TRANSLATE

Multi-CAD for CATIA V5 – JT

USER GUIDE

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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, Defense, Power Generation and Transportation companies and their Supply chains use our products and services daily. Working closely with our customers, to both fully understand their requirements and feed their input into our development processes has significantly contributed to our technology and industry knowledge.

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

Theorem's Product Suite

Theorem have 3 main Product brands. These are:

TRANSLATE

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

See our website for more detail.

PUBLISH

The creation of documents enriched with 3D content

See our website for more detail.

VISUALIZE

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

See our website for more detail.

The CATIA V5 Multi-CAD Bi-directional JT Translator

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

The CATIA V5 Multi-CAD to 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 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.
- 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 Siemens JTOpen API to read and write the respective data formats.
- When creating JT files a number of data types can be generated. A facetted representation, in addition a precise geometry representation using either XT_BREP or JT_BREP format can be created.

- 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.

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 – 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:

https://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.

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 – JT CD

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

Software and License Installation

The software installation and License configuration process is fully defined in a separate document. Please refer to the document titled Installation Guide for complete installation instructions

Running the Product

Once configured and licensed, the product will be ready to be used.

Prior to initially using the translator CATIA V5 must have been launched at least once to allow the selection of the CATIA V5 licenses to be recorded in the user's CATSettings folder. Once the CATSettings are generated the translator will be able to be executed either internally or externally to CATIA V5 until the CATSettings are removed. Therefore if you operate a procedure that programmatically generates the users CATSettings this requirement to maintain a reference to existing CATSettings must be considered when using the product in batch external to the CATIA V5 application.

All specific CATIA V5 environment configuration details are documented in Appendix A of this document.

Using the Product

The CATIA V5 <> JT MultiCAD CADverters can be invoked in a variety of methods, these include;

- Interactively from within CATIA V5 using standard CATIA V5 menu options such as “Insert Existing Component” , “File > Save As” and “File > Open”
- From the Theorem UI
- From a command prompt window using the command line. This mechanism can be used to automate the translation process as part of any automated workflow.

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 JT CATIAV5RXX” (where XX is the release version (29 for V5-6R2019, 30 for V5-6R2020, 31 for V5-6R2021 & 32 for V5-6R2022).

The Start menu item is located under the “Start + All programs + CATIA” folder and is also named “Theorem Multi-CAD JT CATIAV5RXX”.

Options and Configurations

The CATIA V5 Multi-CAD architecture allows you to invoke the Theorem JT 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.

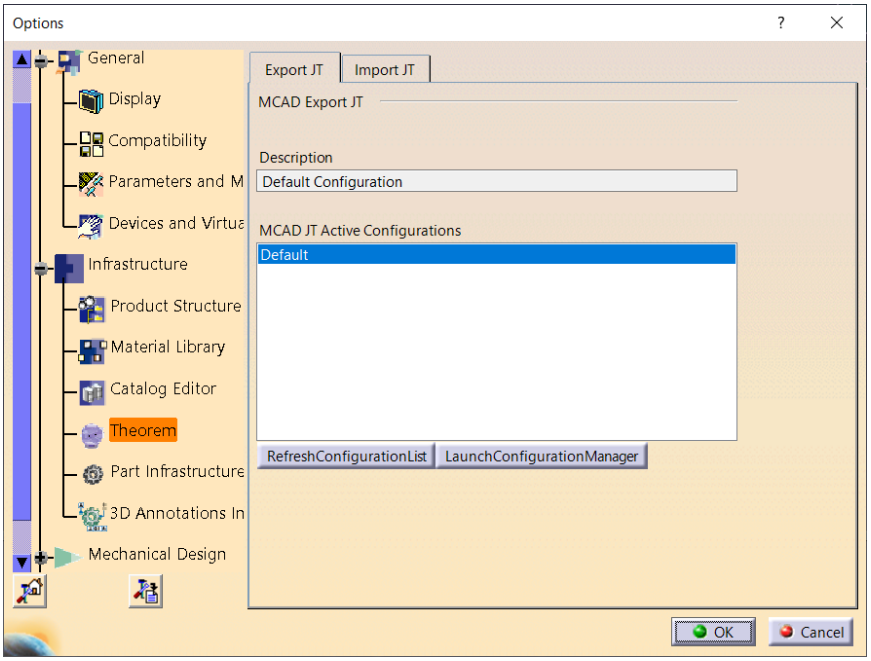
Note that this option page is only available if the TXX-THEOREM-MULTICAx GATEWAY (Theorem Partner) license is available which is provided by the Theorem license manager. Please see Setting Conversion Options in CATIA for a full description of these options.

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

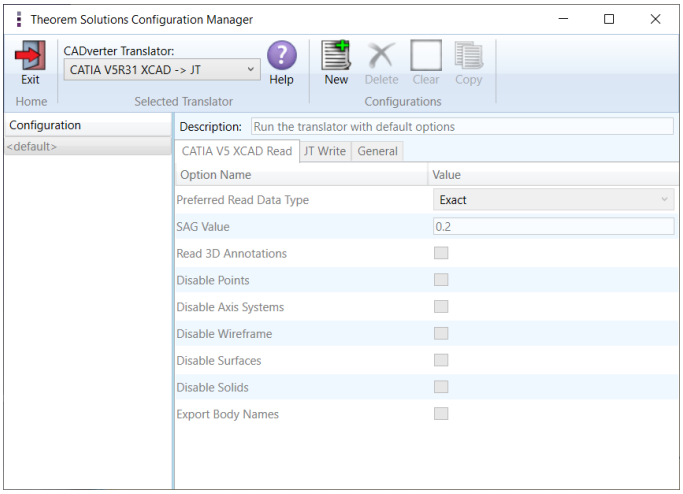
One method to allow you to specify additional options to the JT to V5 Multi-CAD translator is provided by a configuration file that the translator reads during the conversion process.

The configuration file is named **%TS_INST%\data\jt\jt_xcad_opts.txt** and contains information on the various options that can be specified using this method.

In addition a dedicated page of options for **MultiCAD JT** Import and Export operations is also available under the CATIA ‘**Tools>Options>Infrastructure>Theorem**’ menu and is presented as follows:



The 'Export JT and 'Import JT' tabs both contain a list of Configurations used to process the JT data. To create or edit an existing configuration select the 'Launch Configuration Manager' option. This will launch the window below:



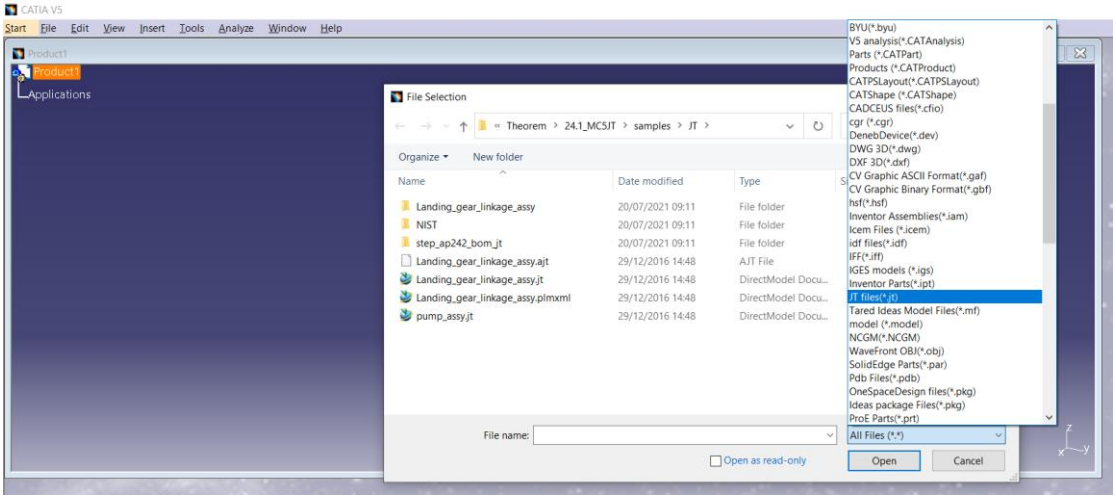
To create and choose options please review the Translator Customization section of this document.

Once the Configuration is completed close the Configuration Manager and select 'Refresh Configuration List' to show the new configuration(s).

To use a Configuration simply select it in the list and press 'OK'.

