User Guide

CATIA V5i - Parasolid

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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.

The CATIA V5 Bi-directional Parasolid Translator

The CATIA V5 to JT translator may be installed on multiple machines each accessing a central network-floating license.

The CATIA V5i to Parasolid Translator is a direct database converter between CATIA V5 and Parasolid. 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.

The Translator can be purchased as a uni-directional, CATIA V5i to Parasolid, or Parasolid to CATIA V5i 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

- 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 Parasolid data this can be done by using the integrated User Interface, which is included with the Translator.

Primary Product benefits?

- Direct conversion between CATIA V5 and Parasolid 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 Visualize 3D for CATIA V5i – Parasolid 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

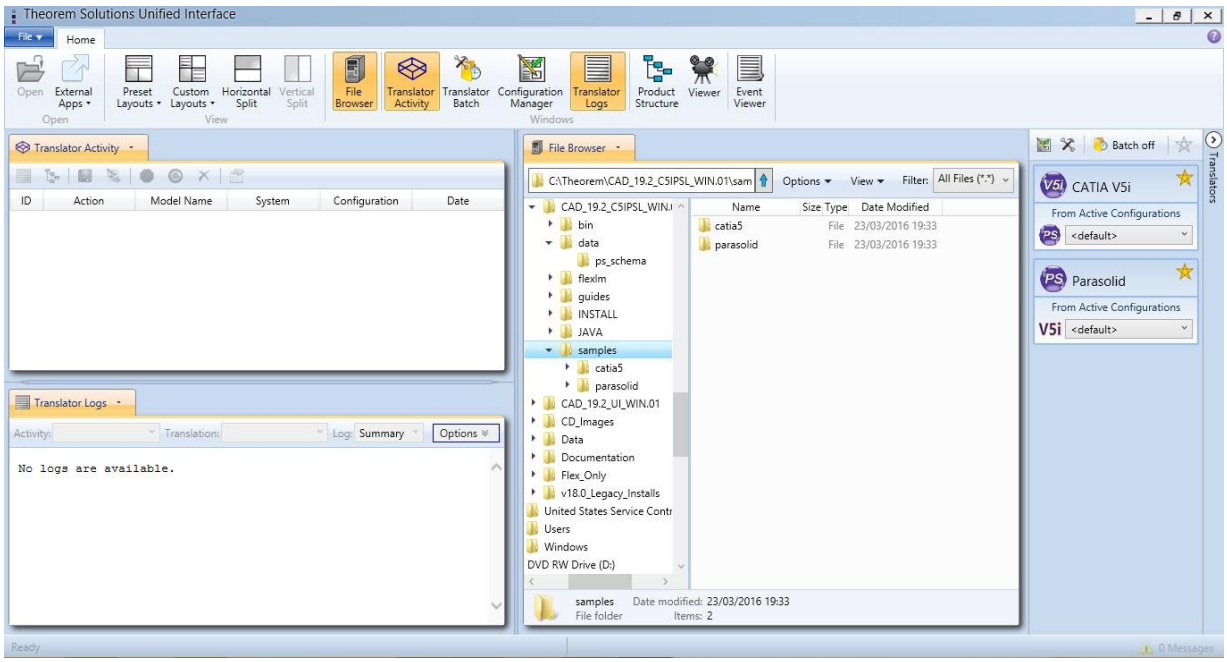
Using the Product

Default Translations

Default Translation – via the Unified Interface

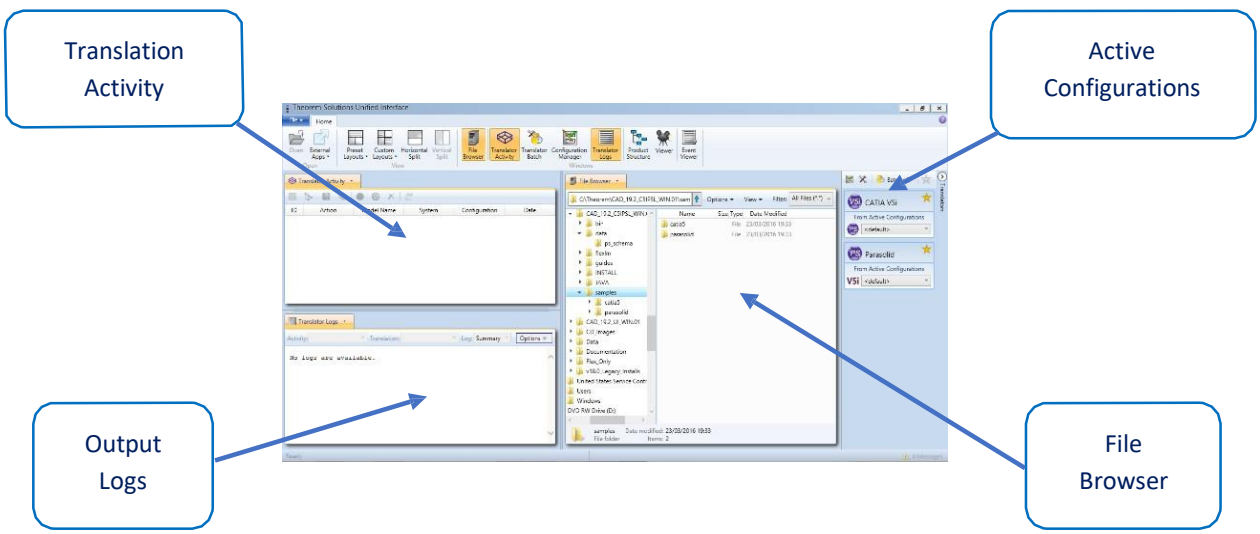
The Unified Interface can be started via the Start Menu – if a shortcut was added during installation.

<UI_installation_directory>\bin\Unified_Interface.cmd

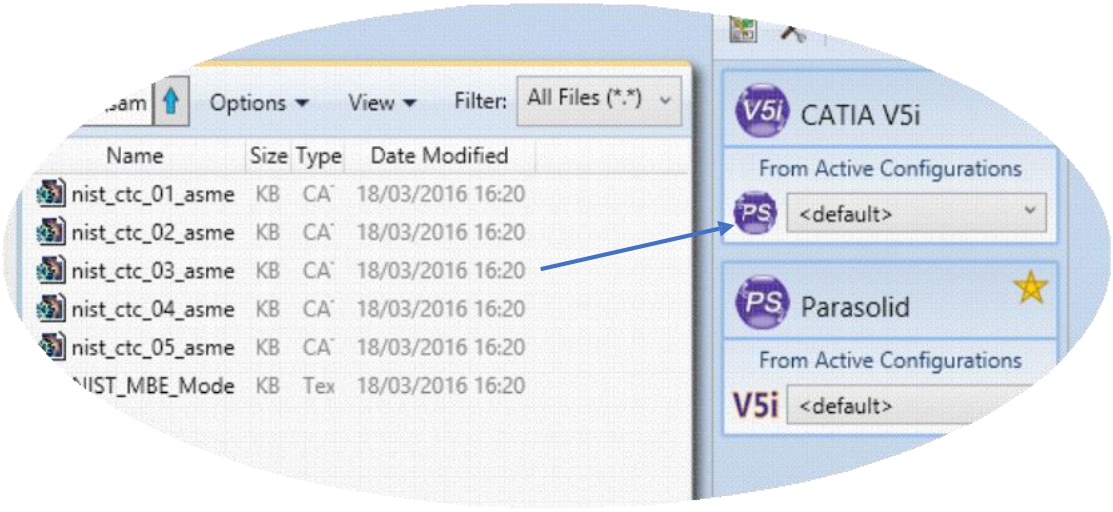


The following interface will be launched:

The default layout is split into 4 primary areas, which can be altered to the user's preference.
Note – When activated the configuration manager will appear as a tab next to the output logs.

