

# (CATIA V5 Adapter)

Product Release Version 26.0



# **USER GUIDE**

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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:



#### TRANSLATE

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

See our <u>website</u> for more detail.



#### PUBLISH

The creation of documents enriched with 3D content

See our <u>website</u> for more detail.



#### VISUALIZE

Visualization for <u>Augmented (AR)</u>, <u>Mixed (MR)</u> and <u>Virtual (VR)</u> Reality applications

See our <u>website</u> for more detail.





#### The CATIA V5 Creo View Adapter

Theorem's Visualize 3D product for CATIA V5 to Creo View is also known as The CATIA V5 Adapter for Creo View.

It is a direct data converter from CATIA V5 to Creo View, PTC's visual collaboration product. The Adapter rapidly and accurately publishes 3D mechanical design geometry parts, assemblies and 2D drawings, together with attribute information to the compact formats used by the Creo View application.

Designed to be compatible with other PTC Creo View Adapters, the Adapter can be integrated into Windchill, DIVISION Graphics Server or other PDM environments.

The Adapter directly accesses native CATIA V5 parts, assemblies and drawing files using the Dassault Systemes supported programming interface. Assembly structure details and geometry colour information is retained during translation. Output file characteristics are configured using the standard PTC Recipe File Editor.

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

#### **Primary Product Features**

- Converts CATPart geometry including analytic data, solid models, and surfaces to the Creo View ".ol" file format
- Converts CATProduct assembly structure and part orientations to the Creo View ".pvs" file format
- Converts attribute data such as meta-data, colour and layer information and CATIA V5 properties
- Retains accuracy of data in Creo View allowing accurate measurements, sections etc.
- Converts CATDrawings to various 2D formats e.g DXF &TIF for viewing within Creo View
- Configuration compatible with other PTC Creo View Adapters
- Optional integration with Windchill, DIVISION Graphics Server as a standard "Worker" application. Also able to be integrated with other PDM environments
- Operates in both command-line and batch modes
- Configurable to output earlier versions of Creo View data
- Able to be initiated from within CATIA V5 menu system
- The Creo View data created using this Adapter can be imported into the Arbortext IsoDraw CADprocess





Primary Product benefits?

- Direct conversion from CATIA V5 to Creo View reduces processing time, simplifies integration and retains accuracy of the model
- Improved communication and collaboration by visualizing CATIA V5 data in Creo View across the enterprise
- Reduce costs and risks associated to accessing the wrong version of data by integrating the publishing process into all related business processes
- 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 CATIA V5 Creo View Adapter. For information regarding any of Theorem's product ranges please contact <a href="mailto:sales@theorem.com">sales@theorem.com</a>





# CATIA V5 to Creo View Adapter Product Modules

The following product modules are available for the Creo View Adapter:

#### Standard Product

The standard product functionality is detailed in this user guide and is available for use in a batch environments.

#### Standard Product for Windchill

The standard product for Windchill functionality is detailed in this user guide and is available for use in a batch and Windchill environments.

The following Service Modules require a prerequisite *'Standard Product for Windchill'* product baseline:

#### CATIA V5 to Creo View PMI Service Module

The PMI module allows PMI, Captures and View states to be read from CATIA V5 and written into Creo View.

#### CATIA V5 to Creo View Post Processing Service Module

The Post Processing module allows alternative formats ("Additional File Types") to be output alongside the standard Creo View output, via a single invocation. In addition bespoke post processing activities can be launched using this method.

#### CATIA V5 to Creo View JT Add On Service 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 V5 to Creo View 3D PDF Add On Service 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.

#### CATIA V5 to Creo View Composites Add On Service Module

The Composites Add On module allows composites data from the specification tree to be created alongside assemblies and parts processed into Creo View. This requires a supporting CATIA composites license.









### What's new in 24.0

#### Flexible Assembly Visualization

This release allows the end-user to make use of Flexible Assembly Visualization in Windchill. This behaviour is turned on by default in the CADVerter and flags are set that are activated in Windchill

Support for Contextual Parts contained in Flexible Assemblies con iare not supported in this revision of CADVerter

Flexible assembly shown in CATIA V5	Fine Proper (Product: 1) Finger (Product: 2) Finger (Proger: 2)	A A A A A A A A A A A A A A A A A A A
Display in Creo View when value for "Flexible assembly features in Dymanic CAD Structures " is set to "No"	<ul> <li>(a) Construct A 1</li> <li>(b) Construct A 1</li> <li>(c) Frage (ATPender, A)</li> </ul>	ted
Display in Creo View along with attributes, when value for "Flexible assembly features in Dymanic CAD Structures " is set to "Yes"	Image: The second sec	Project/SPredent         Project/SPredent           Residence         Notal Network           Notal Network         Notal Network





#### Multi Body Components

Multi-body components are now supported in Creo View and CADVerter takes advantage of this. The expand\_part functionality will be replaced by this Multi-body support and is turned on by default. If this is not required it can be turned off in the recipe file.

🍌 new.rcp (CATIA V5) - rcpedit				
Basic Parameters Advanced				
-Adapter CATIA V5 - Supports Catia V5 version	s R26 to R30			
extitation supports catta vo version	3 120 10 100			
System Paths Copyright V5 Option	ons 1 V5 Options 2 V5	Drawing V5 Post Proces	\$	
Conversion Mode	Minimum Memory	Ŧ		
Read Face Options	Omit Large Faces	Convert Surfaces	Face Colours	
Read Curve Options	Convert Curves	Use 3D Curves	Check 3D Curves	
Read Options	F Hidden Geometry	Read Axis	Mass Properties	
Read PMI Options	PMI + Captures	🔽 Hidden Geom+PM	I	
Read PMI Options	Omit Captures	Read Views		
Write Options	Use Part Names	Disable Inst Attribs		
Write Options	Animations	Empty Parts	Datum Planes	
Write PMI Options	Flip Text In View	Flat To Screen	None Filled	
Multi Bodies	Enabled		<u>±</u>	_
Full Topology	Disabled		±	
Capture Views	Enabled - Geom mask	ON - Local Space	±	
Process Specification Tree	Disabled		±	

#### Each body will be shown with a new icon *and* can be individually addressed







#### View State Comparison

A new Creo View tool, View State Comparison, will start to be supported by CADverter 23.4 and Creo View 7.1.

It allows the PMI in each View State (CATIA Capture) to be compared, when two revisions of the same part are open.

A simple visual report of the two part revisions is clickable to allow each modification to be examined in more detail.

Further details of the View State Comparison tool can be found in the PTC Creo View documentation.







# What's new in 24.2

#### Tesselation now uses default bounding box functionality

This release has moved the default tessellation strategy closer to that of PTC. The tessellation is based upon the solid and surface geometry bounds. In tests most cases have better or equal results than the previous revision. If there is any degradation in the tessellation in your particular case, please contact the support desk who will be able to help.

#### Materials and mass properties are read by default.

Materials and mass properties are now read by default when available. This will give more accurate mass property results in Creo View for single body and multibody components. Material properties only include colours but not texture maps. i.e. a material such as gold will have its colour mapped, whereas a material texture such as honeycomb will not.

#### Material visual properties (colour, opacity) are read.

Inheritance is also in place for multi-body components. At present the translator assumes the CATIA default, which is to propagate the material down to the lower level bodies. See examples below



In the above example because Red Wood has a texture map applied then the translated output will be the default body colour. However the Part level mechanical properties will be correctly washed down to the lower level bodies.







If a material with a colour, such as 'Fire Red' is applied then the colour will be translated as well as the mechanical properties.

#### Recipe File Changes

The Mass properties check box has now been removed from the V5 Options 1 tab of the Recipe Editor as this is now the default.





new.rcp (CATIA V5) - rcpedit asic Parameters Advanced Adapter CATIA V5 - Supports Catia V5 ver System Paths Copyright V5 (	sions R26 to R31 Options 1 V5 Options 2 V5	Drawing V5 Post Process	
Conversion Mode Read Face Options Read Curve Options Read Options Read PMI Options Read PMI Options Write Options Write Options	Minimum Memory Omit Large Faces Convert Curves Hidden Geometry PMI + Captures Omit Captures Use Part Names Animations	Convert Surfaces Convert Surfaces Use 3D Curves Read Axis Hidden Geom+PMI Read Views Disable Inst Attribs Empty Parts	Face Colours Check 3D Curves
Write Options Write PMI Options Multi Bodies Full Topology Capture Views Process Specification Tree	Datum Planes     Flip Text In View     Enabled     Disabled     Enabled - Geom mask     Disabled	Flat To Screen	None Filled





What's new in 25.2

CATIA V5-6 R2022 support CATIA V5-6 R2022

CGR Structures

CGR structures are now displayed consistently in both Standard and Minimum Memory mode

Enhancements to Material properties There is now a button to override the material color rendering







## What's new in 25.4

#### Windchill Smart Platform Support: Locator Conversion

Starting from THEOREM 25.4.01 and WWGM 12.1.2.0 (Narwhal), it is possible to expose Locators information from CATIA V5 Adapter to Windchill for positioning visualization. These locators are used to position model variants within an assembly in Windchill.