Using "Insert Existing Component"

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. The "JT Files" file type should be selected and a JT ".jt" file chosen



The translation will now proceed and the CATIA V5 representation of the JT 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%\JTToNavConfigs.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\jt\JTToNavConfigs.txt will be used. For further information on the CATIA V5 Multi-CAD integration methods and commands see the CATIA Infrastructure Documentation.

The Theorem JT to V5 Multi-CAD translator is also invoked when the CATIA "Update Status Checker" command determines that a JT 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"

Visualizing Inserted Annotations

It should be noted that when you perform the "Insert Existing Component" command with a jt 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.

Using “File + Open”

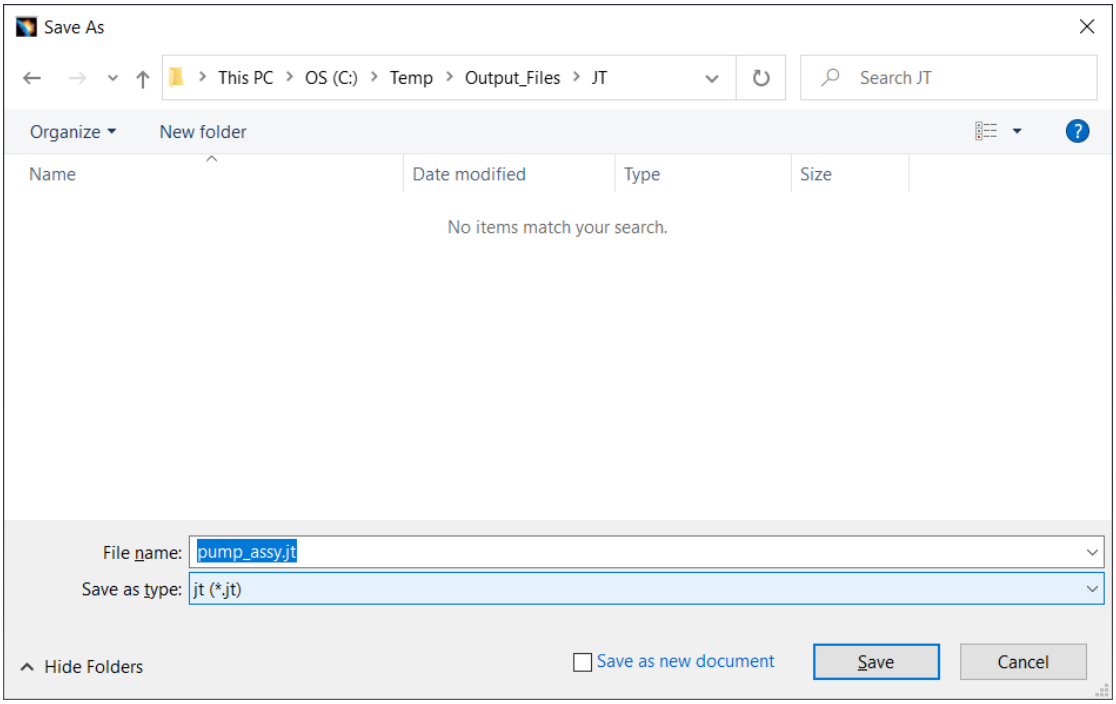
The CATIA “File + Open” menu option allows you to select the file input type “jt”.

The translation will proceed and the CATIA V5 representation of the JT assembly/components will be opened in a new CATProduct window. This can be saved as a CATProduct.

Using “File + Save As”

The CATIA “File + Save As” menu option allows you to select the file output type “jt”.

This will invoke the Theorem CATIA V5 to JT CADverter product using the selected CATPart or CATProduct node in the active CATIA session.



The CATIA V5 XCAD to JT processing messages, errors and warnings are recorded in a ‘.log’ file located in the CATIA CATReport directory. The file is named after the active selected CATIA part/product name. E.g. if the part/product ***Mypart*** were selected, the log file name would be **%CATReport%*Mypart*.log** .

A summary file is also produced which contains key information and the process completion status information. It is named after then log file with the additional suffix .summary. (e.g. **%CATReport%*Mypart*.log.summary**).

The Data saved to JT format will export using the provided Configuration as seen on Page 10.

JT Configuration Files

The Theorem Multi-CAD product also allows the selection of a user defined, or Theorem provided, JT configuration file (.config)

The default jt write configuration file is located as

%TS_INST%\etc\tessCATIA5MultiCAD.config

The options in the Configuration file are detailed in Appendix B.

Property Mapping Files

The user can also manage the mapping of properties written from / to the JT file via a selectable JT property mapping file.

The default property mapping files are located in

%TS_INST%\data\jt\jt_v5_property_mapping.txt

%TS_INST%\data\jt\v5_jt_property_mapping.txt

These text files contains information to show users how to implement a mapping process.

The contents of these option menus can be control by the specification of a jt_mcad_options_configuration.txt file and example of which can also be found in the %TS_INST%\data\jt directory. To define a different file set the environment variable TS_JT_MCAD_OPTIONS_CONFIGURATION

The format of the jt_mcad_options_configuration.txt is

<jt_config_files>
Default TessCATIA5MultiCAD;%TS_INST%\etc\tessCATIAV5MultiCAD.config
</jt_config_files>
<jt_import_property_files>
Default Import Jt Property Filter;%TS_INST%\data\jt\jt_v5_property_mapping.txt
</jt_import_property_files>
<jt_export_property_files>
Default Export Jt Property Filter;%TS_INST%\data\jt\v5_jt_property_mapping.txt
</jt_export_property_files>

There is one option menu entry per line with <Description> ; <Absolute File Path>

Where the <Description> is whats displayed in the option menu and the file path is the location of the jt write configuration file or the property filter. This path definition can include environment variables.

The user can control the mapping of user defined attributes contained in the CATIA product definition and external files during the import and export processes.

The 'JT Import Property Mapping File' and 'JT Export Property Mapping File' are text files of a format described as follows:-

A mapping file is used to control which properties are converted by setting a control value. Setting the control value to 0 will stop a specific property from being exported.

The mapping file can also enable the mapping of property names to new names: this is performed by switching the name between the input name (= field 1) and the output name (= field 2)

The File Line Format is as follows:-

SourceName, TargetName, Control, Dummy, Dummy, Dummy

Lines beginning with a "#" are taken as comment lines

SourceName – is the input attribute name.

TargetName – is the output attribute name (NULL means use *SourceName*)

Control – is flag to control conversion: 0 - Do not convert, 1 – Do convert

Dummy – unused fields

If *SourceName* is given as NULL then any item not included in map will match

So to include all other attributes use

NULL,NULL,1,,,

Or to exclude all other attributes use

NULL,NULL,0,,,

Examples

To exclude the MPARTNAME attribute

MPARTNAME,NULL,0,,,

To include the TAG attribute

TAG,NULL,1,,,

To rename the REFASSYTYPE attribute to ASSYTYPE

REFASSYTYPE,ASSYTYPE,1,,,

From the Command Line

The Theorem User Interface provides the ability to generate conversion command lines that are appended to a nominated batch file for subsequent execution.

%TS_INST%\bin\cad_run.cmd <translator_name>
-i *<input_file_name>*
-o *<output_file_name>*

[*other options*]

E.g. here is a command line example from a batch file:-
call "C:\Theorem\CAD_25.1_MC5JT_WIN.01\bin\cad_run.cmd" JT_CATIA5R32XCAD
-i C:\Theorem\ CAD_25.1_MC5JT_WIN.01\samples\JT\MyFile.jt
-o c:\temp\MyFile.CATProduct
mode_catpart
progress_file c:\temp\ progress.log > c:\temp\ screen.log 2>&1

The <translator_name> will take the form INPUT_OUTPUT as seen in the example above and is case sensitive. For this product the allowed inputs and outputs are

- *CATIA5R29XCAD*
- *CATIA5R30XCAD*
- *CATIA5R31XCAD*
- *CATIA5R32XCAD*
- *JT*

So, to write JT data from CATIA V5-6R2021 (R31) the <translator_name> argument would be:
CATIA5R31XCAD_JT

To write CATIA V5-6R2022 (R32) data the <translator_name> argument would be:
JT_CATIA5R32XCAD

PLMXML from the command line

The command line version of the JT_CATIA5RxxxXCAD convert will also accept the Siemens proprietary **.PLMXML** file format for holding product structure data as the input file.

