



# SPINFIRE CONVERT

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## User Guide CATIA V5 - Creo View

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## SpinFire's Product Suite

SpinFire have 5 main Product brands. These are:



*Automatically Process and Publish Inbound CAD files (Formerly CAD Publisher)*



*Seamless data exchange between CAD and Visualization (Formerly Theorem CADTranslate & CADPublish)*



*Centralised and unified view of product data (Formerly Centro)*



*Comprehensive interrogation of 2D and 3D CAD Files (Formerly Spinfire Ultimate)*



*Augmented, Mixed and Virtual Reality for Engineering (Formerly Theorem XR & Theorem TVP)*

See our [website](#) for more details on these products.

## About SpinFire Convert



SpinFire Convert 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.

SpinFire Convert 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.

## CATIA V5 to Creo View – SpinFire Convert

### The CATIA V5 to Creo View Adapter

The CATIA V5 to Creo View product also known as The CATIA V5 Adapter for Creo View, 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
- Can 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

## 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 environment.
- **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 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.

## Getting Started

### Documentation & Installation Media

Each product has a specific link that provides user documentation in the form of PDF's and Tutorials.

The latest copy of the User Guide for this particular product can be found on our web site at:

[https://docs.techsoft3d.com/spinfire/convert/catia\\_v5/creoview/user\\_guide.html](https://docs.techsoft3d.com/spinfire/convert/catia_v5/creoview/user_guide.html)

Each product has a specific link to the Product Release Notes, which in turn contains a link to the download location of the software.

The latest version of the CATIA V5 to Creo View software can be found in the Product Download Information shortcut via the link below:

[https://docs.techsoft3d.com/spinfire/convert/catia\\_v5/creoview/product\\_release\\_notes.html](https://docs.techsoft3d.com/spinfire/convert/catia_v5/creoview/product_release_notes.html)

## Installation

The installation is run from the .msi file download provided. For full details of the installation process, visit [https://docs.techsoft3d.com/spinfire/convert/catia\\_v5/creoview/index.html](https://docs.techsoft3d.com/spinfire/convert/catia_v5/creoview/index.html)

## License Configuration

To run any product a valid license file and a Flex License Manager installation will be required. The Flex License Manager is run from the .msi file download provided. This can be accessed from the Product Release Notes. For full details of the installation process, visit

[https://docs.techsoft3d.com/spinfire/convert/catia\\_v5/creoview/index.html](https://docs.techsoft3d.com/spinfire/convert/catia_v5/creoview/index.html)

## Using the Product

To use the product, follow the documented steps found in this document or follow the online video tutorials which can be found from [https://docs.techsoft3d.com/spinfire/convert/catia\\_v5/creoview/index.html](https://docs.techsoft3d.com/spinfire/convert/catia_v5/creoview/index.html)

## Running the Product

Once configured and licensed, the product is ready to be run. There are 3 distinct ways of running the translator:

- **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.
  
- **Using the SpinFire Convert UI**
  - The product can be run independently in the SpinFire Convert Unified Interface. This allows the user/administrator to create specific, named configurations for translating different types of data to achieve the best results.
  
- **On the Command Line**
  - A command line method of Invoking the translator is possible. 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***.

## Translation Configuration

It is recommended that the CATIA V5 to Creo View Adapter be run from a pre-created configuration. We 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.

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*.

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

## The Recipe Editor

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.

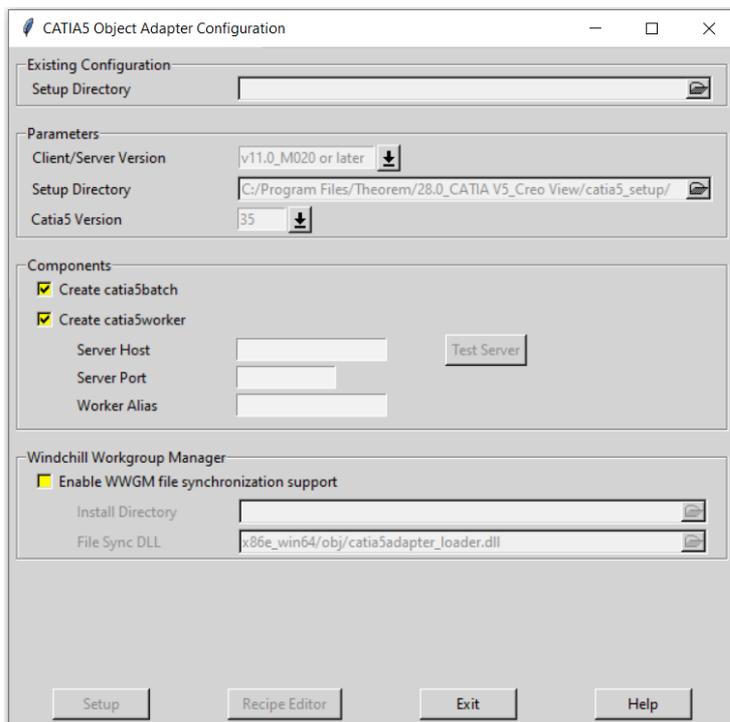
The CATIA V5 Adapter is provided with a master or default recipe file. This file is pre-configured to allow the visualization of most objects. The master recipe file should not be edited. Instead, additional new recipes can be created from this default file using Save As functionality in the recipe editor (*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 <https://www.ptc.com/appserver/cs/doc/refdoc.jsp>. The SpinFire convert Creo View Adapter uses the standard PTC mechanism to Configure translation options. The basic concepts and available options are covered here for convenience.

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

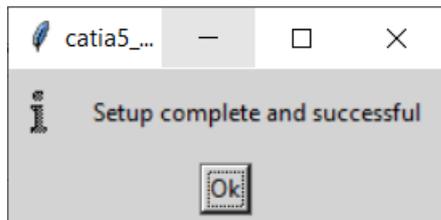
```
<Translator_installation_directory>\bin\catia5_pv_config.cmd
```

The panel below will be displayed:



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 be named **catia5\_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:

rheorem > CAD_18.2_CA5CVW_WIN64.02 > catia5_setup	
Name	Date modified
adapter.pvi	12/06/2015 14:16
catia5_pv.rcp	12/06/2015 14:16
catia5_pv_config.log	12/06/2015 14:16
catia5batch.bat	12/06/2015 14:16
debug_options.txt	12/06/2015 14:16
purge.bat	12/06/2015 14:16

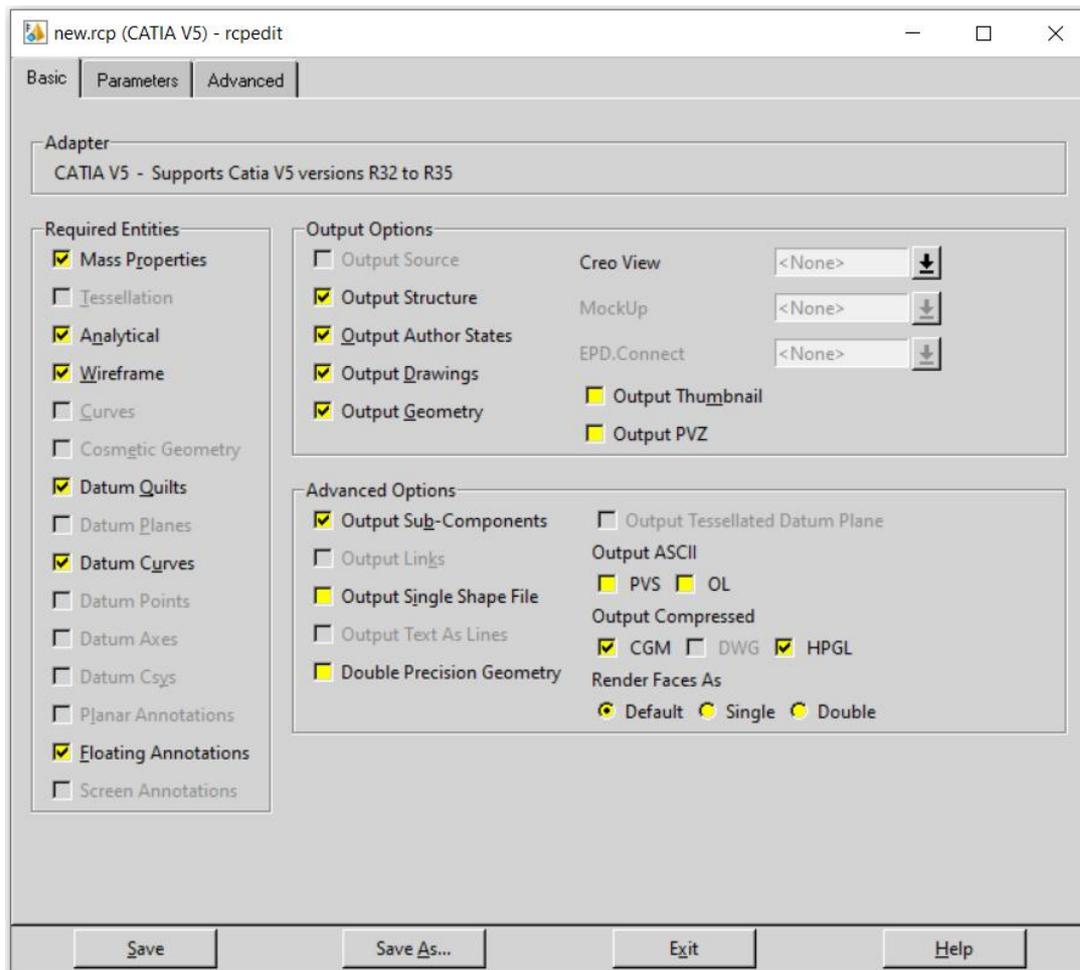
The **catia5batch.bat** script will be used in preference to the SpinFire Convert provided script discussed in the [Default Translation on the Command Line](#) section and will use the configuration options specified in the **catia5\_pv.rcp** (recipe) file.

