USER GUIDE

CREO - 3DEXPERIENCE CATIA V6

Release Version : 24.2

Revision: 2.0

Issued: 17/03/2022

Contents

Overview of Multi-CAD

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:

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 Bi-directional CREO Translator

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

The 3DEXPERIENCE CATIA V6 to CREO Translator is a bi-directional direct database converter between the Dassault Systemes 3DEXPERIENCE CATIA V6 Modelling Application and PTC CREO file format.

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 as a method of collaboration and communication between OEMs and their customers or suppliers.

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

Primary Product Features

- Converts all types of geometry, wire frame, surfaces, trimmed surfaces (faces) and solid models.

- Converts assembly structure between both systems.

- Converts attribute data including colour and layer information.

- Integrated with the 3DEXPERIENCE CATIA V6 installation.

- The conversion process can be run Interactively or in Batch Mode

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

- Uses the 3DEXPERIENCE CATIA V6 API and PTC CREO Toolkit API to read and write data.

Primary Product benefits?

- Being a direct database converter all pre and post processing is eliminated, saving time.

- Reduce costs due to processing time and increase overall conversion success levels by filtering input data and focusing the conversion to only those elements required.

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

- With over 20 years of industrial use Theorem translation products robustness and quality is well proven, reducing your business risk.

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

Getting Started

Documentation

The latest copy of this documentation can be found on our web site at:

http://www.theorem.com/Documentation

Installation Media

The latest copy of Theorem software can be found via our web site at:

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

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

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

Installation

The installation is run from the CD or ZIP file download provided.

To install the translator, select the **Setup.exe** file and follow the installation process. For a full guide to the process, please see our 'Translator Installation Process' demonstration video located here.

License Configuration

In order for the translation to run successfully, the Theorem license file provided to you needs to be configured using FlexLM. For a full guide to this process, please see our 'FlexLM License Set Up and Configuration' demonstration video located here.

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

    o The Interactive Interface provides a direct method of importing and exporting to and from 3DEXPERIENCE.

- Translating from the UI (User Interface)

- In Batch via CATUtil - DataExchangePLMBatch o   The 3DEXPERIENCE DataExchangePLMBatch Interface provides a direct method of invoking the translator. It can be used on an *On Demand* basis to translate single or multiple files

- Translating from the UI (User Interface)

Using the Product

Translating Interactively from within 3DEXPERIENCE

The 3DEXPERIENCE CATIA V6 to CREO translator allows an opened 3DEXPERIENCE CATIA V6 part or Assembly to be exported directly to CREO, and for a CREO part or assembly to be imported directly into the 3DEXPERIENCE CATIA V6 application.

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 be started from a desktop shortcut created at installation time.

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_CREOxx_CATIAV6R<version>(where <version>

(where xx should be substituted with the version of CREO 4,6 or 7)

should be substituted for the version of 3DEXPERIENCE CATIA V6 that you have installed – e.g. 2017x, 2018x, 2019x, etc.):
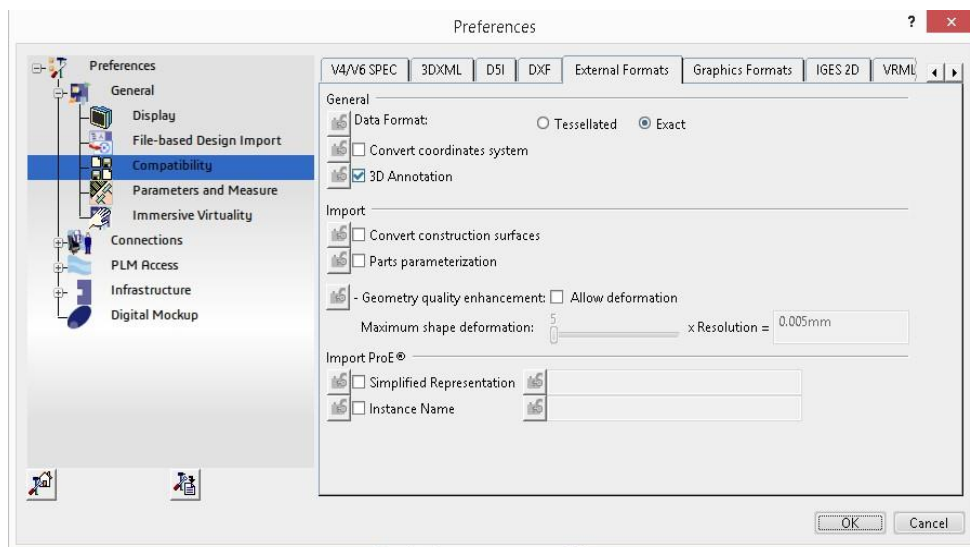
Theorem Interactive Conversion Settings

The 3DEXPERIENCE interface offer a number of setting to enable configuration of the Translation Process.

