User Guide
3DExperience - Creo View

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Contents

About Theorem

Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of our technology & services by world leading customers.

We are proud that the vast majority of the world's leading Automotive, Aerospace, Defence, Power Generation and Transportation companies and their Supply chains use our products and services daily. Working closely with our customers, to both fully understand their requirements and feed their input into our development processes has significantly contributed to our technology and industry knowledge.

Theorem Solutions is an independent UK headquartered company incorporated in 1990, with sales and support offices in the UK and USA. Theorem has strong relationships with the major CAD and PLM vendors, including Autodesk, Dassault Systemes, ICEM Technologies (a Dassault company), PTC, SolidWorks, Spatial Technology and Siemens PLM Software. These relationships enable us to deliver best in class services and solutions to engineering companies worldwide.

Theorem's Product Suite

Theorem have 3 main Product brands. These are:

CAD**Translate**

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

See our website for more detail.

CAD**Publish**

The creation of documents enriched with 3D content

See our website for more detail.

Theorem**XR**

Visualization for Augmented (AR), Mixed (MR) and Virtual (VR) Reality applications

See our website for more detail.

3DExperience to Creo View – CADTranslate

The 3DExperience to Creo View Uni-directional Translator

The 3DExperience to Creo View translator may be installed on a number of machines each accessing a central network-floating license.

The 3DExperience to Creo View Translator is a Uni-directional direct database converter between the Dassault Systemes 3DExperience application and the Creo View file format, used by the PTC Visualisation products.

It enables the user to convert all forms of 3D Mechanical Design Geometry and Assembly data, together with system defined attribute information and colour information, between these two systems. This product is designed for companies using 3DExperience who have selected Creo View to be their main method of visualisation and/or collaboration either internally or between OEMs and their customers or suppliers.

It is also a major method of visualisation and is used by companies using Creo View based solutions to translate their 3DExperience data into the Creo View format.

The translator can be invoked in either an interactive or batch mode.

Primary Product Features

- Converts geometry including analytic data, solid models, and surfaces to the Creo View ".ol" file format

- Converts assembly structures and part orientations to the Creo View ".pvs" file format

- Converts attribute data such as meta-data, colour and layer information and 3DExperience properties. Data can be filtered by layer and entity type during processing. Geometry can be filtered and selectively processed.

- Integrated with the 3DExperience installation.

- The conversion process can be run Interactively (Windchill Independent only) or in Batch Mode.

- Uses the 3DExperience API and PTC Creo View DAPI to read and write data.

- The Creo View data created using this Adapter can be imported into the Arbortext IsoDraw CAD process

Primary Product benefits?

- Direct conversion from 3DExperience to Creo View reduces processing time, simplifies integration and retains accuracy of the model

- Improved communication and collaboration by visualizing 3DExperience data in Creo View across the enterprise

- Reduce costs and risks associated to accessing the wrong version of data by integrating the publishing process into all related business processes

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

The remainder of this document will focus specifically on the use of the 3DExperience to Creo View product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

3DExperience to Creo View Product Modules

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

MC6CVW-U1 – 3DExperience to Creo View Standard Product

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

MC6CVW-A1 – 3DExperience to Creo View PMI Add-On Module

- The PMI module allows PMI, Captures and View states to be read from CATIA V6 and written into Creo View.
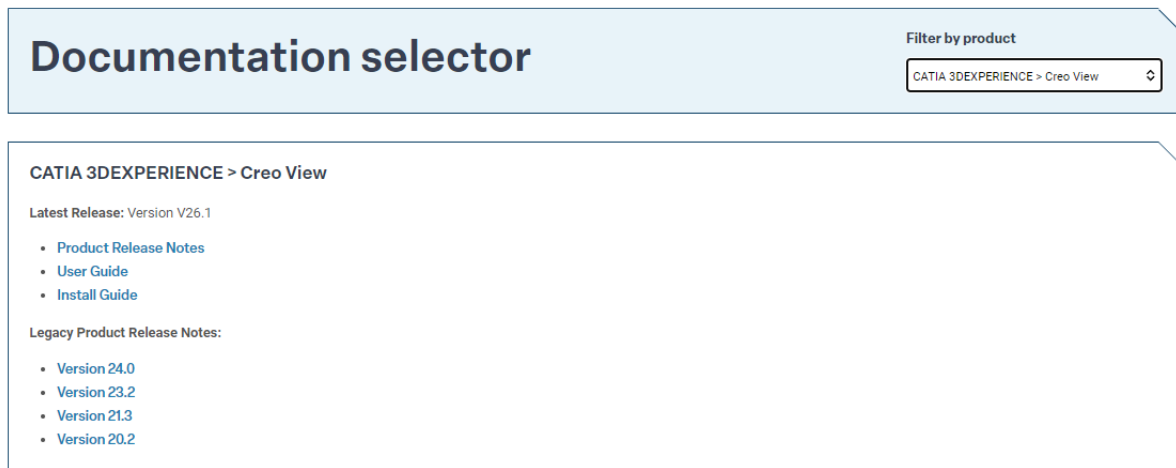
Getting Started

Documentation & Installation Media

The latest copy of the User Guide documentation can be found on our web site at:

http://www.theorem.com/Documentation

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

The latest copy of Theorem software can be found via the link above and by searching for the specific product. See image below:

Each product has a specific link to the Product Release Notes, which contains a link to the download location of the installation CD.

Installation

The installation is run from the .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation

License Configuration

To run any product a valid license file is 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 www.theorem.com/documentation

Using the Product

To use the product, follow the documented steps found in this document or follow the online video tutorials which can be found from www.theorem.com/documentation

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 3DExperience

    o The Interactive Interface provides a direct method exporting from 3DExperience.

- In Batch via CATUtil - DataExchangePLMBatch

    o The 3DExperience DataExchangePLMBatch Interface provides a direct method of invoking the translator. It can be used to translate single or multiple files at once.

- On the Command Line

    o A command line method of Invoking the translator is possible.

Translating Interactively from within 3DExperience

Launching 3DExperience with Theorem Plug-ins

The 3DExperience to Creo View translator allows an opened 3DExperience part or assembly to be exported directly to Creo View.

In order to translate from within 3DExperience, the application must be started using a Theorem environment, so that the appropriate Theorem partner plug-ins are available. (See 3DExperience Environment files)

3DExperience 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:

start_3DEXPERIENCE_Theorem_Multi-CAD_CREOVIEW_CATIAV6R<version>.cmd

(where <version> should be substituted for the version of 3DExperience that you have installed – e.g. 2021x, 2022x, 2023x):

