USER GUIDE

Translate for SOLIDWORKS – JT

Product Release Version 25.1

Revision: 1.0

Issued: 03/05/2022

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, Defence, 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 SOLIDWORKS - JT Translator

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

Theorem's Visualise 3D product for SolidWorks to JT is a direct converter between SolidWorks assemblies (SLDASM. files) and part (.sldprt files) and Siemens JT parts. It enables the user to convert all forms of mechanical design geometry, as well as assembly and attribute information, between these two systems.

The SolidWorks-JT Visualise 3D can be used interactively or in a batch mode, from a standard GUI Interface, offering combined viewing, data filtering and translation capabilities.

Primary Product Features

- Converts all types of geometry, wire frame, surfaces, trimmed surfaces (faces) and solid models
- Converts assembly structure between the systems
- Converts attribute data including colour and layer information
- The conversion process can be run Interactively or in Batch mode
- Data can be filtered by layer and entity type
- Geometry can be filtered and selectively processed

Primary Product benefits?

- Direct conversion between SolidWorks 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 Visualise 3D for SolidWorks-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

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

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.

Currently, there are 2 distinct installation stages that are required.

Currently, there are 2 distinct installation stages that are required.

To install the translator, select the ***CAD_25.1_SWJT_WIN.01.msi*** file and follow the installation process. For a full guide to the process, please see our 'Translator Installation Process' demonstration video located here.

In addition, the Theorem Unified Interface will also need to be installed. The installation process is the same as for the Translator. For a full guide to the process, please see our 'Translator Installation Process' demonstration video located here.

Note: If you need to install on to a single central network location, there is a full installation guide documenting how to do this 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.

- Via the Theorem Unified Interface
  - The Unified Interface offers a Desktop Environment that allows CAD and Visualization data to be viewed pre and post translation
  - For a full guide to this process, please see our 'How to Translate Using the SolidWorks – JT Translator via the User Interface' demonstration video located here.

- Via the the Command Line
  - The Command Line Interface provides a direct method of invoking the translator. It can be used via a DOS shell or called via a third party application as part of a wider process requirement.
  - For a full guide to this process, please see the "Default Translation – via the Command Line" section on page 8

- Interactively from within SolidWorks
  - The Interactive Interface provides a direct method of Translating SolidWorks data to JT and vice versa.
  - For a full guide to this process, please see our 'How to Translate Using the SolidWorks – JT Translator in Interactive Mode' demonstration video located here.

Using the Product

Default Translations

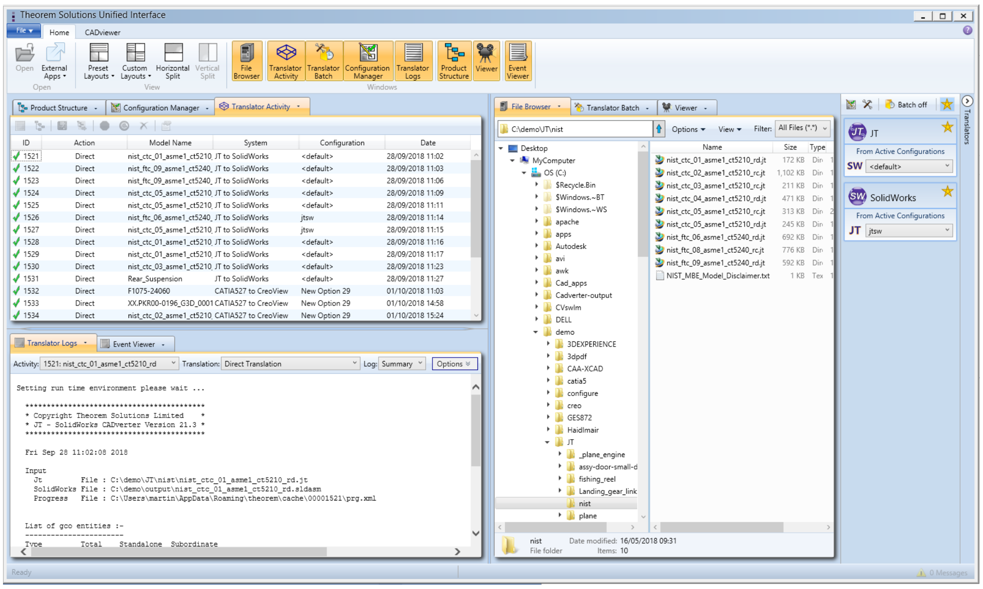
Default Translation – via the Unified Interface

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:



ID	Action	Model Name	System	Configuration	Date
✓ 1521	Direct	nist_ctc_01_asme1_ct5210_	JT to SolidWorks	<default>	28/09/2018 11:02
✓ 1522	Direct	nist_ftc_09_asme1_ct5240_	JT to SolidWorks	<default>	28/09/2018 11:03
✓ 1523	Direct	nist_ftc_09_asme1_ct5240_	JT to SolidWorks	<default>	28/09/2018 11:06
✓ 1524	Direct	nist_ctc_05_asme1_ct5210_	JT to SolidWorks	<default>	28/09/2018 11:09
✓ 1525	Direct	nist_ctc_05_asme1_ct5210_	JT to SolidWorks	<default>	28/09/2018 11:11
✓ 1526	Direct	nist_ftc_06_asme1_ct5240_	JT to SolidWorks	jtsw	28/09/2018 11:14
✓ 1527	Direct	nist_ctc_05_asme1_ct5210_	JT to SolidWorks	jtsw	28/09/2018 11:15
✓ 1528	Direct	nist_ctc_01_asme1_ct5210_	JT to SolidWorks	<default>	28/09/2018 11:16
✓ 1529	Direct	nist_ctc_01_asme1_ct5210_	JT to SolidWorks	<default>	28/09/2018 11:17
✓ 1530	Direct	nist_ctc_03_asme1_ct5210_	JT to SolidWorks	<default>	28/09/2018 11:23
✓ 1531	Direct	Rear_Suspension	JT to SolidWorks	<default>	28/09/2018 11:27
✓ 1532	Direct	F1075-24060	CATIA527 to CreoView	New Option 29	01/10/2018 11:03
✓ 1533	Direct	XX.PKR00-0196_G3D_0001	CATIA527 to CreoView	New Option 29	01/10/2018 14:58
✓ 1534	Direct	nist_ctc_02_asme1_ct5210_	CATIA527 to CreoView	New Option 29	01/10/2018 15:24

Activity: 1521: nist_ctc_01_asme1_ct5210_rd Translation: Direct Translation Log: Summary Options

```
Setting run time environment please wait ...

  *****************************************
  * Copyright Theorem Solutions Limited    *
  * JT - SolidWorks CADverter Version 21.3 *
  *****************************************

  Fri Sep 28 11:02:08 2018

  Input
    Jt         File : C:\demo\JT\nist\nist_ctc_01_asme1_ct5210_rd.jt
    SolidWorks File : C:\demo\output\nist_ctc_01_asme1_ct5210_rd.sldasm
    Progress   File : C:\Users\martin\AppData\Roaming\theorem\cache\00001521\prg.xml


  List of gco entities :-
  -----------------------
  Type          Total    Standalone  Subordinate
```

C:\demo\JT\nist Options View Filter: All Files (*.*)

Desktop
- MyComputer
  - OS (C:)
    - $Recycle.Bin
    - $Windows.~BT
    - $Windows.~WS
    - apache
    - apps
    - Autodesk
    - avi
    - awk
    - Cad_apps
    - Cadverter-output
    - CVswim
    - DELL
    - demo
      - 3DEXPERIENCE
      - 3dpdf
      - CAA-XCAD
      - catia5
      - configure
      - creo
      - GES872
      - Haidlmair
      - JT
        - _plane_engine
        - assy-door-small-d
        - fishing_reel
        - Landing_gear_link
        - nist
        - plane