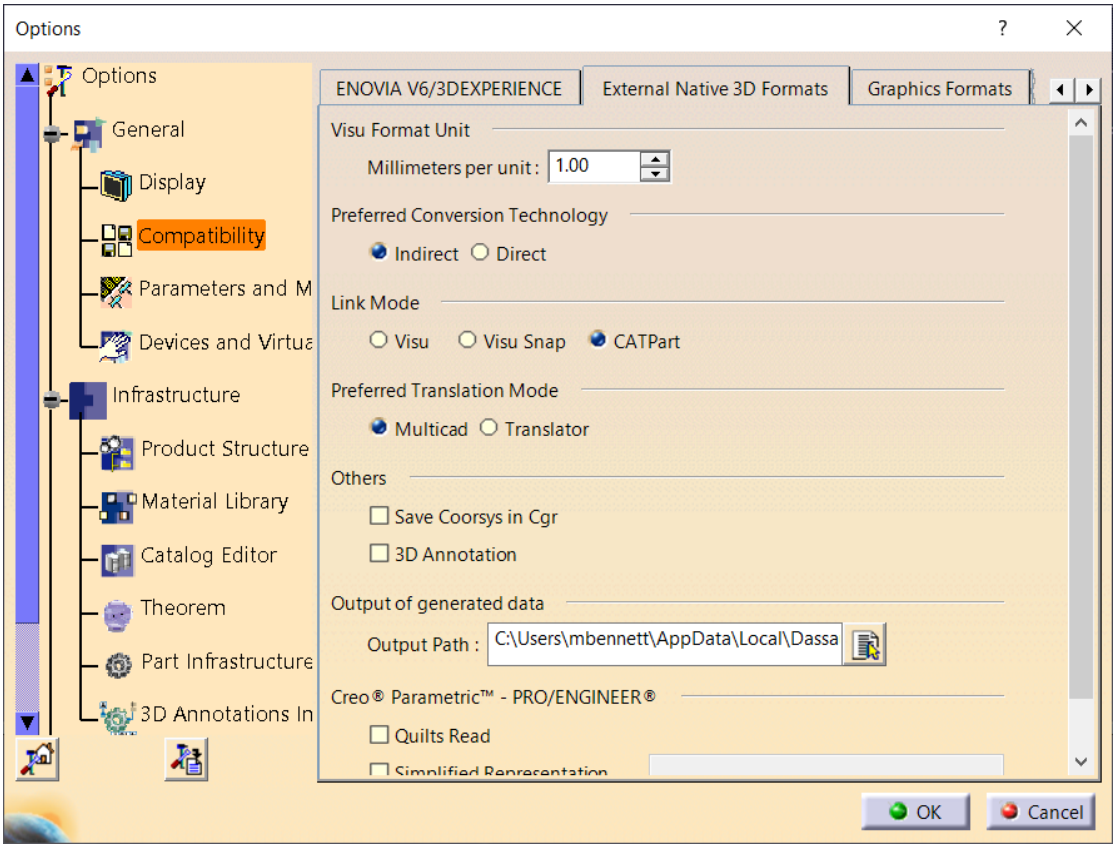
JT to CATIA V5 MultiCAD Usage

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



Visu Format Unit

- Millimeters per unit

The conversion of units from JT parts is incorporated into the Theorem CADverter, so this value should always be set to 1.00

Preferred Conversion Technology

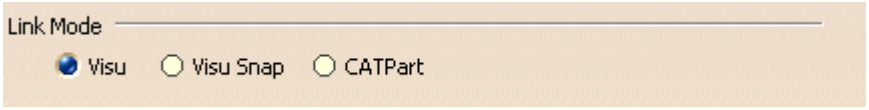
Indirect

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

Direct

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

Link Mode



Visu

Tesselated data. By default, this option is activated.

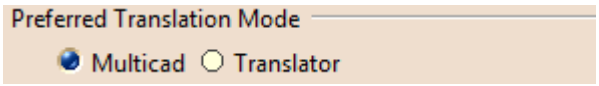
Visu Snap

Selecting this option forces the processing of JT 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 JT 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 JT file to be created in CATPart form.

Preferred Translation Mode



Selecting this option controls if links will be retained between the source jt 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 JT 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 JT, will be imported.

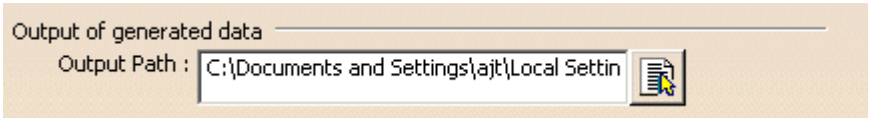
(Note, requires one of the following licenses):

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.

The JT 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 JT 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.JtServerCfile=%TS_INST%\\data\\jt\\jt_xcad_opts.txt

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

Option	Description
info	Outputs extra processing information to log files
debug	Outputs extra debugging information to log files help diagnose problems.
diagnostics	Outputs extra debugging information to log files to help diagnose problems.
single_file	If a JT assembly has mixed units, the user can use the 'single_file' option to force the JT data into a single intermediate representation file with any mirrored and scaled data being exploded out. This results in all unique instances of an assembly with different scales will have their geometry duplicated; with the correct scaling applied. If a monolithic JT assembly with mixed units is imported, this option is required.
noprep	This option effects the reading of XT-Brep data (embedded parasolid brep definitions) within the JT file. By default the brep data is prepared for writing to another system by steps such as removing small edges, splitting complex faces, and converting surfaces to. This option disables this preparation and uses the original parasolid definition.
rd_native_edge	By default some analytic edge definitions are converted to NURBS curves. This option forces the use of native analytic edge definitions.
read_points **read_wire_frame**	By default, points and wireframe curve entities are not read from the JT file. If point entities are required, these options should be uncommented.
read_pmi	Overrides the CATIA Tools - options - Compatibility - “External Formats” “3D Annotations” CATIA setting to invoke read of PMI data.
show_axis_system	By default axis systems are created in CATIA hidden visibility space. This option will create them in the shown visibility space.

The JT to V5 Progress and Log File Outputs

The JT 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.jt were selected, the log and summary file names would be %CATReport%\tea.log and %CATReport%\tea.log.summary.

If the JT 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 JT Progress and Log File Outputs

The CATIA V5 XCAD to JT 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 names would be %CATReport%\Mypart.log and %CATReport%\Mypart.log.summary.

If the CATIA V5 XCAD to JT 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

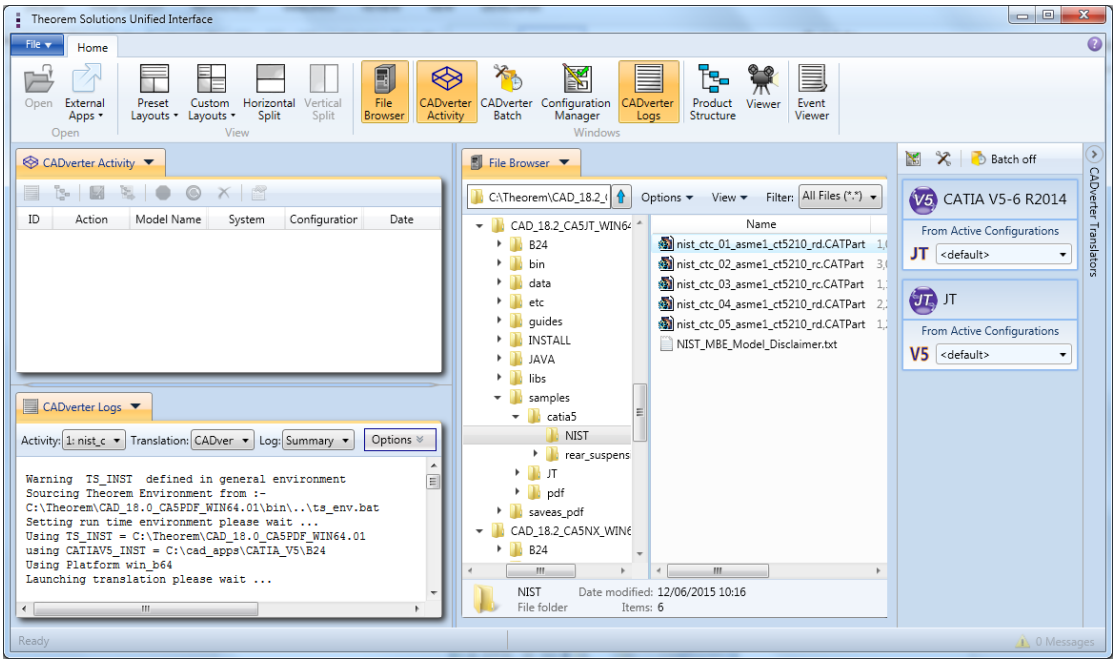
Default Translation – via the Unified Interface

The Unified Interface can be started via the Start Menu – if a shortcut was added during installation.

