



User Guide

CREO - JT

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Overview of TRANSLATE

About Theorem



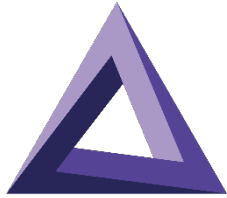
Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of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

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

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

Theorem's Product Suite

Theorem have 3 main Product brands. These are:

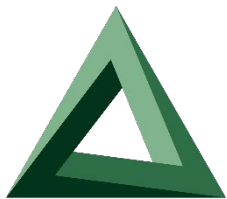


CADTranslate

CADTranslate

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

See our [website](#) for more detail.



CADPublish

CADPublish

The creation of documents enriched with 3D content

See our [website](#) for more detail.



TheoremXR

TheoremXR

Visualization for [Augmented \(AR\)](#), [Mixed \(MR\)](#) and [Virtual \(VR\)](#) Reality applications

See our [website](#) for more detail.

The Creo Bi-directional JT CADverter

The Creo to JT CADverter is a direct database converter between Creo and JT. It enables the user to convert all forms of mechanical design geometry, as well as assembly and attribute information between these two systems.

CADverter can be purchased as a uni-directional, Creo to JT, or JT to Creo product, or as a bi-directional product.

The translator can be invoked in 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

- CADverter converts all geometry
- If assembly data (product structure) is in the file, the assembly structure will be mapped between the two systems as well as colour information
- The user can filter data to optimize the process
- If you wish to visualise and interrogate the JT or Creo data this can be done by using the integrated User Interface, which is included with CADverter
- The conversion process can be run Interactively from the Creo session, in Batch Mode or using the Unified Interface
- Command line interface allows process integration
-

Primary Product benefits?

- Direct conversion between Creo and JT reduces processing time, simplifies integration and retains accuracy of the model
-
- The integrated viewing capability enables visual verification, pre and post translation
- The integrated data filtering options allows selected data ONLY to be processed, enabling optimisation of translations and time savings
-
- By converting all forms of geometry no data is lost, eliminating the time required to recreate missing data
-
- With over 20 years industrial use Theorem's product robustness and quality is well proven, reducing your business risk

This document will focus specifically on guidance for the use of the CADverter for Creo to JT product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

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 Creo – JT product. For information regarding any of Theorem’s product ranges please contact sales@theorem.com

Getting Started

Documentation & Installation Media

The latest copy of the User Guide documentation can be found on our web site at:

<http://www.theorem.com/Documentation>

Each product has a specific link that provides user documentation in the form of PDF and Tutorials.

The latest copy of Theorem software can be found via the link above and by searching for the specific product. Each product has a specific link to the Product Release Document, which contains a link to the download location of the installation CD.

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

Installation

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

License Configuration

To run any product a valid license file is required. The Flex License Manager is run from the .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation

Using the Product

To use the product, follow the documented steps found in this document or follow the online video tutorials which can be found from www.theorem.com/documentation

