



CADverter for CATIA V5i to Creo

Product Release Version 19.2



USER GUIDE

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Overview of CADverter

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.

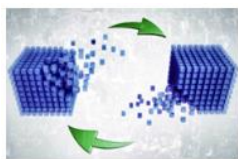


We are proud that the vast majority of the world's leading Automotive, Aerospace, Defense, Power Generation and Transportation companies and their Supply chains use our products and services daily. Working closely with our customers, to both fully understand their requirements and feed their input into 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.

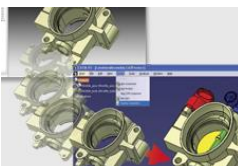
What is CADverter?

CADverter is one of 5 core Theorem brands which consist of:



CADverter

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



Multi-CAD

Interactive integration of non-native 3D data formats into the native CAD system



Visualize 3D

Direct translation of 3D data for the purpose of Visualization



Publish 3D

The creation of documents enriched with 3D content



Process Automation

Applications to automate any Data Exchange and collaboration processes

The CATIA V5i Bi-directional Creo CADverter

The CATIA V5i to CREO CADverter is a direct database converter between CATIA V5 and CREO. 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 CREO, or CREO 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 CREO 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 CREO 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 CREO product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

Getting Started

Documentation

The latest copy of this 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.

Installation Media

The latest copy of Theorem software can be found via our web site at:

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

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 CD or ZIP file download provided.

Currently, there are 2 distinct installation stages that are required.



To install the translator, select the **Setup.exe** file and follow the installation process. For a full guide to the process, please see our 'Translator Installation Process' demonstration video located [here](#).



In addition, the Theorem Unified Interface will also need to be installed. The installation process is the same as for the Translator. For a full guide to the process, please see our 'Translator Installation Process' demonstration video located [here](#).

License Configuration



In order for the translation to run successfully, the Theorem license file provided to you needs to be configured using FlexLM. For a full guide to this process, please see our 'FlexLM License Set Up and Configuration' demonstration video located [here](#).

Running the Product

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

There are 2 distinct ways of running the translator:

- Via the Theorem Unified Interface



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

- Via the Command Line



- 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.

Using the Product

Default Translations

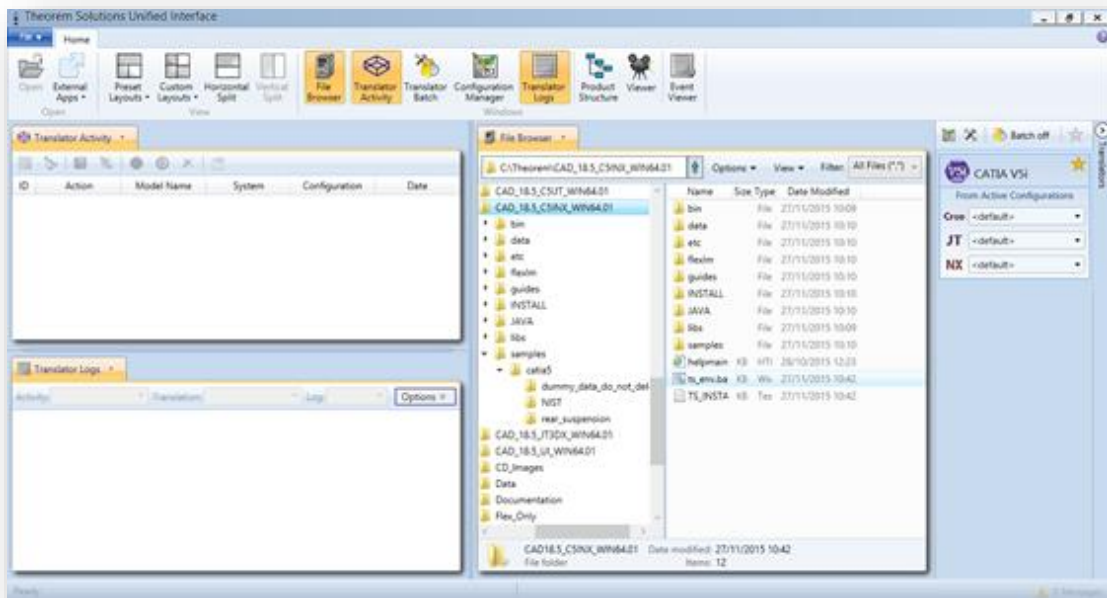
Default Translation – via the Unified Interface

