User Guide

CATIA V5i to CreoView

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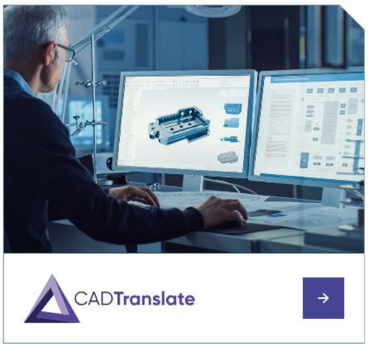


Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of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

The CATIA V5i to CreoView CADverter is a direct database converter between CATIA V5I and CreoView. 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 V5I license.

CADverter can be only purchased as a uni-directional, CATIA V5I to CreoView 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 V5I or CreoView 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

Primary Product benefits?

- Direct conversion between CATIA V5I and CreoView 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 V5i to CreoView product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

CATIA V5I to Creo View PMI Service Module

The PMI module allows PMI, Captures and View states to be read from CATIA V5I and written into Creo View.

CATIA V5I to Creo View JT Add On Module

The JT Add On module works independently of the Post Process module and allows a JT file to be created alongside assemblies and parts processed into Creo View.

CATIA V5I to Creo View 3D PDF Add On Module

The 3D PDF Add On module works independently of the Post Process module and allows a 3D PDF file to be created alongside assemblies and parts processed into Creo View.

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 ***CAD_xx.x_C5ICVW_WIN.xx.msi*** file and follow the installation process. For a full guide to the process, please see our 'Translator Installation' Document 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' 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.

Introduction

Overview

The CATIA V5i CADverter allows the user to translate CATIA V5 assemblies (.CATProduct files), components (.CATPart) from their original CATIA V5 format into the PTC Creo View .pvs, .ed and .ol file formats. The CADverter is developed using Spatial Independent CGM Interface and therefore is independent of the CATIA V5 application.

Support for specific CATIA V5I Release

CATIA V5I Object CADverters are available for support of the following releases of the CATIA V5 Application including all Service Packs at the nominated release.

CATIA V5 revisions R25 up to CATIA V56R2021(R31)

Using the Product

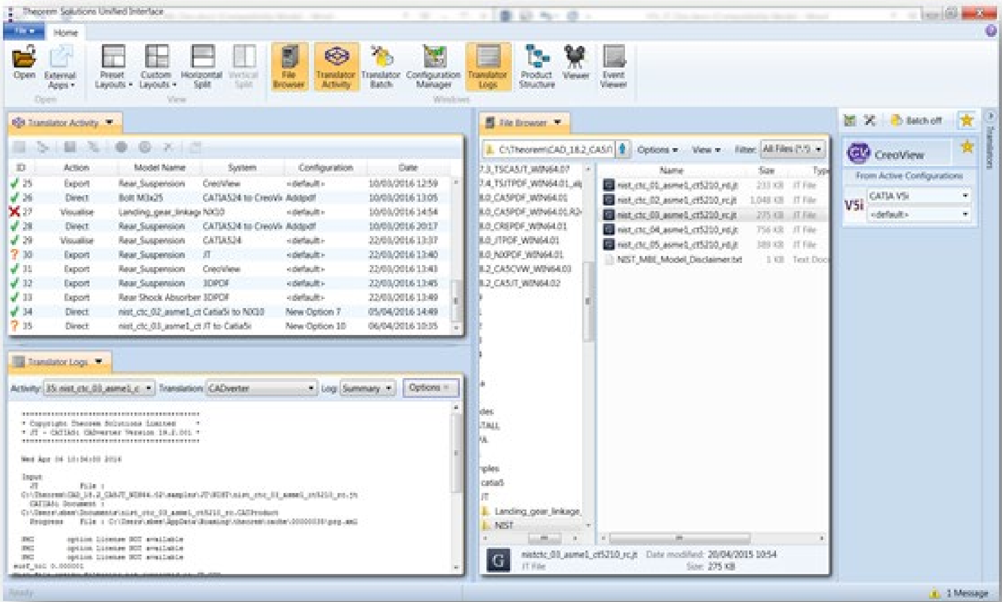
Via the Unified Interface

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

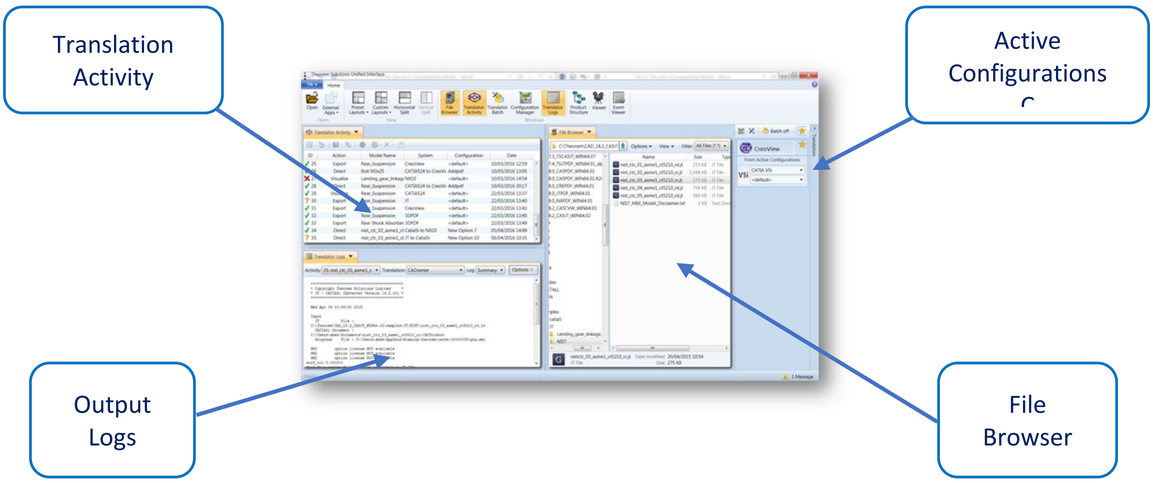
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