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

```
<Translator_installation_directory>\catia5_setup\catia5batch.bat <input_file> -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 SpinFire Convert CATIA V5 Creo View Adapter.



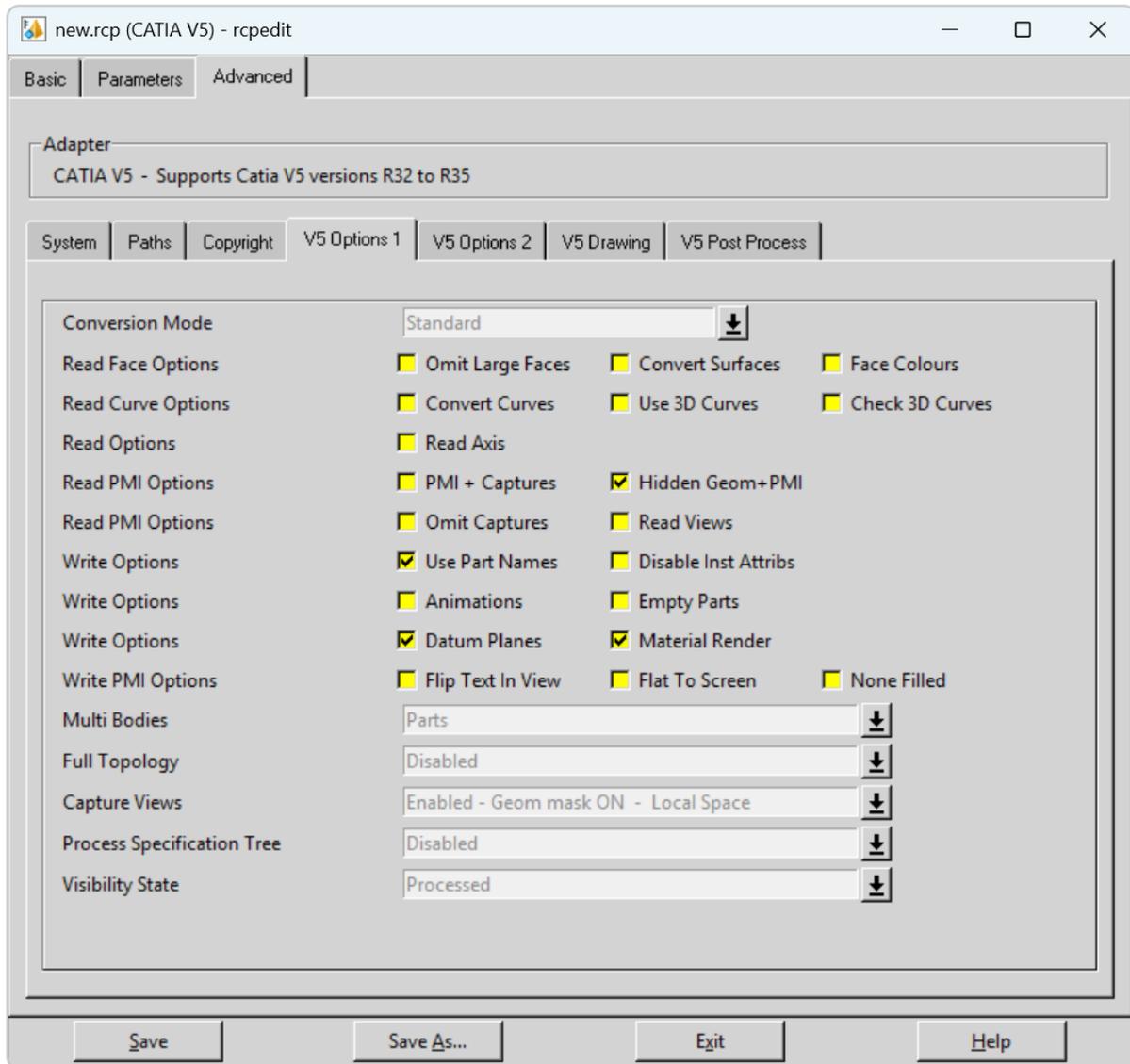
Please note that the Mass Properties option toggled on by default in the Basic tab in the recipe editor will read all the mass validation properties including volume, area, COG and density as per the material applied. This will provide a true mass value in the Creo View output. This will not be the case if this option is toggled off. A mass value will be calculated in Creo View; however, this will not be calculated with the density of the material, consequently meaning that the mass value displayed will not be a true reflection of the actual mass.

The main page provides standard PTC options and 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 <https://www.ptc.com/appserver/cs/doc/refdoc.jsp>.

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

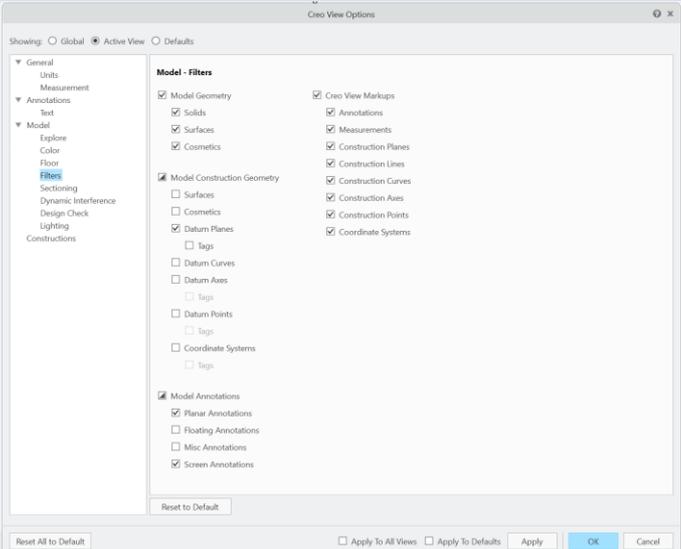
- V5 Options 1 – General translation settings (Including PMI)
- V5 Options 2 – More translation settings
- V5 Drawing – Options relating to CATDrawing processing
- V5 Post Process – Options relating to additional formats

## V5 Options 1 – General translation settings (Including PMI)



Each of these options is described below:

Option	Description
<b>Conversion Mode</b>	<p>The recommended mode for processing assembly information is the <b>Standard</b> conversion mode option. This is the default applied.</p> <p>This reads a CATIA V5 assembly and its entire geometry contents into memory, before writing out all of the data to Creo View.</p>
<b>Read Face Options</b>	<b>Omit Large Faces</b> 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 <i>set_omit_large_planes &lt;value in M&gt;</i> , see additional options for details.
	<b>Convert Surfaces</b> 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. <b>(Often used in conjunction with Convert curves).</b>
	<b>Face Colours</b> By default individual face colours are not read, this option enables individual face colours to override solid colours.
<b>Read Curve Options</b>	<b>Convert Curves</b> 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. <b>(Often used in conjunction with Convert surfaces).</b>
	<b>Use 3D Curves</b> 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.
	<b>Check 3D Curves</b> 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 &lt;value&gt;</i> in the additional option field.
<b>General CATIA V5 Read Options</b>	<b>Read Axis</b> Enables the reading of CATIA V5 axis systems.
<b>Read PMI Options</b>	<b>PMI + Captures</b> <i>Please note!</i> This feature will require additional SpinFire Convert licenses. See the <a href="#">‘CATIA V5 to Creo View with PMI’</a> demonstration video for full details.  Enables the read of PMI stroked and semantic data. Reading of captures is automatically enabled.
	<b>Hidden Geom + PMI</b> Read PMI Annotation set construction geometry. <b>Default is ON.</b>
	<b>Omit Captures</b> Reads only the Views if the option below is checked.
	<b>Read Views</b> Optionally read views, when reading PMI.

<b>Use Part names</b>	Use the CATIA V5 'part number' names for assembly nodes. The default is to use 'Instance name'. <b>Use Part Names is ON by default.</b>
<b>Disable Inst Attribs</b>	Disables the output of any instance attributes.
<b>Animations</b>	<p><b>Please note!</b> this feature will require additional SpinFire Convert licenses.</p> <p>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:</p> <p><b>C:\TEMP\chainsaw.pvs</b></p> <p>Will result in:</p> <p><b>C:\TEMP\chainsaw_removeCover.fra</b></p> <p>The .fra files are created by default in mm.</p>
<b>Empty Parts</b>	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.
<b>Datum Planes</b>	<p>Allows large planes to be created as genuine Creo View datum planes. <b>Default is ON.</b></p> <p>The visibility of these planes in Creo View is determined by the Datum Planes option in the Model – Filters tab within the Active View in the Creo View Options window. This option will need to be toggled on in order to view these datum planes in Creo View.</p> 
<b>Material Render</b>	Enables both the material information and render to be read and displayed in the Creo View output. This option will also try to render the Creo View output using the CATIA material attributes. <b>Default is ON.</b>

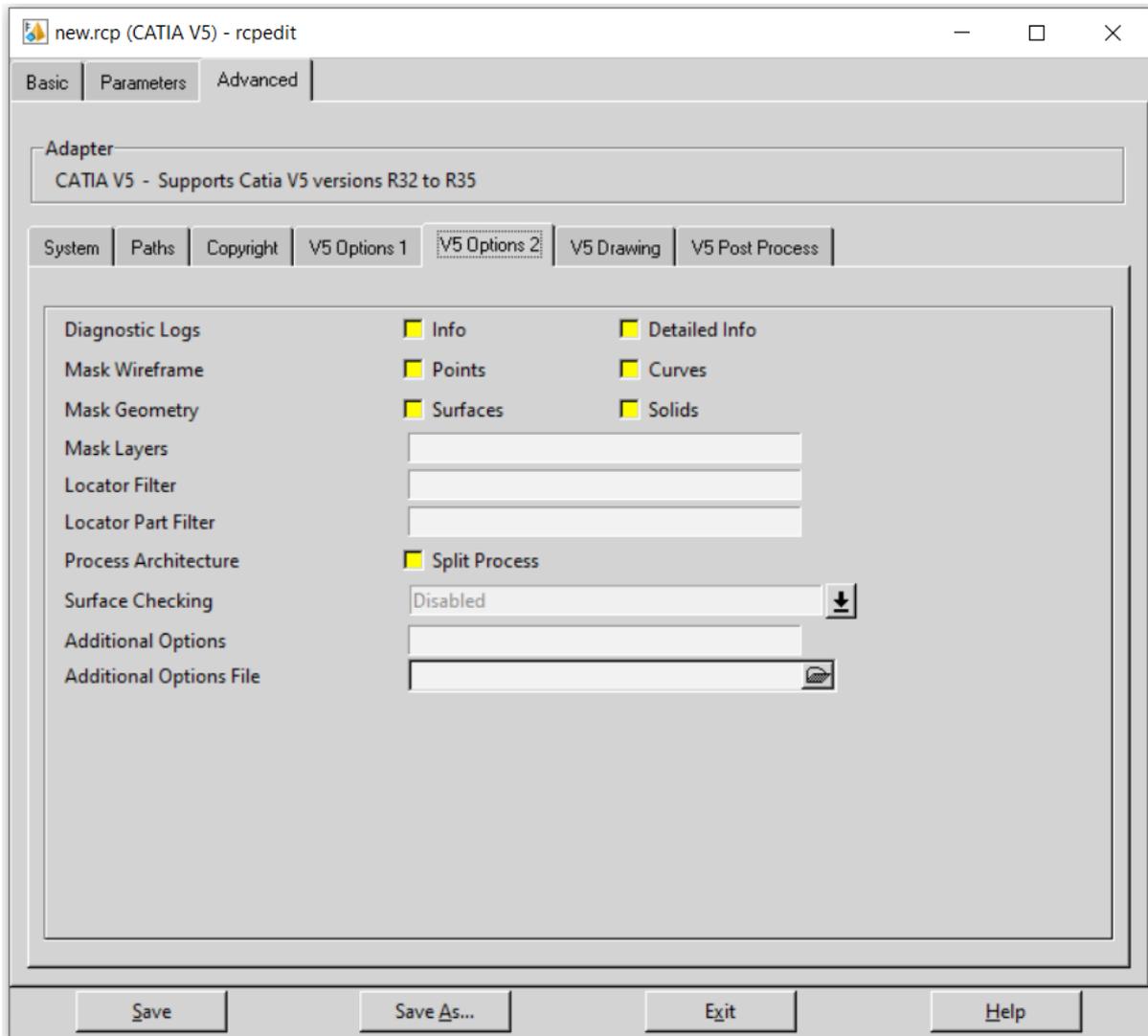
Write PMI Options	<b>Flip Text In View</b>	NOA data read from CATIA V5 may be flipped so that it is readable from any given capture view. This option adjusts the text to mimic the behaviour of the CATIA V5 application.
	<b>Flat To Screen</b>	Text And NOA data read from CATIA V5 that is 'flat to screen' 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.  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 <i>MODEL ANNOTATIONS NAVIGATION FILETRS panel in the Creo Viewer and ensure FLOATING ANNOTATIONS is selected</i> ) It also it may be necessary the select floating annotations and (Right Mouse Button) select 'show' to see them in Creo viewer.
	<b>None Filled</b>	Disable the writing of filled (solid) arrow heads and text, as read from CATIA V5. Only provide an outline style.
Multi Bodies		Enables the display of individual solids within a part.  Options are: <ul style="list-style-type: none"> <li>- Disabled</li> <li>- Parts <b>(Default)</b></li> <li>- Enabled</li> <li>- Fully Expanded</li> </ul>
	<b>Disabled</b>	Will not create extra structure. All geometry will be created under the part node.
	<b>Parts</b>	Will maintain the specification tree structure. Each node will be created whether they include solids or are empty. All surface and wireframe geometry are placed under the part. <b>This is the default setting.</b>
	<b>Enabled</b>	Where top level data has multiple solids, these solids will be displayed as separate nodes in the Creo View specification tree when Enabled is applied. This means that these nodes can be selected individually and subsequently shown or hidden. Any non-solid data will be left at the top level.  As a special case any single face OPEN Solids are not treated as solids.
	<b>Fully Expanded</b>	Similar to Enabled, however with the Fully Expanded option, the single Face OPEN solids are treated as solids and so will have their own node, as will any non-solid data which will be added collectively in an additional sub-nodes.

Full Topology		<p>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.</p> <p>Options are:</p> <ul style="list-style-type: none"> <li>- Disabled (<b>Default</b>)</li> <li>- Enabled - (Solids only)</li> <li>- Enabled - (Solids &amp; Quilts)</li> <li>- Enabled - (Solids only) Fix non-manifold</li> <li>- Enabled - (Solids &amp; Quilts) Fix non-manifold</li> </ul>
	<b>Disabled</b>	<p>Solid faces written with unique edges. <b>This is the default setting.</b></p>
	<b>Enabled</b>	<p><b>(Solids only)</b> – Adjacent faces share edges, such that the resulting Creo View data can support mass properties etc.</p> <p><b>(Solids &amp; Quilts)</b> – Adjacent faces share edges, this includes 'open solids', which will be written into Creo View as quilts.</p> <p><b>(Solids only) Fix non-manifold</b> - As above with additional behaviour needed to resolve non-manifold conditions caused by CATIA V5 add operations.</p> <p><b>(Solids &amp; Quilts) Fix non-manifold</b> – Both solids and open solids processed and resolve non-manifold conditions caused by CATIA V5 add operations.</p> <p><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.</p>

Capture Views		<p>Allows control over the processing of any PMI capture data.</p> <p>Options are:</p> <ul style="list-style-type: none"> <li>- Disabled</li> <li>- Enabled - Geom Mask ON – Local Space <b>(Default)</b></li> <li>- Enabled - Geom Mask OFF – Local Space</li> <li>- Enabled - Geom Mask ON – Global Space</li> <li>- Enabled - Geom Mask OFF – Global Space</li> </ul>
	<b>Disabled</b>	Do NOT write captures.
	<b>Enabled</b>	<p>Geom Mask ON – Local Space : This will ensure that only the geometry associated with a given capture view is displayed in that view. <b>This is the default setting</b></p> <p>Geom Mask OFF – Local Space : Allow users to have all geometry displayed in every capture view.</p> <p>Geom Mask ON – Global Space :</p> <p>Geom Mask OFF – Global Space</p> <p><b>Points to note are :</b></p> <ul style="list-style-type: none"> <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>

Process Specification Tree		<p><b>Please note!</b> this feature will require additional SpinFire Convert licenses.</p> <p>Allows Model Based Definition (MBD) data to be read.</p> <p>Options are:</p> <ul style="list-style-type: none"> <li>- Disabled (<b>Default</b>)</li> <li>- Enabled</li> <li>- Enabled – Expanded</li> </ul>
	<b>Disabled</b>	Does not read the specification tree (MBD). <b>This is the default setting.</b>
	<b>Enabled</b>	<p>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:</p> <p><b><i>Capture views = Enabled – Geom Mask ON</i></b></p>
	<b>Enabled - Expanded</b>	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.
Visibility State		<p>Controls the visibility state of hidden geometry and assembly nodes.</p> <p>Options are:</p> <ul style="list-style-type: none"> <li>- Disabled</li> <li>- Processed (<b>Default</b>)</li> <li>- Enabled – Occurrence</li> </ul>
	<b>Disabled</b>	Will ignore the visibility state of solids and assembly nodes.
	<b>Processed</b>	<p>Reads hidden geometry and assembly nodes and maintains their visibility state when opened in Creo View, mimicking CATIA V5. Only exception is for multi-instanced parts. <b>This is the default setting.</b></p>
	<b>Processed – Occurrence</b>	<p>Similar to Processed, however the Processed – Occurrence option allows multi-instanced parts to maintain their individual visibility state.</p>

## V5 Options 2 – More translation settings



Each of these options is described below:

Option	Description
Diagnostic Logs	<b>Info</b> Generate a more verbose log file
	<b>Detailed info</b> Used as a diagnostics level of detail in the log file ( <i>for debug purposes only</i> )
Mask Wireframe	<b>Points</b> Turn off Point processing. By default, points are translated.
	<b>Curves</b> Turn off Wireframe Curve processing. By default, curves are translated.
Mask Geometry	<b>Surfaces</b> Turn off Surfacing processing. By default, surfaces are translated.
	<b>Solids</b> Turn off Solid processing. By default, solids (or Breps) are translated.
Mask Layers	<p>By default, ALL layers are translated. This field allows the user to select which layers <b>ARE</b> translated.</p> <p>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 ','.</p> <p><b>e.g. 20-30,45,100-300</b></p> <p>Means layers 20 through to 30 AND 45 AND 100 through to 300 WILL be translated.</p> <p><i>The full range of layers is 0-1023.</i></p>
Locator Filter	<p>Enables filtering of locator axis systems and can be filtered using wildcards such as:</p> <p>LOC*                      *TOR                      *CAT*                      LOC*TOR</p>
Locator Part Filter	<p>Enables filtering of parts that contain locator axis systems and can be filtered using wildcards such as:</p> <p>PART*                      *PART                      *AR*                      P*T</p>

Process Architecture	<b>Split Process</b>	This option may aid some specific data issues <b>BUT</b> it should only be used with guidance by the SpinFire Convert support team.
Surface Checking		<p>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.</p> <p>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.</p> <p>The report file is created alongside the progress file and will be named "<b>&lt;progress_name&gt;_surf_check.log</b>". A report summary will be created as follows:</p> <p><i>SURFACE CHECKING COMPLETE</i></p> <p><i>PLEASE CHECK : Some Surface check issues</i></p> <p><i>Found [4434] points &lt;= Gap Tol (green points) [0.001]</i></p> <p><i>Found [1544] points &gt; Gap Tol (red points) [0.001]</i></p> <p><i>Found [0] points &gt; Gap Limit (calc errors) [1]</i></p> <p><i>Largest Gap Valid [0.00992005] found at Idx [1202]</i></p> <p><b>The default is set to Disabled.</b></p>
Additional Options	<b>General</b>	<p>Not ALL options for to the CATIA V5 Creo View Adapter are made available to the recipe editor.</p> <p>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.</p> <p>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. (see below)</p> <p>Some example of additional options are as follows:</p>
	<b>facet_tol &lt;value&gt;</b>	Used in conjunction with Surface Checking to adjust the number of facets created ( <i>min. value = 0.0001</i> ).
	<b>surf_check_max_gap_limit &lt;value&gt;</b>	Used in conjunction with Surface Checking to adjust the ' <b>max gap limit</b> ' value, which allows reported errors to be discarded if they are greater than a set value.

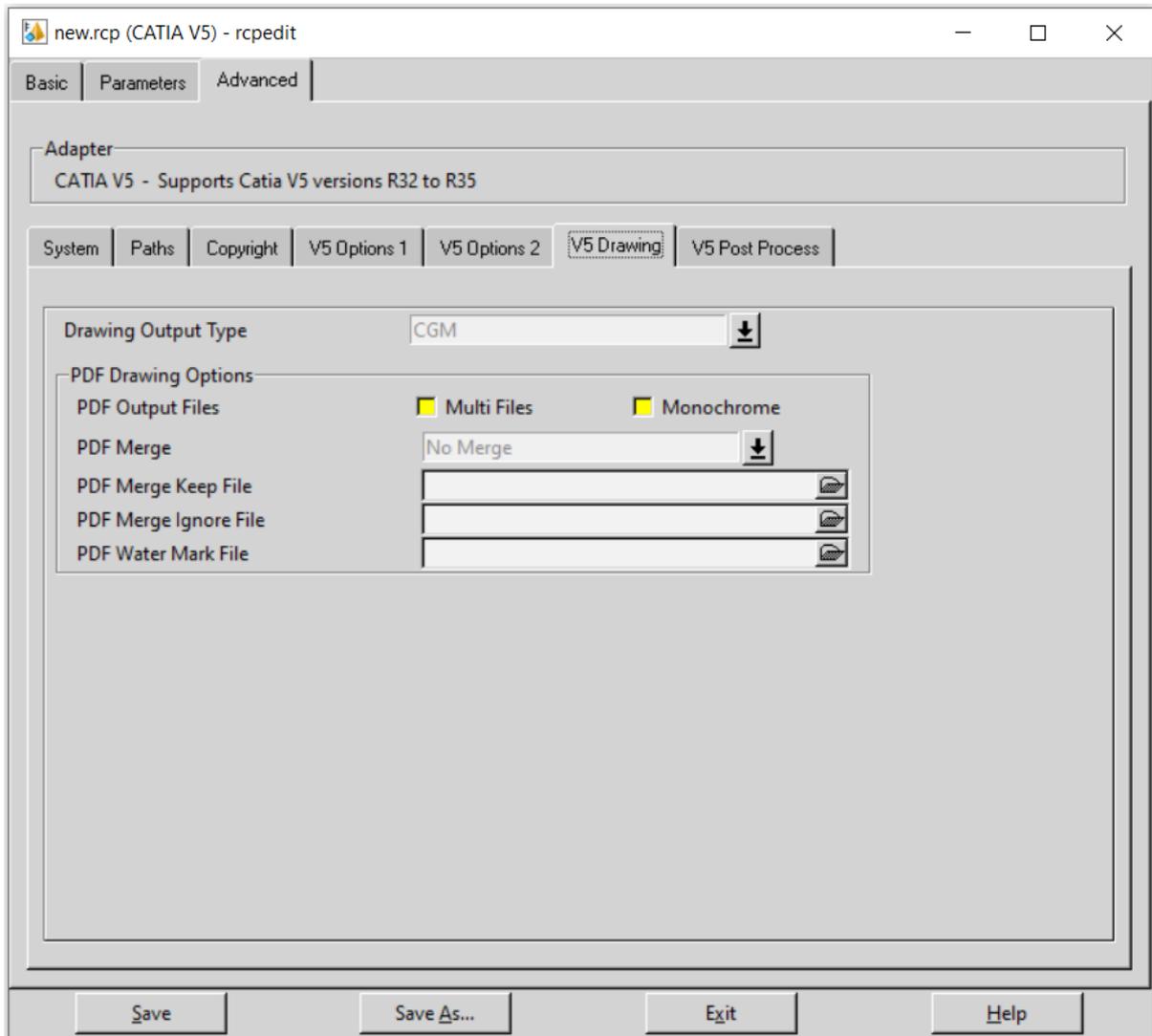
<b>progress_file</b> <file name>	The progress file contains a complete audit trail of the translation identifying each element as it is translated from CATIA V5 into the Creo View format. The file will also contain any error messages that may have been generated during the translation. The default location is:  <b>%TEMP%\tscprogressyj</b>  This option allows a different output location to be specified.
<b>disable_opacity</b>	Disable the writing of opacity settings into Creo View data.
<b>pmi_RGB</b> <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, <b>MUST</b> be given as 3 values 000 to 255 for each of the colours with a '-' character between, e.g.  <b>pmi_RGB 000-000-000</b> – for black text <b>pmi_RGB 255-255-255</b> – for white text
<b>face_opacity</b>	By default individual faces value of opacity is not read. This option enables each face to have its own opacity setting.
<b>opacity_zero</b> <value>	Allows the user to set a minimum value of opacity. Values are allowed in the range of 0.0 to 1.0 (default 0.1). Values below 0.1 will appear invisible in Creo View.
<b>reservations</b>	Enables the conversion of Space Reservations in a faceted form.
<b>reservations_brep</b>	Enables the conversion of Space Reservations in a converted BREP form.
<b>dont_create_udf_axis</b>	Allows Axis Systems to be read as open solids. <b>Note!</b> Axis System read in this way may obscure the Creo View model data – see also <b>udf_axis</b> option.
<b>udf_axis</b>	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.
<b>set_omit_large_planes</b> <value>	Omit large plane(s) greater than the tolerance value ( <i>default is 100m</i> )  e.g. <b>set_omit_large_planes 2000</b> – sets a value of 2km

<b>set_create_datum_planes &lt;value&gt;</b>	<p>Omitted large plane(s) can be replaced by datum plane(s) (default 35mm). The size of that datum plane is controlled with this option.</p> <p>e.g. <i>set_create_datum_planes 150</i> – sets a value of 150mm</p>
