User Guide
3DExperience - 3D PDF

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Contents

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:

CAD**Translate**

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

See our website for more detail.

CAD**Publish**

The creation of documents enriched with 3D content

See our website for more detail.

Theorem**XR**

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

See our website for more detail.

Overview of CADPublish

What is a 3D PDF?

In its simplest form a 3D PDF is a standard Adobe PDF document that contains interactive 3D content.

A PDF is a document format that can contain:

- 2D Graphics.
- 3D Graphics (3D CAD model.)
- Attached files to form a Technical Data Package.
- The ability to interrogate and interact with the document.
- The ability to universally review CAD with a free viewer.
- Compressed data for optimum file size sharing.
- Support for Model Based Definition (MBD).
- The addition of 3D content to a PDF allows your company:
    - To produce documents to fully describe the product.
    - To achieve effective communication regarding product information.
    - To automate specific business processes.
    - To improve understanding and eliminate ambiguity in relation to product data.
    - To create and extend customised, dynamic documents.
    - To create rich documents with 3D content which can be consumed anywhere.
    - To create interactive and engaging documents such as brochures.

It's already used for documents which commonly relate to products e.g.; Request for Quotation, Engineering Change Requests, Bill of Material reports, Inspection Documents, Works instructions, Service manuals, Sales literature, etc.
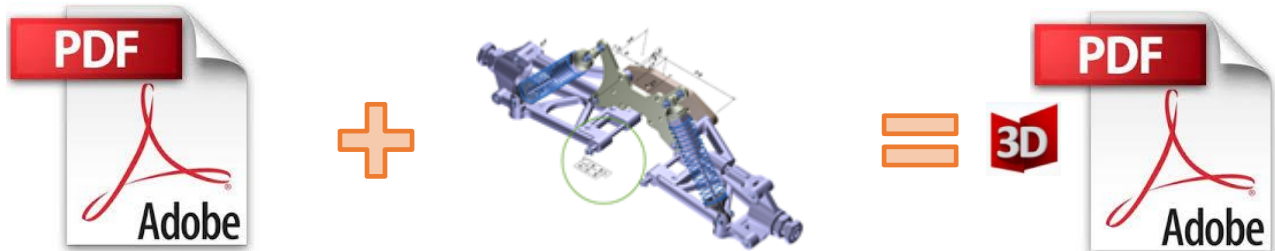
Review our website here **for a short video explaining 3D PDF or review our** Frequently asked questions for more information

What is CADPublish?

Theorem's CADPublish brand offers a 3D PDF publishing solution for organisations who design with 3DExperience, CATIA V5, Creo, NX or who want to extend the use of their JT data to create interactive 3D documentation (3D PDF's.)

The use of 3D PDF means that anybody can now consume, view and mark up 3D data with Adobe's ubiquitous Reader, eliminating the need to install specialist viewing software or to be trained to use a special tool.

Designed for organizations of all sizes, CADPublish solutions are based on the original Adobe® and vendor specific technology, therefore are fully compatible with Acrobat and native CAD systems enabling users to take the intelligence contained within a 3D model directly into the world of PDF.

Theorem's CADPublish suite of products is powered by native Adobe® technology 3D PDF publishing toolkit, which is also used in Adobe Acrobat® and Adobe Reader®.

For more information regarding our CADPublish brand, please contact sales@theorem.com

What does CADPublish provide?

Theorem Solutions' CADPublish suite of applications includes:

- **CADPublish Create**
  - 3D PDF Publishing from within CAD Applications:
    - 3DExperience / CATIA V5 / Creo / NX

- **CADPublish Enhance**
  - Batch Publishing for:
    - 3DExperience / CATIA V5 / Creo / NX / JT

CADPublish supplied templates

Both the CADPublish Create and CADPublish Enhance brands allow the user to create custom configurations. This enables the user to select a specific template in which to translate their source data in to, including the example templates below.

Example templates are provided in the following location:

<installation_directory>\data\publish_3dpdf\templates

The following examples are available:

- BOM – Illustrates how to output an Assembly Bill of Materials.
- Assembly_Viewer – Illustrates JavaScript additions to a template replicating a CAD viewer.
- Technical_Data_Package – Illustrates adding attachments to a template, along with the JavaScript additions
- PMI_Analysis – Illustrates how to output a PMI list and filter either by type or by view

These templates are supplied as examples only and can be used by customers as a guide in creating their own template documents.

Theorem offer a wide range of services to assist customers with their document creation needs. If you would like more information regarding our template design services, please contact sales@theorem.com quoting "3D PDF Template Design".

The remainder of this document will focus on CADPublish for 3DExperience. For further information on other Theorem CADPublish Products please contact sales@theorem.com

3DExperience to 3D PDF – CADPublish

The 3DExperience to 3D PDF Uni-directional Translator

The 3DExperience to 3D PDF translator may be installed on a number of machines accessing a central network-floating license.

The 3DExperience to 3D PDF translator is a Uni-directional direct database converter between the Dassault Systemes 3DExperience application and PDF file format.

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 PDF to be their main method of visualisation and/or collaboration either internally or between OEMs and their customers or suppliers.

It is also a major method of visualisation and is used by companies using Adobe® based solutions to translate their 3DExperience data into PDF format.

Primary Product Features

- Converts 3DExperience data to 3D PDF documents.
- Converts assembly structures to PDF file format.
- Converts attribute data such as meta-data, colour and layer information and 3DExperience properties.
- Integrated with the 3DExperience installation.
- The conversion process can be run Interactively or in Batch Mode.
- Uses the 3DExperience API.

Primary Product benefits?

- Direct conversion from 3DExperience to PDF reduces processing time, simplifies integration and ensures that the correct data is being published.
- Using the freely available Adobe Acrobat Reader application allows the user to visualise the 3D CAD data without Incurring any additional costs.
- Provides improved communication and collaboration by visualising 3DExperience data in a PDF.
- Allows 3D CAD data to be easily shared with non-CAD users.
- Allows for Comments and user-interactions to be saved to the PDF document for downstream updates allowing greater collaboration and communication between users.
- Allows for documents to be password protected improving the security of sensitive Information when sharing the data.
- Enables the data to be converted into a PDF template specified by the user.
- Allows for additional documents to be added as attachments.
- Reduce costs and risks associated to accessing the wrong version of data by integrating the publishing process into all related business processes.
- With over 30 years industrial use Theorem's product robustness and quality is well proven, reducing your business risk.

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's and Tutorials.

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

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

Installation

The installation is run from the .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation

License Configuration

To run any product a valid license file is required. The Flex License Manager is run from the .msi file download provided. This can be accessed from the Product Release Notes. 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

Running the Product

Once configured and licensed, the product is ready to be run. There are 3 distinct ways of running the translator:

- **Interactively from within 3DExperience**
  - The Interactive Interface provides a direct method exporting from 3DExperience.

- **In Batch via CATUtil - DataExchangePLMBatch**
  - The 3DExperience DataExchangePLMBatch Interface provides a direct method of invoking the translator. It can be used to translate single or multiple files at once.

- **On the Command Line**
  - A command line method of Invoking the translator is possible.

Translating Interactively from within 3DExperience

Launching 3DExperience with Theorem Plug-ins

The 3DExperience to 3D PDF translator allows an opened 3DExperience part or assembly to be exported directly to a PDF.

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

3DExperience can be started from a desktop shortcut created during installation.

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

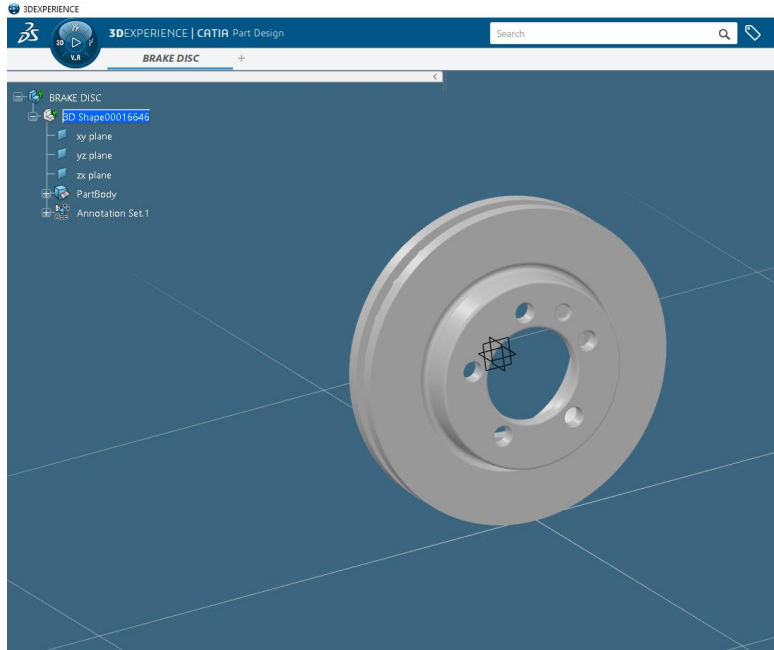***<installation_directory>\bin***

The script name is:

start_3DEXPERIENCE_Theorem_Multi-CAD_3DPDF_CATIAV6R<version>.cmd
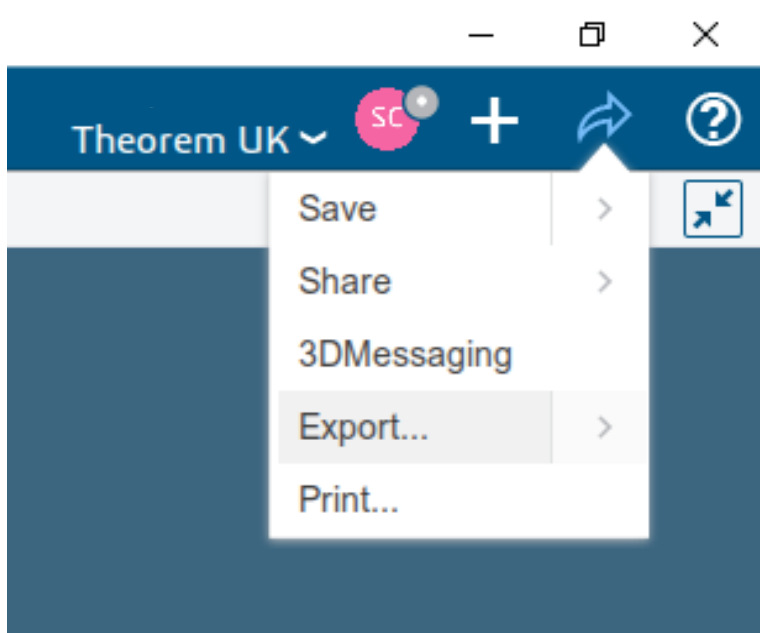
(where <version> should be substituted for the version of 3DExperience that you have installed – e.g. 2021x, 2022x, 2023x):

Interactive Export to 3D PDF

Once the 3DExperience application has been launched, open the product or representation that is going to be exported to PDF.


Select the arrow in the top right corner, then from the list displayed select Export.