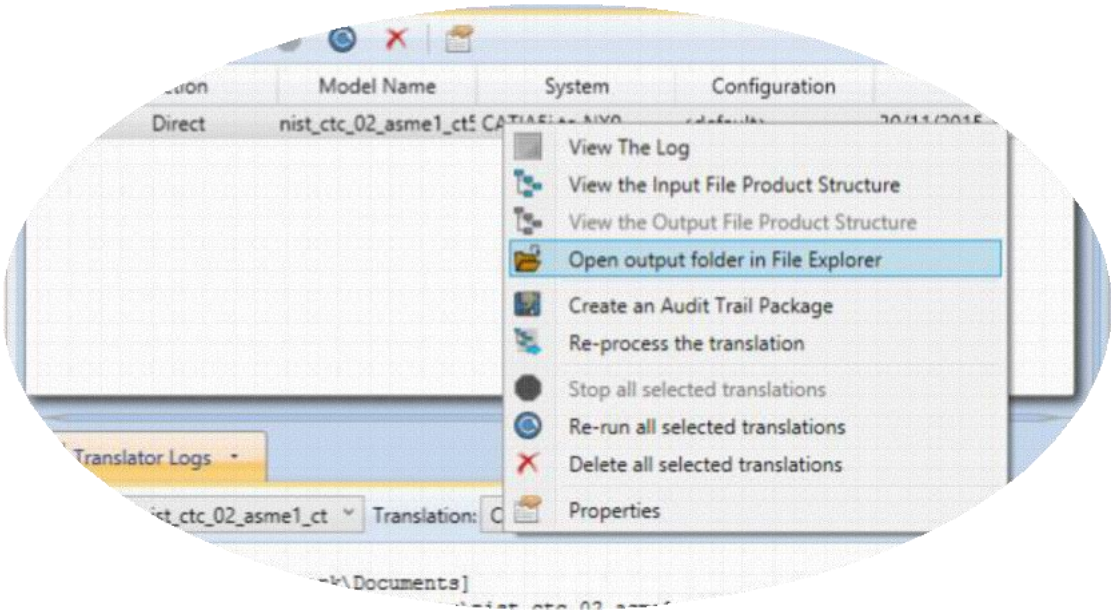


The simplest way to translate from CATIA V5 to PARASOLID is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



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 V5i to PARASOLID:

<Translator_installation_directory>\bin\cad_run.cmd Catia5i_Parasolid –i <input_file> -o <output_file>

```
C:\Users\stephen.clews>"C:\Program Files\Theorem\25.1_CATIA V5i_Parasolid\bin\cad_run.cmd" Catia5i_Parasolid -i "C:
\Program Files\Theorem\25.1_CATIA V5i_Parasolid\samples\catia5\NIST\nist_ctc_01_asme1_ct5210_rd.CATPart" -o C:\temp
\nist_ctc_01_asme1_ct5210_rd
```

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

```
 List of gco entities :-
 -----------------------
 Type         Total    Standalone  Subordinate
 -----------  -------  ----------  -----------
 Curves        418      4           414
 Surfaces      71                   71
 Planes        80                   80
 Faces         151                  151
 Edges         418                  418
 Vertices      277                  277
 Bsolids       1        1
 Axis systems  1        1
 -----------  -------  ----------  -----------

 ****************************************
 * Parasolid file successfully created     *
 * C:\TEMP\NIST_CTC_01_ASME1_CT5210_RD.X_T *
 ****************************************
```

```
Microsoft Windows [Version 10.0.19044.1889]
(c) Microsoft Corporation. All rights reserved.

C:\Users\stephen.clews>"C:\Program Files\Theorem\25.1_CATIA V5i_Parasolid\bin\cad_run.cmd" Catia5i_Parasolid -i "C:
\Program Files\Theorem\25.1_CATIA V5i_Parasolid\samples\catia5\NIST\nist_ctc_01_asme1_ct5210_rd.CATPart" -o C:\temp
\nist_ctc_01_asme1_ct5210_rd

 ****************************************
 * Copyright Theorem Solutions Limited       *
 * CATIA5i - PS CADverter Version 25.1.001   *
 ****************************************

 Tue Aug 16 10:11:12 2022

 Input
   CATIA5i Document : C:\Program Files\Theorem\25.1_CATIA V5i_Parasolid\samples\catia5\NIST\nist_ctc_01_asme1_ct52
10_rd.CATPart
   PS          File : C:\temp\nist_ctc_01_asme1_ct5210_rd.X_T
   Progress File    : C:\Users\STEPHE~1.CLE\AppData\Local\Temp\tscprogressb6.log
```

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

C:\Temp\nist_ctc_02_asme1_ct5210_rd.X_T

The format of the command is as follows when translating from PARASOLID to CATIA V5i:

<Translator_installation_directory>\bin\cad_run.cmd Parasolid_Catia5i –i <input_file> -o <output_file>

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.

