Contents

Overview of Visualise 3D ................................................................................................................. 3
About Theorem ................................................................................................................................. 3
Theorem’s Product Suite ..................................................................................................................... 3
The 3DEXPERIENCE CATIA V6 Bi-directional JT Translator ...................................................... 4
Primary Product Features ............................................................................................................... 4
Primary Product benefits? ................................................................................................................ 5

Getting Started ............................................................................................................................... 6
Documentation ................................................................................................................................. 6
Installation Media ............................................................................................................................. 6
Installation ......................................................................................................................................... 6
License Configuration ....................................................................................................................... 6
Running the Product ......................................................................................................................... 6
Using the Product ............................................................................................................................. 8

Translating Interactively from within 3DEXPERIENCE .................................................................. 8
Theorem Interactive Conversion Settings ....................................................................................... 8
Property Mapping Files ................................................................................................................... 14

PMI Type Filter Files ....................................................................................................................... 15

Advanced User Options ................................................................................................................... 16

Theorem Export ............................................................................................................................... 18
JT Template file usage ..................................................................................................................... 20

Theorem Import ............................................................................................................................... 21
Translating in Batch using CATUtil – PLMBatchDataExchange .................................................... 24
Batch Export to JT ............................................................................................................................ 24
Batch Import from JT ....................................................................................................................... 25
Log File Generation .......................................................................................................................... 25
Export process Log Files .................................................................................................................. 25
Import process Log Files .................................................................................................................. 26

3DEXPERIENCE CATIA V6 Environment Files .......................................................................... 26
Appendix A – JT Configuration File ............................................................................................. 27
Introduction ....................................................................................................................................... 27
The Setup Section ............................................................................................................................ 27
The Level of Detail Section ................................................................................................................. 30
The Filter Section........................................................................................................... 31
The Metadata section...................................................................................................... 32
Overview of Visualise 3D

About Theorem

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

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

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

Theorem’s Product Suite

The 3DEXPERIENCE CATIA V6 to JT translator is part of our Visualize 3D brand which is one of 5 core Theorem brands which consist of:

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

- **Multi-CAD**
  
  Interactive integration of non-native 3D data formats into the native CAD system

- **Visualize 3D**
  
  Direct translation of 3D data for the purpose of Visualization
The 3DEXPERIENCE CATIA V6 Bi-directional JT Translator

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

The 3DEXPERIENCE CATIA V6 to JT Translator is a bi-directional direct database converter between the Dassault Systemes 3DEXPERIENCE CATIA V6 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 and colour information, between these two systems. This product is designed for companies using 3DEXPERIENCE 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 is used by companies using JT based solutions to translate their 3DEXPERIENCE CATIA V6 data into the JT format.

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

Primary Product Features

- Converts all types of geometry, wire frame, surfaces, trimmed surfaces (faces) and solid models.
- Converts assembly structure between both systems.
- Converts attribute data including colour and layer information.
- Integrated with the 3DEXPERIENCE CATIA V6 installation.
- The conversion process can be run Interactively or in Batch Mode
- Data can be filtered by layer and entity type during processing. Geometry can be filtered and selectively processed.
- Uses the 3DEXPERIENCE CATIA V6 API and Siemens JTOpen API to read and write data.
In creating JT files a number of data types can be generated. A facetted representation, a JTBrep definition or an XTBrep definition. As standard a facetted representation is created with an option to select whether JTBrep or XTBrep definition is created.

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 Visualize 3D for the 3DEXPERIENCE CATIA V6 – JT product. For information regarding any of Theorem’s product ranges please contact sales@theorem.com
Getting Started

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

http://www.theorem.com/Documentation

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

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

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

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

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

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

License Configuration

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

Running the Product
Once configured and licensed, the product is ready to be run.

There are 2 distinct ways of running the translator:

- Interactively from within 3DEXPERIENCE CATIA V6
  - The Interactive Interface provides a direct method of importing and exporting to and from 3DEXPERIENCE.
In Batch via CATUtil ·DataExchangePLMBatch

- The 3DEXPERIENCEDataExchangePLMBatch Interface provides a direct method of invoking the translator. It can be used on an *On Demand* basis to translate single or multiple files.
Using the Product

Translating Interactively from within 3DEXPERIENCE

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

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

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

Alternatively, it can be started via the script provided in the Translator installation located in:

\<installation_directory>\bin

The script name is:

start_3DEXPERIENCE_Theorem_Multi-CAD_JT_CATIAV6R<version>.cmd

(where <version> should be substituted for the version of 3DEXPERIENCE CATIA V6 that you have installed – e.g. 2013x, 2014x, 2015x, 2016x):

Theorem Interactive Conversion Settings

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

The method for altering these settings differs between 3DEXPERIENCE versions.