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

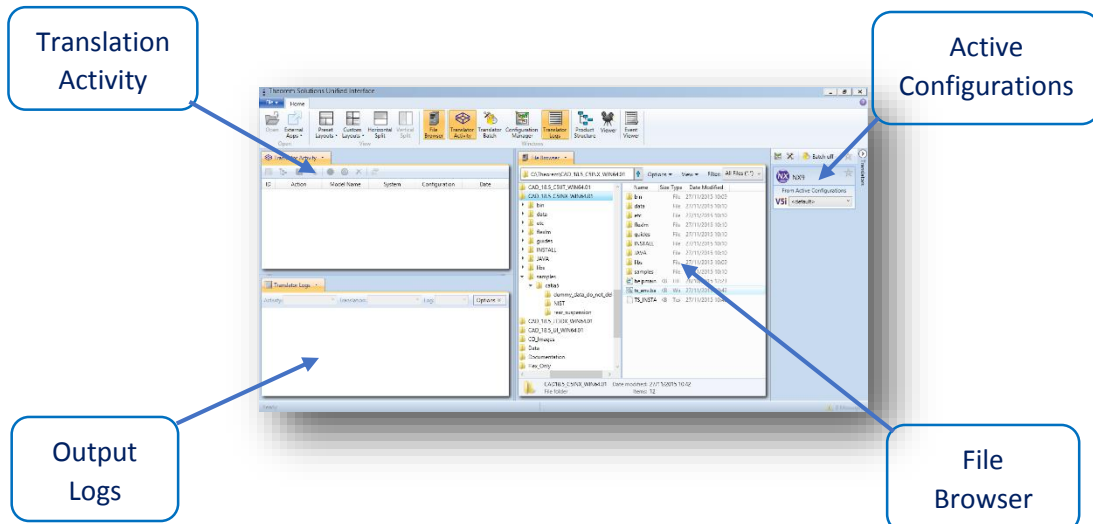
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

```
<UI_installation_directory>\bin\Unified_Interface.cmd
```

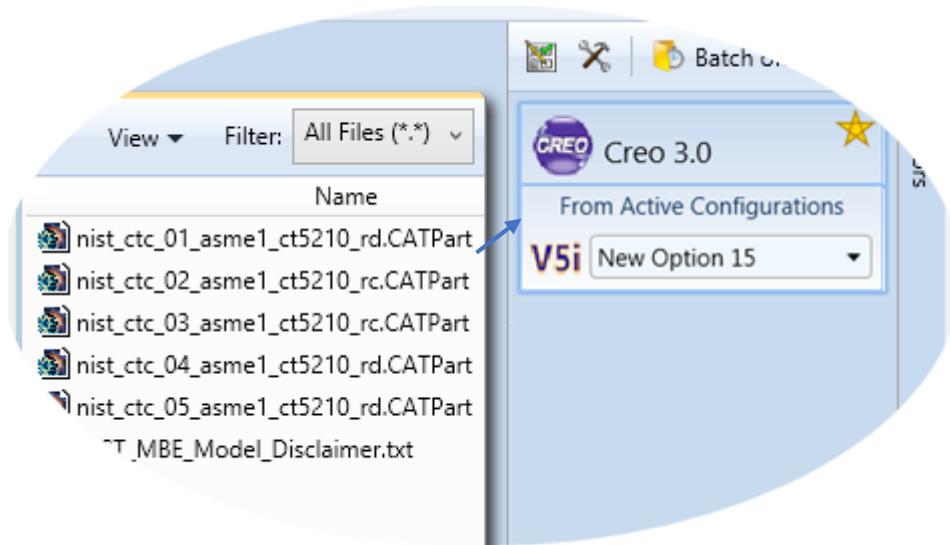
The following interface will be launched:



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

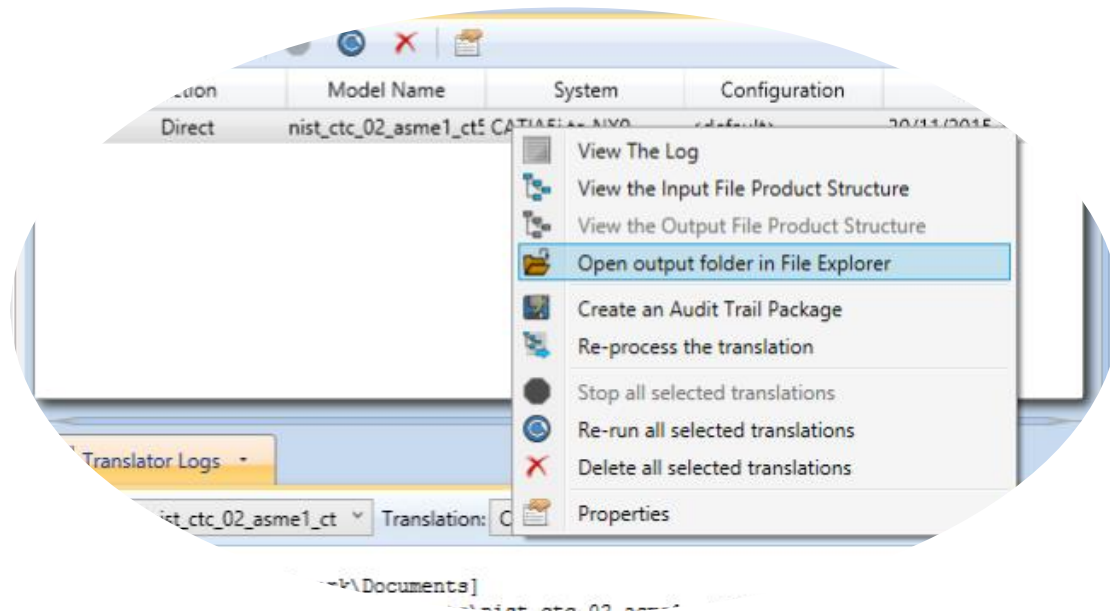


The simplest way to translate from CATIA V5 to CREO is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



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:

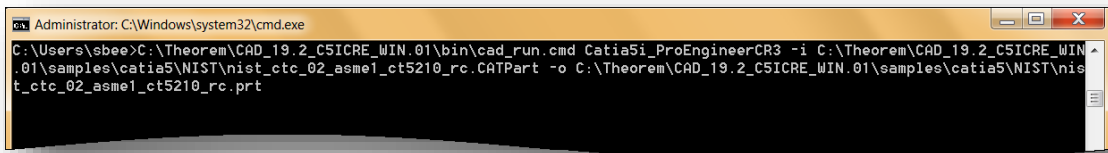


Default Translation – via the Command Line

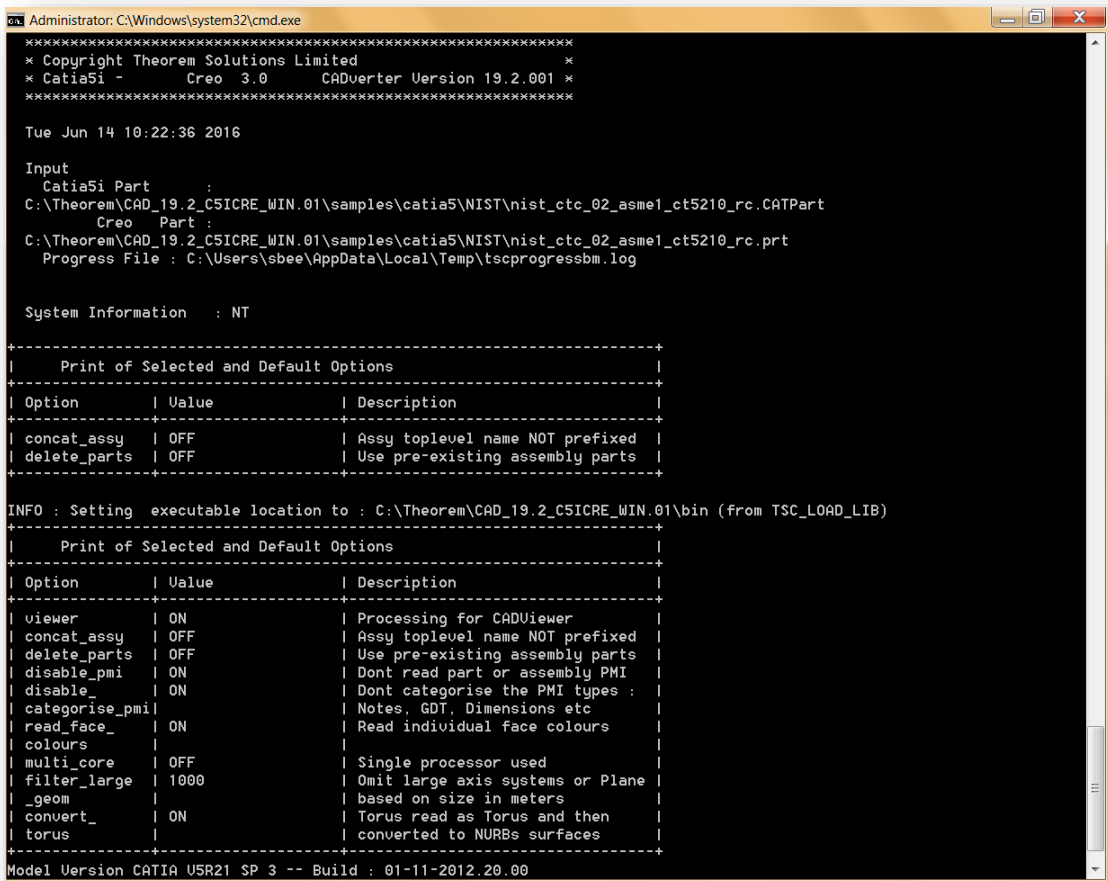
Running a translation via the command line can be carried out via the **cad_run.cmd** file located in the **<installation_directory>\bin** directory. The format of the command is as follows when translating from CATIA V5 to CREO:



**<Translator_installation_directory>\bin\cad_run.cmd Catia5i_ProEngineerCR3 -i
<input_file> -o <output_file>**



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



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

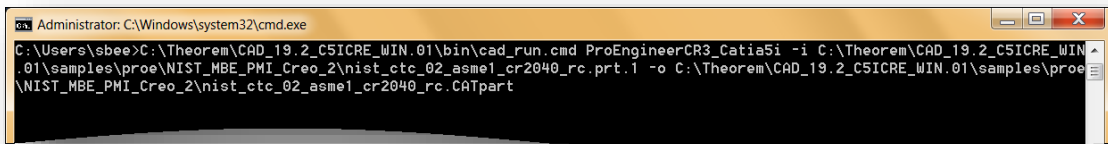
%Theorem_Installation%\samples\catia5\NIST\nist_ctc_02_asme1_ct5210_rc.prt;1



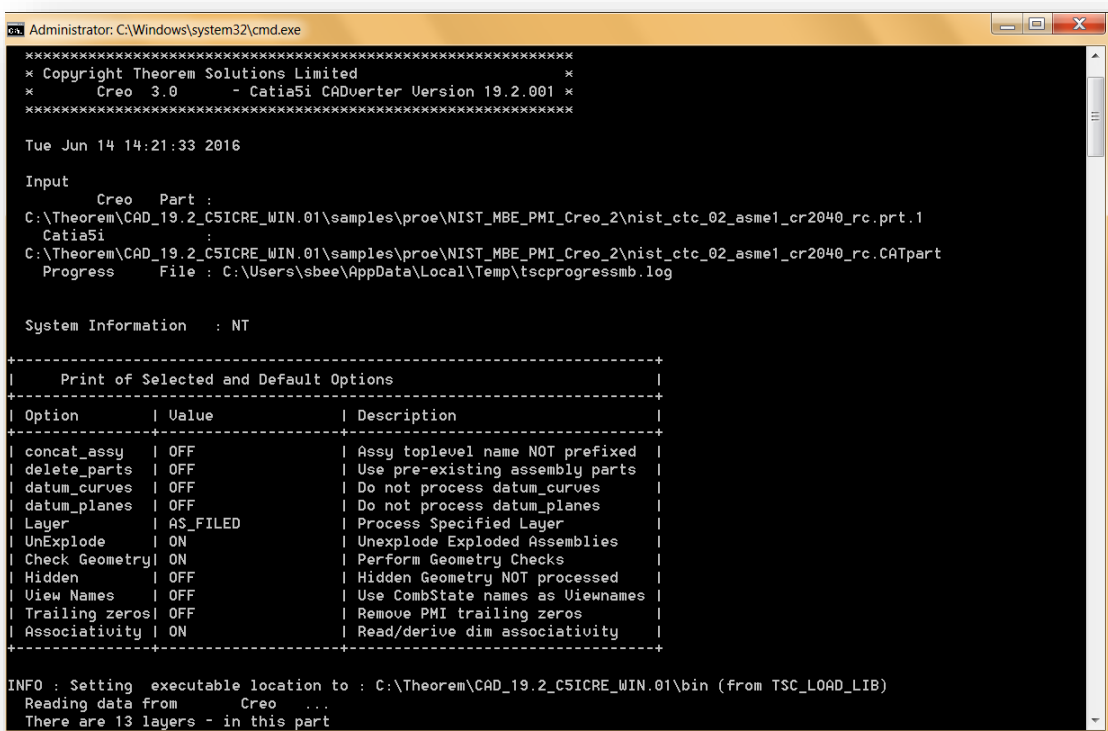
Default Translation – via the Command Line

