User Guide

Multi-Cad CatiaV5 <> CREO

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Overview of TRANSLATE

About Theorem

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

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

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

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Theorem's Product Suite

Theorem have 3 main Product brands. These are:

CADTranslate

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

See our website for more detail.

CADPublish

The creation of documents enriched with 3D content

See our website for more detail.

TheoremXR

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

See our website for more detail.

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The CATIA V5 Multi-CAD Bi-directional CREO Translator

The CATIA V5 Multi-CAD to Creo Parametric (CREO) translator may be installed on a number of machines each accessing a central network floating license.

The CATIA V5 Multi-CAD to CREO Translator is a bi-directional direct
database converter between the Dassault Systemes CATIA V5 Modelling Application and the CREO Modelling Application.

It enables the user to convert all forms of 3D Mechanical Design Geometry and Assembly data, together with system defined attribute information, colour information, between these two systems. This product is designed for companies who require 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 with the command line interface allowing the conversion process to be integrated into any process oriented operation. Alternatively the conversion process may be operated by using the Theorem Unified Interface.

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.
- Converts File Properties.
- Integrated with the CATIA V5 installation.
- The conversion process can be run Interactively, Batch Mode or using the new Unified Interface
- Command line interface allows process integration into any workflow or automated process.
- Uses the Dassault Systems CATIA V5 Multi-CAD API and the Creo Parametric API to read and write the respective data formats.
- When writing CATIA V5 data the user is able to configure the derived geometry to be created in either VISU (CGR), VISU + Snap (CGR + Canonical data to aid positioning) or CATPart format data.

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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 the Multi-CAD for CATIA V5 – CREO product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

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Getting Started

Documentation & Installation Media

The latest copy of each release is specified in the Product Release Document associated with each specific release. A copy of the current product release notes can be found on the Theorem Solutions web site at:
http://www.theorem.com/Product-Release-Notes

Simply download the Product Release Document and select the hyperlink within the document to download the installation media.

For Multi-CAD related products the installation requires 3 CD's to be installed

1. The Theorem Solution TXX Multi-CAD Platform CD
2. The Theorem Solutions Unified Interface CD
3. The Theorem Solutions Multi-CAD CATIA V5 – CREO CD

Alternatively, you can request a copies of the software to be shipped on a physical CD media. Please contact your sales representative mailto:sales@theorem.com to arrange the shipment of the physical CD media.

Installation

The installation is run from the .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation and select UI from the product selection list.

License Configuration

To run any product a valid license file is required. The Flex License Manager is run from the .msi file download provided. 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

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Interactive usage within CATIA V5

Starting CATIA V5
In order to use the Theorem CADverter products interactively within the CATIA application, you must start CATIA V5 with the correct CATEnv environment settings. A desktop icon and a start menu item are both able to be created during the installation process to achieve this.

The desktop icon is labelled "Theorem Multi-CAD CREO CATIAV5Rnn" (where nn is the release version (26 for V5-6R2016, 27 for V5-6R2017, 28 for V5-6R2018).

The Start menu item is located under the "Start + All programs + CATIA" folder and is also named "Theorem Multi-CAD CREO CATIAV5Rnn".

Using "Insert Existing Component"

The CATIA V5 Multi-CAD architecture allows you to invoke the Theorem CREO to V5 Multi-CAD translator to import components or assemblies into an active CATIA V5 CATProduct by using the "Insert > Existing Component…" menu item. Also note that the CATIA associativity (Update Status Checker) mechanism can invoke the translator execution.

In order for the Theorem translator to be invoked from within CATIA V5, the setting of the **"Preferred Conversion Technology"** must be set to **"Indirect"** on the CATIA V5 **Tools > Options > Compatibility > "External Formats"** option page. ***(See section - Setting Conversion Options in CATIA V5)***

Note that this option page is only available if the TXX-THEOREM-MULTICAx GATEWAY (Theorem Partner) license is available which is provided by the Theorem license manager. Please see the installation document named "**CATIA V5 MultiCAD_Installation_Instructions.pdf"** for details.

There are some extra conversion options available to the Theorem User Interface that are not presented on the CATIA V5 External Formats options page.

On invoking the **"Insert Existing Component…"** command in CATIA V5, and selecting a product node for the 'insert' operation, you will see the following file selector.

Note: CREO files may contain a version number at the end of the file name. (e.g.: partname.prt.10) Therefore this is addressed by adding an extra filter to the file selector menu.