For R2013x, the User settings are accessed through the ’Tools->Options’ menu:-
for R2014x and later revisions these are accessed through the User Preferences options:

Of importance for all versions is the Compatibility options page as shown here for R2013x:
And here for R2014x, R2105x and R2016x:-

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

A dedicated page of options for ‘Infrastructure -> Theorem -> MultiCAD Jt’ import and export operations is available under the CATIA ‘Infrastructure’ settings and is presented as follows:-

For R2013x:-
Each of the options in these panels is described below:

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Import JT Conversion Elements - Points</td>
<td>Enables Point entities to be retained. Default is OFF. Enabling this option will read all Point entities. Command Line Syntax: Option data file <code>jt_xcad_opts.txt</code> syntax: &lt;br&gt; ▪ <code>read_points</code> – to turn on</td>
</tr>
<tr>
<td>Import JT Conversion Elements – Wire Frame</td>
<td>Process Wireframe entities during translation (if present). Default is OFF. Option data file <code>jt_xcad_opts.txt</code> syntax: &lt;br&gt; ▪ <code>read_wire_frame</code></td>
</tr>
<tr>
<td>Import JT Conversion Elements – Structure</td>
<td>Enables Assembly Structure to be retained. Default is ON. Disabling this option will remove all assembly structure and collapse ALL geometry into a single selectable object. Option data file <code>jt_xcad_opts.txt</code> syntax: &lt;br&gt; ▪ <code>no_structure</code></td>
</tr>
<tr>
<td>JT Import Property Mapping File</td>
<td>Filter name choices can be set via a configuration file: &lt;br&gt; <code>%TS_INST%/data/jt/jt_mcad_options_configuration.txt</code> The default mapping is located here: &lt;br&gt; <code>%TS_INST%/data/jt/jt_v6_property_mapping.txt</code> This is a file containing a list of CAD properties and information on how they are mapped from the JT file. (See section Property Mapping Files)</td>
</tr>
</tbody>
</table>
The associated “View” button will present the selected filter file content to the user in an editor window.

**JT Import PMI Filter File Name**
Filter name choices which are set via a configuration file: 
%TS_INST%\data\jt\jt_mcad_options_configuration.txt
The default mapping is located here: 
%TS_INST%\data\jt\jt_pmi_filter.txt
This file lists JT PMI type names which are to be ignored by the import process. (See section PMI Type Filter Files)
The associated “View” button will present the selected filter file content to the user in an editor window.

**Export JT**
**Precise BREP Format**
This option allows the user to select the format of precise data that is written to the JT file
- XT-BREP *(Default)*
  - Creates XT-BREP (Parasolid type embedded solid body definitions) in the JT output files.
- JT-BREP
  - Creates Siemens JtBrep data

**Export JT**
**Conversion Elements - Bodies**
Enables CATIA Bodies to be processed *(Default is ON)*

**Export JT**
**Conversion Elements - Surfaces**
Enables CATIA independent Surfaces to be processed *(Default is ON)*

**Export JT**
**Conversion Elements - Curves**
Enables CATIA Wireframe Curves to be processed *(Default is ON)*

**Export JT**
**Conversion Elements - Axes**
Enables CATIA Axes systems to be processed *(Default is ON)*

**Export JT**
**Conversion Elements - Points**
Enables CATIA Points to be processed *(Default is ON)*

**Export JT**
**Conversion Elements - PMI**
Enables CATIA PMI/FTA data to be processed *(Default is ON)*
### Export JT

**Conversion Elements – No Show**

Enables CATIA entities in No show (Hide) state to be converted. *(Default is OFF - i.e. to ignore hidden entities)*

**Export JT**

**Sub-Part Data**

The user can control how Sub Part Data is processed. There are 3 options:

- **None** - Multiple bodies in a single CATIA part representation will result in a single leaf node per CATIA Part in the JT representation. *(Default Option)*
- **Body Names** - The “Body Names” option adds a level of sub structure that can be accessed using the Show Leaf Structure in Jt VisMockup. It creates a leaf node for each CATIA body that is converted.
- **Specification Tree** – The “Specification Tree” option adds a level of sub structure that can be accessed using the Show Leaf Structure in Jt ViewMockup. This option creates a leaf node corresponding to each top level CATIA Part Specification Tree node that contains bodies.