In CATIA V5 terms a Locator is a 'named' Axis System e.g. as shown below 'LEFT\_MOUNTED' or 'RIGHT\_MOUNTED'



The process is then activated by setting two 'Filters'

#### <u>File Filter</u>

This allows the user to limit the parts which the translator will read to search for locators. This filter only applies to top level CATParts. The translator will not attempt to search lowerlevel parts for locators. This is important if the data is large with a flat structure as without this limitation many parts without locators will be read.





- The filter will allow a single asterix anywhere in the root string e.g. Skel\* will find both Skeleton.CATPart and Skeleton\_loc.CATPart or Skel\*loc will only find Skeleton\_loc.CATPart.
- Multiple parts can also be specified but these must be the full name e.g. Skeleton,Loc but not Skel\*,\*Loc.

#### File Locator Filter

- Multiple definitions can be in a comma separated list e.g., \*TOR,L\*3
- This will now allow a single asterix anywhere in the root string e.g., LOC\*, **\*TOR or** L\*R will find a locator called LOCATOR.

Recipe File Changes

The two filters have been added to the recipe file as shown below.

🚯 C:/ProgramDa	ta/Theorem/25.4	/catia5_pv.rcp (CATIA V5) -	rcpedit	_		$\times$
Basic Parameter	s Advanced					
Adapter CATIA V5 - Su System Paths	pports Catia V5 v	rersions R29 to R32 5 Options 1 V5 Options 2	V5 Drawing V5 Post Process			
Diagnostic Lo Mask Wirefra Mask Geome Mask Layers Locator Filter Locator Part I Process Arch Surface Chec Additional Of Additional Of	ngs me try Filter tecture king otions otions File	Linfo Points Surfaces Surfaces Disabled	Detailed Info     Curves     Solids	<u>₹</u>		
<u>S</u> ave		Save <u>A</u> s	E <u>x</u> it		<u>H</u> elp	

There are currently some known limitations.

- It is not possible to convert axis systems to locators using Minimum Memory mode.
- Publishing fails for a Product that contains an empty (without sketches or features) skeleton part.





• Users should set Additional Options to NOSHOW\_AXIS (see below) on Worker Server to be able to expose hidden axis systems or axis systems with hidden references.

Process Architecture	Split Process	
Surface Checking	Disabled	Ŧ
Additional Options	NOSHOW_AXIS	
Additional Options File		

- Locator Filter and Locator Part Filter options do not recognize multiple " \* " wildcard characters in their values.
- Deactivated axis system can be exposed to Windchill if its value matches the Locator Filter option.

#### What's new in 25.4

#### Quick View

QuickView for large assemblies is now available. Please refer to PTC article <u>CS381237</u> for further information.





## 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 videos.

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.



To install the translator, select the *right click* the msi package file and select install. More information can be found in the 'Installation and Configuration Guide' located <u>here</u>.

#### 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 'Installation and Configuration Guide' located <u>here</u>.





#### 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 <u>Appendix A</u> of this document.

There are 2 distinct ways of running the translator:

- Via the Command Line
  - The Command Line Interface provides a direct method of invoking the translator. It can be used directly on the command line or called via a third party application as part of a wider process requirement. Full compatibility with PTC's Windchill Visualization Service (WVS) is supported and is detailed in PTC's *Windchill Installation and Configuration Guide Catia5\_CreoView*.
  - For a full guide to this process, please see our 'How to Translate CATIA V5 to Creo View via the Command Line' later in this document
- Interactively from within CATIA V5
  - The Interactive Interface provides a direct method of Translating CATIA V5 data to Creo View from within CATIA V5 itself.
  - For a full guide to this process, please see our 'How to Translate CATIA V5 to Creo View via Interactive mode' later in this document





### Using the Product

#### Translation Configuration

It is recommended that the CATIA V5 to Creo View Adapter be run from a pre-created configuration. Theorem have adopted the standard PTC Configuration tools which will create a *batch* script for running the Adapter on the command line and also a *worker* script to allow the Adapter to be run with Windchill.

 For full details of this process, please see our 'How to Configure the CATIA V5 to Creo View via CADverter' later in this document

To take full advantage of the configuration tools and to configure the 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 (Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for V5 R24, change to 24):

#### <Translator\_installation\_directory>\bin\ catia5rXX\_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:





option license available FCHK 1A I N Copyright Theorem Solutions Limited \* Catia5 - CreoView Creo 3.0 - M030 (13.2.30.38) CADverter Version 18.2.002 \* Tue Jun 02 18:23:48 2015 Input Catia5 File : C:/Theorem/CAD\_18.2\_CA5CUW\_WIN64.02/samples/catia5/NIST/nist\_ctc\_02\_asme1\_ct5210\_rc.CATPart CreoView File : C:/Output/nist\_ctc\_02\_asme1\_ct5210\_rc Progress File : C:\Users\mark\AppData\Local\Temp\tscprogressyj.log CATIA U5 Library Ver. : 18.2.002 CATIA U5 Build Release: B24.SP0 64 \_Read 1 V5 ( 1 V5 1 Read US Cache Mode : Disabled US Env Name : TheoremCatia5R24 US Env Dir : C:\Users\mark\AppData\Roaming\DassaultSystemes\CATEnv US Saved Version: U5R21 SP3 BD 01-11-2012.20.00 atia5\_Read COMPLETE List of gco entities :-Standalone Туре Total Subordinate Points 8 452 8 Arcs Conics Lines 452 4 688 368 4 689 1 Curves Surfaces 368 ones ylinders lanes Faces Edges Vertices Bsolids 1 eoView\_Write ... eoView\_Write : COMPLETE

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

#### C:\Output\nist\_ctc\_02\_asme1\_ct5210\_rc.pvs

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:

Setting	Result
<@File>	Read Options from the response file <pre></pre>
-?-h	For basic help page. UNIX may try to export the "?" so –h should be typed in quotes, for example, "h".
-d <depth></depth>	Set the conversion file depth. When converting an assembly file determines

#### <Translator\_installation\_directory>\bin\ catia5rXX\_pv.cmd -h





	to what depth the hierarchy should be
	traversed. The default is all.
-H	For extended help options
-o <name></name>	Set output file base name (number of
	input files must be 1).
-p <name></name>	Set output base path.
-r <name></name>	Set recipe to <name></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></file>	Direct all printed output to <file></file>
-vL <file></file>	Concatenates all printed output to
	<file></file>
-vn <file></file>	Direct all printed output to new log file
	<file>-#.log.</file>
-vp <n></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></name>	Get job from <name> .paj file</name>
-epdconnect	Enable EPD.Connect orientated
	conversion.
-mockup	Enable MockUp oriented conversion
	process





### Configuring the CATIA V5 Creo View Adapter using the Recipe Editor

Full details of how to Configure the CATIA V5 Creo View Adapator is contained later in this document

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 V5 Adapter is provided with a master or default recipe file. This file is preconfigured 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 at <u>https://www.ptc.com/appserver/cs/doc/refdoc.jsp</u>. Theorem's Creo View Adapters use the standard PTC mechanism to Configure translation options. 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\catia5\_pv\_config.cmd

The panel below will be displayed:

xisting Configuration			
Setup Directory			
arameters			
Client/Server Version	v11.0_M020 or later		
Setup Directory	C:/Program Files/Theorem/22.1/catia5_setup/		
Catia5 Version	29 🛓		
Components			
Create catia5batch			
Create catia5worker			
Server Host	Test Server		
Server Port			
Vindchill Workgroup Mana	der		
Enable WWGM file syn	chronization support		
Install Directory		 	
File Sync DLL	x86e_win64/obj/catia5adapter_loader.dll		

The Configuration Tool allows the CATIA V5 Creo View Adapter 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 Catia5\_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 catia5batch'** 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 **catia5\_setup**. Subsequent configurations will will named **catia5\_setup'n'** (where 'n' is a unique number). In this manner many different configurations can be created. The configuration directory 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:



The *catia5batch.bat* script will be used in preference to the Theorem provided script discussed in the <u>Default Translation – via the Command Line</u> section and will use configuration options specified in the *catia5\_pv.rcp* (recipe) file.

