USER GUIDE

3DEXPERIENCE CATIA V6 – Creo View

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Contents

Overview of Visualise 3D

About Theorem

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

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

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

Theorem's Product Suite

Theorem have 3 main Product brands. These are:

TRANSLATE

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

See our website for more detail.

PUBLISH

The creation of documents enriched with 3D content

See our website for more detail.

VISUALIZE

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

See our website for more detail.

The 3DEXPERIENCE CATIA V6 Uni-directional Creo View Translator

The 3DEXPERIENCE CATIA V6 to Creo View translator may be installed on a number of machines each accessing a central network-floating license.

The 3DEXPERIENCE CATIA V6 to Creo View Translator is a Uni-directional direct database converter between the Dassault Systemes 3DEXPERIENCE CATIA V6 Modelling Application and the Creo View file format, used by the PTC Visualization 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 visualization and is used by companies using Creo View based solutions to translate their 3DEXPERIENCE CATIA V6 data into the Creo View format.

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

Primary Product Features

- Converts CATIA 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 V6 properties. Data can be filtered by layer and entity type during processing. Geometry can be filtered and selectively processed.

- Integrated with the 3DEXPERIENCE CATIA V6 installation.

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

- Uses the 3DEXPERIENCE CATIA V6 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 CATIA V6 to Creo View reduces processing time, simplifies integration and retains accuracy of the model

- Improved communication and collaboration by visualizing CATIA V6 data in Creo View across the enterprise

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

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

This document will focus specifically on guidance for the use of Visualize 3D for the 3DEXPERIENCE CATIA V6 – Creo View product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

CATIA V6 to Creo View Adapter Product Modules

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

MC6CVW-U1   CATIA V6 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   CATIA V6 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
The latest copy of this documentation can be found on our web site at:

http://www.theorem.com/Documentation

Release Notes
The latest copy of the release notes and Theorem software can be found via our web site at:

http://www.theorem.com/Product-Release-Notes

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

Alternatively, you can request a copy of the software to be shipped on a physical CD.

Installation & License Configuration
The installation and license configuration of this product are run from separate .msi packages. More information regarding these steps can be found in the following document

[CATIA V6 to Creo View Installation and licensing 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 CATIA V6

  - The Interactive Interface provides a direct method exporting from 3DEXPERIENCE. See the 'Running the Translator in Interactive Mode' Section of this document

- In Batch using Job Files

  - The 3DEXPERIENCE – Creo View product provides a command line interface for invoking the translator. This is similar to the method used by Windchill. See the 'Running the Translator from the Command Line' Section of this document

- Publishing data from within Windchill

  - The Windchill interface allows the user to convert CATIA V6 data directly from within the Windchill environment. **For further information regarding instruction on how to implement this option, please see your PTC representative.**

Running the Translator in Interactive Mode

The 3DEXPERIENCE CATIA V6 to Creo View translator allows an open 3DEXPERIENCE CATIA V6 Part or Assembly to be exported directly to Creo View.

In order to translate from within 3DEXPERIENCE CATIA V6, the 3DEXPERIENCE CATIA V6 application must be started using a Theorem environment, so that the appropriate Theorem partner plug-ins are available. (See 3DEXPERIENCE CATIA V6 Environment files)

3DEXPERIENCE CATIA V6 can also be started from a desktop shortcut created at installation time. E.g.

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 CATIA V6 that you have installed – e.g. 2017x, 2018x, 2019X etc.):