<b>v5_face_fixup</b>	Geometry fixup for faces only. It is rare that this option would be used.
<b>v5_fixup</b>	Geometry fixup for solids and faces. It is rare that this option would be used.
<b>convert_curve_tol &lt;tol&gt;</b>	Allows the <i>convert_curves</i> recipe option default tolerance of 0.00001 to be altered.
<b>convert_surface_deg &lt;degree&gt;</b>	Allows the degree for converted NURBS surfaces to be set.
<b>convert_surface_tol &lt;tol&gt;</b>	Allows the <i>convert_surfaces</i> recipe option default tolerance of 0.00001 to be altered.
<b>omit_bad_faces</b>	<p>The Adapter can detect a condition where 2D curves have been created, but are outside of the required tolerance, by default these faces will be processed using Creo View's own 2D curves.</p> <p>This option allows this to be disabled. The face then will be omitted from the solid.</p>
<b>disable_view_zoom</b>	By default the Adapter will zoom to display the viewable PMI/Geometry in a view, this can be disabled using this option.
<b>single_jt_file_in_pvoa</b>	If this option is selected <b>ONLY</b> 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.
<b>report_non_critical_errors</b>	Enables reporting of errors that are deemed not to be critical to receiving a valid output, e.g. omitting faces
<b>dont_convert_spheres</b>	<p><b>Note!</b> This option cannot be used in combination with <i>convert_curves</i> or <i>convert_surfaces</i>.</p> <p>When the user wishes to take measurements of the Creo View data, it is preferable to retain any analytical data. This option retains spherical surfaces in their analytic form.</p>

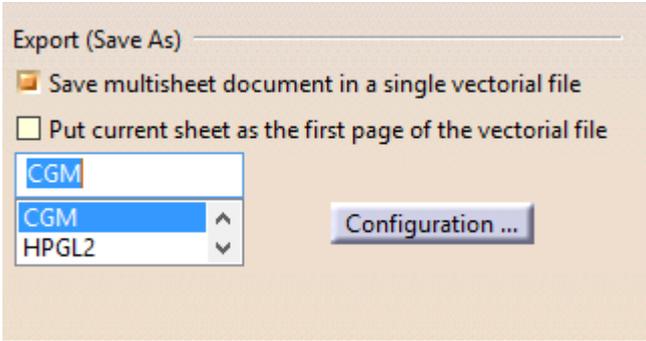
<b>dont_convert_torus</b>	<p><b>Note!</b> This option cannot be used in combination with <i>convert_curves</i> or <i>convert_surfaces</i>.</p> <p>When the user wishes to take measurements of the Creo View data, it is preferable to retain any analytical data. This option retains torus surfaces in their analytic form.</p>								
<b>dont_convert_fillets</b>	<p><b>Note!</b> This option cannot be used in combination with <i>convert_curves</i> or <i>convert_surfaces</i>.</p> <p>When the user wishes to take measurements of the Creo View data, it is preferable to retain any analytical data. This option retains fillet surfaces in their analytic form.</p>								
<b>attr_filter_file &lt;filter file&gt;</b>	<p>This is a method to define which attributes are masked or included during translation.</p> <p>By default, there is an attribute filter file installed at:  <b>%TS_INST%/data/creoView/defaultAttrFilter.txt</b></p> <p>The default locations can be overridden by specifying a value to the attr_filter_file command.</p> <p>Filtering can be disabled by specifying a non-existing file with 'attr_filter_file' OR deleting the defaultAttrFilter.txt file.</p> <p>The essential settings for an attr_filter_file are:</p> <p><b>Attribute Name; New Attribute Name; Mode</b></p> <p>Where:</p> <ul style="list-style-type: none"> <li>- Mode = 0 – Delete named attribute</li> <li>- Mode = 1 – Rename attribute to New Name</li> </ul> <p>The file format (in blue) is best explained by means of examples:</p> <table style="width: 100%; border: none;"> <tr> <td style="color: blue;"><b>MPARTNAME,,0,,</b></td> <td><b>(Delete MPARTNAME attribute)</b></td> </tr> <tr> <td style="color: blue;"><b>FILENAME,FILE (name),1,,</b></td> <td><b>(Rename 'FILENAME' to 'FILE (name)')</b></td> </tr> <tr> <td style="color: blue;"><b>*END,,0,,</b></td> <td><b>(Delete all attributes that end with 'END')</b></td> </tr> <tr> <td style="color: blue;"><b>SpinFire*,,0,,</b></td> <td><b>(Delete all attributes that start with 'SpinFire')</b></td> </tr> </table>	<b>MPARTNAME,,0,,</b>	<b>(Delete MPARTNAME attribute)</b>	<b>FILENAME,FILE (name),1,,</b>	<b>(Rename 'FILENAME' to 'FILE (name)')</b>	<b>*END,,0,,</b>	<b>(Delete all attributes that end with 'END')</b>	<b>SpinFire*,,0,,</b>	<b>(Delete all attributes that start with 'SpinFire')</b>
<b>MPARTNAME,,0,,</b>	<b>(Delete MPARTNAME attribute)</b>								
<b>FILENAME,FILE (name),1,,</b>	<b>(Rename 'FILENAME' to 'FILE (name)')</b>								
<b>*END,,0,,</b>	<b>(Delete all attributes that end with 'END')</b>								
<b>SpinFire*,,0,,</b>	<b>(Delete all attributes that start with 'SpinFire')</b>								
<b>model_based_definitions_2</b>	<p>Setting this option allows the user to remove path data from Specification Tree (Model Based Definition) related attributes. By default the attributes read are unchanged. This should <b>ONLY</b> be used in conjunction <i>with Process Specification Tree Enabled or Enabled – Expanded</i></p>								

	<b>disable_zoomable_pmi</b>	<p>For flat to screen PMI. If set, PMI Items identified as 'zoomable' will behave such that they do not overlap when zoomed to in the Creo Viewer (A viewer version of &gt; 3.1 needed)</p> <p>In addition, relationships between PMI entities will also be maintained, such that they move together.</p> <p>This option is provided to disable this behaviour if it is not required.</p>
	<b>zoomable_pmi_std</b>	This option reverses the child-parent in associated PMI entities and is provided as a temporary option while the behaviour is being delivered in an advanced Creo View viewer that supports this behaviour.
	<b>set TS_CREOVIEW_MODE L_BBOX_VALUE=1000</b>	This option is an environment variable that will enable the user to set the absolute value of the bounding box so that all data is tessellated to the same accuracy e.g. the value of 1000 would force a bounding box of 1 m <sup>3</sup> for tessellation calculations.
	<b>read_composites</b>	Read composites data from the CATIA specification tree.
	<b>read_parameters</b>	Read additional parameters from the CATIA specification tree.
	<b>output_mbd_leaf_nodes</b>	Read sub-part level geometry.
	<b>enable_view_zoom</b>	Views are zoomed to the part geometry.
	<b>create_rosette_graphics</b>	Creates the ply-direction rosette.
	<b>model_based_definitions_1</b>	Read additional mbd data for composites.
	<b>exclude_list</b>	Excludes a list of named nodes from the output tree.
	<b>COMPOSITE_PLY_LINE WIDTH=n</b>	Thickens the composite ply boundaries by a factor (n is between 1 and 5).
<b>Additional Options File</b>		Any of the Additional Options can be specified in an Additional Options File, known as a 'ts_cfile'. See <a href="#">Appendix B</a> for the format of a 'SpinFire Convert Configuration File'.

## V5 Drawing – Options relating to CATDrawing processing



Each of these options is described below:

Option	Description
<b>Drawings Output Type</b>	<p>Allows the output format for Drawing processing to be specified. The default is CGM.</p> <p>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.</p>
<b>Drawing Output Files</b>	<p>2 options are available to select. These are Multi-Files and Monochrome.</p> <p><b>Multi-Files</b> – When processing drawing files with multiple sheets, the output is controlled by a combination of the CATIA5 option ‘<i>save multi-sheet document in a single vectorial file</i>’ (v5-save-single-file) and the Multi-Files recipe setting. The CATIA5 <i>v5-save-single-file</i> option can be set in the <b>Tools-&gt;Options-&gt;Compatibility-&gt;Graphics Formats</b> tab:</p>  <p>And controls whether files are written as one per sheet or all sheets in one file.</p> <p>See <a href="#">Appendix C</a> – Drawing Processing Options for how these values should be used.</p> <p><b>Monochrome</b> – Will change the PDF to a monochrome output using a 3<sup>rd</sup> party application (Not provided by Tech Soft 3D).</p>
<b>PDF Merge</b>	<p><b>Note!</b> 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.</p> <p><b>Note!</b> The Multi-Files option cannot be selected for this option.</p> <p>The following options are available from the pull-down:</p> <ul style="list-style-type: none"> <li>- <b>No Merge</b> - Default do nothing, this MUST be selected when Drawing type is NOT PDF or PDF Merge is NOT required.</li> <li>- <b>PDF Merge</b> - 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>

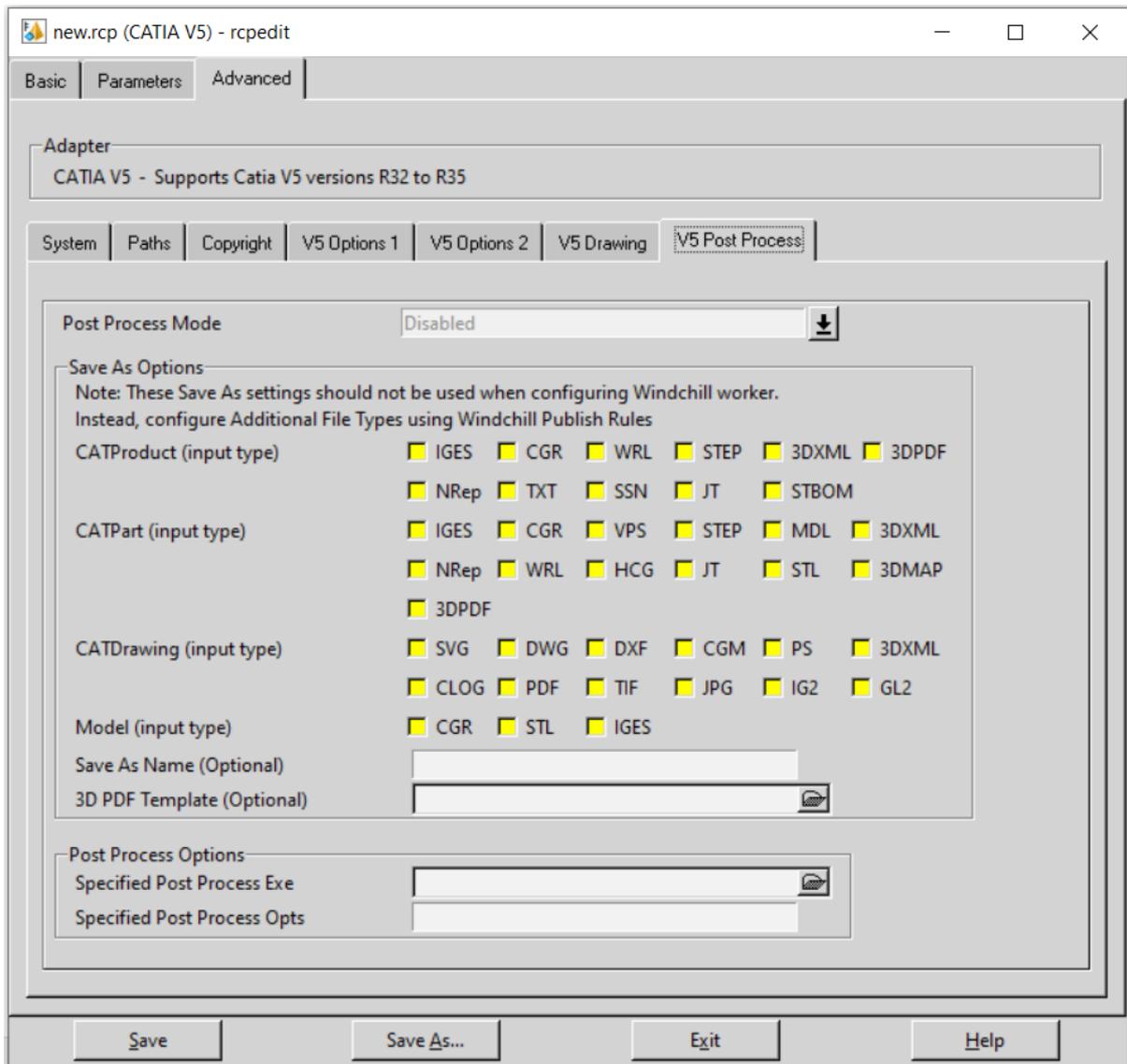
	<ul style="list-style-type: none"> <li>- <b>PDF Merge Ignore Details</b> - If the drawing has multiple sheets and multiple PDF files are saved, this option will automatically ignore any CATIA V5 drawing sheets in the CATDrawing, that are "Detail" sheets, from being included in the resultant merged PDF output.</li>   <li>- <b>PDF Merge Keep Files</b> - 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 <b>PDF Merge Keep File</b> field (See below). The keep file is a simple text file with the drawing sheet names listed one per line, e.g.                       sheet.1                      sheet.2                      sheet.4                       This example will keep sheet's 1,2 and 4 only, in the resultant merged PDF output.</li>   <li>- <b>PDF Merge Ignore Files</b> - 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 <b>PDF Merge Ignore File</b> field (See below). The ignore file is a simple text file with the drawing sheet names listed one per line, e.g.                       sheet.3                      sheet.4                       This example will ignore (omit) sheet's 3 and 4, the resultant merged PDF output will contain all sheets except sheets 3 and 4.</li> </ul>
<b>PDF Merge Keep File</b>	Specify the 'keep' file list for the <b>PDF Merge -&gt; Keep Files</b> setting
<b>PDF Merge Ignore File</b>	Specify the 'ignore' file for the <b>PDF Merge -&gt; Ignore Files</b> setting
<b>PDF Water Mark File</b>	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 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 Post Process – Options relating to additional formats

**Please note!** This Processing feature will require additional SpinFire Convert licenses.

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

The 3DPDF export is provided via the CATIA V5 Creo View Adapter – 3D PDF Add On module and requires an additional SpinFire Convert license. 3DPDF export also allows for templates to be selected to control the page layout and export options.



Each of these options is described below:

Option	Description
<b>Post Process Mode</b>	<p>There are 3 modes available for Post Processing:</p> <p><b>Disabled</b></p> <ul style="list-style-type: none"> <li>– Post Process is disabled</li> </ul> <p><b>Saves As</b></p> <ul style="list-style-type: none"> <li>– Activates the following selections in the page:                             <ul style="list-style-type: none"> <li>○ CATProduct Save As</li> <li>○ CATPart Save As</li> <li>○ CATDrawing Save As</li> <li>○ Model Save As</li> <li>○ Save As Name (optional)</li> </ul> </li> </ul> <p><b>Post Process</b></p> <ul style="list-style-type: none"> <li>– Activates the following selections in the page:                             <ul style="list-style-type: none"> <li>○ Specified Post Process Exe</li> <li>○ Specified Post Process Opts</li> </ul> </li> </ul>
<b>Save As CATProduct (input type)</b>	<p>This set of check boxes enables the selection of file types that are required to be created (i.e. JT, 3DPDF) when Processing <b>CATProducts</b> with <b>SaveAs</b> selected as the Post Process Mode.</p> <p><b>Note!</b> SSN denotes a CATIA Session file.</p>
<b>Save As CATPart (input type)</b>	<p>This set of check boxes enables the selection of file types that are required to be created (i.e. JT, 3DPDF) when Processing <b>CATParts</b> with <b>SaveAs</b> selected as the Post Process Mode.</p> <p><b>Note!</b> NRep denotes a CATIA NavRep file.</p>
<b>Save As CATDrawing (input type)</b>	<p>This set of check boxes allows the selection of file types that are required to be created (i.e. DWG, JPG) when Processing <b>CATDrawings</b> with <b>SaveAs</b> selected as the Post Process Mode.</p> <p><b>Note!</b> CLOG denotes a CATIA CATOLOG file</p>
<b>Save As Model (input type)</b>	<p>This set of check boxes allows the selection of file types that are required to be created (i.e. STL, IGES) when Processing <b>Model</b> files with <b>SaveAs</b> selected as the Post Process Mode.</p>
<b>SaveAs Name</b>	<p>This field allows the base name of the 'SaveAs' files to be specified, the default being the same name as the input file.</p>
<b>3D PDF Template</b>	<p>Enables a PDF template to be specified when saving to the 3DPDF file type.</p>

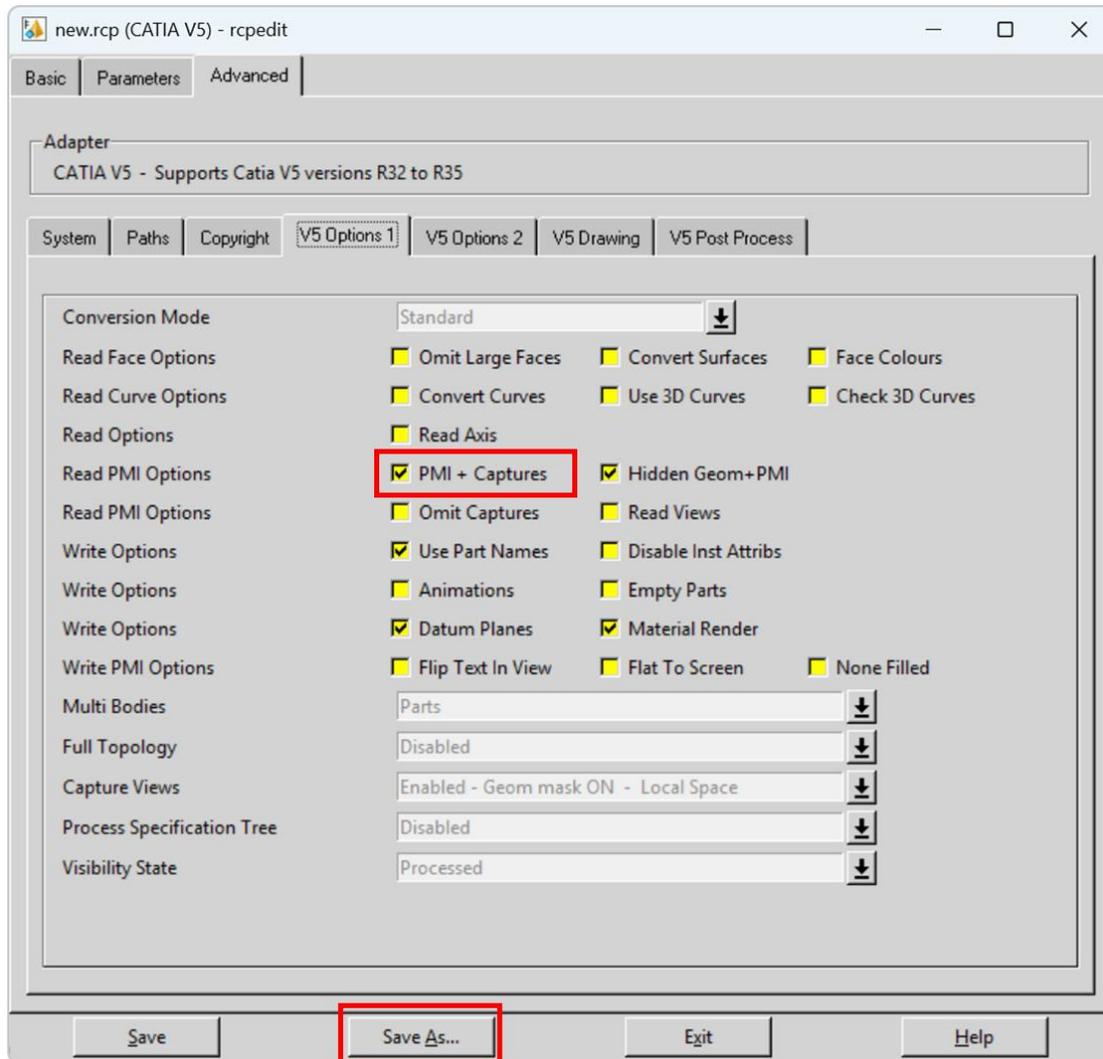
<p><b>Specified Post Process Exe</b></p>	<p>When Post Process Mode is set to Post Process, this field allows ANY post process executable or script to be specified. This allows customers to link in their own post processing behaviour.</p> <p>The specified Post Process executable will be called as follows:</p> <p><b><i>&lt;post_process_exe&gt; &lt;input_file name&gt; &lt;output_folder&gt; &lt;specified arguments&gt; v5_version &lt;CATIA V5 version&gt;</i></b></p>
<p><b>Specified Post Process Opts</b></p>	<p>Specify arguments to be passed to the Specified Post Process Executable</p>

## Creating a New Recipe File

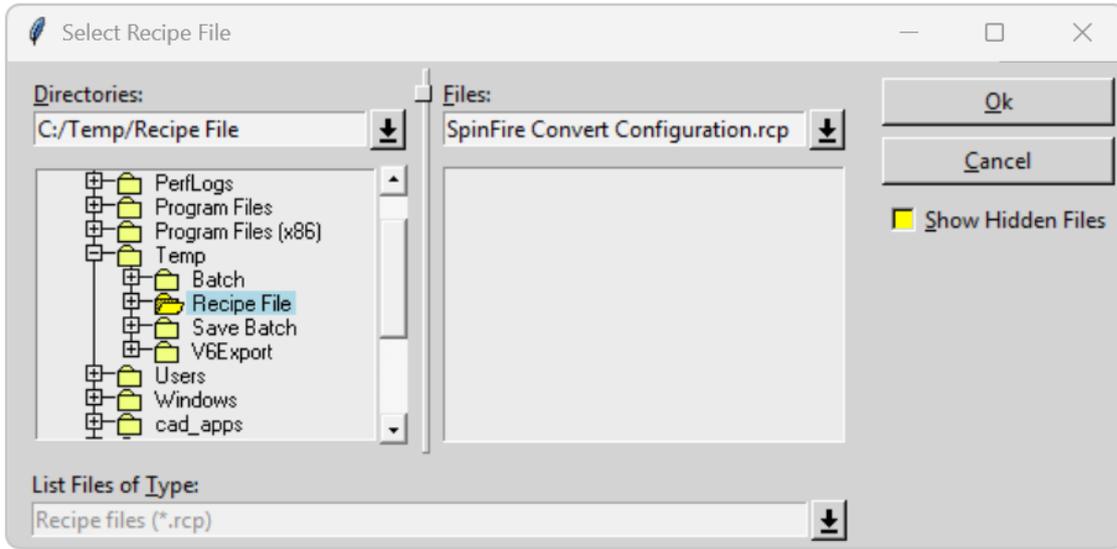
If a new recipe file is required, launch the **rcpedit.exe** command from within the **<Translator\_installation\_directory>\bin** directory in your SpinFire Convert install.

File Name	Date/Time	Type	Size
pvsthumb.exe	21/11/2024 10:13	Application	413 KB
rcpedit.exe	21/11/2024 10:11	Application	413 KB
theorem_catstart.exe	24/02/2025 12:30	Application	515 KB

Select the relevant options in the **rcpedit** window, then select **Save As**.



Select the required directory and input a filename, then click OK.



Name	Date modified	Type	Size
 SpinFire Convert Configuration.rcp	06/03/2026 15:19	RCP File	1 KB

## 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.

This .paj file shows CGR, IGES, STBOM, TXT, PDF and JT files being created for the 'as1.CATProduct' input file used in the example job file above. All of the alternate formats (**Additional File Types**) will be packaged into a .pvoa file which will also include the pvs and ol files.

It is important to check that the correct CATIA V5 licenses are available, since some alternative 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**.

Further details can be found in article **CS111916 - How to configure Additional File Types to be published for specific CAD Document types in Windchill PDMLink?**

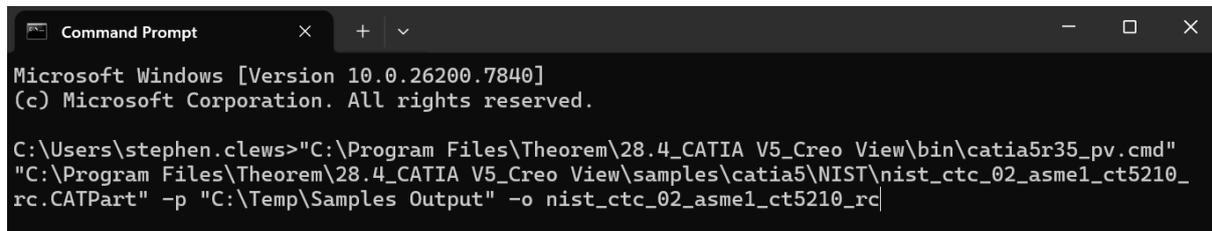
<https://www.ptc.com/en/support/article/CS111916>

## Translating on the Command Line

### Default Translation on 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 **<Translator installation\_directory>\bin** directory. The format of the command is as follows. (Note! The [XX] seen in the example below will be replaced with the version of CATIA V5 that you are using. For example, if you are using R35 then catia5r35 will be displayed.)

**<Translator installation\_directory>\bin\catia5rXX\_pv.cmd <input\_file> -p <output\_path> -o <output\_file>**