These are accessed through the User Preferences options (Preferences/Legacy Preferences for 2019x onwards) :-
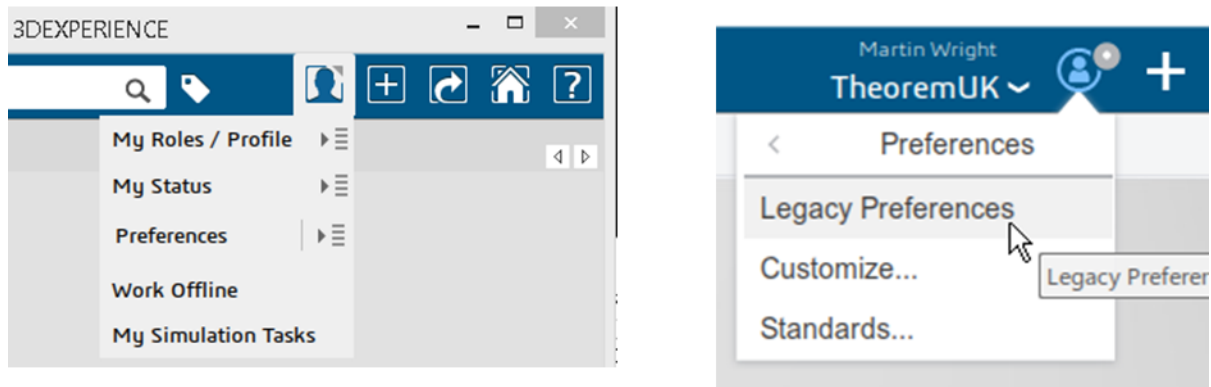

Of importance for all versions is the Compatibility options page as shown below:-

This page sets the preferred mode of conversion (in this case Exact), and also the general options "Convert coordinate systems" and "3D Annotation".

A dedicated page of options for 'Infrastructure -> Theorem ' Export and Import CREO' operations is available under the 'Infrastructure' settings and is presented as follows:-
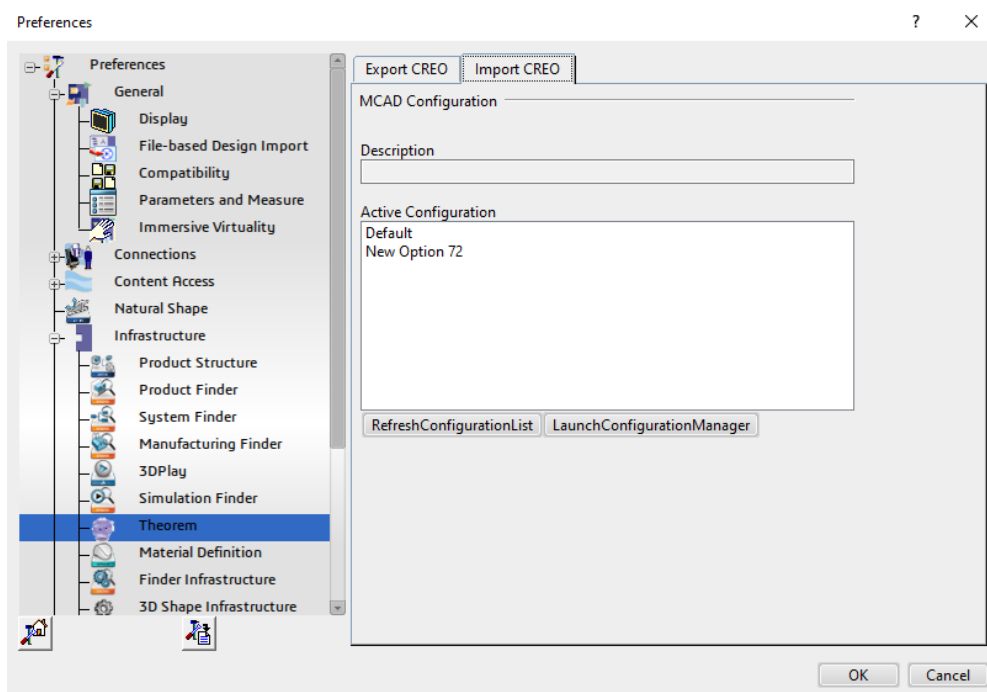
From this Panel, the user can select a predefined configuration, these are set from within the UI (see section UI) or Configuration Manager. You can also launch and refresh the Configuration Manager.
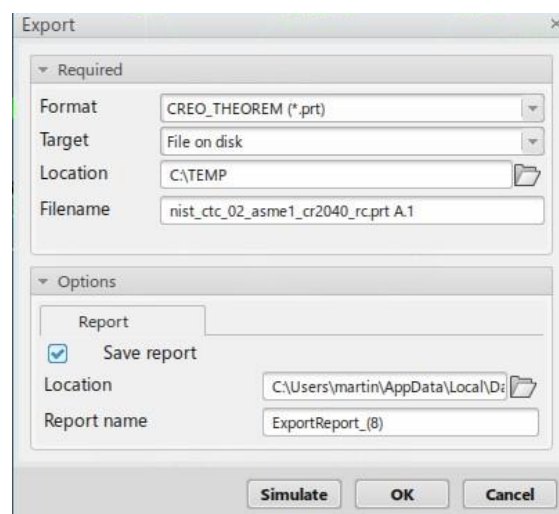
Theorem Export

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

The user selects the menu item as shown here:-

This menu action will then present the user with a Save As dialog box similar to this:-
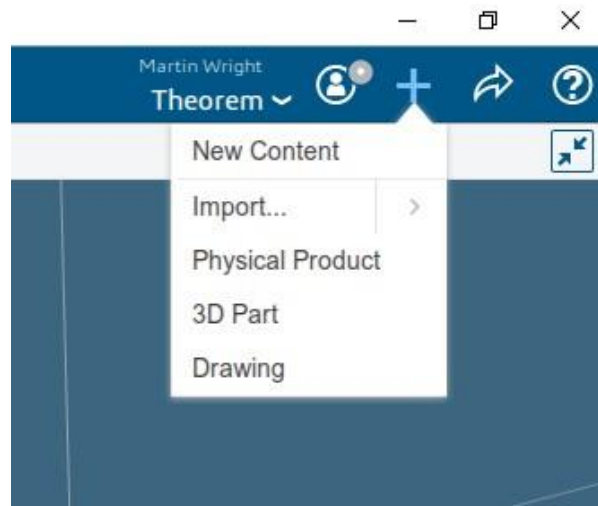

The "Save as types:" or "Format" selection should be set to CREO_THEOREM (*.prt or *.asm).