Theorem Interactive Conversion Settings

The 3DEXPERIENCE interface does not currently require the user to apply any Creo View specific settings for the translation. There are some general settings that should be checked if required (e.g. for PMI conversion.)For R2015x and later revisions these are accessed through Preferences>Legacy Preferences>General> Compatibility > External Native 3D Formats :
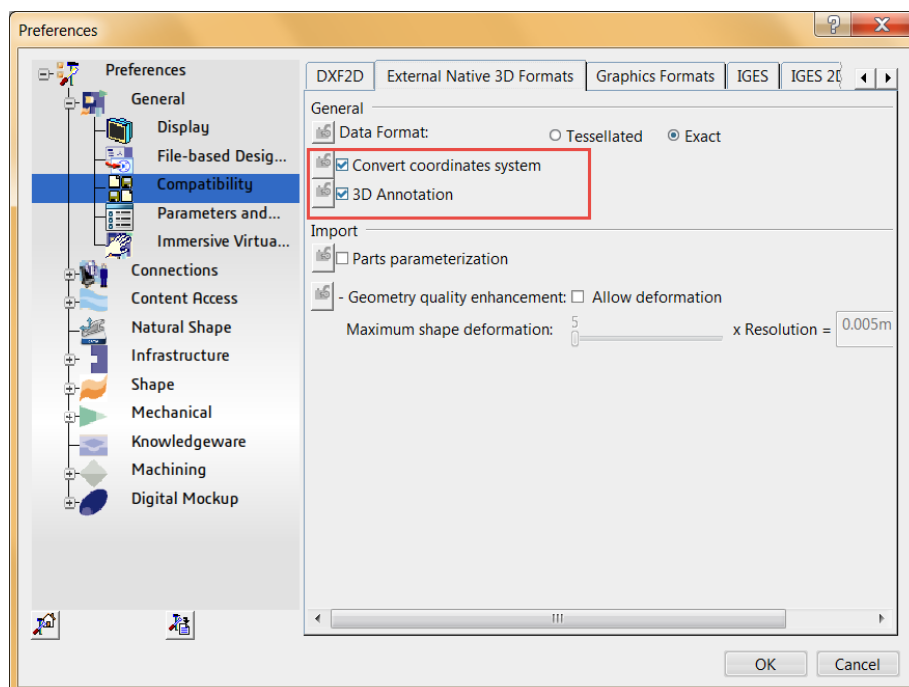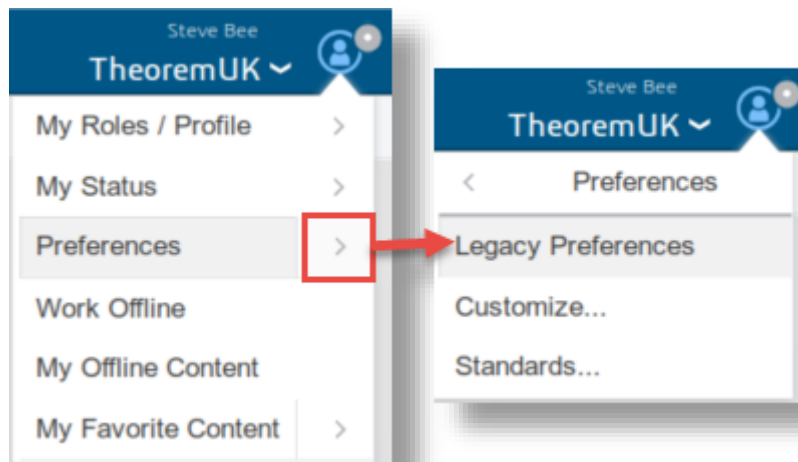
This page is a standard Dassault Page that sets the preferred mode of conversion (in this case exact) this will allow tessellation to be controlled by the Creo View DAPI.

The general options "Convert coordinate systems" and "3D Annotation" allow conversion of these items.

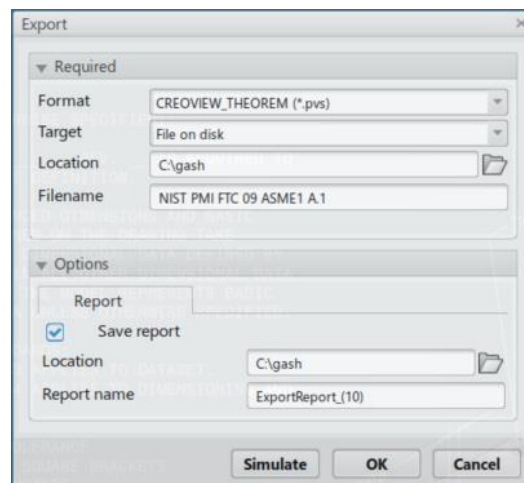The Import section is not applicable to the Creo View format

Theorem Export

Once 3DEXPERIENCE CATIA V6 has been started it is possible for an opened product to be exported to Creo View.

If 3DEXPERIENCE CATIA V6 R2017x / R2018x is being used, the user selects the menu item as shown here:-
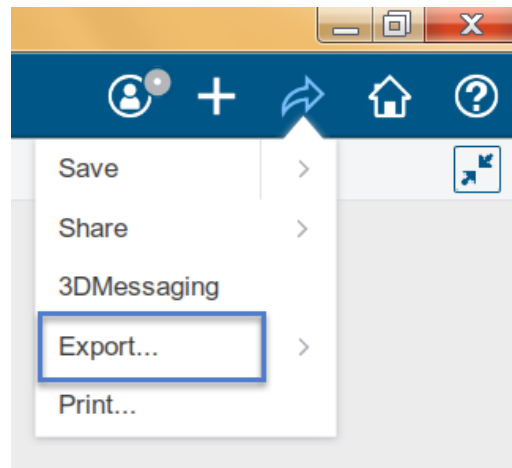
The following dialog appears

The "Format" selection should be set to CREOVIEW_THEOREM (*.pvs). The output "Filename" and "Location" can also be specified.

