User Guide
CATIA V5 <> JT

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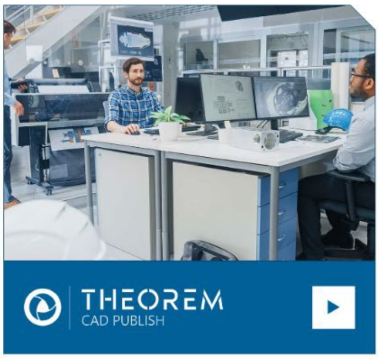
Overview of CADTranslate

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:

CAD**Translate**

CAD Data Exchange: Seamless and robust data translation between CAD and Visualization formats.

See our website for more details.

CAD**Publish**

Interactive Documentation: 3D PDF Publisher and Composer. Making CAD data accessible to non-CAD users.

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Theorem**XR**

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

Extended Reality – XR: Augmented, Mixed and Virtual Reality for the Engineering Metaverse.

See our website for more details.

CADTranslate for CATIA V5 - JT

The CATIA V5 to JT translator may be installed on a number of machines each accessing a central network-floating license.

The CATIA V5-JT Translator is a bi-directional direct database converter between the Dassault Systèmes CATIA V5 Modelling Application and the JT file format, used by the Siemens Teamcenter Visualization products.

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 using CATIA V5 who have selected JT to be their main method of collaboration and communication between OEMs and their customers or suppliers.

It is also a major method of visualization and therefore companies using JT based solutions need to translate their CATIA V5 data into the JT format.

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, from Theorem R18 the conversion process may be operated by using the new 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.
- 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.
- Data can be filtered by layer and entity type during processing. Geometry can be filtered and selectively processed.
- Uses the CATIA V5 API and Siemens JTOpen API to read and write data.
- In creating JT files a number of data types can be generated. A facetted representation, a JT Brep definition or an XT Brep definition. As standard a facetted representation is created with the user selecting whether JT or XT Brep definition is created. Converts all types of geometry, wire frame, surfaces, trimmed surfaces (faces) and solid models.

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 Visualize 3D for CATIA V5 – JT product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

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 and Tutorials.

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

Using the Product

Default Translations

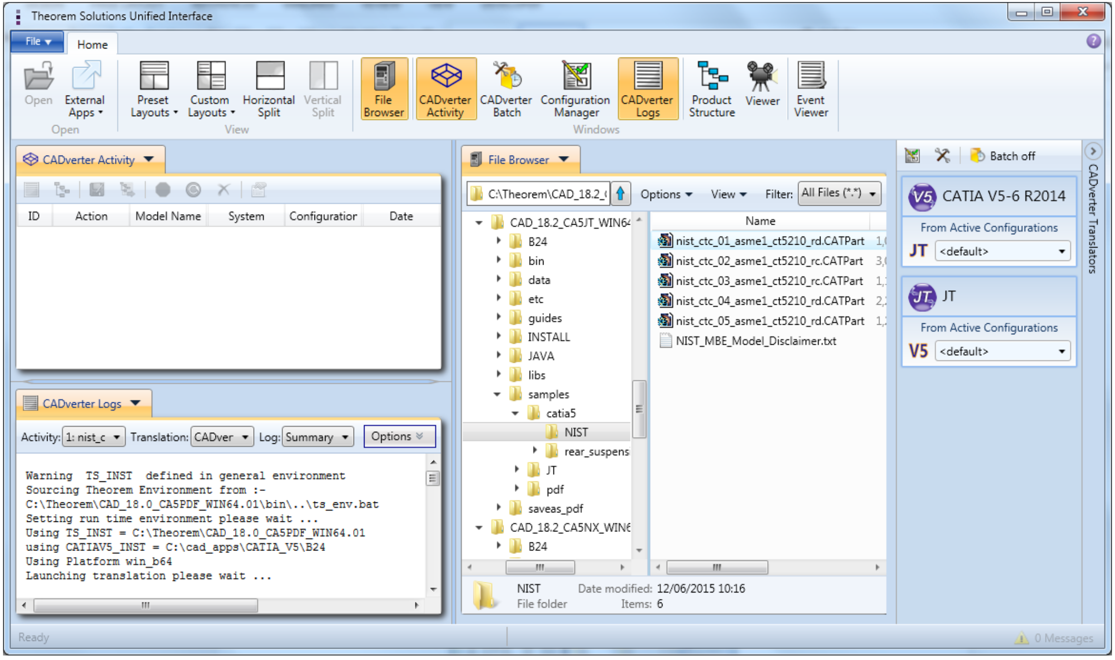
Default Translation – via the Unified Interface

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

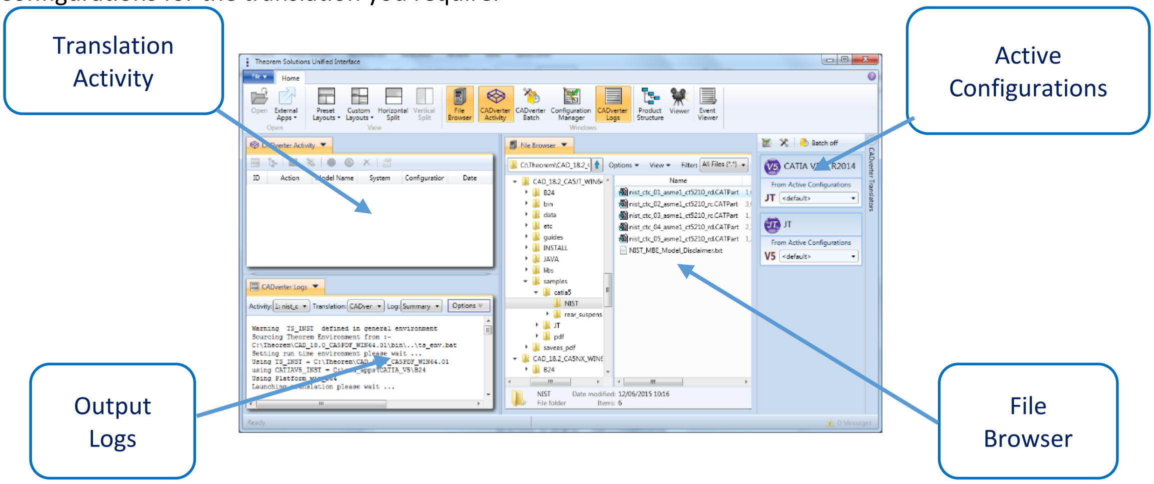
The following interface will be launched:

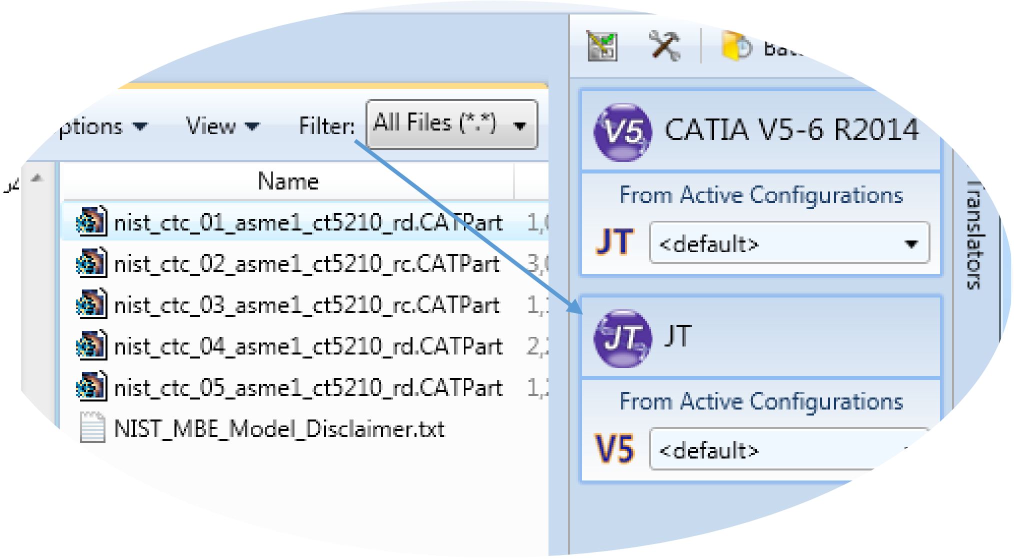


```
Warning  TS_INST  defined in general environment
Sourcing Theorem Environment from :-
C:\Theorem\CAD_18.0_CA5PDF_WIN64.01\bin\..\ts_env.bat
Setting run time environment please wait ...
Using TS_INST = C:\Theorem\CAD_18.0_CA5PDF_WIN64.01
using CATIAV5_INST = C:\cad_apps\CATIA_V5\B24
Using Platform win_b64
Launching translation please wait ...
```

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

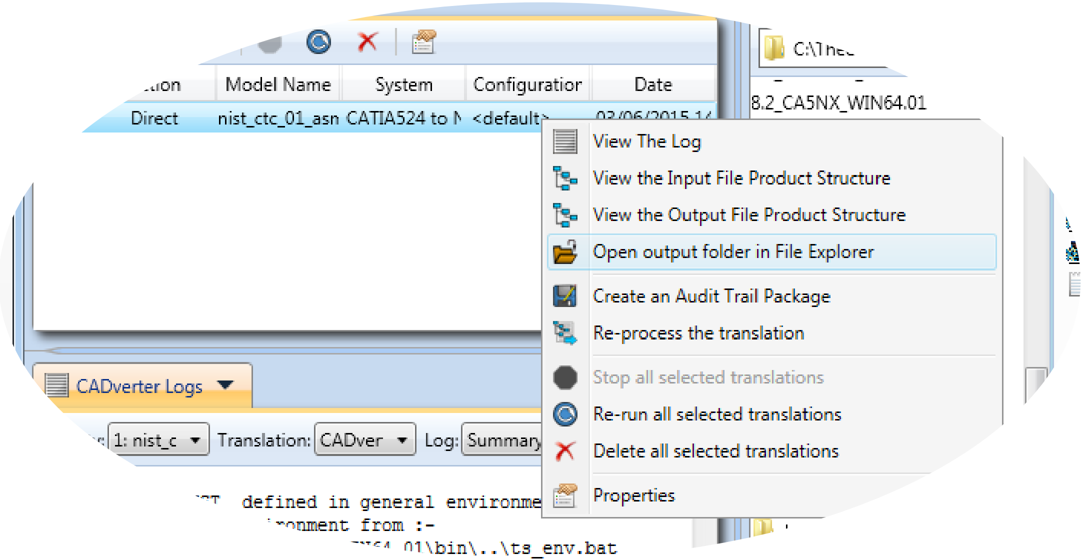
The simplest way to translate from CATIA V5 or JT is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.




ptions
View
Filter: All Files (*.*)
Name
nist_ctc_01_asme1_ct5210_rd.CATPart
nist_ctc_02_asme1_ct5210_rc.CATPart
nist_ctc_03_asme1_ct5210_rc.CATPart
nist_ctc_04_asme1_ct5210_rd.CATPart
nist_ctc_05_asme1_ct5210_rd.CATPart
NIST_MBE_Model_Disclaimer.txt
CATIA V5-6 R2014
From Active Configurations
JT <default>
JT
From Active Configurations
V5 <default>
Translators

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 and opening the output folder:


Model Name
System
Configuration
Date
Direct
nist_ctc_01_asn
CATIA524 to N
<default>
C:\The
8.2_CA5NX_WIN64.01
View The Log
View the Input File Product Structure
View the Output File Product Structure
Open output folder in File Explorer
Create an Audit Trail Package
Re-process the translation
Stop all selected translations
Re-run all selected translations
Delete all selected translations
Properties
CADverter Logs
1: nist_c
Translation: CADver
Log: Summary
defined in general environme
ronment from :-
01\bin\..\ts_env.bat

Default Translation – via the Command Line

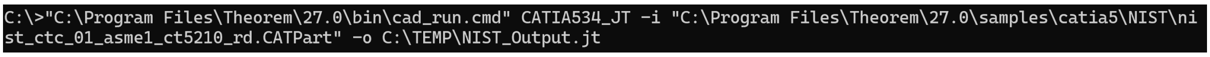
Running a translation via the command line can be carried out via the ***cad_run.cmd*** file located in the ***<installation_directory>\bin*** directory. The format of the command is as follows when translating from CATIA V5 to JT:

<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_JT –i <input_file> -o <output_file>

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

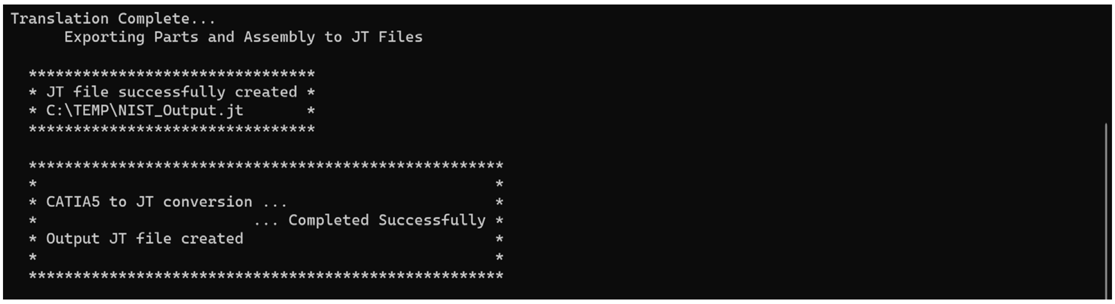
<Translator_installation_directory>\bin\cad_run.cmd JT_CATIA5[XX] –i <input_file> -o <output_file>

(Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for CATIA V5 R34 (V5-6 R2024), change to CATIA534):



```
C:\>"C:\Program Files\Theorem\27.0\bin\cad_run.cmd" CATIA534_JT -i "C:\Program Files\Theorem\27.0\samples\catia5\NIST\nist_ctc_01_asme1_ct5210_rd.CATPart" -o C:\TEMP\NIST_Output.jt
```

The example above will translate a CATIA V5 sample file provided within the installation and finish with the following screen output when successful:



```
Translation Complete...
      Exporting Parts and Assembly to JT Files

  ********************************
  * JT file successfully created *
  * C:\TEMP\NIST_Output.jt       *
  ********************************

  *****************************************************
  *                                                   *
  * CATIA5 to JT conversion ...                       *
  *                        ... Completed Successfully *
  * Output JT file created                            *
  *                                                   *
  *****************************************************
```

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

C:\Temp\NIST_Output.jt

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 JT

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing CATIA V5 data into JT are grouped into 4 areas:

CATIA V5 Read – Those arguments that affect how data is read from CATIA V5

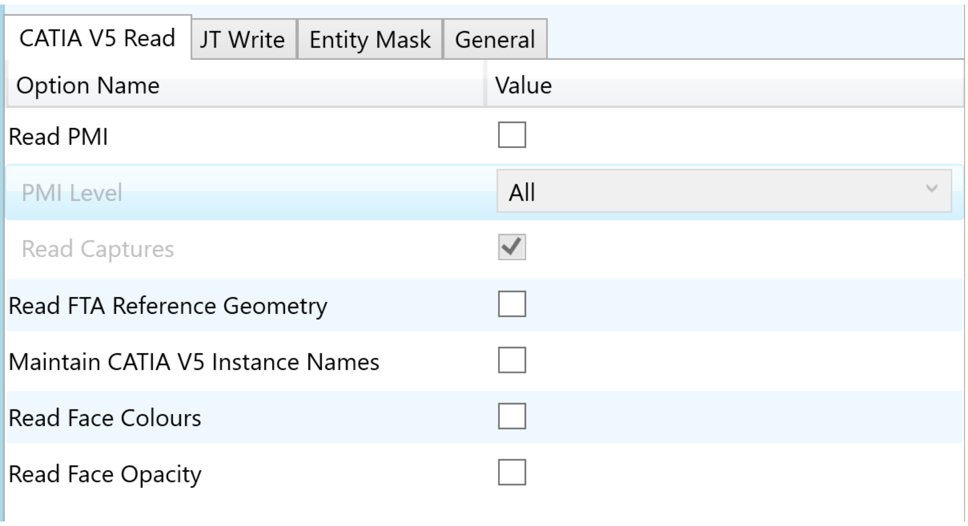
JT Write – Those arguments that affect how the data is written to JT

Entity Mask – Those arguments that allow specific read entities to be masked

General – Those arguments that are common to ALL Publishing activities regardless of source data.

CATIA V5 Read Arguments

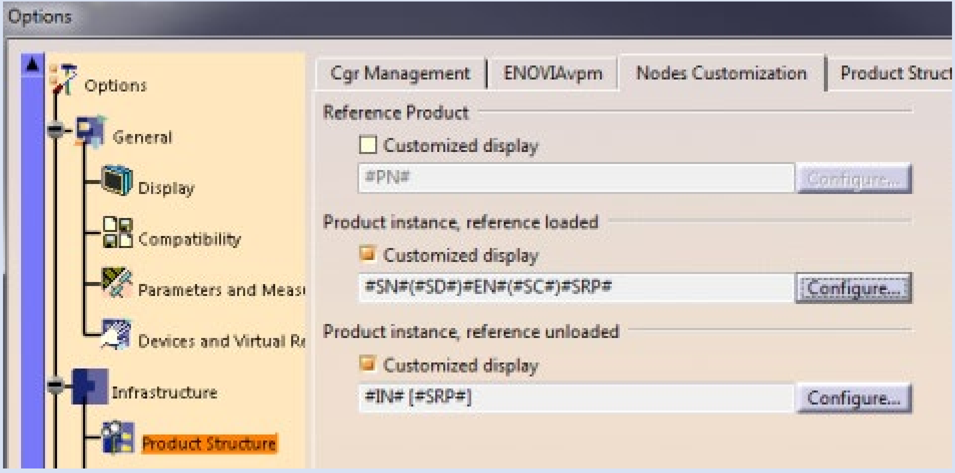
The image below shows the CATIA V5 Read arguments that are available, with their default settings:



Option Name	Value
Read PMI	☐
PMI Level	All
Read Captures	☑
Read FTA Reference Geometry	☐
Maintain CATIA V5 Instance Names	☐
Read Face Colours	☐
Read Face Opacity	☐

Each of these options is described below:

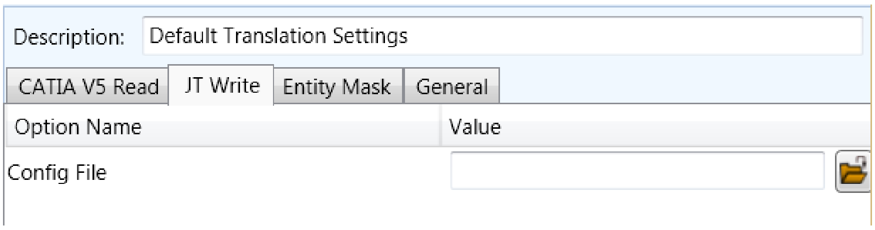
Option	Description
Read PMI	Enables PMI data read from the V5 file. *(Default is ON).* • Command Line Syntax: ○ dont_read_pmi – to turn off

	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 *'dont_fill_pmi_text'*
PMI Level	A secondary argument to 'Read PMI' and allows control of the level of PMI to be read. Default is ALL when 'Read PMI' is marked as ON. • Options Available (command line syntax in italics and square brackets next to the option) ○ All - [read_pmi] ○ Part Level - [read_part_pmi] ○ Assembly Level - [read_assy_pmi] ○ Assembly Set (From CATPart) - [read_part_assy_pmi] ○ Assembly Set (All) - [read_all_assy_pmi]
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 ○ dont_read_captures – to turn off
Read FTA Reference Geometry	Enables reading of FTA Reference Geometry *(Default is Off)* • Command Line Syntax: ○ read_geometry – to turn on
Maintain CATIA V5 Instance Names	Honours CATIA V5 *Tools->Options->Infrastructure->Product Structure->Nodes Customization* panel settings *(Default is Off)*  Options Options General Display Compatibility Parameters and Meas Devices and Virtual R Infrastructure Product Structure Cgr Management ENOVIAvpm Nodes Customization Product Struct Reference Product □ Customized display #PN# Configure... Product instance, reference loaded ☑ Customized display #SN#(#SD#)#EN#(#SC#)#SRP# Configure... Product instance, reference unloaded ☑ Customized display #IN# [#SRP#] Configure... • Command Line Syntax: ○ ditto_naming V5 – to turn on
Read Face Colours	Read the face colours (Default is OFF) • Command Line Syntax: ○ read_face_colours – to turn on
Read Face Opactity	Read the opacity of the face (Default is OFF)

	- Command Line Syntax: - read_materials – to turn on

JT Write Arguments

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



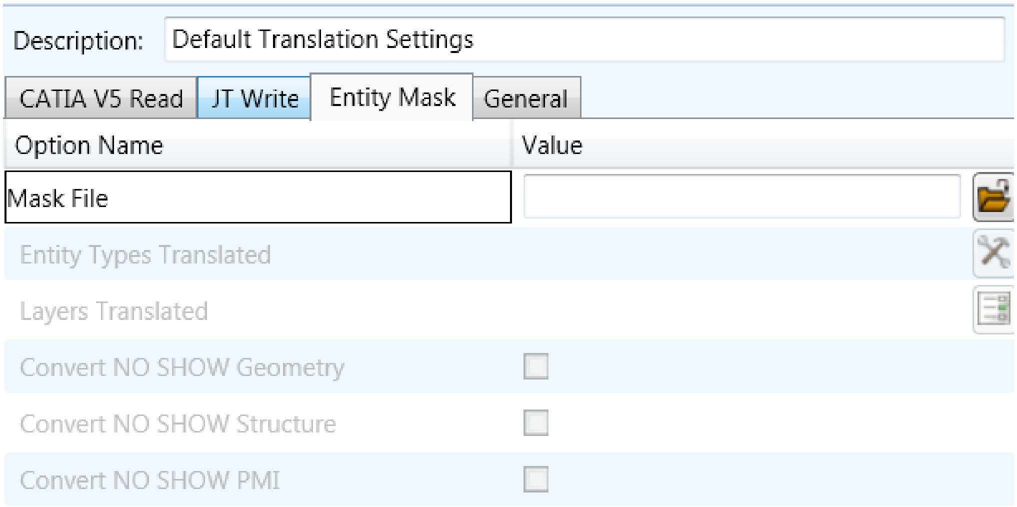
Description: Default Translation Settings

Each of these options is described below:

Option	Description
Config File	Allows a JT configuration file to be specified. Please see Appendix B for a full description of the JT config file format. - Command Line Syntax - -z [path_to_file]