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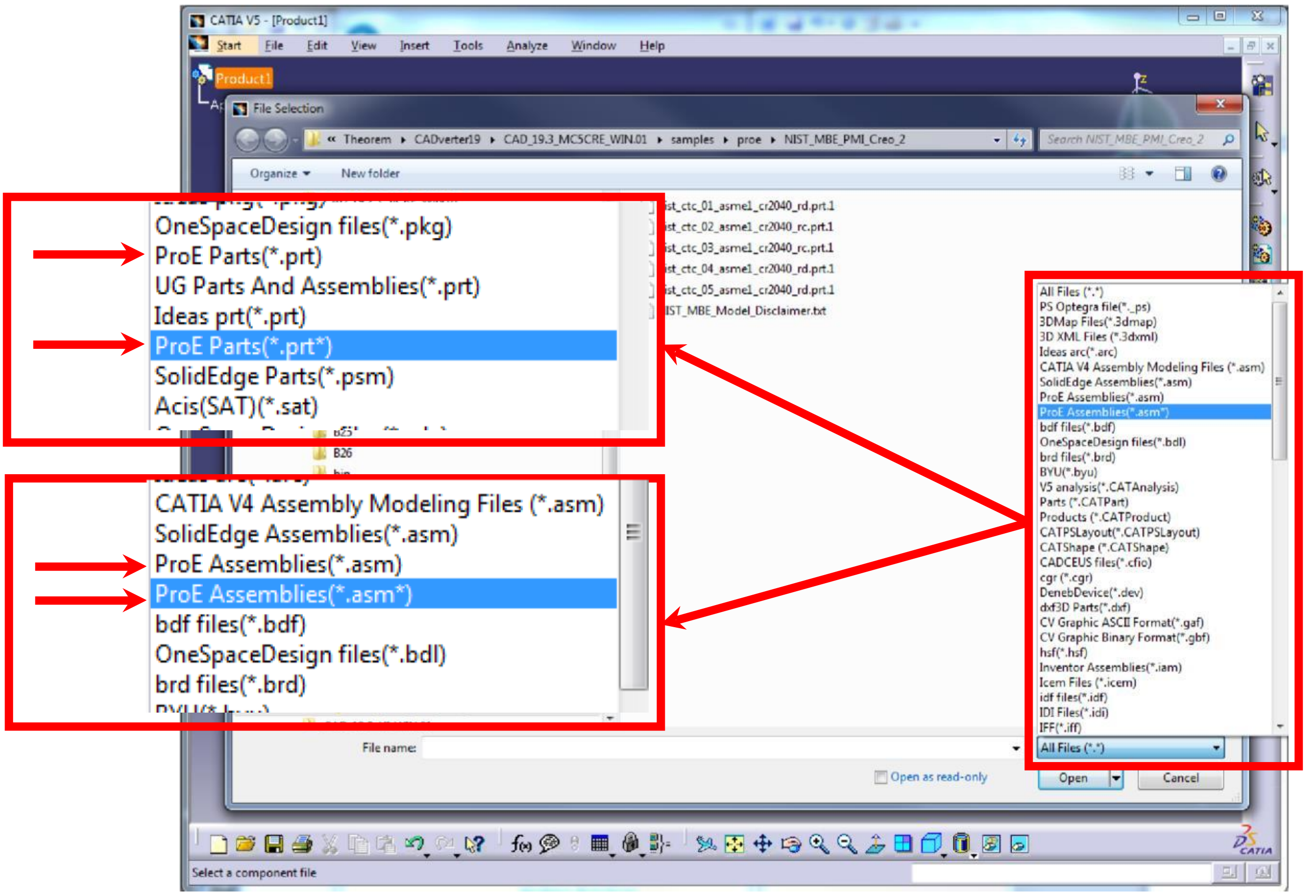

The translation will now proceed and the CATIA V5 representation of the CREO assembly/components will be incorporated in the active CATProduct.

It is worth noting here that CATIA launches and manages a conversion sub-process that feeds the option page settings into the Theorem Multi-CAD translator via a temporary file named ***%CATTemp%\ProEToNavConfigs.txt***
This is referenced using the ***'–c <config_file>'*** command line option.

The Theorem User Interface allows you to select a CATIA generated configuration file as described above, but if none is selected, a default configuration file installed with the Theorem software under %TS_INST%\data\proe\ProEToNavConfigs.txt will be used. For further information on the CATIA V5 Multi-CAD integration methods and commands see the CATIA Infrastructure Documentation.

The Theorem CREO to V5 Multi-CAD translator is also invoked when the CATIA "Update Status Checker" command determines that a CREO part file that was previously inserted using "Insert Existing Component" has been subsequently modified outside of the CATIA environment.

For further information on the usage of the CATIA commands "Insert Existing Component" and "Update Status Checker", consult the CATIA Documentation under headings:

- Mechanical Design → Assembly Design • Here you will find topics ○ "Insert an Existing Component" ○ "User Tasks" → "Updating an Assembly"

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Visualizing Inserted Annotations

It should be noted that when you perform the “Insert Existing Component” command with a CREO file containing PMI data, you must perform the following operations to visualise it if the Link Mode used is ‘Visu’ or Visu Snap’

- Activate a workbench supporting FTA operations such as with menu item “Start > Mechanical Design > Product Functional Tolerancing & Annotation”.
- Use menu “Insert > Visualization > List Annotation Set Switch On/Switch Off”
- Enable the listed Annotation Sets and apply the change.

CREO to CATIA V5 MultiCAD Usage

Setting Conversion Options in CATIA

If you have the TXX-THEOREM-MULTICAx license, the following categories of options will be visible from the CATIA menu - Tools > Options on the Compatibility “External Formats” page

- **Visu Format Unit**
- Preferred Conversion Technology
- Link Mode
- Preferred Translation Mode
- Others
- Output of generated data
- Ideas®
- ProEngineer®
- Some of these settings are accessible to the Theorem CATIA V5 to CREO Multi-CAD