```

Microsoft Windows [Version 10.0.26200.7840]
(c) Microsoft Corporation. All rights reserved.

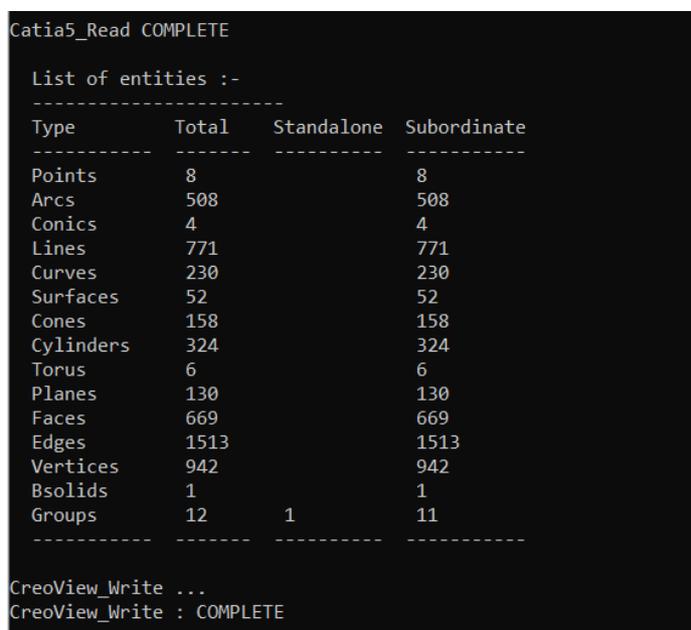
C:\Users\stephen.clews>"C:\Program Files\Theorem\28.4_CATIA V5_Creo View\bin\catia5r35_pv.cmd"
"C:\Program Files\Theorem\28.4_CATIA V5_Creo View\samples\catia5\NIST\nist_ctc_02_asme1_ct5210_
rc.CATPart" -p "C:\Temp\Samples Output" -o nist_ctc_02_asme1_ct5210_rc
    
```

The example above will translate the file to the output path specified. In this case:

**C:\temp\samples output\nist\_ctc\_02\_asme1\_ct5210\_rc.pvs**  
**C:\temp\samples output\nist\_ctc\_02\_asme1\_ct5210\_rc.ol**

Name	Date modified	Type	Si:
 nist_ctc_02_asme1_ct5210_rc.ol	05/08/2025 12:23	PTC Creo View Ge...	
 nist_ctc_02_asme1_ct5210_rc.pvs	05/08/2025 12:23	PTC Creo View Str...	

The following screen output should be expected when successfully translated:



```

Catia5_Read COMPLETE

List of entities :-
-----
Type          Total    Standalone  Subordinate
-----
Points        8         8
Arcs          508       508
Conics        4         4
Lines         771       771
Curves       230       230
Surfaces      52        52
Cones         158       158
Cylinders     324       324
Torus         6         6
Planes        130       130
Faces         669       669
Edges         1513      1513
Vertices      942       942
Bsolids       1         1
Groups        12        1         11
-----

CreoView_Write ...
CreoView_Write : COMPLETE
    
```

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

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

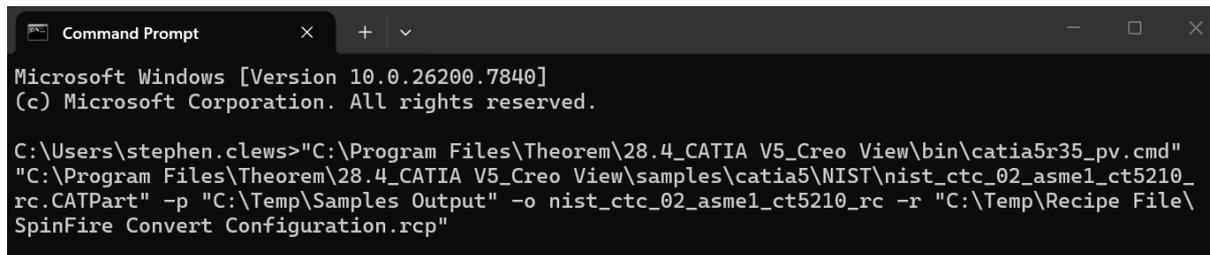
Setting	Result
<b>&lt;@File&gt;</b>	Read Options from the response file <file>
<b>-?-h</b>	For basic help page. UNIX may try to export the “?” so –h should be typed in quotes, for example, “h”.
<b>-d &lt;depth&gt;</b>	Set the conversion file depth. When converting an assembly file determines to what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

## Translating with a Recipe File on the Command Line

A recipe file which includes a set of options defined by the user in the recipe editor (*rcpedit*) can also be added to a command in the command prompt window using the correct syntax.

The format of the command which includes a recipe file is as follows. (Note! The [XX] seen in the example below will be replaced with the version of CATIA V5 that you are using. For example, if you are using R34 then catia5r34 will be displayed.)

**<Translator installation\_directory>\bin\catia5rXX\_pv.cmd <input\_file> -p <output\_path> -o <output\_file> -r <recipe\_file>**



```

Microsoft Windows [Version 10.0.26200.7840]
(c) Microsoft Corporation. All rights reserved.

C:\Users\stephen.clews>"C:\Program Files\Theorem\28.4_CATIA V5_Creo View\bin\catia5r35_pv.cmd"
"C:\Program Files\Theorem\28.4_CATIA V5_Creo View\samples\catia5\NIST\nist_ctc_02_asme1_ct5210_
rc.CATPart" -p "C:\Temp\Samples Output" -o nist_ctc_02_asme1_ct5210_rc -r "C:\Temp\Recipe File\
SpinFire Convert Configuration.rcp"
    