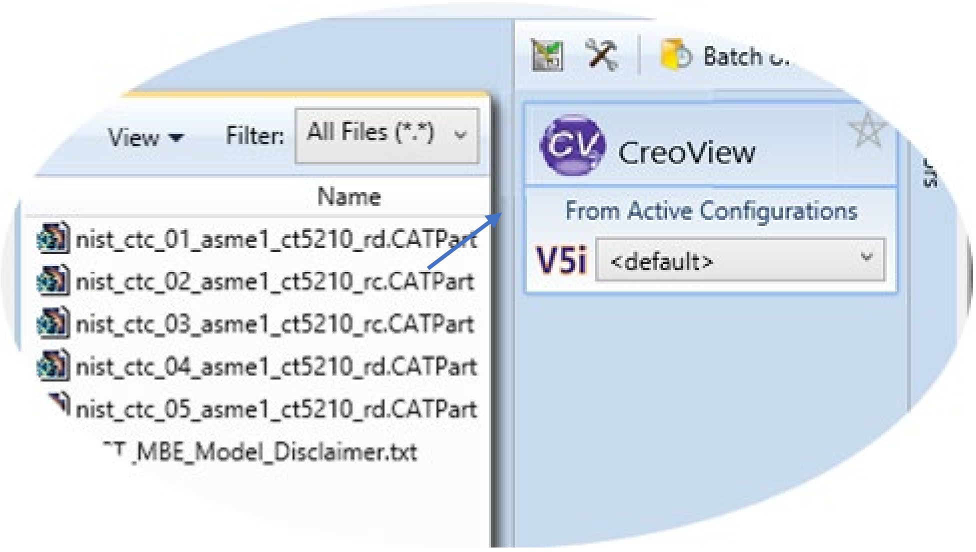
The following interface will be launched:



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

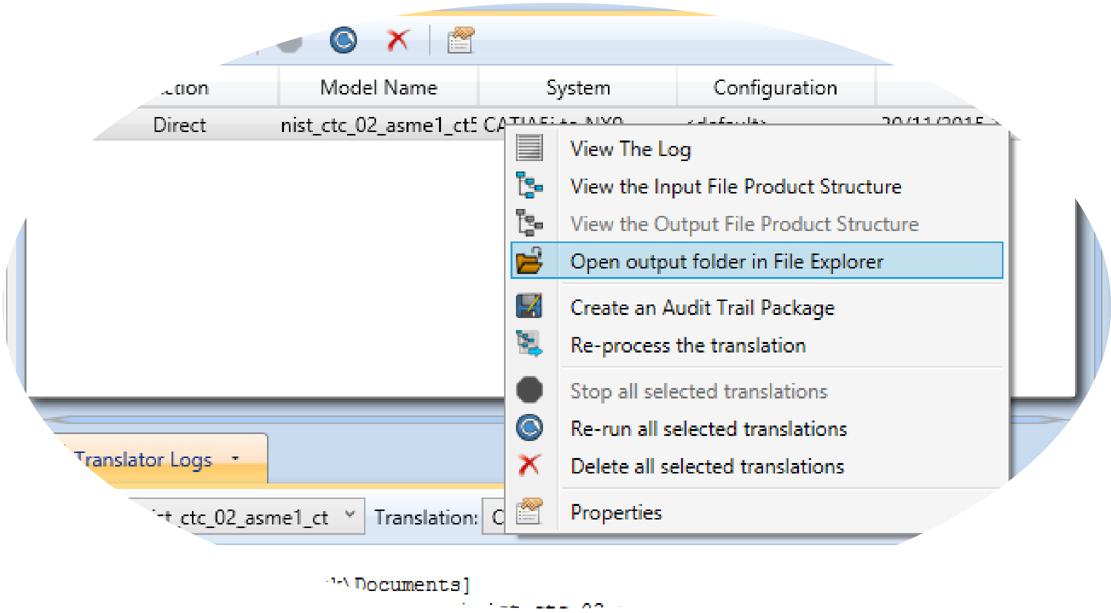


The simplest way to translate from CATIA V5i to Creo View is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



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:



Translation Configuration

It is recommended that the CATIA V5I to Creo View CADverter be run from a pre-created configuration. Theorem have adopted the standard PTC Configuration tools which will create a ***batch*** script for running the CADverter on the command line and also a ***worker*** script to allow the CADverter to be run with WindChill.

- For full details of this process, please see our 'How to Configure the CATIA V5I to Creo View via CADverter' demonstration video located here.

To take full advantage of the configuration tools and to configure the CADverter as an adapter for use as a Windchill Worker please contact your PTC representative to provide the *Windchill Installation and Configuration Guide Catia5_CreoView*.

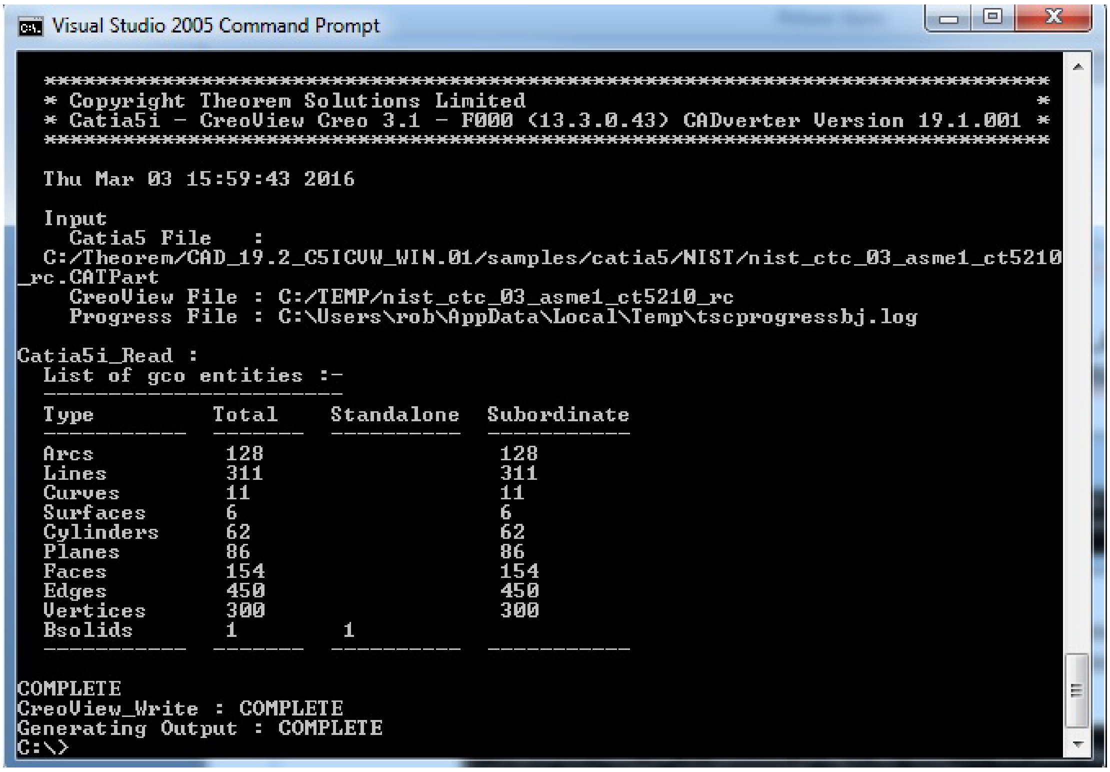
Default Translation

Default Translation – via the Command Line

Running a translation via the command line can be carried out without using a pre-created configuration. This will use the default translator settings. This is achieved by directly running the script file located in the ***<installation_directory>\bin*** directory. The format of the command is as follows:

<Translator_installation_directory>\bin\catia5i_pv.cmd <input_files> -p <output_path> -o <output_file>

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




```
*****************************************************************************
* Copyright Theorem Solutions Limited                                       *
* Catia5i - CreoView Creo 3.1 - F000 (13.3.0.43) CADverter Version 19.1.001 *
*****************************************************************************

Thu Mar 03 15:59:43 2016

Input
  Catia5 File    :
C:/Theorem/CAD_19.2_C5ICVW_WIN.01/samples/catia5/NIST/nist_ctc_03_asme1_ct5210
_rc.CATPart
  CreoView File : C:/TEMP/nist_ctc_03_asme1_ct5210_rc
  Progress File : C:\Users\rob\AppData\Local\Temp\tscprogressbj.log

Catia5i_Read :
  List of gco entities :-
  -----------------------
  Type          Total    Standalone  Subordinate
  -----------   -------  ----------  -----------
  Arcs           128                  128
  Lines          311                  311
  Curves         11                   11
  Surfaces       6                    6
  Cylinders      62                   62
  Planes         86                   86
  Faces          154                  154
  Edges          450                  450
  Vertices       300                  300
  Bsolids        1        1
  -----------   -------  ----------  -----------

COMPLETE
CreoView_Write : COMPLETE
Generating Output : COMPLETE
C:\>
```

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

C:\TEMP\nist_ctc_03_asme1_ct5210_rc

The above example provides the minimum command line arguments required to create an output. In order to support the PTC Windchill interface, Theorem have also adopted the PTC Adapter command line syntax, a full list of available options is shown below and can be displayed by issuing the following command:

<Translator_installation_directory>\bin\ catia5i_pv.cmd -h

Setting	Result
<@File>	Read Options from the response file <file>
-?-h	For basic help page. UNIX may try to export the "?" so –h should be typed in quotes, for example, "h".
-d <depth>	Set the conversion file depth. When converting an assembly file determines to what depth the hierarchy should be traversed. The default is all.
-H	For extended help options
-o<name>	Set output file base name (number of input files must be 1).
-p<name>	Set output base path.
-r<name>	Set recipe to <name>
-vc	Disable all console print-out.
-vc1	Redirect all console print-out to stdout.
-vc2	Redirect all console print-out to stderr (default)
-ve[n]	Increment or set (if[n]is given) the error reporting level. –ve0 disables all error reporting, default 1.
-vl<file>	Direct all printed output to <file>
-vL<file>	Concatenates all printed output to <file>
-vn<file>	Direct all printed output to new log file <file>-#.log.
-vp<n>	Set the process verbosity flag. List the modules by –I. Flags are listed in the source code.
-vw[n]	Increment or set the warning reporting level, see -vc
-vt	Give the current date/time stamp with all print outs.
-j<name>	Get job from <name> .paj file
-epdconnect	Enable EPD.Connect orientated conversion.
-mockup	Enable MockUp oriented conversion process

Translator Customization

In the case of the Creo View Translators, they are all configured using a recipe file, consequently the UI allows the user to point to the required recipe file for the translation settings.

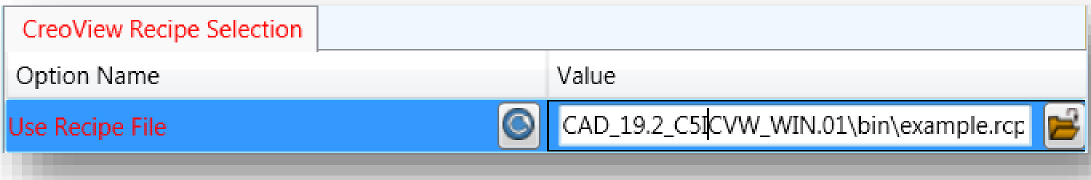
Common Options for CATIA V5i to Creo View

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

- CreoView Recipe selection – Those arguments that affect how data is read from CATIA V5. The setting up of this recipe file to achieve the desired results is explained later in this document:

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
Use recipe file	Use a specific recipe file to create the required reults ○ Command Line Syntax ▪ *-r recipe_file*

Configuring the CATIA V5I Creo View CADverter using the Recipe Editor

For completeness this section of the User Guide describes the available configuration options provided by the recipe editor.

A recipe is a set of user-defined rules that drive the individual CAD Adapter. The recipe concept provides a solution to the problem of efficiently converting CAD data into a form suitable for viewing on a wide range of computer platforms. Like its analogy in cooking, gaining a desired result requires cooking to a specific recipe. While most CAD parts will convert into an efficient form for large-scale visualization, some parts require modifications to the standard visualization recipe to be viewed effectively.

The CATIA V5i Adapter is provided with a master or default recipe file. This file is pre-configured to allow the visualization of most objects. The master recipe file should not be edited. Intead, additional new recipes can be created from this default file using Save As function in the recipe editor (***rcpedit***) provided with the translator.

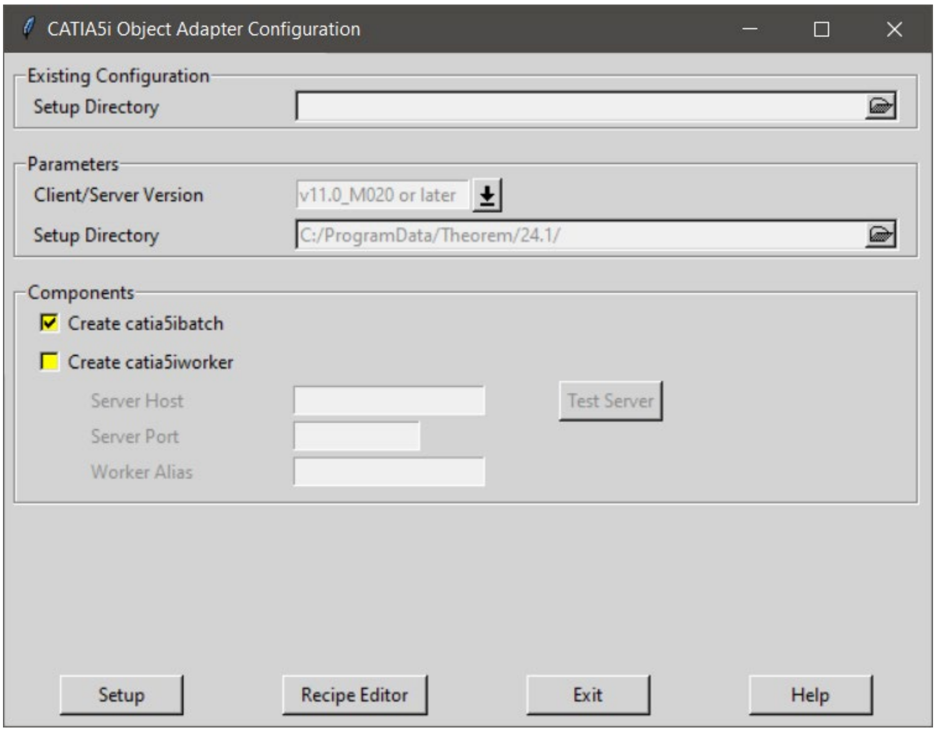
For full details concerning the Recipe Editor, please refer to the *'Creo View MCAD Adapters Installation and Configuration Guide'* document, which can be obtained via the PTC Reference Documents Site.