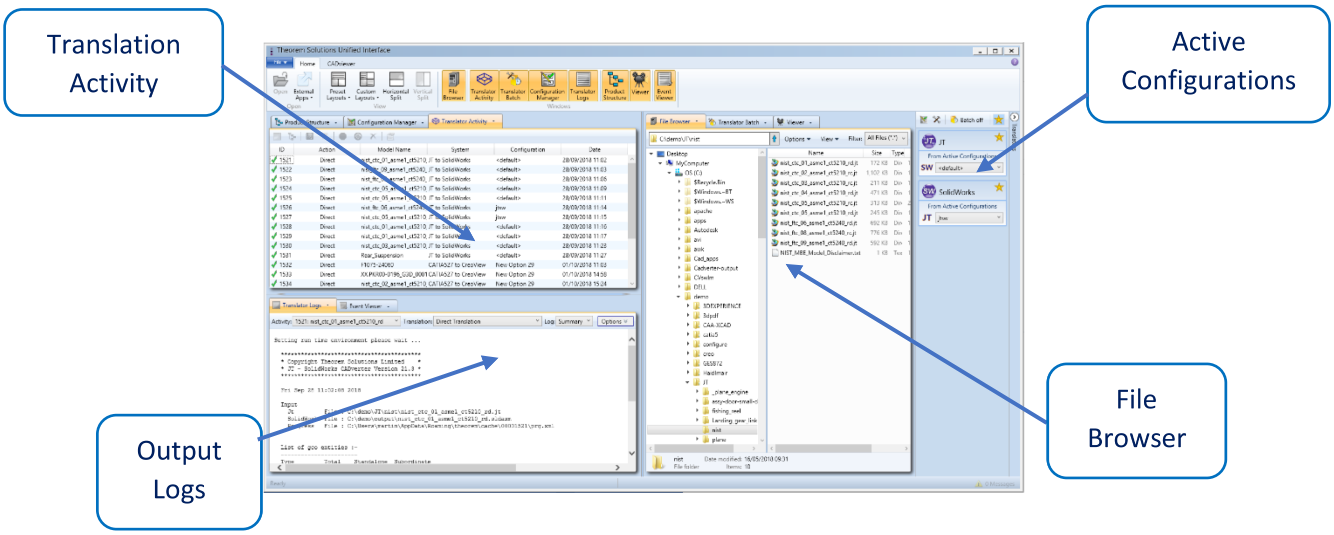
Name	Size	Type
nist_ctc_01_asme1_ct5210_rd.jt	172 KB	Din
nist_ctc_02_asme1_ct5210_rc.jt	1,102 KB	Din
nist_ctc_03_asme1_ct5210_rc.jt	211 KB	Din
nist_ctc_04_asme1_ct5210_rd.jt	471 KB	Din
nist_ctc_05_asme1_ct5210_rc.jt	313 KB	Din
nist_ctc_05_asme1_ct5210_rd.jt	245 KB	Din
nist_ftc_06_asme1_ct5240_rd.jt	692 KB	Din
nist_ftc_08_asme1_ct5240_rc.jt	776 KB	Din
nist_ftc_09_asme1_ct5240_rd.jt	592 KB	Din
NIST_MBE_Model_Disclaimer.txt	1 KB	Tex

nist Date modified: 16/05/2018 09:31
File folder Items: 10

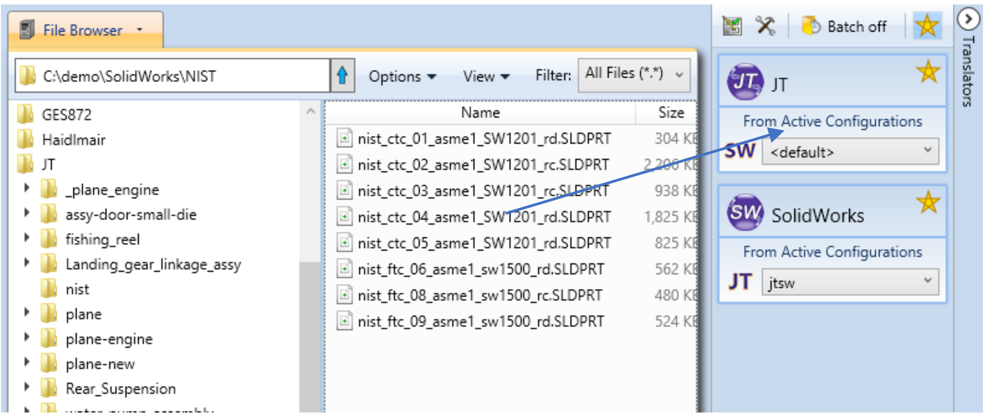
JT
From Active Configurations
SW <default>

SolidWorks
From Active Configurations
JT jtsw

The default layout is split into 4 primary areas that can be altered to the user’s preference:



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



C:\demo\SolidWorks\NIST

- GES872
- Haidlmair
- JT
  - _plane_engine
  - assy-door-small-die
  - fishing_reel
  - Landing_gear_linkage_assy
  - nist
  - plane
  - plane-engine
  - plane-new
  - Rear_Suspension

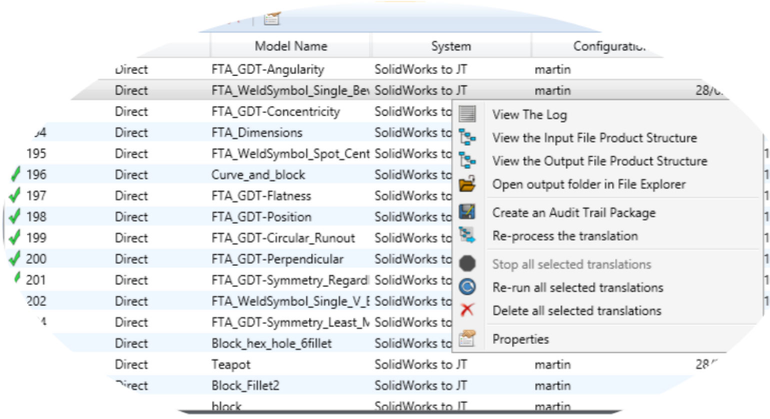
Name	Size
nist_ctc_01_asme1_SW1201_rd.SLDPRT	304 KE
nist_ctc_02_asme1_SW1201_rc.SLDPRT	2,206 KE
nist_ctc_03_asme1_SW1201_rc.SLDPRT	938 KE
nist_ctc_04_asme1_SW1201_rd.SLDPRT	1,825 KE
nist_ctc_05_asme1_SW1201_rd.SLDPRT	825 KE
nist_ftc_06_asme1_sw1500_rd.SLDPRT	562 KE
nist_ftc_08_asme1_sw1500_rc.SLDPRT	480 KE
nist_ftc_09_asme1_sw1500_rd.SLDPRT	524 KE

JT
From Active Configurations
SW <default>

SolidWorks
From Active Configurations
JT jtsw

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

The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:



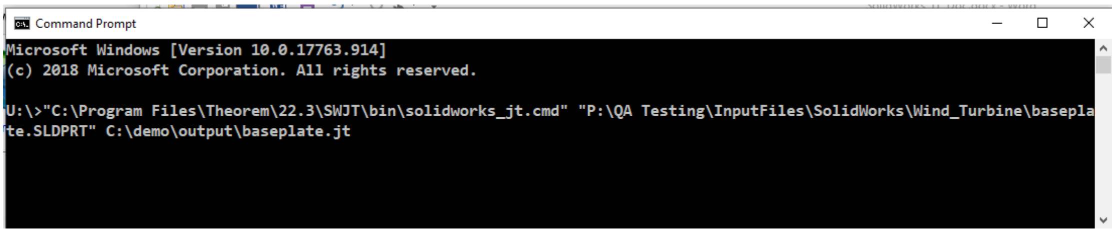
		Model Name	System	Configuratio	
	Direct	FTA_GDT-Angularity	SolidWorks to JT	martin	
	Direct	FTA_WeldSymbol_Single_Be	SolidWorks to JT	martin	28/
	Direct	FTA_GDT-Concentricity	SolidWorks to		
4	Direct	FTA_Dimensions	SolidWorks to		
195	Direct	FTA_WeldSymbol_Spot_Cent	SolidWorks to		10.
✓ 196	Direct	Curve_and_block	SolidWorks to		10:0.
✓ 197	Direct	FTA_GDT-Flatness	SolidWorks to		10:09
✓ 198	Direct	FTA_GDT-Position	SolidWorks to		10:09
✓ 199	Direct	FTA_GDT-Circular_Runout	SolidWorks to		10:09
✓ 200	Direct	FTA_GDT-Perpendicular	SolidWorks to		10:09
✓ 201	Direct	FTA_GDT-Symmetry_Regard	SolidWorks to		10:0
202	Direct	FTA_WeldSymbol_Single_V_E	SolidWorks to		1
4	Direct	FTA_GDT-Symmetry_Least_M	SolidWorks to		
	Direct	Block_hex_hole_6fillet	SolidWorks to		
	Direct	Teapot	SolidWorks to JT	martin	28
	Direct	Block_Fillet2	SolidWorks to JT	martin	
		block	SolidWorks to JT	martin	

View The Log
View the Input File Product Structure
View the Output File Product Structure
Open output folder in File Explorer
Create an Audit Trail Package
Re-process the translation
Stop all selected translations
Re-run all selected translations
Delete all selected translations
Properties

Default Translation – via the Command Line SolidWorks-JT

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

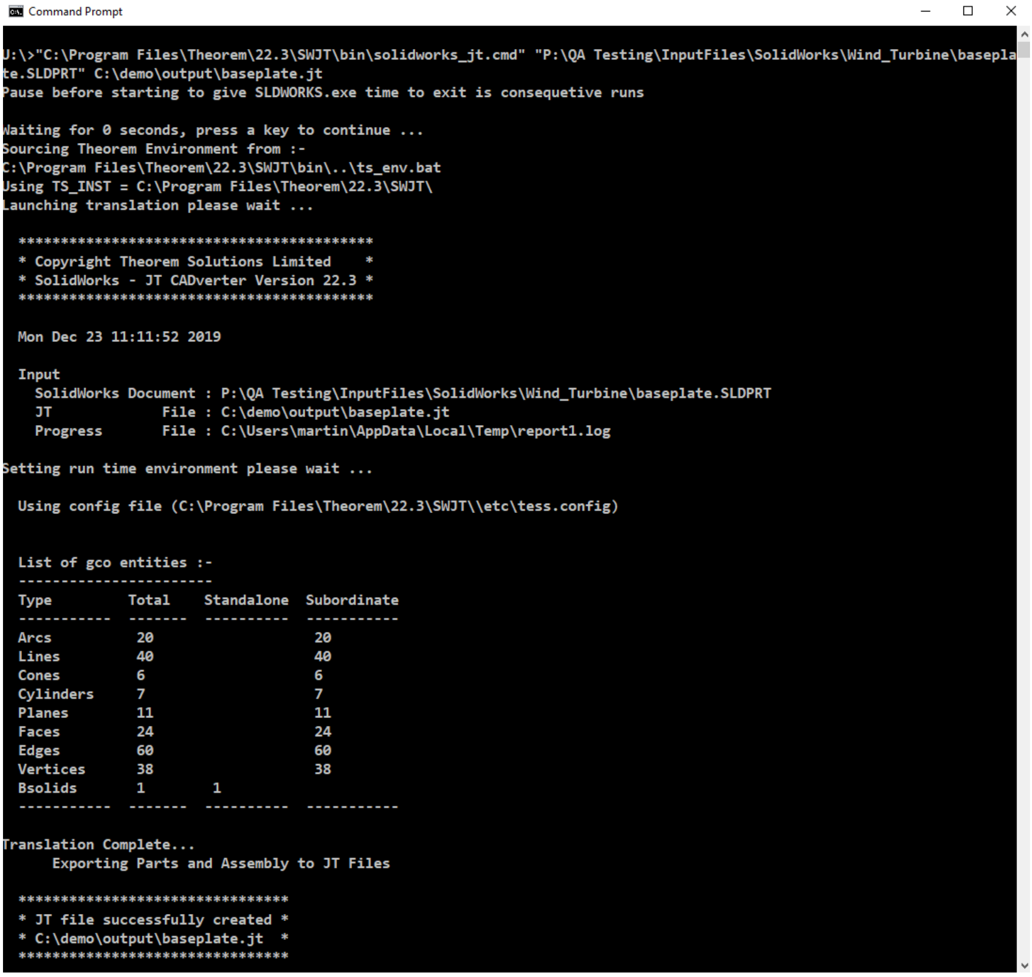
<Translator_installation_directory>\bin\solidworks_jt.cmd <input_file> <output_file>



```
Microsoft Windows [Version 10.0.17763.914]
(c) 2018 Microsoft Corporation. All rights reserved.

U:\>"C:\Program Files\Theorem\22.3\SWJT\bin\solidworks_jt.cmd" "P:\QA Testing\InputFiles\SolidWorks\Wind_Turbine\basepla
te.SLDPRT" C:\demo\output\baseplate.jt
```

The example above will translate an SolidWorks sample file provided within the installation and produce the following screen output:



```
U:\>"C:\Program Files\Theorem\22.3\SWJT\bin\solidworks_jt.cmd" "P:\QA Testing\InputFiles\SolidWorks\Wind_Turbine\basepla
te.SLDPRT" C:\demo\output\baseplate.jt
Pause before starting to give SLDWORKS.exe time to exit is consequetive runs

Waiting for 0 seconds, press a key to continue ...
Sourcing Theorem Environment from :-
C:\Program Files\Theorem\22.3\SWJT\bin\..\ts_env.bat
Using TS_INST = C:\Program Files\Theorem\22.3\SWJT\
Launching translation please wait ...

  ******************************************
  * Copyright Theorem Solutions Limited    *
  * SolidWorks - JT CADverter Version 22.3 *
  ******************************************

  Mon Dec 23 11:11:52 2019

  Input
    SolidWorks Document : P:\QA Testing\InputFiles\SolidWorks\Wind_Turbine\baseplate.SLDPRT
    JT             File : C:\demo\output\baseplate.jt
    Progress       File : C:\Users\martin\AppData\Local\Temp\report1.log

Setting run time environment please wait ...

  Using config file (C:\Program Files\Theorem\22.3\SWJT\\etc\tess.config)


  List of gco entities :-
  -----------------------
  Type         Total    Standalone  Subordinate
  ----------- -------  ----------  -----------
  Arcs          20                   20
  Lines         40                   40
  Cones         6                    6
  Cylinders     7                    7
  Planes        11                   11
  Faces         24                   24
  Edges         60                   60
  Vertices      38                   38
  Bsolids       1        1
  ----------- -------  ----------  -----------

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

  ********************************
  * JT file successfully created *
  * C:\demo\output\baseplate.jt  *
  ********************************
```

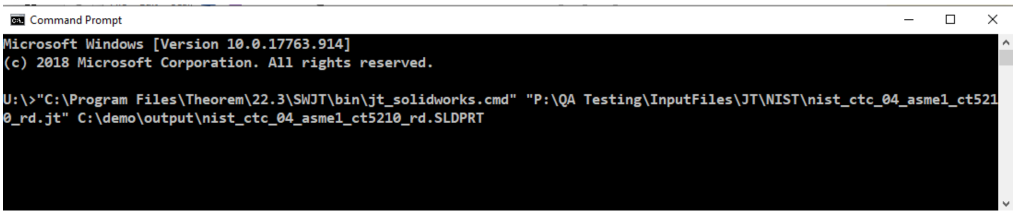
The file will be output to the target location. In this case:

C:\demo\output\baseplate.jt

JT-SolidWorks

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

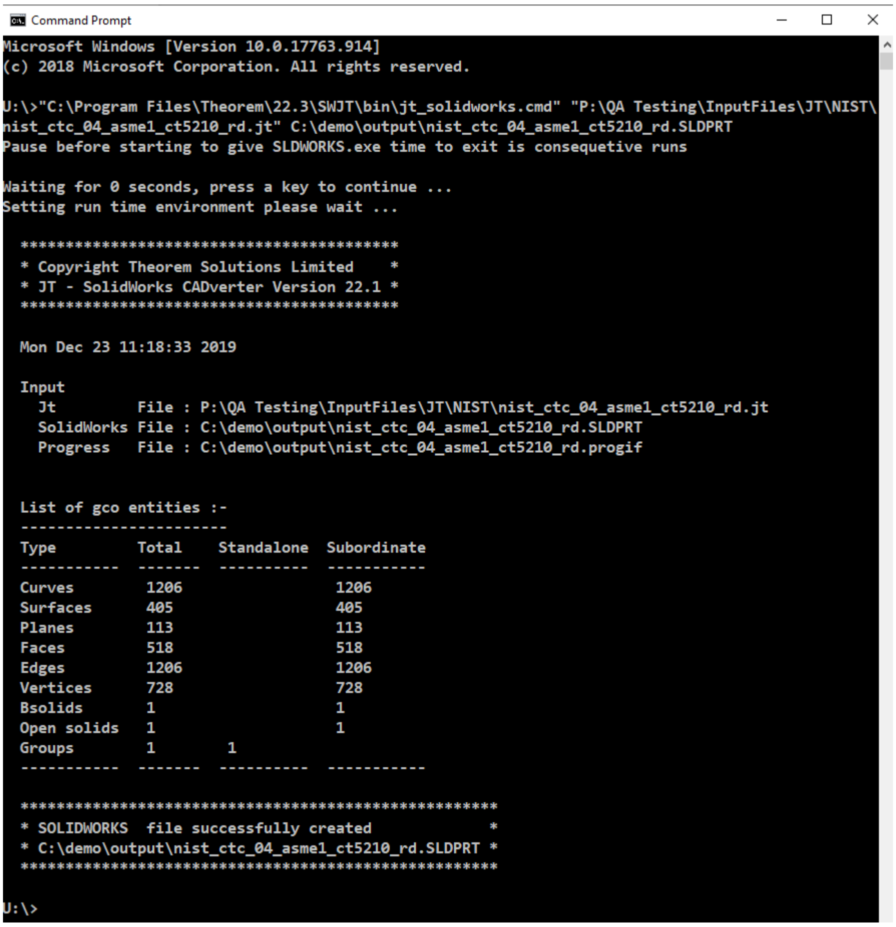
<Translator_installation_directory>\bin\jt_solidworks.cmd <input_file> <output_file>



```
Microsoft Windows [Version 10.0.17763.914]
(c) 2018 Microsoft Corporation. All rights reserved.

U:\>"C:\Program Files\Theorem\22.3\SWJT\bin\jt_solidworks.cmd" "P:\QA Testing\InputFiles\JT\NIST\nist_ctc_04_asme1_ct521
0_rd.jt" C:\demo\output\nist_ctc_04_asme1_ct5210_rd.SLDPRT
```

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



```
Microsoft Windows [Version 10.0.17763.914]
(c) 2018 Microsoft Corporation. All rights reserved.

U:\>"C:\Program Files\Theorem\22.3\SWJT\bin\jt_solidworks.cmd" "P:\QA Testing\InputFiles\JT\NIST\
nist_ctc_04_asme1_ct5210_rd.jt" C:\demo\output\nist_ctc_04_asme1_ct5210_rd.SLDPRT
Pause before starting to give SLDWORKS.exe time to exit is consequetive runs

Waiting for 0 seconds, press a key to continue ...
Setting run time environment please wait ...

  ******************************************
  * Copyright Theorem Solutions Limited    *
  * JT - SolidWorks CADverter Version 22.1 *
  ******************************************

  Mon Dec 23 11:18:33 2019

  Input
    Jt         File : P:\QA Testing\InputFiles\JT\NIST\nist_ctc_04_asme1_ct5210_rd.jt
    SolidWorks File : C:\demo\output\nist_ctc_04_asme1_ct5210_rd.SLDPRT
    Progress   File : C:\demo\output\nist_ctc_04_asme1_ct5210_rd.progif


  List of gco entities :-
  ----------------------
  Type         Total    Standalone  Subordinate
  -----------  -------  ----------  -----------
  Curves        1206                 1206
  Surfaces      405                  405
  Planes        113                  113
  Faces         518                  518
  Edges         1206                 1206
  Vertices      728                  728
  Bsolids       1                    1
  Open solids   1                    1
  Groups        1         1
  -----------  -------  ----------  -----------

  *****************************************************
  * SOLIDWORKS  file successfully created             *
  * C:\demo\output\nist_ctc_04_asme1_ct5210_rd.SLDPRT *
  *****************************************************

U:\>
```

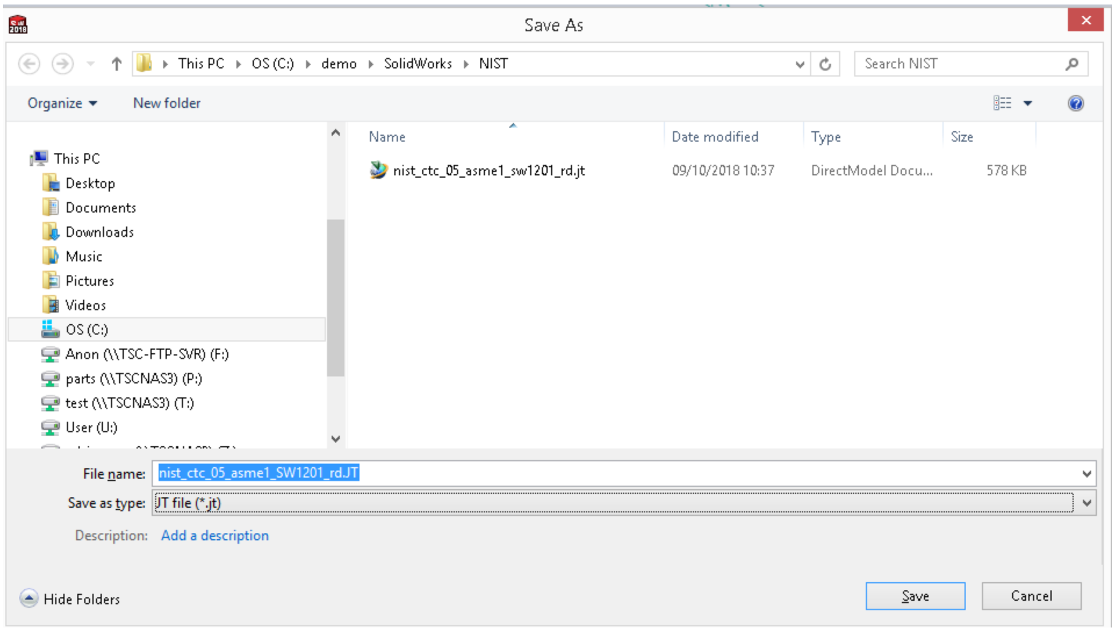
The file will be output to the target location. In this case:

C:\demo\output\nist_ctc_04_asme1_ct5210_rd.SLDPRT

Default Translation – Interactively from SolidWorks

Once the CD is built, in the “**bin**” folder is a batch file called **RegTheoremSWAddIn.bat**. Run this to register the Theorem SolidWorks Add-In that will then be added on the next time SolidWorks is booted up.