Using the Product

Default Translations

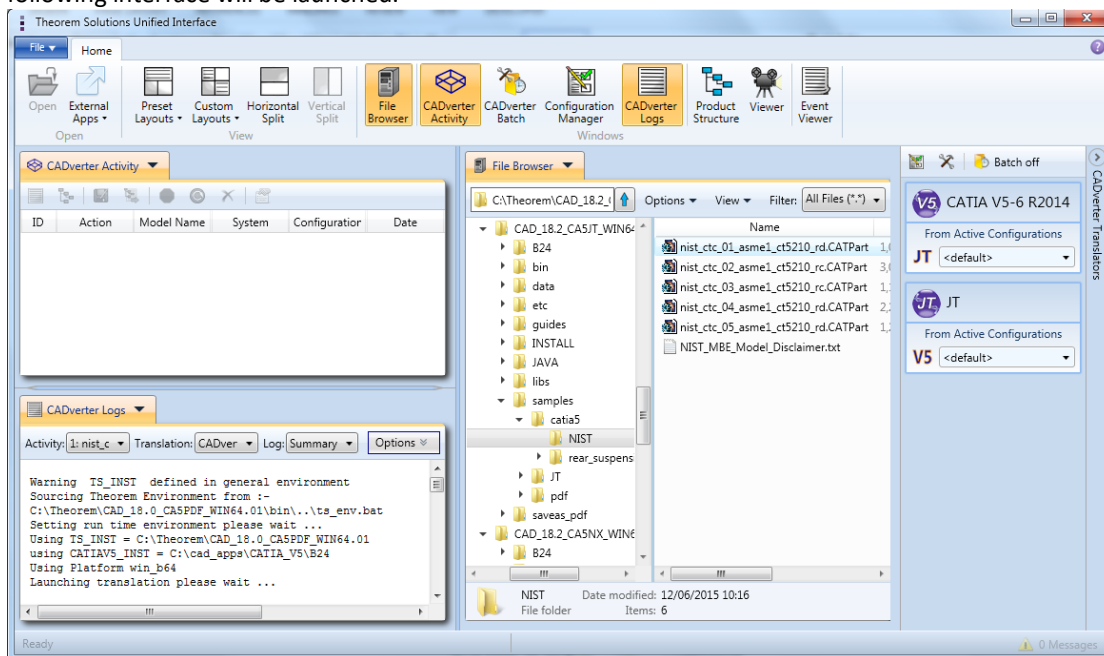
Default Translation – via the Unified Interface

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

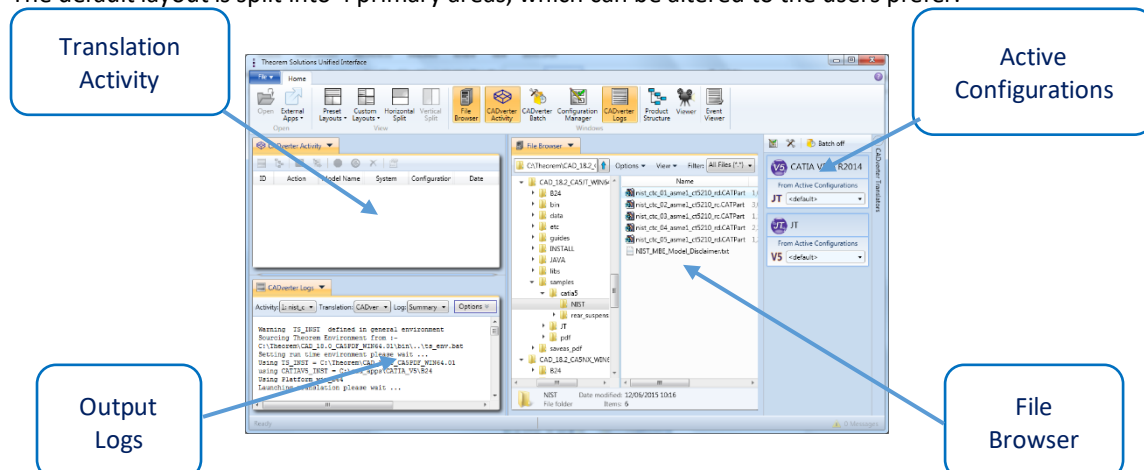
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

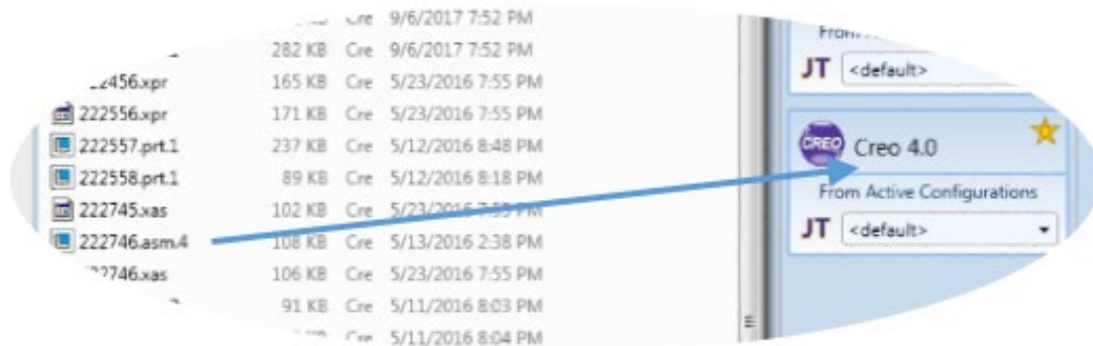
The following interface will be launched:



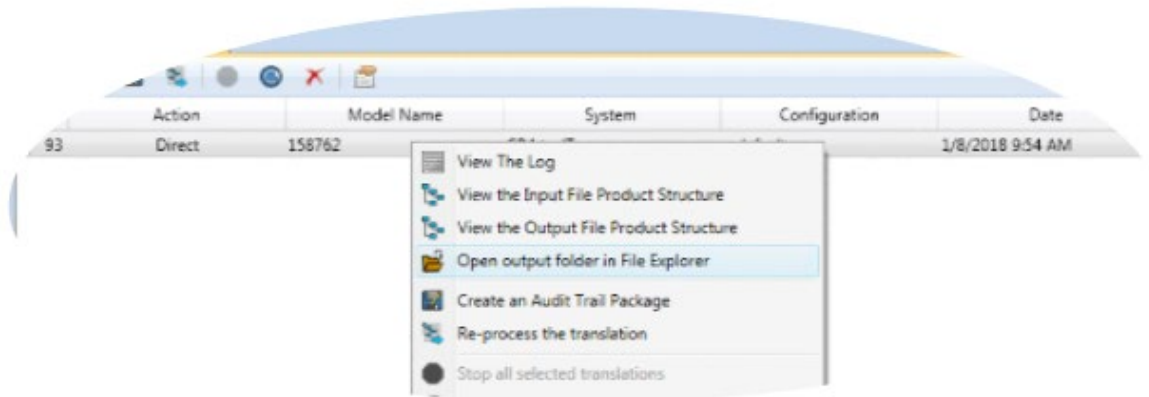
The default layout is split into 4 primary areas, which can be altered to the users prefer:



The simplest way to translate from Creo to 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 the Command Line

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

<Translator_installation_directory>\bin\cad_run.cmd CR[X]_JT -i <input_file> -o <output_file>

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

<Translator_installation_directory>\bin\cad_run.cmd JT_CR[X] -i <input_file> -o <output_file>

Note! Replace the [X] seen in the example with the version of CREO you are using E.g. for CREO 6 change to CR6.

```
C:\Windows\system32\cmd.exe
C:\>"C:\Program Files\Theorem\23.3\bin\cad_run.cmd" CR6_JT -i "C:\Program Files\Theorem\23.3\samples\Creo\Alternator\alternator.asm" -o C:\TEMP\alternator.jt
```

The example above will translate a CREO sample file provided within the installation and produce the following output to the target location. In this case:

C:\Temp\alternator.prt

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 Creo to JT

Within the Configuration Manager panel of the Unified Interface, arguments that can be specified when publishing Creo data into JT are grouped into the following areas:

- **Creo Read** – Those arguments that affect how data is read from JT
- **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

CREO Read Arguments

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

Creo Read JT Write General	
Option Name	Value
Transfer Solids	<input checked="" type="checkbox"/>
Transfer Quilts	<input checked="" type="checkbox"/>
Transfer Datum Curves	<input checked="" type="checkbox"/>
Transfer Datum Surfaces	<input checked="" type="checkbox"/>
Read PMI	<input type="checkbox"/>
Fill PMI Text	<input type="checkbox"/>
Read Cables	<input checked="" type="checkbox"/>
Process SimpRep (Interactive only)	<input type="checkbox"/>
Create Empty Nodes	<input type="checkbox"/>
Read Sub Assembly Colours	<input type="checkbox"/>
Read Part Colours	<input checked="" type="checkbox"/>
Read Surface Colours	<input checked="" type="checkbox"/>
Instance Processing	Off <input type="button" value="v"/>
Instance name	<input type="text"/>