Interactive Export to Creo View

Once the 3DExperience application has been launched, open the product or representation that is going to be exported to Creo View.
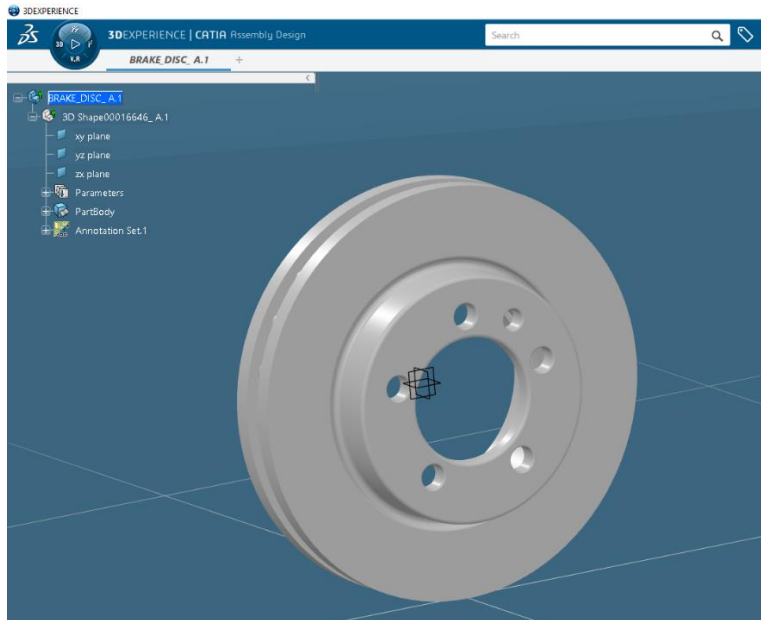
Select the arrow in the top right corner, then from the list displayed select Export.
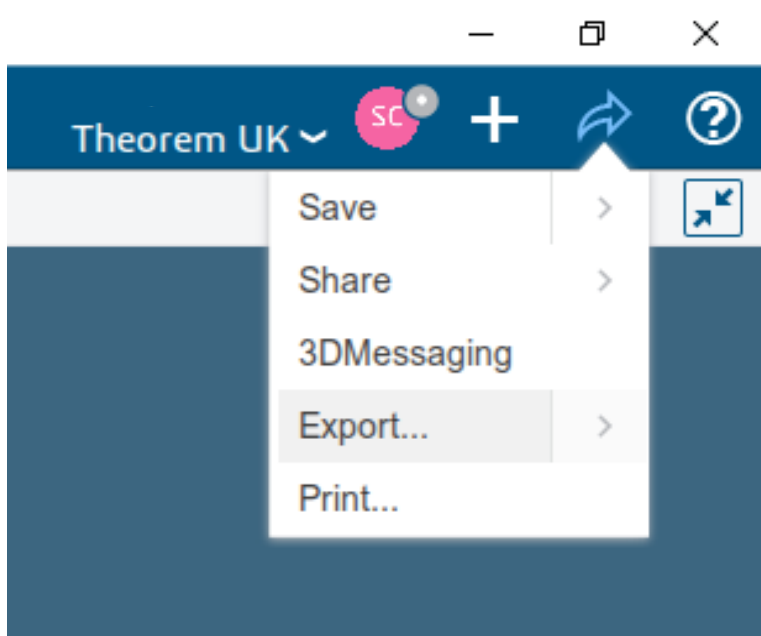
In the Export dialog box is displayed. Ensure the "Format" is set to CREOVIEW_THEOREM (*.pvs). Select the required location for the files and ensure the Filename displayed is correct. Click OK to initiate the translation.
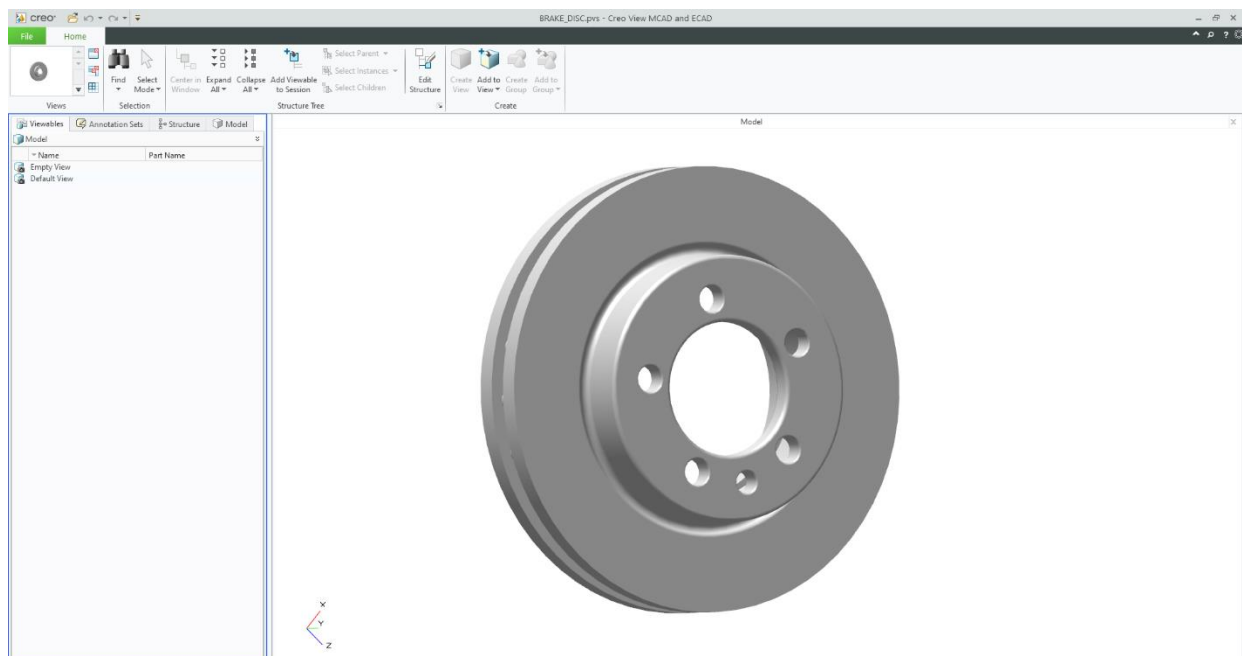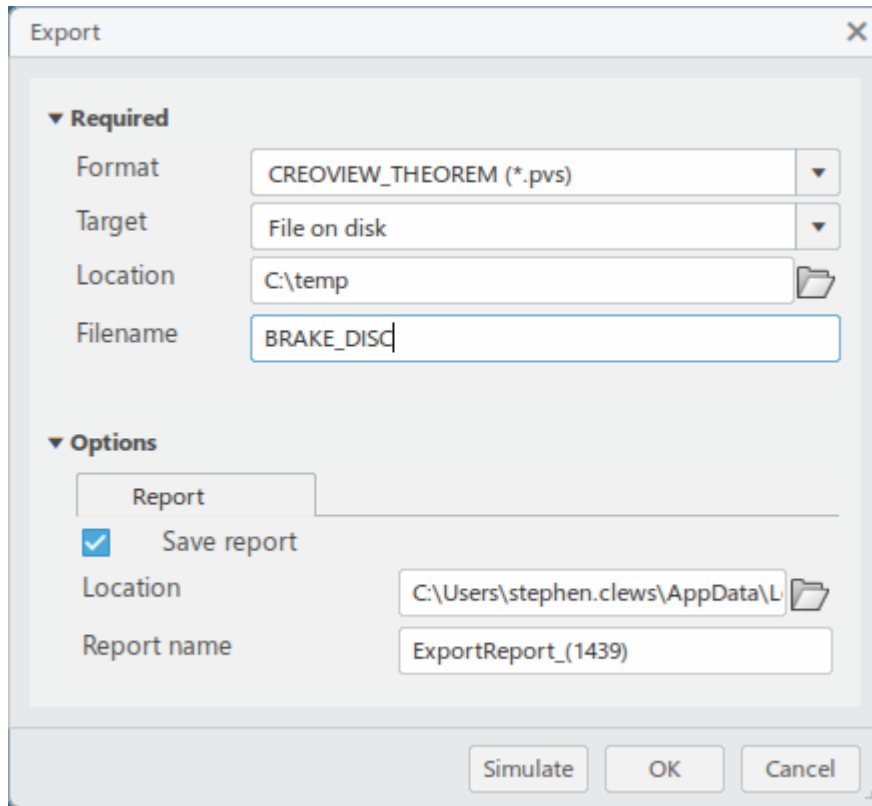
Theorem Interactive Conversion Settings

The 3DExperience interface does not currently require the user to apply any specific settings for the translation. There are some general settings that should be checked if required (e.g. for PMI conversion.) These are accessed through Preferences>Legacy Preferences>General>Compatibility>External Native 3D Formats. There are also some options that can be applied within the Theorem setting. These are accessed through Preferences>Legacy Preferences> Infrastructure>Theorem:
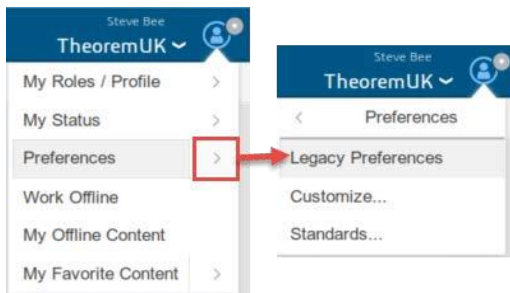