The Add-In can save and convert the current SolidWorks file into the JT format using the <File><Save As> (see below)



Save As

This PC ▸ OS (C:) ▸ demo ▸ SolidWorks ▸ NIST

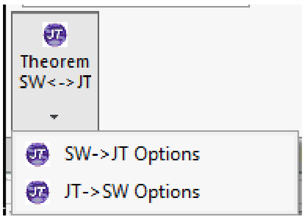
Name	Date modified	Type	Size
nist_ctc_05_asme1_sw1201_rd.jt	09/10/2018 10:37	DirectModel Docu...	578 KB

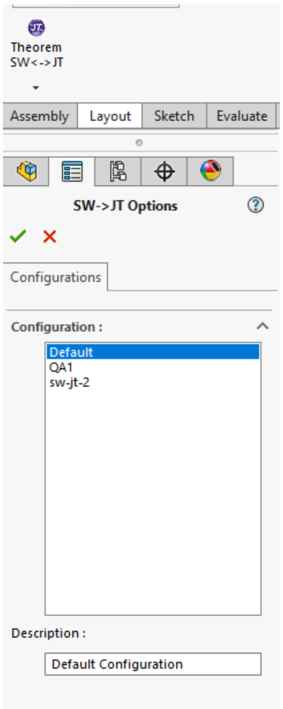
File name: nist_ctc_05_asme1_SW1201_rd.JT

Save as type: JT file (*.jt)

Description: Add a description

In order to translate from within SolidWorks, the SolidWorks application must be started, so that the appropriate SolidWorks menus are loaded. Select the “SW>JT Options” or “JT->SW Options” tab to display the translator options.





Configuration :

Default
QA1
sw-jt-2

Description :

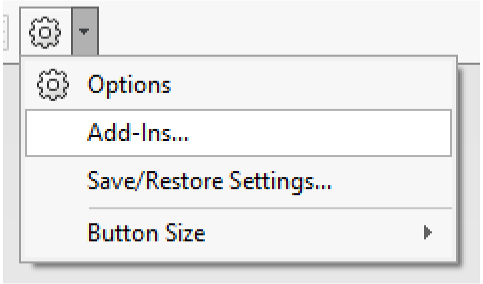
Default Configuration

This page gives you to the SW>JT or JT>SW Configurations
Note: Configurations are created from within the GUI – see below

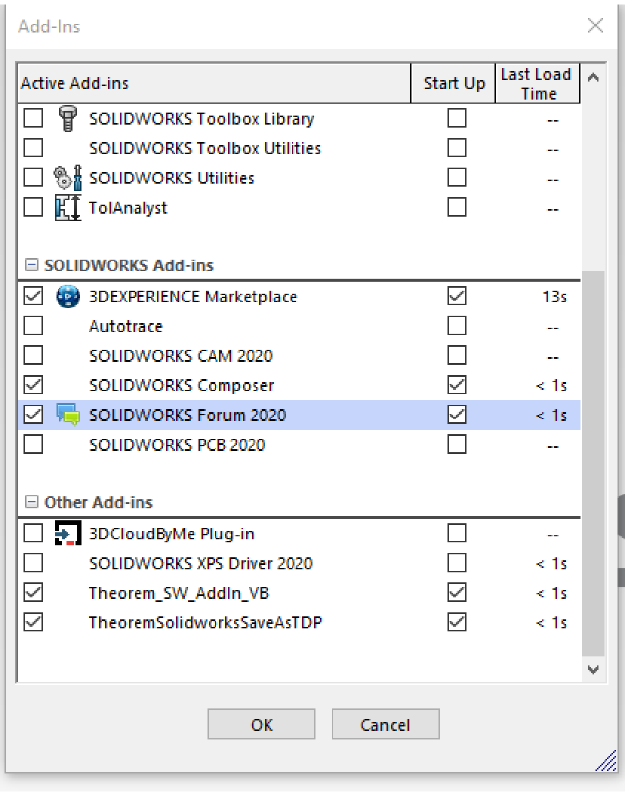
Theorem XR – Interactively from SolidWorks

If you have the Theorem Visualisation Pipeline server setup in your orgnisation, you can configure Solidworks to upload data interactively to the server.

Go to Add-Ins:



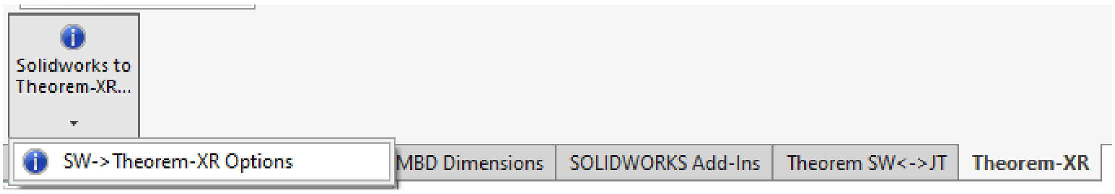
In the Add Ins window, find TheoremSolidworksSaveAsTDP and toggle on the Active check box and Start up check box:



Add-Ins

Active Add-ins	Start Up	Last Load Time
□ SOLIDWORKS Toolbox Library	□	--
□ SOLIDWORKS Toolbox Utilities	□	--
□ SOLIDWORKS Utilities	□	--
□ TolAnalyst	□	--
⊟ SOLIDWORKS Add-ins		
☑ 3DEXPERIENCE Marketplace	☑	13s
□ Autotrace	□	--
□ SOLIDWORKS CAM 2020	□	--
☑ SOLIDWORKS Composer	☑	< 1s
☑ SOLIDWORKS Forum 2020	☑	< 1s
□ SOLIDWORKS PCB 2020	□	--
⊟ Other Add-ins		
□ 3DCloudByMe Plug-in	□	--
□ SOLIDWORKS XPS Driver 2020	□	< 1s
☑ Theorem_SW_AddIn_VB	☑	< 1s
☑ TheoremSolidworksSaveAsTDP	☑	< 1s

For any part/assembly you have open in Solidworks, you can upload to the server by going to Theorem XR and selecting SW -> TheoremXR options:



The window panel on the left will display Configurations options which you can edit:

Read Motion: If the data includes animations you want to upload, select this box.

ServerURL: enter the server you want to upload the data to.

User Email: the email for the server access.

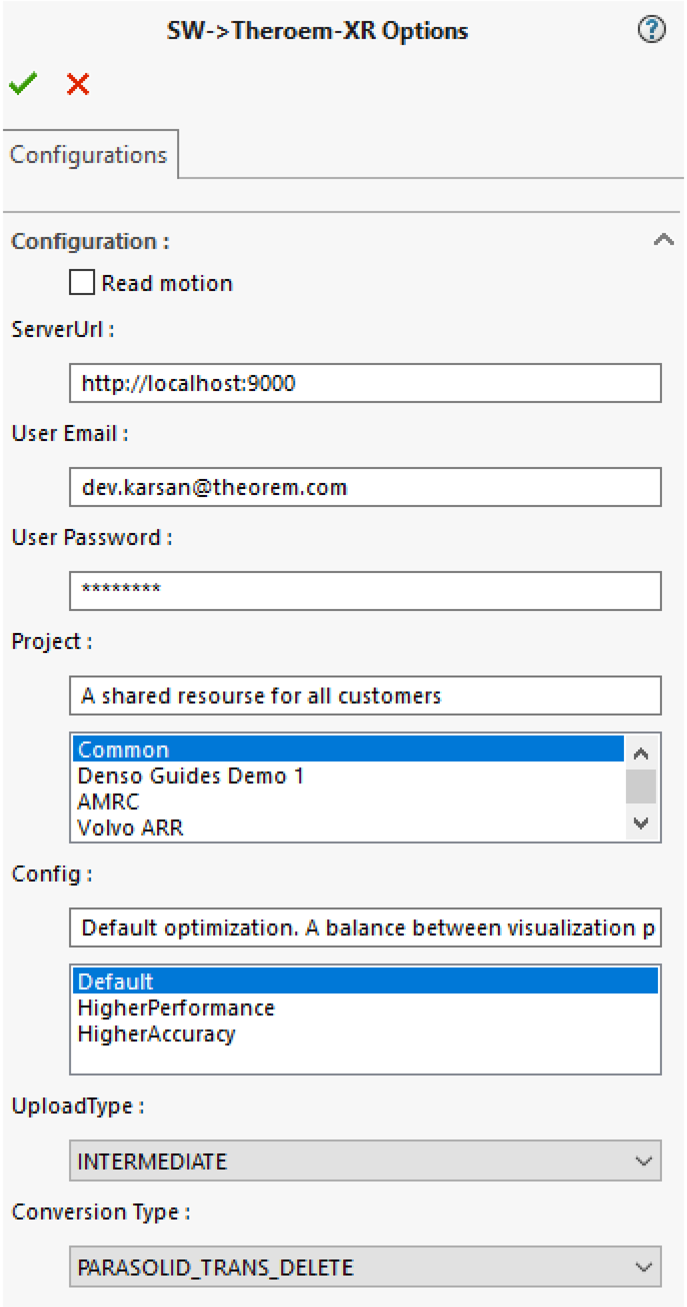
User Password: Password associated with the email.