In the Export dialog box displayed, ensure the 'Format' is set to PDF_THEOREM (*.pdf). Select the required location for the file and ensure the Filename displayed is correct. Click OK to initiate the export.
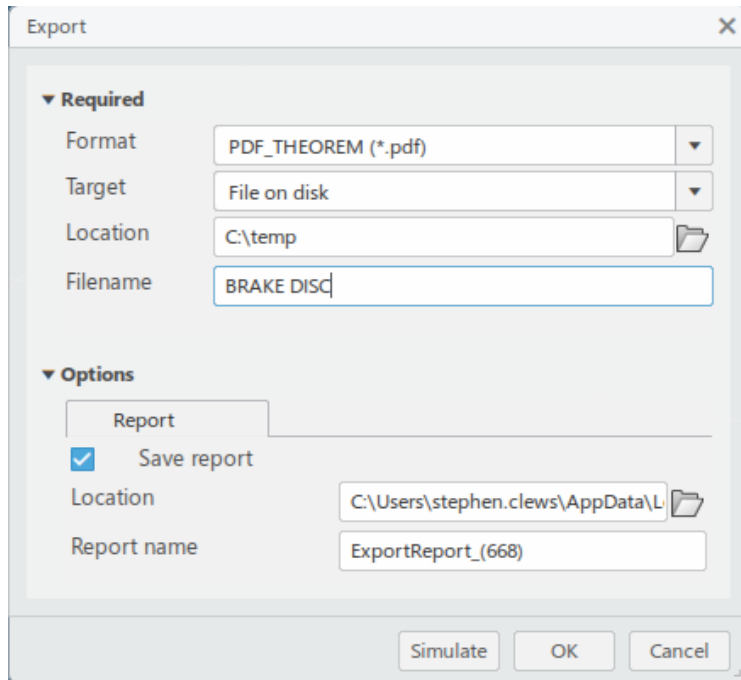

Export status of Succeeded displayed in Operation report window. PDF created in the location specified.
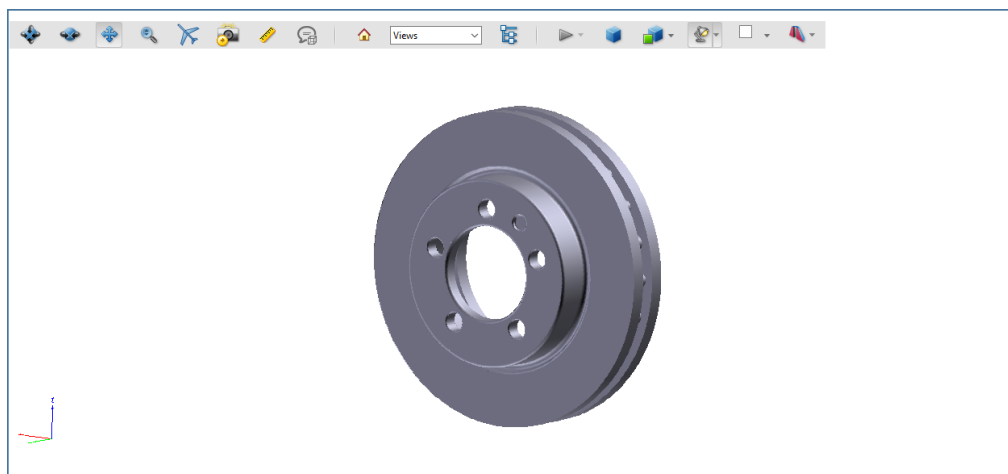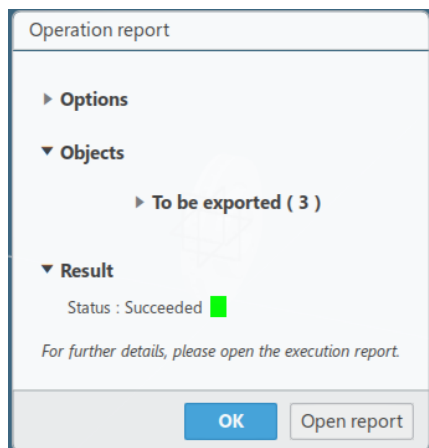
Theorem Interactive Conversion Settings

The 3DExperience interface does not currently require the user to apply any specific settings for the translation. There are some general settings that should be checked if required (e.g. for PMI conversion.) These are accessed through **Preferences>Legacy Preferences>General>Compatibility>External Native 3D Formats.**
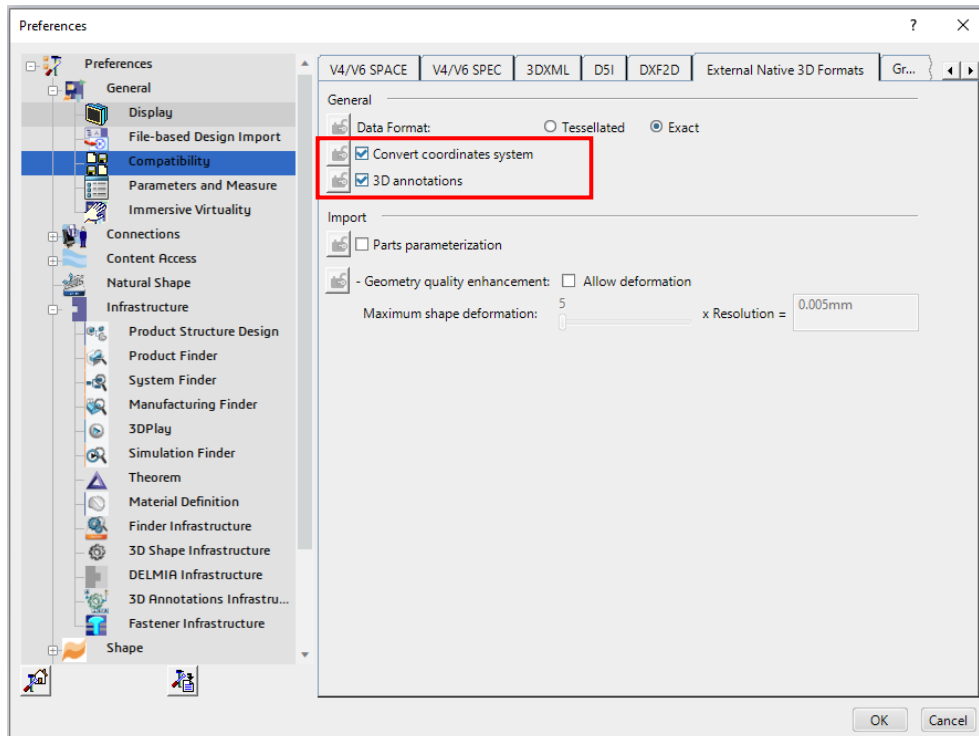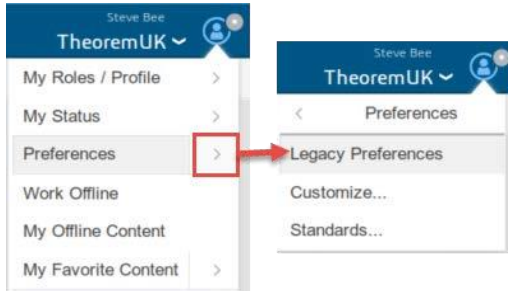

This page is a standard Dassault Page that sets the preferred mode of conversion (in this case Exact), and also the general options **'Convert coordinates system'** and **'3D annotations'**. Both of which are toggled on.

There are also some Theorem settings that can be applied. These are accessed through **_Preferences>Legacy Preferences> Infrastructure>Theorem_**

A dedicated tab under **_'Theorem'_** allows the user access to Theorem Configurations. From this Panel, the user can select a predefined configuration or create a new configuration.

To create a new configuration, select the **_'Launch Configuration Manager'_** command.
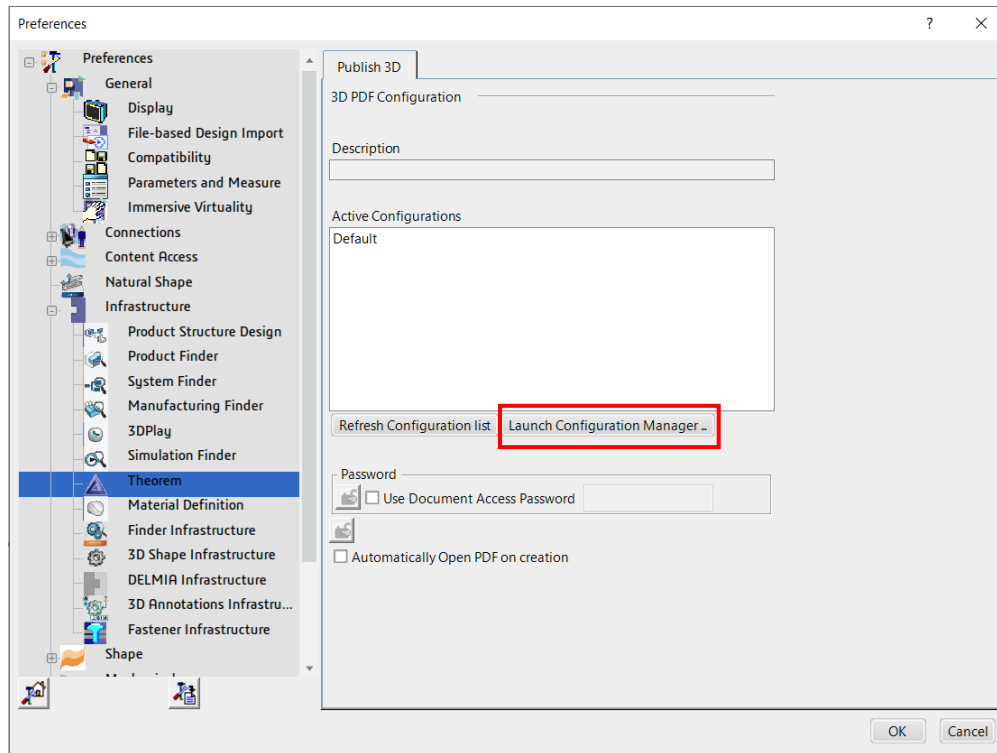
In the configuration manager window, select New. Rename as required.
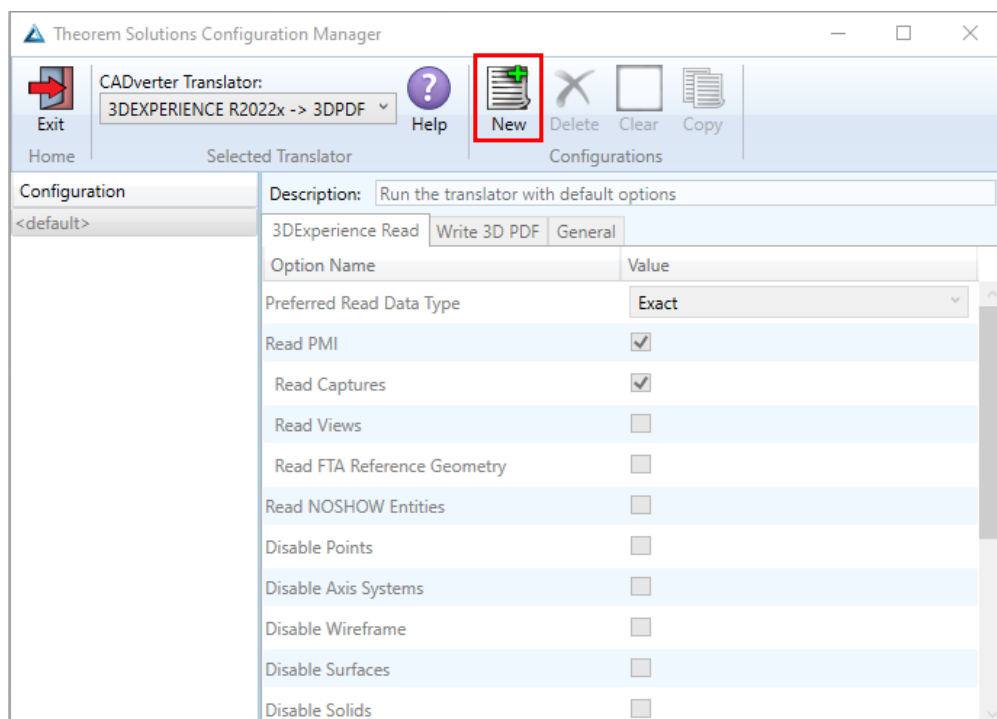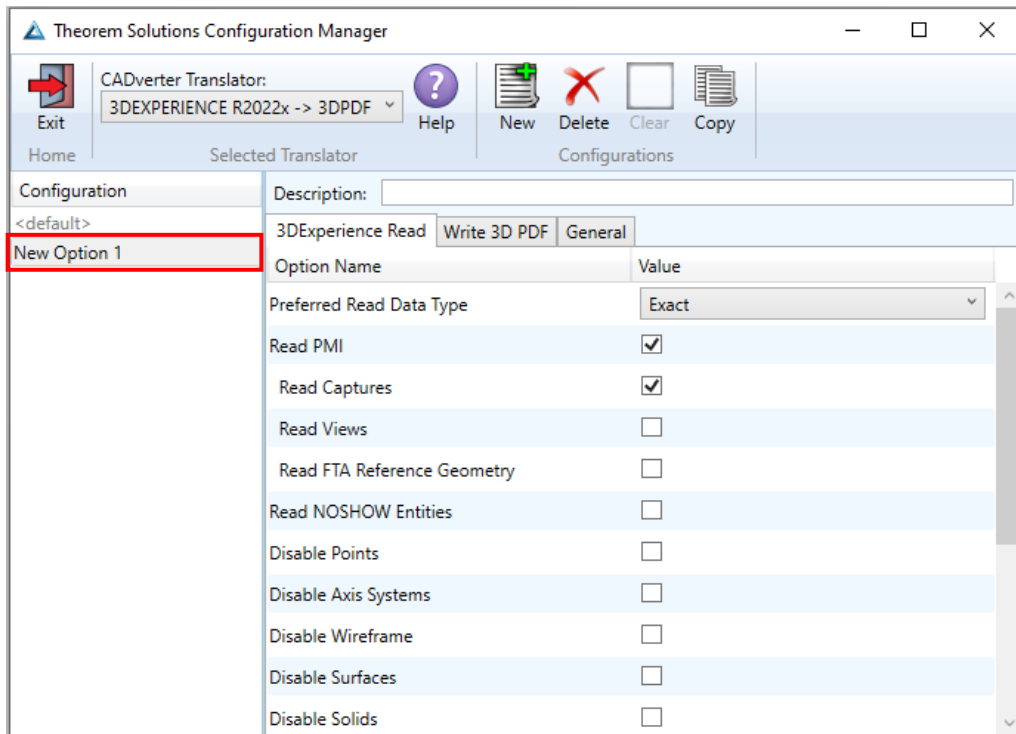
Any new configurations created will be displayed in the Active Configurations list once it has been refreshed.
To do this select *'Refresh Configuration list'*
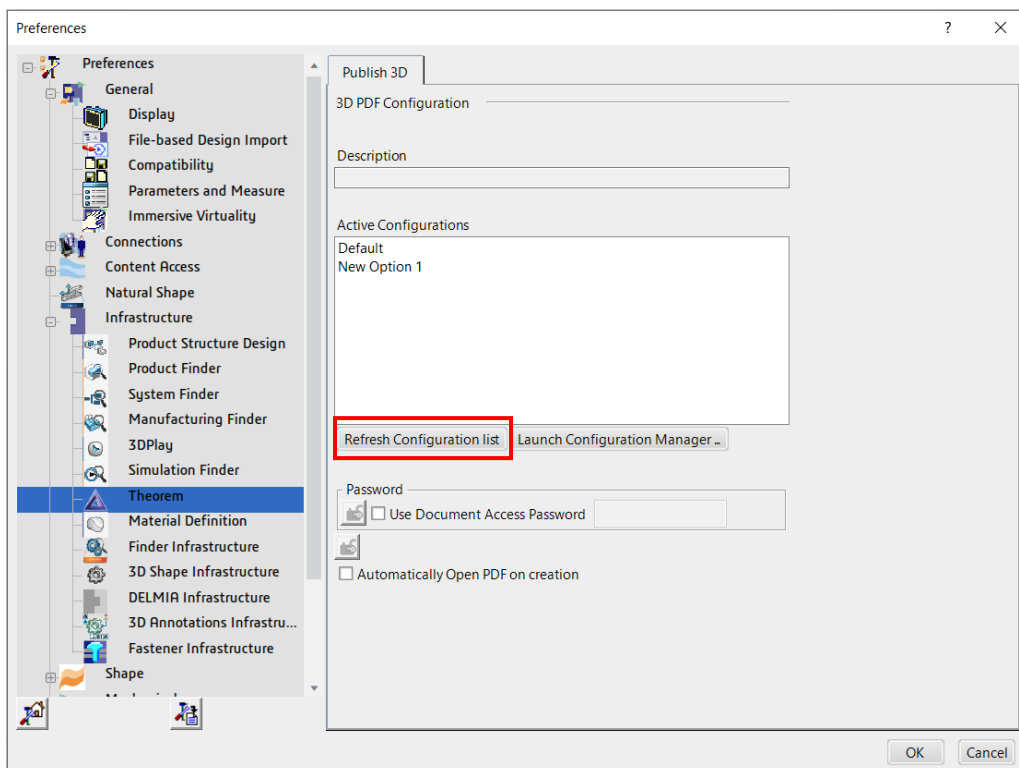
Select the required configuration to make it the active configuration. This will be highlighted in blue when selected.