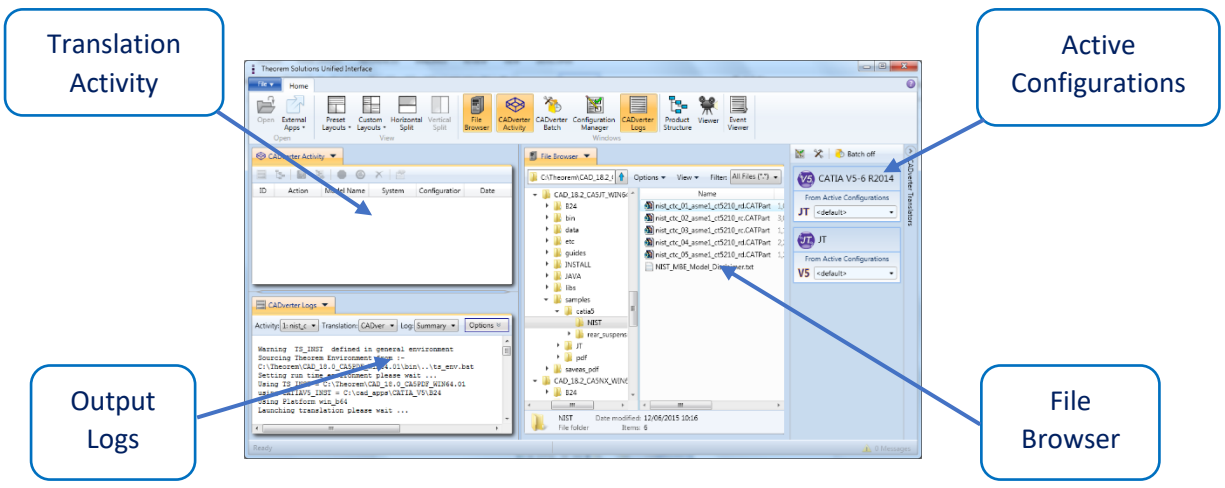
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

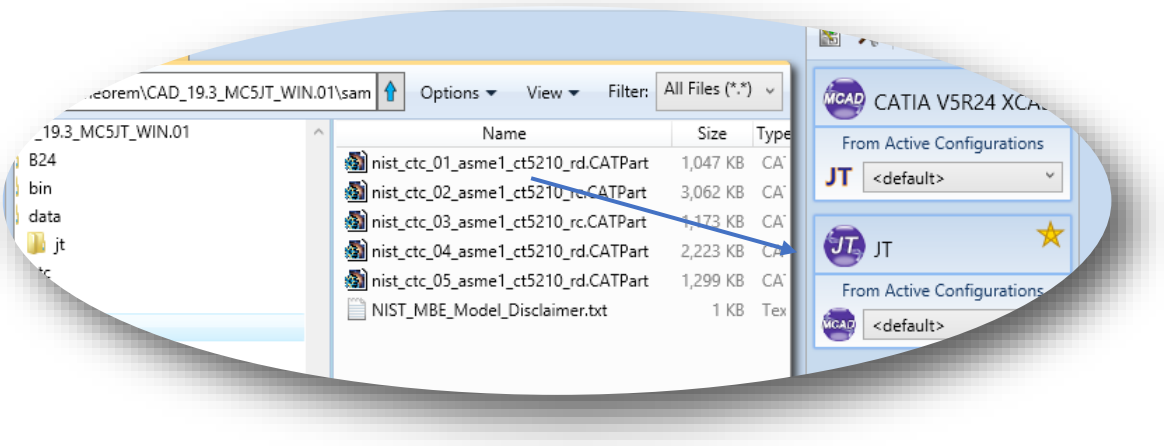
The following interface will be launched:



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

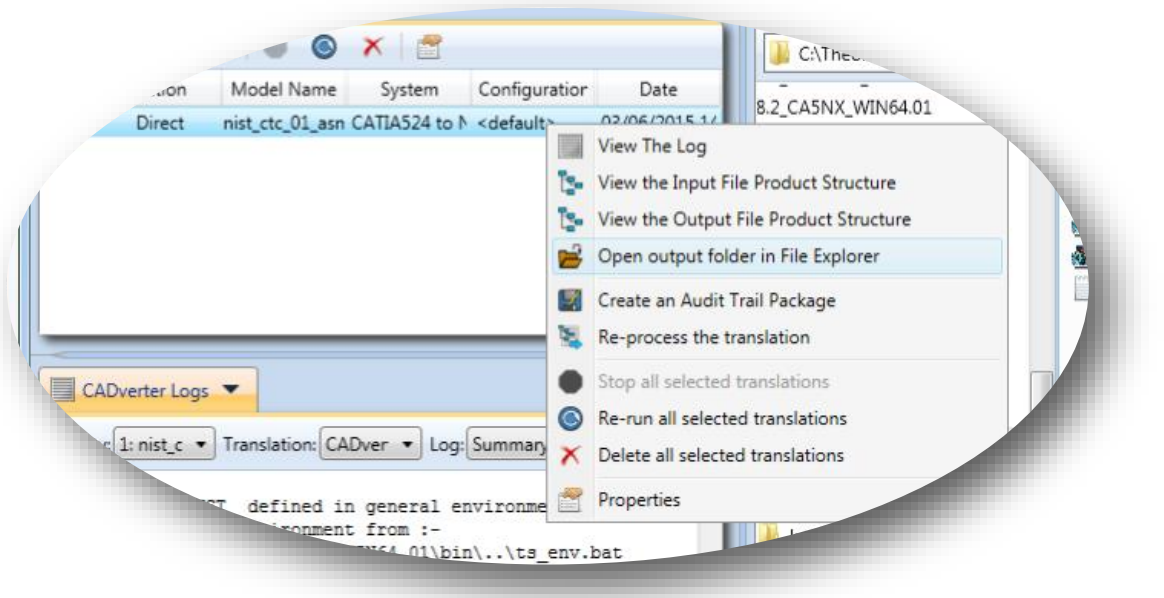


The simplest way to translate from CATIA V5 or JT is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



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:



Default Translation – via Command Line Interface

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 CATIA5R[XX]XCAD_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_CATIA5R[XX]XCAD –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 R30, change to CATIA530):

The example above will translate a CATIA V5 sample file provided within the installation and will be output to the target location. In this case:

C:\TEMP\Rear_Suspension.jt

JT to V5 translations using the CATIA V5 Executable on Command Line

To run the translator from the command line you must ensure that the CATIA V5 environment is set correctly. The easiest method to achieve this is to use the standard CATIA start-up process. This uses the following syntax;

%V5_INSTALL_PATH%win_b64\code\bin\CATSTART.exe
-run "***%TS_INST%*\B***%V5_VER%***win_b64\code\bin\JTToNav.exe
%***COMMAND_LINE_ARGUMENTS***%"
-env Theorem_Multi-CAD_JT_CATIAV5R***%V5_VER%***
-direnv "***%TS_INST%*\B***%V5_VER%***win_b64\CATEnv"

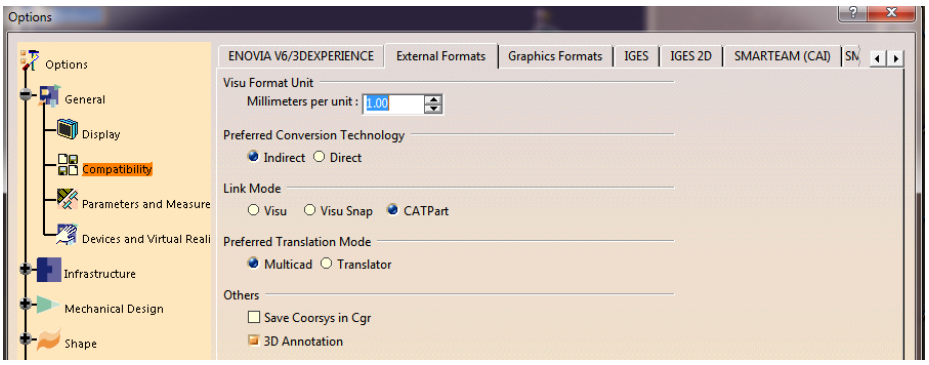
Where;

