CADverter for CATIA V5 - CADDS

Product Release Version 22.1

USER GUIDE

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Contents

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:
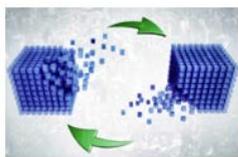
CADverter

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

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Publish 3D

The creation of documents enriched with 3D content

Process Automation

Applications to automate any Data Exchange and collaboration processes

The CATIA V5 Bi-directional CADDS Translator

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

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

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

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

Primary Product Features

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

- Converts assembly structure between the systems

- Converts attribute data including colour and layer information

- The conversion process can be run in Batch mode

- Data can be filtered by layer and entity type

- Geometry can be filtered and selectively processed

Primary Product benefits?

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

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

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

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

- With over 20 years industrial use Theorem's product robustness and quality is well proven, reducing your business risk

This document will focus specifically on guidance for the use of the CADverter for CATIA V5 – CADDS 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.

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

There are 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

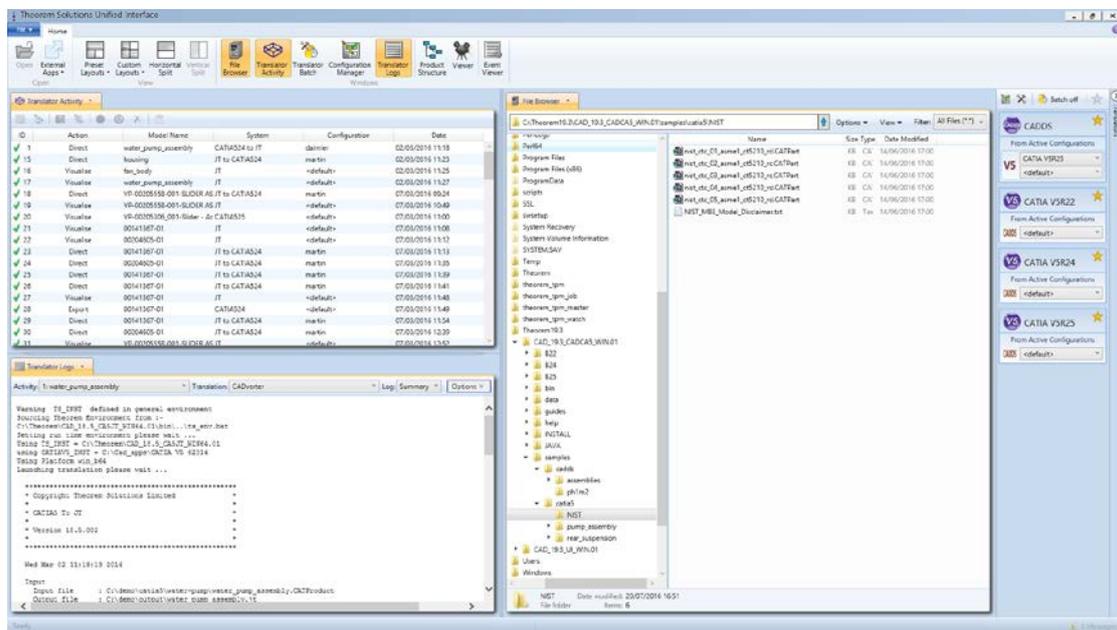### Default Translation – via the Unified Interface