CATIA V5 to JT Entity Masking Arguments

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



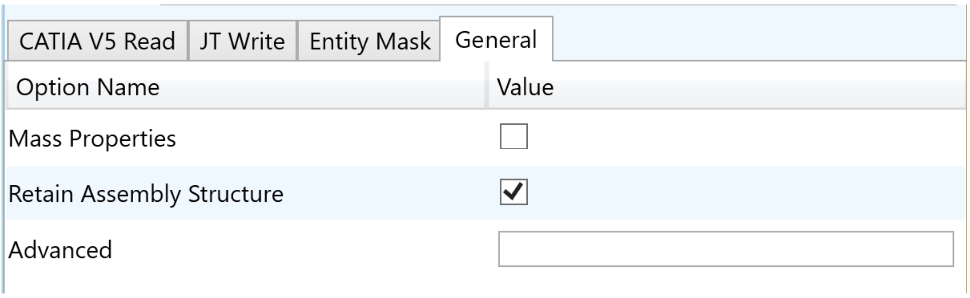
Description: Default Translation Settings

Each of these options is described below:

Option	Description

Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: • Command Line Syntax: ○ Mask <filename>
Entity Types Translated	Specifies a selection list from which to select which entity types are to be processed. The following types are available: "POI","CUR",”SKI”,"SOL",”ISO”,"TEX","AXI" • Command Line Syntax: ○ Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g. ON POI to ensure they are considered in the translation
Layers Translated	Specifies a selection list from which to select which layers are to be processed. • Command Line Syntax: ○ A single entry of ON ALL LAY must precede any Layer Mask command. ○ Add a list or range of numbers representing layer to be processed to the specified mask file to ensure they are NOT considered in the translation e.g. OFF LAY 114,149,166,167,168
Convert NO SHOW Geometry	Enables Hidden geometry to be processed *(Default is Off)* • Command Line Syntax: ○ Add the following entry to the Mask file: ▪ **ON NOSHOW**
Convert NO SHOW Structure	Enables Hidden Assembly Structure to be processed *(Default is Off)* • Command Line Syntax: ○ Add the following entry to the Mask file: ▪ **ON NOSHOW STR**
Convert NO SHOW PMI	Enables Hidden PMI to be processed *(Default = Off)* • Command Line Syntax: ○ Add the following entry to the Mask file: ▪ **ON NOSHOW PMI**

CATIA V5 to JT General Arguments



Option Name	Value
Mass Properties	□
Retain Assembly Structure	☑
Advanced	

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

Each of these options is described below:

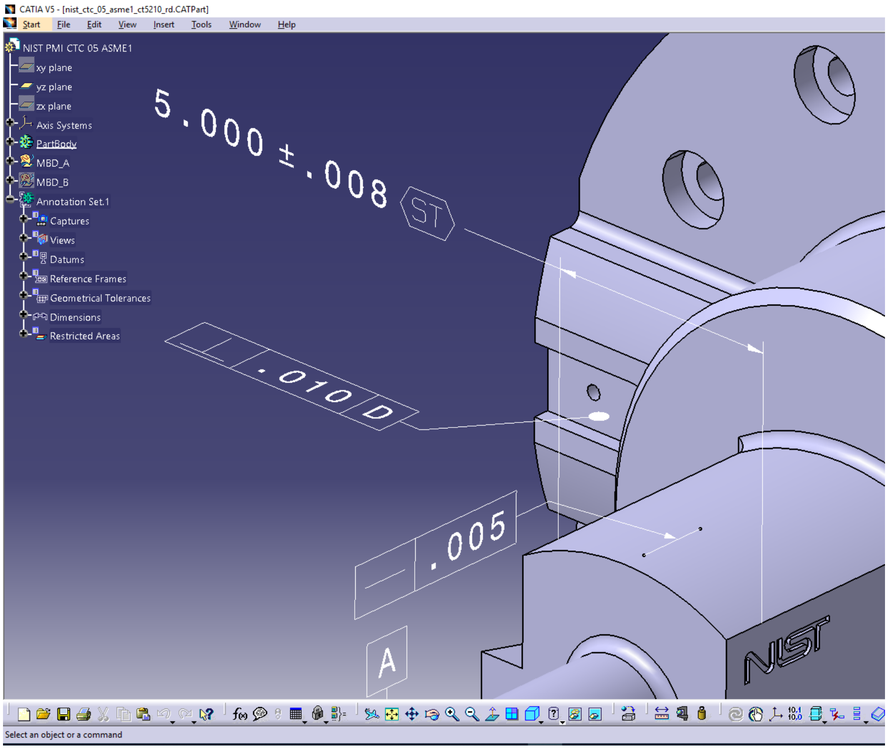
Option	Description
Mass Properties	Allows Mass Property information to be read from the source data and written as attributes *(Default is OFF)* • Command Line Syntax: ○ mprops
Retain Assembly Structure	Maintain the assembly structure during translation. *(Default is ON)* • Command Line Syntax: ○ offditto – to turn off
Advanced	Allows any of the Command Line Advanced arguments documented to be passed to the Unified Interface invocation.

Processing CATIA V5 FTA data to JT PMI

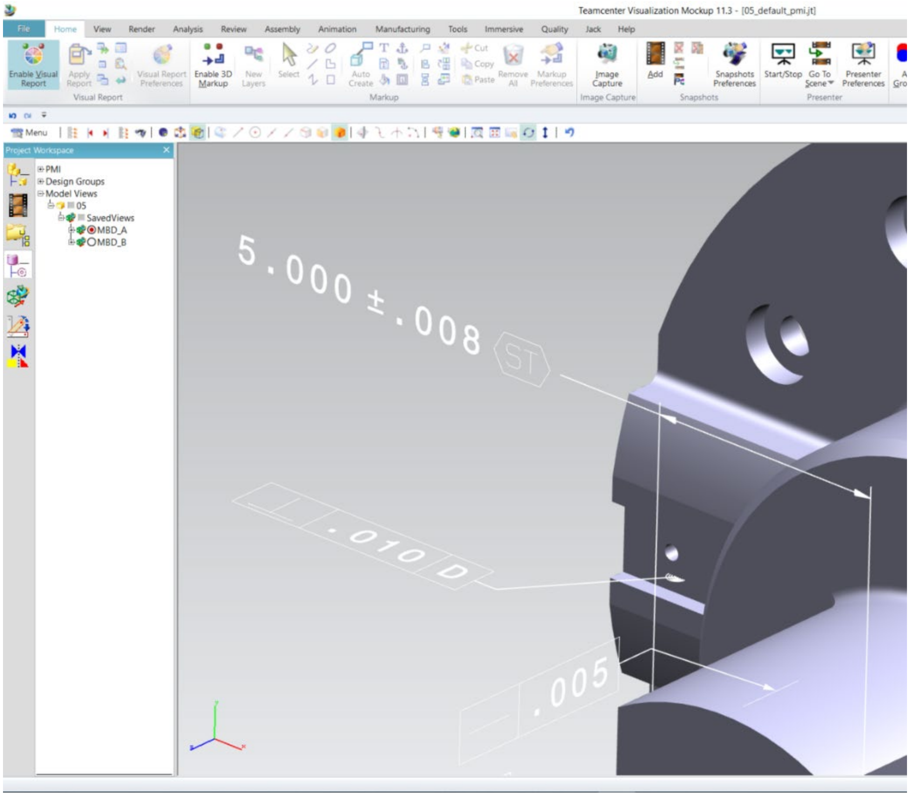
When using the optional CATIA V5 to JT PMI add-on module all CATIA V5 FTA data can be translated to JT PMI output. This includes the translation of 3D dimensions and annotations which are mapped to the equivalent JT PMI entities. In addition, CATIA V5 3D sections are also able to be translated. The CATIA V5 Capture definitions, which manage the visibility of selective FTA elements as well as the view zoom and orientation, are directly mapped to JT Model Views.

Options for Processing FTA Data with Filled Text

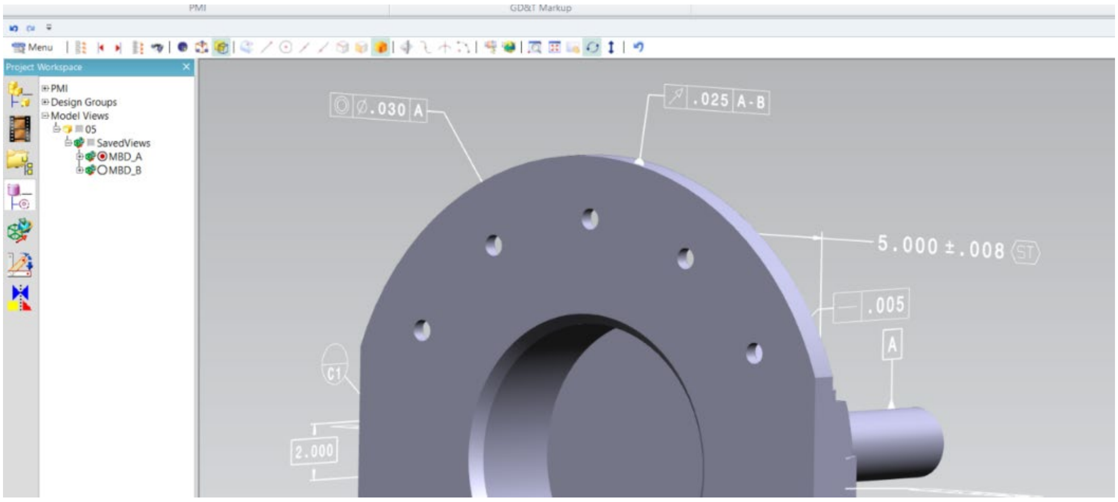
When creating 3D dimensions and annotations with the CATIA V5 FTA module the user can create the text using either stick or filled text fonts. By default, the processing of CATIA V5 filled fonts will recreate the JT output using JT text glyphs. One benefit of using JT glyph-based text definitions is that when the model is reoriented, such that the initial view of the text is reversed, the user can select the annotations to be automatically reoriented. If the filled font representation was created with a simple polygon fill only the character outline would be reoriented and the polygon fill would remain in its original location making the output unreadable.



Picture showing typical CATIA V5 3D Dimensions and Annotations. Note, in the upper dimension the dimensions values 5.000 +- .008 are created using a filled text font. The characters “ST” in the frame are created using an unfilled stick font.