```

The example above will translate the file with the options defined in the recipe file to the output path specified. In this case:

**C:\temp\samples output\nist\_ctc\_02\_asme1\_ct5210\_rc.pvs**  
**C:\temp\samples output\nist\_ctc\_02\_asme1\_ct5210\_rc.ol**  
**C:\temp\samples output\nist\_ctc\_02\_asme1\_ct5210\_rc.pva**  
**C:\temp\samples output\nist\_ctc\_02\_asme1\_ct5210\_rc\_2.pva**  
**C:\temp\samples output\nist\_ctc\_02\_asme1\_ct5210\_rc\_3.pva**

Name	Date modified	Type	Si
 nist_ctc_02_asme1_ct5210_rc.ol	05/08/2025 12:19	PTC Creo View Ge...	
 nist_ctc_02_asme1_ct5210_rc.pva	05/08/2025 12:19	PVA File	
 nist_ctc_02_asme1_ct5210_rc.pvs	05/08/2025 12:19	PTC Creo View Str...	
 nist_ctc_02_asme1_ct5210_rc_2.pva	05/08/2025 12:19	PVA File	
 nist_ctc_02_asme1_ct5210_rc_3.pva	05/08/2025 12:19	PVA File	

## Translating Interactively from within CATIA V5

### Launching CATIA V5 with SpinFire Convert plug-ins

The CATIA V5 to Creo View adapter allows an active CATIA V5 Part or Assembly to be exported directly to Creo View from within the CATIA V5 application.

In order to translate from within CATIA V5, the application must be started using a SpinFire Convert environment, so that the appropriate plug-ins are available.

CATIA V5 can be started from a desktop shortcut created during installation.

Alternatively, it can be started via the script provided in the translator installation located in:

***<installation\_directory>\bin***

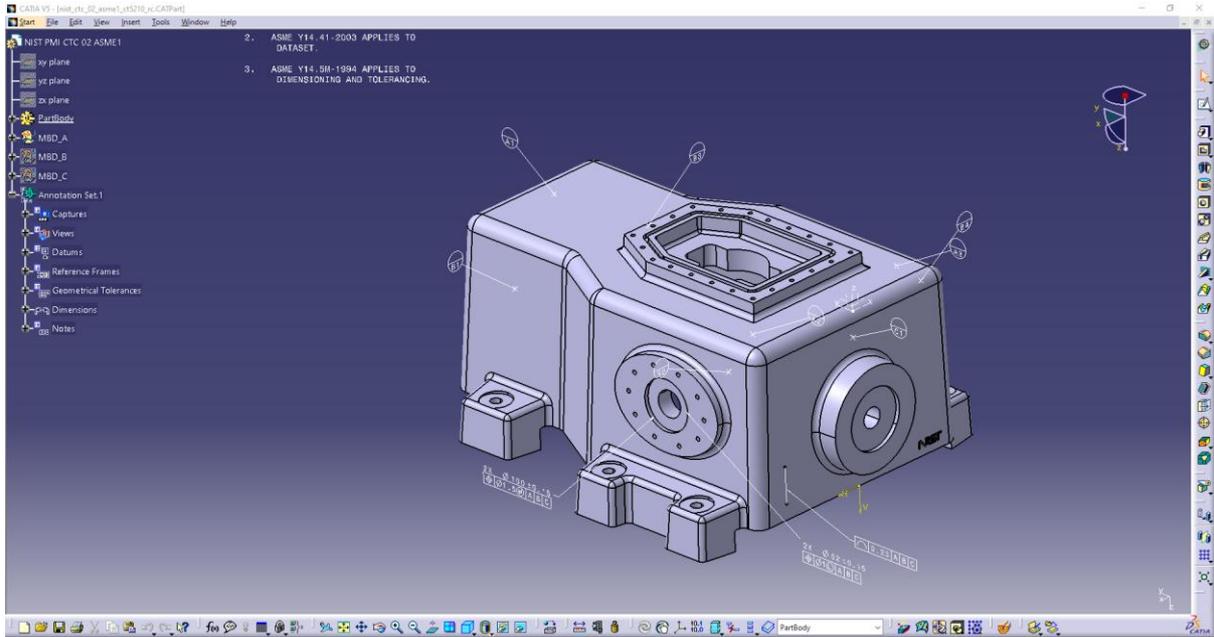
The script name is:

***catia5r<version>\_start.cmd***

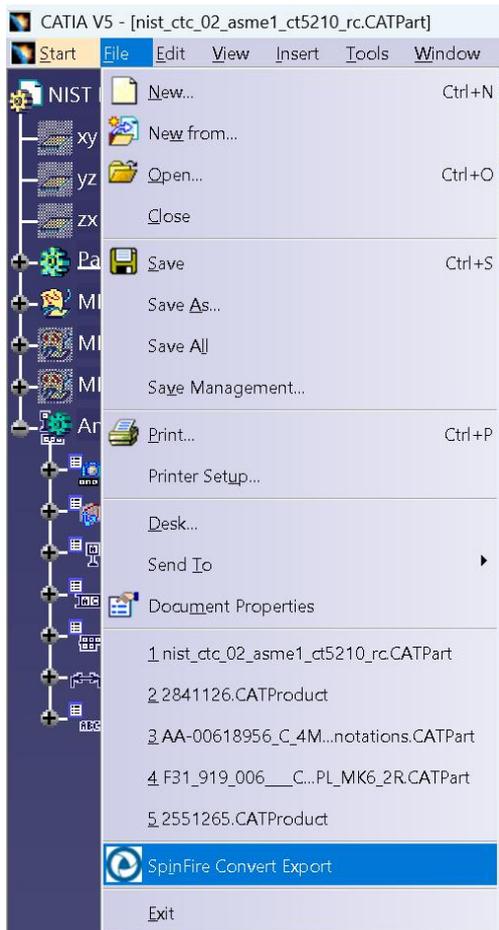
Where <version> is the version of CATIA V5 that you have installed – e.g. 32 for V5-6R2022, 33 for V5-6R2023, 34 for V5-6R2024 35 for V5-6R2025 etc.

## Default Translation from CATIA V5

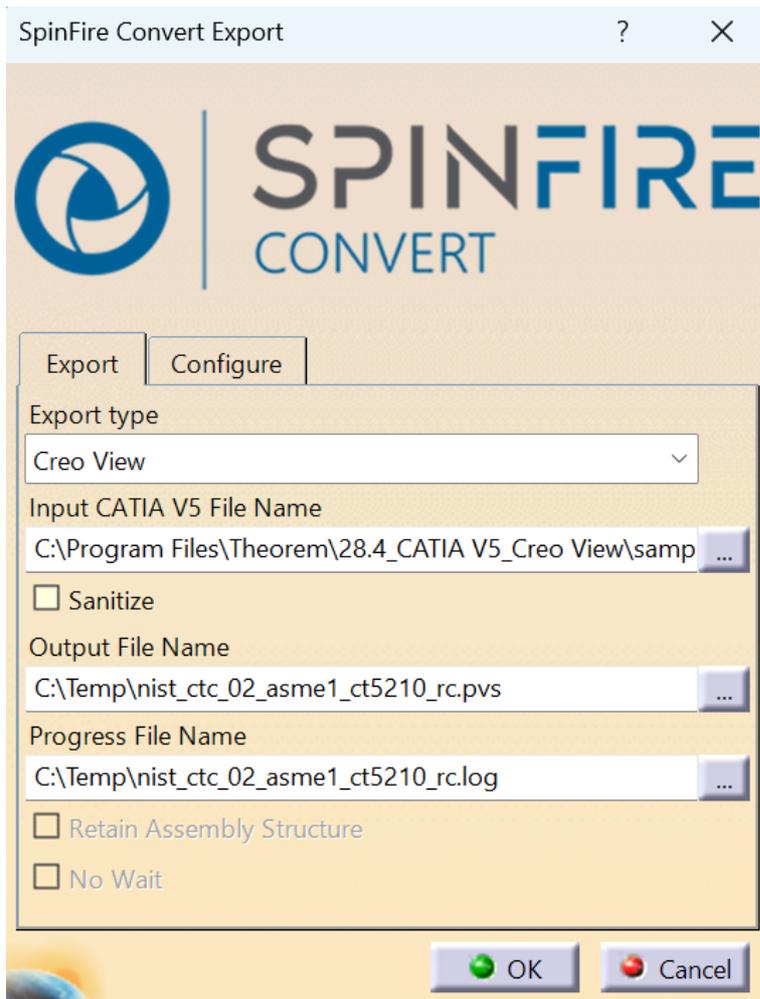
Once CATIA V5 has been started, open the part or assembly that is going to be exported to Creo View.



Select File, then SpinFire Convert Export.



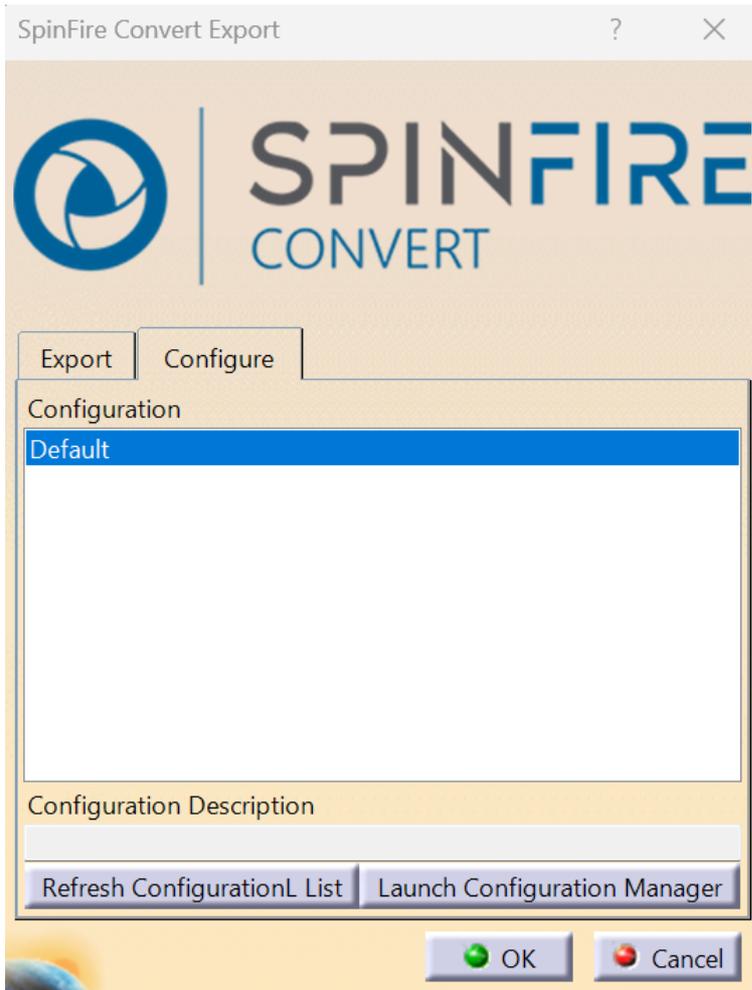
This will launch the SpinFire Convert Export panel.



The **Export** tab displays the Export type, Input CATIA V5 File Name, Output File Name and Progress File Name.

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, the Output File location and Progress File location can be modified by clicking on the  icon.

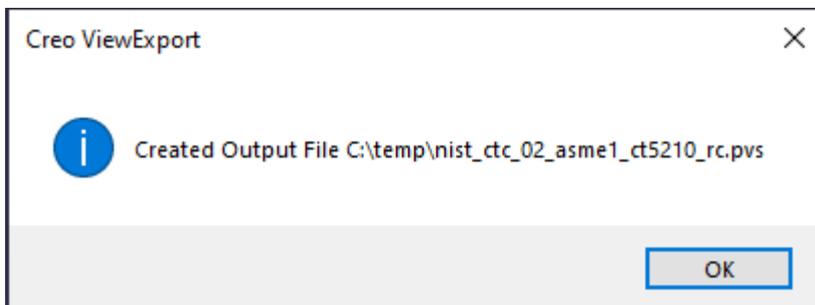
The **Configure** tab allows the user to select a Configuration.



In this tab, a list of configurations that are available to use will be displayed including the standard Default configuration.

When a configuration has been selected, click OK to initiate the translation. The configuration will be highlighted in blue once it has been selected.

The active part or assembly will then be translated to Creo View with pvs and ol files created in the output directory specified. A pop-up message will be displayed to confirm when the translation is complete.

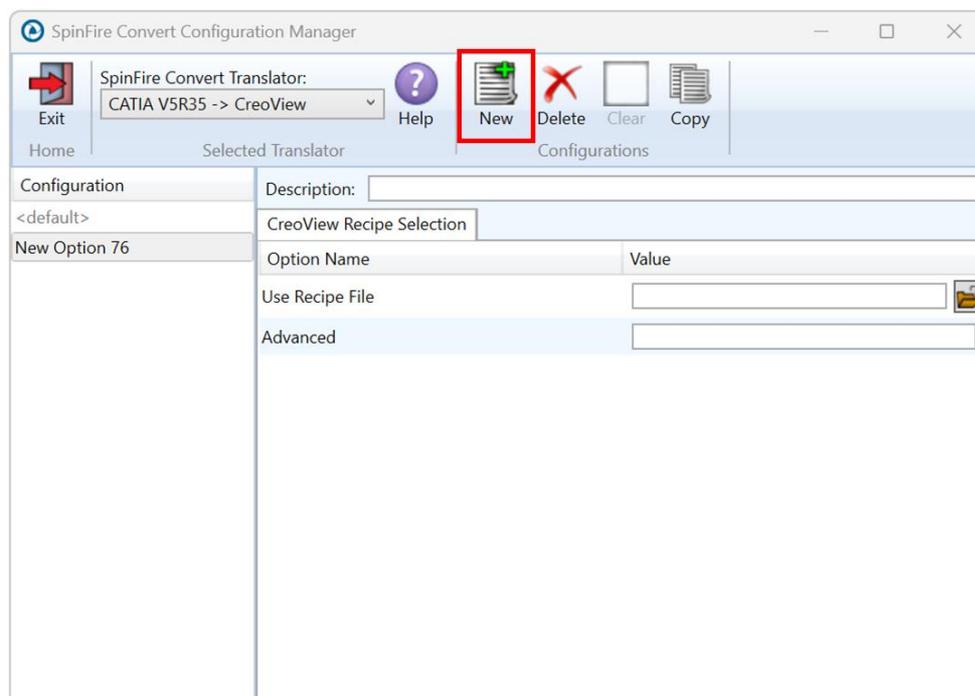


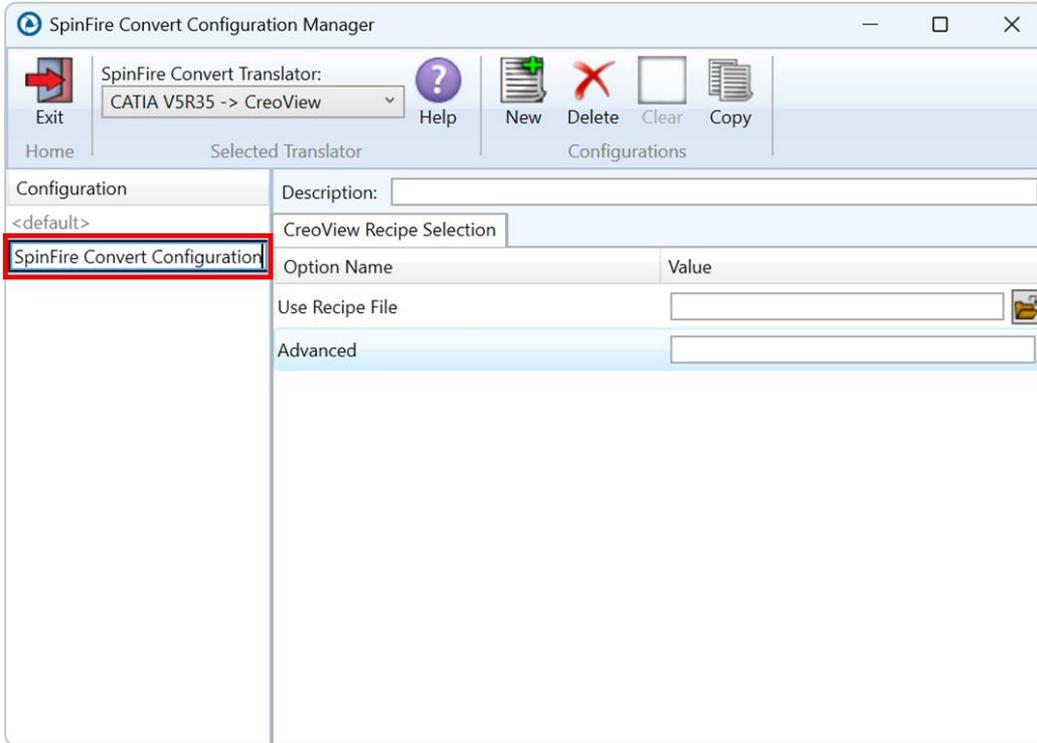
## Configuration Manager

If a suitable Configuration is not available or if a different configuration is required, then a new Configuration can be created by the user. To create a new configuration, select the **'Launch Configuration Manager'** command.

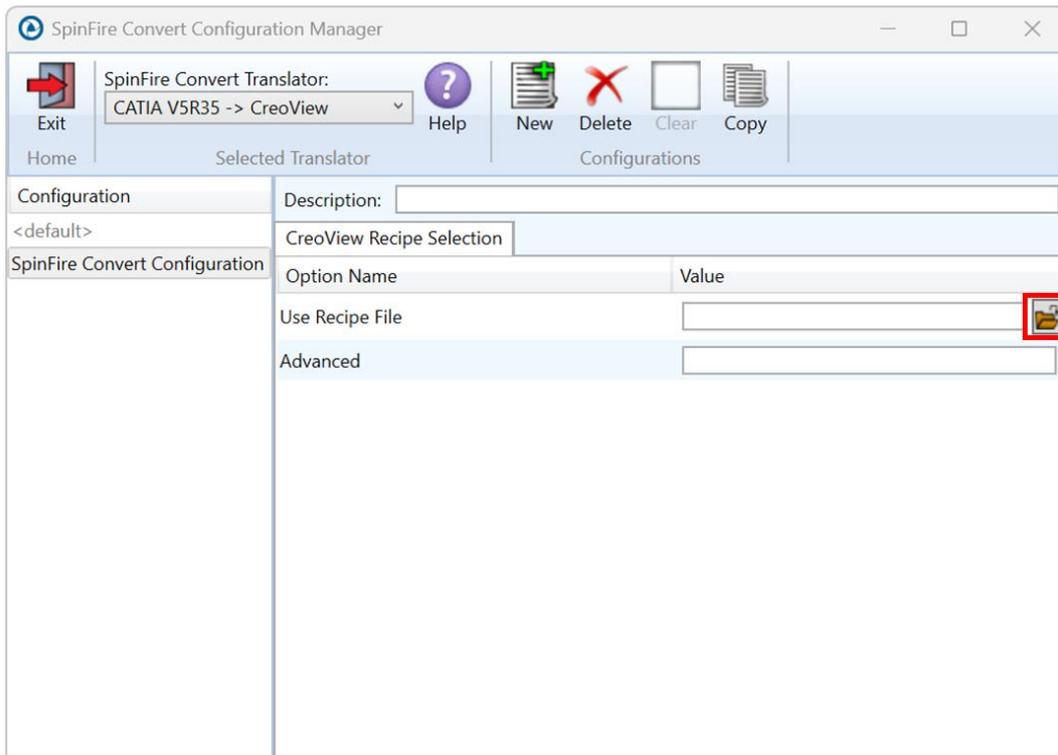


In the configuration manager window, select New, then rename the configuration as required.

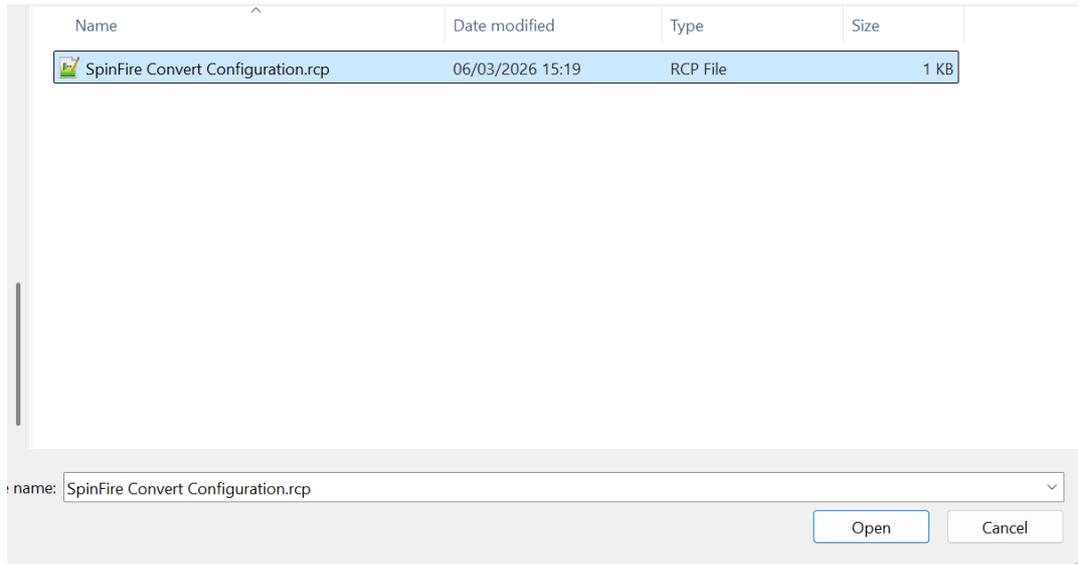




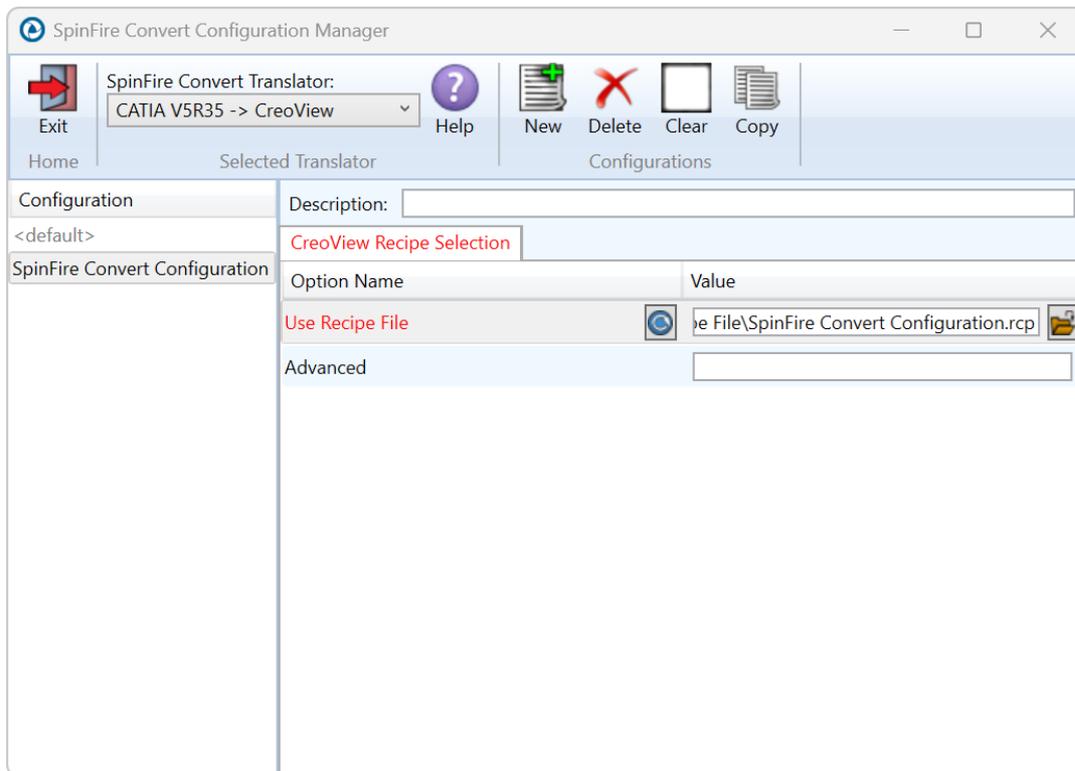
In the configuration manager window, only 1 tab will be displayed named CreoView Recipe Selection which will include a Use Recipe File option. To add a recipe file to the configuration, select the folder icon next to the Use Recipe File option field.



In the Select a CreoView Recipe file pop up window, navigate to the directory where the recipe file is saved. Select the file and click Open.



The recipe file will then be added to the configuration. Click Exit to close the Configuration Manager window.



Any new configurations created will be displayed in the Configurations list once it has been refreshed. To do this select **'Refresh Configuration List'**.



Select the new configuration to make it the active configuration.



## 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.

If the 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 in 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 including CGM, HPGL, PDF or TIF. These can be selected via the recipe editor.

## 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 “tscsummaryj” 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 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 SpinFire Convert messages added to the worker logs are always prefixed by **'TS:'** and are written at two levels of detail 0x01 and 0x10. These messages are enabled via the `-vm` command, 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 the positive value of the summary code,

***8 => No entities found.***

**Process Timeouts** – Timeouts allow a user to control when an individual translation invoked from a Windchill environment should timeout.

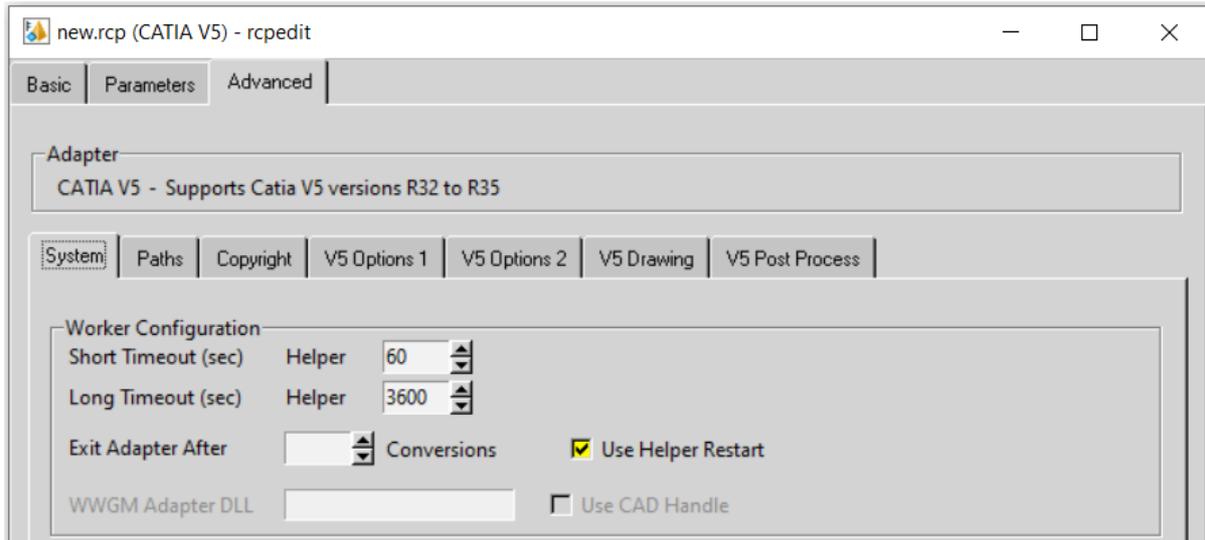
The Windchill interface allows 2 distinct timeout types to be defined, Long and Short.

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 CATIA V5 Adapter and click create
3. Having selected CATIA V5 from the adapter pull-down, select the Advanced tab followed by the System Tab, the recipe editor GUI will then 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.

**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

This Appendix details how to define and configure the CATIA V5 and SpinFire Convert environments 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. 32, 33, 34, 35 etc.

**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.

**SpinFire Convert Installation directory** - The SpinFire Convert translator installation directory is set during the install process and is subsequently displayed in the *ts\_env.bat* file after the install has been completed. 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 SpinFire Convert 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.

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 sections.

## CATIA V5 Environment DIRENV & ENV

The default location for CATIA V5 to store its global environment files is in:

```
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 translator by defining in the **ts\_env.bat** the environment variable **CATIAV5\_DIRENV**:

```
set CATIAV5_DIRENV=/some/directory
```

If the SpinFire Convert 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 SpinFire Convert translator will attempt to create its own environment file called **TheoremCatia5R<XX>.txt**. The user must therefore have write permissions 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.V5R32.B32
```

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.V5R32.B32
```

## Checking the CATIA V5 Environment

A script is provided to check that the CATIA V5 environment is set up correctly. In a command prompt window run the command script:

```
%TS_INST%\bin\checkcatia5r<XX>env.cmd
```

## Checking the SpinFire Convert Shared Library

A script is provided to ensure that the CATIA V5 environment is compatible with the SpinFire Convert shared libraries. In a command prompt 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 SpinFire Convert translator correctly and that the correct version of the CATIA V5 translator products has been installed.

## Appendix B – SpinFire Convert Configuration File (ts\_cfile)

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 CATIA V5\_Creo View Adapter configuration file (ts\_cfile). To invoke these controls, it is necessary to create the configuration file and also set 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 **V5 Options 2 – More translation settings, Additional Options file**.

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:\TEMP\configuration.txt
```

## Appendix C – Drawing Processing Options

The following table shows the effect of the 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 – SpinFire Convert Support Advanced Options

The following environment variables are available to modify the Adapters behaviour under the guidance of the SpinFire Convert support team.

It is recommended, under guidance from the SpinFire Convert Support team 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 <b>Diagnostic Logs -&gt; Info</b> 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 <b>-vm &lt;level&gt;</b> 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).
TS_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>	Causes the named representation for a node to be read. If this does not exist, then 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 end 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 then this 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 an alternative file 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 be in a .pvoa file.

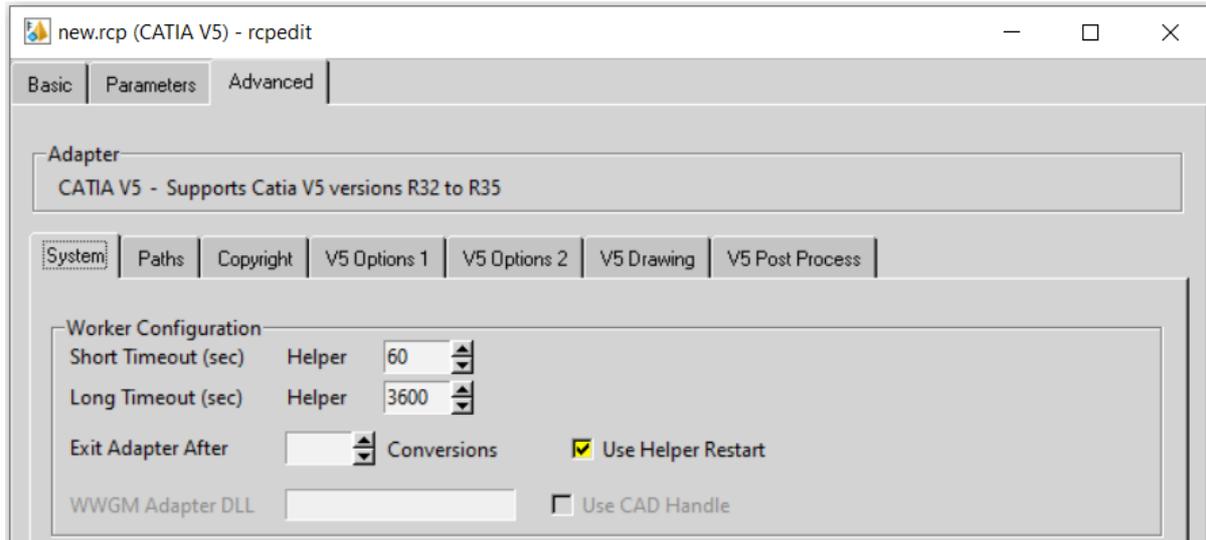
This string of invalid characters used by the JT Add On can be set by the user, via the environment setting :