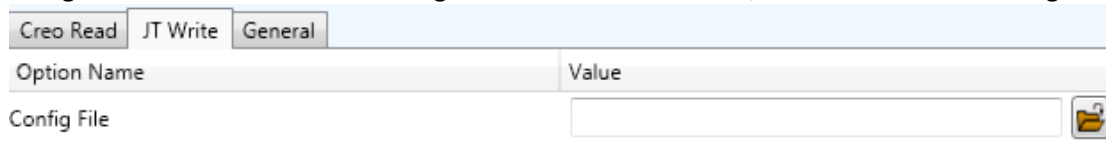
Each of these options is described below:

Option	Description
Transfer Solids	Enables solid processing. (Default is on). <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ no_solids – to Disable
Transfer Quilts	Enables quilt processing. (Default is on). <ul style="list-style-type: none"> • Command Line Syntax:

	<ul style="list-style-type: none"> ○ no_quilts – to Disable
Transfer Datum Curves	<p>Enables Datum Curve processing. (Default is on).</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ no_datum_curves – to Disable
Transfer Datum Surfaces	<p>Enables Datum Surface processing. (Default is on).</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ no_datum_surfaces – to Disable
Read PMI	<p>Enables reading of PMI. (Default is off).</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ read_pmi
Fill PMI Text	<p>Enabled when 'Read PMI' is selected. Improves the quality of PMI, but increases output size and processing time. (Default is off)</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ fill_pmi_text
Read Cables	<p>Enables the reading of Cable data from Creo. (Default is off). Note that in the Creo Configuration Editor, the setting display_thick_cables should be set to yes.</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ read_cables
Process Simprep	<p>Enables the reading of a specified Simplified Representation. This is only available when processing data interactively and using the option via the Configuration Manager.</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ process_simprep
Create Empty Nodes	<p>If a part is missing, create an empty leaf node for it. (Default is off)</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ create_empty_part
Read Sub Assembly Colours	<p>Read colour information set on sub-assembly level (Default is off)</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ sub_assy_colours ○ sub_assy_colours_off (default)
Read Part Colours	<p>Read colour information set on parts. (Default is on)</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ part_colours (default) ○ part_colours_off
Read Surface Colours	<p>Read colour information set on surfaces. (Default is on)</p> <ul style="list-style-type: none"> • Command Line Syntax: <ul style="list-style-type: none"> ○ surface_colours (default) ○ surface_colours_off
Instance Processing	<p>Process a defined instance. Contains 3 options: (Not to be used in conjunction with Process Specified Instance). (Default is off).</p> <ul style="list-style-type: none"> • Off • List all instances in log file - List all instances of Family table to the progress file. <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ instance LIST_ALL • Process Specified Instance – Only process the specified instance. The Text Box Instance Name will become active. <ul style="list-style-type: none"> ○ Command Line Syntax ○ instance [instance_name]
Instance Name	<p>Enter the instance name to process. Only activates when 'Process Specified Instance' selected.</p>

JT Write Arguments

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



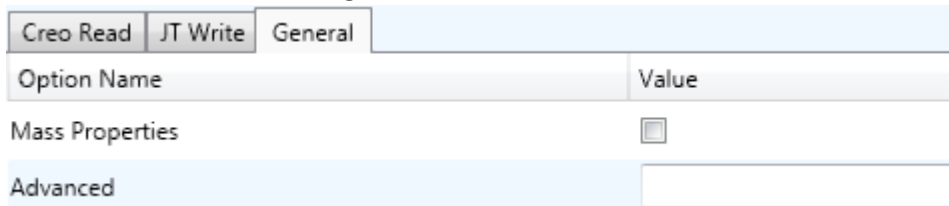
Option Name	Value
Config File	<input type="text"/>

Each of these options is described below:

Option	Description
Config File	<p>Allows a JT configuration file to be specified. Please see Appendix A for a full description of the JT config file format.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>-z [path_to_file]</i>

Creo JT General Arguments

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



Option Name	Value
Mass Properties	<input type="checkbox"/>
Advanced	<input type="text"/>

Each of these options is described below:

Option	Description
Mass Properties	<p>Mass properties (volume/area CofG) are read and any applied materials, using this option, in cases where a part has multiple solids, volume and area values are summed, but CofG data is invalid.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>mprops</i>
Advanced	<p>Allows any of the Command Line Advanced arguments documented to be passed to the Unified Interface invocation.</p>