Running a translation using the *catia5batch.bat* script can be achieved using the following command

#### <Translator\_installation\_directory>\catia5\_setup\catia5batch.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.





Changing the translation options in the configuration recipe file is achieved by selecting the 'Recipe Editor' button.

This action will display a number of panels that are of interest to the Theorem CATIA V5 Creo View Adapter.

Mass P	P <u>roperties</u> ation ical	Output Source     Output Structure     Output Author States	Creo View MockUp	Creo 4.0	Ŧ	
<u>T</u> essell A <u>n</u> alyti <u>W</u> irefra	ical	Output Structure     Output Author States	MockUp	(Maria)		
A <u>n</u> alyti	ical	Output Author States		<none></none>	*	
Wirefra		<u>- output Author States</u>	EPD.Connect	<none></none>	+	
_	ame	Output <u>D</u> rawings			<u> </u>	
<u>C</u> urves	;	Output Geometry	Cutput Inume	nali		
Cosme	tic Geometry		Uutput PV2			
Z Datum	<u>Q</u> uilts	Advanced Options				
Datum	<u>P</u> lanes	Output Sub-Components	Output Tesse	llated Datum Plane	2	
Z Datum	C <u>u</u> rves	Cutput Lin <u>k</u> s	Output ASCII			
Datum	Points	Output Single Shape File	Output Compress	ed		
Datum	Axes	Output Text As Lines		WG 🔽 HPGL		
Datum	Cs <u>y</u> s	Double Precision Geometry	Render Faces As			
Planar	Annotations		🖲 Default 🧧 S	ingle 🧧 Double		
Eloatin	g Annotations					
Screen	Annotations					
ocreen						

The main page provides standard PTC options that we will not discuss in this document, as these are well documented in PTC's *'Creo View MCAD Adapters Installation and Configuration Guide'* document, which can be obtained via the PTC Reference Documents Site at <u>https://www.ptc.com/appserver/cs/doc/refdoc.jsp</u>.

Instead, we will focus on the Theorem specific settings that effect the output. These are accessible via the *Advanced Tab* and are grouped into 4 sub pages:

- V5 Options 1 General translation settings (including PMI)
- V5 Options 2 more translation settings





- V5 Drawing Options relating to CATDrawing processing
- V5 SaveAs Options relation to the post process and SaveAs translation behaviour

V5 Options 1 – General translation settings (including PMI)

Basic Parameters Advanced			
Adapter			
CATIA V5 - Supports Catia V5 versi	ons R26 to R31		
System Paths Copyright V5.0	ptions 1 V5 Options 2 V5	Drawing V5 Post Process	s
Conversion Mode	Minimum Memory	Ŧ	
Read Face Options	Omit Large Faces	Convert Surfaces	Face Colours
Read Curve Options	Convert Curves	Use 3D Curves	Check 3D Curves
Read Options	F Hidden Geometry	Read Axis	
Read PMI Options	PMI + Captures	Hidden Geom+PM	l i i i i i i i i i i i i i i i i i i i
Read PMI Options	Omit Captures	Read Views	
Write Options	🔽 Use Part Names	Disable Inst Attribs	
Write Options	Animations	Empty Parts	
Write Options	Datum Planes		
Write PMI Options	Flip Text In View	Flat To Screen	None Filled
Multi Bodies	Enabled	Enabled	
Full Topology	Disabled		±
Capture Views	Enabled - Geom mask	Enabled - Geom mask ON - Local Space	
Process Specification Tree	Disabled		Ŧ

#### Each of these options is described below: Option Description

Conversion ModeThe user has the option to process Assembly information in one of 2 modes: <ul><li>Minimum Memory</li><li>Standard</li></ul> <li>Minimum Memory: The default method for assembly processing and a more efficient way of processing the data is provided via the Minimum Memory conversion mode selection. This mechanism reads in the assembly data then processes each ".CATPart" file referenced by the assembly on a part by part basis. Standard: This reads a CATIA V5 assembly and its entire geometry contents into memory, before writing out all of the data to Creo View. See the section on Assembly Processing Best Practises for more info.</li>		
	Conversion Mode	<ul> <li>The user has the option to process Assembly information in one of 2 modes: <ul> <li>Minimum Memory</li> <li>Standard</li> </ul> </li> <li>Minimum Memory: The default method for assembly processing and a more efficient way of processing the data is provided via the Minimum Memory conversion mode selection. This mechanism reads in the assembly data then processes each ".CATPart" file referenced by the assembly on a part by part basis.</li> <li>Standard: This reads a CATIA V5 assembly and its entire geometry contents into memory, before writing out all of the data to Creo View.</li> <li>See the section on Assembly Processing Best Practises for more info.</li> </ul>





Option		Description	
Options	Omit Large Faces	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 <b>set_omit_large_planes <value in="" m=""></value></b> , see additional options for details.	
Read Face	Convert Surfaces	Allows the CATIA V5 API to convert analytical surfaces as NURBs, this may be useful from time to time if face/surface errors occur in the CATIA V5 read. (Often used in along with Convert curves)	
	Face Colours	By default individual face colours are not read, this option enables individual face colours to override solid colours.	
10	Convert Curves	Allows the CATIA V5 API to convert analytical curves as PCURVES, this may be useful from time to time if edge/curve errors occur in the CATIA V5 read. (Often used in along with Convert surfaces)	
rve Option	Use 3D Curves	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.	
Read Cur	Check 3D Curves	This option allows the Adapter 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 <i>validate_3D_curve_tol <value></value></i> in the additional option field.	
neral TIA V5 cead tions	Hidden Geometry	Allow the translation of hidden nodes	
Ge CAI Op	Read Axis	Enables the reading of CATIA V5 axis systems.	
tions	PMI + Captures	Please note!This feature will require additionalTheorem licenses. See the 'CATIA V5 to Creo View withPMI' demonstration video for full details.Enables the read of PMI stroked and semantic data.Reading of captures is automatically enabled.	
MI Read Op	Hidden Geometry + PMI	Optionally read PMI Annotation set construction geometry	
£	Omit Captures	Reads only the Views if the option below is checked	
	Read Views	Optionally read views, when reading PMI	





w Write Options	Use Part names	Use the CATIA V5 'part number' names for assembly nodes. The default is to use 'Instance name'.
	Disable Inst Attribs	Disables the output of any instance attributes.
	Animations	Please note! this feature will require additional         Theorem licenses.         Outputs one or more .fra animation files, which can be         imported into any Creo View viewer that supports         animations.         The .fra files will be created alongside the output .pvs         file, in the same location and with the same base name.         If the animation has a title (name) this will be appended         to the base name, for example :         C:\TEMP\chainsaw_pvs         Will result in:         C:\TEMP\chainsaw_removeCover.fra         The .fra files are created by default in mm.
Creo Vi	Empty Parts	Creates an empty (contains a single point at X0,Y0,Z0) dummy .ol file for empty nodes, to force the geometry node creation, where no geometry exists
	Datum Planes	Allows large planes, that are by default omitted, to be written as Datum Planes. Selecting this option will replace PLANES where the bounding box diagonal exceeds 0.1Km with a Datum PLANE 35mm square. The bounding box size can be adjusted by using the additional argument <i>set_omit_large_planes <value i="" in<=""> <i>mm&gt;</i></value></i>
		If the size of the created datum PLANE can be adjusted by using the additional argument <b>set_create_datum_planes <value in="" mm=""></value></b> , see additional options for details.
PMI Filters	Flip Text In View	NOA data read from CATIA V5 may be flipped so that it is readable from any given capture view. This option adjusts the text to mimic the behaviour of the CATIA V5 application.
	Flat To Screen	Text And NOA data read from CATIA V5 that is 'flat to screen' will be written 'flat to screen' with this option set. In order for this option to work, the semantic PMI MUST be read, which means the Read PMI option MUST be set to Enabled.





	None Filled	Please note that these entities are Floating Markup in the Creo viewer. In order to see them in the viewer these entities should be 'enabled' (see MODEL ANNOTATIONS NAVIGATION FILETRS panel in the Creo Viewer and ensure FLOATING ANNOTATIONS is selected) It also it may be necessary the select floating annotations and (Right Mouse Button) select 'show' to see them in Creo viewer.
	None Filled	as read from CATIA V5. Only provide an outline style
		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.
	Disabled	(Default setting) Solid faces written with unique edges.
Full Topology	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. (Solids only) Fix non-manifold - As above with additional behaviour needed to resolve non-manifold conditions caused by CATIA V5 add operations. (Solids & Quilts) Fix non-manifold – Both solids and open solids processed and resolve non-manifold conditions caused by CATIA V5 add operations. <b>NOTE!</b> 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.