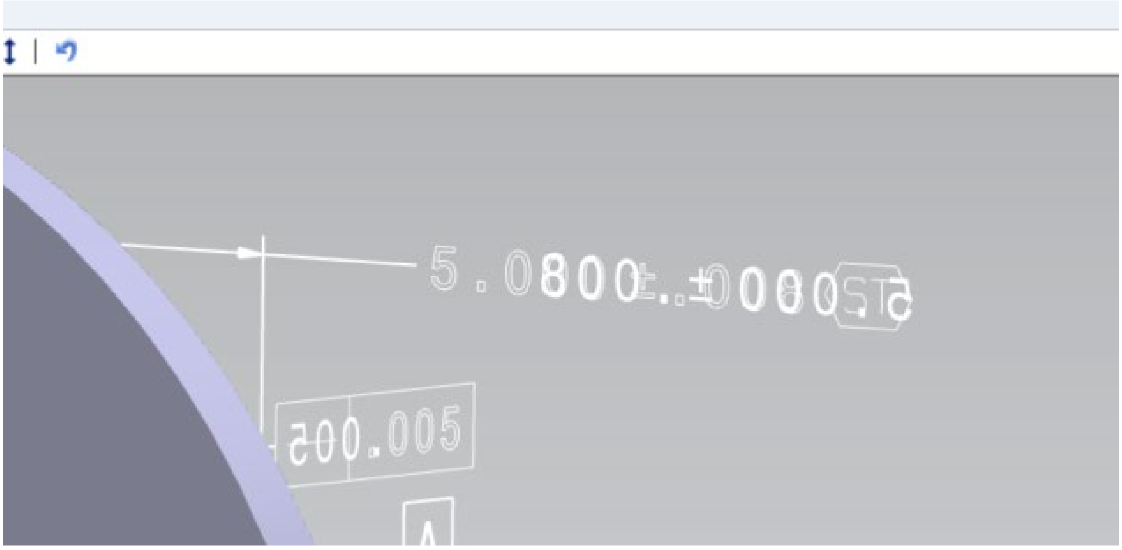

Teamcenter Visualization Mockup 11.3 - [05_default_pmi.jt]
File Home View Render Analysis Review Assembly Animation Manufacturing Tools Immersive Quality Jack Help
Project Workspace
PMI
Design Groups
Model Views
05
SavedViews
MBD_A
MBD_B
5.000 ± .008
ST
.010 D
.005

Picture showing default JT output of V5 FTA data mapped to JT PMI representation



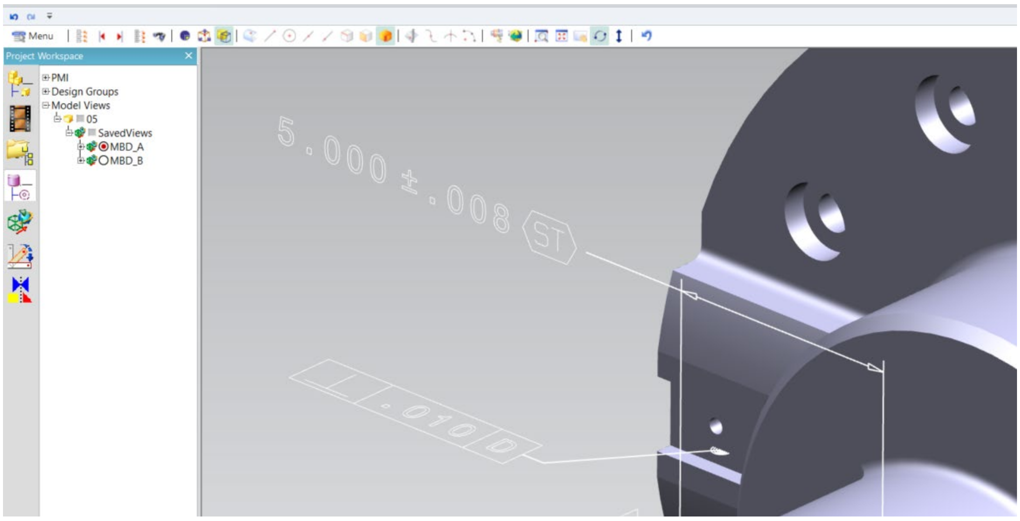
Picture showing the reorientation of the model with the text also reoriented. Note the text characters have been rotated 180 degrees and are now readable with filled character representation.

To recreate the JT filled text using simple polygons, rather than JT text glyph representations, use the options **pmi_glyphs_off pmi_polygons** together. The output representation will appear filled and look correct when viewed in the authored orientation. However, filled text created with simple polygons, will not display correctly if the data is reoriented and the text switching in JT is selected. The picture below shows an example of the display when the data is rotated, and the text is reoriented. As you will see the simple polygon fill is still in its original orientation making the dimension unreadable.



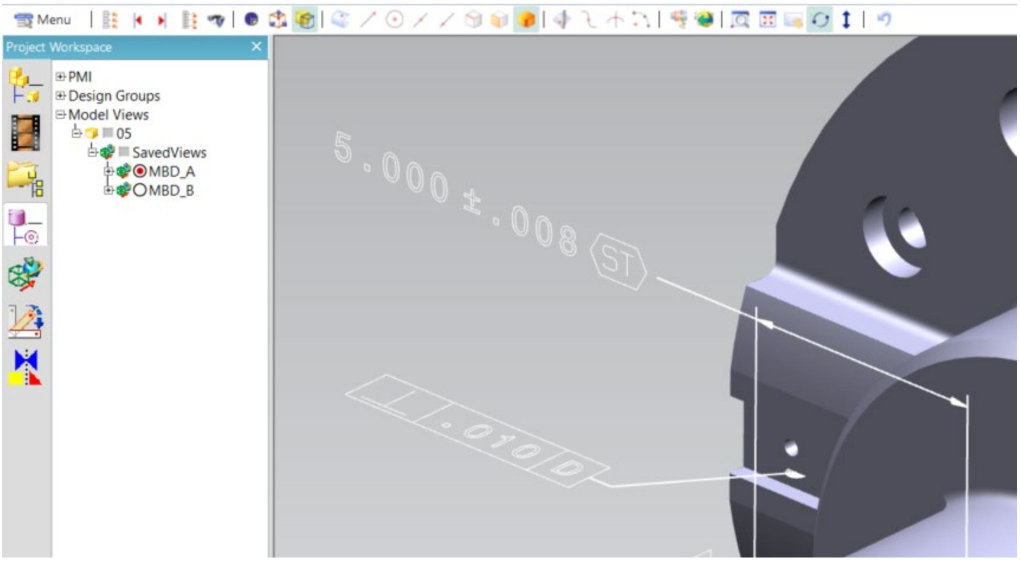
Options for Processing FTA Data using Outline Text

When processing V5 FTA text data created with filled font definition the user can select to generate the JT text representation as outline text, therefore removing the internal rendering. This is achieved by using the option **pmi_glyphs_off**.



Picture showing JT PMI with filled text rendered in outline only. Note the option pmi_glyphs_off also removes the filled arrowhead rendering.

If you would prefer to maintain filled arrowheads and only impact the characters to be recreated in outline mode, then use the option **dont_fill_pmi_text** instead of the option **pmi_glyphs_off**. The resultant output is shown in the picture below with the text drawn in outline and the arrowheads filled.



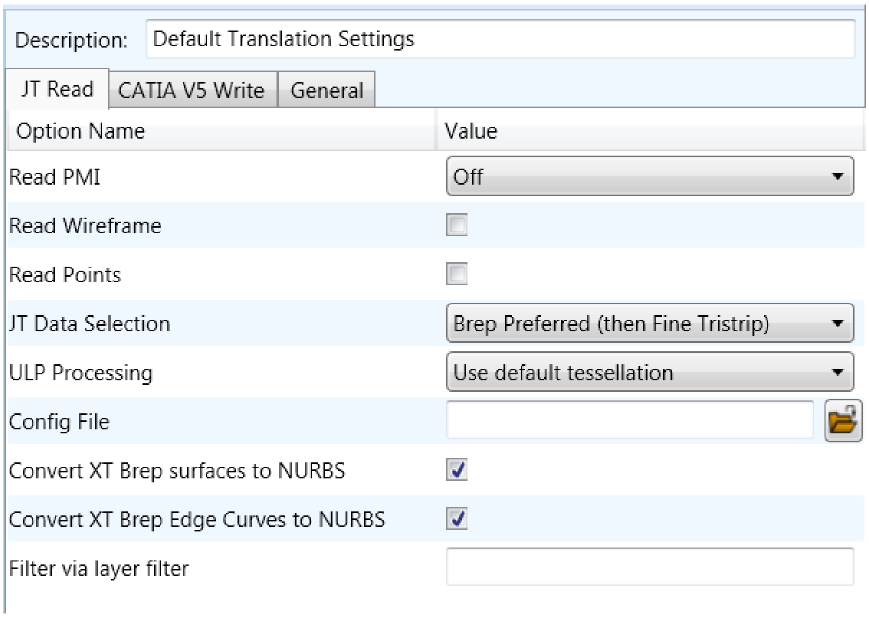
Common Options for JT to CATIA V5

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing JT data into CATIA V5 are grouped into 4 areas:

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

JT Read Arguments

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



Description: Default Translation Settings

Option Name	Value
Read PMI	Off
Read Wireframe	□
Read Points	□
JT Data Selection	Brep Preferred (then Fine Tristrip)
ULP Processing	Use default tessellation
Config File	
Convert XT Brep surfaces to NURBS	☑
Convert XT Brep Edge Curves to NURBS	☑
Filter via layer filter	

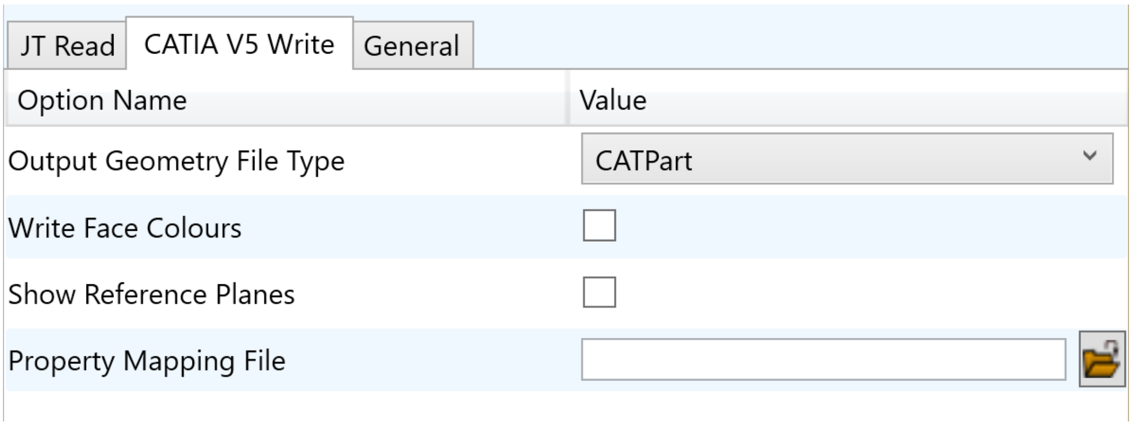
Each of these options is described below.

Option	Description
Read PMI	Reads 3D PMI. Default is OFF. • Command Line Syntax ○ read_pmi dim2_pmi
Read Wireframe	Read JT wireframe data. Default is OFF. • Command Line Syntax ○ read_wire_frame
Read Points	Read JT point data. Default is OFF. • Command Line Syntax ○ read_points
JT Data Selection	Select Brep or tessellated data read. Default is 'Brep Preferred (then fine facet)'.

