PUBLISH 3D for CATIA V5

USER GUIDE

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Overview of Publish 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.

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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.

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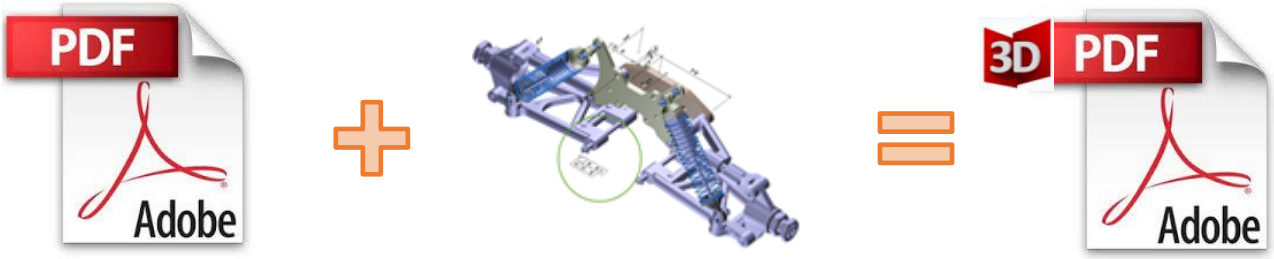
What is a 3D PDF?

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 for sharing
- Support for Model Based Definition (MBD)
- The addition of 3D content to 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 more information on 3D PDF.

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What is Publish 3D?

Theorem's **Publish 3D** 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, Publish 3D 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 Publish 3D 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 Publish 3D please contact sales@theorem.com

What does Publish 3D provide?

A number of Adobe 3D PDF products and solutions are now available as part of the Theorem Solutions Publish 3D suite of applications:

- Publish 3D - **Interactive**
  - 3D PDF Publishing from within CAD Applications:
    - 3DExperience / CATIA V5 / Creo / NX
- Publish 3D - **On Demand**
  - Batch Publishing for:
    - 3DExperience / CATIA V5 / Creo / NX / JT
- Publish 3D - **Automated**
  - High volume batch 3D PDF Publishing Complex Workflow Support

The remainder of this document will focus on Publish 3D Support for CATIA V5 to 3D PDF. For further information on other Theorem Publish 3D Products please contact sales@theorem.com

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Publish 3D supplied templates

The "**Interactive**" Publish 3D for CATIA V5 product in its basic form is limited to translating source data using a default template.

The configuration manager add on allows a user to create custom configurations using other templates alongside the interactive product functionality.

The "**On-Demand**" product will allow the use of other templates without the need for the configuration manager add on plus allow further translation options such as attachments.

Example templates are provided in the following location:

<installation_directory>\data\publish_3dpdf\templates

The following examples are available:

- publishAttachments – Illustrates adding attachments to a template
- publishBOM – Illustrates how to output an Assembly Bill of Materials
- publishMotion – Illustrates using CAD data with motion in PDF
- publishViewer – Illustrates JavaScript additions to a template replicating a CAD viewer

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".