Command Line Advanced Arguments

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

Creo to JT Advanced Arguments

Creo Read Argument	Description
pmi_pcurves	Store non planar PMI graphics (leaders not in the plane of the annotation)
unique_occ	Read multiple occurrences, eg support for occurrence PMI associations
read_assy_pmi	reads PMI in lower level assembly parts
ignore_std_views	Disable reading views with standard names (TOP, LEFT, etc).
ignore_view_list <file>	supply a list of view names to be ignored
no_exploded_views	Disables exploded views.
part_level_views	Default: off Enable the processing of Part Level Views within an assembly.
part_level_views_moved	Process part level views and move into assembly space, such that only one part instance's views are displayed.
part_level_pmi	Default: off Enable the processing of PMI on parts within an assembly.
view_part_name	Uses the part name in the view names to help identify the views when selected in Creo
views_geom_exploded <on off hybrid hybrid2>	Default: hybrid Geometry grouped into assembly nodes for views is by default done on a view bases, which re-uses geometry where possible. This option creates a node for every item of geometry so that the views can hide/show them in the views. This can (depending upon the data and views) reduce the resulting file size and shorten the translation time. <i>exploded - off</i> - create a node per view with all geometry / wire etc for that view <i>explode - on</i> - create nodes for very solid / wireframe etc that can be referenced by views <i>hybrid (default)</i> - mixture of explode on/off - solids being exploded and wireframe grouped (best compromise) <i>hybrind2</i> - as hybrid with points also grouped

JT Write Argument	Description
Parasolid Tolerant Modelling	A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output is specified. Enables Parasolid tolerant modelling. Default is ON <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>nopstolmodel</i> – to turn off
Factor	Specify the factor level of Parasolid Tolerant Modelling when turned on. Default is 3. <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>pstolmodel 3</i>
Sew Parasolid Bodies	A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output is specified. Enables the sewing of Parasolid bodies. Default is ON <ul style="list-style-type: none"> Command Line Syntax <ul style="list-style-type: none"> <i>nosew</i> – to turn off

Tolerance	<p>Specify the tolerance for the sew command above. Default is 0.01.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>pssew 0.01</i>
Incremental Sewing	<p>Enables incremental sewing when used with Sew Parasolid Bodies. Default is ON.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>no_sew_increm – to turn off</i>
Split Discontinuous Surfaces	<p>A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output specified. Splits discontinuous surfaces. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>brep_prep – to turn on</i> ○ <i>no_brep_prep – to turn off</i>
Force body creation (No check of Parasolid entities)	<p>A secondary option enabled when XT Brep (Theorem) output specified. Removes the checking of Parasolid entities. Default is ON.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>nocheck – (force body creation without checking = Default)</i> ○ <i>check – (doesn't force the body creation - Parasolid checking is enabled)</i>
Fix Degenerate Edges	<p>A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output specified. On face create failure, check and fix any degenerate edges. Default is ON.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>fix_degen</i> ○ <i>no_fix_degen – to turn off</i>
Specify a Face Edge Tolerance	<p>A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output specified. Specify an edge tolerance to be used when creating faces. Default is ON.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>Please see Edge Tolerance below</i>
Edge Tolerance	<p>A secondary option used with Specify a Face Edge Tolerance where the tolerance value is assigned. Default is 0.000006.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>face_edge_tol 0.000006</i>
Fix small features in solids	<p>A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output specified. Remove small edges, sliver and spike faces from solid bodies. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>ps_fix_small – to turn on</i> ○ <i>no_ps_fix_small - default</i>
Fix small features in open solids	<p>A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output specified. Remove small edges, sliver and spike faces from open solids. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>ps_fix_osol – to turn on</i> ○ <i>no_ps_fix_osol - default</i>
Simplify Geometry	<p>A secondary option to be used when <i>Brep Type = XT Brep (Theorem)</i> output specified. Simplify Geometry. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>simplify_solids – to turn on</i>
Verbose Report	<p>Defines the report. Default is 'Config File setting'</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>Config File Setting: Default</i> ○ <i>No: -VerboseReporting false</i>