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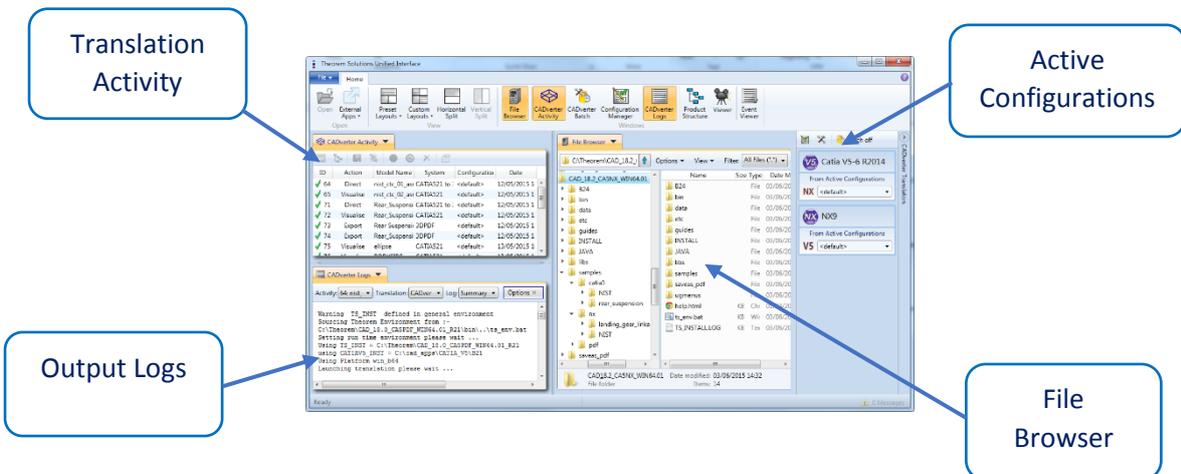
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

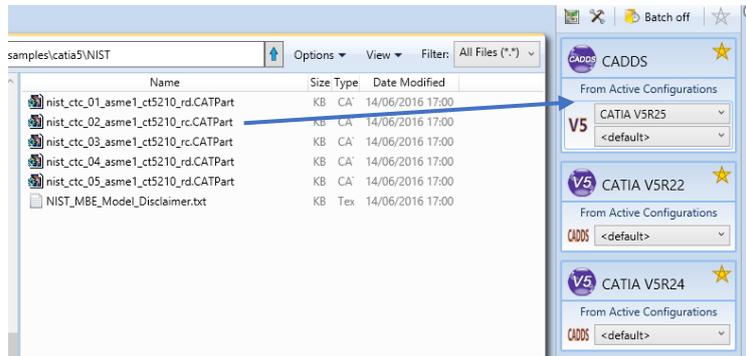**<UI_installation_directory>\bin\Unified_Interface.cmd**

The following interface will be launched:

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

The simplest way to translate from CATIA V5 or CADDS 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:
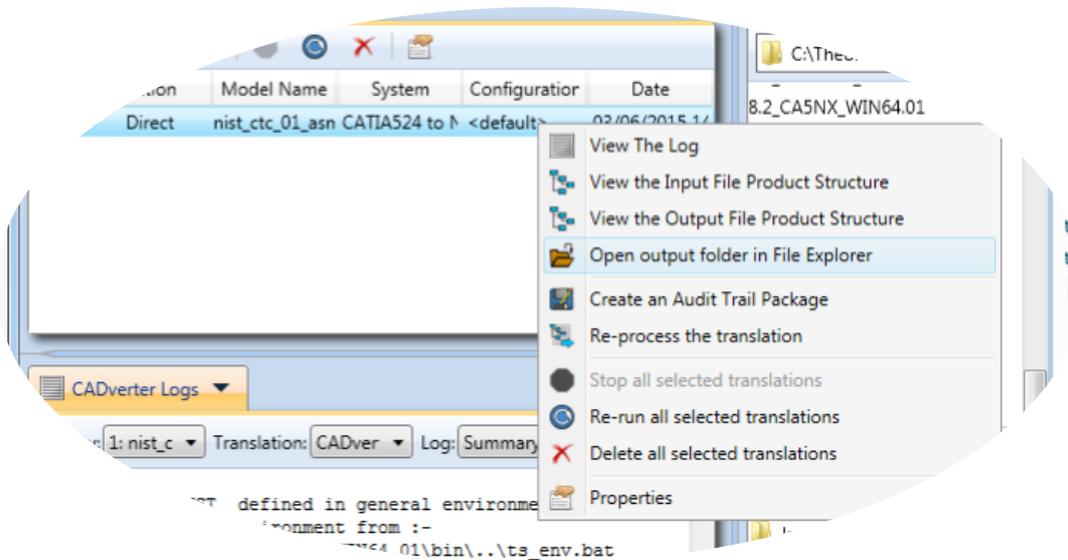
Default Translation – via the Command Line