Project: choose a specific Project folder you want to upload the data to.

Config: Select the optimisation config you want applied to the data when uploaded.

UploadType: Choose between Source (Solidworks), Intermediate (Viewer) or JT type of data you want uploaded. Depending on the option you choose, the data will be converted locally and then pushed to the server for final processing.

Conversion Type: Options on how the data is read before uploading to XR. The default is to read the data out into a Parasolid file.



Configuration :

□ Read motion

ServerUrl : http://localhost:9000

User Email : dev.karsan@theorem.com

User Password : ********

Project : A shared resourse for all customers

- Common
- Denso Guides Demo 1
- AMRC
- Volvo ARR

Config : Default optimization. A balance between visualization p

- Default
- HigherPerformance
- HigherAccuracy

UploadType : INTERMEDIATE

Conversion Type : PARASOLID_TRANS_DELETE

After the desired Configuration options have been selected, press the ✓ to save the changes. Go to File > Save as and save the part/assembly as a .tdp file, this will then push the saved file onto the server and process for Theorem XR use.

Visualise 3D Customization

Visualise 3D 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.

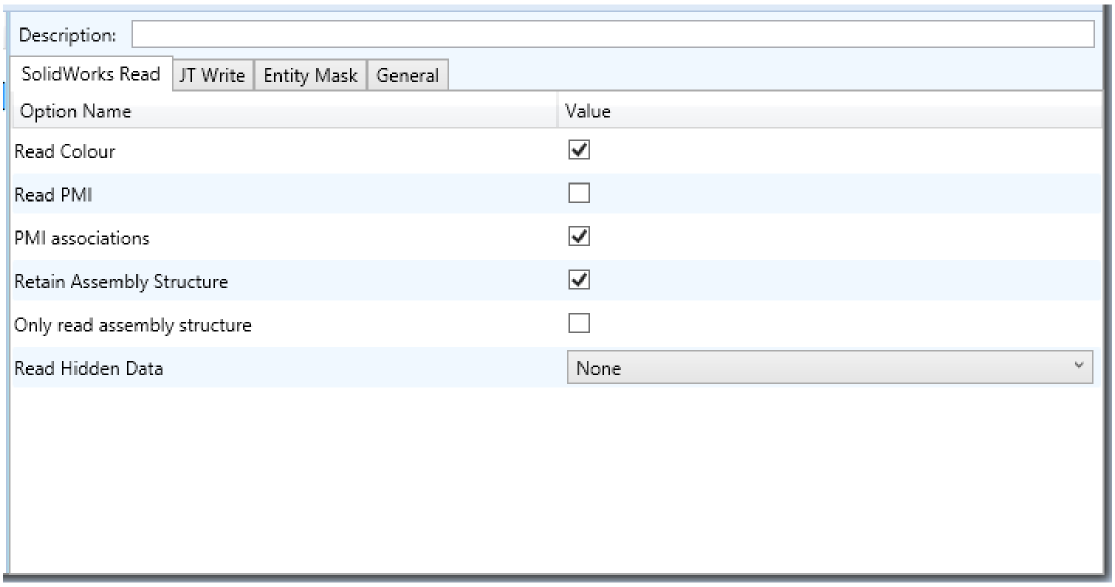
Common Options for SOLIDWORKS to JT

Within the Configuration Manager panel of the Unified Interface, arguments that can be specified when publishing SolidWorks data into JT are grouped into 3 areas:

- SOLIDWORKS Read – Those arguments that affect how data is read from SOLIDWORKS
- 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 dat

SOLIDWORKS Read Arguments

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



Description:

Option Name	Value
Read Colour	☑
Read PMI	□
PMI associations	☑
Retain Assembly Structure	☑
Only read assembly structure	□
Read Hidden Data	None

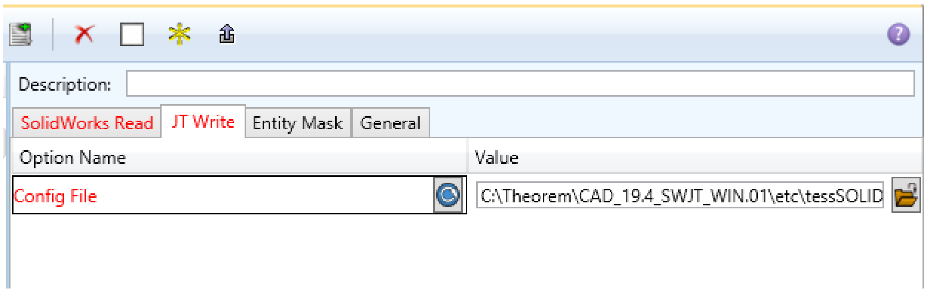
Each of these options is described below:

Option	Description
Read Colour	This allows the user to read the SolidWorks colour ○ Command Line Syntax: ▪ *Project [file_name]*
Read PMI	Enables PMI data read from the SolidWorks file. (Default is OFF). o Command Line Syntax: □ read_pmi – to turn on
PMI Associations	Switches on/off the association between the PMI and the geometry display in JT
Retain Assembly Structure	Enables Assembly Structure to be retained. Deselecting this option will remove all assembly structure and collapse ALL geometry into a single selectable object
Only read assembly structure	Enables Assembly Geometry to be removed This will translate the Assembly Structure only

Read Hidden Data	No show Default – Data will be read as displayed in SolidWorks Read all hidden – Read hidden geometry/structure/pmi Read hidden structure – Read hidden assembly structure only Read hidden geometry – Read hidden geometry only Read hidden pmi – Read hidden pmi only

JT Write Arguments

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



Description:

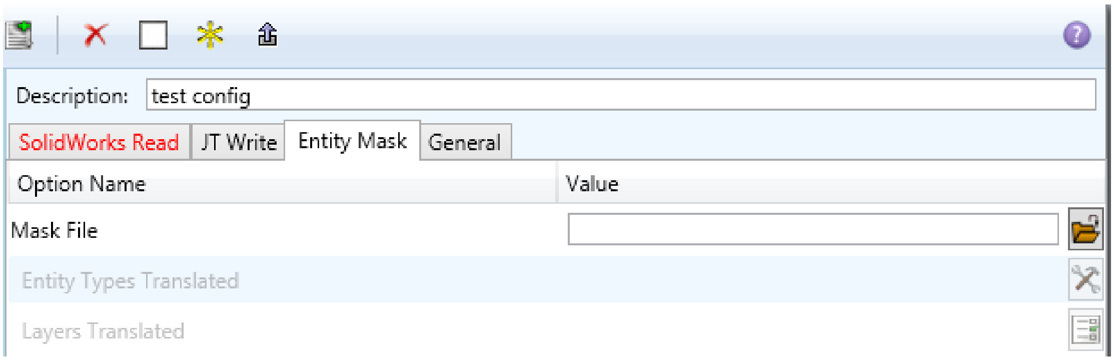
Option Name	Value
Config File	C:\Theorem\CAD_19.4_SWJT_WIN.01\etc\tessSOLID