	Disabled	Allows control over the processing of any PMI capture data. <b>Please Note!</b> The following limitations apply: to correctly view captures with Theorem's v14.1 (onwards) translator it is necessary to use CreoView 1.0 M020 or later Options are:
	Disablea	Do NOT write captures
Capture Views	Enabled	<ul> <li>Geom Mask ON – Local Space : (Default setting) This will ensure that only the geometry associated with a given capture view is displayed in that view.</li> <li>Geom Mask OFF – Local Space : Allow user to have all geometry displayed in every capture view.</li> <li>Geom Mask ON – Global Space :</li> <li>Geom Mask OFF – Global Space :</li> <li>Geom Mask OFF – Global Space :</li> <li>Local Space presents the view relative to the geometry at the assembly node (Consistent with CATIA V5 up to and including R19)</li> <li>Global Space transforms the view into Global or World view (Consistent with CATIA V5 since R19)</li> <li>Any Hidden PMI will automatically be read when captures are enabled, however the user should enable the Hidden Geometry option if they wish to include this data in the captures</li> </ul>
		<b>Please note!</b> this feature will require additional Theorem licenses. Allows Model Based Definition (MBD) data to be read. Options are:
n Tree	Disabled	(Default setting) do not read the specification tree (MBD).
Process Specificatio	Enabled	The specification Tree setting will provide a true representation of the Geometry and PMI entities in the capture views when used in conjunction with the setting: Capture views = Enabled – Geom Mask ON
	Enabled - Expanded	The specification Tree is read to a deeper level which supports the 'initial view state' of views - please note this option can produce many more output files.





### V5 Options 2 – more translation settings

Basic Parameters Advanced		
Adapter CATIA V5 - Supports Catia V5 versions R System Paths Copyright V5 Options	R26 to R31 s 1 V5 Options 2 V5 Drawing V5 Post Proc	cess
Diagnostic Logs Mask Wireframe Mask Geometry Mask Lavers	Info     Detailed Info       Points     Curves       Surfaces     Solids	_
Process Architecture Surface Checking Additional Options Additional Options File	Disabled	±

#### Each of these options is described below: Opt

ostic s	Info	Generate a more verbose log files
Diagno Log	Detailed info	Used as a diagnostics level of detail in the log file (for debug purposes only)
sk rame	Points	Turn off Point processing. By default points are translated.
Ma Wirefi	Curves	Turn off Wireframe Curve processing. By default curves are translated.
ask netry	Surfaces	Turn off Surfacing processing. By default curves are translated.
Ma Georr	Solids	Turn off Solid processing. By default solids (or Breps) are translated.
		By default, ALL layers are translated. This field allows the user to select which layers ARE translated.
Mask Layers	To translate specific layers	The syntax for this field is limited to ranges of layer numbers (separated by '-') and individual layer numbers; each range and individual number being separated by a comma ',' . e.g. 20-30,45,100-300 Means layers 20 through to 30 AND 45 AND 100 through to 300 WILL be translated. The full range of layers is 0-1023




Process Architecture	Split Process	This option may aid some specific data issues <b><i>BUT</i></b> it should only be used with guidance by Theorem support.
Surface Checking	The default is disabled.	Generates a report file that details any points on the output Creo View surfaces that deviate by a distance greater than a selected tolerance. Also, optionally, the output Creo View data can be annotated with 'surface check' points to indicate where these points are. This is achieved by selecting one of the tolerance suffixed with ( <i>plot points</i> ). The 'surface check' points are coloured green for within selected tolerance and red for outside selected tolerance. The report file is created alongside the Theorem progress file and will be named " <progress_name>_surf_check.log". A report summary will be created as follows: SURFACE CHECK : Some Surface check issues Found [4434] points &lt;= Gap Tol (green points) [0.001] Found [1544] points &gt; Gap Tol (red points) [0.001] Found [0] points &gt; Gap Limit (calc errors) [1] Largest Gap Valid [0.00992005] found at Idx [1202]</progress_name>
Additional Options	General	<ul> <li>Not ALL options for to the CATIA V5 Creo View Adapter are made available to the recipe editor.</li> <li>These options are not in common use, but are included here should they ever be required. If additional options are required, then these can be added to the Additional Options field, delimiting each option with a space " " character.</li> <li>This field is functionally equivalent to the additional options file field, however settings made here will override the settings in the options file. This field allows for quick transient tests to be performed, without the need to edit the additional options file. <i>(see below)</i></li> <li>Some example of additional options are as follows:</li> </ul>
	<pre>racet_toi <value></value></pre>	number of facets created ( <i>min. value = 0.0001</i> )





surf_check_ max_gap_lim it <i><value></value></i>	Used in conjunction with Surface Checking to adjust the <i>'max gap limit'</i> value, which allows reported errors to be discarded if they are greater than a set value.
progress_file <file name=""></file>	The progress file contains a complete audit trail of the translation identifying each element as it is translated from CATIA V5 into the Creo View format. The file will also contain any error messages that may have been generated during the translation. The default location is:
	%TEMP%\tscprogressyj
	This option allows a different output location to be specified
disable_opaci ty	Disable the writing of opacity settings into Creo View data
pmi_RGB	Set a default colour for PMI text and graphics, this will
<rrr-ggg-< th=""><th>override the colours read from CATIA V5. The argument rrr-</th></rrr-ggg-<>	override the colours read from CATIA V5. The argument rrr-
bbb>	ggg-bbb, MUST be given as 3 values 000 to 255 for each of the
	colours with a '-' char between, for example:
	<i>pmi_RGB 000-000-000</i> – for black text
face enacity	<b>pmi_RGB 255-255-255</b> – for white text
	option enables each face to have its own opacity setting.
opacity_zero	Allows the user to set a minimum value of opacity. Values are
<value></value>	allowed in the range of 0.0 to 1.0 (default 0.1). Values below 0.1 will appear invisible in Creo View
reservations	Enables the conversion of Space Reservations in a faceted form.
reservations_ brep	Enables the conversion of Space Reservations in a converted BREP form.
dont_create_	Allow Axis Systems to be read as open solids. Note! Axis
udf_axis	System read in this way may obscure the Creo View model data – see also <i>udf_axis</i> option.
udf_axis	Ensures Axis Systems are created as axis systems. This ensures
	that the model in Creo View isn't obscured by very large
	planes that make up the axis system.
set_omit_lar	Omit large PLANEs greater than the tolerance value ( <i>default is</i>
ge_planes	100 M)
<value></value>	o a sat amit large planes 2000
	- sets a value of 2km





set_create_d atum_planes <i><value></value></i>	Omitted large PLANEs can be replaced by datum PLANEs (default 35mm). The size of that datum PLANE is controlled with this option.
	e.g. <i>set_create_datum_planes 150</i> - sets a value of 150mm
v5_face_fixu	Geometry fixup for faces only.
p	It is rare that this option would be used.
v5_fixup	Geometry fixup for solids and faces.
	It is rare that this option would be used.
convert_curv e_tol <i><tol></tol></i>	Allows the <i>convert_curves</i> recipe option default tolerance of 0. 00001 to be altered
convert_surf ace_deg <degree></degree>	Allows the degree for converted NURBS surfaces to be set.
convert_surf ace_tol < <i>tol&gt;</i>	Allows the <i>convert_surfaces</i> recipe option default tolerance of 0. 00001 to be altered
omit_bad_fa ces	The Adapter can detect a condition where 2D curves have been created, but are outside of the required tolerance, by default these faces will processed using Creo View's own 2D curves - This option allows this to be disabled, and the face then will be omitted from the solid.
disable_view _zoom	By default the Adapter will zoom to display the viewable PMI/Geometry in a view, this can be disabled using this option
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 stored in the .pvoa file - so that a whole assembly is stored in the .pvoa file.
report_non_c ritical_errors	Enables reporting of errors that are deemed not to be critical to receiving a valid output – e.g. omitting faces
dont_convert _spheres	<b>Note!</b> This option cannot be used in combination with convert_curves or convert_surfaces. When the user wishes to take measurements of the Creo View data, it is preferable retain any analytical data. This option retains spherical surfaces in their analytic form.
dont_convert _torus	<b>Note!</b> This option cannot be used in combination with convert_curves or convert_surfaces. When the user wishes to take measurements of the Creo View data, it is preferable retain any analytical data. This option retains torus surfaces in their analytic form.