	• Command Line Syntax ○ Brep Preferred (then Fine Facet): brep_pref ○ Brep preferred (then Fine Tristrip): brep_pref_tri ○ Brep Only: brep_only ○ Fine Tristrip: fine_tristrips ○ Coarse Tristrip: coarse_tristrips ○ Fine Facet: fine_facets ○ Coarse Facet: coarse_facets
ULP Processing	How to process ULP data in the JT part. Default is 'Use Default Tessellation'. • Command Line Syntax ○ Use Default Tessellation: Default ○ Tessellate ULP: tess_ulp
Config File	Config File for Brep or ULP tessellation. • Command Line Syntax ▪ -z [path to file]
Convert XT Brep surfaces to NURBS	Read XT Brep surfaces as NURBS surfaces (else read in native form). Default is ON. • Command Line Syntax ○ noprep – to turn off
Convert XT Brep Edge Curves to NURBS	Read XT Brep edge curves as NURBS curves (else read in native form. Default is ON. • Command Line Syntax ○ rd_native_edge – to turn off
Filter via layer filter	Supply layer filter(s) separated by commas and double quoted. Default is OFF. • Command Line Syntax ○ layer_filter

CATIA V5 Write Arguments

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

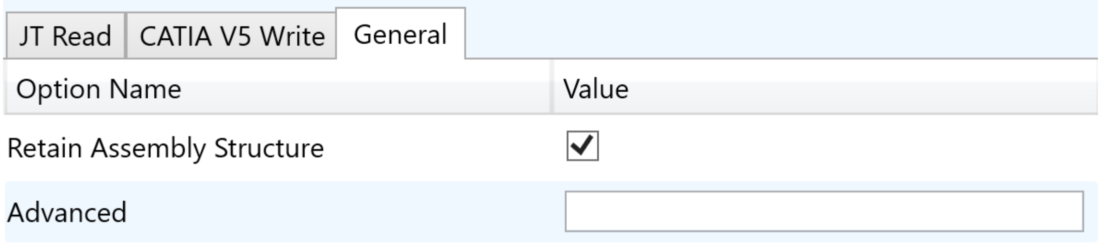


Option Name	Value
Output Geometry File Type	CATPart
Write Face Colours	□
Show Reference Planes	□
Property Mapping File	

Each of these options is described below:

Option	Description
Output Geometry File Type	Output Geometry file type. Default is CATPart. • Command Line Syntax ○ CATPart: output_type CATPart ○ Model: output_type model ○ Cgr: output_type cgr ○ Igs: output_type igs ○ CATShape: output_type CATShape ○ Tessellated: create_cgr
Write Face Colours	Writes face colours. Default is ON. • Command Line Syntax ○ FACE_COLOUR ○ SOLID_COLOUR – to turn off
Show Reference Planes	Creates reference planes. Default is OFF. • Command Line Syntax ○ Show: dont_blank_planes ○ No Show: Default
Property Mapping File	Path to Property Mapping File • Command Line Syntax ○ cad_prop_map_file <file>

JT to CATIA V5 General Arguments



Option Name	Value
Retain Assembly Structure	☑
Advanced	

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

The option is described below:

Option	Description
Retain Assembly Structure	Maintain the assembly structure during translation. *(Default is ON)* • Command Line Syntax: ○ offditto – to turn off
Advanced	Allows any of the Command Line Advanced arguments documented below to be passed to the Unified Interface invocation

Command Line Advanced Arguments

Advanced arguments can be added to the Command Line or the UI General->Advanced field.

CATIA V5 Advanced Arguments

Argument	Description
convert_curves	Converts curves to NURBS form
convert_surfaces	Converts surfaces to NURBS form
dont_fill_pmi_arrows	Disables the read of filled arrow information
dont_fill_pmi_text	Disables the read of filled text information
dont_read_captures	Disables the read of PMI Capture information
face_opacity	Read face opacity
no_face_colour	Sets the default to SOLID colours
noshow noshow_geom noshow_struct noshow_pmi	Reads hidden geometry / structure / pmi
output_mbd	Allows sub-part specification tree information to be read and presented to JT as product structure information.
read_geometry_edges	CATIA V5 allows the display of shaded surfaces and edges. This option allows the translator to mimic this for FTA construction geometry by promoting the edge curves to standalone wireframe.
cad_prop_map_file <file>	Creates a cad property mapping file containing all attributes if one doesn’t exist.
off_ditto	Removes all assembly structure and collapses ALL geometry into a single selectable object

JT Advanced Arguments

Argument	Description
read_wire_frame	Enables reading of Wire Frame entities
read_ref_planes	Reads reference planes

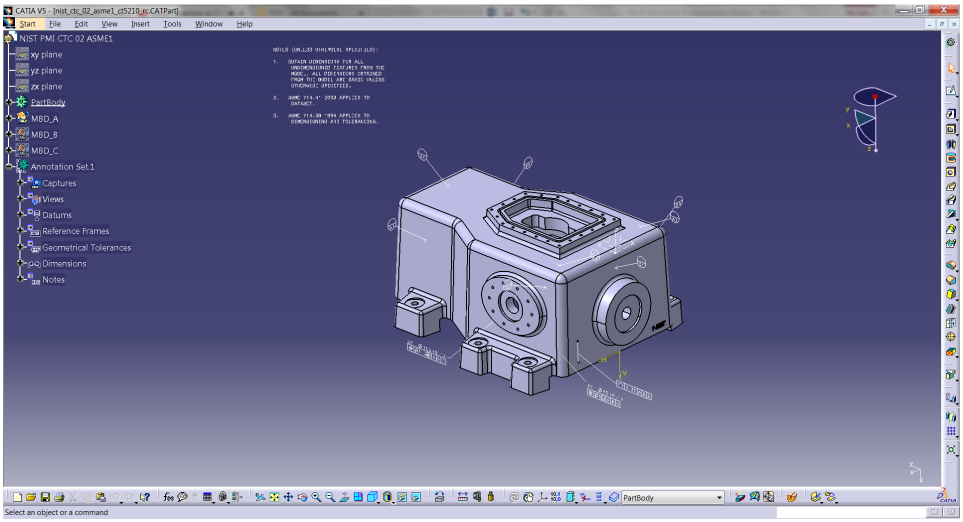
Translating Interactively from within CATIA V5

The CATIA V5 to JT translator allows an active CATIA V5 Part or Assembly to be translated directly into JT or a JT Part or assembly to be imported, directly from the CATIA V5 application.
In order to translate from within CATIA V5, the CATIA V5 application must be started from within a Theorem environment, so that the appropriate CATIA V5 menus are loaded.

CATIA V5 can be started from a shortcut, if requested at installation time. Alternatively, it can be started via the script provided in the Translator installation at:

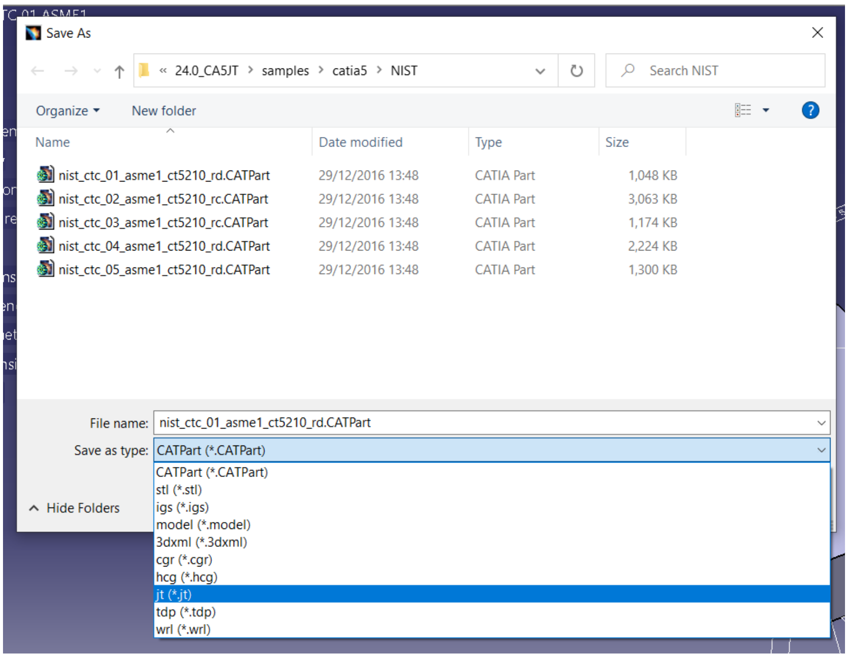
<installation_directory>\bin\catia5r[version]_start.cmd

(where [version] should be substituted for the version of CATIA V5 that you have installed – e.g. 28, 29, 30, 31):



Save As JT

Once CATIA V5 has been started it is possible for a model to be saved to JT. To do this, the user selects the *File -> Save As* Menu Option. From the file browser window that opens, change the *'Save as type'* option to be **jt (*.jt)**.



Save As

« 24.0_CA5JT › samples › catia5 › NIST

Search NIST

Organize ▾ New folder

Name	Date modified	Type	Size
nist_ctc_01_asme1_ct5210_rd.CATPart	29/12/2016 13:48	CATIA Part	1,048 KB
nist_ctc_02_asme1_ct5210_rc.CATPart	29/12/2016 13:48	CATIA Part	3,063 KB
nist_ctc_03_asme1_ct5210_rc.CATPart	29/12/2016 13:48	CATIA Part	1,174 KB
nist_ctc_04_asme1_ct5210_rd.CATPart	29/12/2016 13:48	CATIA Part	2,224 KB
nist_ctc_05_asme1_ct5210_rd.CATPart	29/12/2016 13:48	CATIA Part	1,300 KB

File name: nist_ctc_01_asme1_ct5210_rd.CATPart

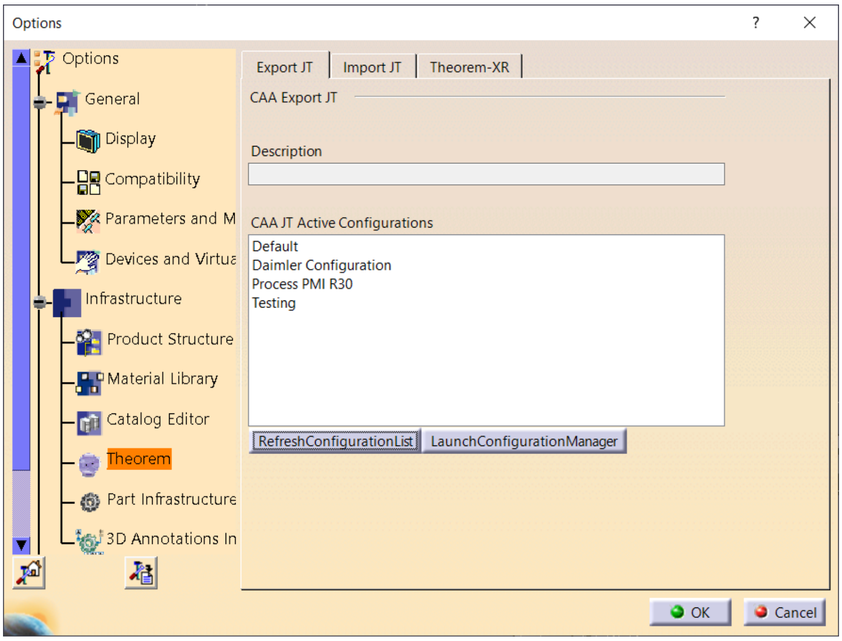
Save as type: CATPart (*.CATPart)

- CATPart (*.CATPart)
- stl (*.stl)
- igs (*.igs)
- model (*.model)
- 3dxml (*.3dxml)
- cgr (*.cgr)
- hcg (*.hcg)
- jt (*.jt)
- tdp (*.tdp)
- wrl (*.wrl)

Hide Folders

Once selected, press the ‘Save’ button to create the JT data.

To add a user defined configuration to the process (created from the Theorem UI), this can be selected from the **Tools>Options>Infrastructure>Theorem>CAA Export JT** menu. The process will then use this defined configuration to create the JT data as required.