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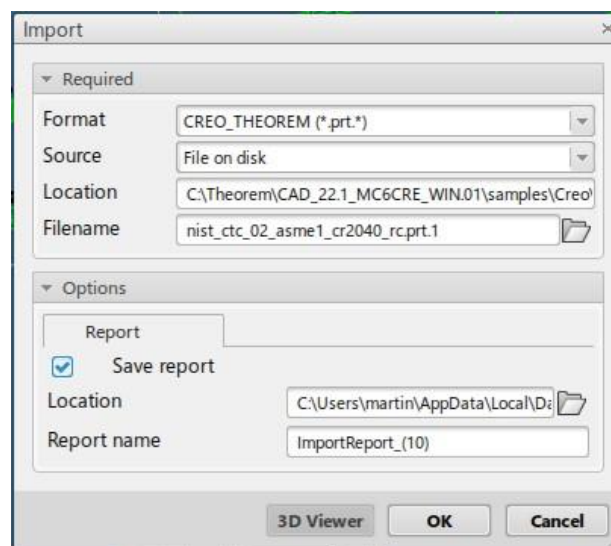
Theorem Import

Once 3DEXPERIENCE CATIA V6 has been started it is possible for a model to be imported from NX.

In order to import, the user selects the following Menu Option:

This launches the Theorem Import Menu:

On selecting **Open** or **OK** the on the File selection Panel the NX data will be translated and imported into the 3DEXPERIENCE CATIA V6 database. It will also be opened in the user's session.

Translating from the UI (User Interface)

For CATIA V6 there are no physical files on disk. So to run a User Interface 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

CADverter allows the information that is read from the source system and written to the target system to be tailored via a set of user specified arguments. Commonly used arguments are supported via the Unified Interface, with Advanced Arguments being described within this document for use in the Unified Interface or via the Command Line invocation.

Configuration Manager

The configuration Manager can be run from either the Unified Interface or by running "<theorem_install>/bin/ConfigurationManager.cmd". Select **CADverter Translator** required**,** select the **New** option to create a new configuration file and then select the options required. The config file will then be available from the <Preferences><Infrastructure><Theorem> tab.
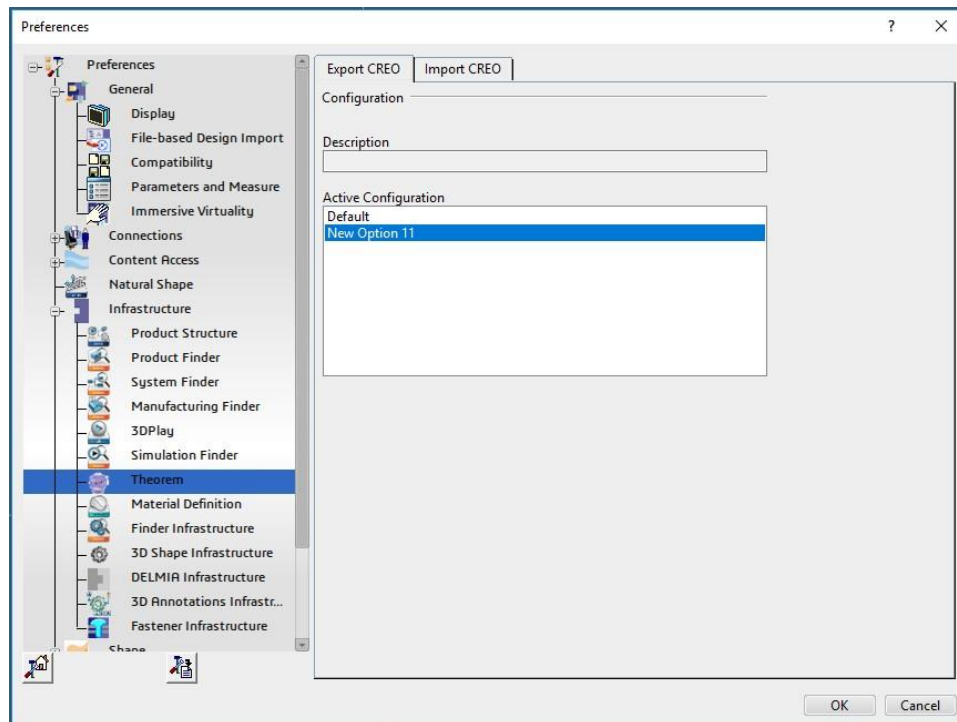
3DExperience Read Arguments