Options for using a password to open the document and for automatically opening the PDF upon creation are also available to toggle on within this tab.

Configuration Manager

CADPublish 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 options. Commonly used options are supported via the Configuration Manager, with Advanced Arguments being described within this document for use in the Configuration Manager or via Command Line invocation.
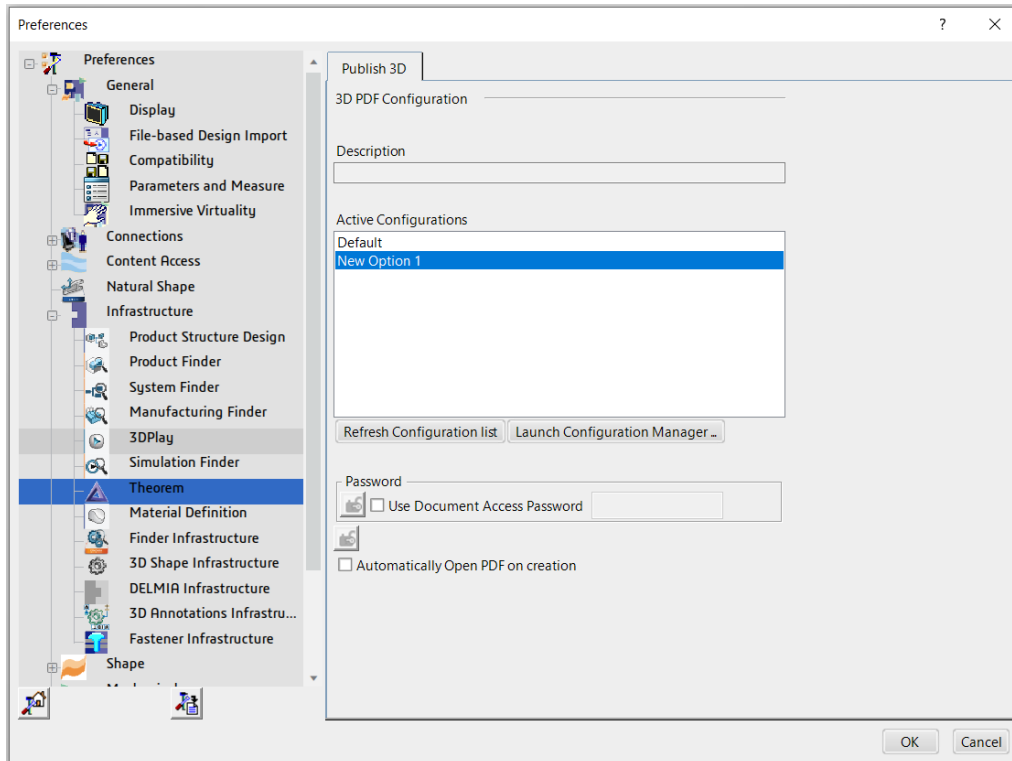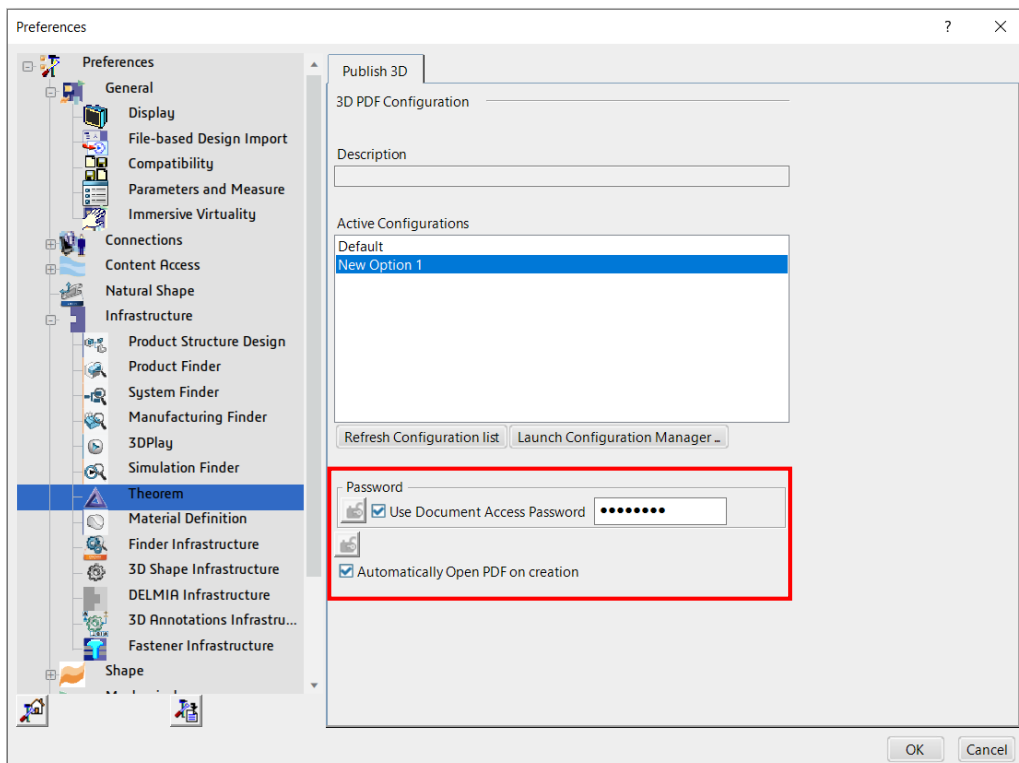
Within the Configuration Manager, the options that are available to use are grouped into the following 3 areas when translating data from 3DExperience to 3D PDF.

- 3DExperience Read – Those options that affect how data is read from 3DExperience.
- Write 3D PDF – Those options that affect how data is written to 3DPDF
- General – Those options that are common to ALL publishing activities regardless of source data. Advanced arguments are added in this tab.

3DExperience Read

Each of the options displayed in the image above are described below:

Option	Description
Preferred Read Data Type	The setting options are EXACT (default) or TESSELATED. Command Line Syntax: • *read_tess*
Read PMI	Enables PMI data read (Default is ON.) Command Line Syntax: • *read_pmi – default* • *read_pmi_off – to turn off* Note – When **'read_pmi'** is enabled it also enables the **'fill_pmi_arrows'**, **'fill_pmi_text'** and **'pmi_filled_text'** options. These can be overridden by setting the Advanced arguments **'dont_fill_pmi_arrows'** and/or **'dont_fill_pmi_text'**
Read Captures	A secondary argument to **'Read PMI'** and allows control over whether captures are read as part of the process. Default is ON when **'Read PMI'** is marked as ON. Captures can ONLY be read when **'Read PMI'** is ON. Command Line Syntax: • *read_captures - default* • *dont_read_captures – to turn off*
Read Views	A secondary argument to **'Read PMI'** and allows control over whether views are read as part of the process. Default is OFF. Views can ONLY be read when **'Read PMI'** is ON. Command Line Syntax: • *read_views – to turn on*
Read FTA Reference Geometry	A secondary argument to **'Read PMI'** and enables the reading of FTA Reference Geometry. Default is OFF. FTA Reference Geometry can ONLY be read when **'Read PMI'** is ON. Command Line Syntax: • *read_geometry – to turn on*
Read NOSHOW Entities	Read any entities that are in NOSHOW. Default is not to read NOSHOW entities. Command Line Syntax: • *noshow*
Disable Points	Switches off Point processing. Command Line Syntax: • *disable_points*
Disable Axis Systems	Switches off Axis System processing. Command Line Syntax: • *disable_axes*