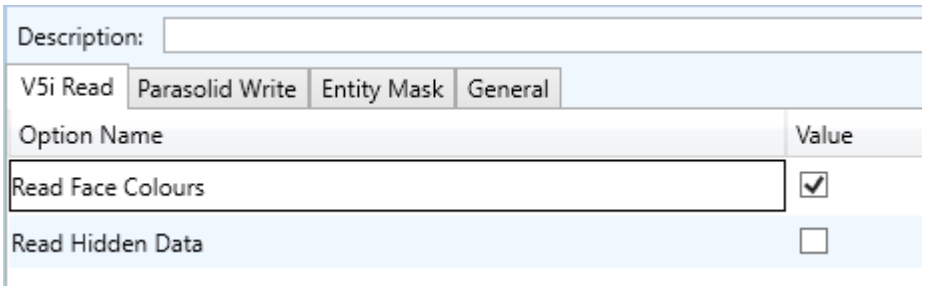
Common Options for CATIA V5i to Parasolid

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

- CATIA V5i Read – Those arguments that affect how data is read from CATIA V5.
- Parasolid Write – Those arguments that affect how the data is written to Parasolid.
- 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 V5i Read Arguments

The image below shows the CATIA V5i Read arguments that are available, with their default settings:

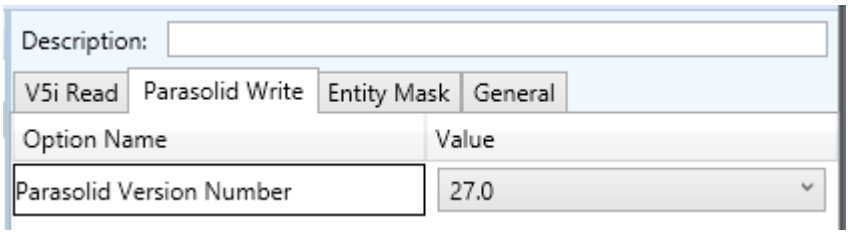


Each of these options is described below:

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 – to turn off.

Parasolid Write Arguments

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

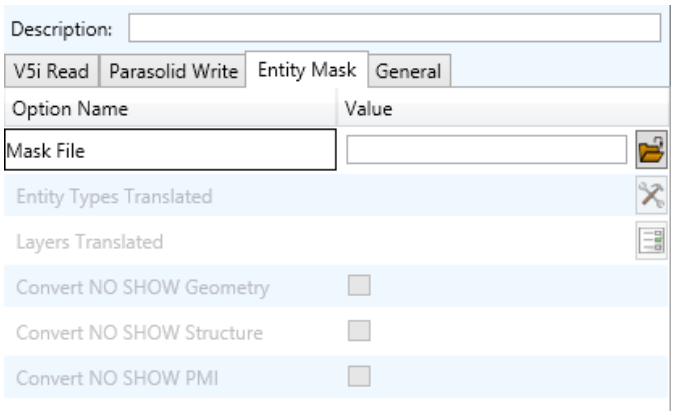


Each of these options is described below:

Option	Description
Parasolid Version Number	Parasolid version number to 'Save As' (default latest - 35) Command Line Syntax ▪ psver <number>

CATIA V5i to Parasolid Entity Masking Arguments

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



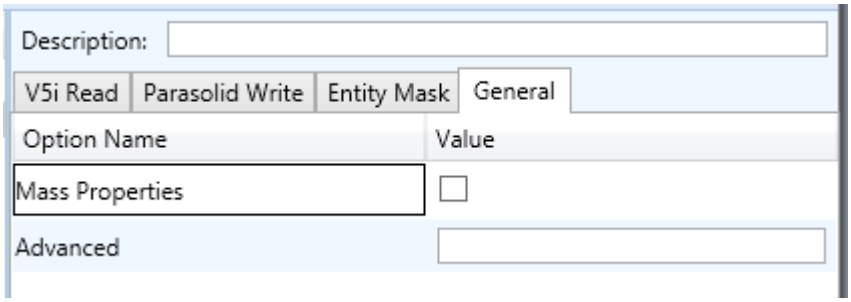
Each of these options is described below:

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 is Off)* Command Line Syntax: ▪ Add the following entry to the Mask file ▪ ON NOSHOW
Convert NO SHOW Structure	Enables Hidden Assembly Structure to be processed *(Default is Off)* Command Line Syntax: ▪ Add the following entry to the Mask file ▪ ON NOSHOW STR
Convert NO SHOW PMI	Enables Hidden PMI to be processed *(Default = Off)* Command Line Syntax: ▪ Add the following entry to the Mask file ▪ ON NOSHOW PMI

CATIA V5i to Parasolid 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.