At later versions the report directory can also be specified

The **OK** button can now be pressed to initiate the conversion to the selected output file name and location.

Running the Batch Utility Translator from the Command Line

Batch Export to Creo View - Some background

The batch utilities are started via the script provided in the Translator installation located in:
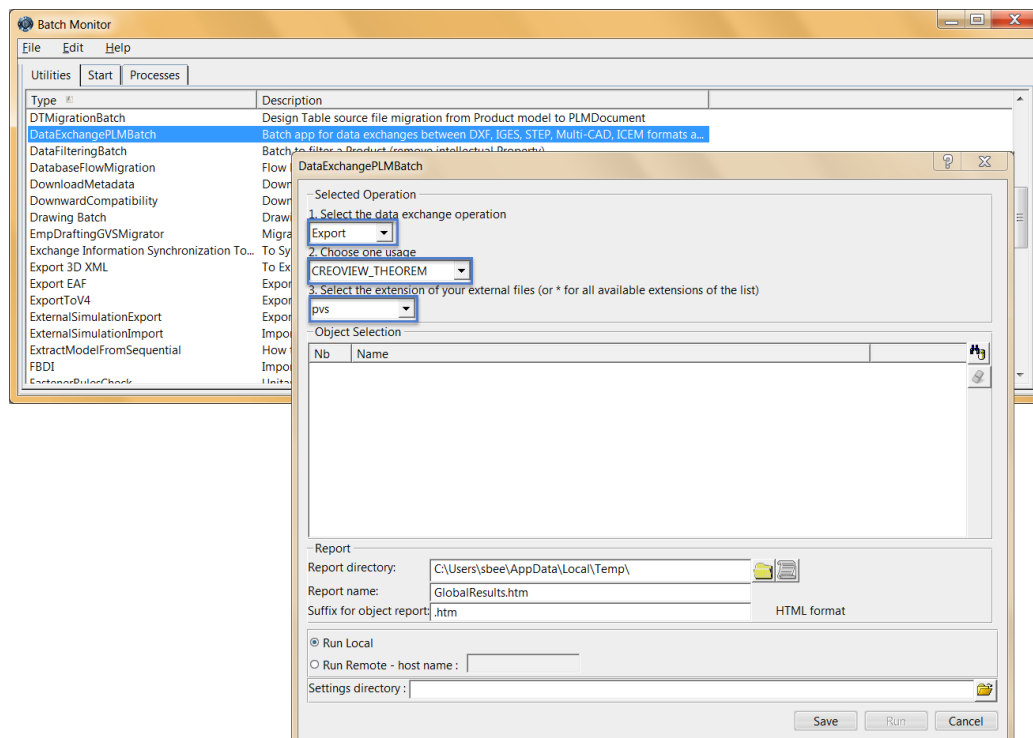
<installation_directory>\bin

The script name is:

**start_CATUTIL_Theorem_Multi-CAD_CREOVIEW_CATIAV6R<version>.cmd**

(where <version> should be substituted for the version of 3DEXPERIENCE CATIA V6 that you have installed – e.g. 2013x, 2017x, 2018x, etc.):

Below is an example of a DataExchangePLMBatch session showing the appropriate settings in preparation for a database selection and an export "save" or "run" :-


Note selection of "Export" operation, usage "CREOVIEW_THEOREM" and extension "pvs". These should all be set correctly.