Each of these options is described below:

Option	Description
Config File	Allows a JT write 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]*

SOLIDWORKS to JT Entity Mask

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



Description: test config

Option Name	Value
Mask File	
Entity Types Translated	
Layers Translated	

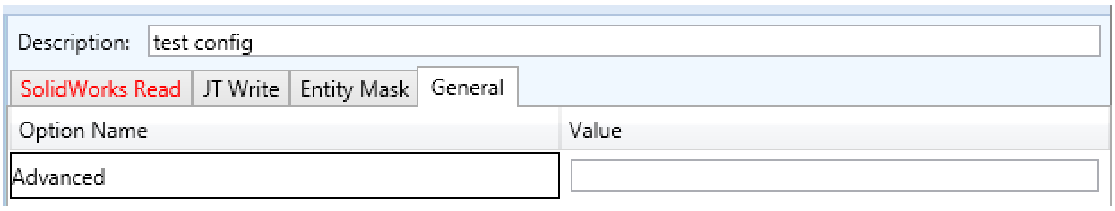
Each of these options is described below:

Option	Description
Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: ○ Command Line Syntax: ▪ *Mask <filename>*
Entity Types Translated	Specifies a selection list from which to select which entity types are to be processed. The following types are available: "POI","LIN","ARC","CON","CUR","SUR","FAC","SOL" ○ Command Line Syntax: ▪ *Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g.: **ON POI** to ensure they are considered in the translation*
Layers Translated	Specifies a selection list from which to select which layers are to be processed. ○ Command Line Syntax: ▪ *A single entry of **ON ALL LAY** Must precede any Layer Mask command.* ▪ *Add a list or range of numbers representing layer to be processed to the specified mask file to ensure*

		they are NOT considered in the translation *e.g.:* ***OFF LAY 114,149,166,167,168***

SOLIDWORKS to JT General Arguments

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



Description: test config

Option Name	Value
Advanced	

Each of these options is described below:

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

SOLIDWORKS to JT Advanced Arguments

Theorem's SOLIDWORKS to JT translator has been configured with default settings that optimise the translation process. However, there are times when a satisfactory result cannot be obtained, so it may be required to deploy one or more Advanced Arguments to improve the translated result.

The following table describes useful Advanced Arguments that can be entered into the General Tab -> Advanced field:

Option	Description
Parasolid Tolerant Modelling	A secondary option to be used when *Brep Type = XT Brep (Theorem)* output is specified. Enables Parasolid tolerant modelling. Default is ON ○ Command Line Syntax

	▪ *nopstolmodel – to turn off*
Factor	Specify the factor level of Parasolid Tolerant Modelling when turned on. Default is 3. ○ Command Line Syntax ▪ *pstolmodel 3*
Sew Parasolid Bodies	A secondary option to be used when *Brep Type = XT Brep (Theorem)* output is specified. Enables the sewing of Parasolid bodies. Default is ON ○ Command Line Syntax ▪ *nosew – to turn off*
Tolerance	Specify the tolerance for the sew command above. Default is 0.01. ○ Command Line Syntax ▪ *pssew 0.01*
Incremental Sewing	Enables incremental sewing when used with Sew Parasolid Bodies. Default is ON. ○ Command Line Syntax ▪ *no_sew_increm – to turn off*
Split Discontinuous Surfaces	A secondary option to be used when *Brep Type = XT Brep (Theorem)* output specified. Splits discontinuous surfaces. Default is OFF. ○ Command Line Syntax ▪ *brep_prep – to turn on* ▪ *no_brep_prep – to turn off*
Force body creation (No check of Parasolid entities)	A secondary option enabled when XT Brep (Theorem) output specified. Removes the checking of Parasolid entities. Default is ON. ○ Command Line Syntax ▪ *nocheck* – (force body creation without checking = Default)

	▪ *check* – (doesn't force the body creation - Parasolid checking is enabled)
Fix Degenerate Edges	A secondary option to be used when *Brep Type = XT Brep (Theorem)* output specified. On face create failure, check and fix any degenerate edges. Default is ON. ○ Command Line Syntax ▪ *fix_degen* ▪ *no_fix_degen – to turn off*
Specify a Face Edge Tolerance	A secondary option to be used when *Brep Type = XT Brep (Theorem)* output specified. Specify an edge tolerance to be used when creating faces. Default is ON. ○ Command Line Syntax ▪ *Please see Edge Tolerance below*
Edge Tolerance	A secondary option used with Specify a Face Edge Tolerance where the tolerance value is assigned. Default is 0.000006. ○ Command Line Syntax ▪ *face_edge_tol 0.000006*
Fix small features in solids	A secondary option to be used when *Brep Type = XT Brep (Theorem)* output specified. Remove small edges, sliver and spike faces from solid bodies. Default is OFF. ○ Command Line Syntax ▪ *ps_fix_small – to turn on* ▪ *no_ps_fix_small - default*
Fix small features in open solids	A secondary option to be used when *Brep Type = XT Brep (Theorem)* output specified. Remove small edges, sliver and spike faces from open solids. Default is OFF. ○ Command Line Syntax ▪ *ps_fix_osol – to turn on* ▪ *no_ps_fix_osol - default*

Parasolid Trans Delete	This option causes the CADverter to check for any hidden bodies in the .sldprt file and delete these before exporting the .sldprt file to a parasolid .x_t file. This is because hidden bodies became visible in the export. This option will reduce the processing time for large assemblies. o Command Line Syntax ▪ parasolid_trans_delete
Simplify Geometry	A secondary option to be used when *Brep Type* = *XT Brep (Theorem)* output specified. Simplify Geometry. Default is OFF. ○ Command Line Syntax ▪ *simplify_solids – to turn on*
SolidWorks Configurations	Configurations allow you to create multiple variations of a part or assembly model within a single document. Configurations provide a convenient way to develop and manage families of models with different dimensions, components, or other parameters. • *configuration* <Configuration Name> cause the specified configuration to be applied to the component prior to conversion • *configuration* <File Name // ; separated list of Configurations> converts the part using the named configurations where this list can be specified on the command line separated by ; or in a file and saved as OUTPUT_PNAME_CONFIGURATION NAME • *all_configurations* causes all configurations to be converted and saved as OUTPUT_PNAME_CONFIGURATION NAME
Speedpaks	A SpeedPak configuration is essentially a subset of the parts, faces, reference geometry, sketches, and curves of an assembly. • *speedpaks_on* • *speedpaks_off (default)*