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 CADDS:

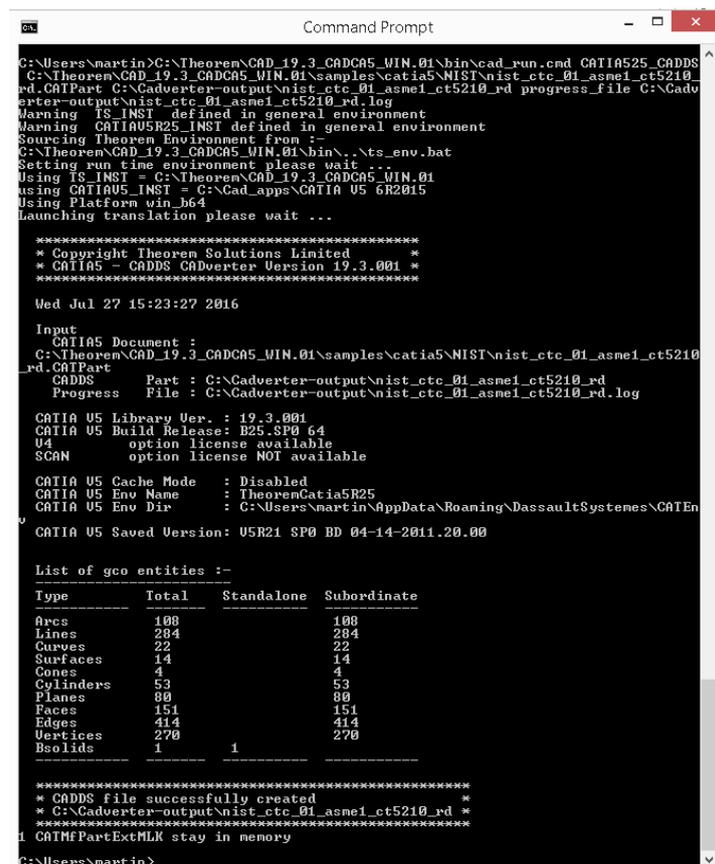**<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_CADDS <input_file> <output_file>**

The format of the command is as follows when translating from CADDS to CATIA5:

<Translator_installation_directory>\bin\cad_run.cmd CADDS_CATIA5[XX] <input_file>
<output_file>

(Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for CATIA V5 R28, change to CATIA528):

```
C:\>C:\Theorem\CAD_19.3_CADCA5_WIN.01\bin\cad_run.cmd CATIA525_CADDS C:\Theorem\
CAD_19.3_CADCA5_WIN.01\samples\catia5\NIST\nist_ctc_01_asme1_ct5210_rd.CATPart C
:\Cadverter-output\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:

```
C:\Users\martin>C:\Theorem\CAD_19.3_CADCA5_WIN.01\bin\cad_run.cmd CATIA525_CADDS
 C:\Theorem\CAD_19.3_CADCA5_WIN.01\samples\catia5\NIST\nist_ctc_01_asme1_ct5210_
rd.CATPart C:\Cadverter-output\nist_ctc_01_asme1_ct5210_rd progress_file C:\Cadv
erter-output\nist_ctc_01_asme1_ct5210_rd.log
Warning  TS_INST  defined in general environment
Warning  CATIAU5R25_INST defined in general environment
Sourcing Theorem Environment from :-
C:\Theorem\CAD_19.3_CADCA5_WIN.01\bin\..\ts_env.bat
Setting run time environment please wait ...
Using TS_INST = C:\Theorem\CAD_19.3_CADCA5_WIN.01
using CATIAU5_INST = C:\Cad_apps\CATIA V5 6R2015
Using Platform win_b64
Launching translation please wait ...