This page is a standard Dassault Page that sets the preferred mode of conversion (in this case Exact), and also the general options "Convert coordinates system" and "3D annotations". Both of which are toggled on.
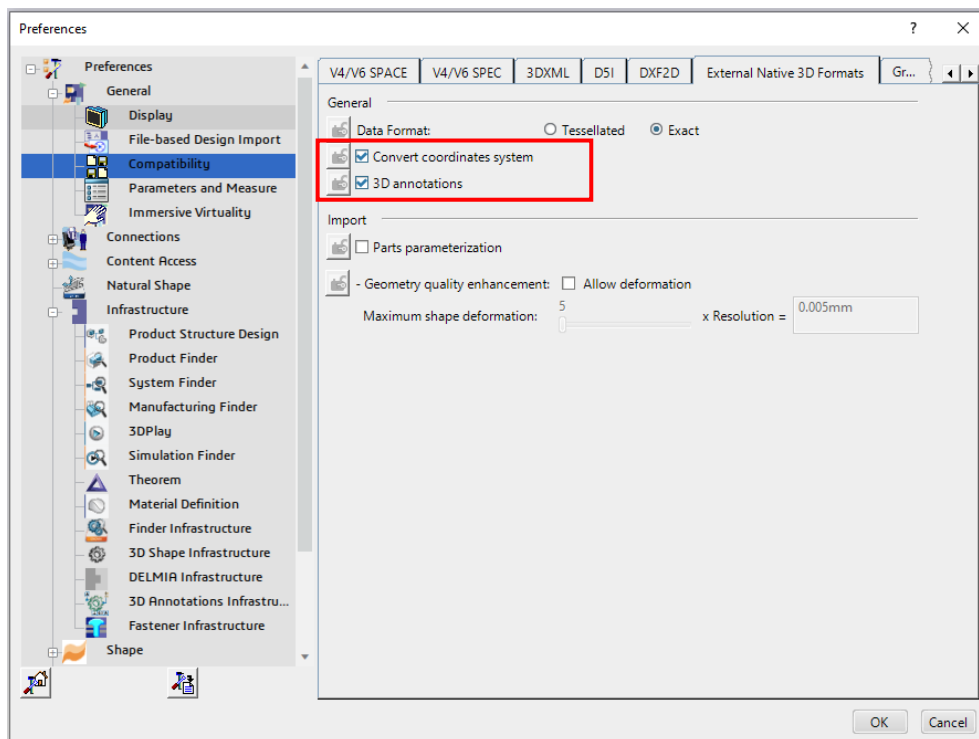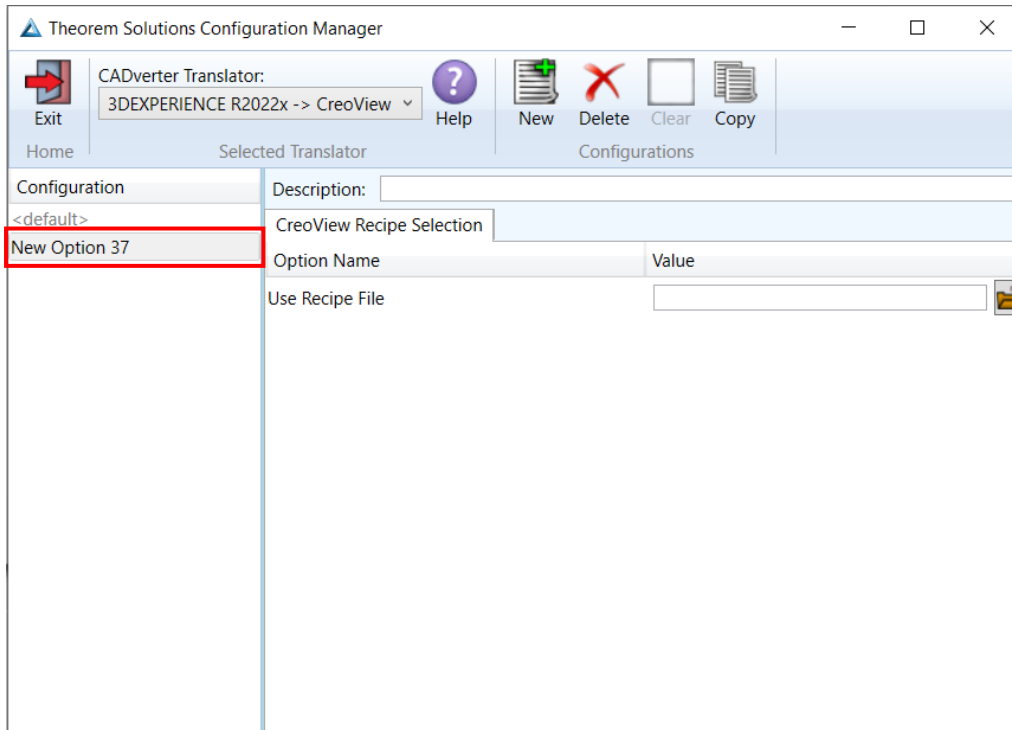
A dedicated tab under *'Infrastructure>Theorem'* allow the user access to Theorem Configurations for Publish 3D. From this Panel, the user can select a predefined configuration or create a new configuration. To create a new configuration, select the *'Launch Configuration Manager'* command.
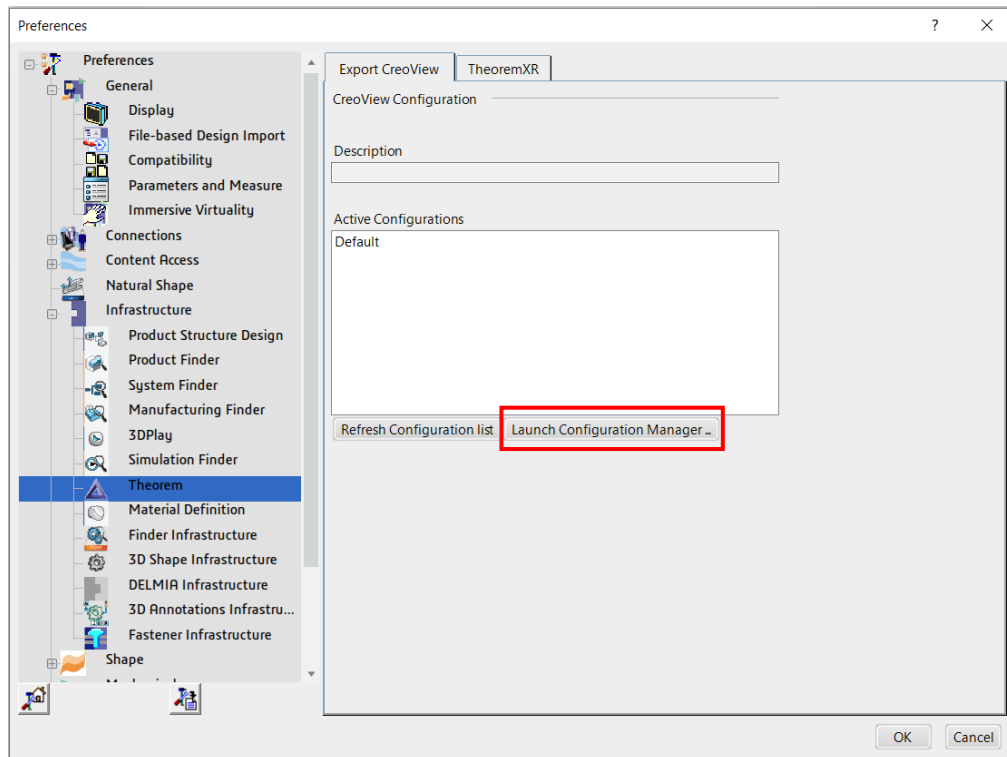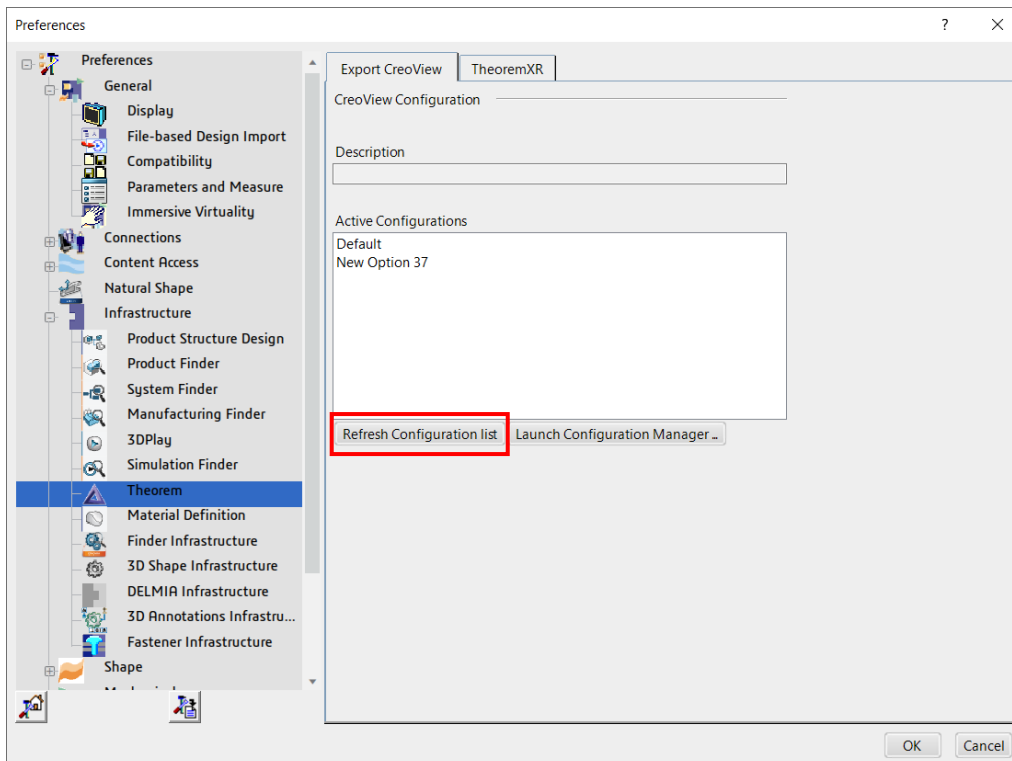