*Note! This option is only supported by CATIA V5-6R2014 SP4 HF30.*

To enable this option, it is necessary to edit the CATIA environment file *(located by default in %TS_INST%\B24\win_b64\CATEnv\Theorem_MultiCAD_JT_CATIAV5R24.txt)*

This contains a commented variable as follows:-

```
! Enable Sub-Part Data option if supported (requires min CATIA version V5-6R2014 SP4 HF30
! XCAD_ExposePartStructureInNameWithSeparator=#
If this line is uncommented by removing the “!” character, the “Specification Tree” option will be enabled
```

### Export JT

**JT config File**

JT configuration file name choices and selection are set via a configuration file:

```
%TS_INST%\data\jt\jt_mcad_options_configuration.txt
```

The default configuration file is located here:

```
%TS_INST%\etc\tessCATIAV6MultiCAD.config
```

Please see Appendix A for a full description of the JT config file.

The associated “View” button will present the selected configuration file content to the user in an editor window.

### Export JT

**Property Mapping File**

Filter name choices which are set via a configuration file:

```
%TS_INST%\data\jt\jt_mcad_options_configuration.txt
```

The default configuration file is located here:

```
%TS_INST%\data\v6\jt_property_mapping.txt
```
This is a file containing a list of CAD properties and information on how they are mapped to the JT file. (See section Property Mapping Files)
The associated “View” button will present the selected filter file content to the user in an editor window.

**Note!** Administrator control locking is available for this tools option page and is control by specification of a **CATReferenceSettingPath** and running CATIA with the **–admin** option. Please refer to standard 3DEXPERIENCE CATIA V6 documentation for more information.

**Note!** It is important that the selections on this option page are saved in the user’s CATSettings data prior to executing conversions by clicking the **“OK”** button at the bottom of the page.

### Property Mapping Files

The selections of the displayed JT Config File (.config) Property Mapping files and PMI Type Filter Files are configurable through the configuration file:

```
%TS_INST%\data\jt\jt_mcad_options_configuration.txt
```

The format of the `jt_mcad_options_configuration.txt` is:

```
<jt_config_files>
Default TessCATIA6MultiCAD;\%TS_INST\%\etc\tessCATIAV6MultiCAD.config
</jt_config_files>
<jt_import_property_files>
Default Import Jt Property Filter;\%TS_INST%\data\jt\jt_v6_property_mapping.txt
</jt_import_property_files>
<jt_export_property_files>
Default Export Jt Property Filter;\%TS_INST%\data\jt\v6_jt_property_mapping.txt
</jt_export_property_files>
<jt_import_pmi_files>
Default Import PMI Type Filter;\%TS_INST%\data\jt\jt_pmi_filter.txt
</jt_import_pmi_files>
```

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

Where the **<Description>** is the text to be 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 PLM part 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 below:-
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,,,**

PMI Type Filter Files

The user can control the filtering of PMI types on import from JT by specifying an appropriate filter file. A default filter file is provided with the installation located as `%TS_INST%\data\jt\jt_pmi_filter.txt`. This file contains a list of all PMI types by name, and can be edited to exclude different named types by removing a ‘#’ (comment character) from the type name not required to be imported.

e.g. if the file is edited as follows

```plaintext
# File for filtering on PMI type via the command : pmi_filter_file "file name"
# Line Format:-
# "pmi type"
#
# Lines that start with a '#' are ignored.
# The supplied file contains all possible PMI types in alphabetical order preceded by a '#'
# To prevent a particular PMI type from being processed, remove the '#' from that type
#
#arc spot weld
#attribute note
#balloon
#bead
```
Any PMI entities of type ‘center point’ would NOT be imported.

This facility was introduced to reduce processing time due to large numbers of ‘redundant’ PMI entities in a JT file.

Note that default settings that can control the JT Export plugin, are also read from the data file %TS_INST%\data\jt\xcad_jt_opts.txt.

Default settings that can control the JT Import plugin, are also read from the data file %TS_INST%\data\jt\jt_xcad_opts.txt.

Advanced User Options

An advanced option panel is available by setting the environment variable

TS_JT_MCAD_OPTIONS_ADVANCED_USER

Here for example is the R2013x presented options:-

![Advanced User Options Panel](image-url)
Additional options are displayed that allow the specification of the JT File Structure created:

Options are:

As Configured, MONOLITHIC, PER_PART, FULL_SHATTER or MIMIC

An option is provided to override the configured JT LOD settings.