The image below shows the 3DExperience Read arguments that are available, with their default settings:
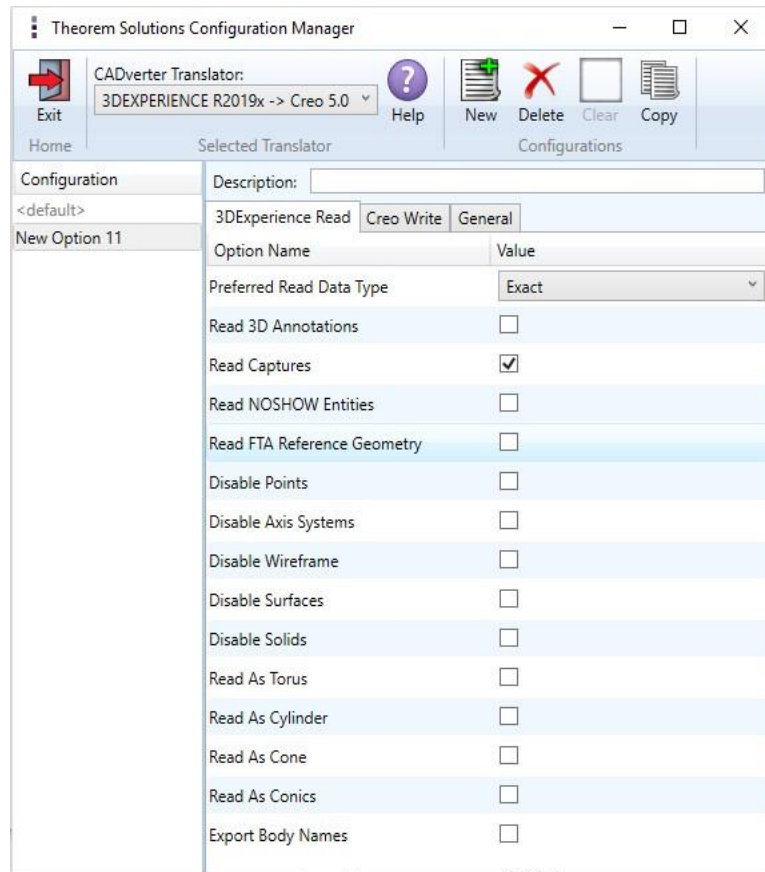
Each of these options is described below:

Option	Description
Prefered Read Data type	Select data to be read as either **Exact** or **Tesselated** • Command Line Syntax: *read_tess*

Read 3D Annotations	Enables PMI data read from the V5 file. *(Default is OFF).*
	  o Command Line Syntax:
	    ▪ *dont_read_pmi – to turn off*
	    ▪ *read_pmi- to turn on*

	Note! When *'read_pmi'* is enabled it also enables the *'fill_pmi_arrows'*, *'fill_pmi_text'* and *'pmi_filled_text'* options. These can be overridden by setting the Advanced arguments: *'dont_fill_pmi_arrows'* and/or

Read Captures	A secondary argument to 'Read PMI' and allows the control over whether captures are read as part of the process. Default is ON when 'Read PMI' is marked as ON.
	 • Command Line Syntax:
	  read_captures - default
	  dont_read_captures – to turn off

Read NOSHOW entities	Read any entities that are in NOSHOW. Default is to not read NOSHOW entities • Command Line Syntax: *noshow*
Read FTA Reference Geometry	Enables reading of FTA Reference Geometry *(Default is Off)* • Command Line Syntax: *read_geometry – to turn on*
Disable Points	Switches off Point processing • Command Line Syntax: *disable_points*
Disable Axis Systems	Switches off Axis System processing • Command Line Syntax: *disable_axes*
Disable wireframe	Switches off Wireframe processing • Command Line Syntax: *disable_wireframe*
Disable Surfaces	Switches off Surface processing • Command Line Syntax: disable_surfaces

Disable Solids	Switches off Solids processing • Command Line Syntax: *disable_solids*
Read as Torus	Read Toroidal surfaces in analytical form (default is NURBS) • Command Line Syntax: *read_torus*
Read as Cylinder	Read Cylindrical surfaces in analytical form (default is NURBS) • Command Line Syntax: *read_cylinder*
Read as Cone	Read Cone surfaces in analytical form (default is NURBS) • Command Line Syntax: *read_cone*
Read as Conic	Read surfaces generated from a Conic in analytical form (default is NURBS) • Command Line Syntax: *read_conics*
Export Body Names	Maintains body names for parts that consist of multiple bodies. • Command Line Syntax: *body_names*

CREO Write Arguments
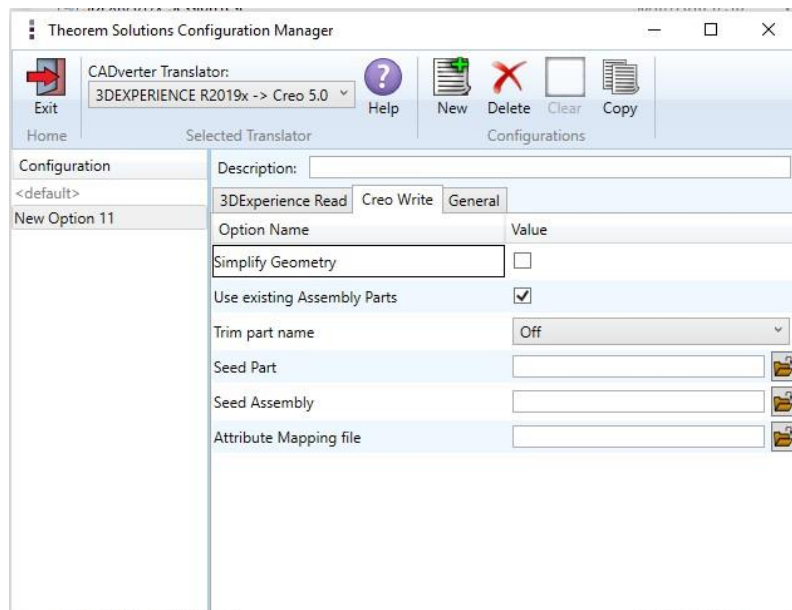
| Simplify Geometry	Attempt to write Analytical Geometry where possible (*Default is Off*) • Command Line Syntax o *simplify*
Use existing Assembly Parts	If Output Folder already contains output Files of the same name, do not Overwrite those Files (*Default is Off*) • Command Line Syntax o *use_parts*
Trim part name	If the part name is over 30 characters, use the first X characters and the final (X-30) characters to reach the 30 character limit.
Seed Part	*Use a Creo Seed Part file when creating the Creo output* 🞂 *Command Line Syntax* o *seed_prt*