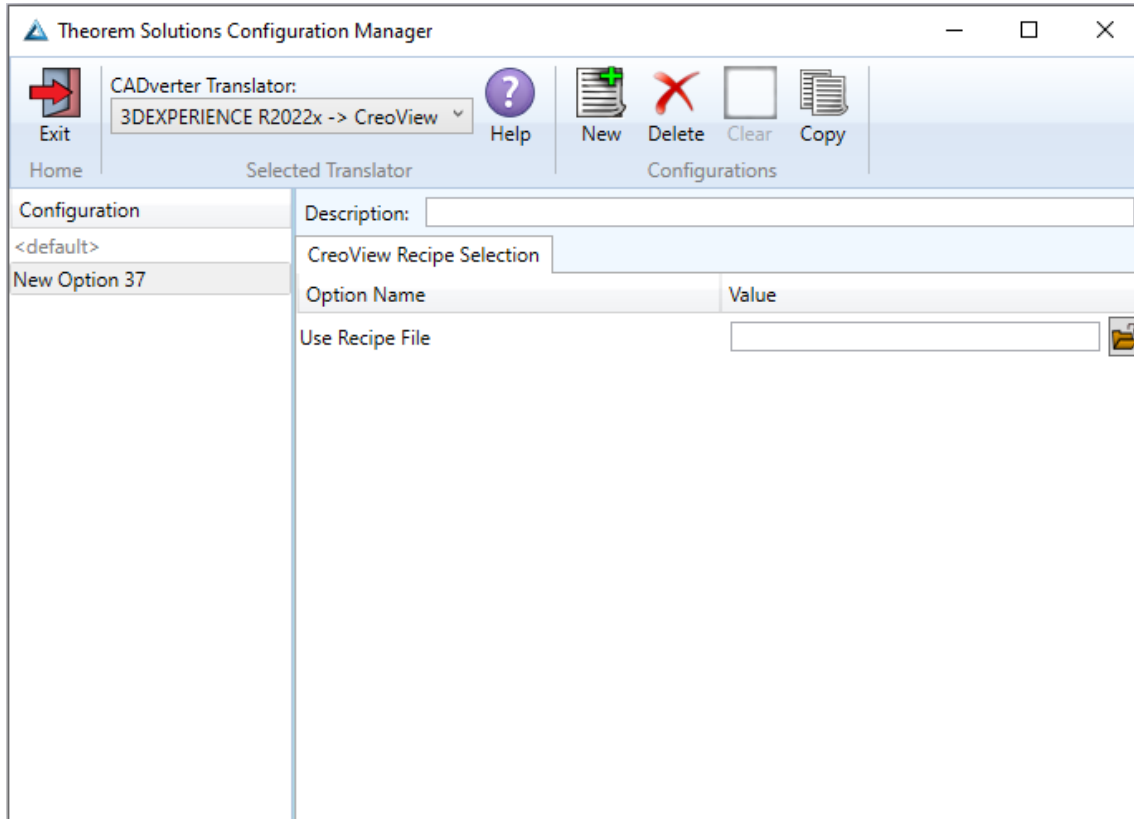In the configuration manager window, select New. Rename as required.
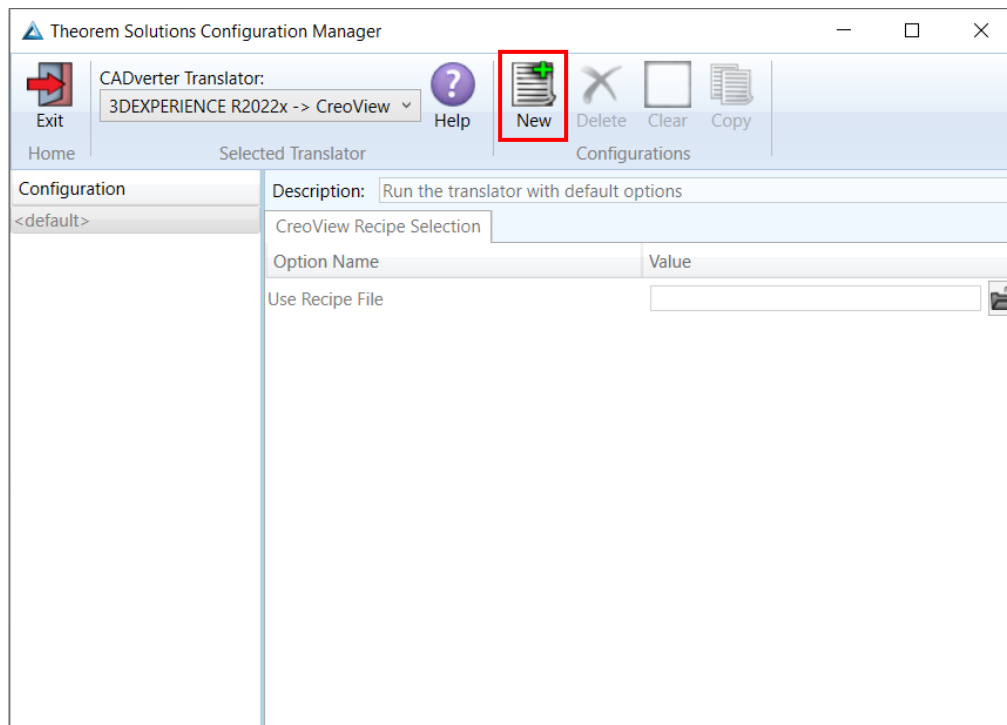
Any new configurations created will be displayed in the Active Configurations list once it has been refreshed.
To do this select *'Refresh Configuration list'*

Configuration Manager

Within the Configuration Manager, only a CreoView Recipe Selection tab is available where a saved recipe file (.rcp) can be added.