If the Number of LODS is set to a value of 0, the LOD settings contained in the selected JT Config File will be used.

Please see Appendix A for a full description of the JT Config file.
Theorem Export

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

V6 2013x

If 3DEXPERIENCE CATIA V6 R2013x is being used, in order to export, the user selects the PLMAccess - Export option as shown here:-

V6 2014x and later

If 3DEXPERIENCE CATIA V6 R2014x or later is being used, the user selects the menu item as shown here:-
This menu action will then present the user with a Save As dialog box similar to this:
- For R2013x and R2014x:
- For R2015x:
For R2016x:

The “Save as types:” or “Format” selection should be set to JT_THEOREM (*.jt).

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

JT Template file usage

If, when a representation is read from 3DEXPERIENCE CATIA V6, no geometry is found, or a major write error occurs preventing a JT file being produced, an empty JT file (named by default template.jt) will be copied to the expected output file name. This enables the conversion process to complete successfully and maintains the expected file outputs.

This behaviour is implemented via an environment variable set in the Theorem CATEnv file e.g. %TS_INST%\B214\win_b64\CATEnv\Theorem_Multi-CAD_JT_CATIAV6R<ver>.txt file, typically as follows:

```
TS_JT_TEMPLATE_FILE=C:\Theorem\CAD_19.3_MC6JT_WIN.01\data\jt\template.jt
```

The user can change the content of this JT file or the location of this file and its name as required.
Theorem Import

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

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

For 3DEXPERIENCE CATIA V6 R2013x:-

For 2014x and later:-
This launches the Theorem Import Menu:

For R2013x and R2014x:-

For R2015x:-

For R2016x:-
On selecting **Open** or **OK** the on the File selection Panel the JT data will be translated and imported into the 3DEXPERIENCE CATIA V6 database. It will also be opened in the user’s session.
Translating in Batch using CATUtil – PLMBatchDataExchange

The Dassault Systemes CATUTIL batch management tool can be launched with the correct Theorem enabled environment using the scripts provided in the Translator installation at:

```
<installation_directory>\bin\start_CATUTIL_Theorem_Multi-CAD_JT_CATIAV6R<revision>.cmd
```

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

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

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

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

```
TS_IGNORE_JT_CATSETTINGS=1
```

Batch Export to JT

Here is an example showing the appropriate settings in preparation for a database selection and an export “save” or “run”:–

Note selection of “Export” operation, usage “JT_THEOREM” and extension “jt”.

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Note selection of “Export” operation, usage “JT_THEOREM” and extension “jt”. 
Batch Import from JT

Here is an example showing the appropriate settings in preparation for an input JT file selection and an export “save” or “run”:-

Note selection of “Import” operation, usage “JT_THEOREM” and extension “jt”.

Log File Generation

Export process Log Files
In the process of exporting the selected 3DEXPERIENCE CATIA V6 part or assembly, the following log files are generated by the JT Export plug-in.

The process log and error messages are recorded in a ‘.err’ file located in the 3DEXPERIENCE CATIA V6 CATReport directory. The file is named after the active CATIA component.

   e.g. %CATReport%\model1.err

Additional log files are created in the TSC_TEMP_DIR directory. This directory is defined in the %TS_INST%\ts_env.bat file.

   %TSC_TEMP_DIR%\Read_to_viewer_<input_part_name>.log

   Where: <input_part_name> is the name of the input part (or the active part name in interactive usage)
This contains information describing the 3DEXPERIENCE CATIA V6 'data read' processing into Theorem Intermediate data format

%TSC_TEMP_DIR%\viewer_<part-name>_screen_output.log

Where <part-name> is the selected output file name
This contains the screen output of the process of writing the data to JT.

%TSC_TEMP_DIR%\viewer_<part-name>.log

Where <part-name> is the selected output file name
This contains detailed process information of the write of the data to JT.

Import process Log Files
In the process of importing a JT file, the following log files are generated by the JT Import plug-in.

The process log and error messages are recorded in a '.err' file located in the 3DEXPERIENCE CATIA V6 CATReport directory. The file is named after the selected input file name. e.g. if the file Mypart.jt were selected, the log file names would be %CATReport%Mypart.err and %CATReport%Mypart.rpt.