Theorem's Creo View CADverters use the standard PTC mechanism to Configure translation option. The basic concepts and available options are covered here for convenience.

Theorem provide a configuration script to allow a recipe file to be created. Running the following script will launch the Recipe Editor Configuration Tool:

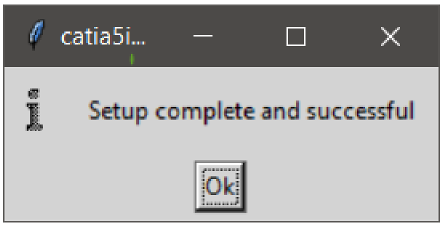
<Translator_installation_directory>\bin\catia5i_pv_config.cmd

The panel below will be displayed:



The Configuration Tool allows the CATIA V5i Creo View CADverter to be configured for use in batch *(via the command line)* and/or for use in a Windchill environment *(catia5worker).* Please contact your PTC representative to provide the *Windchill Installation and Configuration Guide Catia5i_CreoView* for full details on configuring in a Windchill environment. This guide will focus on running the translator from the command line, but all of the configuration options are available in both environments.

The ***'Create catia5ibatch'*** selection will create a recipe file for batch and the ***'Create catia5worker'*** will create a recipe file for a Windchill invocation. Having selected either of these options (and provided a valid Windchill Host and Port) the ***'Setup'*** button will become active. Selection of the 'Setup' button will launch the following panel:



This can be accepted and the ***'Recipe Editor'*** button will become active.

The ***'Setup'*** action will create a new directory beneath the translator installation directory. So, the user that creates new configurations will need write access to the translator installation directory. The first configuration directory will be named ***catia5i_setup***. Subsequent configurations will will named ***catia5i_setup'n'*** (where 'n' is a unique number). In this manner many different configurations can be created. The configuration directory will contain an invocation script that will deliver a default Configuration that uses default translation settings. Selection of the ***'Recipe Editor'*** button will allow the user to set specific translation settings.

The contents of a Configuration folder are:

Name	Date modified	Type	Size
adapter.pvi	03/03/2016 10:49	PVI File	1 KB
catia5i_pv.rcp	03/03/2016 11:50	RCP File	1 KB
catia5i_pv_config.log	03/03/2016 10:49	Text Document	0 KB
catia5ibatch.bat	03/03/2016 10:49	Windows Batch File	1 KB
debug_options.txt	03/03/2016 10:49	Text Document	1 KB
purge.bat	03/03/2016 10:49	Windows Batch File	1 KB

The ***catia5ibatch.bat*** script will be used in preference to the Theorem provided script discussed in the Default Translation – via the Command Line section and will use configuration options specified in the ***catia5i_pv.rcp*** (recipe) file.

Running a translation using the ***catia5ibatch.bat*** script can be achieved using the following command

<Translator_installation_directory>\catia5i_setup\catia5ibatch.bat <input_files> -p <output_path> -o <output_file>

The results and screen output will be the same as that noted for the Default Translation.

Configuring the Recipe File for the CATIA V5I Object Adapter

A recipe is a set of user-defined rules that drive the individual CAD translation tools. The recipe concept provides a solution to the problem of efficiently converting CAD data into a form suitable for viewing on a wide range of computer platforms. Like its analogy in cooking, gaining a desired result requires cooking to a specific recipe. While most CAD parts will convert into an efficient form for large-scale visualization, some parts require modifications to the standard visualization recipe to be viewed effectively.

The CATIA V5I Object Adapter is provided with a master or default recipe file. This file is pre-configured to allow the visualization of most objects. You should not edit the master recipe. Additional new recipes can be created from this default file using Save As function in the recipe editor (rcpedit).

The Using the Recipe Editor document describes the steps you should take to create a new recipe file for your particular configuration. Using the Recipe Editor explains the recipe concept and describes the settings available for each object adapter. A copy of the document "Using the Recipe Editor with CAD Object Adapters" can be obtained by contacting your local PTC customer support department.

Customizing Your Object Adapter Output

After installing, configuring, and testing you object adapter, you may decide that you want to further customize the output. You can customize your object adapter output by using the recipe editor. Using the Recipe Editor describes a number of specific customizations users will be interested in configuring. The list provided next is not meant to be exhaustive, but includes the following specific recipe configuration instructions:

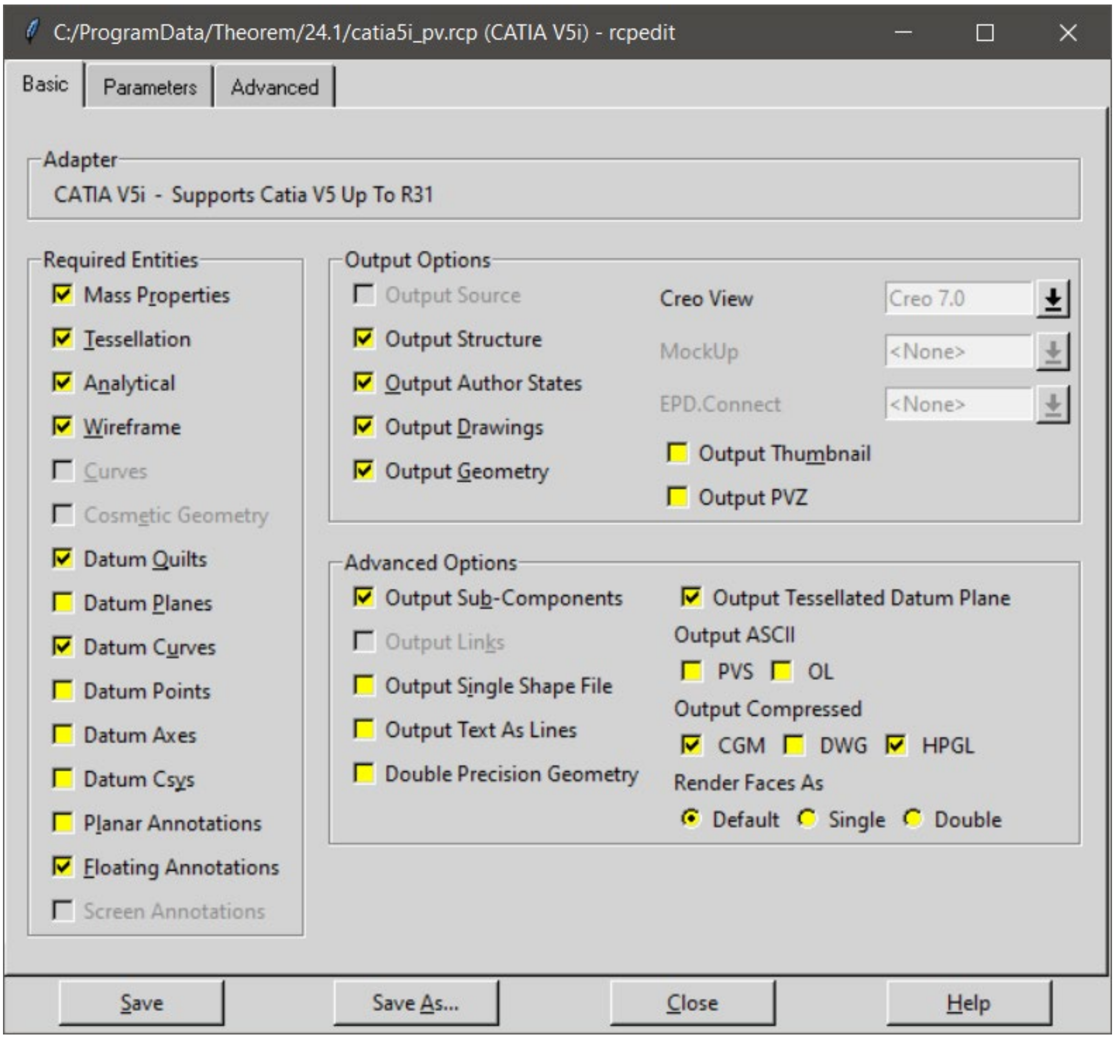
- Changing what information is published by an object adapter.
- Optimizing for viewing on Windows or UNIX platforms.
- Changing the accuracy of the visual representation.
- Fixing any conversion issue such as holes or cracks.
- Enabling or disabling CATIA V5I layer states.

The information available in Using the Recipe Editor is periodically updated. For the latest information, download the document from the PTC web-site noted above.

Using The Recipe Editor To Configure Options

The recipe editor has an advanced options tab, which contains a number of sub menus which allow the user to control the Theorem translation settings. To access these options :

1. Launch the recipe editor:
    rcpedit.exe <recipe file name>
2. The recipe editor GUI will be displayed as follows (example):



Specifying a Recipe file on the command line

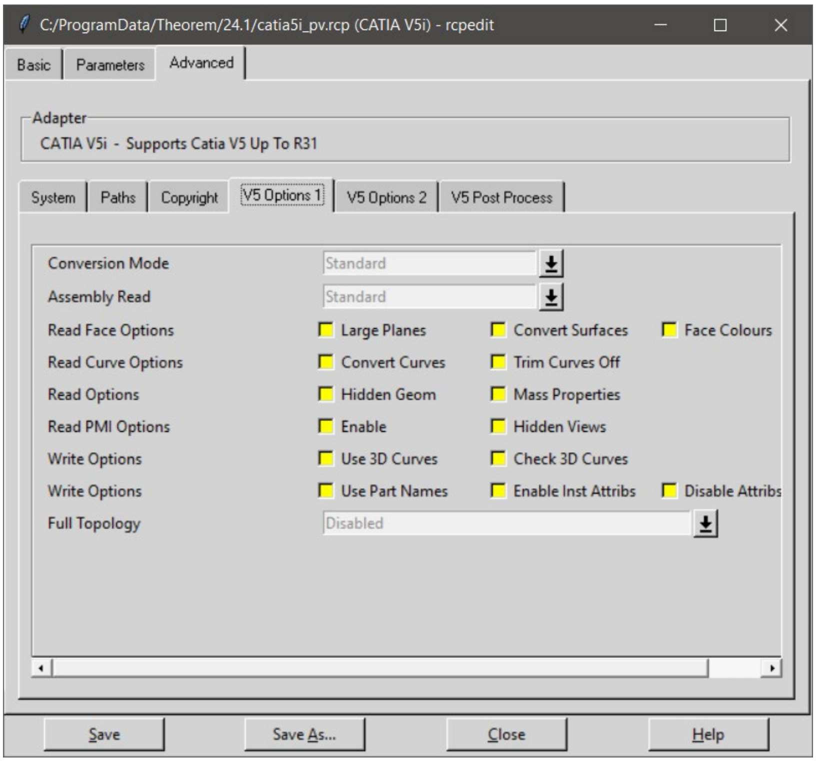
For any option set in a recipe file, once the recipe file has been saved, the translator can take advantage of the settings by referring to the recipe file on the command line:

e.g. catia5i_pv.cmd <input file> -r <recipe file name>

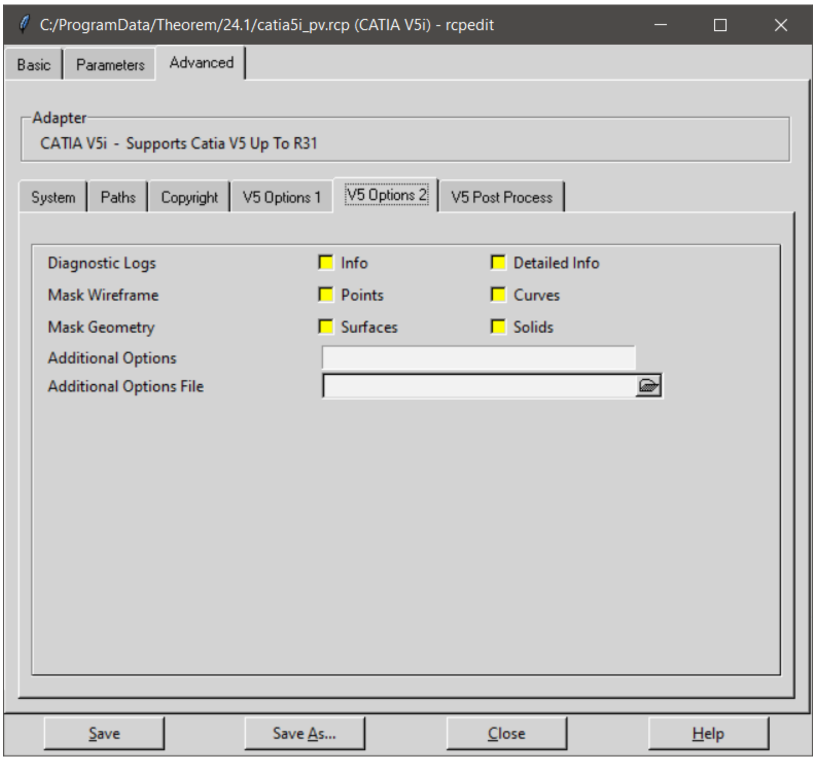
Theorem Recipe V5I Options

Select the V5I Options Tabs :