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 CREO to CATIA V5:

```
<Translator_installation_directory>\bin\cad_run.cmd ProEngineerCR3_Catia5i -i
<input_file> -o <output_file>
```



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



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

```
%Theorem_Installation%\samples\ proe\NIST_MBE_PMI_Creo_2\
nist_ctc_02_asme1_cr2040_rc.CATPart
```



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 CREO

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

- CATIA V5i Read – Those arguments that affect how data is read from CATIA V5
- CREO Write – Those arguments that affect how the data is written to CREO
- 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:

V5i Read		Creo Write	Entity Mask	General
Option Name	Value			
Retain Assembly Structure	<input checked="" type="checkbox"/>			
Read Face Colours	<input checked="" type="checkbox"/>			
Read PMI	<input type="checkbox"/>			

Each of these options is described below:

Option	Description
Retain Assembly Structure	Retain the assembly structure. Default is ON. <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>offditto</i> (to disable) – reduces an assembly to a single Part
Read Face Colours	Process face colours in preference to body colours. Default is ON <ul style="list-style-type: none"> ▪ Command Line Syntax to turn off <ul style="list-style-type: none"> ▪ <i>disable_face_colours</i>
Read PMI	Reads PMI data from V5 <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>read_pmi</i>

CREO Write Arguments

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

V5i Read		Creo Write	Entity Mask	General
Option Name	Value			
Simplify Geometry	<input type="checkbox"/>			
Use existing Assembly Parts	<input type="checkbox"/>			

Each of these options is described below:

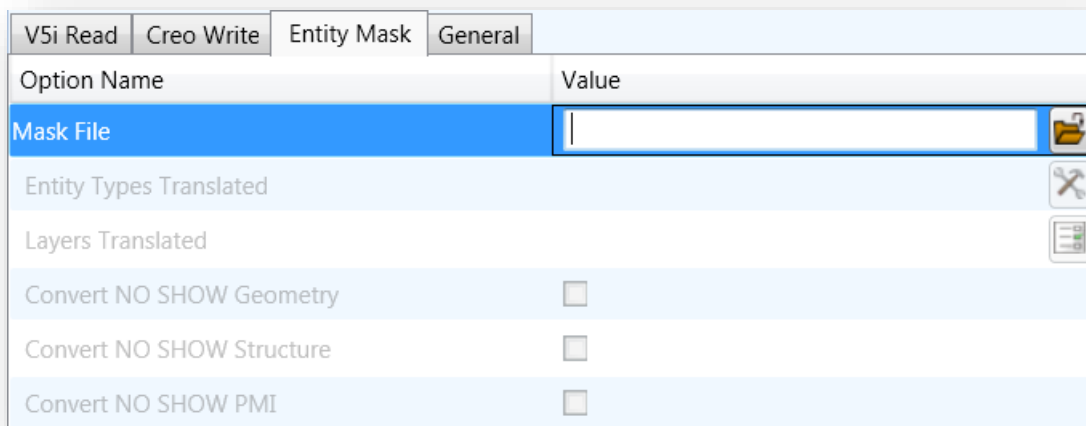
Option	Description
Simplify Geometry	Convert NURBS curves to analytics. Default is OFF. <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>simplify</i>