%TS_TEMP_DIR%\<input_file_name>_viewer.log

%TS_TEMP_DIR%\<input_file_name>_viewer_screen_output.log

%CATReport%\<input_file_name>.err

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

3DEXPERIENCE CATIA V6 Environment Files
As part of the Theorem installation process, a set of 3DEXPERIENCE CATIA V6 environment files are created which are subsequently used in the launch of 3DEXPERIENCE and CATUtil sessions to support the Theorem partner plug-ins for JT import and export.

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

These environment files are located in

<installation_directory>\B214\win_b64\Theorem_Multi-CAD_JT_CATIAV6R2013x.txt

<installation_directory>\B216\win_b64\Theorem_Multi-CAD_JT_CATIAV6R2014x.txt

<installation_directory>\B417\win_b64\Theorem_Multi-CAD_JT_CATIAV6R2015x.txt

<installation_directory>\B418\win_b64\Theorem_Multi-CAD_JT_CATIAV6R2016x.txt
Appendix A – 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.

The configuration file name can be defined in the interactive User Preferences/Tools Options form, which is in turn defined in the TS_INST\data\jt\jt_mcad_options_configuration.txt file (where TS_INST = Installed directory).

By default this is set to TS_INST\etc\tessCATIAV6MultiCAD.config

Alternatively, if in batch mode, with the CATIA environment variable TS_IGNORE_JT_CATSETTINGS=1 set, the interactive User Preferences/Tools Options form will be ignored, and the content of the TS_INST\data\jt\xcad_jt_opts.txt file will be used.

In this file, the default is also defined as

-z "%TS_INST%\etc\tessCATIAV6MultiCAD.config"

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

<table>
<thead>
<tr>
<th>Option name</th>
<th>Keywords</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>EAITranslator</td>
<td>EAITranslator</td>
<td>EAITranslator</td>
</tr>
<tr>
<td>OutputDirectory</td>
<td>&quot;path to directory&quot;</td>
<td>OutputDirectory = &quot;/home/&lt;user&gt;/&quot;</td>
</tr>
<tr>
<td>CommonPartsPath</td>
<td>&quot;path to directory&quot;</td>
<td>CommonPartsPath = &quot;/myaccount/jtparts/&quot;</td>
</tr>
<tr>
<td>chordalOption</td>
<td>&quot;RELATIVE&quot;</td>
<td>chordalOption = &quot;RELATIVE&quot;</td>
</tr>
<tr>
<td>structureOption</td>
<td>&quot;PER_PART&quot;</td>
<td>structureOption = &quot;MONOLITHIC&quot;</td>
</tr>
<tr>
<td>Variable</td>
<td>Options</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------</td>
<td>------------------------------------</td>
</tr>
<tr>
<td>WriteWhichFiles</td>
<td>&quot;ALL&quot;</td>
<td>WriteWhichFiles = &quot;ALL&quot;</td>
</tr>
<tr>
<td></td>
<td>&quot;ASSEMBLY_ONLY&quot;</td>
<td></td>
</tr>
<tr>
<td></td>
<td>&quot;PARTS_ONLY&quot;</td>
<td></td>
</tr>
<tr>
<td>compression</td>
<td>true</td>
<td>compression = true</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FALSE</td>
<td></td>
</tr>
<tr>
<td>triStripOpt</td>
<td>true</td>
<td>triStripOpt = false</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FALSE</td>
<td></td>
</tr>
<tr>
<td>seamSewing</td>
<td>true</td>
<td>seamSewing = true</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FALSE</td>
<td></td>
</tr>
<tr>
<td>Note: Not available for Unigraphics.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>seamSewingTol</td>
<td>any integer</td>
<td>seamSewingTol = 0.001</td>
</tr>
<tr>
<td>includeBrep</td>
<td>true</td>
<td>includeBrep = false</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FALSE</td>
<td></td>
</tr>
<tr>
<td>brepPrecision</td>
<td>&quot;SINGLE&quot;</td>
<td>brepPrecision = &quot;SINGLE&quot;</td>
</tr>
<tr>
<td></td>
<td>&quot;DOUBLE&quot;</td>
<td></td>
</tr>
<tr>
<td>autoNameSanitize</td>
<td>true</td>
<td>autoNameSanitize = true</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FALSE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>true</td>
<td>FALSE</td>
</tr>
<tr>
<td>------------------------</td>
<td>------</td>
<td>-------</td>
</tr>
<tr>
<td><strong>updateChangedPartsOnly</strong></td>
<td>true</td>
<td>FALSE</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>updateChangedPartsOnly = false</strong></td>
<td></td>
</tr>
<tr>
<td><strong>verboseReporting</strong></td>
<td>true</td>
<td>FALSE</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>verboseReporting = false</strong></td>
<td></td>
</tr>
<tr>
<td><strong>writeAsciiAssembly</strong></td>
<td>true</td>
<td>FALSE</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>writeAsciiAssembly = false</strong></td>
<td></td>
</tr>
<tr>
<td><strong>singlePartsNoAssem</strong></td>
<td>true</td>
<td>FALSE</td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>singlePartsNoAssem = false</strong></td>
<td></td>
</tr>
<tr>
<td><strong>smartLODgeneration</strong></td>
<td>true</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>smartLODgeneration = true</strong></td>
<td></td>
</tr>
<tr>
<td><strong>autoLowLODgeneration</strong></td>
<td>true</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TRUE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>false</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>autoLowLODgeneration = true</strong></td>
<td></td>
</tr>
<tr>
<td><strong>numLODs</strong></td>
<td>any integer</td>
<td></td>
</tr>
<tr>
<td></td>
<td>numLODs = 3</td>
<td></td>
</tr>
<tr>
<td><strong>close brace</strong></td>
<td>}</td>
<td></td>
</tr>
</tbody>
</table>
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