%V5_INSTALL_PATH% = the path to the CATIA V5 installation e.g C:\Program Files\Dassault Systems\v5r26\B26
%TS_INST% = the path to the Theorem Solutions installation e.g. C:\Theorem\CAD_19.3_MC5JT_WIN.01
%V5_VER% = the version of CATIA V5 that is being used e.g. 24, 25, or 26
%COMMAND_LINE_ARGUMENTS% = these will depend upon the type of Link Mode data type that you want to create in the output e.g. VISU, VISU+SNAP or CATPart

E.g. To translate a JT file to CATIA V5 to create the geometry in CATPart format and process PMI/FTA data the command line syntax would be;

e.g. %V5_INSTALL_PATH%\win_b64\code\bin\CATSTART.exe -run
"%TS_INST%\B26\win_b64\code\bin\JTToNav.exe
-c %TS_INST%\data\jt\JTToNavConfigs_CATPart.txt
–i %TS_INST%\samples\jt\NIST\ nist_ctc_01_asme1_ct5210_rd.jt
–oproduct C:\temp\NIST_01.CATProduct –dpart C:\temp read_pmi"
-env Theorem_Multi-CAD_JT_CATIAV5R26
-direnv "%TS_INST%\B26\win_b64\CATEnv"

Using Link Mode: CATPart



Command Line Arguments

-c %TS_INST%\data\jt\JTToNavConfigs_CATPart.txt

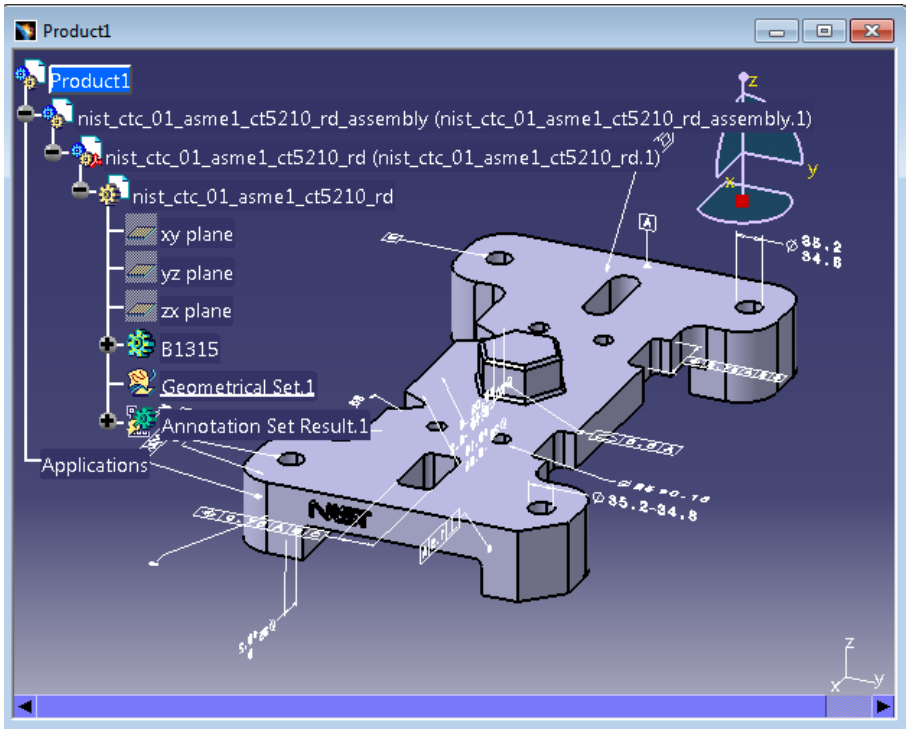
-i <input_JT_filename>

-oproduct <output_CATProduct_filename>

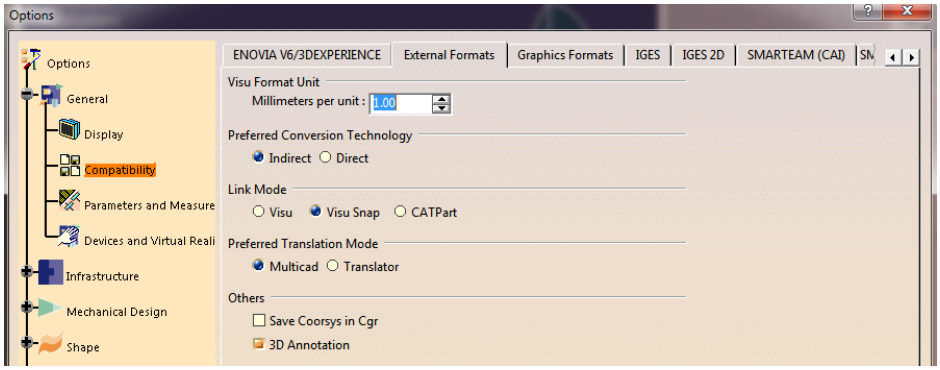
-dpart <path_to_output_folder>

read_pmi - optional argument to process JT PMI data. Note this requires access to a V5 FTA license

e.g –c %TS_INST%\data\jt\JTToNavConfigs_CATPart.txt –i %TS_INST%\samples\jt\NIST\ nist_ctc_01_asme1_ct5210_rd.jt –oproduct C:\temp\NIST_01.CATProduct –dpart C:\temp read_pmi



Link Mode: Visu Snap



Command Line Arguments

-c %TS_INST%\data\jt\JTToNavConfigs_VISUSNAP.txt

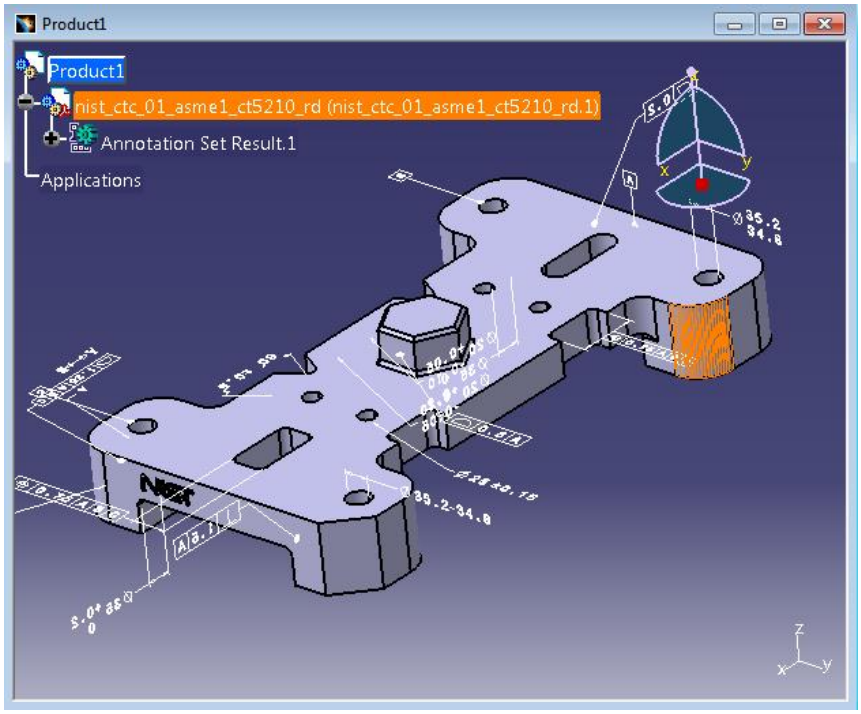
-i <input_JT_filename>

-oproduct <output_CATProduct_filename>

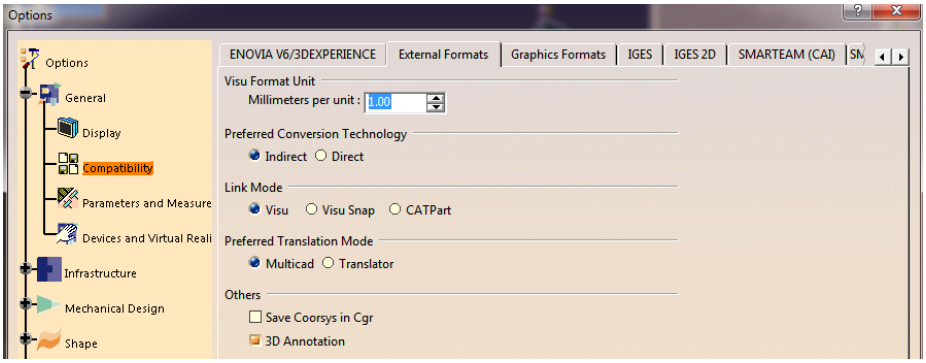
-d <path_to_output_folder>

read_pmi - optional argument to process JT PMI data. Note this requires access to a V5 FTA license

e.g –c %TS_INST%\data\jt\JTToNavConfigs_VISUSNAP.txt –i %TS_INST%\samples\jt\NIST\nist_ctc_01_asme1_ct5210_rd.jt –oproduct C:\temp\NIST_01.CATProduct –d C:\temp read_pmi