dont_convert	Note! This option cannot be used in combination with
_fillets	convert_curves or convert_surfaces. When the user wishes to take measurements of the Creo View data, it is preferable retain any analytical data. This option retains fillet surfaces in their analytic form.
attr_filter_file <filter file=""></filter>	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
model_based	Setting this option allows the user to remove path data from
_definitions_2	By default attributes read are unchanged. This should ONLY be used in conjunction with <b>Process Specification Tree</b>
disable zoom	Enabled or Enabled - Expanded.
able_pmi	<i>'zoomable'</i> will behave such that they do not overlap when zoomed to in the Creo Viewer (A viewer version of > 3.1
	In addition relationships between PMI entities will also be maintained, such that the move togother.
	This option is provided to disable this behaviour if it is not required.
zoomable_pm	This option reverses the child-parent in associated PMI
i_std	entities and is provided as a temporary option while the behaviour is being delivered in advanced of a Creo View viewer that supports this behaviour.
set	It is now possible (from 23.1 to use an environment variable
IS_CREOVIEW_ MODEL BBOX	to set the absolute value of the bounding box so that all data
VALUE=1000	is tessenated to the same accuracy e.g. the value of 1000 would force a bounding box of 1 m <sup>3</sup> for tessellation
	calculations





## Additional Options For Composites

read_composites	Read composites data from the CATIA specification tree.
read_parameters	Read additional parameters from the CATIA specification tree.
output_mbd_leaf_nodes	Read sub-part level geometry
enable_view_zoom	Views are zoomed to the part geometry
create_rosette_graphics	Creates the ply-direction rosette
model_based_definitions_1	Read additional mbd data for composites
exclude_list	Excludes a list of named nodes from the output tree
COMPOSITE_PLY_LINEWIDTH=n	Thickens the composite ply boundaries by a factor (n is between 1 and 5)

## Additional Options file

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. See <u>Appendix B</u> for the
	format of a 'Theorem Configuration File'.





## V5 Drawings – Options Related to CATDrawing Processing

Basic Parameters Advanced		
Adapter CATIA V5 - Supports Catia V5 versio	ns R26 to R31	
System Paths Copyright V5.0p	ions 1 V5 Options 2 V5 Drawing V5 Post Process	
Drawing Output Type	CGM	
PDF Drawing Options		
PDF Output Files	Multi Files	
PDF Merge	No Merge 👱	
PDF Merge Keep File		
PDF Merge Ignore File		≥
PDF Water Mark File	<u> </u>	

Each of these options is described below:

Option	Description			
Drawings Output Type	Allows the output format for Drawing processing to be specified. The default is DXF. CGM, PDF, HPGL or TIF can be specified, these options, however, do not offer the multi-sheet output that is present in the DXF format.			
Drawing Output Files – Multi-Files	however, do not offer the multi-sheet output that is present in the DXF format. When processing drawing files with multiple sheets, to output is controlled by a combination of the CATIA5 option 'save multi-sheet document in a single vectoria file' (v5-save-single-file) and the Multi-Files recipe setting. The CATIA5 v5-save-single-file option can be in the Tools->Options->Compatility->Graphics Formation tab: Export (Save As) Save multisheet document in a single vectorial file Put current sheet as the first page of the vectorial file CGM CGM HPGL2 And controls whether files are written as one per shee or all sheets in one file. See Appendix C – Drawing Processing Options for how			
	these values should be used			





PDF Merge	<ul> <li>Note! This option is only available when Drawing Output Type is set to PDF, it is used to add additional control over the election of multiple sheets</li> <li>Note! The Multi-Files option cannot be selected for this option to operate</li> <li>The following options are available from the pull-down: No Merge - Default do nothing, this MUST be selected when Drawing type is NOT PDF or PDF Merge is NOT required.</li> <li>PDF Merge <ul> <li>If the drawing has multiple sheets and multiple PDF files are saved, this option will merge them back to a single PDF document.</li> </ul> </li> <li>PDF Merge Ignore Details <ul> <li>If the drawing has multiple sheets and multiple PDF files are saved, this option will automatically ignore any CATIA V5 drawing sheets in the CATDrawing, that are "Detail" sheets, from being included in the resultant merged PDF output.</li> </ul> </li> <li>PDF Merge Keep Files <ul> <li>If the drawing has multiple sheets and multiple PDF files are saved this option will allow selected sheets to be merged - using the file specified in the PDF Merge Keep File field (See below). The keep file is a simple text file with the drawing sheet names listed one per line, for example : sheet.1 sheet.2 sheet.4</li> <li>This example will keep sheet's 1,2 and 4 only, in the resultant merged PDF output.</li> </ul> </li> <li>PDF Merge Ignore Files <ul> <li>If the drawing has multiple sheets and multiple PDF files are saved this option will allow selected sheets to be ignored - using the file specified in the PDF Merge Ignore File field (See below). The keep file is a simple text file with the drawing sheet names listed one per line, for example : sheet.1 sheet.2 sheet.4</li> <li>This example will keep sheet's 1,2 and 4 only, in the resultant merged PDF output.</li> </ul> </li> </ul>
	<ul> <li>This example will ignore (omit) sheet s 3 and 4, the resultant merged PDF output will contain all sheets avant sheets 2 and 4.</li> </ul>
	sneets except sneets 3 and 4.
PDF Merge Keep File	Specify the 'keep' file list for the <b>PDF Merge -&gt; Keep</b> Files setting





PDF Merge Ignore File	Specify the 'ignore' file for the <b>PDF Merge -&gt; Ignore Files</b> setting
PDF Water Mark File	Allows the selection of an image file, in either JPG or PNG format, that will be merged into the PDF file as a watermark. This option only works if one of the the PDF Merge options are selected (i.e. it will NOT work if No Merge is selected). <b>Note!</b> The watermark files are not scaled, the user must provide the correct page size/format to match the input drawing.





V5 SaveAs – Options relating to the Additional formats

## Please note! This Processing feature will require additional Theorem licenses.

st Process Mode	Disabled				Ŧ		
ave As Options Note: These Save As settings should nstead, configure Additional File T	d not be used wh ypes using Wind	en config chill Publi	juring Wir ish Rules	ndchill worl	cer.		
CATProduct (input type)	☐ IGES	CGR	WRL	F STEP	G 3DXM	IL 📕 3DPDF	
	🗖 NRep 🛛	ТХТ	SSN	п л	F STBO	М	
ATPart (input type)	🗖 IGES 🖡	CGR	VPS	<b>STEP</b>		<b>3DXML</b>	
	🗖 NRep 🛛	WRL	F HCG	TL 🗖	F STL	<b>3DMAP</b>	
	3DPDF						
ATDrawing (input type)	SVG 🔽	DWG	DXF	□ ССМ	F PS	<b>3DXML</b>	
	CLOG	PDF	TIF	🗖 JPG	🗖 1G2	GL2	
/lodel (input type)	CGR	STL	IGES				
ave As Name (Optional)					_		
D PDF Template (Optional)					Ø		
ost Process Ontions						_	
pecified Post Process Exe							
pecified Post Process Opts							

Each of these options is described below:

*Please Note!* JT export is provided via the CATIA V5 Creo View Adapter – JT Add On module and requires an additional Theorem license. JT export also requires a configuration file to control the output. The default configuration file is located at **%TS\_INST%/etc/tessCATIA5.config.** The user can edit this, to suit their tessellation quality and output requirements.

**Please Note!** 3D PDF export is provided via the CATIA V5 Creo View Adapter – 3D PDF Add On module and requires an additional Theorem license. 3D PDF export also allows for templates to be selected to control the page layout and export options.