The user can then select a part or assembly from CATIAV6 using the "Object Selection" section. When the section is complete hitting "Apply" will retrieve the results.
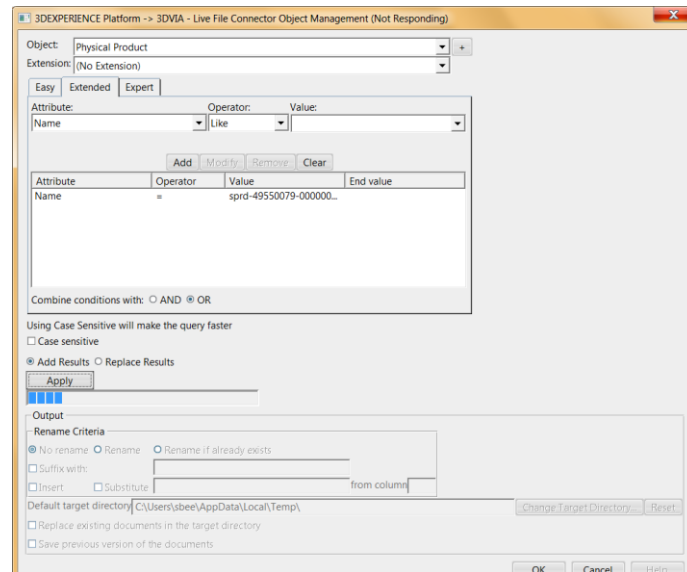
When the search is complete this panel can be dismissed, and the results are shown in the Object Selection Panel.
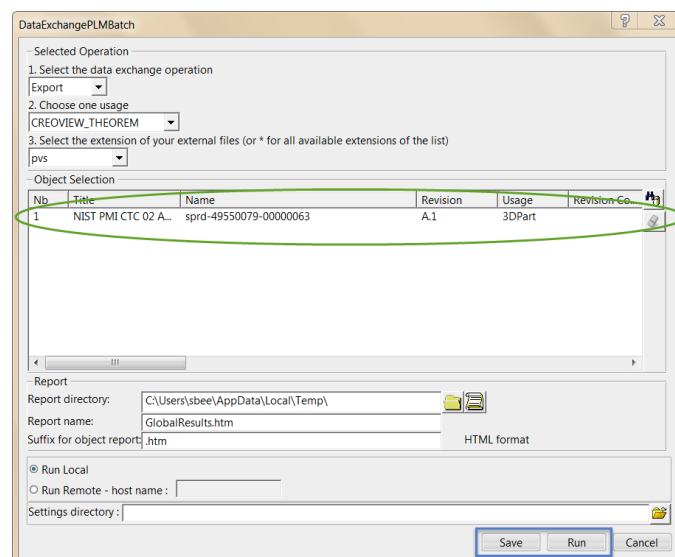
The batch file can now be run or saved for execution at a later time.

The utilities shown above are a valid method of converting data using standard Dassault batches.

This method also provides the basis for Theorem command line processing, which will now be described in more detail.

Further information regarding DataExchangePLMBatch can be found is the standard Dassault Systems documentation.

Introduction to Command Line Processing

For CATIA V6 there are no physical files on disk. So to run a command line process a job file is used instead of CAD files.

This job file contains all the necessary information to locate, open and translate the V6 data. Running a translation with a job file is similar to running CADverter with file based CAD models e.g. for CATIA V5. More about the job file and its contents is described in Appendix A

The basic command line syntax is

| Option	Description
"C:\Program Files\Theorem\23.2\bin\catia6r2020x_pv.cmd"	Command to run
C:\parts\ 2017xjobs\AS1 _2017x.xml	Input job file
-p C:\output\CreoView\2017x	output directory
-o AS1_2017xTest	Creo View pvs name
-r \parts\ 2017xrecipes\2017x.rcp	recipe name

Recipe Files

Recipe files will be familiar to users of the CATIA V5 to Creo View product.

For those new to this concept, the recipe file controls the translation options to produce the desired output.

