofG) are read and any applied materials, using this option, in cases where a part has multiple solids, volume and area values are summed, but CofG data is invalid. ○ Command Line Syntax ▪ *mprops*
Advanced	Allows any of the Command Line Advanced arguments documented 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.

CATIA V5i to NX Advanced Arguments

Option	Description
Simplify Curves	Convert NURBS curves to conics. Default is OFF. ○ Command Line Syntax ▪ *simplify_curve*
Convert Curves to NURBS	Convert curves to NURBS. Default is OFF. ○ Command Line Syntax ▪ *convert_curves*
Conversion Tolerance	A secondary argument to *'Convert Curves'* defining the conversion tolerance. Default is 0.00001 ○ Command Line Syntax ▪ *convert_curve_tol 0.00001*
Convert Surfaces to NURBS	Process data (read) types as NURBS. Data type is selected from options. Default is Fillets. ○ Command Line Syntax ▪ *None: dont_convert_fillets* ▪ *Fillets: Default Option.* ▪ *Spheres: dont_convert_fillets convert_spheres* ▪ *Fillets + Spheres: convert_spheres* ▪ *All: convert_surfaces*
Convert Torus to NURBS	Even when data is read as NURBS data, the Torus types are converted to NURBS by default, this can be disabled using the command line ○ Command Line Syntax *dont_convert_torus*
Conversion Tolerance	A secondary option to 'Convert Surfaces to NURBS'. Defines the conversion tolerance. Default is 0.00001. ○ Command Line Syntax ▪ *convert_surface_tol 0.00001*
Trim Face Surfaces	Trims face surfaces. Default is ON. ○ Command Line Syntax ▪ *dont_trim_surfaces*
Process Large Faces	Enable reading of faces larger than 1km. Default is OFF. ○ Command Line Syntax ▪ *allow_large_faces*
UDF Axis Systems	Enable reading of User Defined Axis systems. Default is OFF. ○ Command Line Syntax ▪ *read_udf_axis – to turn on*
Graphical Read	By default the BREP data will be read. It is possible to read the CATIA V5 data as a graphical representation using this option ○ Command Line Syntax ▪ *enable_graphical*
Filter Geometry	It is possible to filter large planes (construction planes) larger than a given size using (default being 1000 meters) ○ Command Line Syntax ▪ *filter_large_geom <meters>* There is a special case for PLANES (typically construction planes) which by default are not read, these can be enabled using ○ Command Line Syntax

	▪ *read_planes*

NX to CATIA V5i Advanced Arguments

Advanced Option	Description
Small Curves	Report Small curves as errors. (default OFF) ○ Command Line Syntax ▪ small_curves (to enable)
Extend Nurb Surfaces	Extends NURBS surfaces beyond face limits for curve projection (default state) ○ Command Line Syntax ▪ no_extend_nurb -(*Don't extend NURBS surfaces to face limits)* ▪ extend_nurb <int> - (trims NURBS surfaces to <int> * 0.0001 face extents in u and v)
Remove Groups	Remove Group entities into assembly structure. (default OFF) ○ Command Line Syntax ▪ remove_groups (to enable)
Use Ref Name	Uses file name from input system to name files (default OFF) ○ Command Line Syntax ▪ use_ref_name (to enable)

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