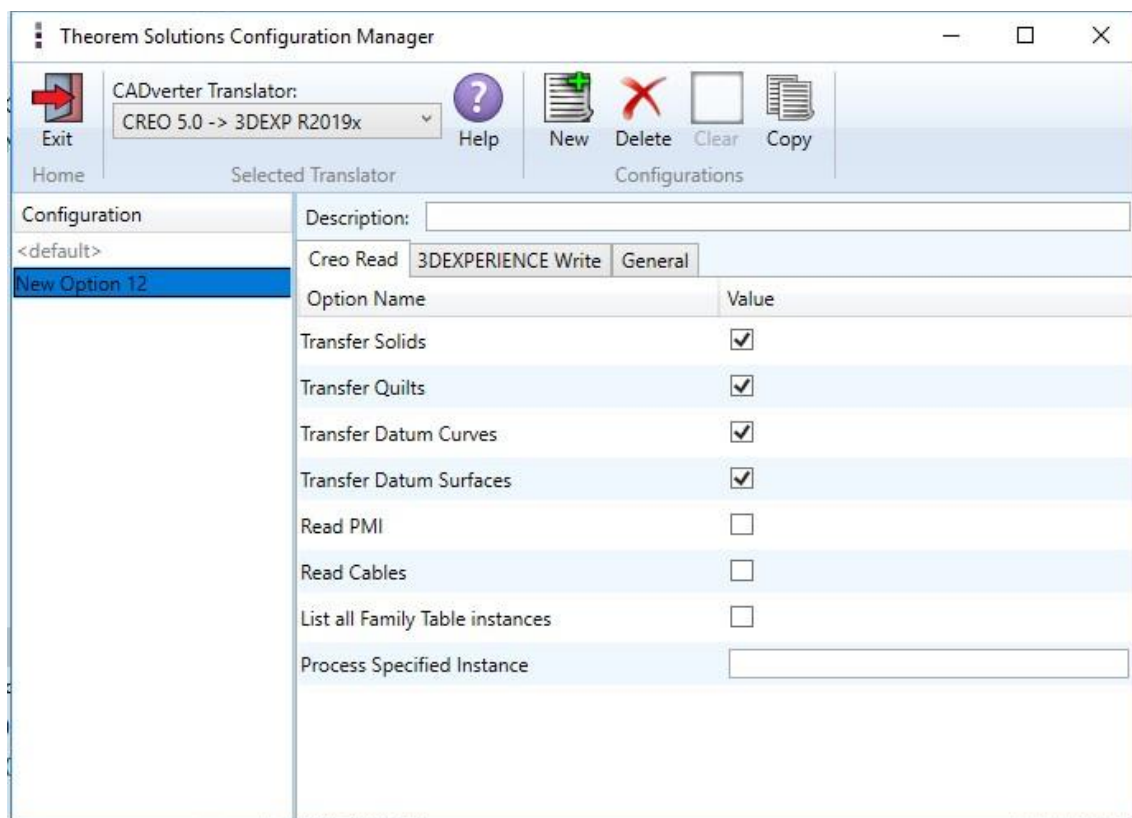|	
Seed Assembly	*Use a Creo Seed Assembly file when creating the Creo output ⬛ Command Line Syntax* *o seed_asm*
Attribute Mapping File	*Select a standard property mapping file* *⬛ Command Line Syntax* *o cad_prop_map_file*
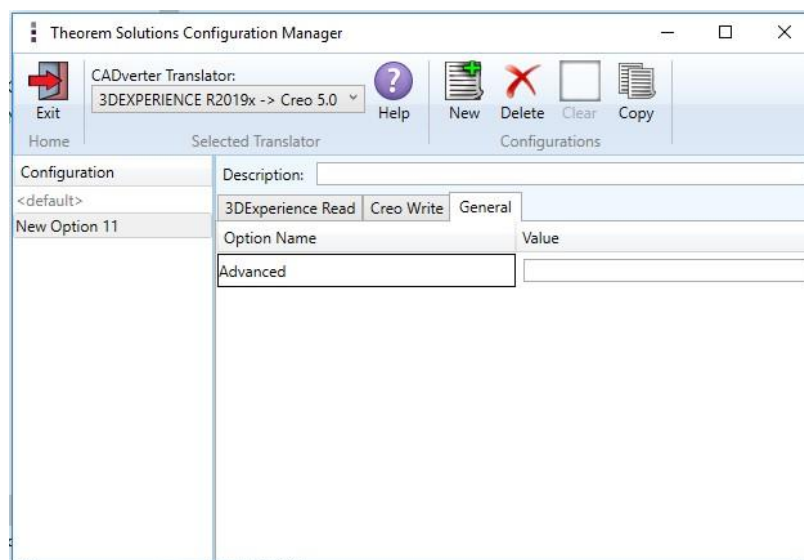
Advanced options

Any of the advanced arguments can be added to the Command Line Invocation or to the General->Advanced field when run from within the User Interface.