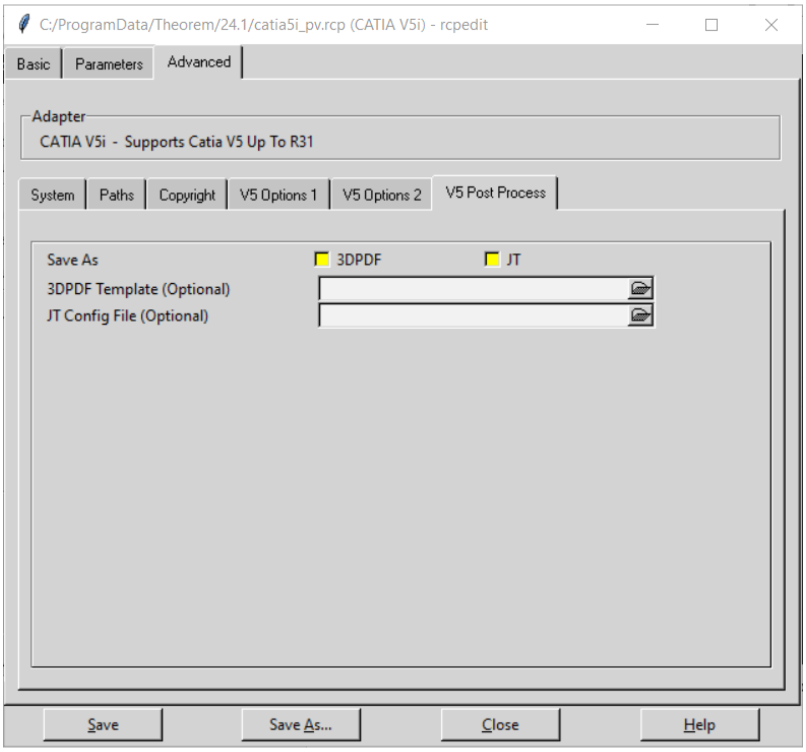
Options 1 contains the main functions.



Options 2 allows masking and additional options



Post Process allows conversion of the same data to 3D PDF or JT if you have purchased the additonal add-on packages.



A description of these commands are shown below

Option		**Description**
Conversion Mode		The user has the option to process Assembly information in one of 2 modes: • Standard • Minimum Memory *Standard:* The default method for assembly processing reads a CATIA V5I assembly and its entire geometry contents into memory, before writing out all of the data to Creo View. *Minimum Memory:* A more efficient way of processing the data has been provided via the Minimum Memory conversion mode selection. This mechanism reads in the assembly data then process each ".CATPart" file referenced by the assembly on a part by part basis. *See the section on Large Assembly Processing Best Practises for more info.*
Assembly Read		The user 3 option to process Assembly information : • Standard • Lite - Structure Only • Lite - Structure & Geom *Standard* - uses a direct read of the Catia5 data *Lite - Structure Only* - Uses a lite weight read designed to read large assemblies quickly, this setting will just read the assembly structure. *Lite - Structure & Geom* - Uses a lite weight read designed to read large assemblies quickly,followed by minimum memory processing for the parts.
Read Options	Large Planes	Allows the user to omit large PLANEs where the bounding exceeds 0.1Km in any direction. These faces are most often construction planes. The bounding box size can be adjusted by using the additional argument ***set_omit_large_planes <value in M>,*** see additional options for details.
	Hidden Geom	Allow selective data types to be read regardless of hide/show state. Default is OFF. ○ Command Line Syntax ▪ *read_hidden_geometry*
	Hidden Views	Reads any Views that are hidden

Read Options	**Convert Curves**	**Convert curves to NURBS. Default is OFF.** ○ **Command Line Syntax** ▪ ***convert_curves***
	Convert Surfaces	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*
	Trim Curves Off	Trims face surfaces. Default is ON. ○ Command Line Syntax *dont_trim_surfaces*
Read PMI	**Enable**	Read CATIA V5I PMI, such as annotations, GDT and Dimensions.
Write Options	**Use 3D Curve**	This option allows Creo View API to generate its own 2D curves. This option is most likely only ever used as a work-around when poor data is encountered.
	Check 3D Curves	This option allows the CADverter to test the data and if necessary automatically enable Use 3D Curves. A default tolerance of 0.01 (1%) face/surface overlap being used for these checks. This tolerance can be adjusted with ***validate_3D_curve_tol <value>*** in the additional option field.
	Use Part Names	Use the CATIA V5I 'part number' names for assembly nodes. The default is to use 'Instance name'.
	Instance Attributes	Enables the output of any instance attributes.
Diagnostic Info	**info**	Add information messages to the progress file
	Detailed info	Add detailed information (diagnostics) messages to the progress file

Full Topology	**Controls how the topology is written into Creo View data. By default this is 'disabled', which is consistent with earlier releases of the product. The default setting generates good viewable data, but since each face of a solid is written separately it is not possible to calculate mass properties such as volume, correctly view capping sections or perform 3D Compare or Interference Detection in the Creo View client.** **Options are:** ***Disabled*** • ***(Default setting)*** **Solid faces written with unique edges.** ***Enabled*** • ***(Solids only)*** **– Adjacent faces share edges, such that the resulting Creo View data can support mass properties etc.** • ***(Solids & Quilts)*** **– Adjacent faces share edges, this includes 'open solids', which will be written into Creo View as quilts.** **Enabling full topology can increase translation times, so we suggest that the user selects the minimum setting that required. For example, don't use the Fix non-manifold settings unless necessary and don't use the Quilts setting unless required.**
Additional Options	Not ALL options for to the CATIA V5i Creo View CADverter are made available to the recipe editor. These options are not in common use, but can be included here should they ever be required, please delimiting each option with a space " " character. Some example of additional options are detailed in a separate section :
Additional Options File	Any of the Additional Options can be specified in an Additional Options File, known as a *'Theorem Configuration File'*. Use this option to specify a configuration file to use.