Use existing Assembly Parts	Use any existing parts created by a previous translation <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>use_parts</i>
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CATIA V5i to CREO Entity Mask Arguments

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



Each of these options is described below:

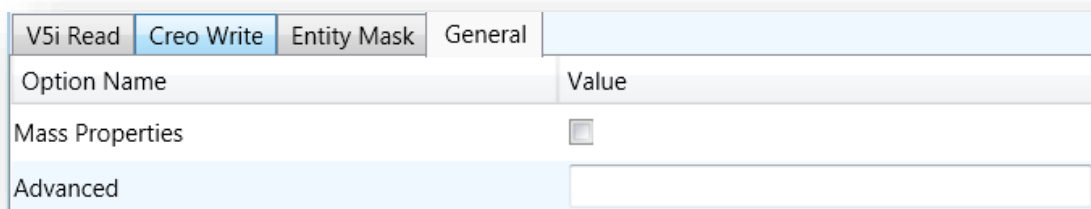
Option	Description
Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>Mask <filename></i>
Entity Types Translated	Specifies a selection list from which to select which entity types are to be processed. The following types are available: <ul style="list-style-type: none"> "POI" - Masks any 0D entity "CUR" - Masks any 1D entity "SKIN" - Masks any 2D entity "SOL" - Masks any 3D entity "ISOL" - Masks Isolated facettted solids "TEXT" - Masks PMI Text "AXIS" - Masks Axis Systems ▪ 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.:



	<p>ON POI to ensure they are considered in the translation</p>
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 <p>e.g.: OFF LAY 114,149,166,167,168</p>
Convert No Show Geometry	<p>Enables Hidden Geometry to be processed (Default = Off)</p> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ Add the following entry to the Mask file ON NOSHOW
Convert No Show Structure	<p>Enables Hidden Assembly Structure to be processed (Default = Off)</p> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ Add the following entry to the Mask file ON NOSHOW STR
Convert No Show PMI	<p>Enables Hidden PMI to be processed (Default = Off)</p> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ Add the following entry to the Mask file ON NOSHOW PMI

CATIA V5i to CREO 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



	<p>part has multiple solids, volume and area values are summed, but CofG data is invalid.</p> <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>mprops</i>
Advanced	<p>Allows any of the Command Line Advanced arguments documented to be passed to the Unified Interface invocation.</p>



Common Options for CREO to CATIA V5i

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

- CREO Read – Those arguments that affect how data is read from CREO
- Catia5i Write – Those arguments that affect how the data is written to Catia5
- General – Those arguments that are common to ALL Publishing activities regardless of source data

CREO Read Arguments

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

Option Name	Value
Transfer Solids	<input checked="" type="checkbox"/>
Transfer Quilts	<input checked="" type="checkbox"/>
Transfer Datum Curves	<input checked="" type="checkbox"/>
Transfer Datum Surfaces	<input checked="" type="checkbox"/>
Retain Assembly Structure	<input checked="" type="checkbox"/>
Read blanked Layers	<input type="checkbox"/>

Each of these options is described below.

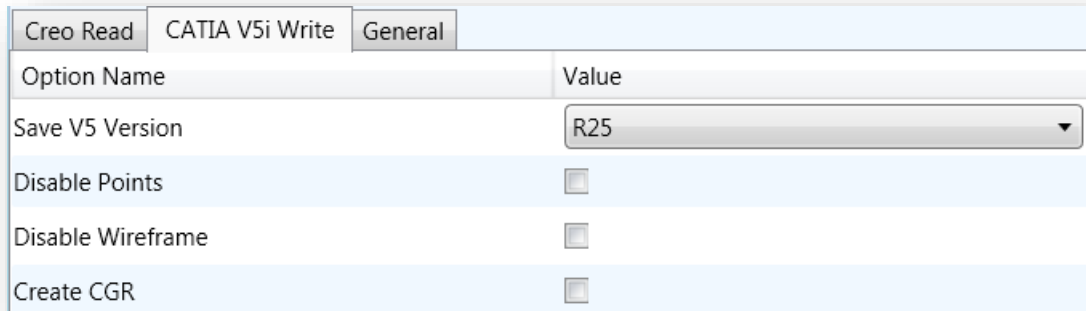
Option	Description
Transfer Solids	Enables solid processing. <i>(Default is on).</i> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>no_solids</i> – to turn off
Transfer Quilts	Enables quilt processing. <i>(Default is on).</i> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>no_quilts</i> – to turn off
Transfer Datum Curves	Enables solid processing. <i>(Default is on).</i> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>no_datum_curves</i> – to turn off
Transfer Datum Surfaces	Enables solid processing. <i>(Default is on).</i> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>no_datum_surfaces</i> – to turn off
Retain assembly structure	This will retain the structure of an assembly. <i>(Default is on).</i> <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>Noditto</i> – to turn off