Option	Description
edge to nurbs	Convert curves to NURBS. Default is OFF
surf to nurbs	Convert surfaces to NURBS. Default is off.

no_check_geom	Disable solid geometry checking during CREO read operations. Possibly improve throughput by allowing the write of geometry which may not pass the geometry checks.
no_datum_curve	Disable the processing of Datum Curves
disable_default_colours	Disable applying default colours to un-coloured entities.
process_hidden_geom	Process all geometry, including hidden layers.
read_pmi	Process PMI data.
read_assy_pmi	Process PMI found at the assembly level.
write_stroked_pmi	Causes PMI information to be stored in stroked(rendered) form as required by MultiCAD.

CREO Read Arguments

Each of these options is described   below.

Option	Description
Transfer Solids	Enables solid processing. (Default is on). ⏹ Command Line Syntax: o no_solids – to Disable
Transfer Quilts	Enables quilt processing. (Default is on). ⏹ Command Line Syntax: o no_quilts – to Disable
Transfer Datum Curves	Enables Datum Curve processing. (Default is on). ⏹ Command Line Syntax: o no_datum_curves – to Disable
Transfer Datum Surfaces	Enables Datum Surface processing. (Default is on). ⏹ Command Line Syntax: o no_datum_surfaces – to Disable
Read PMI	Enables Datum Surface processing. (Default is off). ⏹ Command Line Syntax: o read_pmi
Read Cables	Enables the reading of Cable data from Creo. (Default is off). Note that in the Creo Configuration Editor, the setting display_thick_cables should be set to yes. ⏹ Command Line Syntax: o read_cables

| List all Family Table instances	List all instances of Family table to the progress file. (Not to be used in conjunction with Process Specified Instance). (Default is off). ⬚ Command Line Syntax: o instance LIST_ALL
Process Specified Instance	Process Specified Instance (Not to be used in conjunction with 'List all Family Table instances') ⬚ Command Line Syntax: `o instance [name_of_instance]`

3DExperience Write Arguments
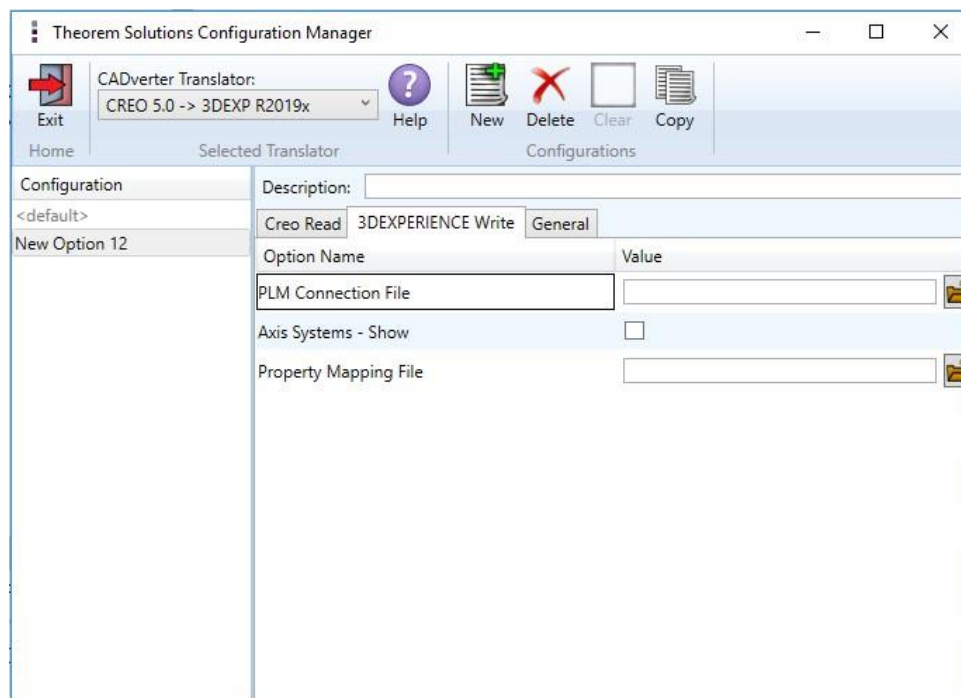
PLM Connection File	The PLM Connection file is one of the xml files defining the connection parameters to the PLM database. It is required to create a connection and logging into a PLM session programmatically.
Axis Systems - Show	Show all translated Axis Systems
Property Mapping File	Assign a property mapping file ❑ Command Line Syntax o cad_prop_map_file [Path to file]

Translating in Batch using CATUtil – PLMBatchDataExchange

The Dassault Systemes CATUTIL batch management 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_CREO_CATIAV6R<revision>.cmd

(where [revision] should be substituted for the version of 3DEXPERIENCE that you have installed – e.g. 2013x, 2014x, 2015x, 2016x)

The settings made in the user options page "MultiCAD CREO" described in the interactive usage above, are also used in BATCH mode.

In some cases, it may be desirable to ignore the interactive settings while running in batch mode.