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Getting Started

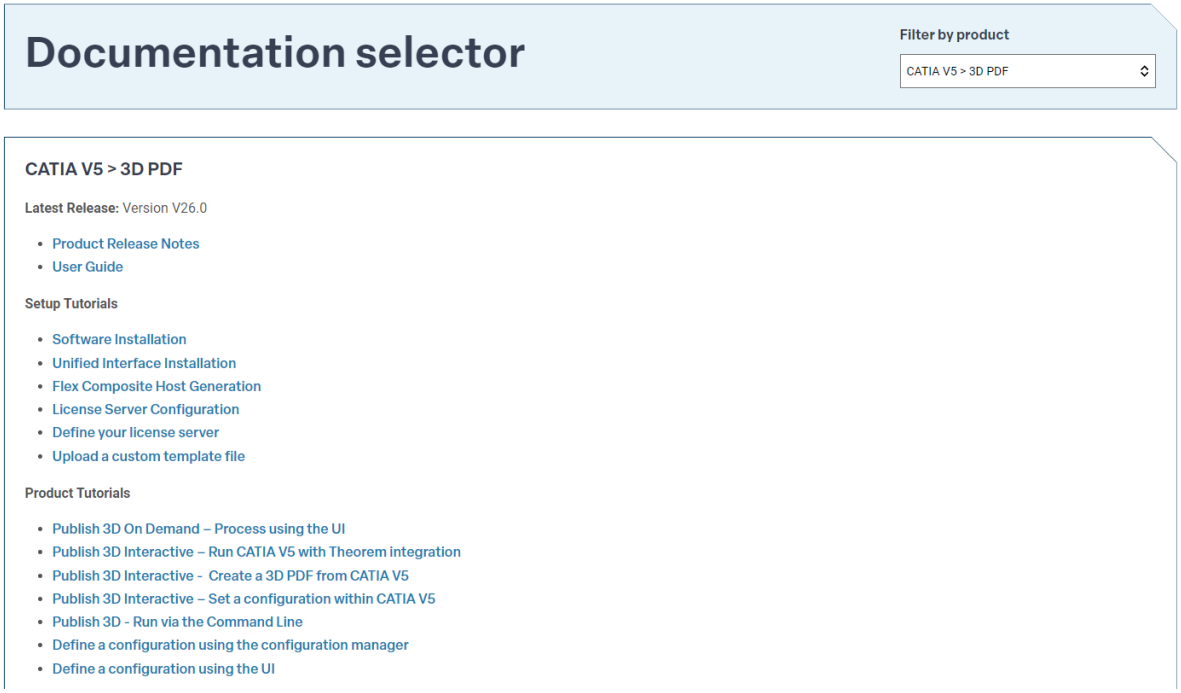
Documentation & Installation Media

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 software.

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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

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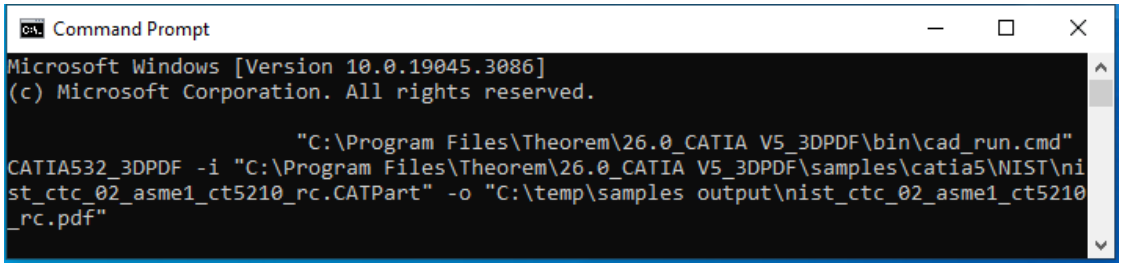
Translating on the Command Line

Default Translation on the Command Line

Running a translation via the command line can be carried out via the ***cad_run.cmd*** file located in the ***<installation_directory>\bin*** directory. The format of the command is as follows (Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for V5 R32, change to CATIA532):

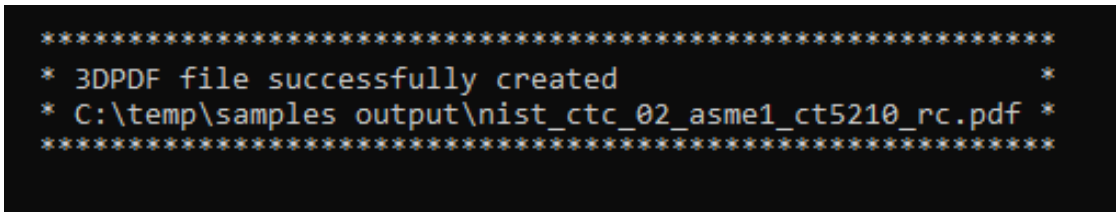
<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_3DPDF –i <input_file> -o <output_file>