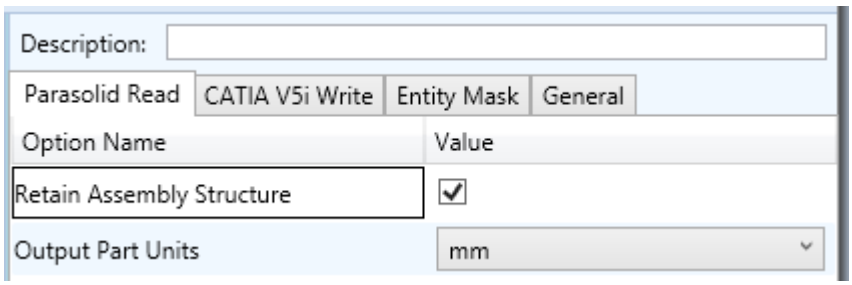
Common Options for Parasolid to CATIA V5i

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

- Parasolid Read – Those arguments that affect how data is read from Parasolid
- Catia5i Write – Those arguments that affect how the data is written to Catia V5
- 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

Parasolid Read Arguments

The image below shows the CATIA V5i Read arguments that are available, with their default settings:

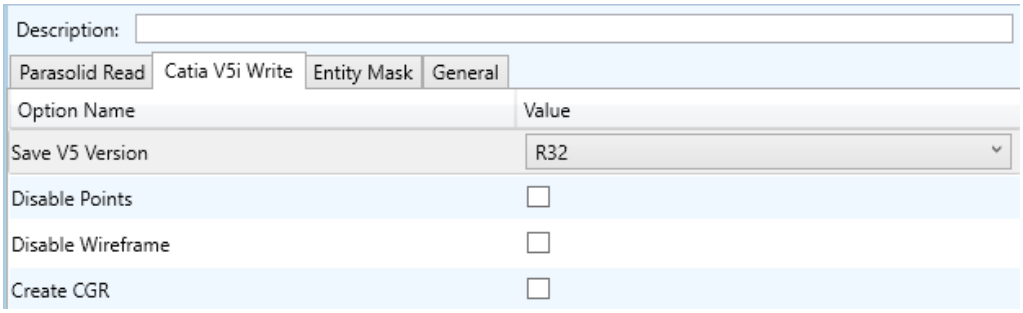


Each of these options is described below:

Option	Description
Retain Assembly Structure	Retain Assembly Structure. Default is ON. Command Line Syntax to disable (remove structure): ▪ noditto
Output Part Units	Define the output part units (default mm) Command Line Syntax: ▪ mm ▪ inches ▪ metres

Catia5i Write Arguments

The image below shows the CATIA V5i Read arguments that are available, with their default settings:



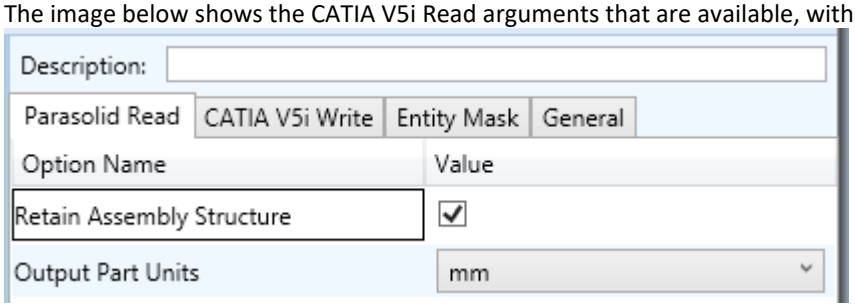
Each of these options is described below.