NOTE! If you are intending to run this product with Windchill, please contact your PTC representative for the relevant documentation.
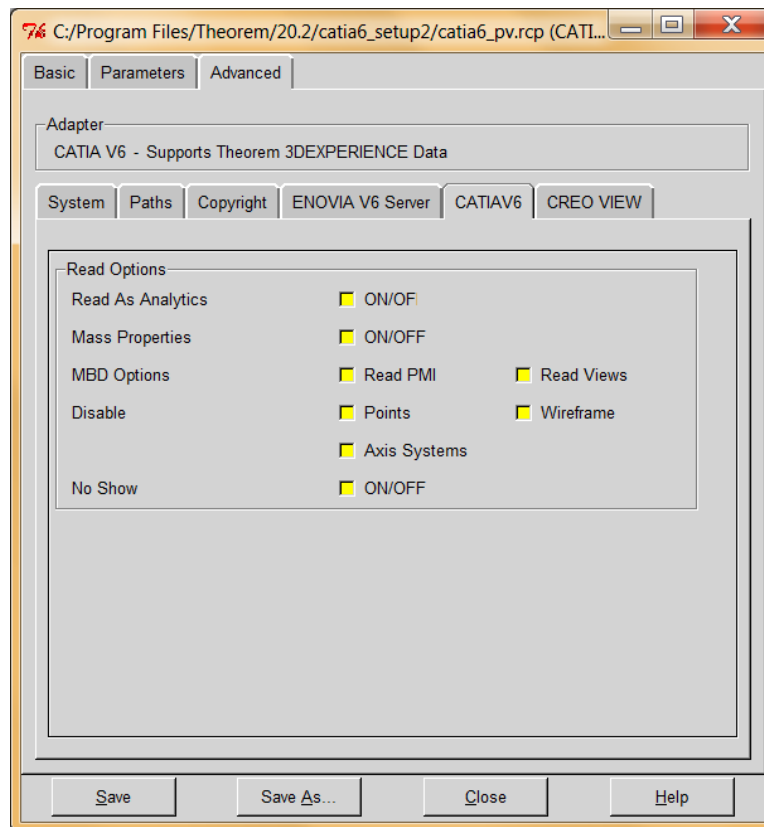
First run catia6_pv_config.cmd.exe
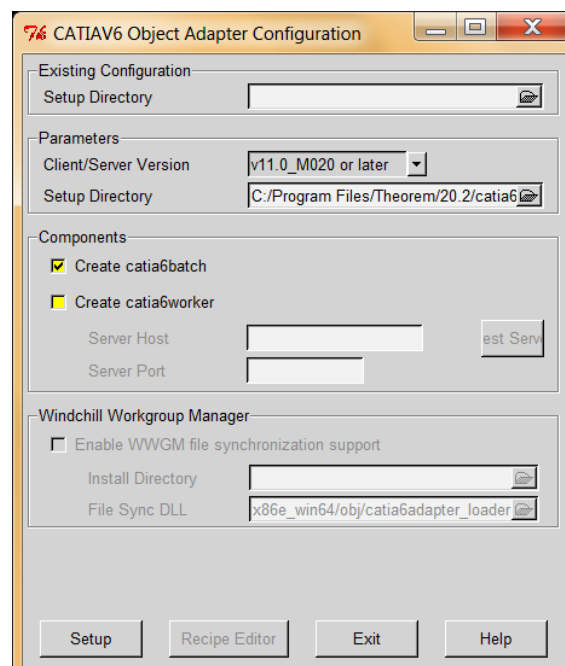
You will need to specify a setup directory. This is where your recipe file be stored. Uncheck the Create catia6worker box. Then click 'Setup'. This will enable the 'Recipe Editor' button which you should click to create your own recipe file.

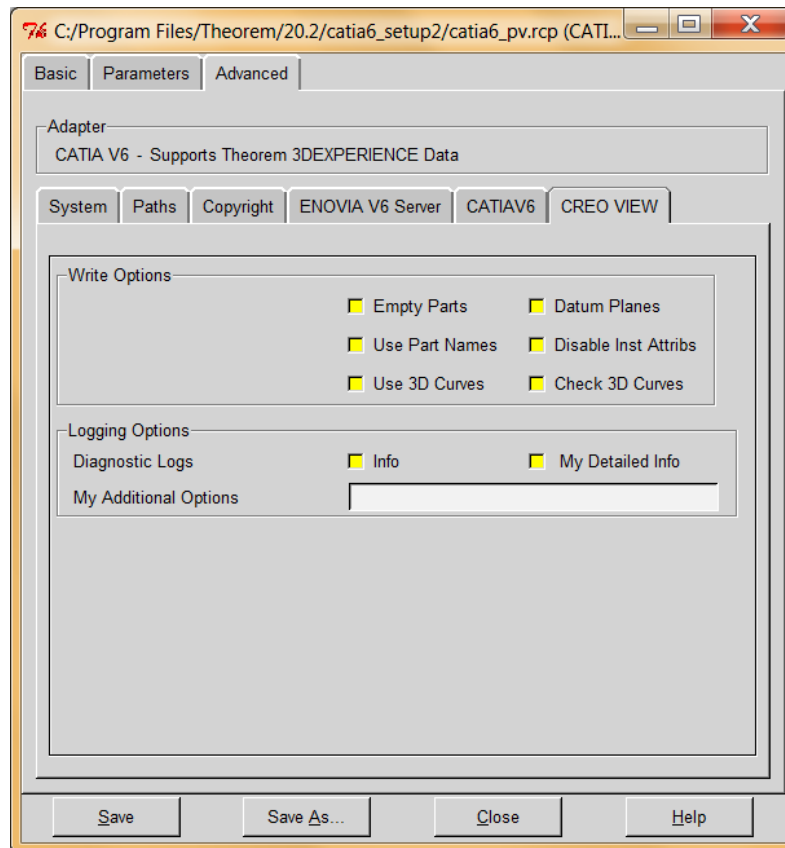There are two pages which affect the basic output

Firstly V6 read

The Options are described below

Option	Description
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	Reads data that has been hidden.