     *******************************************
     * Copyright Theorem Solutions Limited      *
     * CATIA5 - CADDS CADverter Version 19.3.001 *
     *******************************************

     Wed Jul 27 15:23:27 2016

     Input
         CATIA5 Document :
     C:\Theorem\CAD_19.3_CADCA5_WIN.01\samples\catia5\NIST\nist_ctc_01_asme1_ct5210
     _rd.CATPart
         CADDS      Part : C:\Cadverter-output\nist_ctc_01_asme1_ct5210_rd
         Progress   File : C:\Cadverter-output\nist_ctc_01_asme1_ct5210_rd.log

     CATIA V5 Library Ver. : 19.3.001
     CATIA V5 Build Release: B25.SP0 64
     U4          option license available
     SCAN        option license NOT available

     CATIA V5 Cache Mode    : Disabled
     CATIA V5 Env Name      : TheoremCatia5R25
     CATIA V5 Env Dir       : C:\Users\martin\AppData\Roaming\DassaultSystemes\CATEn

     CATIA V5 Saved Version: V5R21 SP0 BD 04-14-2011.20.00


     List of gco entities :-
     ------------------------------------
     Type         Total   Standalone  Subordinate
     ---------   ------   ----------  -----------
     Arcs         108                 108
     Lines        284                 284
     Curves       22                  22
     Surfaces     14                  14
     Cones        4                   4
     Cylinders    53                  53
     Planes       80                  80
     Faces        151                 151
     Edges        414                 414
     Vertices     270                 270
     Bsolids      1       1
     ---------   ------   ----------  -----------

     *******************************************************
     * CADDS file successfully created                     *
     * C:\Cadverter-output\nist_ctc_01_asme1_ct5210_rd     *
     *******************************************************
1 CATMfPartExtMLK stay in memory