Option	Description
Write Catia5 Version	Save a specified version of Catia V5 data Command Line Syntax ▪ save_catia5_version <version> ▪ Where versions are: ▪ R28 ▪ R29 ▪ R30 ▪ R31 ▪ R32 ▪ R33

Disable Points	Disable Point processing (default is On) Command Line Syntax to disable Points: ▪ disable_points
Disable Wireframe Processing	Disable Wireframe processing (default is On) Command Line Syntax to disable Wireframe: ▪ disable_wireframe
Create CGR	Create a tessellated CGR file output (default is OFF) Command Line Syntax to create a CGR: ▪ Create_CGR

Parasolid Read Arguments

The image below shows the CATIA V5i Read arguments that are available, with their default settings:

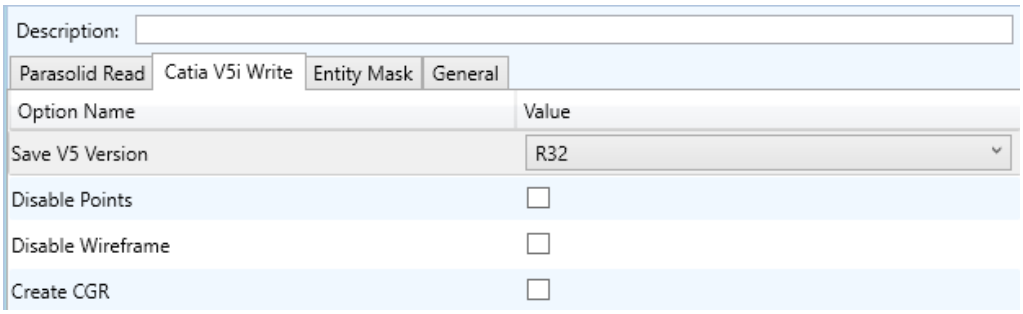


Each of these options is described below:

Option	Description
Retain Assembly Structure	Retain Assembly Structure. Default is ON. Command Line Syntax to disable (remove structure): ▪ noditto
Output Part Units	Define the output part units (default mm) Command Line Syntax: ▪ mm ▪ inches ▪ metres

Catia5i Write Arguments

The image below shows the CATIA V5i Read arguments that are available, with their default settings:

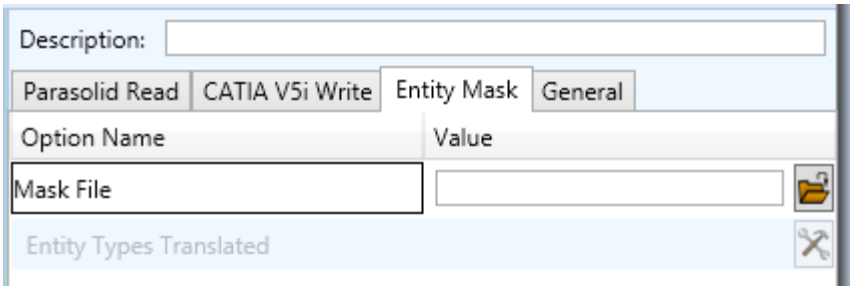


Each of these options is described below.

Option	Description
Write Catia5 Version	Save a specified version of Catia V5 data Command Line Syntax ▪ save_catia5_version <version> ▪ Where versions are: ▪ R28 ▪ R29 ▪ R30 ▪ R31 ▪ R32 ▪ R33
Disable Points	Disable Point processing (default is On) Command Line Syntax to disable Points: ▪ disable_points
Disable Wireframe Processing	Disable Wireframe processing (default is On) Command Line Syntax to disable Wireframe: ▪ disable_wireframe
Create CGR	Create a tessellated CGR file output (default is OFF) Command Line Syntax to create a CGR: ▪ Create_CGR

Parasolid to CATIA V5i Entity Mask Arguments

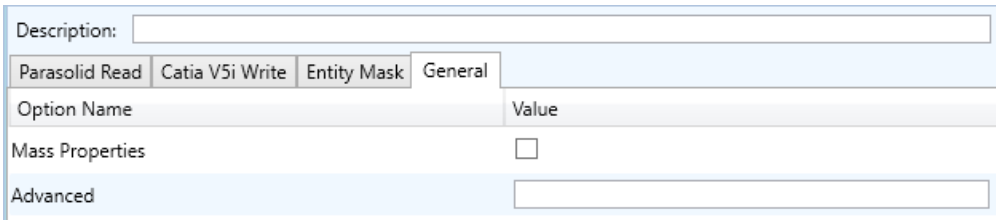
The image below shows the Parasolid to CATIA V5i Entity Mask arguments that are available, with their default settings:



Each of these options is described below:

Option	Description
Output Geometry File Type	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>
Write Face Colours	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.