Theorem Recipe V5I Filter Options

Select the V5I Filter Options Tab :

Mask Off	**Face Colours**	**Disable the processing of face colours.**
	Attributes	Disable the reading of attributes
	Points	Disable the reading of points
	Solids	Disable the reading of solids
	surfaces	Disable the reading of free surfaces
	Wireframe	Disable the reading of wirefrane

Theorem Recipe V5I Post Process

Select the V5I Post Process Tab :

Save as	**Save As PDF**	**Output PDF file as well as Creo View Data**
	Save As JT	Output JT file as well as Creo View Data
	3D PDF Template	Define a non default template (XML) file used to control the options and publish style in writing the PDF file.
	JT Config File	Define a non default config file (tess.config) used to control the options in writing the JT file.

Save As JT & PDF Files

When these options are selected, the V5I data is read, with a common set of options and then written with write options defined in the related JT configure file or PDF template file.

The default JT configuration file is found in :

%TS_INST%\etc\tessCATIA5ISaveAs.config

The recipe editor allows any JT configuration file to be used, if the default is not suitable.

The default PDF template file is found in :

%TS_INST%\data\publish_3dpdf\defaultManifest.xml

Which in itself references :

%TS_INST%\data\publish_3dpdf\publish3d_options.txt

Which defines the PDF write leg options

The recipe editor allows any PDF template to be selected.

Save As Option Licenses

NOTE: Additional license(s) are required to use save as JT and PDF

CATIA V5i to CREOVIEW Advanced Arguments

Theorem's CATIA V5i to CREOVIEW translator has been configured with default settings that optimises the translation process. However, there are times when a satisfactory result cannot be obtained, so it may be required to deploy one or more Advanced Arguments to improve the translated result. (**Using the Additional Options field in the recipe editor**)

Option	Description
Conversion Tolerance	A secondary argument to *'Convert Curves'* defining the conversion tolerance. Default is 0.00001 ○ Command Line Syntax ▪ *convert_curve_tol 0.00001*
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*
UDF Axis Systems	Enable reading of User Defined Axis systems. Default is OFF. ○ Command Line Syntax ▪ *read_udf_axis – to turn on*
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
Categorise PMI	PMI is categorised by type. Default is ON. Only selectable if *'Read PMI'* is selected. This can be switched off using ○ Command Line Syntax ▪ *disable_catagorise_pmi*

single_jt_file_in_pvoa	**If this option is selected ONLY the top level JT file is added to a .pvoa file when a job file (.paj) is processed, otherwise all subordinate part JT files are store in the .pvoa file - so that a whole assembly is stored in the .pvoa file.**
attr_filter_file *<filter file>*	A default Attribute Filter file is supplied in: ***%TS_INST%/data/creoView/defaultAttrFilter.txt*** The default locations can be overridden by specifying a value to the attr_filter_file command. Filtering can be disabled by specifying a non-existing file with 'attr_filter_file' OR deleting the defaultAttrFilter.txt The essential settings for an attr_filter_file are: ***Attribute Name; New Attribute Name; Mode*** Where: Mode = 0 – omit named attribute from Mode = 1 – Rename to New Name
set_omit_large_planes *<value>*	Omit large PLANEs greater than the tolerance value *(default is 100 M)* e.g. ***set_omit_large_planes 2000*** - sets a value of 2km
disable_opacity	Disable the writing of opacity settings into Creo View data
pmi_RGB *<rrr-ggg-bbb>*	Set a default colour for PMI text and graphics, this will override the colours read from CATIA V5. The argument rrr-ggg-bbb, MUST be given as 3 values 000 to 255 for each of the colours with a '-' char between, for example: ***pmi_RGB 000-000-000*** – for black text ***pmi_RGB 255-255-255*** – for white text
Mass Properties	CATIA V5I 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*

Assembly Processing

Processing CATIA V5I Assemblies (.CATProduct files)

Assuming that the input to the CADverter was a single assembly named ***test_assembly.CATProduct*** related to many subordinate parts (.CATPart) files then the output from the translator will be a single Creo View assembly file ***test_assembly.pvs*** plus many geometry .ol files, one for each part file processed.

Given that the example assembly file had additional assembly files subordinate to it then all of the accumulated assembly hierarchy information would be output into the top level Creo View .pvs file.

The CATIA V5I Creo View CADverter takes advantage of the latest Creo View dAPI which writes .pvs files (Creo View binary assembly structure files) by default. If a user wishes to write out earlier .pvs versions or .ol files, this can be achieved via the appropriate setting the recipe editor.

Processing CATIA V5I Parts (.CATPart files)

For each part (.CATPart) file processed individually then the output from the translator will be a single Creo View assembly .pvs file and a single geometry .ol file. Therefore assuming that the file being processed was named ***test_component.CATPart*** then the output would be ***test_component.pvs*** and ***test_component.ol***

Efficient Large Assembly Processing

If an assembly is opened using the CATIA V5I application or API (CAA), all subordinate assembly and all related geometry will be loaded into memory. This is very inefficient in terms of memory usage.

Theorem have made improvements to assembly read efficiency in the following areas:

- Each .ol file is now written and all associated memory freed on a file by file basis, such that in writing to Creo View there is no build up of memory.

- Although the translation process is constrained by the CATIA V5I API reading all assembly and geometry information into memory, it is now possible to write out each geometry and assembly file on a file by file basis. This functionality can be invoked by either selecting the the “minimum memory” conversion mode option.

The combination of the 2 efficiency improvements detailed here provides in excess of a 20% memory saving on larger (>50MB) assemblies. In general the larger the input assembly the greater the saving will be.

Large Assembly Processing Best Practices

Processing large CATIA V5I assemblies often requires access to large amounts of resources. This can be a combination of both available disk space as well as memory resources. As described above, the default method that the CATIA V5I API uses to consume assembly data compounds this issue. Therefore the following steps are advised as best practices for working with large assemblies:

1. Use the Minimum Memory Mode recipe setting described in section "Running In Minimum Memory Mode"

   For further assistance with this setting, please refer to your CATIA V5I on-line help.

Minimum Memory Mode for Very Large Assemblies

Additional Large Assembly read options are provided that may provide some benefits, particularly in Windchill implementations that deploy Positioning Assemblies.

These options should only be considered for use having liaised with your PTC Windchill support team.

Error Tracking and Management

A method of tracking and managing errors output from the CATIA V5I to Creo View process has been provided. This is implemented by setting exit status codes from the CADverter and additionally the creation of a summary file for each translation task. The structure of the summary file enables detailed analysis of the translation task to be verified.