<p>Output Units</p>	<ul style="list-style-type: none"> ○ Yes: -VerboseReporting true <p>Output unit definition. Default is 'Config File setting'</p> <ul style="list-style-type: none"> ● Command Line Syntax <ul style="list-style-type: none"> ○ Config File Setting: Default ○ As Input: -OutputUnits <inputUnits> ○ Millimeters: -OutputUnits mm ○ Centimeters: -OutputUnits cm ○ Meters: -OutputUnits m ○ Inches: -OutputUnits inches ○ Feet: -OutputUnits feet ○ Yards: -OutputUnits yards
<p>Structure Output Type</p>	<p>Specifies the type of assembly structure to be output. Default is JT.</p> <ul style="list-style-type: none"> ● Selectable options are: <ul style="list-style-type: none"> <i>JT: Default</i> <i>PLMXML (with external references to JT files)</i> <i>STEP BOM (with external references to JT files)</i> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ JT: Default ▪ PLMXML: <See PLMXML Ref Type> ▪ STEP BOM: write_stepbom
<p>PLMXML Reference Type</p>	<p>Only active if a Structure Type of PLMXML is selected.</p> <ul style="list-style-type: none"> ● Selectable options are: <ol style="list-style-type: none"> 1. <i>PLMXML referencing JT Parts in the same folder</i> 2. <i>PLMXML in addition to the JT Assembly File</i> <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ 1: plmxml_only ▪ 2: write_plmxml
<p>PLMXML Property Mapping File</p>	<p>Allows a Property Mapping file to be selected. Only active if a Structure Type of PLMXML is selected. (See Appendix B for the file format)</p> <ul style="list-style-type: none"> ● Command Line Syntax <ul style="list-style-type: none"> ○ plmxml_prop_map_file [File] <p>Note! When this option is unset, the file "plmxml_property_mapping.txt" in the 'data\jt folder' will be used as the mapping file. This file contains lines with mappings that are mandatory for certain downstream applications and mappings to remove attributes used solely in the translation process. Therefore it is a good idea to start with a copy of this file when creating a new mapping file.</p>
<p>Brep Type</p>	<p>Specifies the BREP type in the resultant JT Files. Default is XT Brep.</p> <ul style="list-style-type: none"> ● Selectable options are: <ul style="list-style-type: none"> <i>JT Brep</i> <i>XT Brep (Theorem)</i> <i>JT Brep (JT Open)</i> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ JT Brep: Default ▪ XT Brep (Theorem): xt_brep no_fixup

	<ul style="list-style-type: none"> ▪ <i>JT Brep (JT open): jt_xt_brep</i>
Explode Solids to Faces	<p>A secondary option enabled when XT Brep (Theorem) output specified. Explodes solids to faces. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>split_brep</i>
Brep Wireframe	<p>Store wireframe on JT Brep. Default is YES.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>Yes: Default</i> ○ <i>No (Tessellated): no_brep_wire</i>
Produce Tessellated Output	<p>Convert Brep data directly to faceted data. Default is OFF</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>tess_output</i>
Expand Part	<p>Process multi-solid parts as an assembly. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>expand_part</i>
Reuse Solids	<p>Used with Expand Part to re-use existing solids. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>reuse_solids</i>
CAD Property Mapping File	<p>A file containing a list of CAD properties and information on how they are mapped to the JT file. Not used by default.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>cad_prop_map_file [File]</i>

JT to Creo Advanced Arguments

JT Read Option	Description
Small Curves	<p>Report Small curves as errors. (default OFF)</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>small_curves</i> (to enable)
Extend Nurb Surfaces	<p>Extends NURBS surfaces beyond face limits for curve projection (default state)</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>no_extend_nurb</i> -(Don't extend NURBS surfaces to face limits) ○ <i>extend_nurb <int></i> - (trims NURBS surfaces to <int> * 0.0001 face extents in u and v)
Remove Groups	<p>Remove Group entities into assembly structure. (default OFF)</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>remove_groups</i> (to enable)
Use Ref Name	<p>Uses file name from input system to name files (default OFF)</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>use_ref_name</i> (to enable)
Re-tessellate Brep Data	<p>Allows Brep data to be re-tessellated to create a link between the faces in the Brep and the tessellated representation. There is a choice between processing just the tessellated form or both the tessellated and Brep forms with the links between them. Default is 'Off'.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>On – Re-tessellate Brep Data: tess_brep</i>