# CADVerter 26.0 for CATIA V5 to Creo View



Option	Description			
Post Process Mode	There are 3 modes available for Post Processing: Disabled Post Process is disabled Saves As Activates the following selections in the page: CATproduct SaveAs CATPart SaveAs CATDrawing SaveAs SaveAs Name (optional) Post Process Specified Post Process Exe Specified Post Process Onts			
Post Processing CATProducts	This set of check boxes allows the selection of the file types to be created when <i>Processing CATProducts</i> AND <i>Post Process Mode is SaveAs</i> <i>Note!</i> SSN denotes a CATIA Session file			
Post Processing CATParts	This set of check boxes allows the selection of the file types to be created when <i>Processing CATPart</i> is selected AND <i>Post Process Mode is SaveAs</i> <i>Note!</i> When processing assemblies and using minimum memory mode, any settings in the SaveAs CATPart section will be applied to every part <i>Note!</i> NRep denotes a CATIA NavRep file			
Post Processing CATDrawings	This set of check boxes allows the selection of the file types to be created when <i>Processing CATDrawing</i> AND <i>Post Process Mode is SaveAs</i> <i>Note!</i> CLOG denotes a CATIA CATOLOG file			
SaveAs Name	This field allows the base name of the 'SaveAs' files to be specified, the default being the same name as the input file. When processing assemblies and using minimum memory mode, this name will ONLY apply to the top level assembly – this is to avoid the same name being used for all parts and the assembly			
Specified Post Process Exe	When Post Process Mode is set to Post Process, this field allows ANY post process executable or script to be specified – This allows customers to link in their own post processing behaviour. The specified Post Process executable will be called as follows: <post_process_exe> <input_file name=""> <output_folder> <specified arguments=""> v5_version <catia v5="" version=""></catia></specified></output_folder></input_file></post_process_exe>			
Specified Post Process Opts	Specify arguments to be passed to the Specified Post Process Executable			





Alternative File Output (Additional File Types)

The Adapter supports the generation of the same alternative file types (Additional File Types) as listed in the Post Processing SaveAs section, using a WVS Publish Rules 'job' file.

The syntax of invoking the Adapter with a job file is as follows :

#### catia5batch.cmd -j <job\_file.paj>

Where an example job file format is:

<?xml version="1.0" encoding="UTF-8"?> <publish> <input filename="as1.CATProduct"/> <output typename="ALTFILE"> <file display-name="drw1" type="CGR" output-prefix="cc" output-suffix="ss"> </file> <file display-name="drw4" type="IGES" output-prefix="m" output-suffix="uu"> </file> <file display-name="drw5" type="STBOM" output-prefix="h" output-suffix="uu"> </file> <file display-name="drw6" type="TXT" output-prefix="t" output-suffix="uu"> </file> <file display-name="as1" type="PDF" output-prefix="t" output-suffix="uu"> </file> <file display-name="as1" type="JT" output-prefix="t" output-suffix="uu"> </file> </output> </publish>

The *<file...>* elements used in the .paj file result from the same entries being used in Windchill's WVS Publish Rules XML definition for the *<additional-files...>* elements. Further details can be found in the **WVS Publish Rules** documentation in the **Windchill Visualization Services Guide** available from the PTC Reference Documents Site:

#### https://www.ptc.com/appserver/cs/doc/refdoc.jsp

This .paj file shows CGR, IGES, STBOM, TXT, PDF and JT files being created for the 'as1.CATProduct' input file. All of the alternate formats (*Additional File Types*) will be packaged into a .pvoa file.

It is important to check that the correct CATIA V5 licenses are available, since some alternative file types (*Additional File Types*) are licensed, This can be carried out via the 'SaveAs' menu option from the CATIA V5 application. If the required type can be saved interactively then the 'Alternative File Output' mechanism should operate successfully.

**Please note!** The output prefix and output suffix values are added to the output file names generated by the translation, so in the JT example above the resultant file name would be **t\_as1\_uu.jt**. If these strings are empty the pre/post fix is an "\_" (underscore). So, for the JT example above, if both strings were empty the resultant file name would be **\_as1\_.jt**.





## Translating Interactively from CATIA V5

The CATIA V5 to Creo View Adapter allows an active CATIA V5 Part or Assembly to be translated directly into Creo View, directly from the CATIA V5 application.

In order to translate from within CATIA V5, the CATIA V5 application must be started from within a Theorem environment, so that the appropriate CATIA V5 menus are loaded.

CATIA V5 can be started from a shortcut, if requested at installation time. Alternatively, it can be started via the script provided in the CADverter installation at:

#### <installation\_directory>\bin\catia5r[version]\_start.cmd

(where [version] should be substituted for the version of CATIA V5 that you have installed – e.g. 19, 23, 24):



Once CATIA V5 has been started and a model loaded, the active Part or Assembly can be exported to Creo View.

In order to export, the user selects the File -> Export Menu Option:







.ac_ctc_02_asme1_ct5210_rc.CATPas.	
Theorem Import	
Theo <u>r</u> em Export	
<u>E</u> xit	

Which in turn launches the Theorem Export panel:

Theorem Export ? ×	
Theorem Solutions	
Export Recipe File	
Export type	
Creo View 🗸	
Input CATIA V5 File Name	
C:\Theorem\CAD_18.2_CA5CVW_WIN64.02\sar	
🖾 Sanatize	
Output File Name	
C:\Theorem\CAD_18.2_CA5CVW_WIN64.02\sar	
Progress File Name	
C:\Theorem\CAD_18.2_CA5CVW_WIN64.log	
Retain Assembly Structure	
OK Gancel	1

The *Input CATIA V5 File Name, Output File Name* and *Progress File Name* fields will be prepopulated if a model is already loaded into the CATIA V5 session. However, these locations can all be modified prior to selecting the *OK* button.

The *Recipe File* tab allows the user to select a Creo View recipe file:



# CADVerter 26.0 for CATIA V5 to Creo View



	xport	Recipe File				
	Recipe Fil	e File Name				
	C:\Theorem\CAD_18.2_CA5CVW_WIN64.02\rec					
	Recipe Fil	e Options				
	file=catia	a5_pv-mockup	-rcp	^		
	ver=5.0					
	adapter/outputPView=0					
	adapter/outputWireframe=0					
	adapter/outputLink=1					
adapter/outputAnalytical=0						
	· · · · · · · · · · · · · · · · · · ·					
		Recipe E	ditor			
			3			

Having selected a recipe file. The recipe editor can be launched via the *Recipe Editor* button, to allow the selection to be edited:







Tessellation				-	
	Output Structure	MockUn	<none></none>		
A <u>n</u> alytical	Output Author States	EDD Consect	Alarea	<u> </u>	
<u>W</u> ireframe	✓ Output Drawings	EPD.Connect	<none></none>	*	
<u>C</u> urves	Output <u>G</u> eometry	Output Thumbr	nail		
Cosm <u>e</u> tic Geometry		Output PVZ			
Datum <u>Q</u> uilts	Advanced Options				
Datum <u>P</u> lanes	✓ Output Sub-Components	Cutput Tessell	lated Datum Plane	1	
Datum C <u>u</u> rves	Cutput Lin <u>k</u> s	Output ASCII			
Datum Points	Output Single Shape File	PVS COL			
Datum Axes	Cutput Text As Lines		ea VG 🔽 HPGI		
Datum Cs <u>y</u> s	Double Precision Geometry	Render Faces As			
Planar Annotations		🖲 Default 🧧 Si	ngle 🧿 Double		
Eloating Annotations					

If a user saves or creates a recipe file, the resultant file should be selected in the *Recipe File Name* field.

On selecting **OK** the on the Export Panel the active Part or Assembly will be written to Creo View using the into the selected output directory.





## Assembly Processing

## Processing CATIA V5 Assemblies (.CATProduct files)

Assuming that the input to the Adapter 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 V5 Creo View Adapter 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 V5 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** 

## Processing CATIA V5 Drawings (.CATDrawing files)

For each drawing (.CATDrawing) file processed individually the translator will output a (by default) DXF (.dxf) file per sheet found in the .CATDrawing file. The final output to the user is a Creo View assembly file and one or many .dxf format files.

Therefore, assuming that the file being processed was named **test.CATDrawing** then the output would be **test.pvs** and **test\_sheet1.dxf**, **test\_sheet2.dxf**, etc.

The user can alternatively elect to output drawing files in several formats inclusing CGM, HPGL, PDF or TIF. These can be selected via the recipe editor.

## Efficient Large Assembly Processing

If an assembly is opened using the CATIA V5 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.

The Theorem translator has the capability to read assemblies 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 V5 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 selecting 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 V5 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 V5 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".
- 2. Enable CATIA V5 Cache Mode and ensure that the assembly has been accessed at least once to generate the cached CGR files.

Cache Mode can be set via the CATIA V5 Tool -> Options -> Infrastructure -> Products Structure -. Cache Management tab, as displayed below:

Options	Cache Management   Cgr Management   ENOVIAvpm   Nodes Customization   Pr
	Cache Activation
📕 🚰 General	Work with the cache system
	,
Display	Cache Location
	Path to the local cache C:\Users\mark\AppData\Local\DassaultSy V
Compatibility	Dath to the released cache
Parameters and Mea	
Devices and Virtual	
-	_ Configure
Infrastructure	Cache Size
-52	Check Maximum size
Product Structure	Maximum size 500 MB
Material Library	
	Timestamp
Catalog Editor	Check timestamps
	GMT timestamp format
Part Infrastructure	

For further assistance with this setting, please refer to your CATIA V5 on-line help.