Link Mode: Visu



Command Line Arguments

-c %TS_INST%\data\jt\JTToNavConfigs_VISU.txt

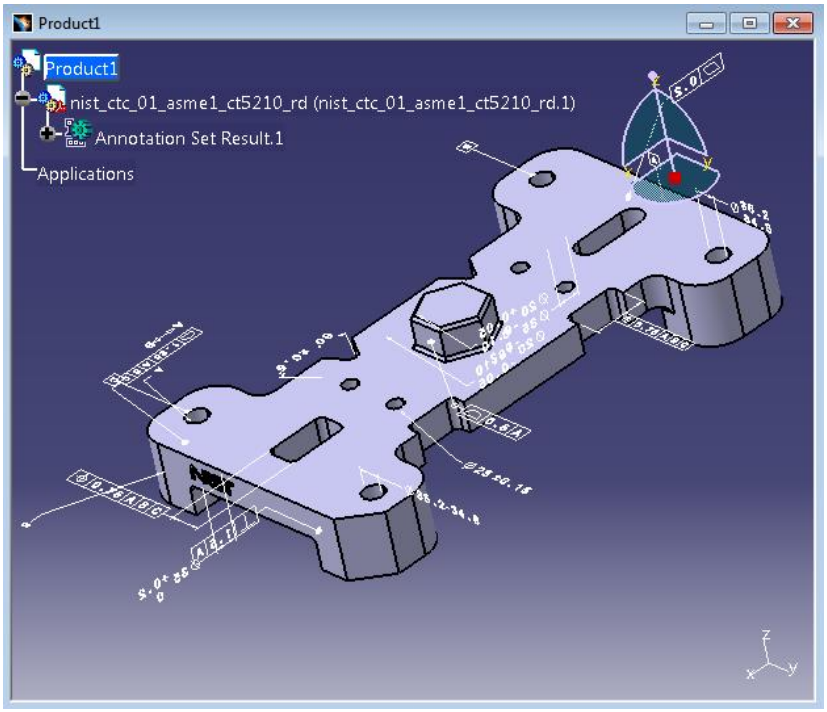
-i <input_JT_filename>

-oproduct <output_CATProduct_filename>

-d <path_to_output_folder>

read_pmi - optional argument to process JT PMI data. Note this requires access to a V5 FTA license

e.g –c %TS_INST%\data\jt\JTToNavConfigs_VISU.txt –i %TS_INST%\samples\jt\NIST\nist_ctc_01_asme1_ct5210_rd.jt –oproduct C:\temp\NIST_01.CATProduct –d C:\temp read_pmi



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.

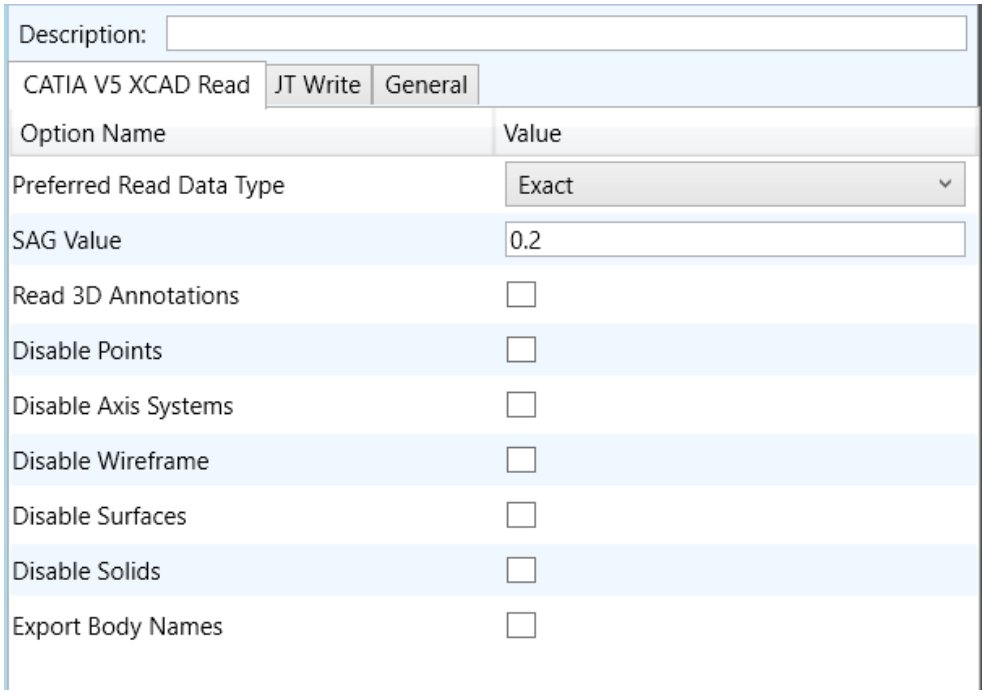
Common Options for CATIA V5 to JT

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

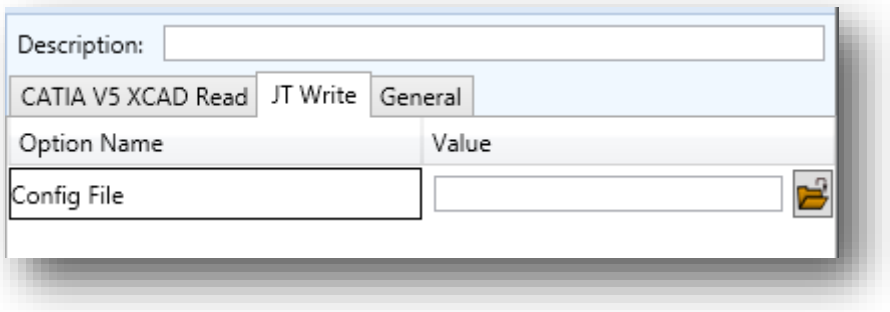


Each of these options is described below:

Option	Description
Preferred Read Data Type	Allows the user to specify Exact or Tessellated read
SAG Value	Set SAG value
Read 3D Annotations	Read PMI
Disable Points	Disable the processing of standalone Points
Disable Axis Systems	Disable the processing of Axis Systems
Disable Wireframe	Disable the processing of standalone Wireframe
Disable Surfaces	Disable the processing of standalone Surfaces
Disable Solids	Disable the processing of solids
Export Body Names	Create Body Named Containers

JT Write Arguments

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

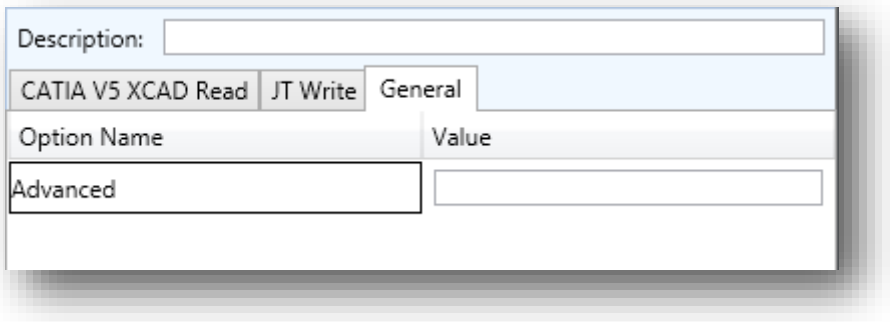


This option is described below:

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 General Arguments

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



This option is described below:

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

Exporting CGR Data to JT

When processing CGR files the default process will write a single JT "part" file for each CGR. However, it is possible to expand the JT output such that an individual JT part file is created for each of the bodies in defined in the original CGR file. To achieve this, use the command line options **expand_part body_names**. When using the UI select the option "**Export Body Names**" in the **CATIA V5 XCAD Read** options and set the **expand_part** option using the the Advanced option in the **General** options panel.