To add a recipe file, click on the folder icon and navigate to the directory where the file is saved. Select the file then click Open to add it to the configuration selected in the configuration manager.

Recipe Files

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. While most CAD parts will convert into an efficient form for large-scale visualisation, some parts require modifications to the standard visualisation recipe to be viewed effectively.

The Adapter is provided with a master or default recipe file. This file is preconfigured to allow the visualisation of most objects. The master recipe file should not be edited. Instead, new recipe files can be created using the Save As function in the recipe editor (rcpedit.exe) provided with the translator. Launching the rcpedit.exe will display a number of tabs with the available translation options displayed under these tabs.

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

Instead, we will focus on the Theorem specific settings that affect the output. These are accessible via the Advanced Tab and are grouped into 2 sub pages:
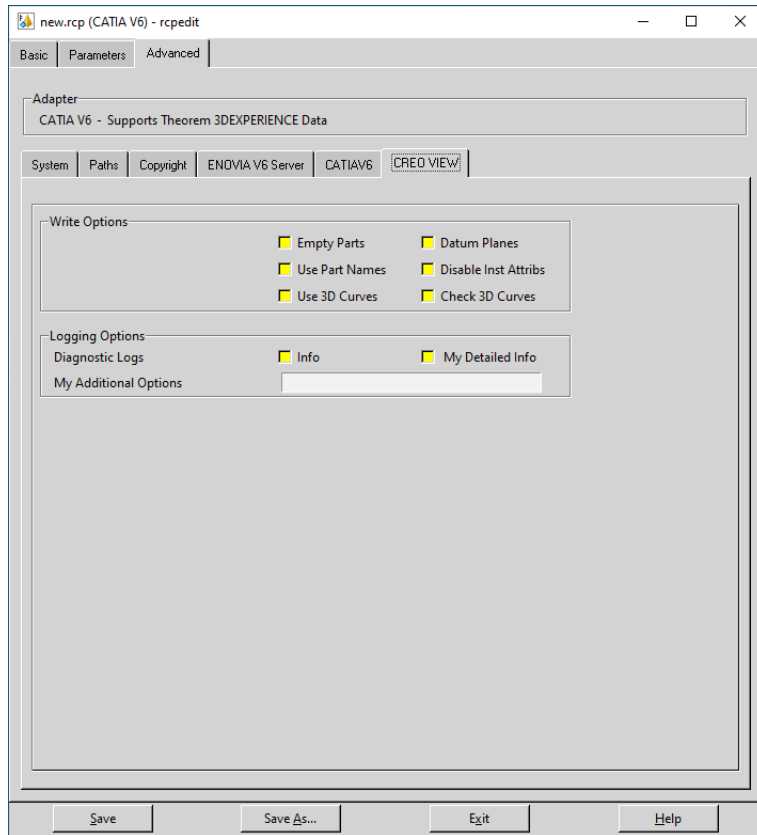
- CATIAV6

- CREO VIEW

CATIAV6
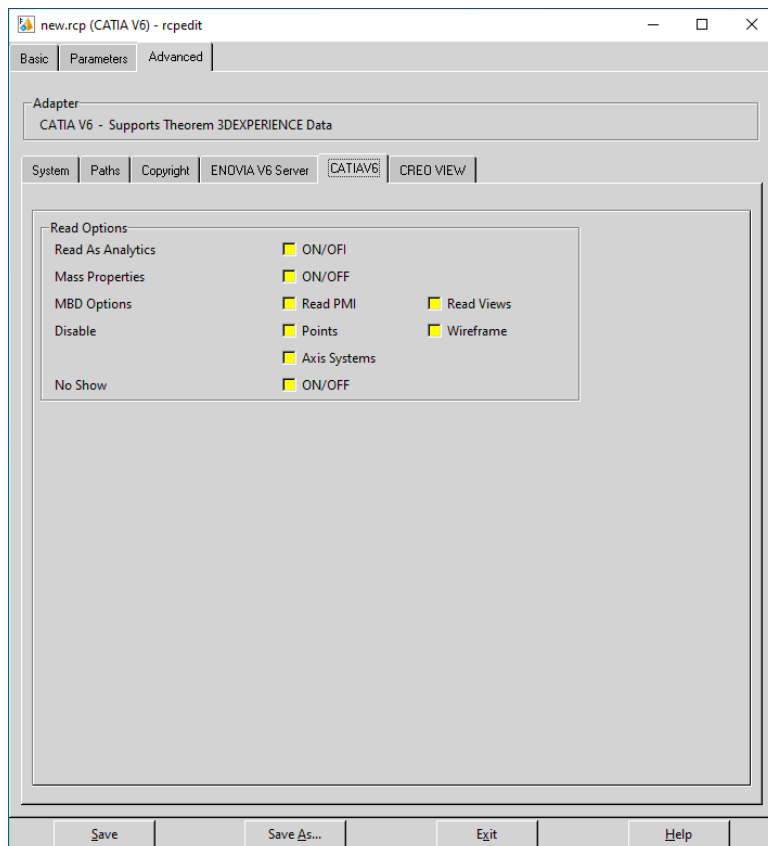
Each of the options displayed in the image above are described below:

Option	Description
Read Options	
Read as Analytics	If checked this causes the data to be read as Analytic data where possible.
Mass Properties	Reads the Mass properties from the CAD data where available.
MBD Options	When checked reads PMI and Views/Capture data if available. Note this is only available with an add on license.
Disable	Stops Points, Wire Frame and/or Axis systems from being read.
No Show	Read's data that has been hidden.

CREO VIEW

Each of the options displayed in the image above are described below:

Option	Description
Write Options	
Empty Parts	Creates an empty (contains a single point at X0, Y0, Z0) dummy .ol file for empty nodes, to force the geometry node creation, where no geometry exists.
Datum Planes	Allows large planes, that are by default omitted, to be written as Datum Planes. Selecting this option will replace PLANES where the bounding box diagonal exceeds 0.1Km with a Datum PLANE 35mm square. The bounding box size can be adjusted by using the additional argument. ***set_omit_large_planes \<value in mm>*** If the size of the created datum PLANE can be adjusted by using the additional argument. ***set_create_datum_planes \<value in mm>*** see additional options for details.
Use Part Names	Use the 'Part Number' names for assembly nodes. The default Is to use 'Instance Names'.
Disable Inst Attribs	Disables the output of any Instance attributes.
Use 3D Curves	This option allows the Creo View API to generate Its own 2D curves. This option Is most likely only ever used as a work around when poor data Is encountered.
Check 3D Curves	This option allows the 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 ***validate_3D_curve_tol \<value>*** in the additional option field.

Logging Options	
Info	Disables the output of any Instance attributes. Generate a more verbose log.
My Detailed Info	Disables the output of any Instance attributes. Generate a more verbose log. Used as a diagnostics level of detail in the log file *(for debug purposes only.)*
My Additional Options	
set_omit_large_planes *<value>*	Omit large PLANEs greater than the tolerance value (default is 100 M) e.g. ***set_omit_large_planes 2000*** - sets a value of 2km
set_create_datum_planes *<value>*	Omit large PLANEs greater than the tolerance value (default is 100 M) Omitted large PLANEs can be replaced by datum PLANEs (default 35mm). The size of that datum PLANE is controlled with this option. e.g. ***set_create_datum_planes 150*** - sets a value of 150mm

Translating in Batch using CATUtil – DataExchangePLMBatch

Launching DataExchangePLMBatch

The Dassault Systemes CATUTIL Batch Monitor tool can be launched with the correct Theorem enabled environment using the scripts provided in the Translator installation at:

<installation_directory>\bin\start_CATUTIL_Theorem_Multi-CAD_CREOVIEW_CATIAV6R<revision>.cmd

(where *<revision>* should be substituted for the version of 3DExperience that you have installed – e.g. 2021x, 2022x, 2023x)

It can also be started from the Tools>Utility workbench within an Interactive 3DExperience session.
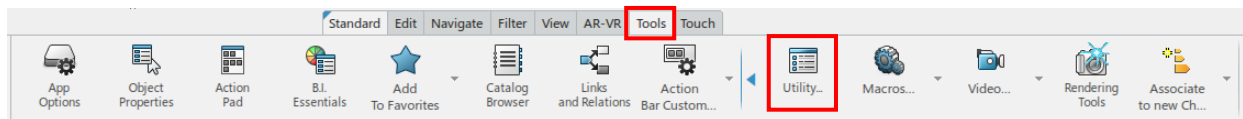
Batch Export to Creo View

Here is an example showing what is required in preparation for a 3DExperience database selection and batch export.