Options

- Options
  - General
    - Display
    - Compatibility
    - Parameters and M
    - Devices and Virtu
  - Infrastructure
    - Product Structure
    - Material Library
    - Catalog Editor
    - Theorem
    - Part Infrastructure
    - 3D Annotations In

Export JT | Import JT | Theorem-XR

CAA Export JT

Description

CAA JT Active Configurations

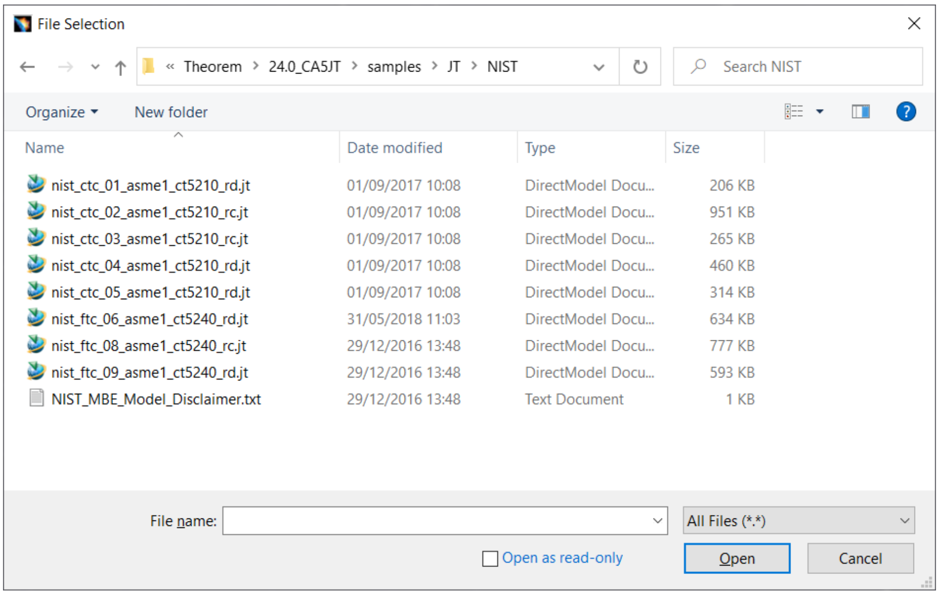
Default
Daimler Configuration
Process PMI R30
Testing

RefreshConfigurationList LaunchConfigurationManager

OK Cancel

Open JT Data

Once CATIA V5 has been started it is possible for a model to be imported from JT using the *File -> Open* menu. Once selected, find the JT data you wish to import into CATIA V5 and select it from the file browser. When ready, press the ‘Open’ button from the menu to import.

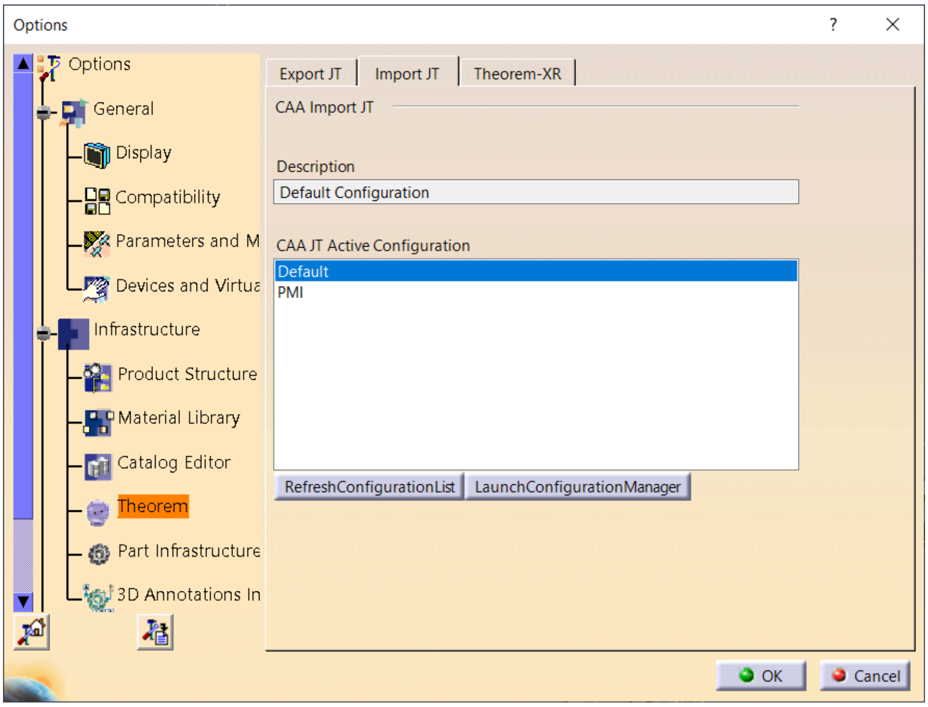


File Selection

« Theorem › 24.0_CA5JT › samples › JT › NIST

Name	Date modified	Type	Size
nist_ctc_01_asme1_ct5210_rd.jt	01/09/2017 10:08	DirectModel Docu...	206 KB
nist_ctc_02_asme1_ct5210_rc.jt	01/09/2017 10:08	DirectModel Docu...	951 KB
nist_ctc_03_asme1_ct5210_rc.jt	01/09/2017 10:08	DirectModel Docu...	265 KB
nist_ctc_04_asme1_ct5210_rd.jt	01/09/2017 10:08	DirectModel Docu...	460 KB
nist_ctc_05_asme1_ct5210_rd.jt	01/09/2017 10:08	DirectModel Docu...	314 KB
nist_ftc_06_asme1_ct5240_rd.jt	31/05/2018 11:03	DirectModel Docu...	634 KB
nist_ftc_08_asme1_ct5240_rc.jt	29/12/2016 13:48	DirectModel Docu...	777 KB
nist_ftc_09_asme1_ct5240_rd.jt	29/12/2016 13:48	DirectModel Docu...	593 KB
NIST_MBE_Model_Disclaimer.txt	29/12/2016 13:48	Text Document	1 KB

To add a user defined configuration to the process (created from the Theorem UI), this can be selected from the **Tools>Options>Infrastructure>Theorem>CAA Import JT** menu. The process will then use this defined configuration to create the JT data as required.



Options

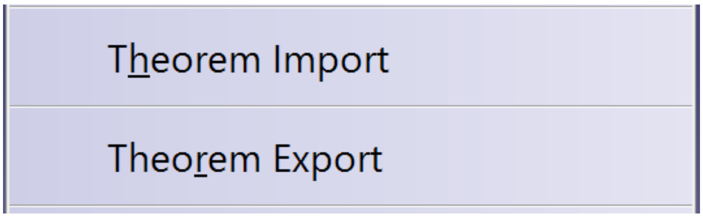
CAA Import JT

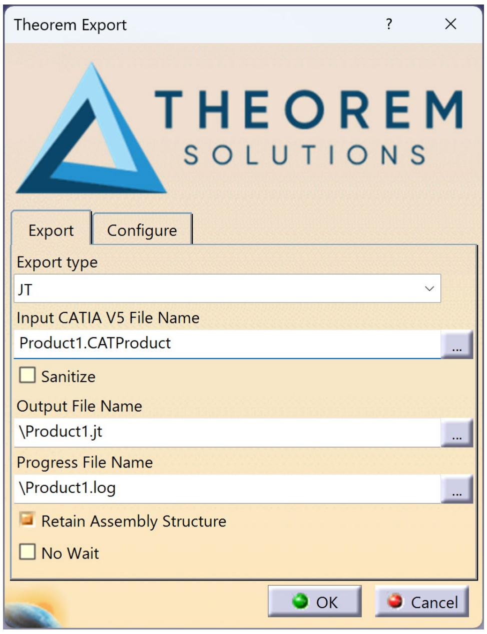
Description: Default Configuration

CAA JT Active Configuration: Default (selected), PMI

Theorem Import and Export

From the 'File' menu in CATIA V5 there are two options named 'Theorem Import' and 'Theorem Export'. This allows you to import JT data or export to JT from the File menu. Select 'File' > 'Theorem Export' or 'Theorem Import' to open the menu.





Export type: JT

Input CATIA V5 File Name: Product1.CATProduct

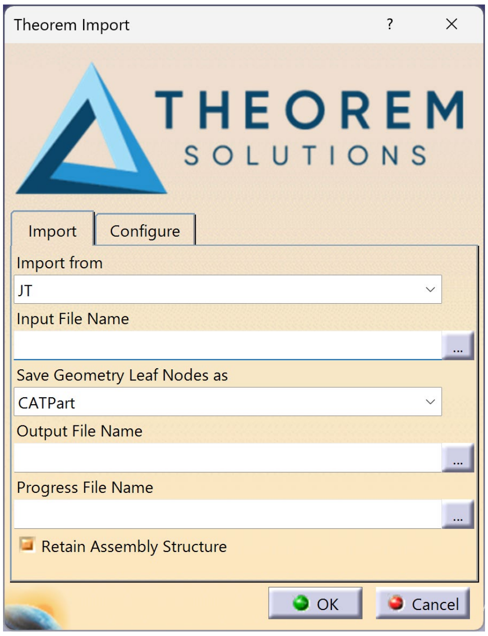
□ Sanitize

Output File Name: \Product1.jt

Progress File Name: \Product1.log

☑ Retain Assembly Structure

□ No Wait



Import from: JT

Input File Name:

Save Geometry Leaf Nodes as: CATPart

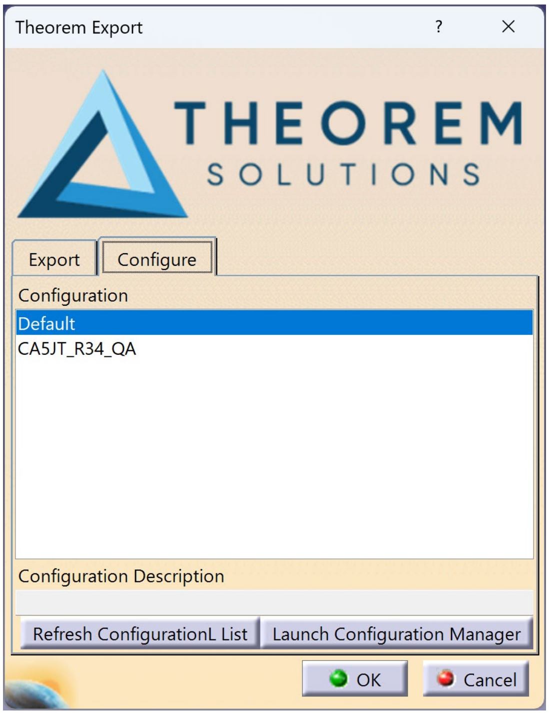
Output File Name:

Progress File Name:

☑ Retain Assembly Structure

Provide output file name and log file location, then press 'OK' to export the data.

To add a custom configuration, select the 'Configure' tab.



Configuration

Default

CA5JT_R34_QA

Configuration Description

This allows the user to select a configuration, or create a new one by selecting the 'Launch Configuration Manager' option. Once a configuration is decided simply select it from the list for it to be applied.

Appendix A – CATIA V5 Configuration

Introduction

This Appendix details how to define and configure the CATIA V5 and Theorem environment to work together.

Conventions

Release of CATIA V5
To indicate a release of CATIA V5 the notation <XX> shall be used. This needs to be replaced with the specific release to be used i.e.31, 32, 33, 34, 35.

Platform specific directory
Within the installation directory of CATIA V5 there is a platform specific directory i.e. win_b64. This directory shall be referred to as ***<OSDS>*** in this Appendix.