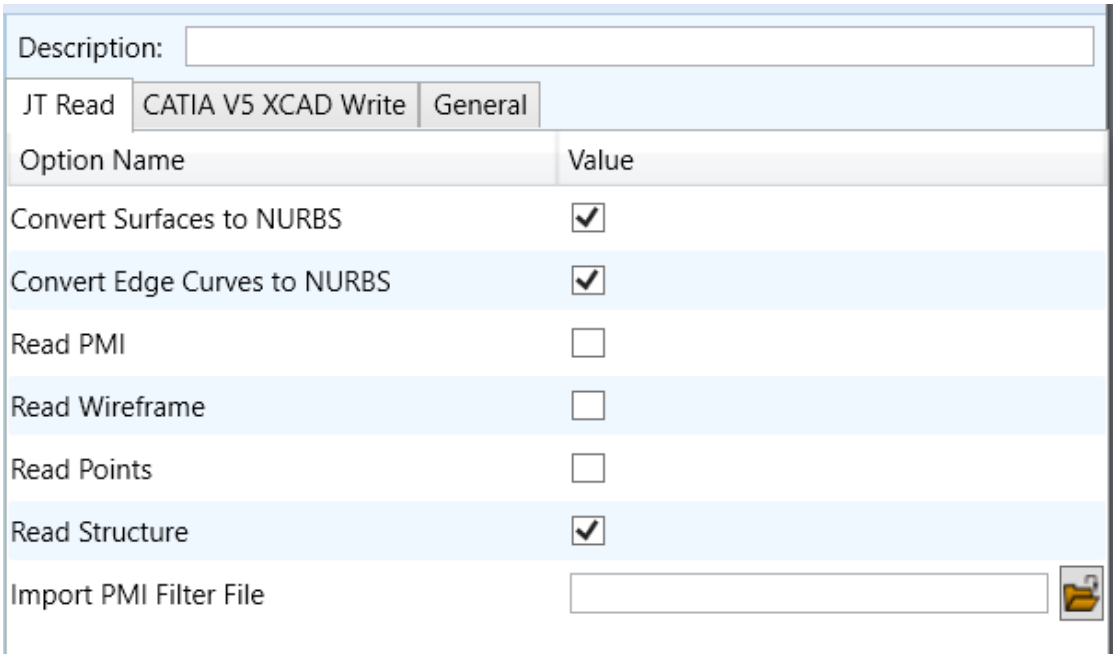
Common Options for JT to CATIA V5

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



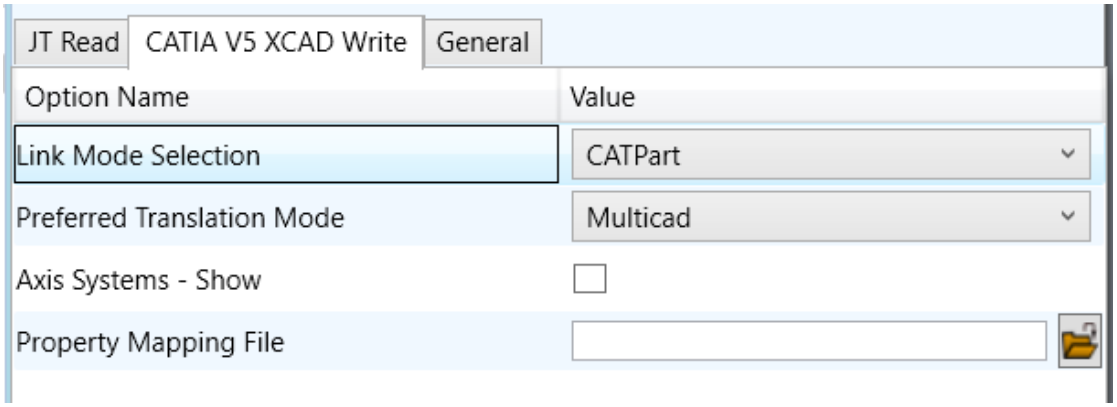
Each of these options is described below.

Option	Description
Convert 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 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*
Read PMI	Reads 3D PMI. Default is OFF. ○ Command Line Syntax ▪ *read_pmi*

Read Wireframe	Read JT wireframe data. Default is OFF. ○ Command Line Syntax ▪ *read_wire_frame*
Read Points	Read Points. Default is OFF. ○ Command Line Syntax ▪ *read_points*
Read Structure	Read assembly structure. Default is ON. ○ Command Line Syntax ▪ *structure* ▪ *no_structure – to turn off*
Import PMI Filter File	JT Import PMI Filter File Name. Default is OFF ○ Command Line Syntax ▪ *pmi_filter_file <file>*

CATIA V5 Write Arguments

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



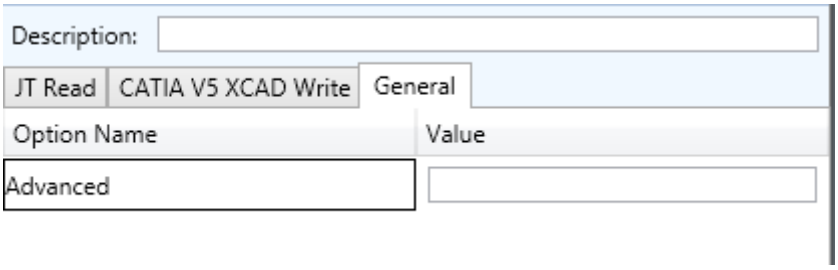
Each of these options is described below:

Option	**Description**
Link Mode Selection	Choose the Link Mode to translate with (See Link Mode for details) ○ Options are: ▪ Visu ▪ Visu Snap ▪ *CATPart (Default)* ○ Command Line Syntax ▪ *mode_visu* ▪ *mode_visu_snap* ▪ *mode_catpart*

Preferred Translation Mode	Choose the mode to translate with (See Translator Mode for details) ○ Options are: ▪ *Multicad (Default)* ▪ *Translator* ○ Command Line Syntax ▪ *translator_mode_off* ▪ *translator_mode_on*
Axis Systems Show	Show Axis Systems in output. Default is OFF ○ Command Line Syntax ▪ show_axis_system
Property Mapping File	Specify a file which allows filtering of Detail user attributes. Default is OFF. ○ Command Line Syntax ▪ *cad_prop_map_file [file]*

JT 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

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. 29, 30, 31, 32.

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.

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:

%TS_INST%\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:

%APPDATA%\DassaultSystemes\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.V5R29.B29

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

set CATIAV5R19_ENV=CATIA.V5R29.B29

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)

tessCATIA5.config in the directory where the translator is run

tess.config in the directory where the translator is run

tessCATIA5.config in TS_INST\etc directory

tess.config in TS_INST\etc directory

Two example config files are provided in the **TS_INST\etc** directory, a standard **tessCATIA5.config** one, and one that illustrates the options required for large assembly processing,**tessLargeAssmCATIA5.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"`	`structureOption = "MONOLITHIC"`

	"MONOLITHIC" "FULL_SHATTER"	
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 false FALSE	seamSewing = true
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 TRUE false FALSE	singlePartsNoAssem = false
smartLODgeneration	true TRUE false FALSE	smartLODgeneration = true
autoLowLODgeneration	true TRUE false FALSE	autoLowLODgeneration = true
numLODs	*any integer*	numLODs = 3
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 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 Special Section

The special section of the configuration file contains lines that are unique to this translator.