This can be achieved by setting the following variable in the CATIA Environment being used:-

**TS_IGNORE_CREO_CATSETTINGS=1**

Batch Export to CREO

Here is an example showing the appropriate settings in preparation for a database selection and an export "save" or "run":-
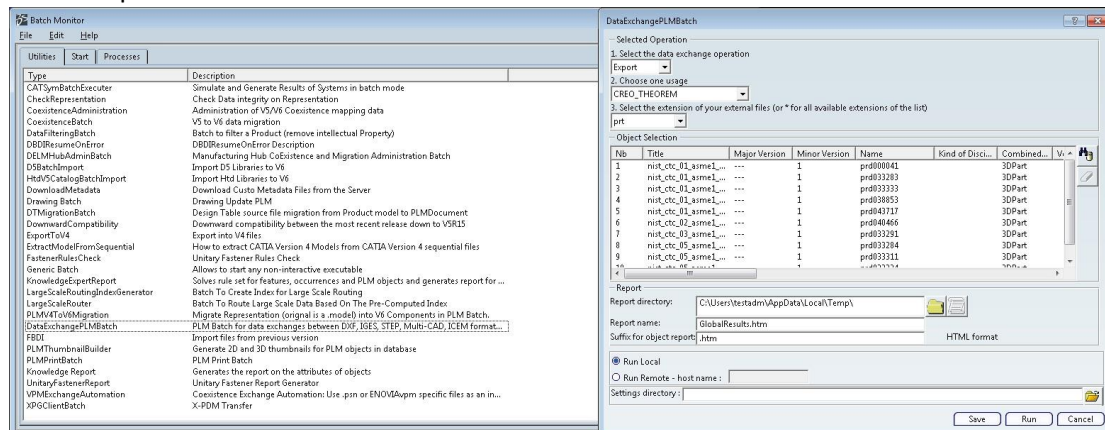
Note selection of "Export" operation, usage "CREO_THEOREM" and extension "prt".

Batch Import from CREO

Here is an example showing the appropriate settings in preparation for an input CREO file selection and an export "save" or "run":-
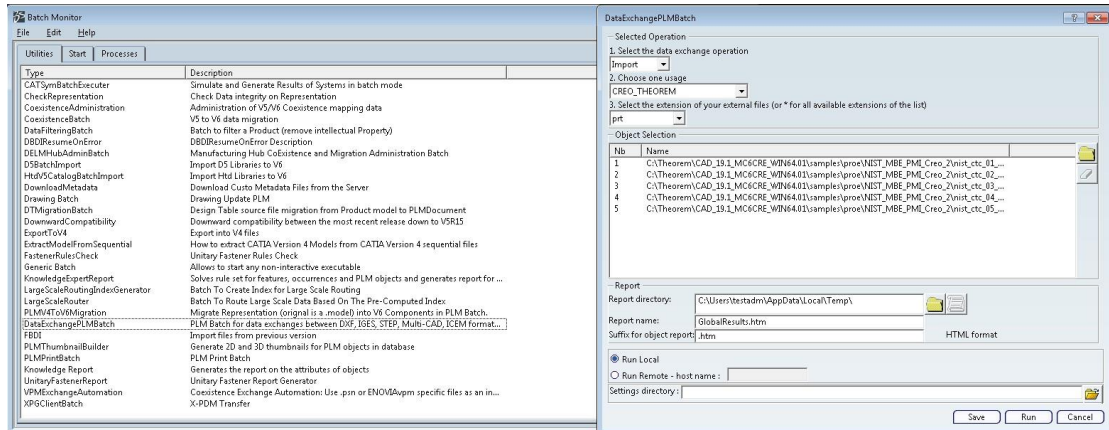
Note selection of "Import" operation, usage "CREO_THEOREM" and extension "prt".

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 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.

%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.

Import process Log Files

In the process of importing a CREO file, the following log files are generated by the CREO Import 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 selected input file name. e.g. if the file Mypart.prt were selected, the log file names would be *%CATReport%\Mypart.err and %CATReport%\Mypart.rpt.*

%TS_TEMP_DIR%\ *<input_file_name>*_viewer.log

%TS_TEMP_DIR%\ *<input_file_name>*_viewer_screen_output.log

%CATReport%*<input_file_name>*.err

Where *<Input_file_name>* - is the base file name of the imported file.

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 import and export.