	<ul style="list-style-type: none"> ○ <i>On – Re-tessellate and Read Brep Data:</i> <i>tess_and_read_brep</i>
Read as Bounding Boxes	<p>This option allows the user to read the JT file as a very light weight bounding box representation, with each solid having its own bounding box.</p> <p>This option may be useful when the part's details are not of interest, as much as the <u>physical space</u>, the part occupies.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>bounding_box</i>
Exclude small solids (by absolute value)	<p>Exclude solids that have bounding boxes smaller than value. Default is OFF.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>bb_exclude_value</i>
Exclude small solids (by bounding box fraction)	<p>This option allows the user to exclude small solids from the JT read, the excluded solids being smaller than a specified fractional size of the overall bounding box of the part.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>bounding_box_frac [value]</i>
Exclude parts File for bounding box checks	<p>This option allows the user to supply an input file containing a list of JT parts, one per line. The parts specified in the file will be excluded from the checks to remove small solids. This option is used in conjunction with the "Exclude Small Solids" options.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>bb_exclusion_file '[path_to_file]</i>
Creo Write Option	Description
Collapse Assembly Structure	<p>If the Input CAD data contains any assembly structure, then by default assembly structure will be created in the Output CAD format. Running this option causes the assembly structure to be "exploded" into a flat single component file.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>noditto</i>
Group All Geometry	<p>Creo default is to create one part per Solid body. This option will allow all Geometry to be written into one part.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>mult_feat_on</i> ○ <i>mult_feat_off</i> (default)
Group Open Solids into a Single Creo Part	<p>This option will allow all "Open Solid" (Quilt) Geometry to be written into one part.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>mult_open_on</i> ○ <i>mult_open_off</i> (default)
Group Closed Solids into a Single Creo Part	<p>This option will allow all "Closed Solid" Geometry to be written into one part.</p> <ul style="list-style-type: none"> • Command Line Syntax <ul style="list-style-type: none"> ○ <i>mult_brep_on</i>

- *mult_brep_off* (default)

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`.

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

tess.config in the directory where the translator is run

tess.config in TS_INST\etc directory

Two example config files are provided in the **TS_INST\etc** directory, a

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

The Setup Section

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

To edit setup options

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

Option name	Keywords	Example
EAITranslator	EAITranslator {	EAITranslator {
OutputDirectory	"path to directory"	OutputDirectory = "/home/<user>/"
CommonPartsPath	"path to directory"	CommonPartsPath= "/myaccount/jtparts/"
chordalOption	"RELATIVE" "ABSOLUTE"	chordalOption = "RELATIVE"
structureOption	"PER_PART" "MONOLITHIC" "FULL_SHATTER"	structureOption = "MONOLITHIC"
WriteWhichFiles	"ALL" "ASSEMBLY_ONLY" "PARTS_ONLY"	WriteWhichFiles = "ALL"
compression	true TRUE false FALSE	compression = true
advCompression	true TRUE	advcompression = true

	false FALSE	
advCompressionLevel	<i>any number</i>	advCompressionLevel = 0.0
JtFileFormat	<i>Any JT file version</i>	JtFileFormat = "95"
triStripOpt	true TRUE false FALSE	triStripOpt = false
seamSewing	true TRUE false FALSE	seamSewing = true
seamSewingTol	<i>any number</i>	seamSewingTol = 0.001
includeBrep	true TRUE false FALSE	includeBrep = true
IncludeGeom	true TRUE false FALSE	includeGeom = true
autoXtBrep	true TRUE false FALSE	autoXtBrep = false
brepPrecision	"SINGLE" "DOUBLE"	brepPrecision = "SINGLE"
autoNameSanitize	true	autoNameSanitize = true