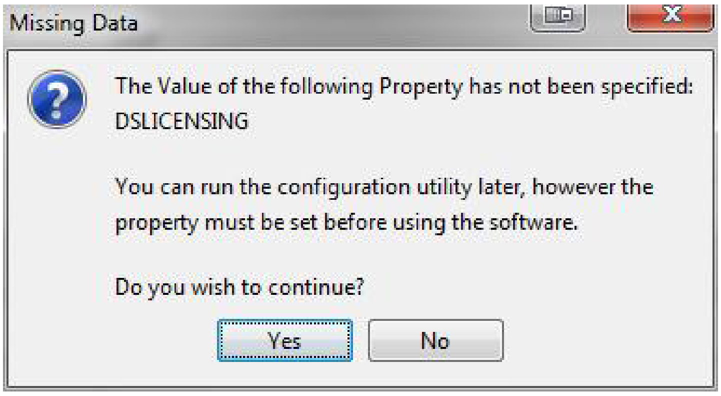
Theorem Installation directory
The Theorem translator installation directory is set at installation time in the translator ***ts_env.bat*** file. This directory shall be noted as ***<%TS_INST%>*** in this Appendix.

CATIA V5 Installation Directory

Upon installation of a CATIA V5 product the user will be asked to specify the installation directory. This is the directory which contains the platform specific <OSDS> directory.

Having selected the CATIA V5 installation directory via the browse button, the installation process will record the location of the CATIA V5 installation directory in the ts_env.bat file. This file is located in the Theorem translator installation directory. If the location of CATIA V5 subsequently changes, the translator can be guided to the changed location by modifying this file using a text editor to modify the ***ts_env.bat*** that is located in the translator installation directory.

If no entry is included for DSLICENSING a warning dialog will be displayed which warns of the empty field. Selecting ***Yes*** to continue will allow the installation to continue.



Missing Data

The Value of the following Property has not been specified:
DSLICENSING

You can run the configuration utility later, however the property must be set before using the software.

Do you wish to continue?

Running CATIA V5 Translators

Before running the translator the user must run CATIA V5 interactively at least once to configure the CATIA V5 environment and license settings. This can be achieved by running the catia5r<XX>_start script as follows:

<Translator_Installation_Directory>\bin\catia5r<XX>_start.cmd

Once CATIA has been run the Translator can run as described in the relevant product User Guide.

CATIA V5 Environment DIRENV & ENV

The default location for CATIA V5 to store its global environment files is in the global directory:
Windows 10:

C:\ProgramData\DassaultSystemes\CATEnv
Or
%APPDATA%\CATEnv

You can find this location by running:

%CATIAV5_INST%\<OSDS>\code\bin\setcatenv -h

The environment files are named in the form ***CATIA.V5RN.B<XX>.txt***
If when installing CATIA V5 the default environment file location was replaced with another location then this location needs to be indicated to the Translator by defining in the ***ts_env.bat*** the environment variable CATIAV5_DIRENV:

set CATIAV5_DIRENV=/some/directory

If the Theorem installation is needed to support multiple releases of CATIA. Then the user can define release specific locations using:

set CATIAV5R<XX>_DIRENV=/some/directory

The Theorem translator will attempts to create its own environment file called ***TheoremCatia5R<XX>.txt.*** The user must therefore have write permission to the CATEnv directory. If this is not possible an existing environment file can be specified using the variable **CATIAV5_ENV**. e.g.

set CATIAV5_ENV=CATIA.V5R31.B31

Note. the extension ***.txt*** is not required. The user can specify a release specific name using ***CATIAV5R<XX>_ENV*** e.g.

set CATIAV5R31_ENV=CATIA.V5R31.B31

Checking the CATIA V5 Environment

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

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

Checking the Theorem Shared Library

A script is provided to ensure that the CATIA V5 environment is compatible with the Theorem shared library. In a command window run the command script:

%TS_INST%\bin\checkcatia5r<XX>cadverter.cmd

A successful output is an indication that the location for CATIA V5 has been specified to the Theorem translator correctly and that the correct version of the Theorem CATIA V5 translator products have been installed.

Appendix B – JT Configuration File

Introduction

A configuration file contains the settings for your translations. The configuration file can be specified using the command line option –config or -z.

If this is not supplied the following directories will be searched in the specified order for the named configuration files: (TS_INST = Installed directory)

tess.config in the directory where the translator is run
tess.config in TS_INST\etc directory

Two example config files are provided in the **TS_INST\etc** directory, a standard **tess.config** one, and one that illustrates the options required for large assembly processing, **tessLargeAssm.config** which is documented by some comments within it.

The JT configuration file contains various sections, each containing different settings based on the section.

The Setup Section

The setup options in the configuration file define how your files are translated. The setup section is the first part of the configuration file and contains a series of standard translator options.

To edit setup options

1. Open an existing configuration file with a text editor.
2. Edit the configuration file options listed in the table below.
3. Save the configuration with a .config extension

Option name	Keywords	Example
EAITranslator	EAITranslator {	EAITranslator {
OutputDirectory	"*path to directory*"	OutputDirectory = "/home/<user>/"
CommonPartsPath	"*path to directory*"	CommonPartsPath= "/myaccount/jtparts/"
chordalOption	"RELATIVE" "ABSOLUTE"	chordalOption = "RELATIVE"
structureOption	"PER_PART" "MONOLITHIC" "FULL_SHATTER"	structureOption = "MONOLITHIC"
WriteWhichFiles	"ALL" "ASSEMBLY_ONLY" "PARTS_ONLY"	WriteWhichFiles = "ALL"
compression	true TRUE false FALSE	compression = true
triStripOpt	true TRUE false FALSE	triStripOpt = false
seamSewing **Note: Not available for Unigraphics.**	true TRUE	seamSewing = true

	false FALSE	
seamSewingTol	*any integer*	seamSewingTol = 0.001
includeBrep	true TRUE false FALSE	includeBrep = false
brepPrecision	"SINGLE" "DOUBLE"	brepPrecision = "SINGLE"
autoNameSanitize	true TRUE false FALSE	autoNameSanitize = true
updateChangedPartsOnly	true TRUE false FALSE	updateChangedPartsOnly = false
verboseReporting	true TRUE false FALSE	verboseReporting = false
writeAsciiAssembly	true TRUE false FALSE	writeAsciiAssembly = false
singlePartsNoAssem	true	singlePartsNoAssem = false

	TRUE false FALSE	
smartLODgeneration	true TRUE false FALSE	smartLODgeneration = true
autoLowLODgeneration	true TRUE false FALSE	autoLowLODgeneration = true
numLODs	*any integer*	numLODs = 3
close brace	}	}

The Filter Section

The filter section of the configuration file contains the filename and metadata filtering information. Edit this section if you want to change how the translator sanitizes filenames and filters metadata keys.

To edit filter options

1. Open an existing configuration file with a text editor.
2. Edit the configuration file options from the table below.
3. Save the configuration with a .config extension

Option name	Keywords	Example
Filter	Filter {	Filter {
FilenameSanitizeSet	*"string of characters"*	FilenameSanitizeSet = "abc123."
FilenameSanitizeSetAdd	*"string of characters"*	FilenameSanitizeSetAdd = "4l"
FilenameSanitizeSetDelete	*"string of characters"*	FilenameSanitizeSetDelete = "c"
MetadataKey	*"string of characters"*	MetadataKey = "*metadata key to exclude*"
close brace	}	}

The Metadata Section

The metadata section sets which metadata to attach to all parts, assemblies and nodes of the model.

Note: Be sure to add these options to the configuration file in pairs: one line to define the metadata key and one line to define the metadata value.

To edit metadata options

1. Open an existing configuration file (.CONFIG) in a text editor.
2. Edit the configuration file options shown in the table below.
3. Save the configuration with a .config extension

Option name	Keywords	Example
Metadata	Metadata {	Metadata {

AddToParts	*"string of characters"*	AddToParts = "<metadata key>"
		AddToParts = "<metadata value>"

AddToAssemblies	*"string of characters"*	AddToAssemblies = "<metadata key>"

		AddToAssemblies = "<metadata value>"

AddToAllNodes	*"string of characters"*	AddToAllNodes = "<metadata key>"
		AddToAllNodes = "<metadata value>"

close brace	}	}

The Level of Detail Section

The level of detail section of the configuration file contains the tessellation and simplification information for each level of detail in the file.

This section consists of several sets of level of detail (LOD) information, and the number of these sets depends on the number you specified on the numLODs line in the configuration file.

To edit level of detail options

1. Open an existing configuration file in a text editor.
2. Edit the configuration file options listed below.
3. Save the configuration with a .config extension

Option name	Keywords	Example
LOD	LOD "*lod number*" {	LOD "1" {
Level	*any integer*	Level = 1
Chordal	*any number*	Chordal = 0.001
Angular	*any number*	Angular = 25
Length	*any number*	Length = 1
FeatureSuppression	*any integer*	FeatureSuppression = 0
Simplify	*any number*	Simplify = 0.60
close brace	}	}

The JT Options Section

The JT Options section of the configuration file contains arguments for JT data.

To edit special options

1. Open an existing configuration file with a text editor.
2. Edit the configuration file options shown in the table below.
3. Save the configuration with a .config file extension.

Option	Keyword	Example	Default Value
JtOptions	JtOptions {	JtOptions {	
AddSemanticPMI	true TRUE false FALSE	AddSemanticPMI = true	true
AnnotationPlanesViewsetName	Character string	AnnotationPlanesViewsetName = “Views”	Views
AppendCADExtension	true TRUE false FALSE	AppendCADExtension = false	true
BrepType	XT JT XTJT	BrepType = true	XT
BrepWireframe	true TRUE false FALSE	BrepWireframe = false	false
CadPropertyMappingFile	Character String	CadPropertyMappingFile = "%TS_INST%\data\jt\jt_property_mapping.txt"	
CollapseHierarchy	SOLtoDetail toSets toPart SURandFACtoDetail toPartOptimized expandPart	CollapseHierarchy = SOLtoDetail	SOLtoDetail
DefaultLayerFilter	"Character String"	DefaultLayerFilter = “”	“”
EnhanceColour	true TRUE false FALSE	EnhanceColour = true	true
ExpandPart	true	ExpandPart = false	false

	TRUE false FALSE		
ExplodeSolidsToFaces	true TRUE false FALSE	ExplodeSolidsToFaces = false	false
ExternalDetailNaming	Character string	ExternalDetailNaming = “”	“”
FaceEdgeTol	Real	FaceEdgeTol = 0.000006	0.000006
FixDegenerateEdges	true TRUE false FALSE	FixDegenerateEdges = true	true
FixSmallFeaturesOpenSolids	true TRUE false FALSE	FixSmallFeaturesOpenSolids = false	false
FixSmallFeaturesSolids	true TRUE false FALSE	FixSmallFeaturesSolids = false	false
ForceBodyCreation	true TRUE false FALSE	ForceBodyCreation = true	true
IncrementalSewing	true TRUE false FALSE	IncrementalSewing = true	true
IncrementalSewingNoOfIterations	Integer	IncrementalSewingNoOfIterations = 5	5
JtBrepFixup	true TRUE false FALSE	JtBrepFixup = False	False
LayerFilter	true TRUE false FALSE	LayerFilter = false	false
ModelViews	true TRUE false FALSE	ModelViews = true	true