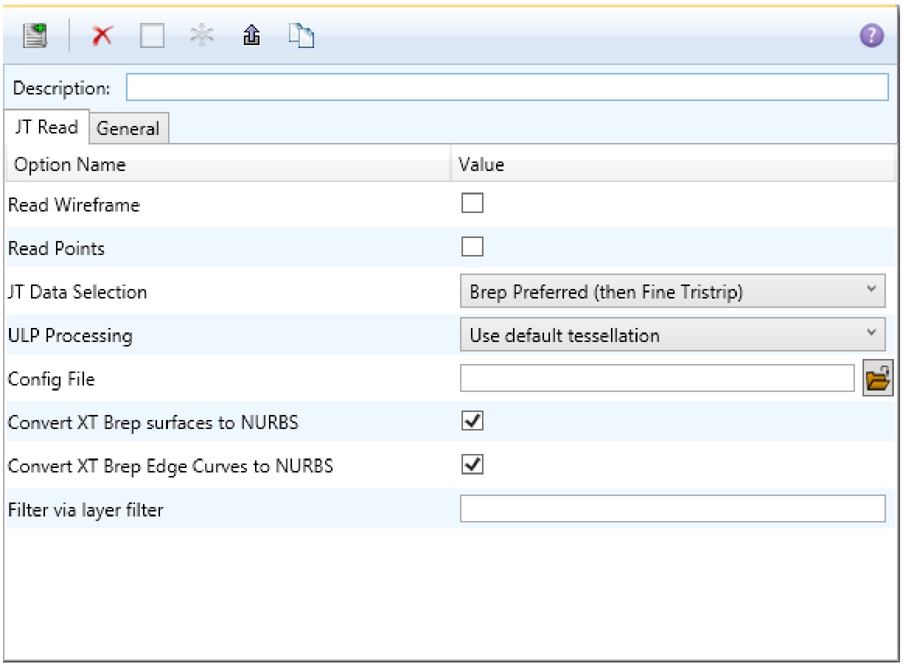
Common Options for JT to SOLIDWORKS

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing JT data into SolidWorks are grouped into 2 areas:

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

JT Read Arguments

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



Description:

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

Each of these options is described below:

Option	Description
Read Wireframe	Read JT wireframe data. Default is OFF. o Command Line Syntax read_wire_frame
Read Points	Read JT point data. Default is OFF. o Command Line Syntax

	▪ *read_points*
JT Data Selection	Select Brep or tessellated data read. Default is 'Brep Preferred (then fine facet)'. ○ Command Line Syntax ○ Brep Preferred (then Fine Facet): brep_pref ○ Brep preferred (then Fine Tristrip): brep_pref_tri ○ Brep Only: brep_only ○ Fine Tristrip: fine_tristrips ○ Coarse Tristrip: coarse_tristrips ○ Fine Facet: fine_facets ○ Coarse Facet: coarse_facets
ULP Processing	How to process ULP data in the JT part. Default is 'Use Default Tessellation'. ○ Command Line Syntax ▪ *Use Default Tessellation: Default* ▪ *Tessellate ULP: tess_ulp*
Config File	Config File for Brep or ULP tessellation. ○ `Command Line Syntax` ○ `-z [path to file]`
Retain Assembly Structure	Enables Assembly Structure to be removed. Selecting this option will remove all assembly structure and collapse ALL geometry into a single selectable object
Convert XT Brep surfaces to NURBS	Read XT Brep surfaces as NURBS surfaces (else read in native form). Default is ON. ○ Command Line Syntax ▪ *noprep – to turn off*
Convert XT Brep Edge Curves to NURBS	Read XT Brep edge curves as NURBS curves (else read in native form. Default is ON. ○ `Command Line Syntax` ▪ *rd_native_edge – to turn off*
Filter via layer filter	Supply layer filter(s) separated by commas and double quoted. Default is OFF. ○ Command Line Syntax ▪ *layer_filter*

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 config file will be used:-

> **tessSOLIDWORKS.config** in ***%TS_INST%\etc*** directory (where TS_INST = Installed directory)

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

The Setup Section

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

To edit setup options

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

Option name	Keywords	Example
EAITranslator	EAITranslator {	EAITranslator {
OutputDirectory	"*path to directory*"	OutputDirectory = "/home/<user>/"
CommonPartsPath	"*path to directory*"	CommonPartsPath= "/myaccount/jtparts/"
chordalOption	"RELATIVE" "ABSOLUTE"	chordalOption = "RELATIVE"
structureOption	"PER_PART" "MONOLITHIC" "FULL_SHATTER"	structureOption = "MONOLITHIC"
WriteWhichFiles	"ALL" "ASSEMBLY_ONLY" "PARTS_ONLY"	WriteWhichFiles = "ALL"
compression	true TRUE false FALSE	compression = true
triStripOpt	true TRUE false FALSE	triStripOpt = false
seamSewing	true TRUE false FALSE	seamSewing = true
seamSewingTol	*any integer*	seamSewingTol = 0.001
includeBrep	true	includeBrep = false

	TRUE false 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
JtFileFormat	64,70,80,81,82,90, 91,92,93,94,95,100	JtFileFormat = “95”
includeULP	PASSTHROUGH TRUE FALSE	includeULP = “PASSTHROUGH”
ulpPrecision	Real Value	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	*any integer*	Level = 1
Chordal	*any number*	Chordal = 0.001
Angular	*any number*	Angular = 25
Length	*any number*	Length = 1
Label	*any string*	“ud_FINE”
FeatureSuppression	*any integer*	FeatureSuppression = 0
Simplify	*any number*	Simplify = 0.60
AdvCompressionLevel	*any number*	AdvCompressionLevel = 0.0
ULP	true 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	*"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
SolidWorksOptions	SolidWorksOptions {	SolidWorksOptions {	
ProjectFile	Full system file path	Windows example ProjectFile = "P:\apps\SolidWorks2009\Samples.ipj"	“”
IgnoreMigration	true/TRUE false/FALSE	IgnoreMigration = true	false
ReportFilename	Full system file path	Windows example ReportFilename = P:\caddata\translation\result\part55	Windows system C:%TEMP%\tscprogressyi
OutputUnits	mm millimetres cm centimetres m metre metres inches feet yards inputUnits	OutputUnits = mm	inputUnits
StructureOutputType	JT PLMXML PLMXMLJT	StructureOutputType = JT	JT
PLMXMLPropertyMappingFile	*File Name*	PLMXMLPropertyMappingFile = “mapping_file.txt”	“”
brepType	XT JT XTJT	brepType = XT	JT

ParasolidTolerantModelling	true/TRUE false/FALSE	ParasolidTolerantModelling = true	true
ParasolidTolerantModellingFactor	*Any positive integer*	ParasolidTolerantModellingFactor = 3	3
SewParasolidBodies	true/TRUE false/FALSE	SewParasolidBodies = true	true
SewParasolidBodiesTol	*Any number*	SewParasolidBodiesTol = 0.01	0.01
IncrementalSewing	true/TRUE false/FALSE	IncrementalSewing = true	true
IncrementalSewingNoOfIterations	true/TRUE false/FALSE	IncrementalSewingNoOfIterations = 5	5
ExplodeSolidstoFaces	true/TRUE false/FALSE	ExplodeSolidstoFaces = false	false
SplitDiscontinuousSurfaces	true/TRUE false/FALSE	SplitDiscontinuousSurfaces = true	false
ForceBodyCreation	true/TRUE false/FALSE	ForceBodyCreation = true	true
FixDegenerateEdges	true/TRUE false/FALSE	FixDegenerateEdges = true	true
FaceEdgeTol	*Any number*	FaceEdgeTol = 0.000006	0.000006
FixSmallFeaturesSolids	true/TRUE false/FALSE	FixSmallFeaturesSolids = false	false
FixSmallFeaturesOpenSolids	true/TRUE false/FALSE	FixSmallFeaturesOpenSolids = false	false
SimplifyGeometry	true/TRUE false/FALSE	SimplifyGeometry = false	false
BrepWireframe	true/TRUE false/FALSE	BrepWireframe = true	true
ProduceTessellatedOutput	true/TRUE false/FALSE	ProduceTessellatedOutput = false	false
ExpandPart	true/TRUE false/FALSE	ExpandPart = false	false
ReuseSolids	true/TRUE false/FALSE	ReuseSolids = false	false
CADPropertyMappingFile	*File Name*	CADPropertyMappingFile = “mapping_file.txt”	“”
SavedViewsViewSetName	*"string of characters"*	SavedViewsViewSetName	"SavedViews"
AnnotationPlanesViewSetName	*"string of characters"*	AnnotationPlanesViewSetName	"AnnotationPlanes"
Close brace	}	}	

Appendix C – 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,,
```