CADverter Exit Status Codes

The software will return one of the following exit status codes:

0 = Translation completion without errors
1 = Translation completed with errors

These codes will be returned regardless of the type of data being processed, either single parts or assemblies. If the error code returned is 1 (e.g. Completed with errors) the user will be directed to look at a summary file that details the exact reason for failure.

Summary File Definition

Each translation creates a summary file using the standard name “tscsummarybj” located in the temporary directory. The user can override the default name using the environment variable TSC_SUMMARY_FILE.

The name of the active summary file is recorded in the progress file:

WINDOWS default name=%TEMP%\tscsummary

Output is recorded in the summary file with a single line reporting a status for each item processed. Each line is defined using 4 fields, separated by a “,” character. Each field represents the following data:

Field 1 = Input File Name
Field 2 = Error Code *(See Summary File Error Codes)*
Field 3 = Error Description *(See Summary File Error Codes)*
Field 4 = Progress File name

e.g. C:\myparts\sample.CATPart,0,Completed with no errors,/usr/data/sample.CATPart.log

When processing either single parts or assemblies using the default recipe file settings, only one line will appear in the summary file. However for assemblies processed using the minimum memory mode methodology or with links enabled the summary file will contain a line for each “.CATPart” and “.CATProduct” file translated.

Summary File Error Codes

The following Error Codes are output the Summary File:

0 = Completed with no errors
-1 = Command line syntax error
-2 = Licensing Error
-3 = Input File Not Found

-4 = Failed to Open Progress File
-6 = General Read Error
-7 = General Write Error
-8 = No entities Found
-10 = Failed to open CATIA V5I file
-12 = Solid validation error
-13 = Some solid degradation
-14 = One or more faces omitted
-15 = One or more geometry files not found in an assembly

Worker Logs

The Adapter writes key messages to the PTC worker logs, these include the summary error codes (positive values are used in these logs, e.g. 3 = input file not found).

The Theorem messages added to the worker logs are always prefixed by ***'TS:'***, for clarity, and are written at two levels of detail 0x01 and 0x10. These messages are enabled via the –vm command line with the correct bit mask level for logs required.

e.g. ***–vm 1 will enable all 0x01 messages***
-vm 11 will enable all 0x01 AND 0x10 messages.

The –vL <log file> command line can be used to re-direct these messages to a file.
In the event of an error the summary code will be written to the worker log he positive value of the summary code :
e.g.
8 => No entities found.

Process Timeouts

Timeouts allow a user to control when an individual translation invoked from a WindChill environment should timeout.

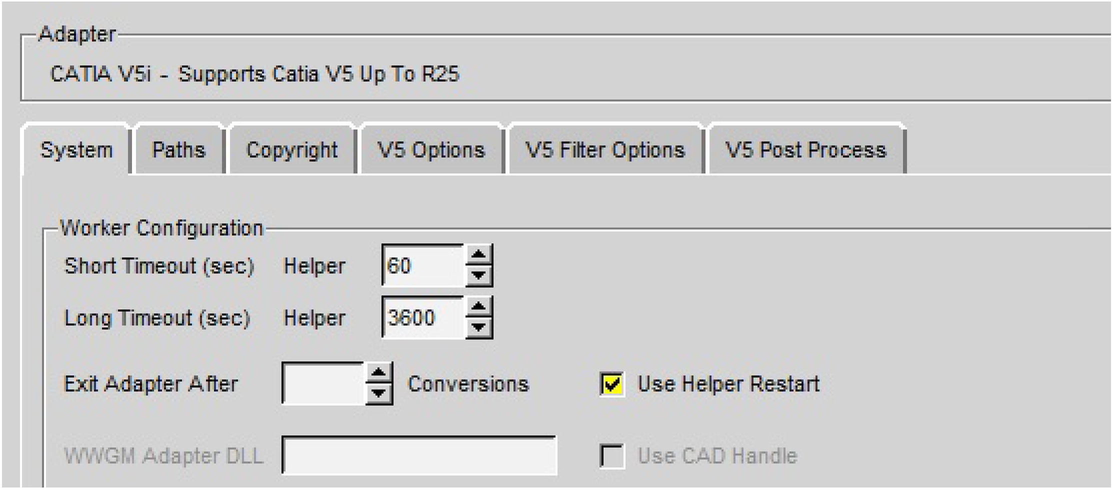
The WindChill interface allows 2 distinct timeout types to be defined, Long and Short.

Three simple timeouts have been allocated to the CATIA V5->Creo View translator, one using the Short Timeout value and two using the Long Timeout value setting:

Long Timeout – ***Catia5i_Read***, to read each file from CATIA V5.
Long Timeout – **Creo*View_Write,*** to write each file into Creo View

The ***Timeout* values** can be set as follows:

1. Launch the recipe editor: **rcpedit.exe *<recipe file name>***
2. Select the Adapter Tab
3. Having selected CATIA V5I from the adapter pull-down, followed by the System Tab, the recipe editor GUI will be displayed as follows:



Apply an appropriate time for the Short and Long timeout (in seconds).

Note! These times are totally dependent upon the user data. Some trial and error may be required to define the best times for a specific user environment.
It is recommended that timeouts should NOT be used when running the software with its default settings. They should only be deployed when running translations in minimum memory mode or with links enabled. This will allow the LONG timeout setting to be reduced, as using these options will instruct the translator to process assemblies one part at a time.

Appendix A – Theorem Support Advanced Options

Introduction

The following environment variables are available to modify the CADverters behaviour under the guidance of the Theorem support team.

It is recommended, under guidance from Theorem Support that these variables be set in the ***ts_env.bat*** file, if required.

Where no value is suggested, set the variable with a value of 1:

e.g.

set TSC_DEBUG=1

Diagnostics

Variable	Value	Description
TSC_DEBUG	1	Annotate logs and screen output with debug data Note! Much of the TSC_DEBUG info is now re-directed via the ***–vm <level>*** command line option into the worker log
TS_V5I_CVW_KEEP_VWR_FILE	1	Retain any intermediate GCO files

Testing

Variable	Value	Description
TS_TEST_WORKER_MODE	1	**Simulate worker mode**
TS_TEST_POSITIONAL_ASSY	1	Simulate Positional Assembly Mode In WindChill
TS_POSITIONAL_ASSY_DEPTH	1..n	Set a depth for Positional Assembly default is 1

3D PDF Publish

Variable	Value	Description
TS_V5_CV_SAVE_3DPDF_PUBLISH_MODE_OFF	1	**Disable 3D PDF publish mode, create 3D PDF file without publish template**

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