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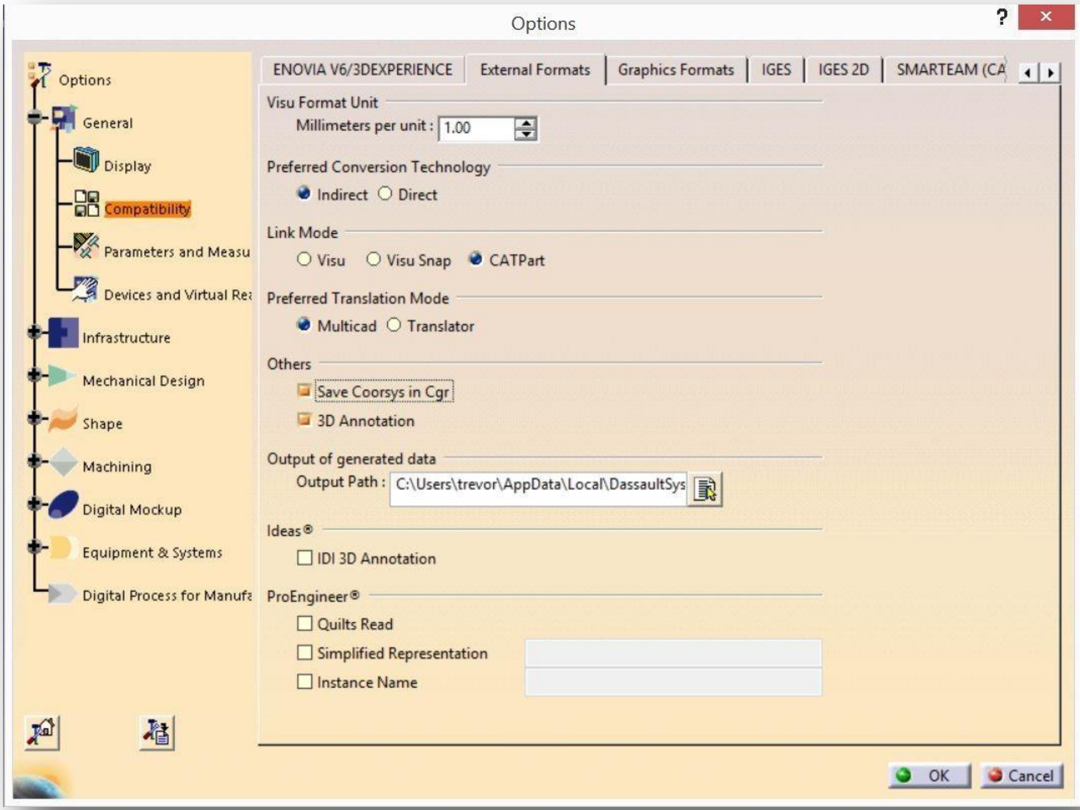

translator when it is run in both interactive and command line (batch) mode.

Visu Format Unit

- Millimeters per unit
  The conversion of units from CREO parts is incorporated into the Theorem CADverter, so this value should always be set to 1.00

Preferred Conversion Technology

Indirect

This parameter determines that the 3rd party Theorem converters will be used in conversion operations.

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Direct

This parameter determines that the Dassault Systemes converters will be used in conversion operations.

Link Mode

Visu

Tessellated data.
By default, this option is activated.

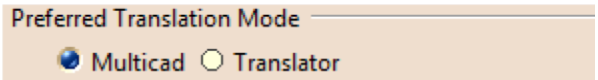
Visu Snap

Selecting this option forces the processing of CREO precise BREP geometry to generate a CATIA V5 CGR output. In addition canonical shape properties are also generated in the CGR file. Consequently the CGR data resulting from importing CREO will have additional geometry snapping capabilities. After selecting this option, this message appears: "Please, restart session to take modifications into account".

CATPart

This option causes the import of exact/brep geometry from the CREO file to be created in CATPart form.

Preferred Translation Mode



Selecting this option controls if links will be retained between the source CREO files which are imported, and the resulting CATIA files generated. In Multicad mode, links are retained and modifications are locked, in Translator mode, links are not retained and modifications are allowed.

Others

Save Coorsys in Cgr

This option saves the CREO PMI co-ordinate system data into the derived CGR representation when Visu output is selected. Note this feature would also require the 3D Annotation setting to be selected to create the required output.

3D Annotation

This parameter determines whether 3D annotation data, referred to as “PMI” data in CREO, will be imported.
(Note, requires one of the following licenses):

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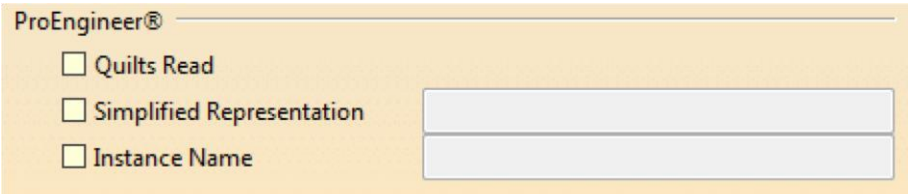
CATIA: FT1.prd or FTA.prd
ENOVIA: DT1.prd
DELMIA: MTR.prd or MFT.prd

Output of generated data

Output Path

Setting the Output Path location enables you to customize the folder location that is used when writing the derived CATIA V5 generated data. It specifies the location where CGR, CATPart and CATProducts will be generated.

ProEngineer



Quilts Read

Enable the reading of Quilts or Surfaces from Creo.

Simplified Representation

Specify which Simplified Representation is to be read from Creo.

Instance Name

If the file to be translated contains a “Family Table” this option allows the user to specify which “Instance” is to be processed from the Family Table.

The CREO to CATIA V5 Multi-CAD options file

There are some extra conversion options available to the Theorem CADverter GUI that are not presented to the user on the CATIA External Formats options page. A mechanism to allow you to specify additional options to the CREO to V5 Multi-CAD translator is provided via a text file that the translator reads during the conversion process.
The default location of this options file is defined in the %TS_INST%\bin\TheoremProps.txt file by the setting:

Theorem. ProeServerCfile=%TS_INST%\\data\\proe\\proe_xcad_opts.txt

This text file can be edited to contain any of the following command line options on a separate line:- **info**

Outputs extra processing information to log files.

debug

Outputs extra debugging information to log files help diagnose problems.