<table>
<thead>
<tr>
<th>Option name</th>
<th>Keywords</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOD</td>
<td>LOD &quot;lod number&quot;</td>
<td>LOD &quot;1&quot; {</td>
</tr>
<tr>
<td>Level</td>
<td>any integer</td>
<td>Level = 1</td>
</tr>
<tr>
<td>Chordal</td>
<td>any number</td>
<td>Chordal = 0.001</td>
</tr>
<tr>
<td>Angular</td>
<td>any number</td>
<td>Angular = 25</td>
</tr>
<tr>
<td>Length</td>
<td>any number</td>
<td>Length = 1</td>
</tr>
<tr>
<td>FeatureSuppression</td>
<td>any integer</td>
<td>FeatureSuppression = 0</td>
</tr>
<tr>
<td>Simplify</td>
<td>any number</td>
<td>Simplify = 0.60</td>
</tr>
<tr>
<td>close brace</td>
<td>}</td>
<td>}</td>
</tr>
</tbody>
</table>
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

<table>
<thead>
<tr>
<th>Option name</th>
<th>Keywords</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter</td>
<td>Filter {</td>
<td>Filter {</td>
</tr>
<tr>
<td>FilenameSanitizeSet</td>
<td>&quot;string of characters&quot;</td>
<td>FilenameSanitizeSet = &quot;abc123.&quot;</td>
</tr>
<tr>
<td>FilenameSanitizeSetAdd</td>
<td>&quot;string of characters&quot;</td>
<td>FilenameSanitizeSetAdd = &quot;4l&quot;</td>
</tr>
<tr>
<td>FilenameSanitizeSetDelete</td>
<td>&quot;string of characters&quot;</td>
<td>FilenameSanitizeSetDelete = &quot;c&quot;</td>
</tr>
<tr>
<td>MetadataKey</td>
<td>&quot;string of characters&quot;</td>
<td>MetadataKey = &quot;metadata key to exclude&quot;</td>
</tr>
<tr>
<td>close brace</td>
<td>}</td>
<td>}</td>
</tr>
</tbody>
</table>
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

<table>
<thead>
<tr>
<th>Option name</th>
<th>Keywords</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Metadata</td>
<td>Metadata {</td>
<td>Metadata {</td>
</tr>
<tr>
<td>AddToParts</td>
<td>&quot;string of characters&quot;</td>
<td>AddToParts = &quot;&lt;metadata key&gt;&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>AddToParts = &quot;&lt;metadata value&gt;&quot;</td>
</tr>
<tr>
<td>AddToAssemblies</td>
<td>&quot;string of characters&quot;</td>
<td>AddToAssemblies = &quot;&lt;metadata key&gt;&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>AddToAssemblies = &quot;&lt;metadata value&gt;&quot;</td>
</tr>
<tr>
<td>AddToAllNodes</td>
<td>&quot;string of characters&quot;</td>
<td>AddToAllNodes = &quot;&lt;metadata key&gt;&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>AddToAllNodes = &quot;&lt;metadata value&gt;&quot;</td>
</tr>
<tr>
<td>close brace</td>
<td>}</td>
<td>}</td>
</tr>
</tbody>
</table>