```
set TS_V5_CV_JT_INVALID_CHARS="\n\r\t`~!@#$$%^&*()-+=|\\"'";,./<>?|[]{}"
```

This string would have to mirror the Filter>FilenameSanizationSet value in the JT config file. It is suggested that the SpinFire Convert support team should be contacted in the unlikely event that this is 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 be enabled by a recipe setting:



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

## PMI Options

If flat to screen or flipped PMI is enabled then 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 the environment variable:

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

## 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
<b>TS_OUTPUT_STDERR</b>	1	Redirects error messages to the screen log, or a log specified via the <code>-vL</code> 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
<b>TS_MOTION_FILE_UNIT</b>	m, cm, inch, feet, yard or <i>value</i>	Specify a unit or a value. where value is mm/unit required

## PVZ Output

PVZ output can be enabled by setting the recipe editor option:

***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 to be selected.

## 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 it is only used by a 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 customers 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.

An additional option can be used to provide an alternative output directory for the generated 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***

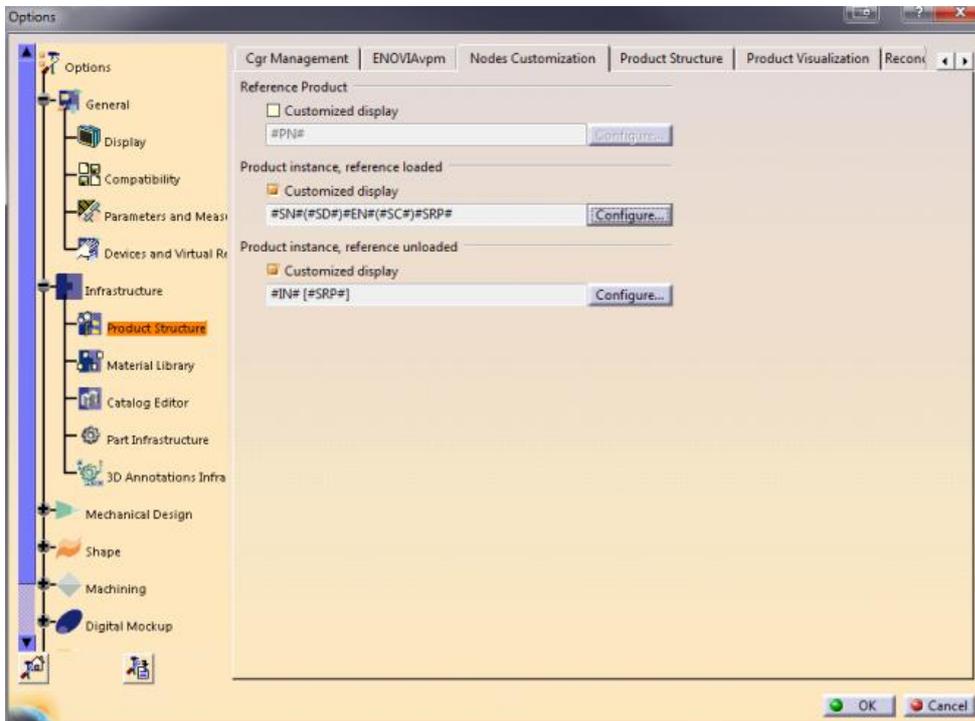
would result in a higher level of tessellation, so consequently a larger file is produced.

***adapter/lod=Standard***  
***adapter/chordHeight=10.0***

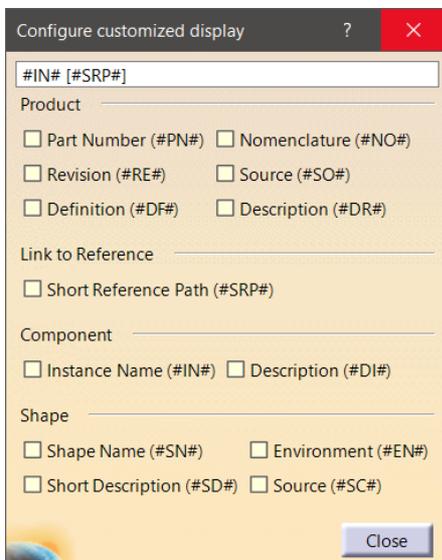
would result in a lower level of tessellation, so consequently a smaller file is produced.

## Instance Naming

Variable	Value	Description
<b>INSTANCE_NAMING</b>	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)
<b>INSTANCE_RENAMING</b>	<STRING>	Allows this to be specified on the command line. Details of the options are shown below



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.

Support for renaming the instance on the command line is also available when using the option INSTANCE\_RENAMING <string>, where <string> (#XX#) 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# - Environment,  
#SD# - Short Description,  
#SC# - Source

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

## Flat to screen OR flipped PMI

If these are enabled the PMI nodes are 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.

This can be disabled via the environment variable:

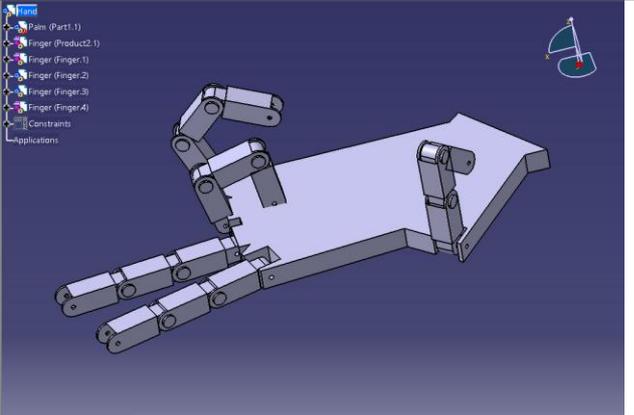
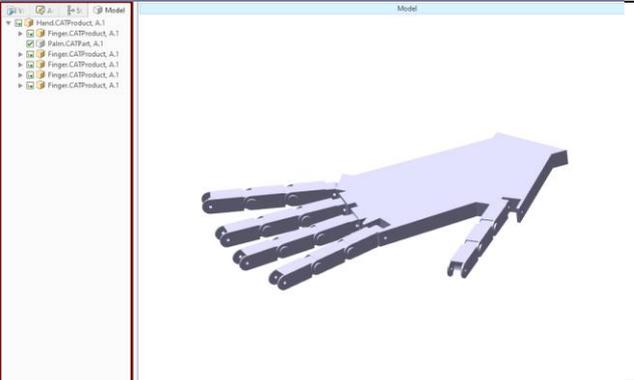
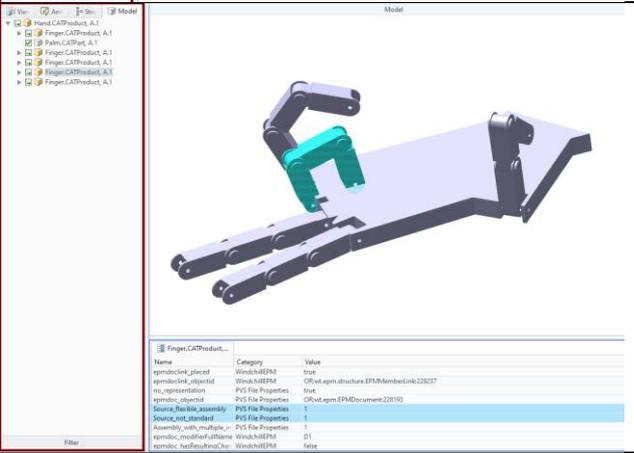
***TS\_V5\_CVW\_PMI\_PMI\_OMIT\_LEADERS=1***

## Appendix E – What’s New

### What’s new in 24.0

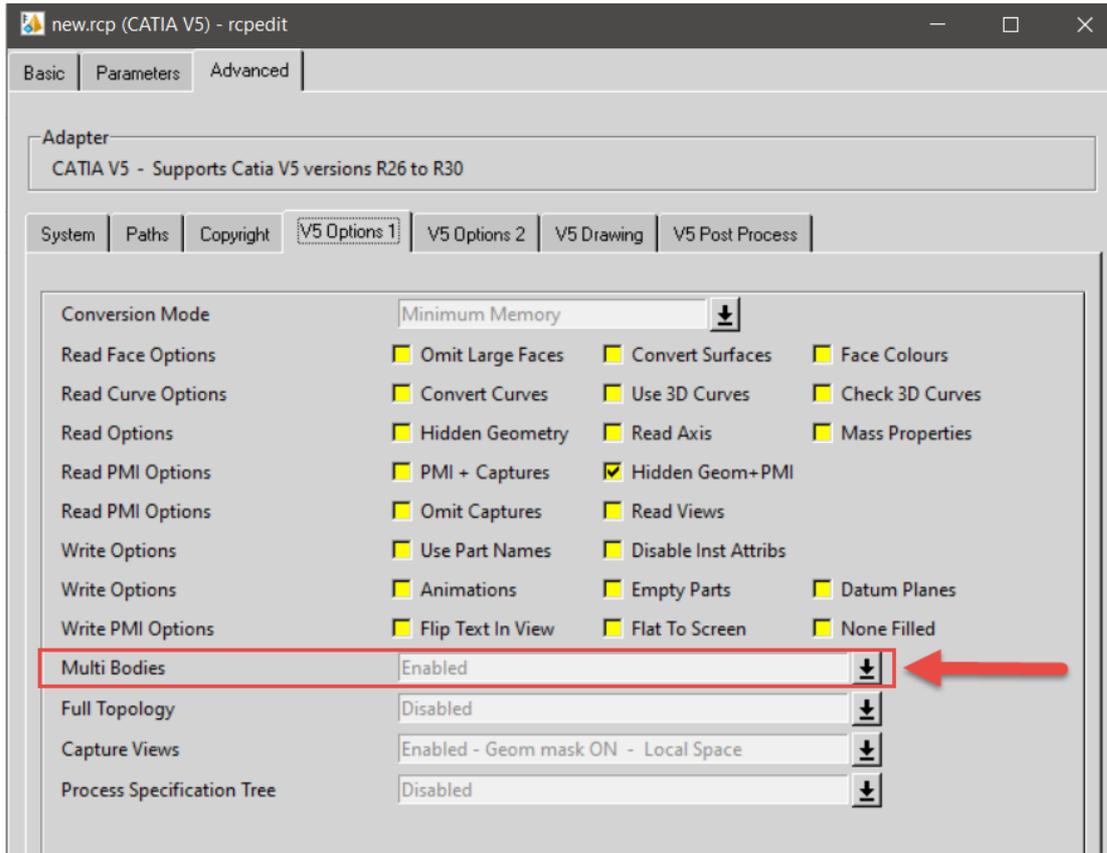
#### Flexible Assembly Visualization

This release allows the end-user to make use of Flexible Assembly Visualization in Windchill. This behaviour is turned on by default in the translator.

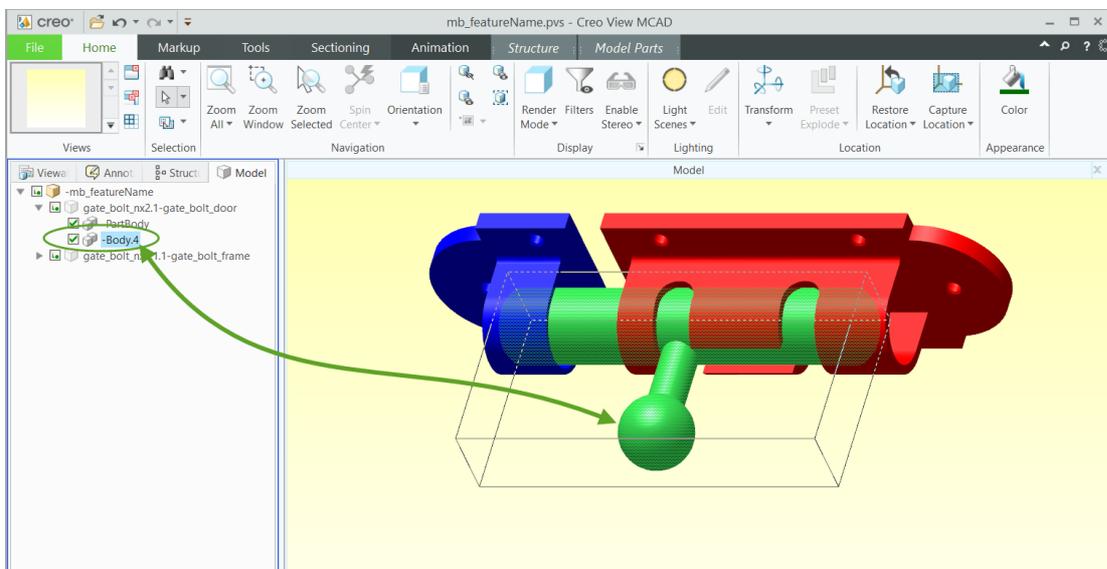
<p>Flexible assembly shown in CATIA V5</p>																															
<p>Display in Creo View when value for “Flexible assembly features in Dynamic CAD Structures ” is set to “No”</p>																															
<p>Display in Creo View along with attributes, when value for “Flexible assembly features in Dynamic CAD Structures ” is set to “Yes”</p>	 <table border="1" data-bbox="853 1720 1332 1832"> <thead> <tr> <th>Name</th> <th>Category</th> <th>Value</th> </tr> </thead> <tbody> <tr> <td>epmdec-link_gleacd</td> <td>WindchillEPM</td> <td>true</td> </tr> <tr> <td>epmdec-link_nolpstd</td> <td>WindchillEPM</td> <td>CRU:epmstructure.EPM/MemberLink.226237</td> </tr> <tr> <td>ru_representation</td> <td>PVS File Properties</td> <td>true</td> </tr> <tr> <td>epmdec_3dobject</td> <td>PVS File Properties</td> <td>CRU:epm.EPM/Document.226193</td> </tr> <tr> <td><b>Flexible assembly</b></td> <td><b>PVS File Properties</b></td> <td><b>1</b></td> </tr> <tr> <td>Source_not_Standard</td> <td>PVS File Properties</td> <td>1</td> </tr> <tr> <td>assembly_with_multiple...</td> <td>PVS File Properties</td> <td>1</td> </tr> <tr> <td>epmdec_modelFullname</td> <td>WindchillEPM</td> <td>D1</td> </tr> <tr> <td>epmdec_topResultCo...</td> <td>WindchillEPM</td> <td>File</td> </tr> </tbody> </table>	Name	Category	Value	epmdec-link_gleacd	WindchillEPM	true	epmdec-link_nolpstd	WindchillEPM	CRU:epmstructure.EPM/MemberLink.226237	ru_representation	PVS File Properties	true	epmdec_3dobject	PVS File Properties	CRU:epm.EPM/Document.226193	<b>Flexible assembly</b>	<b>PVS File Properties</b>	<b>1</b>	Source_not_Standard	PVS File Properties	1	assembly_with_multiple...	PVS File Properties	1	epmdec_modelFullname	WindchillEPM	D1	epmdec_topResultCo...	WindchillEPM	File
Name	Category	Value																													
epmdec-link_gleacd	WindchillEPM	true																													
epmdec-link_nolpstd	WindchillEPM	CRU:epmstructure.EPM/MemberLink.226237																													
ru_representation	PVS File Properties	true																													
epmdec_3dobject	PVS File Properties	CRU:epm.EPM/Document.226193																													