Double click on **DataExchangePLMBatch**.
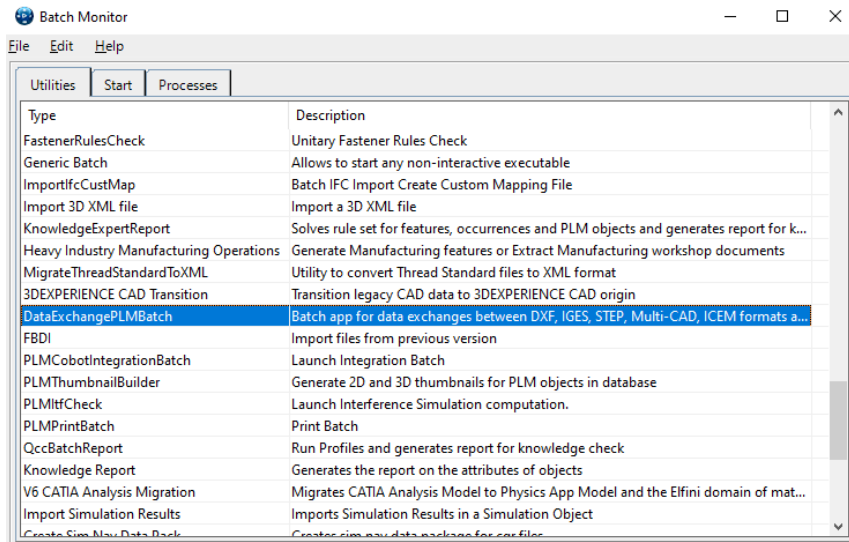
Ensure that **'Export'** is selected as the data exchange operation, **'CREOVIEW_THEOREM'** is selected as the usage and **'pvs'** is selected as the extension.
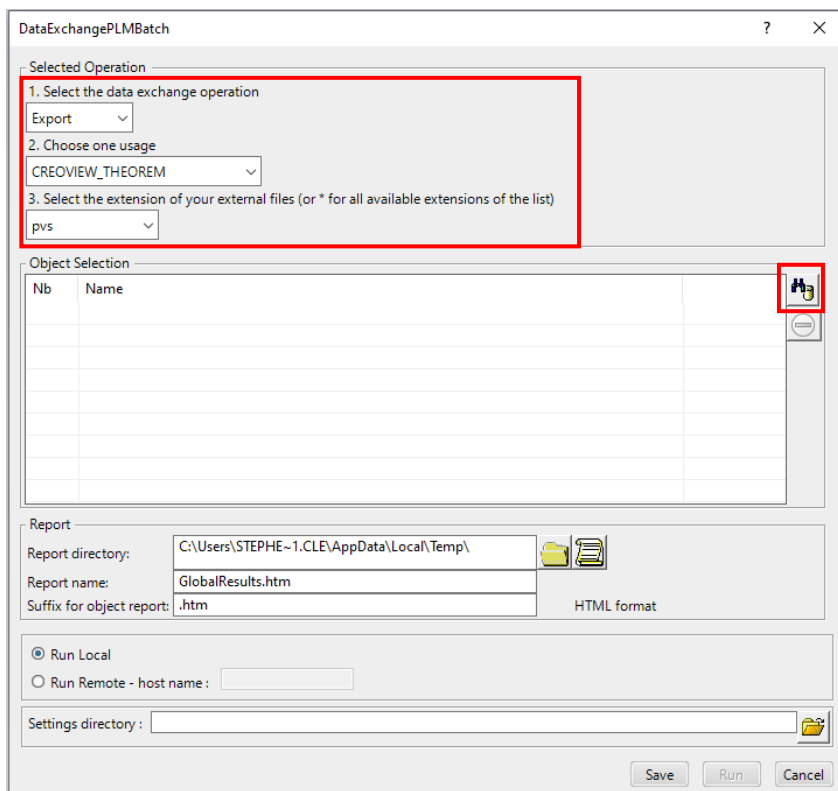
To add the required products for the batch export, select the 3DExperience Platform Object Search icon.