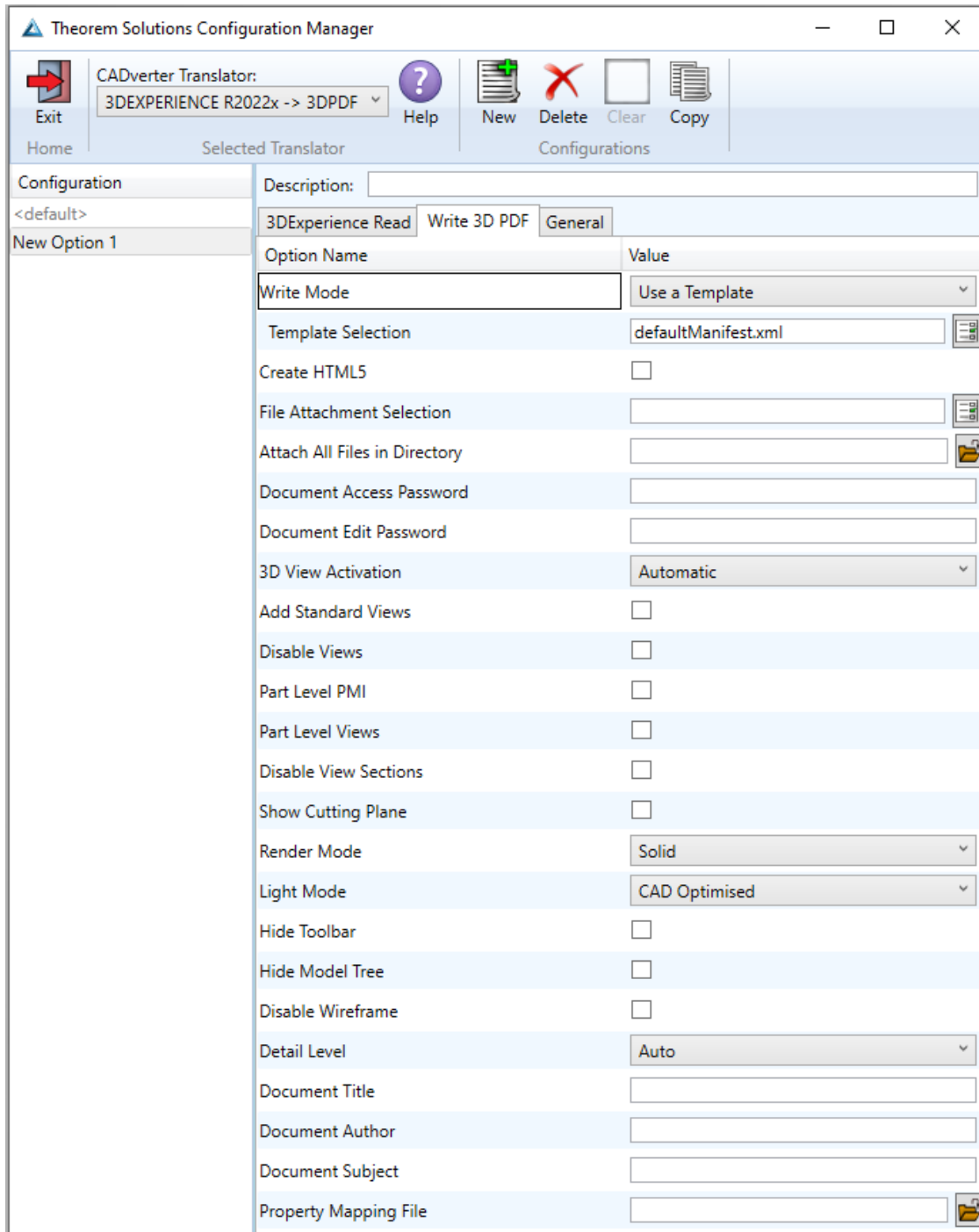|	
Disable Wireframe	Switches off Wireframe processing. Command Line Syntax: • *disable_wireframe*
Disable Surfaces	Switches off Surface processing. Command Line Syntax: • *disable_surfaces*
Disable Solids	Switches off Solids processing. Command Line Syntax: • *disable_solids*
Read As Torus	Read Toroidal surfaces in analytical form (default is NURBS.) Command Line Syntax: • *read_torus*
Read As Cylinder	Read Cylindrical surfaces in analytical form (default is NURBS.) Command Line Syntax: • *read_cylinder*
Read As Cone	Read Cone surfaces in analytical form (default is NURBS.) Command Line Syntax: • *read_cone*
Read As Conics	Read surfaces generated from a Conic in analytical form (default is NURBS.) Command Line Syntax: • *read_conics*
Read Axis Filter	Enables a specified list of axis systems to be processed. Command Line Syntax: • *read_axis <value>* Where *<value>* is the name of the axis system. This can be formatted as per the examples below: • Axis System.1; – Only *'Axis System.1'* will be processed. • *System.2; – Any axis system that includes *'System.2'* at the end of the name will be processed, i.e. Axis System.2 • Axis System*; - Any axis system that includes *'Axis System'* at the start of the name will be processed, i.e. Axis System.1, Axis System.2, Axis System.3, etc • *System*; - Any axis system that includes *'System'* anywhere in the name will be processed. Multiple axis systems can also be processed. This is formatted as per the example below: • Axis System.1;Axis System.2;Axis System.3;

Export Body Names	Maintains body names for parts that consist of multiple bodies. Command Line Syntax: • *body_names*

Write 3D PDF

Each of the options displayed in the image above are described below:

Option	Description
Write Mode	Contains two options: — Use a Template – Allows data to be published to a given template Command Line Syntax: • *publish <XML Manifest file>* — No Template – No template will be used during publishing. If this option is selected, 'Template Selection' is excluded as an option. Command Line Syntax: • *publish off*
Template Selection	Select which templates Publish 3D will use to create the output 3D PDF, if Write Mode is set to *Use a Template* (Default = defaultManifest.xml) Command Line Syntax: • *This is a secondary argument to the publish command. The argument can be a single xml manifest file or a text file containing a list of manifest files*

File Attachment Selection	This option allows files to be attached to the output PDF file together with a brief description of the selected files. (Default is for no files to be attached.)
	Command Line Syntax:
	• *attach_file <Attachment List>*
	
	Where *Attachment_List* is a text file, with a list of full path names followed by an optional description line, For example:
	
	   C:\TEMP\my_image.jpg
	   This is a jpg Image
	   C:\TEMP\report.txt
	   Report Document
	   C:\TEMP\related.pdf
	   C:\TEMP\another.pdf
	   This PDF has a description
	
	Note in this example the 'Related.pdf' attached file doesn't have the optional comment.
	
	The file path may contain environment variables which will be resolved. e.g. %TEMP%/myFile.pdf

Attach All Files in Directory	Attach **ALL** files found in a specified directory. Command Line Syntax: • *attach_files_in_dir <dir>* Where *<dir>* is the location of the files.

Document Access Password	Contains two options selectable via a checkbox:
	
	— No Password – Specifies that no password will be used to encrypt the PDF Document (Default). If this option is selected Password is excluded as an option.
	Command Line Syntax:
	    • *no entry required*
	
	— Use Password – Allow a password to be specified to encrypt the PDF document. If specified, the user will be prompted for the password to open the resultant PDF files.
	Command Line Syntax:
	    • *password <password>*

Document Edit Password	Contains two options selectable via a checkbox:
	
	— No Password – Specifies that no password will be used to controls edits to the PDF Document (Default). If this option is selected Password is excluded as an option.
	Command Line Syntax:
	    • *no entry required*
	
	— Use Password – Allow a password to be specified to controls edits to the PDF Document.
	Command Line Syntax:
	    • *permission_password <password>*

3D View Activation	Controls when the 3D view is activated in the PDF document. Has 3 options:
	
	— Automatic – Activates when the page is opened. (Default) Command Line Syntax:
	    • *activate_mode PAGE_OPEN*
	
	— On Selection – Activates when the user clicks on the model.
	Command Line Syntax:
	    • *activate_mode CLICK (default via the command line)*
	
	— Page Visible – Activates when the page becomes visible to the user
	Command Line Syntax:
	    • *activate_mode PAGE_VISIBLE*

Add Standard Views	Add isometric views to the data being written to PDF (Default is off.) Command Line Syntax: • *add_standard_views*
Disable Views	Switch off any Captures/Views from being written into the PDF. (Default is off.) Command Line Syntax: • *no_views*

Part Level PMI	Enable the processing of PMI on parts within an assembly. (Default is off.) Command Line Syntax: • *part_level_pmi*
Part Level Views	Enable the processing of part level views within an assembly. (Default is off.) Command Line Syntax: • *part_level_views*
Disable View Sections	Switch off any sections within the Captures/Views from being written into the PDF. (Default is off.) Command Line Syntax: • *no_sections*
Show Cutting Plane	Enable cut plane visualisation. (Default is off.) Command Line Syntax: • *cutplane_on*
Render Mode	Controls which Rendering Mode to use for the model. Options are: • Solid (Default) • Transparent • Wireframe • Illustrated • Outline • Shaded Command Line Syntax: • *render_mode <mode>* Where *<mode>* is set to one of the above values.
Light Mode	Controls which Light Mode to use. Options are: • Default • Off • Day • Bright • Prim • Night • Blue • Red • Cube • Head Command Line Syntax: • *light_mode <mode>* Where *<mode>* is set to one of the above values. Note – no command is required for the default value.

Hide Toolbar	Hide the 3D Toolbar in the resultant document. This can be re-enabled in Adobe if required. (Default is off.)
	Command Line Syntax:
	• *hide_toolbar*

Hide Model Tree	Hide the Model Tree in the resultant document. This can be re-enabled in Adobe if required. (Default is off.)
	Command Line Syntax:
	• *hide_model_tree*

Disable Wireframe	Disable Wireframe Processing (Default is to Enable Wireframe processing.)
	Command Line Syntax:
	• *process_wf off*

Detail Level	The Detail Level option can be used to set the render quality of the resultant 3D PDF output. A number of discrete values are made available via the configuration manager. They correspond to a relative chordal deviation (% of the diagonal length across the model bounding box) or absolute chordal deviation, whichever is the smaller. The following options are available:
	• Auto (Corresponds approximately to High)
	• Very High (relative = 0.0001%, absolute = 0.001mm)
	• High (relative = 0.001%, absolute = 0.01mm)
	• Medium (relative = 0.01%, absolute = 0.1mm)
	• Low (relative = 0.1%, absolute = 1mm)
	• Very Low (relative = 0.5%, absolute = 5mm)
	For very large (detailed) parts the use of the options Low or Very Low can significantly reduce the size of the output PDF file and the time taken to produce it, at the cost of quality.
	A setting of Medium is seen as a good compromise, between quality and size. This value is the default value.
	Command Line Syntax:
	• *lod_level <value>*
	Where value = v_low, low, med, high, v_high, off

Document Title	Set the Document Properties 'title' field.
	Command Line Syntax:
	• *doc_description_title <"text">*
	the text should be quoted.

Document Author	Set the Document Properties 'author' field.
	Command Line Syntax:
	• *doc_description_author <"text">*
	the text should be quoted.