C:\Users\martin>_
```

The file will be output to the target location. In this case:
        C:\\nist_ctc_01_asme1_ct5210_rd/_pd

CADverter Customization

CADverter allows the information that is read from the source system and written to the target system to be tailored via a set of user specified arguments. Commonly used arguments are supported via the Unified Interface, with Advanced Arguments being described within this document for use in the Unified Interface or via the Command Line invocation.
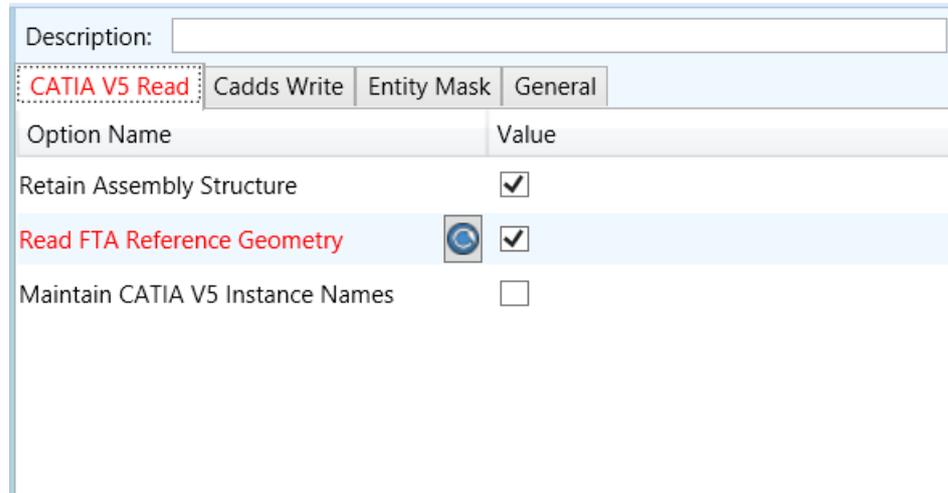
Common Options for CATIA V5 to CADDS

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CATIA V5 data into CADDS are grouped into 4 areas:

- CATIA V5 Read – Those arguments that affect how data is read from CATIA V5

- CADDS Write   – Those arguments that affect how the data is written to CADDS

- Entity Mask   – Those arguments that allow specific read entities to be masked

- General   – Those arguments that are common to ALL Publishing activities regardless of source data

CATIA V5 Read Arguments

The image below shows the CATIA V5 Read arguments that are available, with their default settings:

Each of these options is described below:

Option	Description

Retain Assembly Structure	Enables Assembly Structure to be retain. Default is ON. Disabling this option will remove all assembly structure and collapse ALL geometry into a single selectable object
	    o Command Line Syntax:
	        ▪ *off_ditto – to turn off*

Read FTA Reference Geometry	Enables reading of FTA Reference Geometry *(Default is Off)*
	    o Command Line Syntax:
	    ▪ *read_geometry – to turn on*

Maintain CATIA V5 Instance Names	Honours CATIA V5 *Tools->Options->Infrastructure->Product Structure->Nodes Customization* panel settings *(Default is Off)*
	    o Command Line Syntax:
	    ▪ *ditto_naming V5 – to turn on*

CADDS Write Arguments
The image below shows the Write CADDS arguments that are available, with their default settings.

Each of these options is described below:

Option	Description

Part Format	Defines the format of the output file to be cadds4x or cadds5 - Default is cadds4x
	    o Command Line Syntax
	          ▪ cadds5

Part Precision	Specifies the output part to be written in single or double precision
	Default is double
	       o Command Line Syntax
	          ▪ *single*

Geometry Type	Defines whether NURBS or ASD geometry is written – default is NURBS o Command Line Syntax ▪ *asd*
Simplify Geometry Tolerance	Tolerance value for CADDS simplify of psurfs Default tol =0.001 in part units o `Command Line Syntax` • c4simplify <tol>
Explode	Explode brep to faces Default off o Command Line Syntax ▪ split_brep

Overwrite	Overwrite existing parts default=use existing parts o Command Line Syntax ▪ *no_overwrite – use existing parts* ▪ *overwrite – overwrite existing parts*