# Error Tracking and Management

A method of tracking and managing errors output from the CATIA V5 to Creo View process has been provided. This is implemented by setting exit status codes from the Adapter 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.

## Adapter 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 "tscsummaryyj" 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 <u>(See Summary File Error Codes)</u> Field 3 = Error Description <u>(See Summary File Error Codes)</u> 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
- -5 = CATIA V5 Library incompatibility
- -6 = General Read Error
- -7 = General Write Error
- -8 = No entities Found
- -9 = CATIA V5 Environment problem
- -10 = Failed to open CATIA V5 file
- -11 = CATIA V5 Session failure
- -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
- -16 = Invalid Output Type Specified for Drawing

#### 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:

Short Timeout – *Catia5\_Access*, providing initial access to CATIA V5 Long Timeout – *Catia5\_Read*, to read each file from CATIA V5. Long Timeout – *CreoView\_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 V5 from the adapter pull-down, followed by the System Tab, the recipe editor GUI will be displayed as follows:

System Paths Copyright V5 Options 1 V5	Options 2 V5 Drawing V5 Post Process
Worker Configuration	
Short Timeout (sec) Helper 60	
Evit Adapter After	Line Helper Destart
WWGM Adapter DLL	Use CAD Handle

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 the out-of-the-box default Long and Short Timeouts should NOT be used when running the software with its default settings. They should only be modified 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.

Worker Logs

To aid tracking of problematic crashes an environment variable has been added:

set TS\_V5\_CV\_DEBUG\_DUMP\_GCO\_DATA=1

will create a C:\TEMP\TS\_DUMP\_V5\_PV.vwr (ascii) just prior to the write code commencing. This should only be used under support supervision and recommendation





# 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. 26, 27,28,29 or 30.

#### 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.

For CATIA V5 R21 and R22 a choice is available of using the older LUM Legacy licensing system or the new Dassault Licensing. To enable this enter LEGACY in the DSLICENSING option.

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.





# Appendix B – Theorem Configuration File

## Introduction

There are a number of optional pieces of information that can be provided to the Adapter prior to execution. These will control the information that is finally written into the output Creo View file by selecting specific data to be read from the input files processed.

These options are controlled by settings defined within the Theorem CATIA V5\_Creo View Adapter configuration file. To invoke these controls it is necessary to create the configuration file and also set identify the configuration filename and location.

## Configuration File Format

The configuration file is a simple ASCII text file generated with any available text editor. The format of the file is such that **each configuration command** statement is specified on a **separate line**. Blank characters separate any optional arguments related to the configuration command statement.

e.g.

## progress\_file c:\TEMP\v5\_pv.log pmi\_RGB 000-000-000

## Configuration File Location

The location of the configuration file can be defined in one of 2 ways. The recommended method is to use the recipe editor. See Setting the Additional Options File Name above.

Alternatively, the 'TS\_CFILE' environment variable can be set to point to the CATIA V5 Creo View configuration file prior to starting the translation Adapter:

Syntax: set TS\_CFILE=<configuration\_filename>

e.g. set TS\_CFILE=C:\theorem\data\catia5\configuration.txt





# Appendix C – Drawing Processing Options

## Introduction

The following table shows the effect of combined CATIA V5 Tool->Options and the CATIA V5 Creo View Adapter recipe settings:

Drawing Type	V5 save single file	PV Multi-File	Comment
DXF	Enable/Disabled	Enabled/Disabled	Multiple .dxf files are always created for this file type, so v5-save-single-file and Creo View Multi-File have no effect - OK
CGM	Enabled	Enabled/Disabled	Single .cgm file with multiple sheets - OK
CGM	Disabled	Enabled/Disabled	Single .cgm file with single sheet - Not OK
TIF	Enabled/Disabled	Disabled/Enabled	Single .tif file with single sheet – Not OK
PDF	Disabled	Disabled	Multi .pdf files created but only first sheet is viewable – Not OK
PDF	Disabled	Enabled	Multi .pdf files with a sheet per file - OK
PDF	Enabled	Disabled	Single .pdf file – correctly displayed with multiple sheets - OK
PDF	Enable	Enabled	Single .pdf file displayed as individual pages from same file - OK
HPGL	Disabled	Disabled/Enabled	Single .hpgl file created with only first sheet – Not OK
HPGL	Enabled	Enabled/Disabled	Single .hpgl file created with multi sheets - OK
TIF	Enabled/Disabled	Disabled/Enabled	Single .tif file with single sheet – Not OK





# Appendix D – Theorem Support Advanced Options

## Introduction

The following environment variables are available to modify the Adapters 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\_TIME=1

Diagnostics Variable	Value	Description
TSC_DEBUG_TIME	1	Annotates logs and screen output with time stamps.
		The <i>Diagnostic Logs -&gt; In</i> fo recipe setting must be on
TSC_DEBUG	1 or 2	Annotate logs and screen output with debug data Note! Much of the TSC_DEBUG info is now re-directed via the <i>-vm <level></level></i> command line option into the worker log
TSC_EXT_REF	1	Output specific debug for external references
TSC_LEAVE_GCO	1	Retain any intermediate GCO files

Filtering

Variable	Value	Description
TS_NO_ROUGHNESS	1	Disable PMI roughness types
TS_OMIT_ATTRIBUTES	1	List of attribute names to be omitted
TS_DISABLE_OMIT_ATTRIBUTES	1	Allow all attributes through





## Options

Variable	Value	Description
TSC_EXT_REF_ASSY	1	Include Assemblies as an external reference set
TSC_DISABLE_LARGE_ASSY_PARTS	1	For large assembly processing, omit geometry processing
TS_DISABLE_PRIMSOL_WR	1	Disable the ability to write GCO PRIMSOL data (CATIA V4 pipes)
TSC_CFILE	1	Import general command line options via a file input

## Representations

Variable	Value	Description
REPRESENTATIONS		Causes all representations for a node to be read
REPRESENTATION	<name></name>	Causes the named representation for a node to be read. If this does not exist then else the default representation will be read

## Positional Assembly Testing

This is a special set of options used to test positional assembly outside of Windchill, and possibly provide some control on its behaviour, these options offer NO benefits for users.

Variable	Value	Description
TS_TEST_POSITIONAL_ASSY	1	Simulate Positional Assembly mode in Windchill
TS_TEST_WORKER_MODE	1	Simulate Worker Mode in Windchill
TS_POSITIONAL_ASSY_DEPTH	1	Allow the depth of the read of positional assembly to be adjusted
TS_CATPRODUCT_IGNORE_FALLBACK	8-bits	001 – Ignore REFASSYNAME check 010 – Ignore signal event check 100 – Ignore NETS=0 check 111 – All on

## JT Configuration File

When saving JT files a default JT configuration file will be used, if the user wishes to specify a different configuration file these is achieved by setting the environment variable:

## set TS\_WCV\_JT\_SAVEAS\_OPTIONS=-z %TS\_INST%\etc\tessSomeOther.config

*Note!* The -z <config file> syntax

The additional options field can also be used in the recipe editor to specify a JT configuration file via:





## -z <full path to JT config file>

## Alternate File Format (Additional File Types)

When JT is selected as alternative format using the JT Add On module, the JT configuration file allows the output name to be sanitized.

Some characters are invalid in a JT context and need to be changed during the translation process. The Adapter is shipped with a default set of characters to be mapped to the allowable character of "\_". These characters are:

## \n\r\t`~!@#\$%^&\*()-+=\"\\':;,. /<>?/[]{}

It is important, particularly in a Windchill context, that the resultant JT file can be found, as it will need to into an expected .pvoa file.

This string of invalid characters used by the JT Add On can be set by the user, via the environment setting :

#### set TS\_V5\_CV\_JT\_INVALID\_CHARS="\n\r\t`~!@#\$%^&\*()-+=\"\\':;,. /<>?|[]{}"

This name sanization string would have to mirror the Filter- >FilenameSanizationSet value in the JT config file. It is suggested that Theorem support be contacted in the unlikely event that this should be used.

#### Restart

When the Adapter is used in a Windchill context, in some circumstances the worker process may require a restart. For example, when there are no CATIA V5 licenses available. This behaviour can enabled by a recipe setting:

Worker Configuration			 
Short Timeout (sec) H	elper 60		
Long Timeout (sec) H	elper 3600 👤		
Exit Adapter After	Conversions	🔽 Use Helper Restart	
WWGM Adapter DLL		Use CAD Handle	

In addition to the recipe setting an environment variable setting is required to ensure that the end-user deliberately wants to switch on the restart functionality.