Then Creo View write

The Options are described below

Option	Description
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
Create 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 CATIA V5 'part number' names for assembly nodes. The default is to use 'Instance name'.

Disable Inst Attribs	Disables the output of any instance attributes.
Use 3D Curves	This option allows Creo View API to generate its own 2D curves. This option is most likely only ever used as a work-around when poor data is encountered.
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	Generate a more verbose log file
Detailed Info	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>*	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

Once saved the recipe file can be used:

From the command line using

-r "<mysetupfolder>\recipe\xcad_pv_pmi.rcp"

Interactively by editing the relevant line in the <installation folder>\data\Creoview\ xcad_creoview_opts.txt file.

-r "<mysetupfolder>\recipe\xcad_pv_pmi.rcp"

Log File Generation

Export process Log Files

In the process of exporting the selected 3DEXPERIENCE CATIA V6 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 CATIA V6 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 CATIA V6 'data read' processing into Theorem Intermediate data format

%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.

%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.

3DEXPERIENCE CATIA V6 Environment Files

As part of the Theorem installation process, a set of 3DEXPERIENCE CATIA V6 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 import and export.

A 'CATEnv' file is created for each installed version of 3DEXPERIENCE CATIA V6.

These environment files are located under the Theorem installation directory in

B419\win_b64\Theorem_Multi-CAD_ CREOVIEW _CATIAV6R2017x.txt

B420\win_b64\Theorem_Multi-CAD_ CREOVIEW _CATIAV6R2018x.txt

B421\win_b64\Theorem_Multi-CAD_ CREOVIEW _CATIAV6R2019x.txt

B422\win_b64\Theorem_Multi-CAD_ CREOVIEW _CATIAV6R2020x.txt

Appendix A – Job Files

Introduction

As previously documented, running with a job file is similar to running a CADverter with file based CAD models

The basic command line syntax is

Option	Description
"C:\Program Files\Theorem\21.3\bin\catia6_pv.cmd"	Command to run
C:\parts\ 2018xjobs\AS1 _2018x.xml	Input job file
-p C:\output\CreoView\2018x	output directory
-o AS1_2018xTest	output Creo View pvs name
-r \parts\ 2018xrecipes\2018x.rcp	recipe name

e.g.

"C:\Program Files\Theorem\21.3\bin\catia6_pv.cmd" C:\parts\ 2018xjobs\AS1 _2018x.xml -p C:\output\CreoView\2018x -o AS1_2018xTest -r \parts\ 2018xrecipes\2017x.rcp

Batch Processing

As part of the development of Large Assembly Processing, a new batch processing utility has been created for 3DEXPERIENCE V6 CATIA export to Creo View.

This runs as a command line executable which requires the following inputs:-

<TS_INST>\bin\catia6R2013X_pv.cmd <input_file> -p <output_path> –o <output_file> -r <recipe_file>

Where:-

<TS_INST>

> is the Theorem Solutions software installation directory.

<input_file>
> is an xml file defining the access to a specific object in a specified 3DEXPERIENCE PLM database.
> This file provides user login details (V6R2013x) or a Login Ticket (2015x, 2017x), to a specified Enovia repository
> plus a set of 3 attribute names and values which will uniquely identify the input PLM object of the conversion.