	TRUE false FALSE	
nameSanitizeMacro	<i>Macro name</i>	nameSanitizeMacro = ""
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
autoLowLODgeneration	true TRUE false FALSE	autoLowLODgeneration = true
smartLODgeneration	true TRUE false FALSE	smartLODgeneration = true

numLODs	<i>any integer</i>	numLODs = 3
includeULP	<i>PASSTHROUGH</i>	includeULP = "PASSTHROUGH"
ulpPrecision	<i>any number</i>	ulpPrecision = 0.001
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	<i>any integer</i>	Level = 1
Chordal	<i>any number</i>	Chordal = 0.001
Angular	<i>any number</i>	Angular = 25
Length	<i>any number</i>	Length = 1
FeatureSuppression	<i>any integer</i>	FeatureSuppression = 0
Simplify	<i>any number</i>	Simplify = 0.60
AdvCompressionLevel	<i>any number</i>	AdvCompressionLevel = 0.0
ULP	<i>True / false</i>	ULP = false
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	<i>"string of characters"</i>	FilenameSanitizeSet = "abc123."
FilenameSanitizeSetAdd	<i>"string of characters"</i>	FilenameSanitizeSetAdd = "41"

FilenameSanitizeSetDelete	<i>"string of characters"</i>	FilenameSanitizeSetDelete = "c"
MetadataKey	<i>"string of characters"</i>	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	<i>"string of characters"</i>	AddToParts = "<metadata key>" AddToParts = "<metadata value>"
AddToAssemblies	<i>"string of characters"</i>	AddToAssemblies = "<metadata key>" AddToAssemblies = "<metadata value>"
AddToAllNodes	<i>"string of characters"</i>	AddToAllNodes = "<metadata key>" AddToAllNodes = "<metadata value>"
close brace	}	}

Appendix B – Property Mapping Files

Property mapping files are required for CAD property Mapping and PLMXML Property Mapping

A Property Mapping File is a comma separated text file containing information of how CAD properties from the source system will be mapped into the target file.

The format is as follows:

- Lines beginning with a "#" are treated as comment lines and are ignored.
- Any space characters will be treated as part of the item
- Lines containing a mapping must contain 6 items separated by 5 commas

The six items are :-

Item	Description
Source name	The attribute name in the Source System
Target name	The attribute name in the Target File
Data derived from	0 - Do not convert 1 - Use the source value as given 6 - Use the source value as given and hide the property Note! Value 6 For CAD Mapping Files ONLY (Not PLMXML)
Default Value	Not currently used
Value Type	Not currently used
Default Units	Not currently used


An Example of a mapping file is shown below:-

```
# Mapping from input attribute name to Target property name
#
# Line Format:-
# Source name,Target name,Data derived from,Default Value,Value Type,Default Units
# Data derived from:-
# 0 - Do not convert
# 1 - Use the source values as given
# 6 - Use the source value as given and hide the property
#
_ActivateBOM,NULL,0,0,,
_LastModifier,NULL,0,0,,
_Maturity,NULL,0,0,,
_PrdVersion,NULL,0,0,,
_ReferenceTimeStamp,NULL,0,0,,
_Responsible,NULL,0,0,,
COG M,ud_CAD_CENTER_OF_GRAVITY,1,0,,
COMPONENTS PRINCIPAL AXES ,NULL,0,0,,
DENSITY Kg/M^3,NULL,0,0,,
INERTIA MATRIX KgM2,ud_CAD_MOMENT_OF_INERTIA,1,0,,
INERTIA VOLUME M^3,ud_CAD_VOLUME,1,0,,
INERTIA WET AREA M^2,ud_CAD_SURFACE_AREA,1,0,,
MASS Kg,ud_CAD_MASS,1,0,,
PRINCIPAL MOMENTS KgM^2,NULL,0,0,,
FILESAVETIME,File Last Modified,1,0,,
LOCALE,LOCALE,1,0,,
Masterdata Version,Masterdata Version,1,0,,
```



Material Details,Material Details,1,0,,
PART_NUMBER,PART_NUMBER,1,0,,
MPARTNAME,Source Model Name,1,0,,
Source,SourceName,1,0,,




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