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surf_to_nurbs

Read surfaces in NURBS form, rather than the default analytic forms (cylinder, cone, torus etc.).

edge_to_nurbs

Read edge curves in NURBS form, rather than the default analytic forms (conics). **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_curves

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.

The CREO to V5 Progress and Log File Outputs

The CREO to CATIA V5 XCAD process log and error messages are recorded in a '.log' file located in the CATIA CATReport directory. A process summary file is also produced here which contains the completion status of the conversion. These files are named after the selected input file name. E.g. if the file tea.prt were selected,

the log and summary file names would be %CATReport%\tea.log

and %CATReport%\tea.log.summary.

If the CREO to CATIA V5 XCAD process is run using the CADVerter GUI or using the command line option 'progress_file' <name>, the log file output will honour this name, and the summary file will be named similarly with the suffix '.summary'.

The V5 to CREO Progress and Log File Outputs

The CATIA V5 XCAD to CREO process log and error messages are recorded in a '.log' file located in the CATIA CATReport directory. The file is named after the selected output file name. E.g. if the file Mypart were selected, the log file

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names would be %CATReport%\Mypart.log and
%CATReport%\Mypart.log.summary.
If the CATIA V5 XCAD to CREO process is run using the CADVerter GUI or using the command line option ‘progress_file’ <name>, the process log and error messages will be output to the specified name, and the summary file will also be named after this name with the suffix ‘.

Using the Theorem User Interface

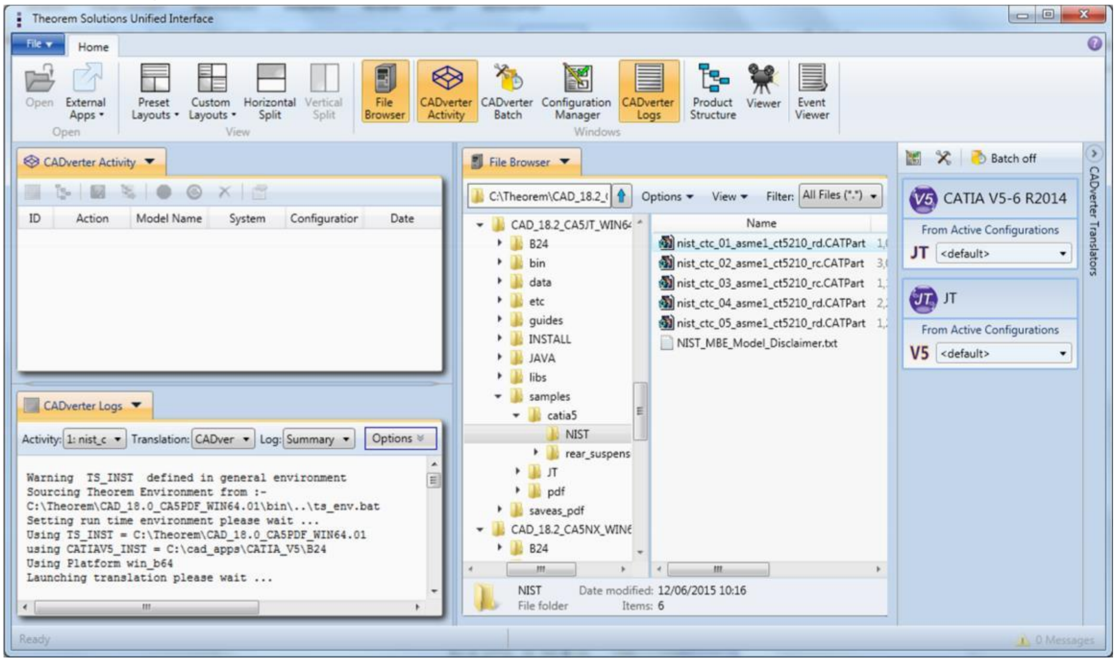
Default Translations

Default Translation – via the Unified Interface

The Unified Interface can be started via the Start Menu – if a shortcut was added during installation. Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

The following interface will be launched:



The default layout is split into 4 primary areas, which can be altered to the users prefer:

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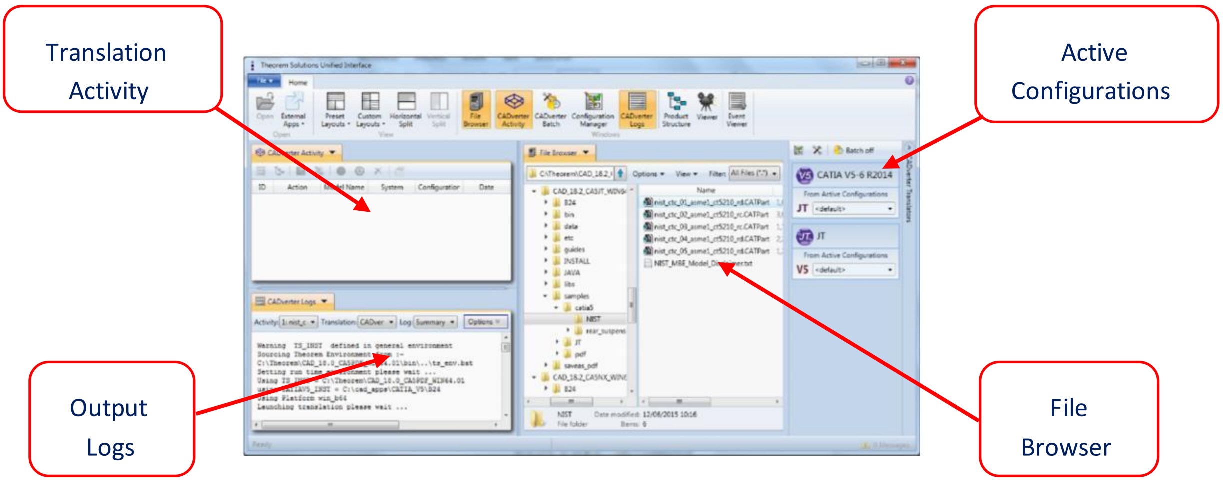
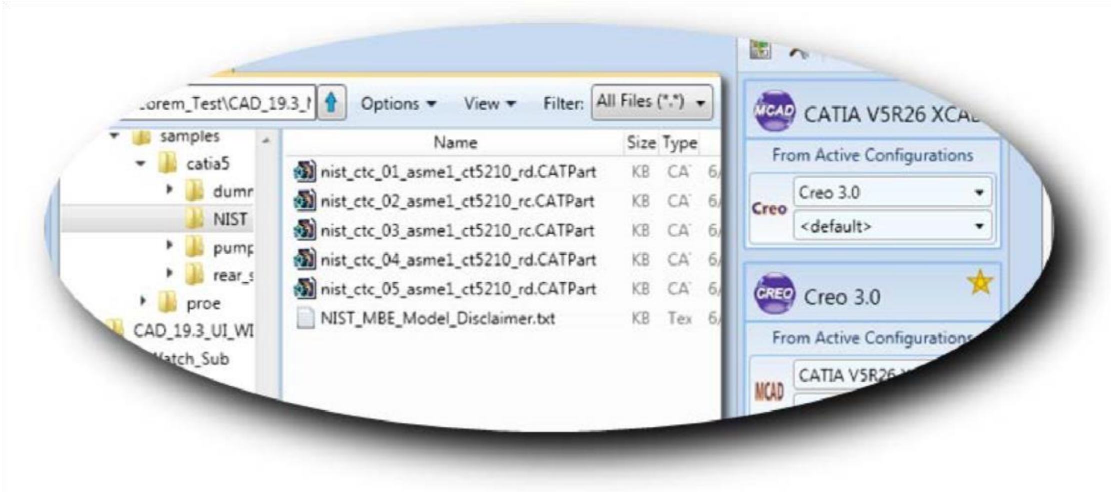


The simplest way to translate from CATIA V5 or CREO is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



On completion, the Unified Interface will display the activity information and details from the log file created during the translation, if requested, in the Translation Activity and Output Log panes, respectively.

The generated output data can be located by selecting the translation from the Activity pane

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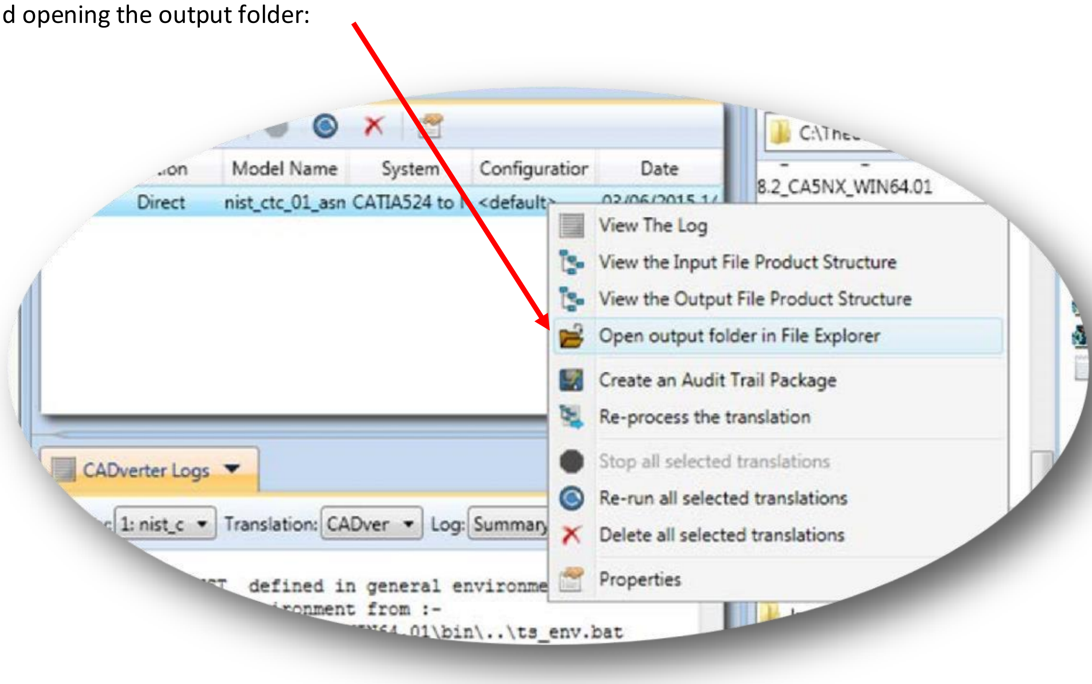
and opening the output folder:



Default Translation – via the Command Line

Running a translation via the command line can be carried out via the ***cad_run.cmd*** file located in the ***<installation_directory>\bin*** directory. The format of the command is as follows when translating from CATIA V5 to CREO:

<Translator_installation_directory>\bin\catia5r[XX]xcad_protkcr(X).cmd <input_file> <output_file>

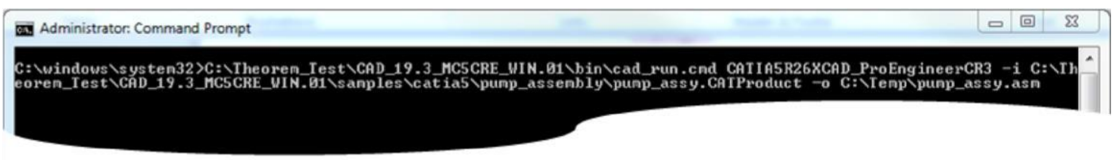
The format of the command is as follows when translating from CREO to CATIA V5:

<Translator_installation_directory>\bin\ protkcr(X)_catia5r[XX]xcad.cmd <input_file> <output_file>

(Notes!

Replace the [XX] with the version of CATIA V5 you are using. E.g. for CATIA V5R26, change to catia5r26xcad

Replace the (X) with the version of CREO. E.g. for CREO3**,** protkcr3):



The example above will translate a CATIA V5 sample file provided within the installation and produce the following screen output:

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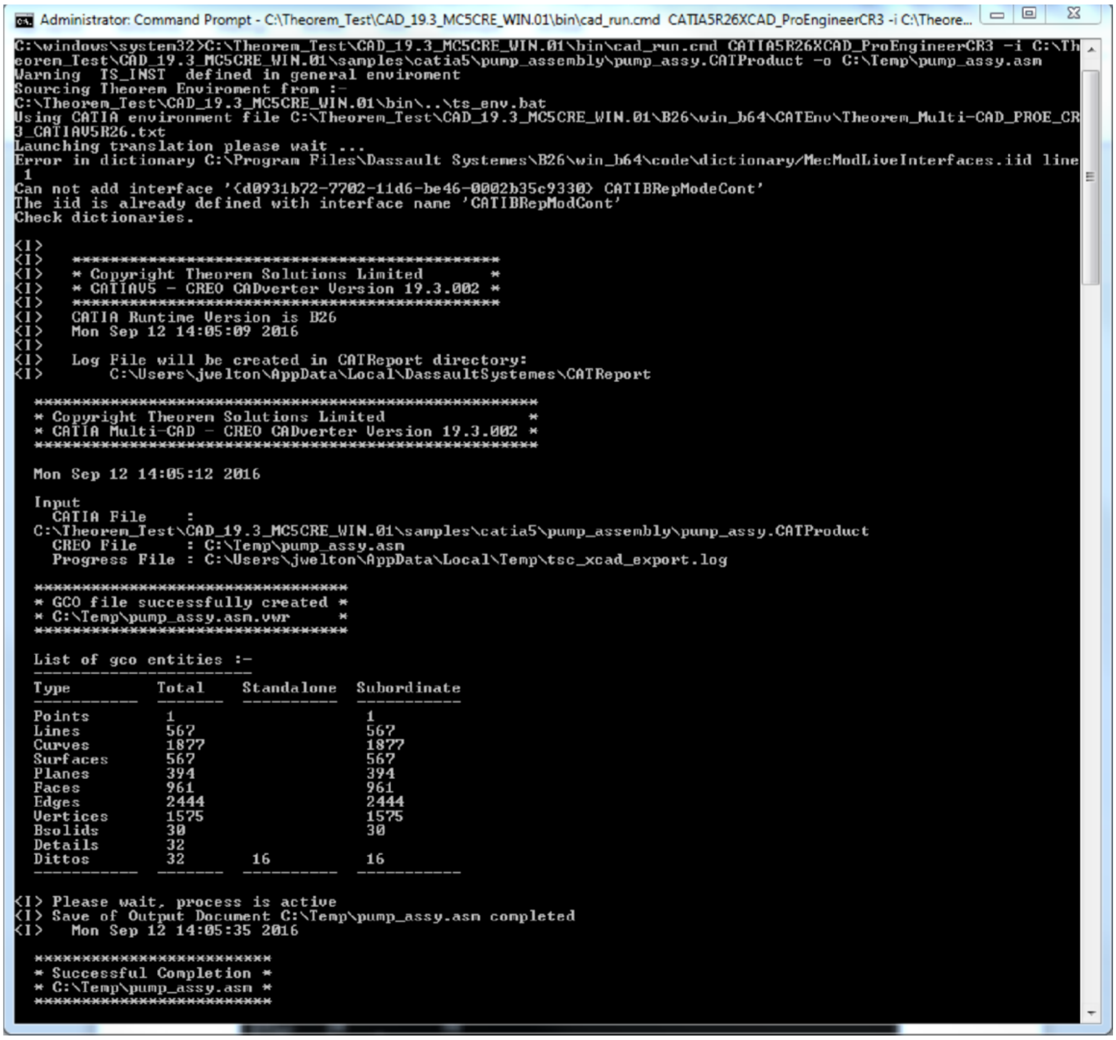

The file will be output to the target location. In this case:

C:\Temp\pump_assy.asm

Translator Customization

The Theorem translator 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.