<b>Flexible assembly</b>	<b>PVS File Properties</b>	<b>1</b>																													
Source_not_Standard	PVS File Properties	1																													
assembly_with_multiple...	PVS File Properties	1																													
epmdec_modelFullname	WindchillEPM	D1																													
epmdec_topResultCo...	WindchillEPM	File																													

## Multi Body Components

Multi-body components are now supported in Creo View and the translator 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 using the **'Disabled'** option. For version 28.0 the default option has been changed to **'Parts'** instead of **'Enabled'**.



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

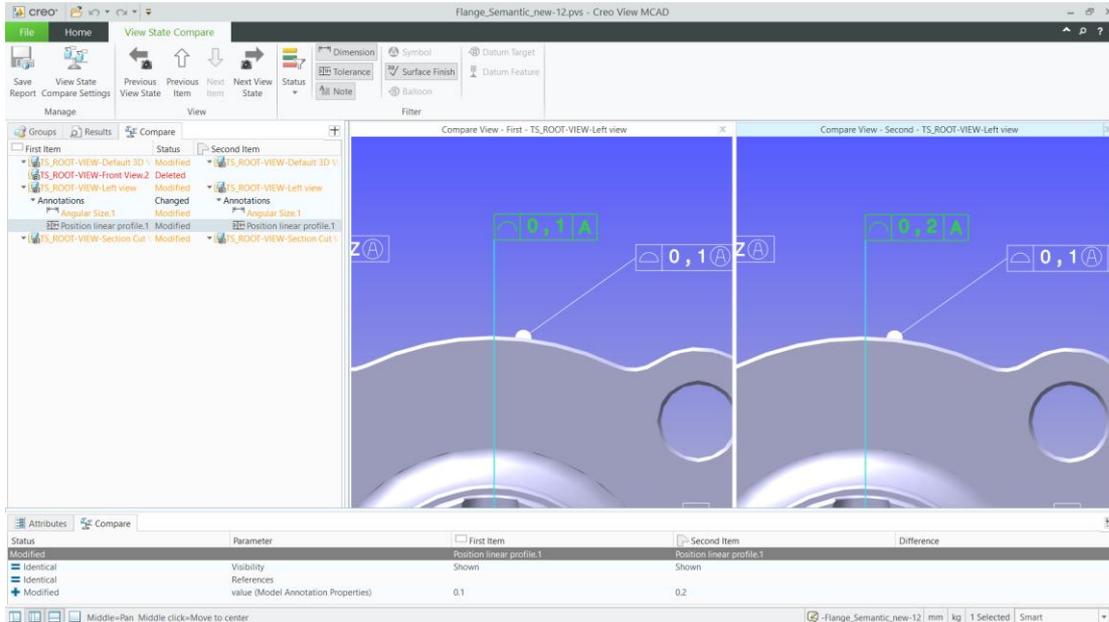


## View State Comparison

A new Creo View tool, View State Comparison 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

### Tessellation now uses default bounding box functionality

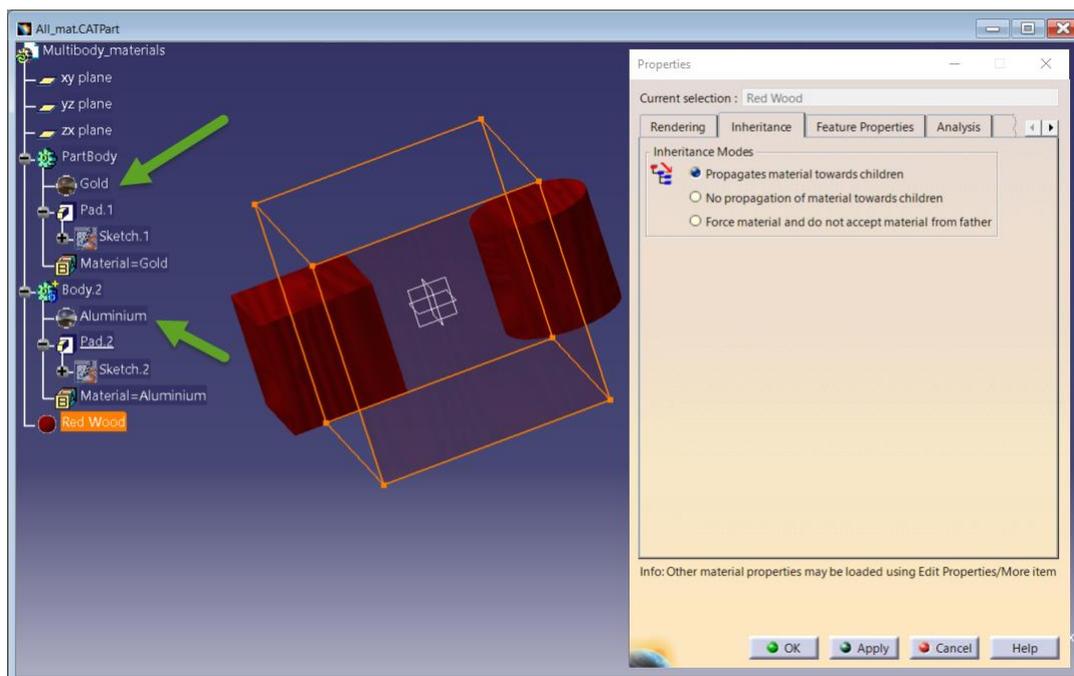
This release has moved the default tessellation strategy closer to that of PTC. The tessellation is based upon the solid and surface geometry bounds. In most cases, better or equal results have been observed when compared with the previous revision. If there is any degradation in the tessellation in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

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

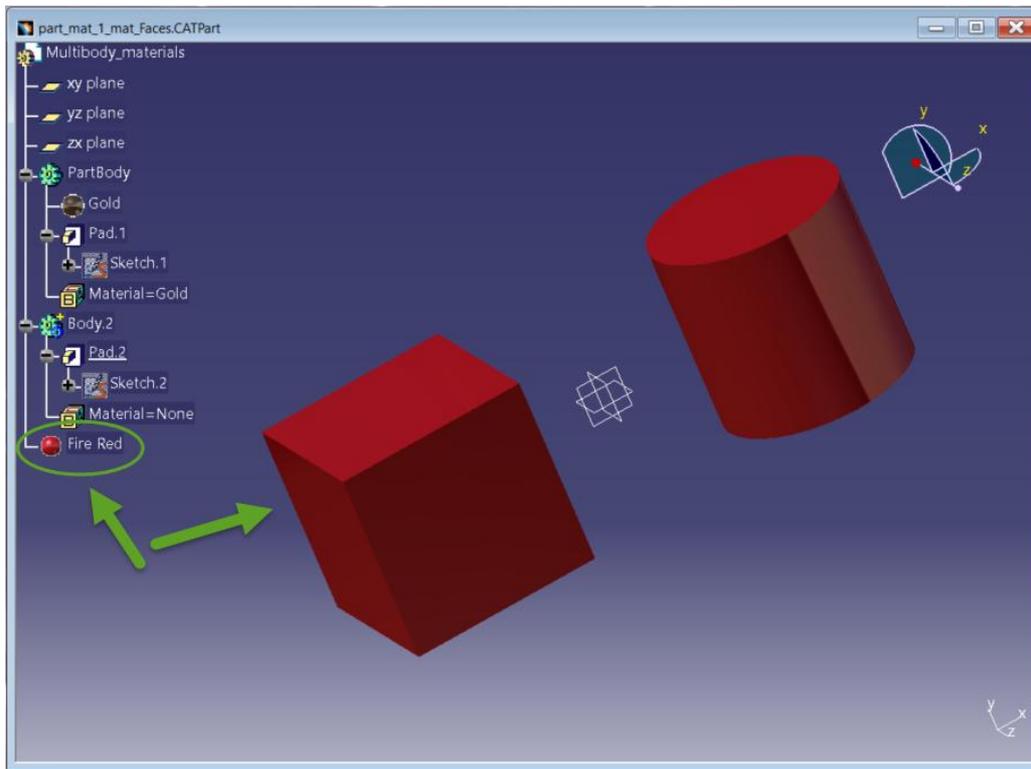
Materials and mass properties are now read by default when available. This will give more accurate mass property results in Creo View for single body and multibody components. Material properties only include colours but not texture maps. i.e. a material such as gold will have its colour mapped, whereas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



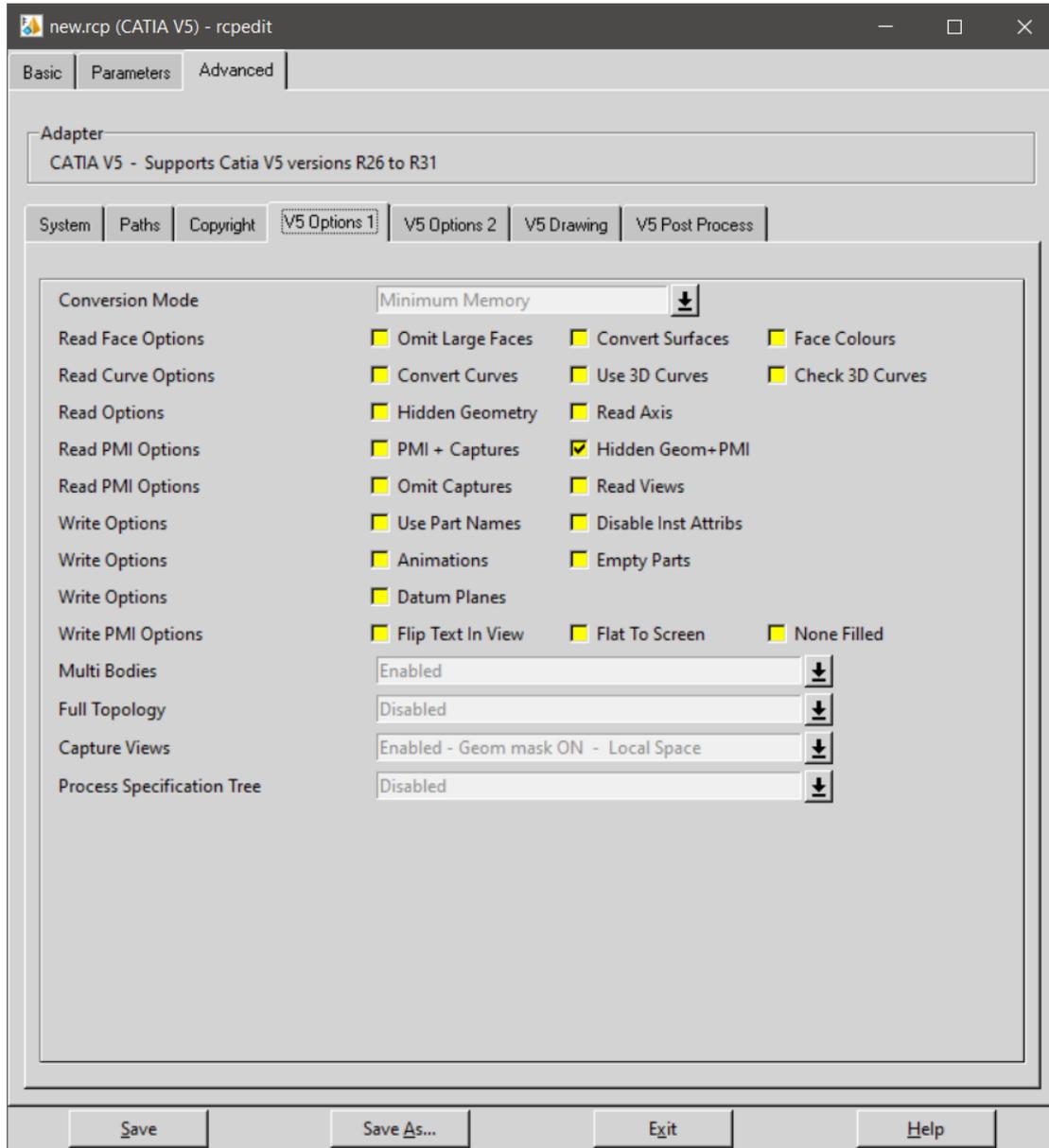
In the above example, because Red Wood has a texture map applied then the translated output will be the default body colour. However the Part level mechanical properties will be correctly washed down to the lower level bodies.



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.



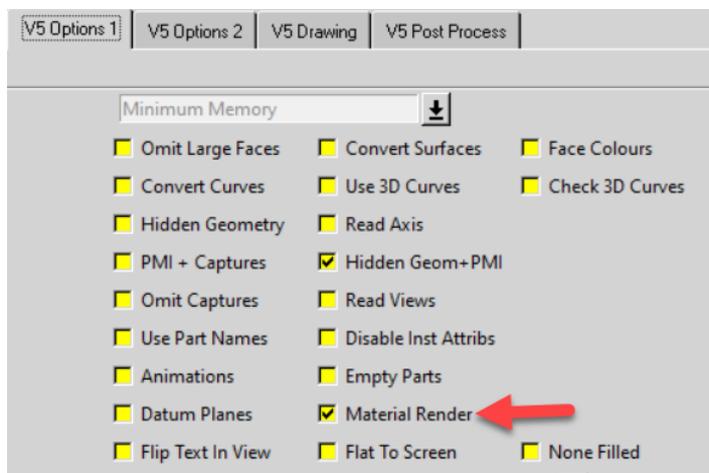
## What's new in 25.2

### 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. This new option is named Material Render, See Image below:

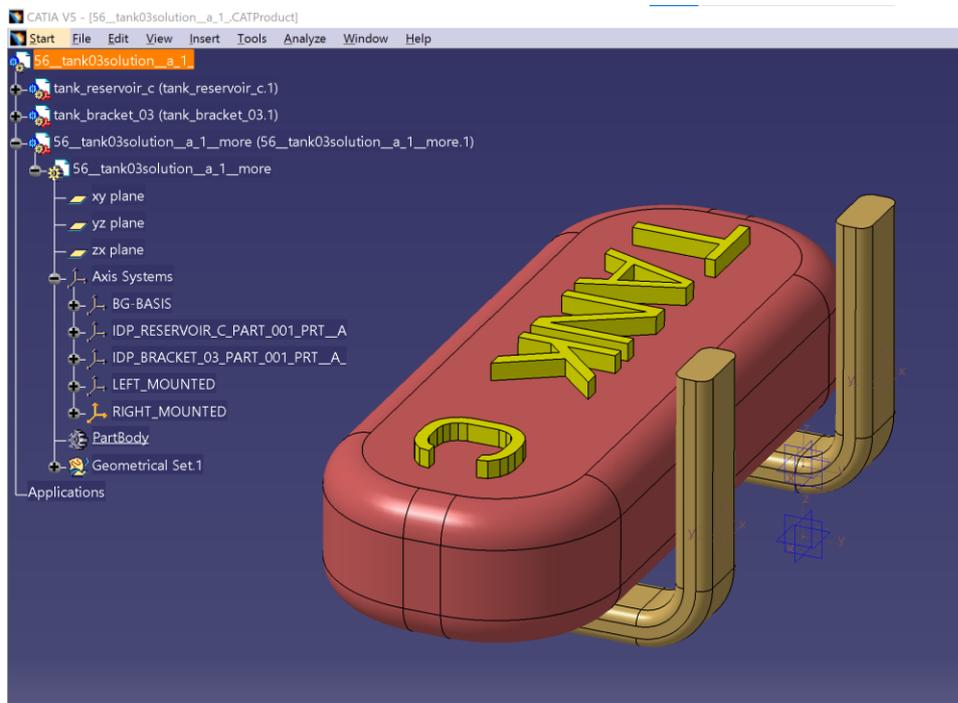


## What's new in 25.4

### 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 a Locator is a 'named' Axis System, for example 'LEFT\_MOUNTED' or 'RIGHT\_MOUNTED' as displayed in the example below:



The process is then activated by setting two 'Filters'

#### File Filter

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 lower-level 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.

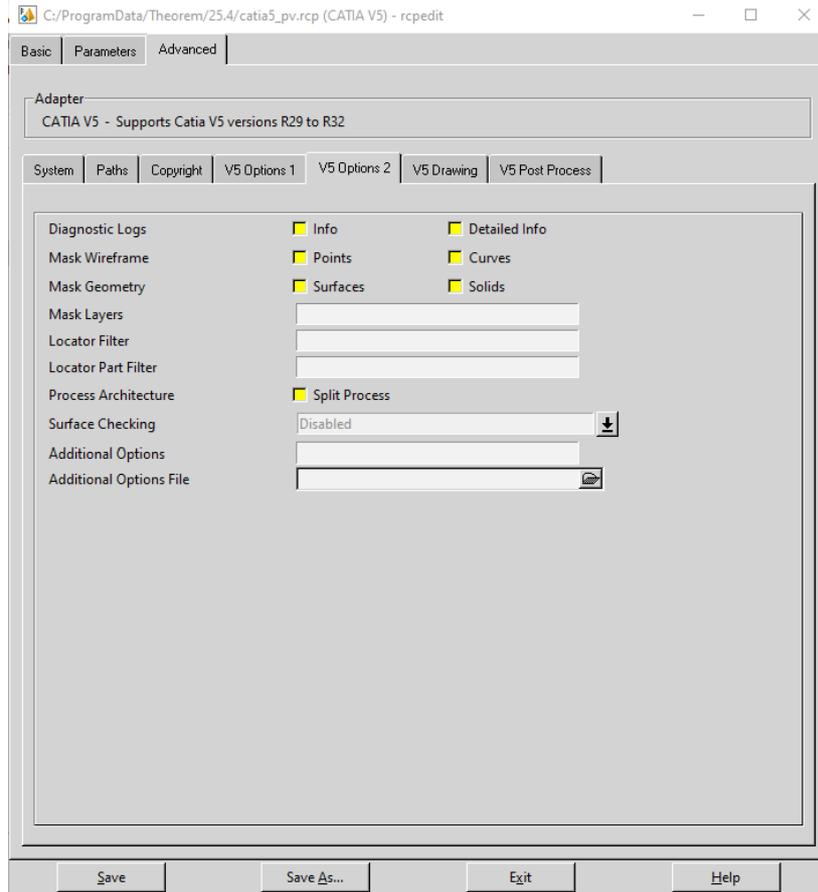
- The filter will allow a single asterix anywhere in the root string e.g. Skel\* will find both Skeleton.CATPart and Skeleton\_loc.CATPart or Skel\*loc will only find Skeleton\_loc.CATPart.
- Multiple parts can also be specified but these must be the full name e.g. Skeleton,Loc but not Skel\*,\*Loc.

#### 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.

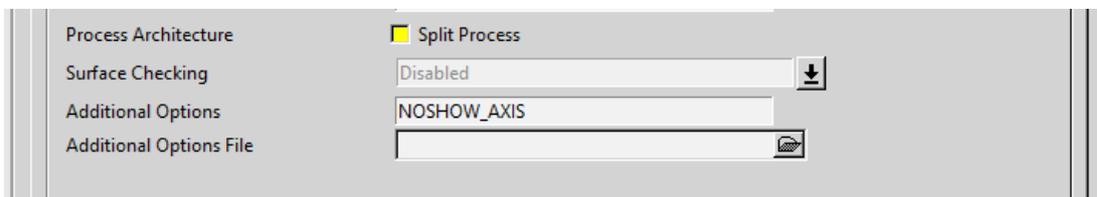
## Recipe File Changes

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



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.



- 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 26.0

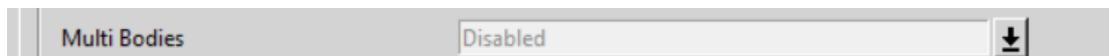
### Quick View

QuickView for large assemblies is now available. Please refer to PTC article [CS381237](#) for further information.

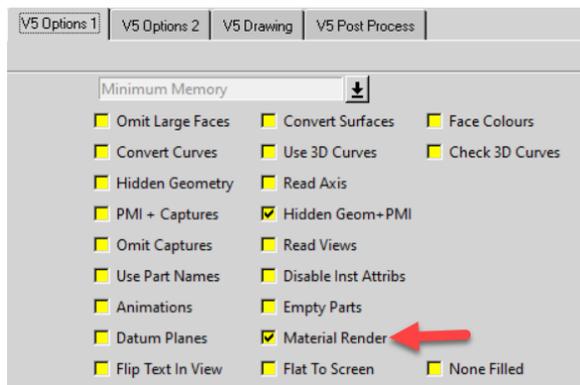
## What's new in 26.3

### Multi-Body Parts and Materials

A reminder that the defaults for translation have changed over the last couple of revisions. Multi-body components are now read/created by default. This can be disabled in the recipe settings.



Materials are now read by default; the translator will also try to render the Creo View output using the CATIA material attributes. This can affect the face colour display in the Creo View output. It is possible to turn this rendering off, so that body and face colours are rendered instead. Toggle off Material Render for this to be applied.



The translator will now always try to create mass properties for each body in a CATPart. For some parts (e.g. solids which have degraded to faces or a mixture of solids and surfaces) this can take a very long time without providing a useful mass properties result.

A new option MPROPS\_OFF has been introduced to allow translation to take place more quickly.

### Locators

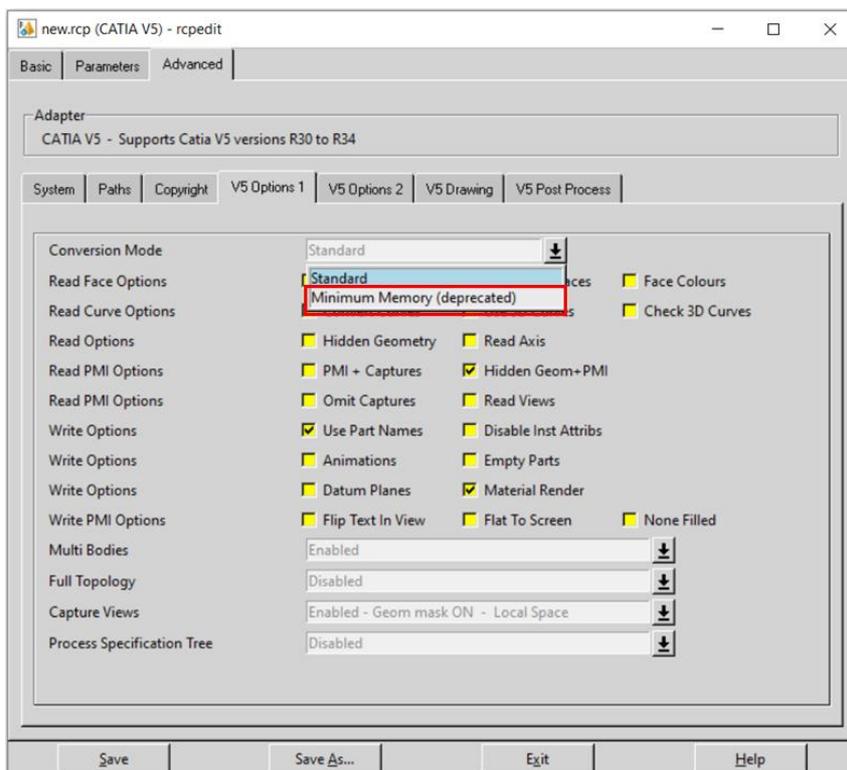
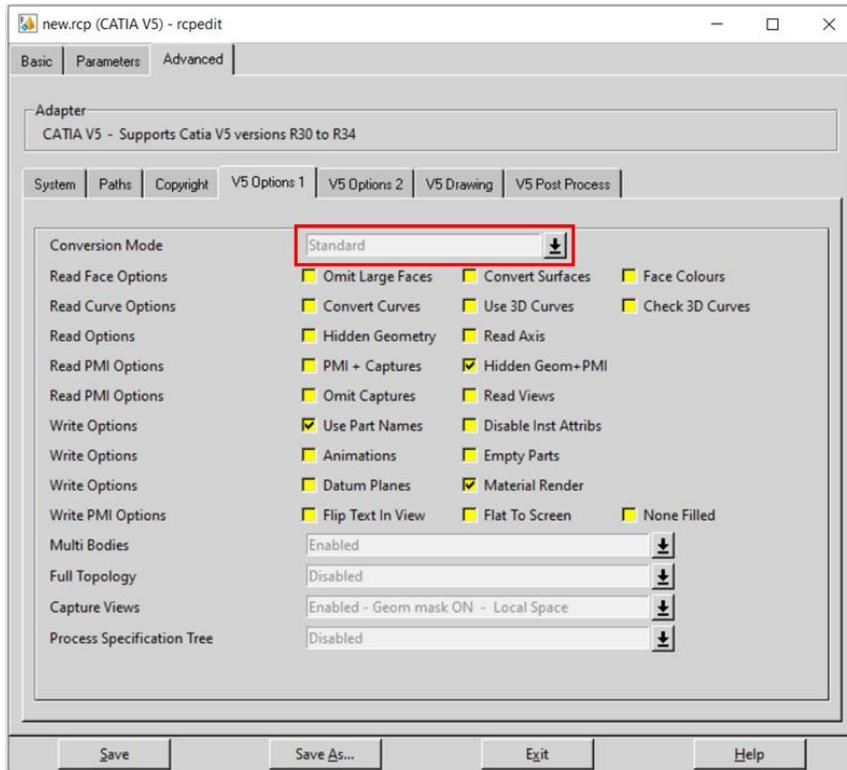
A number of additional options/enhancements are now possible when using locators.

- Multiple Wildcards "\*" are now recognised in Locator Filter and Locator Part Filter e.g. \*LOC\*
- Deactivated axis systems are no longer exposed as locators
- Empty skeleton Parts now publish successfully

## What's new in 27.2

### Conversion Mode

Default option for conversion mode set to **'Standard'** replacing **'Minimum memory'**. Minimum Memory mode also deprecated in this version with this ultimately being removed in future releases.







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