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 {`	

ProcessSolids	`true`
	`TRUE`
	
	`false`
	`FALSE`
		`ProcessSolids = true`	true

ProcessOpenSoilds	`true`
	`TRUE`
	
	`false`
	`FALSE`
		`ProcessOpenSolids = true`	true

ProcessWireFrame	`true`
	`TRUE`
	
	`false`
	`FALSE`
		`ProcessWireFrame = false`	true

ProcessPoints	`true`
	`TRUE`
	
	`false`
	`FALSE`
		ProcessPoints = true	true

ProcessHiddenGeom	`true`
	`TRUE`
	
	`false`
	`FALSE`
		`ProcessHiddenGeom = false`	false

ProcessLayers	`ALL_LAYERS`	`ProcessLayers = 1-10, 20, 30-40, 88`	ALL_LAYERS

	A comma separated list of layer numbers (0-255), using and hyphen '-' to separate number ranges.		
ReportFilename	Full system file path	Unix example ReportFilename = /users/caddata/translation/result/part55 Windows example ReportFilename = P:\caddata\translation\result\part55	Unix system /tmp/tscprogressyi Windows system C:%TEMP%\tscprogressyi
OutputUnits	mm millimetres cm centimetres m metre metres inches feet yards inputUnits	OutputUnits = mm	inputUnits
AppendCADExtension	true TRUE false	AppendCADExtension = false	false

	FALSE		
ProcessPMI	read_pmi read_pmi_1 read_pmi_2 read_pmi_3 false	ProcessPMI = false	false
ProcessCaptures	true TRUE false FALSE	ProcessCaptures = false	false
collapseHierarchy	false expandPart SOLtoDetail toSets toPart SURandFACtoDetail toPartOptimized	CollapseHierarchy = SOLtoDetail	SOLtoDetail
autoExpandPart	threshold value	autoExpandPart = 50	Optional Config Entry
autoRunAssemblyScript	true TRUE false FALSE	autoRunAssemblyScript = true	Optional Config Entry
useExeInAssemblyScript	Full system file path	Unix example useExeInAssemblyScript = /users/translation/exename	Optional Config Entry

		Windows example useExeInAssemblyScript = X:\users\translation\exename.exe	
useLogDirInAssemblyScript	`Full system path`	`Unix example useLogDirInAssemblyScript = /users/translation/logDir` Windows example useLogDirInAssemblyScript = X:\users\translation\logDir	Optional Config Entry
zPart	`Full system file path`	`Unix example zPart = /users/translation/tessPart.config` Windows example zPart = X:\users\translation\tessPart.config	Optional Config Entry

structureOutputType	`JT`
	
	`PLMXML`
	
	`PLMXMLJT`
		`structureOutputType = JT`	JT

plmxmlPropertyMappingFile	`Mapping File for PLMXML Properties`	`Windows example plmxmlPropertyMappingFile = X:\users\translation\plmxml_property_mapping.txt`	Optional Config Entry

brepType	`JT`
	
	`XT`
	
	`XTJT`
		`brepType = XT`	JT

parasolidTolerantModelling	`true`
	
	`TRUE`
	
	`false`
	
	`FALSE`
		parasolidTolerantModelling = true	true

parasolidTolerantModellingFactor	An integer factor	parasolidTolerantModellingFactor = 4	3
sewParasolidBodies	true TRUE false FALSE	sewParasolidBodies = true	true
sewParasolidBodiesTol	A tolerance for sewing in millimetres	sewParasolidBodiesTol = 0.01	0.01
incrementalSewing	true TRUE false FALSE	incrementalSewing = true	true
incrementalSewingNoOfIterations	The maximum number of iterations to be used for incremental sewing	incrementalSewingNoOfIterations = 7	5
explodeSolidsToFaces	true TRUE false FALSE	explodeSolidsToFaces = true	false
splitDiscontinuousSurfaces	true TRUE false FALSE	splitDiscontinuousSurfaces = true	true
forceBodyCreation	true	forceBodyCreation = false	true

	TRUE false FALSE		
fixDegenerateEdges	true TRUE false FALSE	fixDegenerateEdges = false	true
faceEdgeTol	A tolerance for face creation in metres	faceEdgeTol = 0.000004	0.000006
fixSmallFeaturesSolids	true TRUE false FALSE	fixSmallFeaturesSolids = true	false
fixSmallFeaturesOpenSolids	true TRUE false FALSE	fixSmallFeaturesOpenSolids = true	false
simplifyGeometry	true TRUE false FALSE	simplifygeometry = true	false
brepWireframe	true - process wireframe as JT Brep TRUE	brepWireframe = true	false

	false - process wireframe as tessellation FALSE		
produceTessellatedOutput	true TRUE false FALSE	produceTessellatedOutput = true	false
expandPart	true TRUE false FALSE	expandPart = true	false
reuseSolids	true TRUE false FALSE	reuseSolids = true	false
cadPropertyMappingFile	Mapping File for JT Properties	Windows example cadPropertyMappingFile = X:\users\translation\cad_property_mapping.txt	Optional Config Entry
addSemanticPMI	true TRUE false FALSE	addSemanticPMI = true	false
JTBrepFixup	true	JTBrepFixup = false	true

	TRUE false FALSE		
PMIAttributeMap	true TRUE false FALSE	PMIAttributeMap = true	false
PMIAttributeMapFileName	Mapping File for PMI Attributes	Windows example PMIAttributeMapFileName = X:\users\translation\PMIAttributeMap.txt	Optional Config Entry
externalDetailNaming	Mapping File for detail names	Windows example externalDetailNaming = X:\users\translation\NameMap.txt	Optional Config Entry
layerFilter	true TRUE false FALSE	layerFilter = true	false
defaultLayerFilter	Layer Filter Name to be used as the default	Example defaultLayerFilter = LF1	Optional Config Entry
subNode	true TRUE false FALSE	subNode = true	false

retainAssemblyStructure	`true`
	`TRUE`
	`false`
	`FALSE`
		retainAssemblyStructure = false	true

readSpaceReservations	`true`
	`TRUE`
	`false`
	`FALSE`
		readSpaceReservations = true	false

readScanData	`true`
	`TRUE`
	`false`
	`FALSE`
		readScanData = true	false

readCachedCGR	`true`
	`TRUE`
	`false`
	`FALSE`
		readCachedCGR = true	false

readLeafNodeCGR	`true`
	`TRUE`
	`false`
	`FALSE`
		readLeafNodeCGR = true	false

readHiddenData	`None`
	`Geometry`
	`Structure`
	`All`
		readHiddenData = Geometry	None

viewContext	`Default3D`	viewContext = Unfolded	Default3D

	Unfolded		
readFaceColours	true TRUE false FALSE	readFaceColours = true	false
faceOpacity	true TRUE false FALSE	faceOpacity = true	false
opacityZero	The value of opacity to be actually used when opacity is zero	opacityZero = 0.2	0.1
simplifyProgressFile	true TRUE false FALSE	simplifyProgressFile = true	false
convertCurvesToNurbs	true TRUE false FALSE	convertCurvesToNurbs = true	false
convertCurvesToNurbsTol	A tolerance for curve conversion	convertCurvesToNurbsTol = 0.00003	0.00001
simplifyCurves	true TRUE false	simplifyCurves = true	false

	FALSE		
convertSurfacesToNurbs	None Fillets Spheres Fillets+Spheres All	convertSurfacesToNurbs = Spheres	Fillets
maximumNurbsDegree	The maximum value of degree allowed for surface to NURBS conversion	maximumNurbsDegree = 7	5
convertSurfacesToNurbsTol	A tolerance for surface conversion	convertSurfacesToNurbsTol = 0.00003	0.00001
applyTrimmingLimits	true TRUE false FALSE	applyTrimmingLimits = true	false
trimFaceSurfaces	true TRUE false FALSE	trimFaceSurfaces = false	true
processLargeFaces	true TRUE false FALSE	processLargeFaces = false	true
UDFAxisSystems	true	UDFAxisSystems = true	false

	TRUE false FALSE		
reduceNurbsSurfaceDegree	true TRUE false FALSE	reduceNurbsSurfaceDegree = true	false
surfaceChecking	true TRUE false FALSE	surfaceChecking = true	false
surfaceCheckingTol	A tolerance for surface checking	surfaceCheckingTol = 0.00003	
readInfiniteAxis	true TRUE false FALSE	readInfiniteAxis = true	false
nonManifold	true TRUE false FALSE	nonManifold = true	false
readNOA	true TRUE false FALSE	readNOA = true	false

retainTrailingZeros	true TRUE false FALSE	retainTrailingZeros = true	false
decimalSeparator	dot comma	decimalSeparator = comma	dot
readFTAReferenceGeometry	true TRUE false FALSE	readFTAReferenceGeometry = true	false
readMotion	true TRUE false FALSE	readMotion = true	false
readWelds	true TRUE false FALSE	readWelds = true	false
selectionSets	true TRUE false FALSE	selectionSets = true	false
assemblyReportFilename	File for assembly report	Windows example assemblyReportFilename = X:\users\translation\AssemblyReport.txt	Optional Config Entry

progressFileWarnings	`true`
	`TRUE`
	`false`
	`FALSE`
		progressFileWarnings = false	true

progressFileInformation	`true`
	`TRUE`
	`false`
	`FALSE`
		progressFileInformation = false	true

generateEmptyPart	`true`
	`TRUE`
	`false`
	`FALSE`
		generateEmptyPart = true	false

maskFilename	`File containing masking instructions`	`Windows example maskFilename = X:\users\translation\Mask.txt`	Optional Config Entry
close brace	`}`	`}`	