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

These environment files are located in

<install_dir>\B420\win_b64\CATenv\Theorem_Multi-CAD_CREOx_CATIAV6R2018x.txt

<install_dir>\B421\win_b64\CATenv\Theorem_Multi-CAD_CREOx_CATIAV6R2019x.txt

<install_dir>\B422\win_b64\CATenv\Theorem_Multi-CAD_CREOx_CATIAV6R2020x.txt

<install_dir>\B423\win_b64\CATenv\Theorem_Multi-CAD_CREOx_CATIAV6R2021x.txt

Where x = Version of CREO installed

Appendix A – Job Files

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\catia6R2017X_pdf.cmd <input_file> <output_path <output_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 (V6R2019x) or a Login Ticket (2019x), 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 R2019x:-

```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.2019x.theorem.com"/>
  <attribute name="ServerPort" value="447"/>
  <attribute name="ServerRootURI" value="3dspace"/>
  <attribute name="LoginTicket"
value="QzdBRTA4NzEzN0I5NDRFNTlFRTUOM0RFMDRFRDc1N0N8Um9iaW4xNnh8
Um9iaW4xNnh8fHwwfA=="/>
  <attribute name="PLMType" value="VPMReference"/>
  <attribute name="ReportDirectory" value="C:\TEMP\V6Export"/>
  <attribute name="LicenseData" value="LIC"/>
  <attribute name="BatchXMLFileName" value="Default"/>
 </parameters>
```
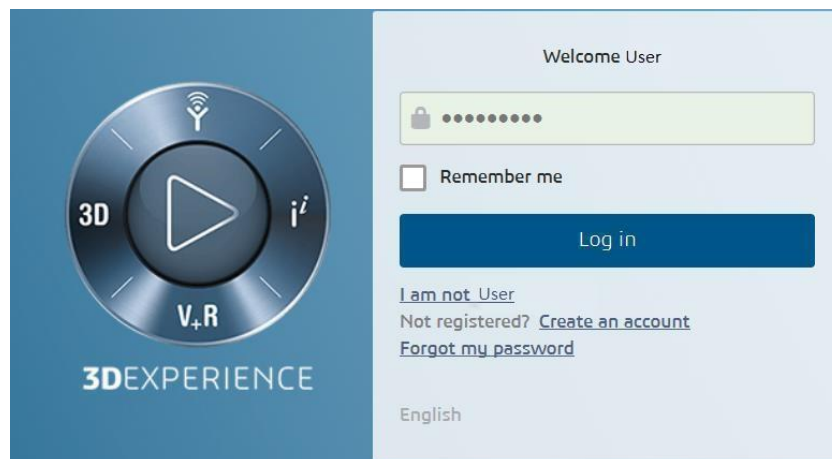
```
    <!-- arguments both key and value are passed to V6 -->
    <arguments>
     <attribute key="PLM_ExternalID" value="sprd-49550079-00000063"/>
     <attribute key="V_version" value="A"/>
     <attribute key="minorrevision" value="1"/>
    </arguments>
   </mc6_read>
```

Appendix B – Creating a Login Ticket

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

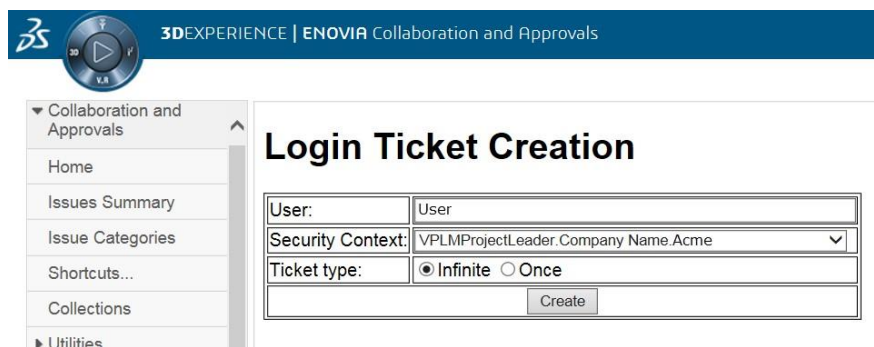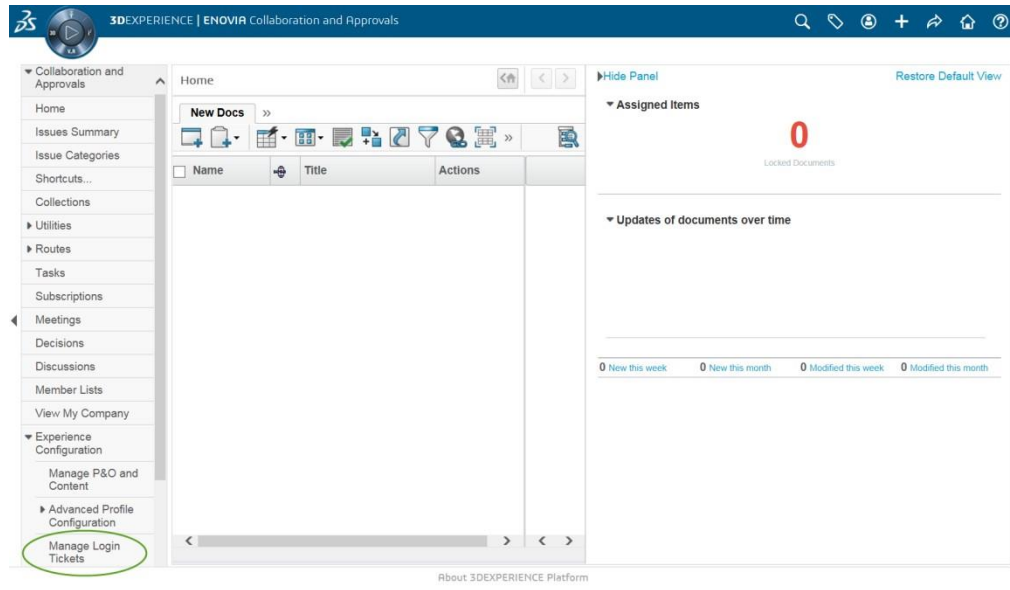https://3dspace.2019x.theorem.com:447/3dspace/common/emxNavigator.jsp

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:-

REEyNzM3STE1MER2NDgxQzlFNzk1QzIwNjZGNATzDAN8Um9iaW58Um9iaW58fHwwfA==

This can then be used as the 'LoginTicket' value in the job xml file described in Appendix A