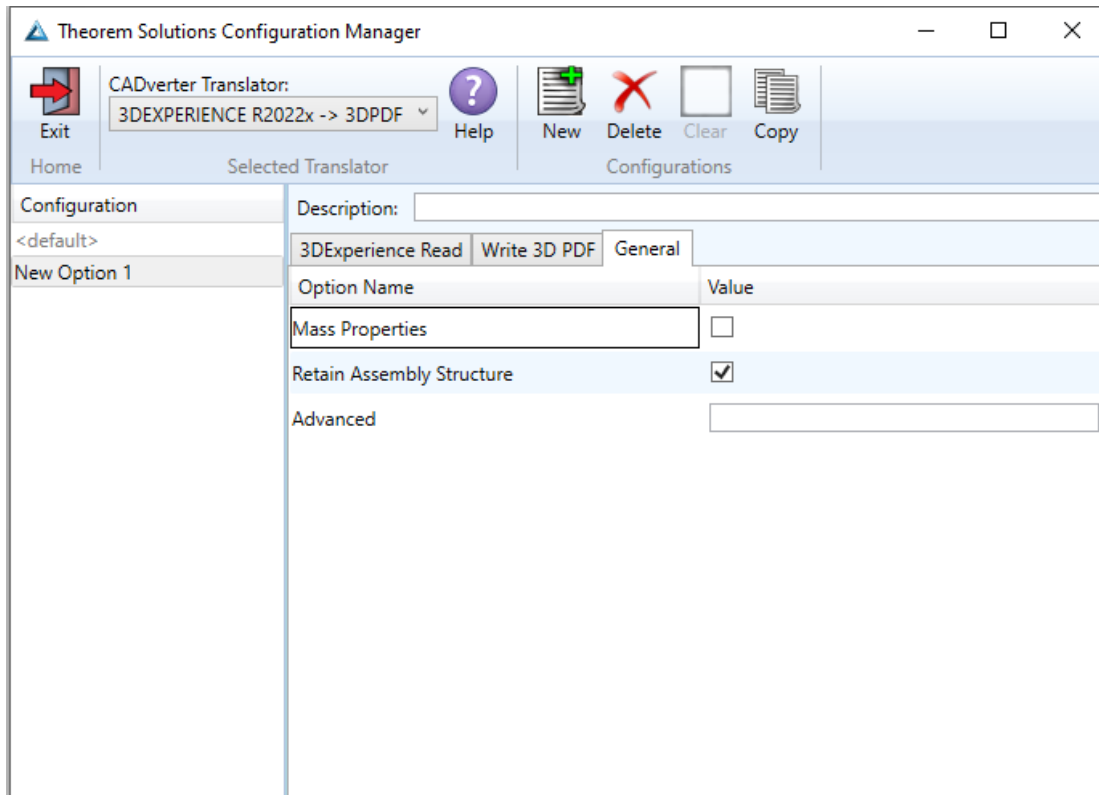|	
Document Subject	Set the Document Properties 'subject' field. Command Line Syntax: • *doc_description_subject <"text">* the text should be quoted.
Property Mapping file	Map CAD properties using a mapping file. Command Line Syntax: • *cad_prop_map_file <file>*

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SOLUTIONS

General

Each of the options displayed in the image above are described below:

Option	Description
Mass Properties	Allows mass property information to be read. Command line syntax: • *mprops*
Retain Assembly Structure	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. Command line syntax: • *off_ditto – to turn off*
Advanced	Allows any of the advanced arguments documented to be added to the configuration manager and applied during the translation.

Advanced Arguments

3DExperience Advanced Arguments

Argument	Description
dont_fill_pmi_arrows	Disables the read of filled arrow information.
dont_fill_pmi_text	Disables the read of filled text information.

3DPDF Advanced Arguments

Argument	Description
design_name	Names the root node after the TAG attribute in the DESIGN entity. Allows the root node to match the CAD system.
progress_file <filename>	Specify the log file for this process.
info	Add 'info' messages to the log file.
no_inst_colours	Switch off the writing of instance colours.
no_colours	Switch off the writing of any colours.
no_attrs	Switch off the writing of attributes into the PDF.
no_pmi_edge_associations	Disables edge highlighting.
no_poster	By default, a 2D poster is generated from the 3D model which can be very time consuming for a complex model. This option disables that process and uses a default JPG called **TS_INST\\data\saveas_3dpdf_images\no_poster.jpg.** The user could replace this JPG file with their own if desired.
background_RGB <r g b>	Where r g b are values 1 to 255, this controls the background colour of the 3D display, which now defaults to pale grey (204,204,204).
pmi_RGB <r g b>	Where r g b are values 1 to 255, this controls the PMI colour. By default, PMI is displayed in the same colour as the source CAD system. When a colour is not read the default will be black. This option allows the PMI colour to be fixed to any colour. This is a single override colour for all PMI.
wireframe_RGB <r g b>	Where r g b are values 1 to 255, this controls the wireframe colour. By default, wireframe is displayed in the same colour(s) as the source CAD system. This option allows the wireframe colour to be fixed to any colour. This is a single override colour for all wireframe.

pmi_flat_to_screen <on/all/off>	This option enables the processing of flat to screen PMI data, the default is off. Please note fonts are not currently supported. **'on'** allows for FLAT TO TEXT types to be processed **'all'** allows all FLAT TO SCREEN types to be processed, however only the text and NOT the frames are processed.
process_wf <value>	*Where value = tess_low, tess_med, tess_high, nurb or off* When this option is selected the wireframe entities are written as lines/conic/nurbs. This results in a wire node for each entity contained under the Wireframe node in the model tree, there could be many wire nodes in a complex model. If any of the settings prefixed tess are used, these options convert wireframe into a number of sampled points on the curve and writes wireframe as a Poly Line, this allows many wires to be combined however a Poly Line can only be a single colour, so wireframe of the same colour will be grouped into a Poly Wire, which will be a wire node contained under the Wireframe node in the model tree. There will be a wire node for each colour, when multi colours exist. If however, all wireframe is the same colour, then all wireframe is contained in the Wireframe node.
attr_filter_file <filter file>	This is a method to define which attributes are masked or included during translation. By default, there is an attribute filter file installed at: *%TS_INST%\ data\publish_3dpdf\attrFilters\ defaultAttrFilter.txt* This command line option allows this file to be overridden, such that the named file is used instead of the default. The file format (in blue) is best explained by means of examples: MPARTNAME,,0,,, (Delete MPARTNAME attribute) FILENAME,F I L E (name),1,,, (Rename 'FILENAME' to 'F I L E (name)) FILESIZE,,3,big,, (Default (always) FILESIZE value to big) *END,,0,,, (Delete all attributes that end with 'END') Theorem*,,0,,, (Delete all attributes that start with 'Theorem')
dump_attr_file <file>	This is a utility which can be used to generate a text file with the Key/Value pairs from the **GCO DESIGN, DITTO** and **DETAIL** entities, this can be used in the 'design' of templates.
compress_tess <value>	Where *value = v_low, low, med, high ,v_high, off* These options can be used to control the tessellation setting, using the values 10, 25, 50, 75, 90. These compression settings apply ONLY to PRC/U3D output files.

add_standard_views_1st	Force views to be first angle projection regardless of source data.
add_standard_views_3rd	Force views to be third angle projection regardless of source data.
views_filter <val>	This option allows a sub-set of views to be included in the output based upon the <val> which describes a part or all of the views name(s). For example: views_filter MBD_* = Includes all views with names beginning '**_MBD_'** views_filter *SECTION = Includes all views with names ending in **'SECTION'** views_filter BLUE = Includes all views with names containing **'BLUE'** Multiple filters can also be applied by adding any number of views_filter arguments. For example: **views_filter A* views_filter *B views_filter FRED** – Includes views that start with an 'A' or end in a 'B'. Views named 'FRED' will also be included.
views_filter_exclude <val>	This option allows a sub-set of views to be excluded from the output based upon the <val> which describes a part or all of the views name(s). For example: views_filter_exclude MBD_* = Excludes all views with names beginning **'MBD_'** views_filter_exclude *SECTION = Excludes all views with names ending in **'SECTION'** views_filter_exclude BLUE = Excludes all views with names containing **'BLUE'** Multiple filters can also be applied by adding any number of views_filter_exclude arguments. For example: **views_filter_exclude A* views_filter_exclude *B views_filter_exclude FRED** – Excludes views that start with an 'A' and end in a 'B'. Views named 'FRED' will also be excluded. Note – It is possible to use both include and exclude filters in the same translation. For Example **views_filter A* views_filter_exclude AXEL** – Includes views that start with an 'A' but excludes views named 'AXEL'

add_meta_data <default or <file>>	Option to append the attributes in PDF with useful data, like part count, which can be consumed by the templates.
	
	<default> - just add part count
	<file> - add the part count and process the file, such that 3 lines are processed.
	
	    part_name // matches the detail (part name)
	    attr_name // name of attribute to be added
	    attr_value // value of the attribute to be added

move_subassy_pmi <on/off/all>	Move the sub assy PMI to top level. **'all'** will reposition all PMI into global space which can be used for true assemblies.
copy_part_attrs <on/off>	Attributes on parts can be copied on to instance nodes with this option (default is off). (The user can enable dump_attr_file <file> option to confirm what attributes are available on part/instances, to check the effect of this option.)
cutplane_capping_off	Disable section capping. (Default is on)
cutplane_RGB <r g b>	When section cutting planes are enabled, the cutting plane colour can be controlled by r g b values in the range of 1 to 255.
cutplane_opacity <value>	When section cutting planes are enabled, the cutting plane opacity can be specified by setting **'value'** between 0.0 and 1.0.
cutplane_intersections	Section cutting intersections can be enabled via this option. (Default is off)
cutplaneIS_RGB <r g b>	When section cutting plane intersections are enabled, the cutting plane intersection colour can be controlled by r g b values in the range of 1 to 255.
enable_brep	By default, BREP data is omitted when the PDF file is generated so this option allows for such data to be retained. **Do not use this option when the generation of BREP data in PDF is forbidden for legal or commercial reasons.**
fixup	There are occasions when the source data presented for publishing to the 3D PDF is invalid. In these circumstances, a validation of the data is carried out by the translator and the user is advised, via a message on the screen to re-run the translator, specifying the **'fixup'** command line option. Specifying this option will allow the translation to complete, omitting any faces that are not topologically correct.

fix_inst_names <on/off>	By default, this option is on. It will detect instances with no names and apply the part name to those instances. This accommodates cases where instance names are not available in the source data.
launch_adobe_viewer	This option will automatically display the resulting PDF file in an Adobe Reader. The environment variable **TS_CMD_PATH_AND_EXE** can be used to specify the Adobe application if it isn't found in a default location.
group_solids	This can be useful when data is created with many open solids (1 per face). This can result in many 100's or even 1000's of solids which causes the 3D PDF file to have a massive entity tree, to be slower and larger than necessary, and with a Bounding Box for each face. Setting this option groups the solids into logical entities.
dont_group_solids	Some systems will group solids by default, so this option allows that setting to be disabled.
pmi_assoc_limit <off or number>	This option is used to limit the number of PMI to Geometry associations that exist for any single PMI node. The default value being 500. It has been found that in rare cases many 1000's of associations exist and this has a significant effect on the translation time. If the limit is exceeded the log file reports the PMI entities names and reports an error, allowing the user to modify the limit, or review the PMI's validity.
set_perm <text option>	Set the permissions for the PDF file, the options being '**PRINT'** or '**LOCK'**, if no options are set then the create PDF file is fully open. The '**PRINT'** setting allows the file **ONLY** to be printed. The '**LOCK'** setting ensures the document cannot be modified. (Be aware that with Adobe Writer it is possible to override any permission setting).
set_perm_mask <int value>	This allows the full control of all the permission settings when creating a PDF file, the in values need to be logically ORed together so 0x2 and 0x1 become 3. values are 0xHex (decimal) 0x01 (1) - **DocumentPermOpen** The user can open and decrypt the document. 0x02 (2) - **DocumentPermSecure** The user can change the document's security settings. 0x04 (4) - **DocumentPermPrint** The user can print the document. Page Setup access is unaffected by this permission, since that affects Acrobat's

preferences - not the document's. In the Document Security dialog, this corresponds to the Printing entry.

0x08 (8) - **DocumentPermEdit** The user can edit the document more than adding or modifying text notes (see also DocumentPermEditNotes). In the Document Security dialog, this corresponds to the Changing the Document entry.

0x10 (16) - **DocumentPermCopy** The user can copy information from the document to the clipboard. In the document restrictions, this corresponds to the Content Copying or Extraction entry.

0x20 (32) - **DocumentPermEditNotes** The user can add, modify, and delete text notes (see also DocumentPermEdit). In the document restrictions, this corresponds to the Authoring Comments and Form Fields entry.

0x40 (64) - **DocumentPermSaveAs** The user can perform a Save As.... If both DocumentPermEdit and DocumentPermEditNotes are disallowed, Save will be disabled but 'Save As...' will be enabled. The 'Save As...' menu item is not necessarily disabled even if the user is not permitted to perform a Save As...