NameRootNode	true TRUE false FALSE	NameRootNode = false	false
OutputUnits	mm cm m inches	OutputUnits = mm	mm
ParasolidTolerantModelling	true TRUE false FALSE	ParasolidTolerantModelling = true	true
ParasolidTolerantModellingFactor	Integer	ParasolidTolerantModellingFactor = 3	3
PlmxmlPart	true TRUE false FALSE	PlmxmlPart = false	false
PlmxmlPropertyMappingFile	Character String	PlmxmlPropertyMappingFile = "%TS_INST%\data\jt\plmxml_property_mapping.txt"	
PmiAttributeMap	true TRUE false FALSE	PmiAttributeMap = false	false
PmiAttributeMapFilename	"Character String"	PmiAttributeMapFilename = “”	
PmiAttributeSplit	true TRUE false FALSE	PmiAttributeSplit = false	false
PmiGlyphs	2D 3D off	PmiGlyphs = 3D	3D
PmiPolygons	true TRUE false FALSE	PmiPolygons = false	false
ProduceTessellatedOutput	true TRUE false FALSE	ProduceTessellatedOutput = false	false
ReuseSolids	true TRUE false FALSE	ReuseSolids = false	false

SavedViewsViewsetName	"Character String"	SavedViewsViewsetName = “Catpures”	Captures
SewParasolidBodies	true TRUE false FALSE	SewParasolidBodies = true	true
SewParasolidBodiesTol	Real	SewParasolidBodiesTol = 0.01	0.01
ShineSetting	Integer	ShineSetting = 30	30
SimplifyGeometry	true TRUE false FALSE	SimplifyGeometry = false	false
SplitDiscontinuousSurfaces	true TRUE false FALSE	SplitDiscontinuousSurfaces = false	false
StructureOutputType	JT PLMXML PLMXMLJT STEPBOM STEPBOM_E2	StructureOutputType = JT	JT
SubNode	true TRUE false FALSE	SubNode = false	false
UseInstanceNames	true TRUE false FALSE	UseInstanceNames = false	false
FlushRootNodeColour	true TRUE false FALSE	FlushRootNodeColour = false	false
ExplodeSolidIntoFaces	true TRUE false FALSE	ExplodeSolidIntoFaces = false	false
AddSolidNodes	true TRUE false FALSE	AddSolidNodes = false	false
WriteSubAssyFiles	true TRUE	WriteSubAssyFiles = false	false

	false FALSE		
create_cadid	true TRUE false FALSE	create_cadid = false	false
cadid_attr	Character String	cadid_attr = “”	
cadid_fname	Character String	cadid_fname = “”	
close brace	}	}	

The Catia5 Options Section

The Catia5 Options section of the configuration file contains arguments for Catia5 data.
To edit special options

1. Open an existing configuration file with a text editor.
2. Edit the configuration file options shown in the table below.
3. Save the configuration with a .config file extension.

Option	**Keyword**	**Example**	**Default Value**
Catia5Options	Catia5Options {	Catia5Options {	
ApplyTrimmingLimits	true TRUE false FALSE	ApplyTrimmingLimits = false	false
AssemblyReportFilename	Character string	AssemblyReportFilename = “”	
ConvertSurfaceFaceLimits	On Off Powerfill	ConvertSurfaceFaceLimits = On	On
ConvertGenericFilletSurfaceTol	Real	ConvertGenericFilletSurfaceTol = 0.0001	0.0001
ConvertCurvesToNurbs	true TRUE false FALSE	ConvertCurvesToNurbs = false	false

ConvertCurvesToNurbsTol	Real	ConvertCurvesToNurbsTol = 0.00001	0.00001
ConvertSurfacesToNurbs	None Fillets Spheres Fillets+Spheres	ConvertSurfacesToNurbs = None	None
ConvertSurfacesToNurbsTol	Real	ConvertSurfacesToNurbsTol = 0.00001	0.00001
DecimalSeparator	dot comma	DecimalSeparator = dot	dot
FaceOpacity	true TRUE false FALSE	FaceOpacity = true	true
FeatureName	true TRUE false FALSE	FeatureName = false	false
GenerateEmptyPart	true TRUE false FALSE	GenerateEmptyPart = false	false
HybridJT	true TRUE false FALSE	HybridJT = false	false
InstanceNaming	Character String	InstanceNaming = “”	
MaximumNurbsDegree	Integer	MaximumNurbsDegree = 5	5
MergeCGRVertices	true TRUE false	MergeCGRVertices = false	false

	FALSE		
NonManifold	true TRUE false FALSE	NonManifold = false	false
OpacityZero	Real	OpacityZero = 0.1	0.1
OutputMBD	true TRUE false FALSE	OutputMBD = false	false
OutputMBDLeafNodes	true TRUE false FALSE	OutputMBDLeafNodes = false	false
PrintInfo	true TRUE false FALSE	PrintInfo = true	true
ProcessCaptures	true TRUE false FALSE	ProcessCaptures = false	false
ProcessEsplice	true TRUE false FALSE	ProcessEsplice = false	false
ProcessElectrical	true TRUE false	ProcessElectrical = false	false

	FALSE		
ProcessHybridJT	true TRUE false FALSE	ProcessHybridJT = false	false
ProcessLargeFaces	HideLargeFaces AllowLargefaces DatumFaces True False	ProcessLargeFaces = Datumfaces	DatumFaces
ProcessLayers	ALL_LAYERS A comma separated list of layer numbers(0 - 255), using hyphen '-' to separate number ranges.	ProcessLayers = ALL_LAYERS	ALL_LAYERS
ProcessMotion	true TRUE false FALSE	ProcessMotion = false	false
ProcessOpenSolids	true TRUE false FALSE	ProcessOpenSolids = true	true
ProcessPMI	true TRUE false FALSE	ProcessPMI = false	false
ProcessPoints	true TRUE false	ProcessPoints = true	true

	FALSE		
ProcessSolids	true TRUE false FALSE	ProcessSolids = true	true
ProcessWelds	true TRUE false FALSE	ProcessWelds = false	false
ProcessWireFrame	true TRUE false FALSE	ProcessWireFrame = true	true
ReadCachedCGR	true TRUE false FALSE	ReadCachedCGR = false	false
ReadCGREdges	true TRUE false FALSE	ReadCGREdges = true	true
ReadFaceColours	true TRUE false FALSE	ReadFaceColours = true	true
ReadFTAReferenceGeometry	true TRUE false FALSE	ReadFTAReferenceGeometry = false	false

ReadHiddenData	None Geometry Structure All	ReadHiddenData = None	None
ReadInfiniteAxis	true TRUE false FALSE	ReadInfiniteAxis = false	false
ReadLeafNodeCGR	true TRUE false FALSE	ReadLeafNodeCGR = true	true
ReadMotion	true TRUE false FALSE	ReadMotion = false	false
ReadNOA	true TRUE false FALSE	ReadNOA = false	false
ReadScanData	true TRUE false FALSE	ReadScanData = true	true
ReadSpaceReservations	true TRUE false FALSE	ReadSpaceReservations = false	false
ReduceNurbsSurfaceDegree	true	ReduceNurbsSurfaceDegree = false	false

	TRUE false FALSE		
RetainAssemblyStructure	true TRUE false FALSE	RetainAssemblyStructure = true	true
RetainTrailingZeros	true TRUE false FALSE	RetainTrailingZeros = false	false
SimplifyCurves	true TRUE false FALSE	SimplifyCurves = false	false
SimplifyProgressFile	true TRUE false FALSE	SimplifyProgressFile = false	false
SelectionSets	true TRUE false FALSE	SelectionSets = false	false
SurfaceChecking	true TRUE false FALSE	SurfaceChecking = false	false
SurfaceCheckingTol	Real	SurfaceCheckingTol = 0.001	0.001
TextName	true	TextName = false	false

	TRUE false FALSE		
TopGroupPerBody	true TRUE false FALSE	TopGroupPerBody = false	false
TrimFaceSurfaces	true TRUE false FALSE	TrimFaceSurfaces = true	true
UDFAxisSystems	true TRUE false FALSE	UDFAxisSystems = false	false
ViewContext	Default3D Unfolded	ViewContext = Default3D	Default3D
close brace	}	}	

The Catia4 Options Section

The Catia4 Options section of the configuration file contains arguments for Catia4 data.

To edit special options

1. Open an existing configuration file with a text editor.
2. Edit the configuration file options shown in the table below.
3. Save the configuration with a .config file extension.

Option	Keyword	Example	Default Value
Catia4Options	Catia4Options {	Catia4Options {	

CatiaAssemblyPartsPath	ASIS SET LAYER ENTITYTYPE VIEW	ColourVisualizationMode = ASIS	ASIS
ColourVisualizationMode	Character string	AssemblyReportFilename = “”	
FacetColour	true TRUE false FALSE	FacetColour = false	false
GroupPipes	groupPipelines groupPipes ungroupPipelines	GroupPipes = groupPipelines	groupPipelines
ProcessEntityNames	ALL_ENTITIES A comma separated list of ENTITY names	ProcessEntityNames = ALL_ENTITIES	ALL_ENTITIES
ProcessFAC	true TRUE false FALSE	ProcessFAC = true	true
ProcessGPR	true TRUE false FALSE	ProcessGPR = true	true
ProcessLayers	ALL_LAYERS CURRENT_LAYER_FILTER A comma separated list of layer numbers(0 - 255), using hyphen '-' to	ProcessLayers = ALL_LAYERS	ALL_LAYERS

	separate number ranges.		
ProcessModelNumber	Integer	ProcessModelNumber = 1	1
ProcessNOSHOW	true TRUE false FALSE	ProcessNOSHOW = false	false
ProcessNRBS	true TRUE false FALSE	ProcessNRBS = true	true
ProcessPIP	true TRUE false FALSE	ProcessPIP = true	true
ProcessPOI	true TRUE false FALSE	ProcessPOI = true	true
ProcessSetNames	ALL_SETS A comma separated lists of SET names	ProcessSetNames = ALL_SETS	ALL_SETS
ProcessSKI	true TRUE false FALSE	ProcessSKI = true	true
ProcessSOL	true TRUE false FALSE	ProcessSOL = true	true

ProcessSTR	true TRUE false FALSE	ProcessSTR = true	true
ProcessSUR	true TRUE false FALSE	ProcessSUR = true	true
ProcessWireFrame	true TRUE false FALSE	ProcessWireFrame = true	true
UseColorMethod	AS_FILED SET LAYER ENTITY	UseColorMethod = AS_FILED	AS_FILED
close brace	}	}	

The General Options Section

The General Options section of the configuration file contains general arguments.

To edit special options

1. Open an existing configuration file with a text editor.
2. Edit the configuration file options shown in the table below.
3. Save the configuration with a .config file extension.

Option	**Keyword**	**Example**	**Default Value**
GeneralOptions	GeneralOptions {	GeneralOptions {	
MassProperties	true TRUE	MassProperties = false	false

	false FALSE		
ProgressFileInformation	true TRUE false FALSE	ProgressFileInformation = true	true
ProgressFileWarnings	true TRUE false FALSE	ProgressFileWarnings = true	true
ReportFilename	Character String	ReportFilename = ”%TEMP%\report1.log”	
AdvancedOptions	Chatacter String	AdvancedOptions = “”	
close brace	}	}	

Appendix C – Error Codes

Common Error Codes

When processing data, the command screen or log file will show a return code as a status of the translation. These are detailed below.

Return Code	Description
0	Success
1	Issues during write leg such as incomplete run or no arguments given
2	Issue obtaining license
3	Input file argument issue or failure to open Progress File
4	Issue during read including no entities read
5	Failure to located secondary process (aka vwr_3dpdf.exe or vwr_jt.cmd)
6	
7	Failure to open CATIA V5 document
8	Failure to find root node of CATIA V5 document
9	Failure to start CATIA V5 session
10	catia5cadverter.dll version doesn't match the catia5_XXX.exe
11	CATIA V4 data given as input file but bi _V4MAIN license available