Common Options for CATIA V5 to CREO

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CATIA V5 data into CREO are grouped into 3 areas:

- CATIA V5 Read – Those arguments that affect how data is read from CATIA V5
- CREO Write – Those arguments that affect how the data is written to CREO
- General – Those arguments that are common to ALL Publishing activities regardless of source data

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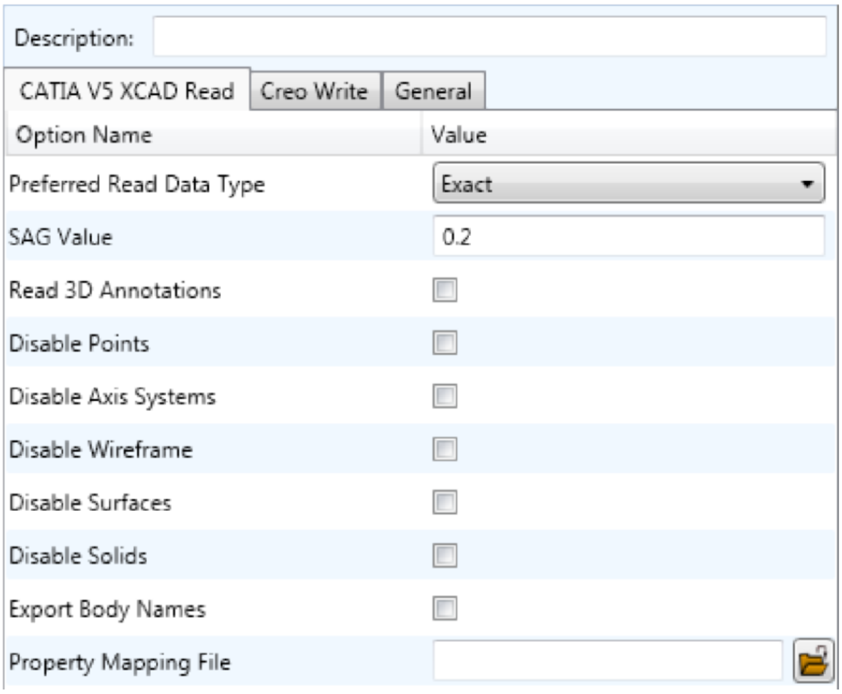
CATIA V5 Read Arguments

The image below shows the CATIA V5 Read arguments that are available, with their default settings:



Each of these options is described below:

Option	**Description**
Preferred Read Data Type	Allows the user to specify Exact or Tessellated read. *(Default is Exact)* • Command Line Syntax ○ *read_tess (to enable Tessellated)* ○ *Exact as is Default and has no argument*
Read 3D Annotations	Read 3D PMI. *(Default is Off)* • Command Line Syntax ○ *read_pmi*
Disable Points	Disable the processing of standalone Points. *(Default is Off)* • Command Line Syntax ○ *disable_points*
Disable Axis Systems	Disable the processing of Axis Systems. *(Default is Off)* • Command Line Syntax ○ *disable_axes*
Disable Wireframe	Disable the processing of standalone Wireframe. *(Default is Off)* • Command Line Syntax ○ *disable_wireframe*

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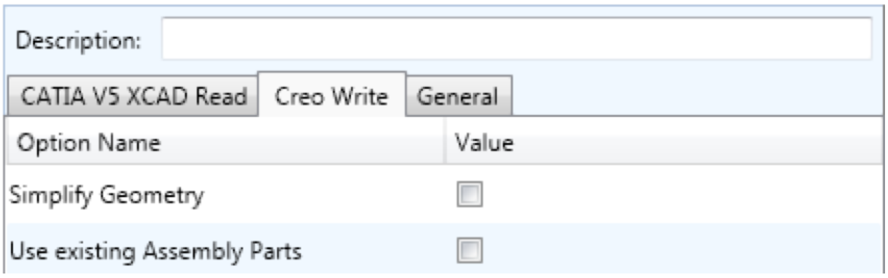
Disable Surfaces	Disable the processing of standalone Surfaces. *(Default is Off)* • Command Line Syntax ○ *disable_surfaces*
Disable Solids	Disable the processing of solids. *(Default is Off)* • Command Line Syntax ○ *disable_solids*
Export Body Names	Create Body Named Containers. *(Default is Off)* • Command Line Syntax ○ *body_names*
Property Mapping File	This is a file containing a list of CAD properties and information on how they are mapped to the CREO file. (See section Property Mapping Files) . • Command Line Syntax ○ *cad_prop_map_file <path\filename>*

CREO Write Arguments

The image below shows the Write CREO arguments that are available, with their default settings:



This option is described below:

Option	**Description**
Simplify Geometry	Attempt to write Analytical Geometry where possible (*Default is Off)* • Command Line Syntax ○ *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 ○ *use_parts*

CATIA V5 to CREO General Arguments

The image below shows the General arguments that are available, with their default settings:

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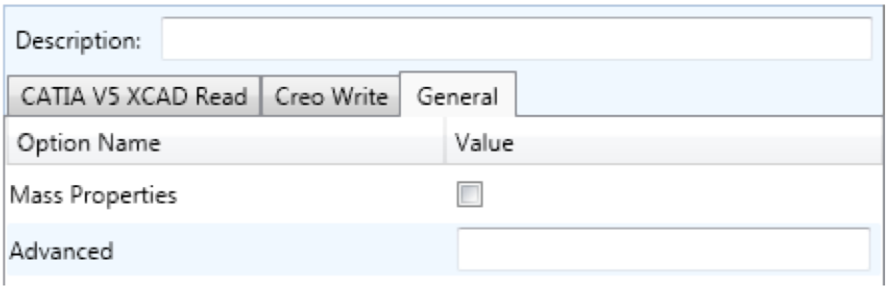


This option is described below:

Option	**Description**
Mass Properties	Allows Mass Property information to be calculated in the source and target data and writes this information in the log file. *(Default is Off)* • Command Line Syntax: ○ *mprops*
Advanced	Allows any of the Command Line Advanced arguments documented to be passed to the Unified Interface invocation.