0x100 (256) - **DocumentPermFillandSign** Overrides other DocumentPerm bits. It allows the user to fill in or sign existing form or signature fields.

0x200 (512) - **DocumentPermAccessible** Overrides DocumentPermCopy to enable the Accessibility API. If a document is saved in Rev2 format (Acrobat 4.0 compatible), only the DocumentPermCopy bit is checked to determine the Accessibility API state.

0x400 (1024) - **DocumentPermDocAssembly** Overrides various DocumentPermEdit bits and allows the following operations: page insert/delete/rotate and create bookmark and thumbnail.

0x800 (2048) - **DocumentPermHighPrint** This bit is a supplement to DocumentPermPrint. If it is clear (disabled) only low quality printing (Print As Image) is allowed. On UNIX platforms where Print As Image doesn't exist, printing is disabled.

0x8000 (32768) - **DocumentPermOwner** The user is permitted to perform all operations, regardless of the permissions specified by the document. Unless this permission is set, the document's permissions will be reset to those in the document after a full save.

0x10000 (65536) - **DocumentPermFormSubmit** This should be set if the user can submit forms outside of the browser. This bit is a supplement to DocumentPermFillandSign.

0x20000 (131072) - **DocumentPermFormSpawnTempl** This should be set if the user can spawn template pages. This bit will allow page template spawning even if DocumentPermEdit and DocumentPermEditNotes are clear.

0xFFFFFFFF - All permissions.

expand_part	This option creates a pseudo assembly structure below a part, such that each solid is modelled by a separate node. The default is to combine all solids that belong to the same part. Using this option can be useful when control over part with multiple solids is required, e.g. view/hide separate solids, however it should be noted that this option may compromise other functionality, such a PMI association.
view_part_name	Prefix a part view within an assembly with the corresponding part name, to help make the view names unique – only need this when the option 'Part Level Views' is also enabled.
zoom_views_x <value>	This option will magnify the view by the value provided, this is defaulted to 2.0 for all CAD systems. Values < 1.0 zoom out. Values > 1.0 zoom in. The purpose of this option is to expose an adjustment factor that a given CAD system might need to best fit the VIEWS in the PDF, this may be affected by the aspect ratio of the page size/template used in PDF write. Every effort has been made to set a good value by default.
optimize_save off	By default, the PDF will be automatically optimized to make The file smaller. This disables the optimization feature.
image_dir <directory>	Import images into a template from a directory.
axis_systems <on>/auto/<size in mm>	Enables the output of axis systems to the PDF. Default is on. The Default value when on is 25mm. When enabled wireframe x(RED),y(GREEN),z(BLUE) are created to represent the axis system location. Using auto will enable the size of the axis system to be generated based upon the bbox within a 5-25mm range. The length of the axis system can also be given in mm, e.g. axis_systems 100 which specifies a length of 100mm for example.

ts_cfile <text file>	This option allows command line arguments to be placed in a text file, with one command or option per line, for example: read_pmi dont_fill_pmi_text disable_points pmi_RGB 255 1 1 wireframe_RGB 255 1 1 Note – the progress_file <file> option cannot be added into the text file because the progress file is opened before the command file is read.
attach_2D_views <on or WxH>	It is possible to automatically generate JPG images for each view and attach them to the output PDF file. This is off by default. If it is enabled, then the options are: *'on'* - this generated a JPG based upon a WxH of 1600x1200. *'WxH'* - allows the image size to be defined e.g. 1200x1800. **The view name is prefixed with the input PDF file name to name the image files.**
attach_2D_views_name_only <on or WxH>	It is possible to automatically generate JPG images for each view and attach them to the output PDF file. This is off by default. If it is enabled, then the options are: *'on'* - this generated a JPG based upon a WxH of 1600x1200. *'WxH'* - allows the image size to be defined e.g. 1200x1800. **ONLY the view name is used when creating the image files.**
cutPlane_FIX <x_val y_val z_val>	This is an option that can be added to resolve section capping issues, where *x_val, y_val, z_val* are the values that can be added to move the cutplane along the desired axis. For example cutPlane_FIX 0.0 0.2 0.0 will move the cutplane by 0.2mm along the y axis. Negative values can also be added, for example cutPlane_FIX 0.0 -0.2 0.0 will move the cutplane by 0.2mm in the opposite direction along the y axis. Only slight adjustments will typically be required, usually between -0.5 and 0.5. Please note that higher values may be required from time to time.

Templates

Publishing to a Specific List of Templates – via the Configuration Manager

It is possible to publish to a single template or to multiple templates via one translation using the Configuration Manager. To do this create a new configuration as described earlier in this user guide, (See Theorem Interactive Conversion Settings), or use an existing configuration if one has been created previously. Having a user generated configuration will enable the user to change the options specified including the template selection. By default, the template selected is defaultManifest.

To change the template, select the *'Write 3D PDF'* tab to display the Write Mode and Template Selection options. Ensure that *'Use a Template'* is selected, then select the Icon next to the Template Selection field.
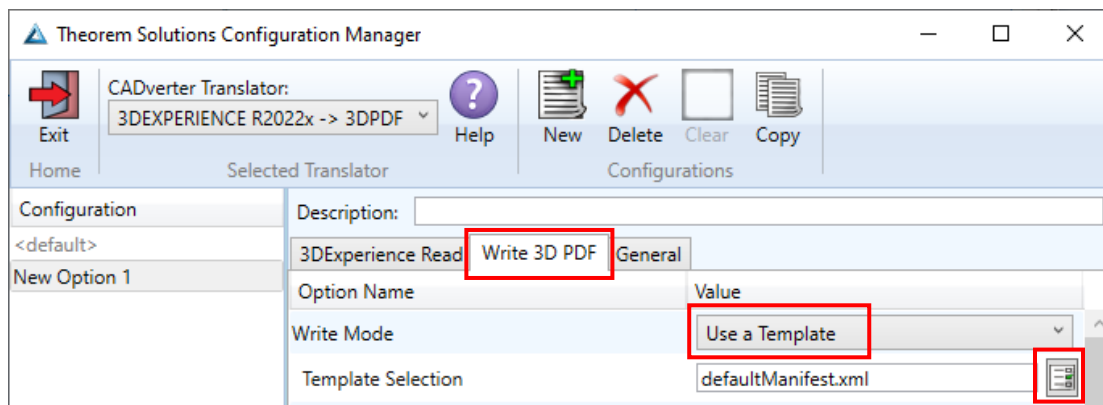
In the Template selection manager window select the icon. Navigate to the required directory and select the template XML file to add it to the list.
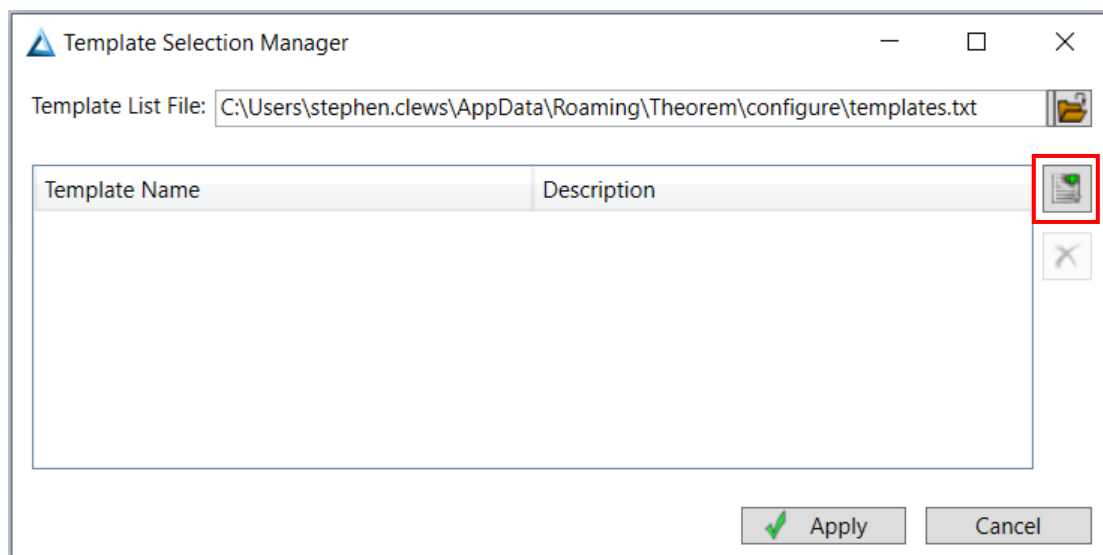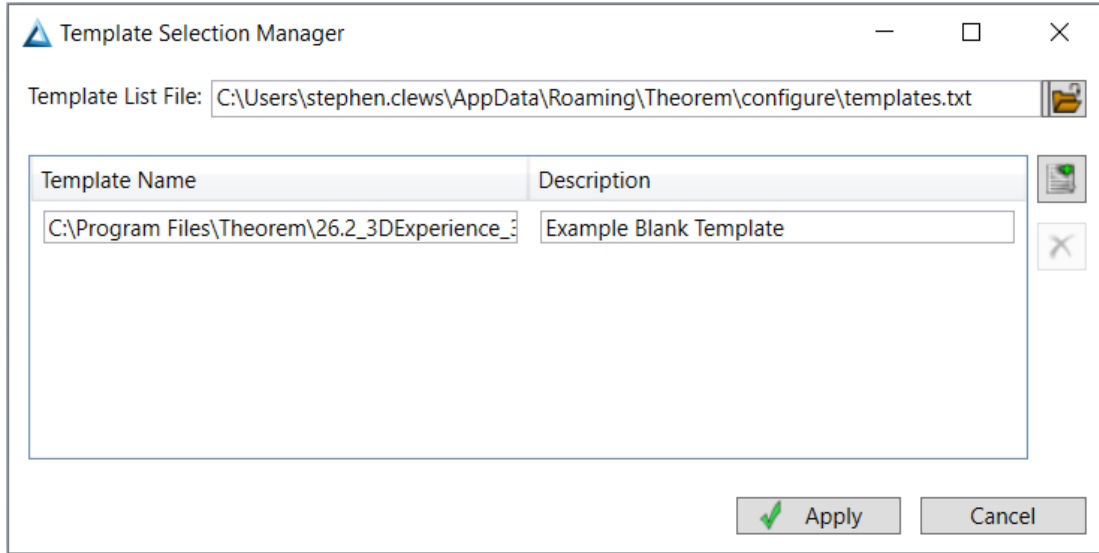
Repeat the process to add more templates if required. Select apply to update the Template Selection.
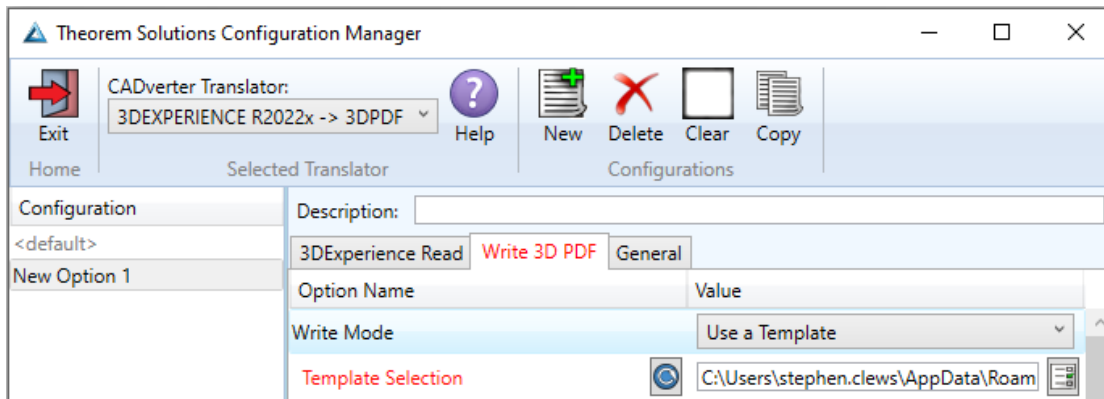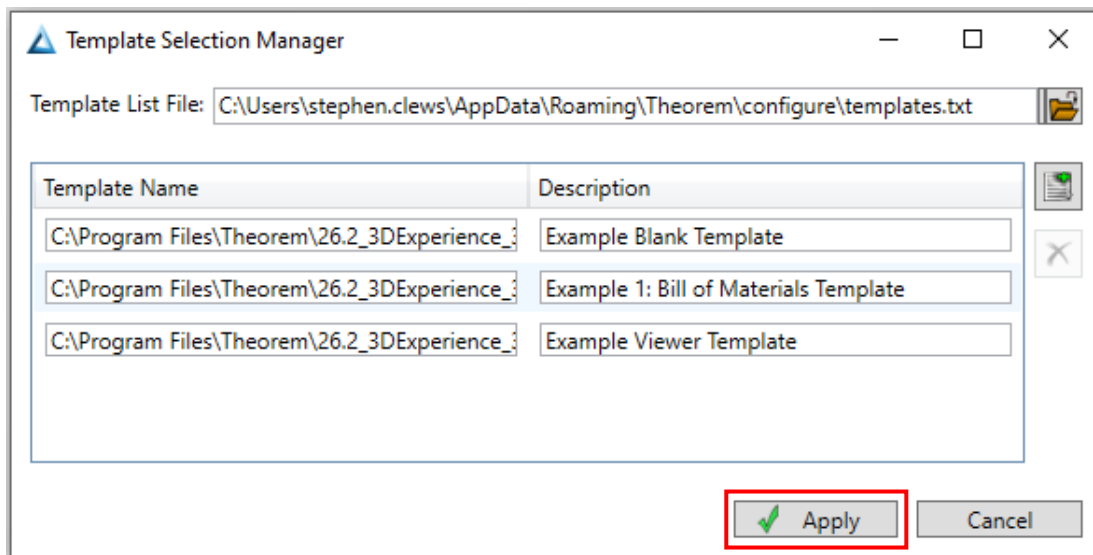
The configuration can then be selected from the Active Configurations list shown previously. (See Theorem Interactive Conversion Settings.)