In the subsequent window displayed, add the required products using the correct attribute, operator and value ensuring that the correct object type has been selected. (E.g. Physical Product.) For multiple products ensure that the **OR** condition has been selected. Select Add results and click Apply to add the list of products to the Object Selection field.
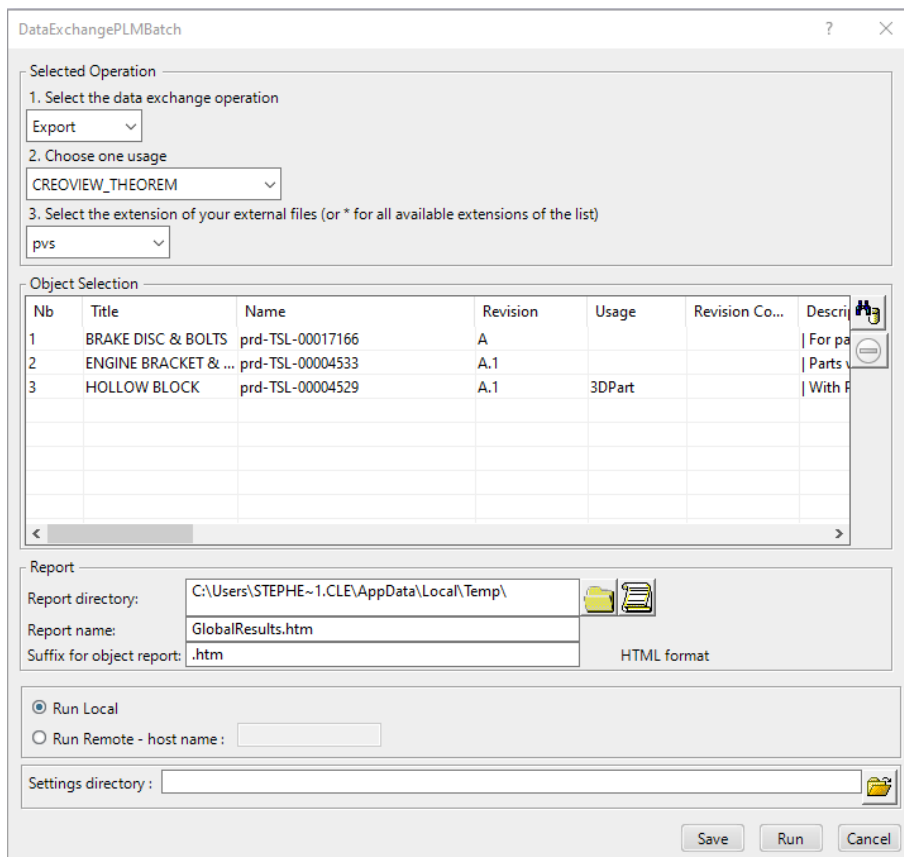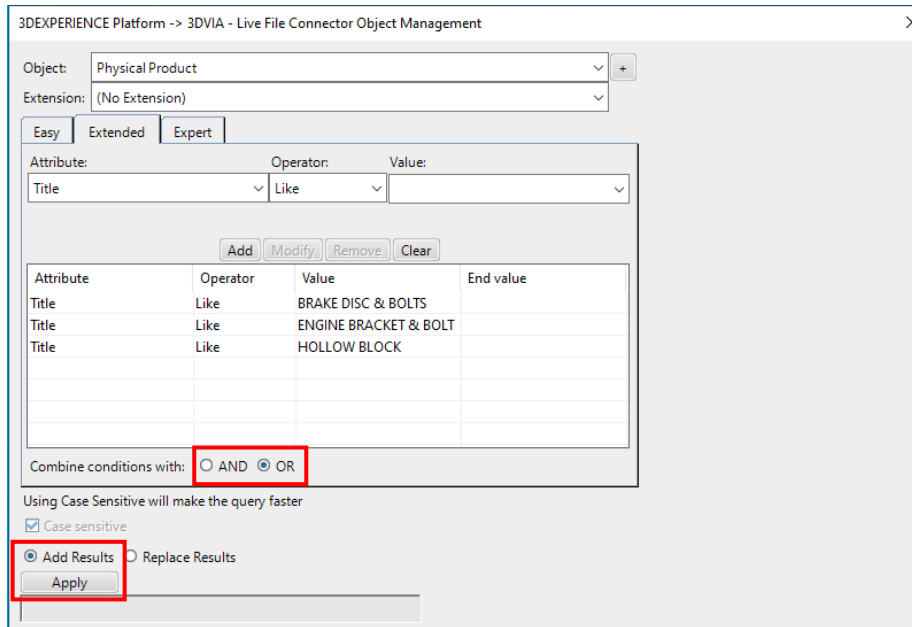
To change the location where both the report and the pvs files will be saved, select the icon next to the Report directory field, then select the required directory.
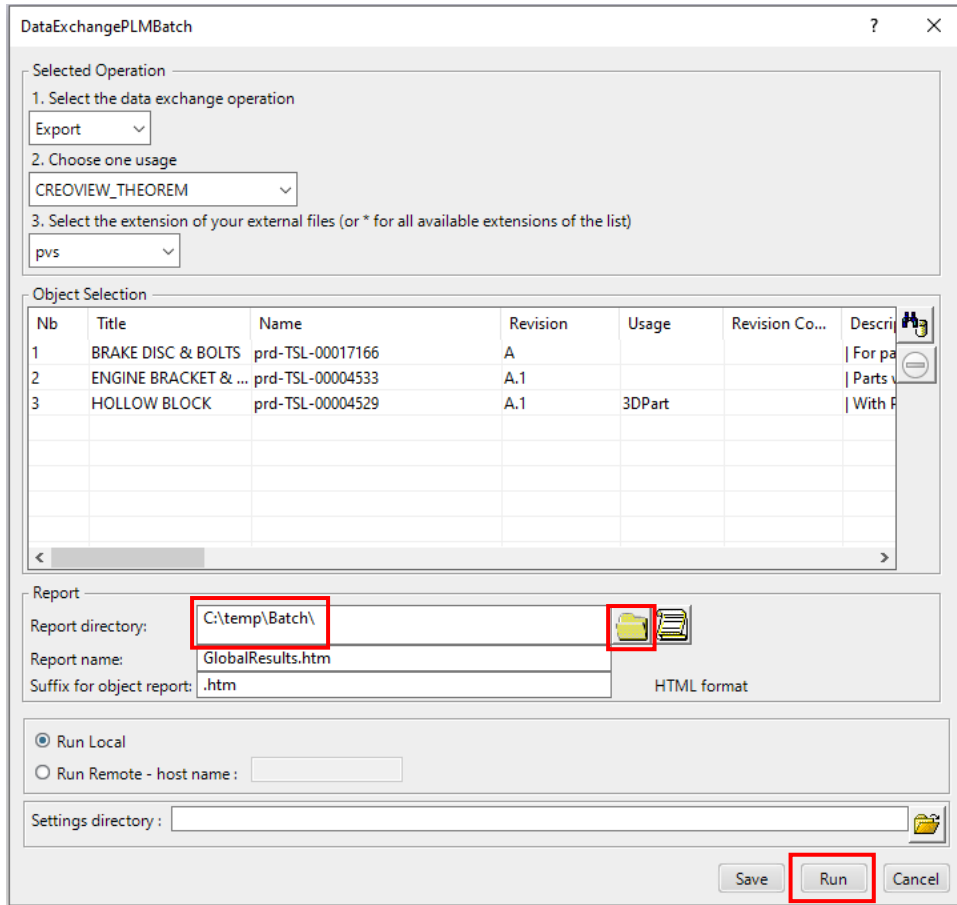
Click Run to initiate the Batch Export.

Adding Options for a Batch Translation

Options displayed in the recipe editor (rcpedit.exe) can be added to the **xcad_creoview_opts.txt** file available in the following location when translating via batch.

<installation_directory>\data\CreoView

To include a new option in a batch translation, simply enter the name of the argument to a new line in the **xcad_3dpdf_opts.txt** file. In the example below the read_pmi argument has been added.

```
1  * CreoView Exporter default options
2  * The first '*' character on a line denotes a comment line
3  *
4  * To specify a non-default CreoView recipe file,
5  * created using the CreoView Recipe Editor (rcpedit.exe), use
6  * -r <recipe_file>
7  * (examples are located in Theorem install directory under folder \recipe)
8  * e.g. The following line to point to an existing recipe file
9  * that will support the conversion of PMI/FTA data
10 -r "%TS_INST%\recipe\gco_pv_pmi.rcp"
11 * End of options
12 *
13 * To Read validation properties
14 *mprops
```

```
1  * CreoView Exporter default options
2  * The first '*' character on a line denotes a comment line
3  *
4  * To specify a non-default CreoView recipe file,
5  * created using the CreoView Recipe Editor (rcpedit.exe), use
6  * -r <recipe_file>
7  * (examples are located in Theorem install directory under folder \recipe)
8  * e.g. The following line to point to an existing recipe file
9  * that will support the conversion of PMI/FTA data
10 -r "%TS_INST%\recipe\gco_pv_pmi.rcp"
11 * End of options
12 *
13 * To Read validation properties
14 *mprops
15 read_pmi
```

Translating on the Command Line

It is possible to run a 3DExperience to Creo View translation on the command line, however as all CAD data is saved in the 3DExperience database an XML file will need to be used as the input file instead of a CAD file. This XML file contains all the necessary information to locate, open and translate the CAD data. This is explained in more detail below. In order to run a translation on the command line the following inputs are required: -

<TS_INST>\bin\catia6r<revision>_pv.cmd <input_file> -p <output_path> -o <output_file> -r <recipe_file>

Where: - <TS_INST> is the Theorem Solutions software installation directory.
Where: - <revision> is the version of 3DExperience that you have installed – e.g. 2021x, 2022x, 2023x.

<input_file>

Is an xml file defining access to a specific object in the 3DExperience database.

This file provides user login details (V6R2013x) or a Login Ticket (2015x onwards) (see appendix A), to a specified Enovia repository plus a set of 3 attribute names and values which will uniquely identify the input object required for the conversion. These are highlighted in <mark>green</mark> in the example below and will need changing to the object and revision being used for the translation. The lines highlighted in <mark>yellow</mark> should be modified with 'your' login details which are required to access Enovia. Examples of the input file XML can be found in the **<TS_INST>\samples\3dexperience** folder. These can be modified and saved on your local drive.

Here is an example of the xml input file for 3DExperience

```xml
<?xml version="1.0" encoding="utf-8"?>

<mc6_read>
 <!-- parameters must be in this ORDER -->
 <!-- parameters only the value passed to V6  -->
 <parameters>
  <attribute name="repository" value="PLM1"/>
  <attribute name="ServerName" value="3dspace.theorem.com"/>
  <attribute name="ServerPort" value="447"/>
  <attribute name="ServerRootURI" value="3dspace"/>
  <attribute name="LoginTicket" value="REEyNzM3M0Q1REM2NDgxQzlFNzk1QzIwNjZGRDYzODN8Um9iaW58Um9iaW558fHwwfA=="/>
  <attribute name="PLMType" value="VPMReference"/>
  <attribute name="ReportDirectory" value="C:\TEMP\V6Export"/>
  <attribute name="LicenseData" value="LIC"/>
  <attribute name="BatchXMLFileName" value="Default"/>
 </parameters>
 <!-- arguments both key and value are passed to V6 -->
 <arguments>
  <attribute key="PLM_ExternalID" value="prd-Interfix"/>
  <attribute key="V_version" value="A"/>
  <attribute key="minorrevision" value="1"/>
 </arguments>
</mc6_read>
```

Note – In some cases, the "minorrevision" value is not required in the XML file, only the V_version value is required. This value could be A.1 for example.