Concatenate Name	concatenate top level assy name to all subcomponents default= no_concat_assy o Command Line Syntax ▪ concat_assy/no_concat_assy

CATIA V5 to CADDS Entity Masking Arguments

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

Each of these options is described below:

Option	Description

Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT:
	
	    o 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: "POI","CUR","SKI","SOL","ISO","TEX","AXI"
	
	    o 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.
	
	    o 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*

16 | P a g e		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)*
		   o 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 = Off)*
		   o 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)*
		   o Command Line Syntax:
		        ▪ *Add the following entry to the Mask file*
		          ON NOSHOW PMI

CATIA V5 to CADDS General Arguments

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

Each of these options is described below:

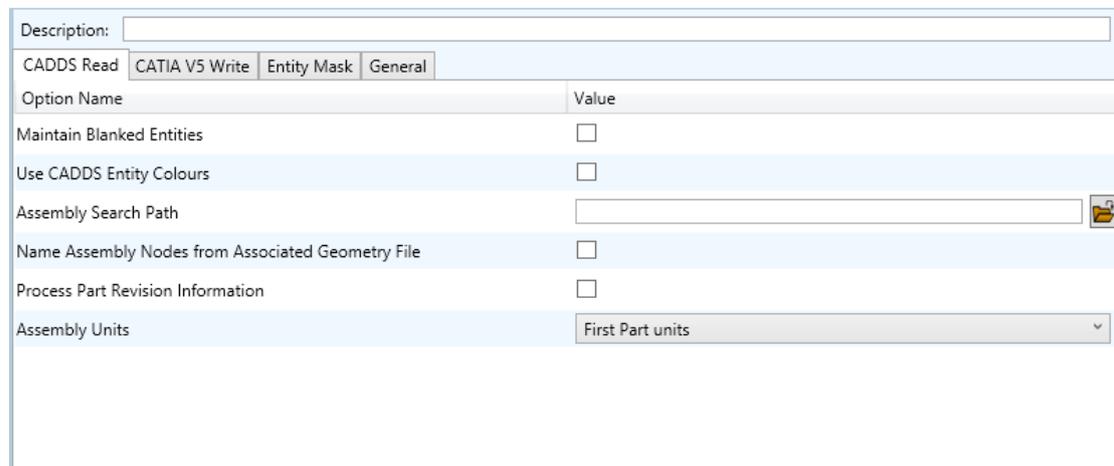| Option	Description

Mass Properties	Allows Mass Property information to be read from the source data and written as attributes to the PDF document. Default is OFF.
	    o Command Line Syntax:
	         ▪ *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'
	    o Command Line Syntax:
	         ▪ *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.
	    o Command Line Syntax:
	         ▪ *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 CADDS to CATIA V5

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CATIA V5 data into CADDS are grouped into 4 areas:

- CADDS Read   – Those arguments that affect how data is read from CADDS

- CATIA V5 Write   – Those arguments that affect how the data is written to CATIA5

- 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

CADDS Read Arguments

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

Each of these options is described below.

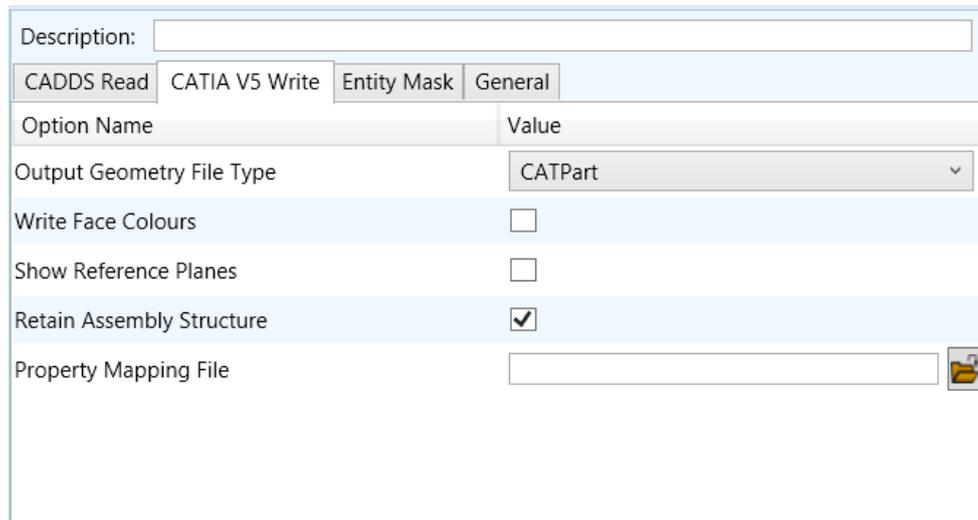| Option	Description

Maintain Blanked Entities	Read blanked entities and maintain their show/hid state Default is OFF
	      o Command Line Syntax
	          ▪ *maintain_blanked*

Use CADDS Entity Colours	Use CADDS entity colours rather than part colours
	      o Command Line Syntax:
	          ▪ *ecol*

Assembly Search Path	Specify the search paths that contain assembly parts
Name Assembly Nodes from Associated Geometry File	o Command Line Syntax: Default is OFF ▪ *mapitem*

Process Part Revison Information	Read assembly revision info from _ps file
	o Command Line Syntax:
	▪ *read_rev*
Assembly Units	Specify the units when reading an assembly
	o Command Line Syntax:
	▪ *assy_units <mm/inch>*

CATIA V5 Write Arguments