> Here is an example of the xml input file for 3DEXPERIENCE V6 R2017x:-

```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="REEyNzM3M0Q1REM2NDgxQzlFNzk1QzIwNjZGRDYzODN8Um9iaW58Um9ia
W58fHwwfA=="/>
   <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="sprd- 51097330-00023109"/>
   <attribute key="V_version" value="A"/>
   <attribute key="minorrevision" value="1"/>
  </arguments>
  </mc6_read>
```

-p <output_path>

> The folder path to the output file below

–o <output_file>

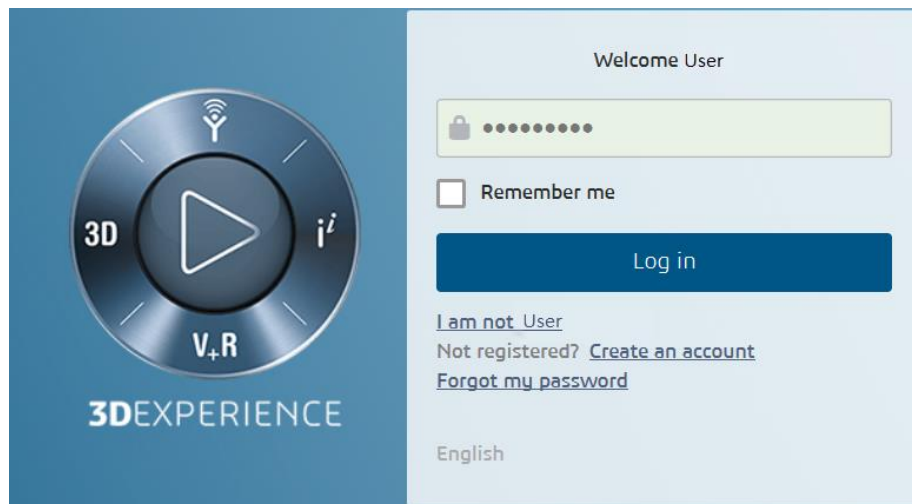   is the required output Creo View file name.

-r <recipe_file>

       Is the name of the recipe file used for controlling the output characteristics
described previously.

Appendix B – Creating a Login Ticket

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

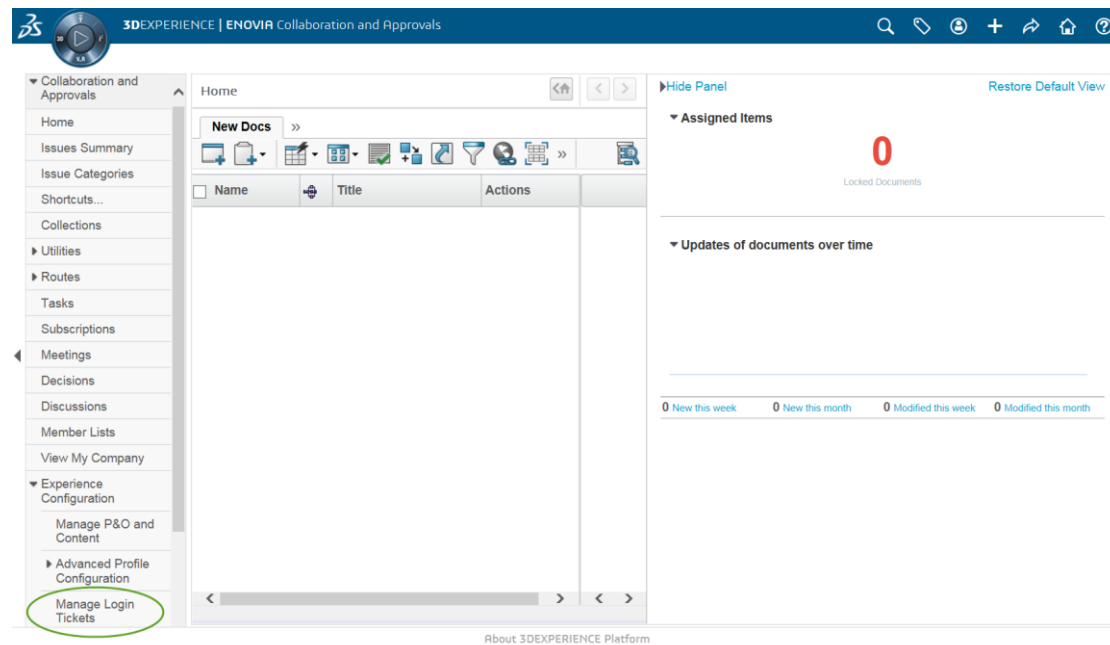https://3dspace.2017x.theorem.com:447/3dspace/common/emxNavigator.jsp

A page will appear in the browser

Login as normal, e.g.