Publishing without a Template – via the Configuration Manager

Running the translation without a template will create a 3D PDF output file with no containing document. To do this in the configuration manager, select the Write 3D PDF tab then from the *'Write Mode'* dropdown, select *'No Template'*.
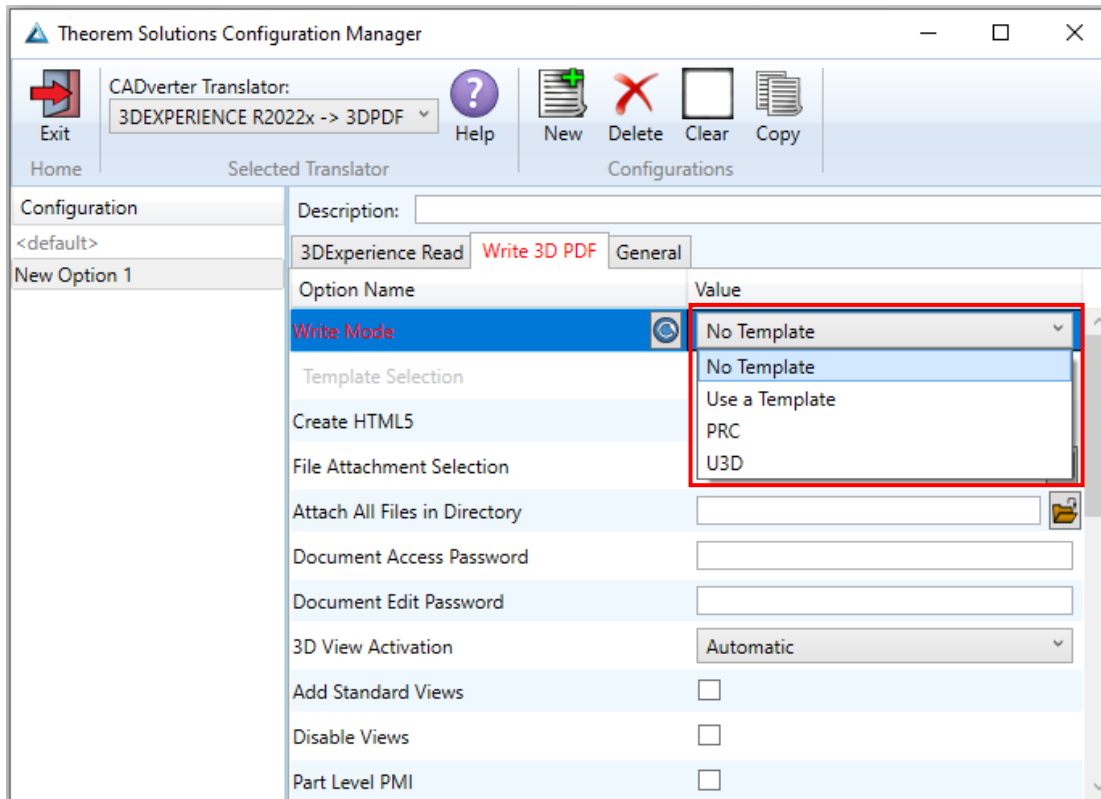
Translating in Batch using CATUtil – DataExchangePLMBatch

Launching DataExchangePLMBatch

The Dassault Systemes CATUtil Batch Monitor 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_3DPDF_CATIAV6R<revision>.cmd

(where *<revision>* should be substituted for the version of 3DExperience that you have installed – e.g. 2021x, 2022x, 2023x.)

It can also be started from the Tools>Utility workbench within an Interactive 3DExperience session.

Batch Export to 3D PDF

Here is an example showing what is required in preparation for a 3DExperience database selection and batch export.

Double click on **DataExchangePLMBatch**.

Ensure that **'Export'** is selected as the data exchange operation, **'PDF_THEOREM'** is selected as the usage and **'pdf'** is selected as the extension.

To add the required products for the batch export, select the 3DExperience Platform Object Search icon.

In the subsequent window displayed, add the required products using the correct attribute, operator and value ensuring that the correct object type has been selected. (E.g. Physical Product). Where required use an Asterix in the value field to identify multiple parts or assemblies.

For multiple products ensure that the **OR** condition has been selected. Select Add results and click Apply to add the list of products to the Object Selection field.

To change the location where both the report and the PDF files will be saved, select the [folder icon] icon next to the Report directory field, then select the required directory.

Click Run to initiate the Batch Export.

Changing the Options Selected for a Batch Export

The default options selected for a batch export can be modified within the **xcad_3dpdf_opts.txt** file available in the following location.

<installation_directory>\data\3dpdf

Within the file a number of arguments are displayed, these include common arguments such as read_pmi which is turned on by default and disable_points, disable_wireframe and disable_axes which are turned off by default. If an argument has the character (!) at the start of the line then this means that the argument is turned off.

```
22   !!!!!!!!!!!!!!!!!!!!!
23   ! V6 Read options
24   !!!!!!!!!!!!!!!!!!!!!
25   !disable_points
26   !disable_wireframe
27   !disable_surfaces
28   !disable_solids
29   !disable_axes
30   read_pmi
31   new_captures
32   !read_axis
33   !read_geometry
34   ! Write the filter for the axis system(s) here (if multiple separate with a ";")
35   !part_level_views
36   !part_level_pmi
37   ! To read geometry in Noshow (hidden) state, uncomment the following line.
38   !noshow
39   ! To read surfaces in analytic form rather than default NURBS form, uncomment the following lines.
40   !read_torus
41   !read_cylinder
42   !read_cone
43   ! To read conic curves in analytic form rather than default NURBS form, uncomment the following lines.
44   !read_conics
45   ! To create output tree names based on CATIA default names for systems using instance naming (e.g. 3dpdf)
46   instance_naming #Ref#(#Inst#)
47   !
```

To turn on a particular argument remove the (!) from that particular line. Consequently to turn off an argument add the (!) to the start of the line. Note – the user will need to be in administrator mode in order to make the change.

```
22   !!!!!!!!!!!!!!!!!!!!!
23   ! V6 Read options
24   !!!!!!!!!!!!!!!!!!!!!
25   disable_points
26   disable_wireframe
27   !disable_surfaces
28   !disable_solids
29   disable_axes
30   !read_pmi
31   new_captures
32   !read_axis
33   !read_geometry
34   ! Write the filter for the axis system(s) here (if multiple separate with a ";")
35   !part_level_views
36   !part_level_pmi
37   ! To read geometry in Noshow (hidden) state, uncomment the following line.
38   !noshow
39   ! To read surfaces in analytic form rather than default NURBS form, uncomment the following lines.
40   !read_torus
41   !read_cylinder
42   !read_cone
43   ! To read conic curves in analytic form rather than default NURBS form, uncomment the following lines.
44   !read_conics
45   ! To create output tree names based on CATIA default names for systems using instance naming (e.g. 3dpdf)
46   instance_naming #Ref#(#Inst#)
47   !
```

Adding Options for a Batch Export

To include a new argument, simply add a new line in the **xcad_3dpdf_opts.txt** file under the V6 Read Options section or the 3D PDF Write Options section depending on the argument used. (See Configuration Manager and Advanced Arguments for more information on 3DExperience read and 3D PDF write arguments). In the example below the ***pmi_RGB 255 1 1*** argument has been added.

```
53    !!!!!!!!!!!!!!!!!!!!!
54    ! 3D PDF Write options
55    !!!!!!!!!!!!!!!!!!!!!
56    ! to present solids and bodies as separate Model Tree items
57    ! uncomment the following 2 lines
58    !body_names
59    !multi_solids
60    ! to use part names for leaf nodes in output tree
61    use_part_names on
62    !
63    pmi_filled_text on
64    !
65    ! To calculate view camera scope from geometry uncomment the following line.
66    !calc_views_camera
67    !
68    ! To name route node from DESIGN ent
69    design_name
70    !
71    ! To display the axis systems
72    axis_systems on
73    !
```

```
53    !!!!!!!!!!!!!!!!!!!!!
54    ! 3D PDF Write options
55    !!!!!!!!!!!!!!!!!!!!!
56    ! to present solids and bodies as separate Model Tree items
57    ! uncomment the following 2 lines
58    !body_names
59    !multi_solids
60    ! to use part names for leaf nodes in output tree
61    use_part_names on
62    !
63    pmi_filled_text on
64    !
65    ! To calculate view camera scope from geometry uncomment the following line.
66    !calc_views_camera
67    !
68    ! To name route node from DESIGN ent
69    design_name
70    !
71    ! To display the axis systems
72    axis_systems on
73    !
74    pmi_RGB 255 1 1
```

Changing the Template Selected for a Batch Export

The template selected for a batch export can also be modified within the **xcad_3dpdf_opts.txt** file.

To change the template, comment out the STD line by adding the character (!), then add in the file path where the template xml file is saved on a new line below. Use quotation marks if there are spaces in the file path.

These changes will consequently mean that the user specified template will be used instead of the default template. Note – the user will need to be in administrator mode in order to make the change.

```
13    ! First option is PUBLISH or VWR_3DPDF
14    PUBLISH
15    ! if PUBLISH, next option is STD, or a file name
16    STD
17    ! Now any additional tail-end options follow
18    ! General Diagnostic options
```

```
13    ! First option is PUBLISH or VWR_3DPDF
14    PUBLISH
15    ! if PUBLISH, next option is STD, or a file name
16    !STD
17    "C:\Program Files\Theorem\26.2_3DExperience_3DPDF\data\publish_3dpdf\templates\Blank.xml"
18    ! Now any additional tail-end options follow
19    ! General Diagnostic options
```

Translating on the Command Line

It is possible to run a 3DExperience to 3D PDF translation on the command line, however as all CAD data is saved in the 3DExperience database an XML file will need to be used as the input file instead of a CAD file. This XML file contains all the necessary information to locate, open and translate the CAD data. This is explained in more detail below. In order to run a translation on the command line the following is required: -

<TS_INST>\bin\catia6r<revision>_3dpdf.cmd <input_file> <output_file> publish <XML Manifest file> <options>

Where: - <TS_INST> is the Theorem Solutions software installation directory.
Where: - <revision> is the version of 3DExperience that you have installed – e.g. 2021x, 2022x, 2023x.