The image below shows the CATIA V5 Write arguments that are available, with their default settings:
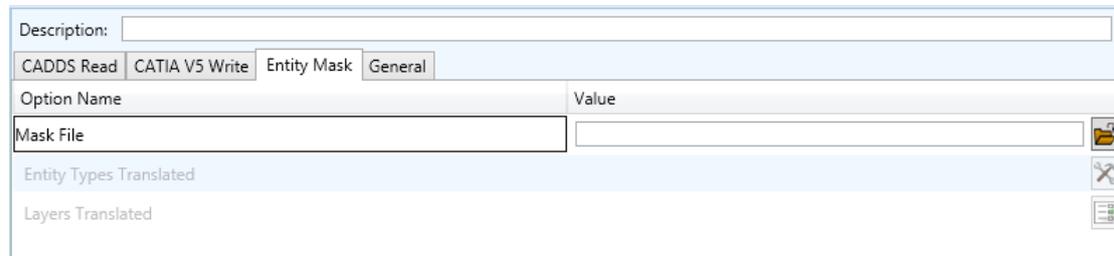

Each of these options is described below:

Option	Description

Output Geometry File Type	Output Geometry file type. Default is CATPart.
	   o Command Line Syntax
	      ▪ *CATPart: output_type CATPart*
	      ▪ *Model: output_type model*
	      ▪ *Cgr: output_type cgr*
	      ▪ *Igs: output_type igs*
	      ▪ *CATShape: output_type CATShape*
	      ▪ *Tessellated: create_cgr*

Write Face Colours	Writes face colours. Default is ON.
	   o Command Line Syntax
	      ▪ *FACE_COLOUR*
	      ▪ *SOLID_COLOUR – to turn off*

Show Reference Planes	Creates reference planes. Default is OFF.
	  Command Line Syntax
	  o Show: *dont_blank_planes*

Retain Assembly Structure	Maintain assembly structure in derive output *(Default is ON)*. Command Line Syntax o `off_ditto`

Property Mapping File	Assign a property mapping file
	  Command Line Syntax
	  o *cad_prop_map_file [Path to file]*

CADDS to CATIA V5 Entity Masking Arguments

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

Each of these options is described below:

Option	Description

Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT:
	   o 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:
	"POI","LIN","ARC","CON","CUR","SUR","FAC","SOL"
	   o 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.
	   o 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**

CADDS to CATIA V5 General Arguments

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

Description:	
CADDS Read	CATIA V5 Write	Entity Mask	**General**
Option Name	Value
Advanced	

The option is described below:

Option	Description
Advanced	Allows any of the Command Line Advanced arguments documented below to be passed to 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. 25, 26, 27, 28 or 29.

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.
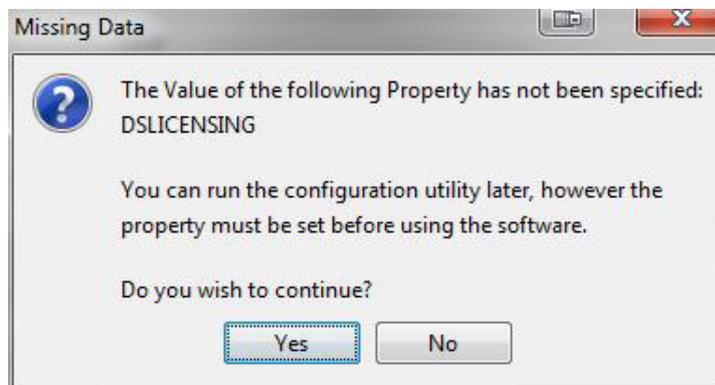
Running CATIA V5 Translators

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 CADverter by defining in the *ts_env.bat* the environment variable CATIAV5_DIRENV:

set CATIAV5_DIRENV=/some/directory

If the Theorem installation is needed to support multiple releases of CATIA. Then the user can define release specific locations using:

set CATIAV5R<XX>_DIRENV=/some/directory

The Theorem translator will attempts to create its own environment file called *TheoremCatia5R<XX>.txt.* The user must therefore have write permission to the CATEnv directory. If this is not possible an existing environment file can be specified using the variable **CATIAV5_ENV**. e.g.

set CATIAV5_ENV=CATIA.V5R19.B19

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.V5R19.B19

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.