> *User*

> *Password*

Select Collaboration and Approvals > Experience Configuration > Manage Login Tickets

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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'
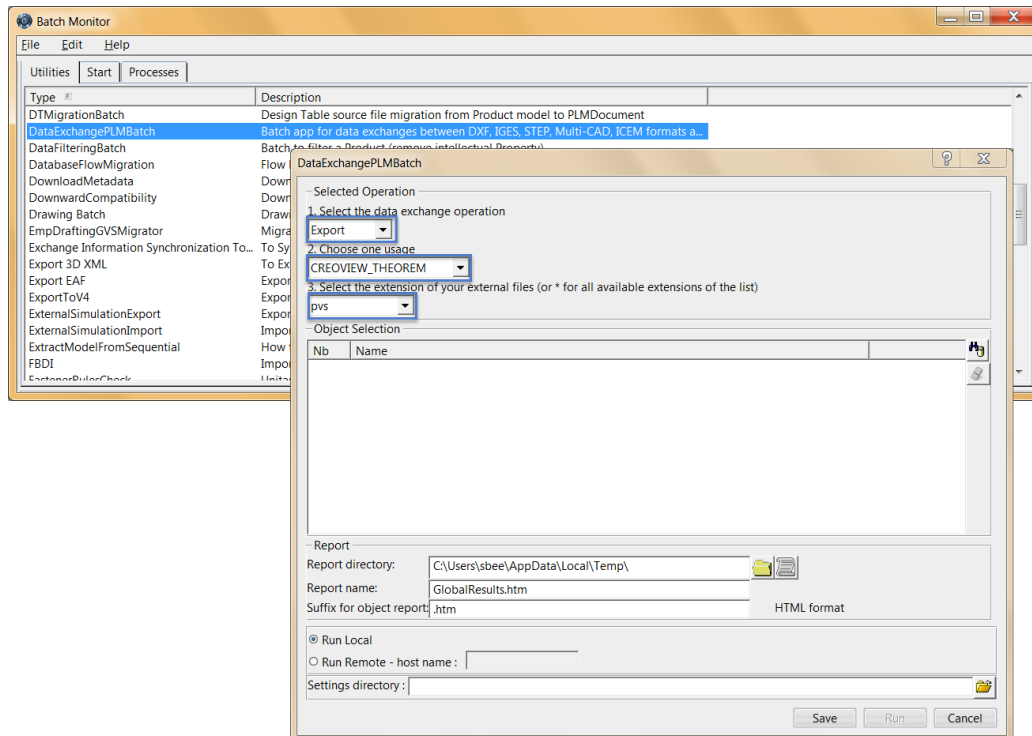
"Create" will produce the ticket:-

REEyNzM3STE1MER2NDgxQzlFNzk1QzIwNjZGNATzDAN8Um9iaW58Um9iaW58fHwwfA==

This can then be used as the 'LoginTicket' value in the job xml file described in Appendix A
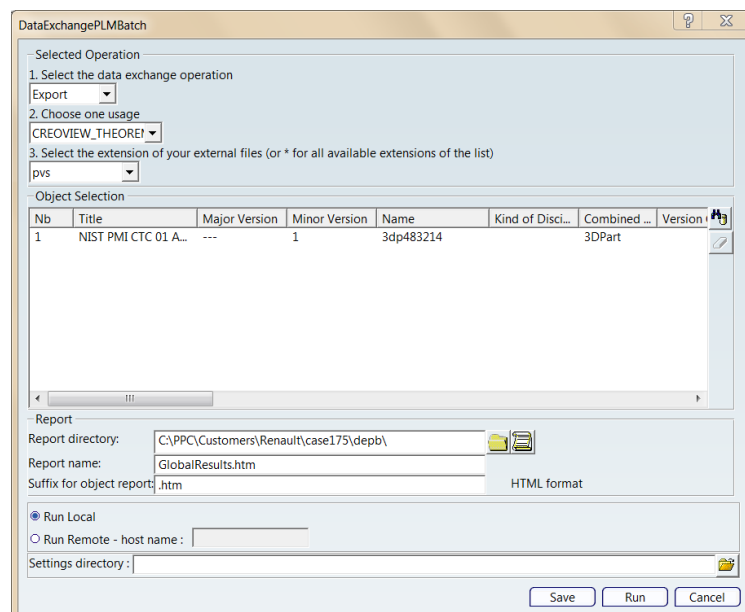
Appendix C – Batch Export to Creo View

Below is an example showing the appropriate settings in preparation for a database selection and an export "save" or "run" :-

Note selection of "Export" operation, usage "CREOVIEW_THEOREM" and extension "pvs"

Once an item is selected the translation can be either run imediately or saved for later execution.