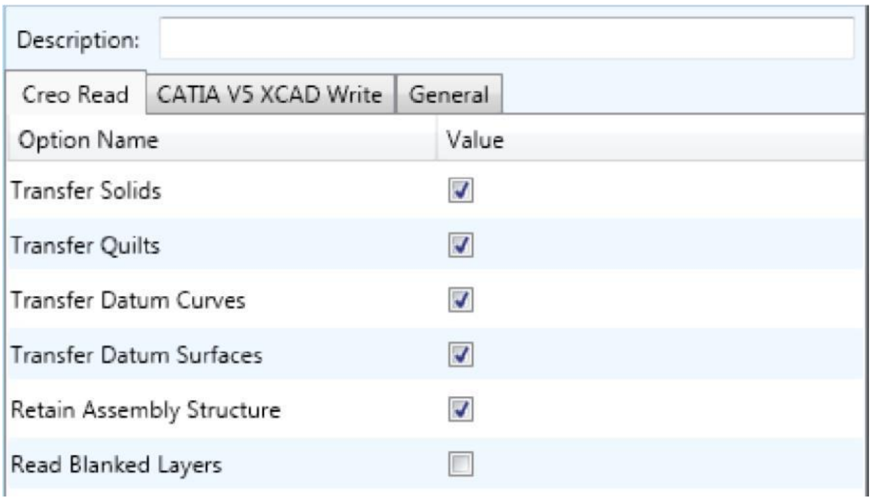
Common Options for CREO to CATIA V5

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CREO data into CATIA V5 are grouped into 3 areas:

- CREO Read – Those arguments that affect how data is read from CREO
- CATIA V5 Write – Those arguments that affect how the data is written to CREO
- General – Those arguments that are common to ALL Publishing activities regardless of source data

CREO Read Arguments

The image below shows the CREO Read arguments that are available, with their default settings:



Each of these options is described below.

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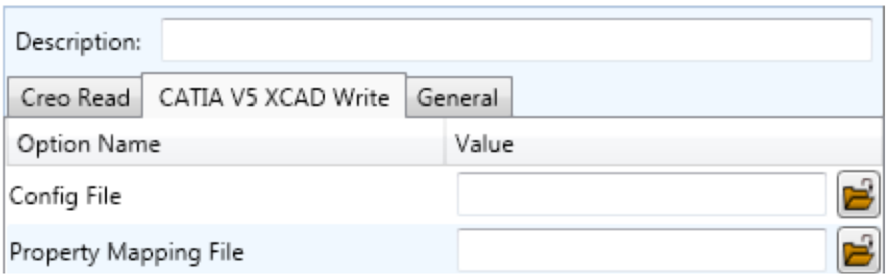
Option	**Description**
Transfer Solids	Solids will be converted *(Default is on)* • Command Line Syntax: ○ *no_solids (disable processing of Solids)*
Transfer Quilts	Quilts will be converted *(Default is on)* • Command Line Syntax: ○ *no_quilts (disable processing of Quilts)*
Transfer Datum Curves	Datum Curves (Wireframe) will be converted *(Default is on)* • Command Line Syntax: ○ *no_datum_curves (disable processing of Datum Curves)*
Transfer Datum Surfaces	Datum Surfaces will be converted *(Default is on)* • Command Line Syntax: ○ *no_datum_surfaces (disable processing of Datum Surfaces)*
Retain Assembly Structure	Maintain assembly structure in target file *(Default is on)* • *Command Line Syntax:* ○ *ditto (default – maintain structure)* ○ *noditto (no structure, single part created)*
Read Blanked Layers	Process data As Saved, or read data from All Layers *(Default is off or As Saved)* • *Command Line Syntax:* ○ Layer ALL (enables all layers)

CATIA V5 Write Arguments

The image below shows the CATIA V5 Write arguments that are available, with their default settings:



Each of these options is described below:

Option	**Description**
Config File	Specify Advanced Arguments in a Config File. • Command Line Syntax ○ *-c <path\filename>*

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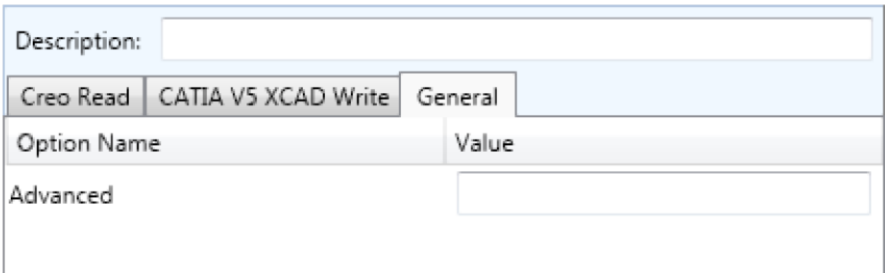
Property Mapping File	Specify a file which allows filtering of Detail user attributes. • Command Line Syntax ○ *-c <path\filename>*

CREO to CATIA V5 General Arguments

The image below shows the General arguments that are available, with their default settings:



The option is described below:

Option	**Description**
Advanced	Allows any of the Command Line Advanced arguments documented below to be passed to the Unified Interface invocation

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