Read Blanked Layers	Reads all layers, including blanked layers. <i>(Default is off)</i> . <ul style="list-style-type: none"> ▪ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>Layer ALL – to turn on</i>
----------------------------	---

Catia V5i Write Arguments

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



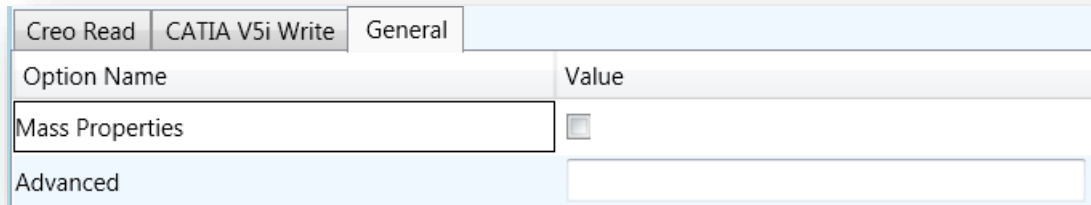
Each of these options is described below:

Option	Description
Save Catia5 Version	Save a specified version of Catia5 data (default R25 (2015)) <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>save_catia5_version <version></i> Where versions are : <ul style="list-style-type: none"> ▪ 16 ▪ 17 ▪ 18 ▪ 19 ▪ 20 ▪ 21 ▪ 22 or 2012 ▪ 23 or 2013 ▪ 24 or 2014 ▪ 25 or 2015
disable_points	Prevents point entities from being written <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>disable_points</i>
disable_wireframe	Prevents wireframe entities from being written <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>disable_wireframe</i>
Create CGR	Writes data as a CGR file <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>Create_CGR</i>



CREO to CATIA V5i General Arguments

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



The option 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. <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>mprops</i>
Advanced	Allows any of the Command Line Advanced arguments documented below 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 CREO Advanced Arguments

Option	Description
Simplify Curves	Convert NURBS curves to conics. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>simplify_curve</i>
Convert Curves to NURBS	Convert curves to NURBS. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>convert_curves</i>
Conversion Tolerance	A secondary argument to 'Convert Curves' defining the conversion tolerance. Default is 0.00001 <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>convert_curve_tol 0.00001</i>
Convert Surfaces to NURBS	Process data (read) types as NURBS. Data type is selected from options. Default is Fillets. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>None: dont_convert_fillets</i> <i>Fillets: Default Option.</i> <i>Spheres: dont_convert_fillets convert_spheres</i> <i>Fillets + Spheres: convert_spheres</i> <i>All: convert_surfaces</i>
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 <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>dont_convert_torus</i>
Conversion Tolerance	A secondary option to 'Convert Surfaces to NURBS'. Defines the conversion tolerance. Default is 0.00001. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>convert_surface_tol 0.00001</i>
Trim Face Surfaces	Trims face surfaces. Default is ON. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>dont_trim_surfaces</i>
Process Large Faces	Enable reading of faces larger than 1km. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>allow_large_faces</i>
UDF Axis Systems	Enable reading of User Defined Axis systems. Default is OFF. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>read_udf_axis – to turn on</i>
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



	<ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>enable_graphical</i>
Filter Geometry	<p>It is possible to filter large planes (construction planes) larger than a given size using (default being 1000 meters)</p> <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>filter_large_geom <meters></i> <p>There is a special case for PLANES (typically construction planes) which by default are not read, these can be enabled using</p> <ul style="list-style-type: none"> ▪ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>read_planes</i>

CREO to CATIA V5i Advanced Arguments

This section will be updated in a future release.

Advanced Option	Description