Variables can also be set to run a user defined script that can carry out specific actions at the time of restart, e.g. stopping other processes and cleaning temporary directories





Variable	Value	Description
TS_CREOVIEW_RESTART_CONFIG	1	Switch on Windchill Restart
TS_V5_CV_RUN_STARTUP_SCRIPT	<name></name>	The name of a script that is run by the catiastart.bat script when the worker is restarted
TS_V5_CV_TEMP_CACHE_DIR	<dir></dir>	Folder to delete when the worker is restarted

## **PMI** Options

if flat to screen OR flipped PMI is enabled the PMI nodes may be split between the PMI and leader data. In these cases, PMI and Leader nodes are by default associated so that if the PMI text is selected, so is the leader.

This can be disabled via - env var :

## TS\_V5\_CVW\_PMI\_PMI\_OMIT\_LEADERS=1

Attribute Filter additions

There have been several additions made to the default attribute filter file

PARENT\_PMI\_ID,,0,,, ASSOCIATED\_GROUP\_ID,,0,,, PART CAPTURE VIEW,,0,,,, INFINITE,,0,,,, LINE\_TYPE,,0,,, LINE\_WIDTH,,0,,, SYMBOL\_TYPE,,0,,, CGR\_EDGE\_TYPE,,0,,,





## Screen Output

By default error messages are reported to a temporary log file: %TSC\_TEMP\_DIR%\ts\_v5\_pv\_stderr.log

Error messages can be redirected by setting the variable below:

Variable	Value	Description
TS_OUTPUT_STDERR	1	Redirects error messages to the screen log, or a log specified via the –vL command line argument

## Worker Logs

If –*vm* <*level>* worker logs are enabled and not re-directed to a log file, then these messages will default to stderr and be written to the ts\_v5\_pc\_stderr.log

## Animation Files

Motion file units can be specified:

Variable	Value	Description
TS_MOTION_FILE_UNIT	m, cm, inch, feet, yard or <b>value</b>	Specify a unit or a value – where value is mm/unit required

## PVZ Output

PVZ Output can be enabled by setting the recipe editor setting:

## adapter/outputPvz=1

## Issues Creating a CATIA V5 Worker

In the unlikely scenario that the recipe editor displays the *create catia5Worker* button greyed out then it is necessary to run the %TS\_INST%\bin\pview.reg file. This updates the registry and will enable this button.

## Managed CGR (aka SuperCGR)

Managed or SuperCGR (PTC terminology) is the capability provided via the Adapter to allow a CGR file to be created for the CATPart or CATProduct at the same time as generating the Creo View output data

This capability is the forerunner to the Post process SaveAs capability, so is only used by few customers. The function is switched on via a hidden recipe file setting:

#### adapter/output\_CGR=my\_cgr\_file\_name

This will result in a file being created in the Creo View output directory named *my\_cgr\_file\_name.cgr* 





#### Alternatively:

## adapter/output\_CGR=

will result in a file being created in the Creo View output directory named after the input CATPart/CATProduct file name.

For customer using Managed CGR, this Adapter setting is set by the Windchill worker mechanism.

When processing assemblies the output will always be a single CGR file, regardless of the conversion mode setting (i.e. standard or minimum memory).

An additional option can be used to provide an alternative output directory for the generate CGR files.

A FULL path when using this option:

#### adapter/output\_CGR\_Path=C:\\TEMP\\target\_cgr\_output\_path

*Note!* double back slashes are required for Windows paths.

**Tessellation Settings** 

In the unlikely event that a user should want to alter the tessellation settings in the Creo View output, the following settings can be manually added and modified in the recipe file under guidance from your PTC representative:

#### adapter/lod=Standard adapter/chordHeight=0.1

would result in a higher level of tessellation, and so a larger file :

#### adapter/lod=Standard adapter/chordHeight=10.0

would result in a lower level of tessellation, and so a smaller file :

#### Instance Naming

Variable	Value	Description
INSTANCE_NAMING	V5	This advanced argument causes the part names in Creo View to be displayed in accordance with the Tools->Options- >Infrastructure->Product Structure- >Nodes Customization panel (see Below)
INSTANCE_RENAMING	<string></string>	Allows this to be specified on the command line. Details of the options are shown below





# CADVerter 26.0 for CATIA V5 to Creo View

Options	Cgr Management ENOVIAvpm	Nodes Customization	Product Structure	Product Visualization Recon	4
-	Reference Product				
General	Customized display				
- Display	#PN#		onfigure		
	Product instance, reference loaded				
Compatibility	Customized display				
Parameters and Measu	#5N#(#SD#)#EN#(#SC#)#SRP#	i C	onfigure		
Devices and Virtual Re	Product instance, reference unloaded				
	Customized display				
Infrastructure	#IN# [#SRP#]	C	onfigure		
Product Structure					
00					
Material Library					
Catalog Editor					
6					
Part Infrastructure					
3D Annotations Infra					
Mechanical Design					
Medianical Design					
🚧 Shape					
H Machining					
Digital Mockup					
Tal					

An example naming configuration is shown below

The code looks for the #XX# type string finds the appropriate attribute and replaces it in the string

Note that care should be taken when assigning these names as you can end up with both duplicated names and/or empty names.

Also support for the rename specification on the command line using the option is available where <string> can be combination of :-

Product,
#PN# - Part Number,
#NO# - Nomenclature,
#RE# - Revision,
#SO# - Source,
#DF# - Definition,
#DR# - "Description",
Link to Reference,
#SRP# - Short Reference Path,





Component,
#IN# - Instance Name,
#DI# - Description,
Shape,
#SN# - Shape Name,
#EN# - Enviroment,
#SD# - Short Description,
#SC# - Source

Setting it to V5 will replicate the naming style from the CATIA V5 CATSettings

#### if flat to screen OR flip PMI is enabled the PMI nodes may be split between PMI and leader data, in these cases, PMI and Leader nodes are by default associated so that if the PMI text is selected, so is the leader. (we can only do this one way) This can be disabled via - env var :

TS\_V5\_CVW\_PMI\_PMI\_OMIT\_LEADERS=1

Solid Naming for Expand Parts

It is possible to name solids created by expand part to a more understandable when solid names have been used in CATIA. This is turned on by the command:

feature\_name

This is intended to be the default and will change in a future revision.





# Appendix E – Troubleshooting

## Introduction

Occasionally users may find translations are not successful, this can caused by many different reasons, and the Adapter is highly configurable to deal with many of these issues. This Appendix details how you can troubleshoot any problems that you may experience with the CATIA V5 Creo View Adapter.

The flow diagrams included in this section aim to guide the user as to which options are likely to resolve more common issues.

## Check List

The following check list should be followed as a precursor to following the troubleshooting flow charts:

Is the CATIA V5 source data version supported by the Adapter?

• (i.e. CATIA V5 Release 21 data cannot be translated with CATIA V5 Release 20 Adapter)

Is the machine performing the translation capable of loading the part/assembly into CATIA V5?

• This usually applies to very large assemblies, if the machine cannot load the CATIA5 data, then the translation is not likely to be successful,

Does the user have the correct permissions to read/write all of the data, including any log files?





## Troubleshooting Problem Parts

If a part fails to translate correctly this chart provides guidance as to the best options to try:







## Troubleshooting Problem Assemblies

If an assembly fails to translate correctly this chart provides guidance to the best options to try, sometimes assembly issues are a result of a 'bad' part, which can also be identified here:






## Troubleshooting Translation Performance

Some options can slow a translation down, this chart should be used to check that these options are not being used unnecessarily:







## Mutually Exclusive Options

Some options are not supported when used in conjunction with others – that is to say they are mutually exclusive, these are detailed in this chart:







When using the different processing methods. There may be several other discrepancies in the area of MBD, associated with PMI and Captures/Views.

There are limitations in converting views because CATIA does not expose all of the information needed to fully convert views into Creo View format, we recommend the use of captures

In Catia there exists the possibility to have PMI and Captures at assembly and sub-assy levels. If Techniques such as positioning assemblies are being used then it may be necessary to turn MBD processing off to get the desired result. Users will need to experiment with their own data to achieve the output required.

Single CATProducts which have a cgr representation cannot be translated as these methods are designed to work for multi-file data sets.

The preferences 'Specification Tree' and 'Component ID' are incompatible. The translation will fail if both are chosen.