<input_file>

Is an xml file defining access to a specific object in the 3DExperience database.

This file provides user login details (V6R2013x) or a Login Ticket (2015x onwards) (see appendix A), to a specified Enovia repository plus a set of 3 attribute names and values which will uniquely identify the input object required for the conversion. These are highlighted in green in the example below and will need changing to the object and revision being used for the translation. The lines highlighted in yellow should be modified with 'your' login details which are required to access Enovia. Examples of the input file XML can be found in the *<TS_INST>\samples\3dexperience* folder. These can be modified and saved on your local drive.

Here is an example of the xml input file for 3DExperience

```xml
<?xml version="1.0" encoding="utf-8"?>

<mc6_read>
 <!-- parameters must be in this ORDER -->
 <!-- parameters only the value passed to V6  -->
 <parameters>
  <attribute name="repository" value="PLM1"/>
  <attribute name="ServerName" value="3dspace.theorem.com"/>
  <attribute name="ServerPort" value="447"/>
  <attribute name="ServerRootURI" value="3dspace"/>
  <attribute name="LoginTicket" value="REEyNzM3M0Q1REM2NDgxQzlFNzk1QzIwNjZGRDYzODN8Um9iaW58Um9iaW58fHwwfA=="/>
  <attribute name="PLMType" value="VPMReference"/>
  <attribute name="ReportDirectory" value="C:\TEMP\V6Export"/>
  <attribute name="LicenseData" value="LIC"/>
  <attribute name="BatchXMLFileName" value="Default"/>
 </parameters>
 <!-- arguments both key and value are passed to V6 -->
 <arguments>
  <attribute key="PLM_ExternalID" value="prd-Interfix"/>
  <attribute key="V_version" value="A"/>
  <attribute key="minorrevision" value="1"/>
 </arguments>
</mc6_read>
```

Note – In some cases, the "minorrevision" value is not required in the XML file, only the V_version value is required. This value could be A.1 for example.

<output_file>

 is the required PDF output file name.

publish <XML Manifest file>

 This is the template that the data will be published to. A text file containing a list of manifest files can also be used to publish to multiple templates.

<options>

 Options displayed in the configuration manager and advanced arguments can be added at the end of the command using the correct syntax, e.g. disable_points, disable_wireframe. (See Configuration Manager and Advanced Arguments for list of command line syntaxes.)

```
C:\Users\stephen.clews>"C:\Program Files\Theorem\25.4_3DExperience_3DPDF\bin\catia6r2022x_3dpdf.cmd"
C:\temp\3dex_2022x_input_Engine_Bracket.xml "C:\temp\ENGINE BRACKET.pdf" publish "C:\Program Files\Th
eorem\25.4_3DExperience_3DPDF\data\publish_3dpdf\templates\publishBlank.xml" disable_wireframe
```

Log File Generation

In the process of exporting the selected 3DExperience part or assembly, the following log files are generated by the PDF Export plug-in.

The process log and error messages are recorded in a '.err' file located in the 3DExperience 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 'data read' processing into Theorem Intermediate data format. Normally a list of entities.

```
List of gco entities :-
----------------------
Type         Total    Standalone   Subordinate
----------   -------  ----------   -----------
Lines        237                   237
Curves       468                   468
Surfaces     189                   189
Planes       81                    81
```

%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 PDF. The status of the translation can be found here

%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 PDF and contains additional information such as modifiers and options used.

3DExperience Environment Files

As part of the Theorem installation process, a set of 3DExperience environment files are created which are subsequently used in the launch of 3DExperience and CATUtil sessions to support the Theorem partner plug-ins for 3D PDF export.

A 'CATEnv' file is created for each installed version of 3DExperience.

These environment files are located in the 3DExperience revision specific folder e.g.

<installation_directory>\B423\win_b64\CATEnv\Theorem_Multi-CAD_3DPDF_CATIAV6R2021x.txt

<installation_directory>\B424\win_b64\CATEnv\Theorem_Multi-CAD_3DPDF_CATIAV6R2022x.txt

<installation_directory>\B425\win_b64\CATEnv\Theorem_Multi-CAD_3DPDF_CATIAV6R2023x.txt

These files consist of the current 3DExperience settings with the required Theorem settings appended at the bottom.

```
!--------------------------------------
! Theorem Additional Multi-CAD Settings
!--------------------------------------
TS_INST=C:\Program Files\Theorem\25.3_3DExperience_3DPDF\
THEOREM_LICENSE_FILE=7601@tscpc04
TSC_TEMP_DIR=%TEMP%
OPTIONS_MULTICAD_PARTNER=1
CNEXTOUTPUT=%TSC_TEMP_DIR%\3DEXPERIENCE_B424_output.log
!TS_MCAD_OPTIONS_PRODUCT_NAMING=C:\Program Files\Theorem\25.3_3DExperience_3DPDF\\data\3dx\export_name_format.txt
MEPREF_LEGACYPANEL=1
```

Appendix A – Creating a Login Ticket

Use a browser link (similar to the one below) to your 3DExperience Server.

https://3dspace.2017x.theorem.com:447/3dspace/common/emxNavigator.jsp
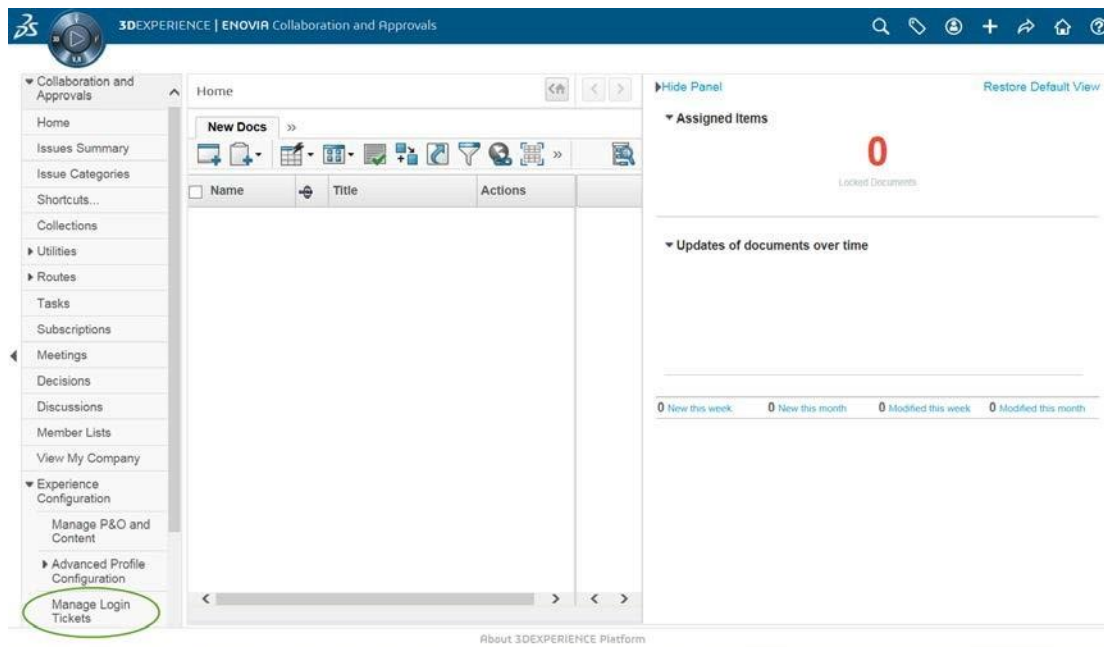
A page will appear in the browser.
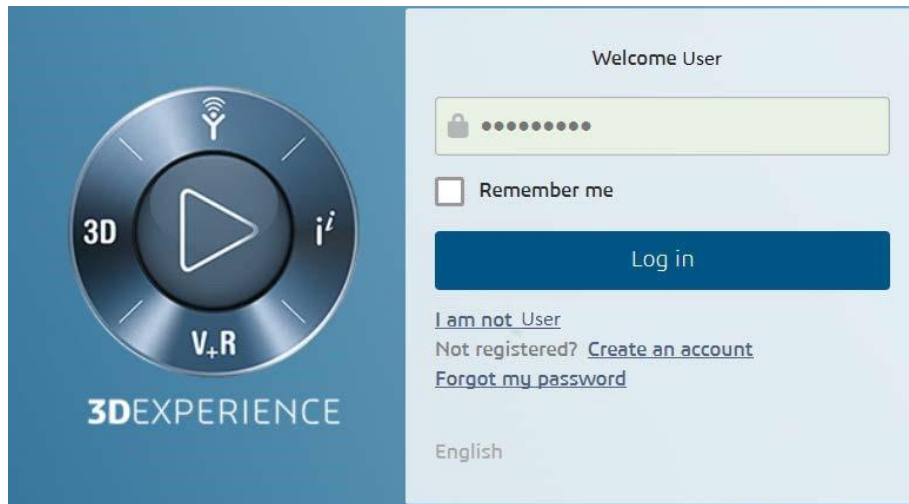
Login as normal, e.g.
>   User
>   Password

Select Collaboration and Approvals > Experience Configuration > Manage Login Tickets.

Select the values required for your user:

User and Security Context should already be set, make sure that Ticket Type: Infinite is selected. Then click on 'Create'.

'Create' will produce the ticket. See example below: -

REEyNzM3STE1MER2NDgxQzlFNzk1QzIwNjZGNATzDAN8Um9iaW58Um9iaW58fHwwfA==

This can then be used as the 'LoginTicket' value explained previously.

Appendix B – Security Changes to Adobe Reader PDF Documents

Adobe have implemented new security options to disable the auto-play of 3D content in PDF files. The following banner will be displayed upon opening a PDF file that contains 3D content:
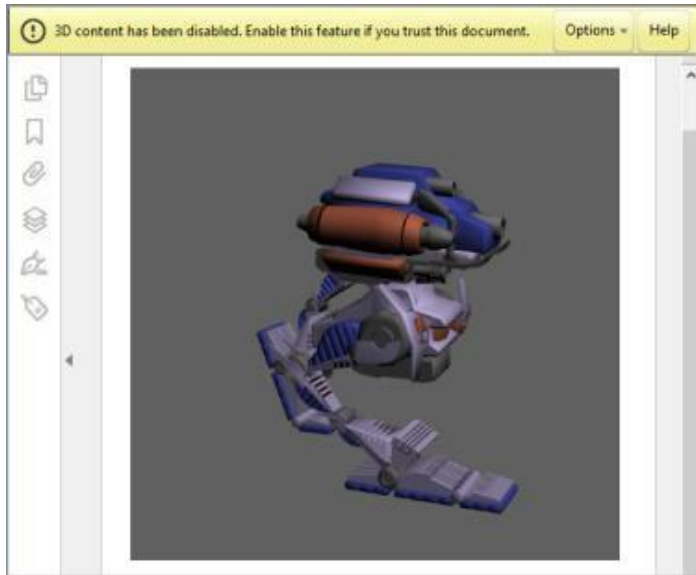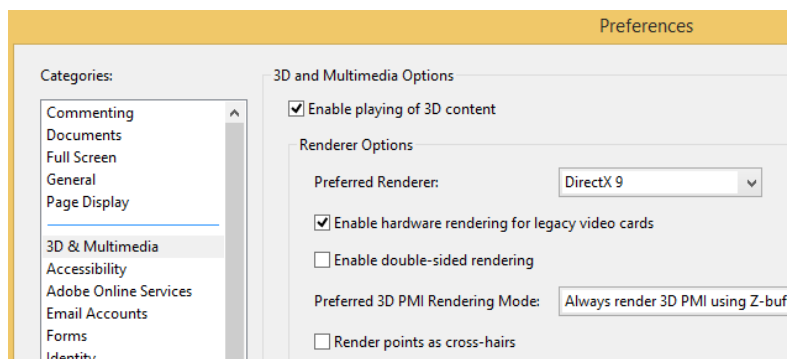

To enable 3D content in the current document:

Click the Options button and then select an appropriate option:

- Trust this document one time only.
- Trust this document always.

To enable 3D content permanently in ALL documents:

- Go to Edit > Preferences > 3D & Multimedia and then select the Enable playing 3D content checkbox.

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