Parasolid 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. Command Line Syntax: ▪ mprops
Advanced	Allows any of the Command Line Advanced arguments documented below to be passed to the Unified Interface invocation

Command Line Advanced Arguments

Advanced arguments can be added to the Command Line or the UI General->Advanced field.

CATIA V5i Advanced Arguments

Option	Description
Small Curves	Report Small curves as errors. Default is OFF. Command Line Syntax to enable: ▪ small_curves
Extend Nurb Surfaces	Extends NURBS surfaces beyond face limits for curve projection (default state) Command Line Syntax: ▪ no_extend_nurb -(Dont 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 is OFF. Command Line Syntax to enable: ▪ remove_groups
Use Ref Name	Uses file name from input system to name files Default is OFF. Command Line Syntax to enable: ▪ use_ref_name
Simplify Curves	Convert NURBS curves to conics. Default is OFF. Command Line Syntax to enable: ▪ simplify_curve
Convert Curves to NURBS	Convert curves to NURBS. Default is ON. Command Line Syntax to disable: ▪ dont_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 All. i.e. convert ALL surfaces to NURBS

	Command Line Syntax: ▪ None: dont_convert_surfaces ▪ Fillets: dont_convert_fillets. ▪ Spheres: dont_convert_spheres ▪ Toruses: dont_convert_torus ▪ All: convert_surfaces ***e.g. dont_convert_spheres + dont_convert_fillets*** will leave fillets and spheres in their analytical form.
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
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. Default is ON. Command Line Syntax to disable: ▪ dont_convert_torus
Trim Face Surfaces	Trims face surfaces. Default is ON. Command Line Syntax to disable: ▪ dont_trim_surfaces
Process Large Faces	Enable reading of faces larger than 1km. Default is OFF. Command Line Syntax to enable: ▪ allow_large_faces
UDF Axis Systems	Enable reading of User Defined Axis systems. Default is OFF. Command Line Syntax to enable. ▪ read_udf_axis
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. Default is OFF. Command Line Syntax to enable: ▪ 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
Parasolid Tolerant Modelling	Enable Parasolid Tolerant Modelling. Default is ON. Command Line Syntax to disable: ▪ nopstolmodel
Sew Parasolid Bodies	Enabled Sewing of Parasolid Bodies. Default is ON. Command Line Syntax to disable: ▪ nosew
Incremental Sewing	Enable incremental Sewing. Default is ON. Command Line Syntax to disable: ▪ no_sew_increm
Incremental Sewing Iterations	No. of iterations for incremental Sewing (default 5) Command Line Syntax: ▪ sew_increm <number>
Explode Solids to Faces	Explode Solids to Individual Faces. Default is OFF. Command Line Syntax to enable: ▪ split_brep
Split Discontinuous Surfaces	Split Discontinuous Surfaces. Default is ON. Command Line Syntax to disable: ▪ no_brep_prep
Force Body Creation	Force body creation (No check of Parasolid entities - Default is ON) Command Line Syntax to disable: ▪ check
Fix Degenerate Edges	On face create failure, check, and fix any degenerate edges. Default is ON. Command Line Syntax to disable: ▪ no_fix_degen

Parasolid to CATIA V5i Advanced Arguments

Option	Description
Convert Surfaces to NURBS	Read surfaces as NURBS surfaces (else read in native form). Default is ON. Command Line Syntax to disable: ▪ noprep
Convert Edge Curves to NURBS	Read edge curves as NURBS curves (else read in native format) (default ON) Command Line Syntax to disable ▪ rd_native_edge
Mass Props Volume Tolerance	Set tolerance for volume percentage comparison to consider as success (default 1.0) Command Line Syntax: ▪ vol_tol <value>
Mass Props Area Tolerance	Set tolerance for area percentage comparison to consider as success (default 0.5) Command Line Syntax: ▪ area_tol <value>
Mass Props CofG Tolerance	Set tolerance for Centre Of Gravity (CofG) percentage comparison to consider as success (default 1.0) Command Line Syntax: ▪ cog_tol <value>

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