-p <output_path>

> Is the path where the output file will be saved.

-o <output_file>

> is the required Creo View output file name.

-r <recipe_file>

> Is the name of the recipe file used for controlling the translator options.


```
C:\Users\stephen.clews>"C:\Program Files\Theorem\26.1_3DExperience_Creo View\bin\catia6r2022x_pv.cmd"
C:\temp\3dex_2022x_input_Engine_Bracket.xml -p C:\temp -o ENGINE_BRACKET -r C:\temp\catia6_pv.rcp
```

Log File Generation

In the process of exporting the selected 3DExperience part or assembly, the following log files are generated by the Creo View Export plug-in.

The process log and error messages are recorded in a '.err' file located in the 3DExperience CATReport directory. The file is named after the active CATIA component.

e.g. *%CATReport%\model1.err*

Additional log files are created in the TSC_TEMP_DIR directory. This directory is defined in the *%TS_INST%\ts_env.bat* file.

%TSC_TEMP_DIR%\Read_to_viewer_<input_part_name>.log

Where: <input_part_name> is the name of the input part (or the active part name in interactive usage)
This contains information describing the 3DExperience 'data read' processing into Theorem Intermediate data format. Normally a list of entities.

```
List of gco entities :-
----------------------
Type          Total    Standalone   Subordinate
----------    -------  ----------   -----------
Lines         237                   237
Curves        468                   468
Surfaces      189                   189
Planes        81                    81
```

%TSC_TEMP_DIR%\viewer_<part-name>_screen_output.log

Where *<part-name>* is the selected output file name.

This contains the screen output of the process of writing the data to Creo View. The status of the translation can be found here

%TSC_TEMP_DIR%\viewer_*<part-name>*.log

Where *<part-name>* is the selected output file name.

This contains detailed process information of the write of the data to Creo View and contains additional information such as modifiers and options used.

3DExperience Environment Files

As part of the Theorem installation process, a set of 3DExperience environment files are created which are subsequently used in the launch of 3DExperience and CATUtil sessions to support the Theorem partner plug-ins for Creo View export.

A 'CATEnv' file is created for each installed version of 3DExperience.

These environment files are located in the 3DExperience revision specific folder e.g.

<installation_directory>\B423\win_b64\CATEnv\Theorem_Multi-CAD_CREOVIEW_CATIAV6R2021x.txt

<installation_directory>\B424\win_b64\CATEnv\Theorem_Multi-CAD_CREOVIEW_CATIAV6R2022x.txt

<installation_directory>\B425\win_b64\CATEnv\Theorem_Multi-CAD_CREOVIEW_CATIAV6R2023x.txt

These files consist of the current 3DExperience settings with the required Theorem settings appended at the bottom.

```
!---------------------------------------
! Theorem Additional Multi-CAD Settings
!---------------------------------------
TS_INST=C:\Program Files\Theorem\26.1_3DExperience_Creo View\
THEOREM_LICENSE_FILE=7601@ts-tam-lic-svr
TSC_TEMP_DIR=%TEMP%
XCAD_FTA_NO_LINKS=1
OPTIONS_MULTICAD_PARTNER=1
PVIEW_HOME=C:\Program Files\Theorem\26.1_3DExperience_Creo View\
```

Appendix A – Creating a Login Ticket

Use a browser link (similar to the one below) to your 3DExperience Server

https://3dspace.2017x.theorem.com:447/3dspace/common/emxNavigator.jsp
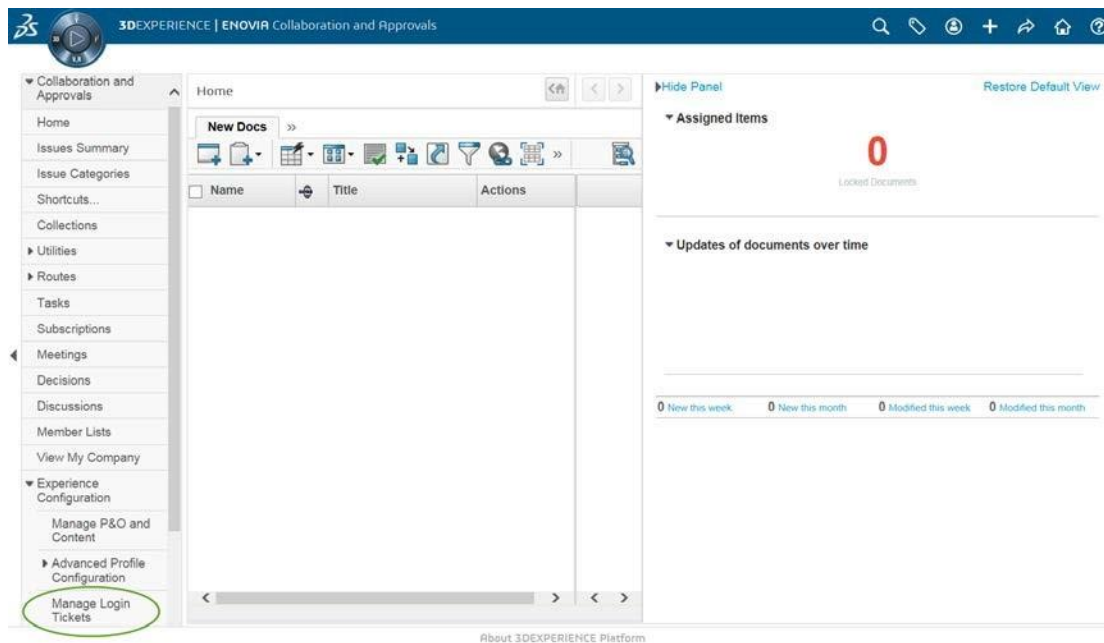
A page will appear in the browser
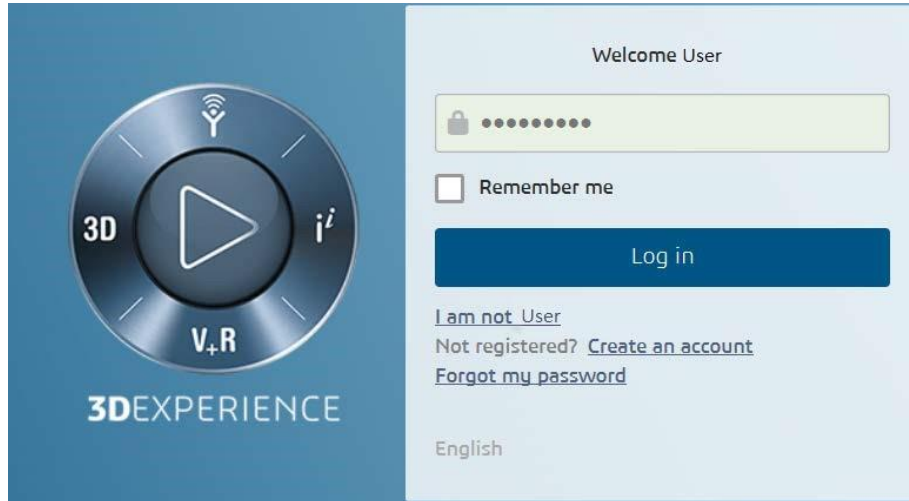
Login as normal, e.g.

User

Password

Select Collaboration and Approvals > Experience Configuration > Manage Login Tickets

Select the values required for your user:

User and Security Context should already be set, make sure that Ticket Type: Infinite is selected. Then click on 'Create'

'Create' will produce the ticket. See example below: -

REEyNzM3STE1MER2NDgxQzlFNzk1QzIwNjZGNATzDAN8Um9iaW58Um9iaW58fHwwfA==

This can then be used as the 'LoginTicket' value explained previously.

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