The example above will translate the file to the output location specified. In this case:

C:\temp\samples output\nist_ctc_02_asme1_ct5210_rc.pdf

The following screen output should be expected when successfully translated:



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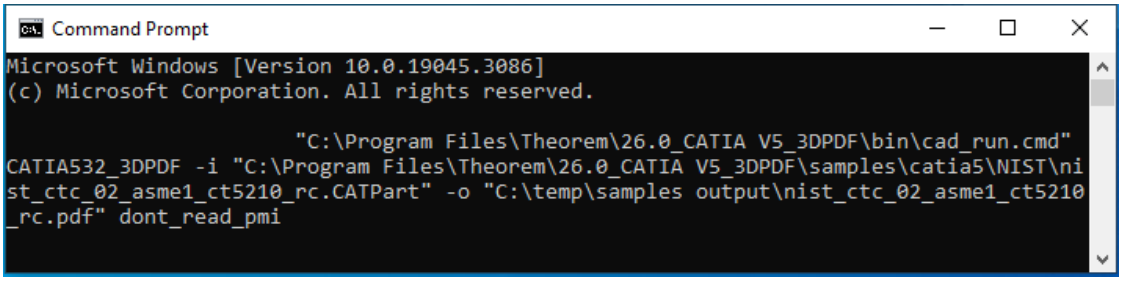
Translating with Options on the Command Line

Options can also be added to a command in the command prompt window using the correct syntax. These are explained in more detail in the Configuration Manager and Advanced Arguments section.

The format of the command with options included is as follows (Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for V5 R32, change to CATIA532):

<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_3DPDF –i <input_file> -o <output_file> <options>



The example above will translate the file to the output location specified. In this case:

C:\temp\samples output\nist_ctc_02_asme1_ct5210_rc.pdf

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Translating in the Unified Interface

Default Translation in the Unified Interface

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

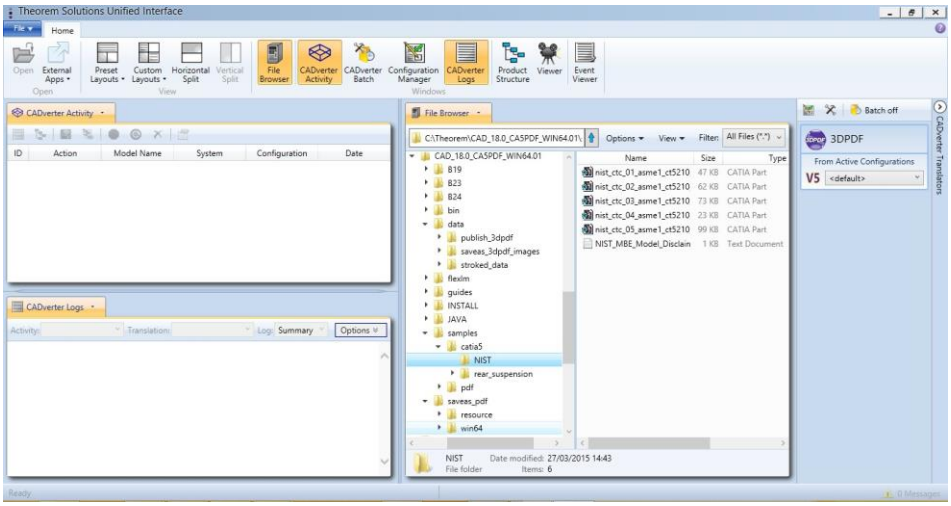
Alternatively, it can be started via the script provided in the UI installation located in:

<UI_installation_directory>\bin

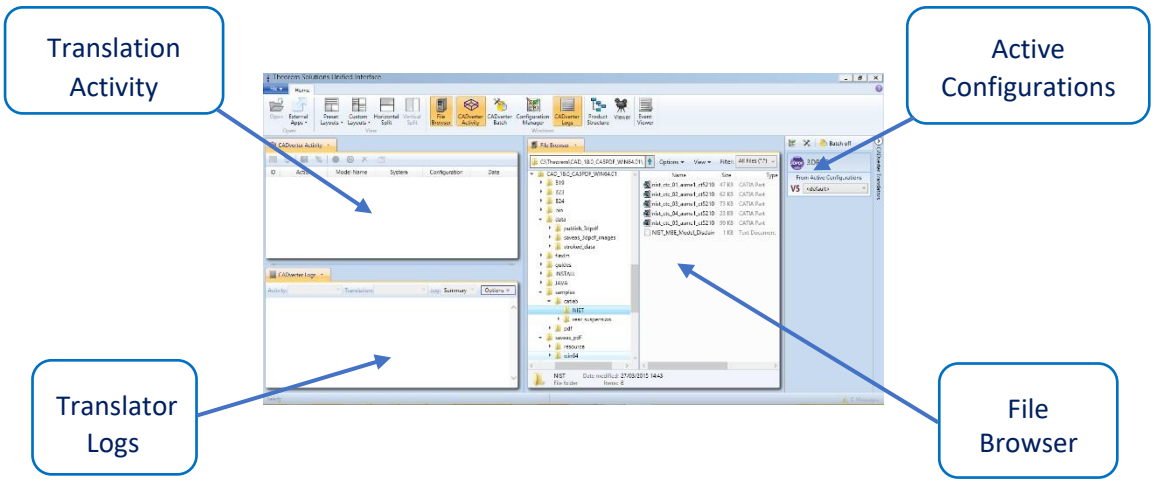
The script name is:

Unified_Interface.cmd

The following interface will be launched:



The default layout is split into 4 primary areas, which can be altered as per the user's preference:



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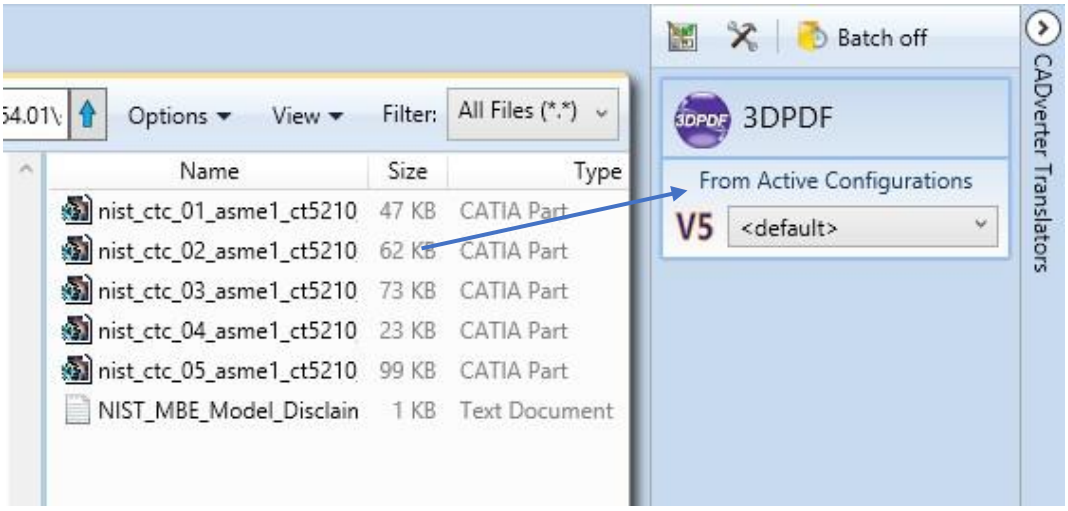
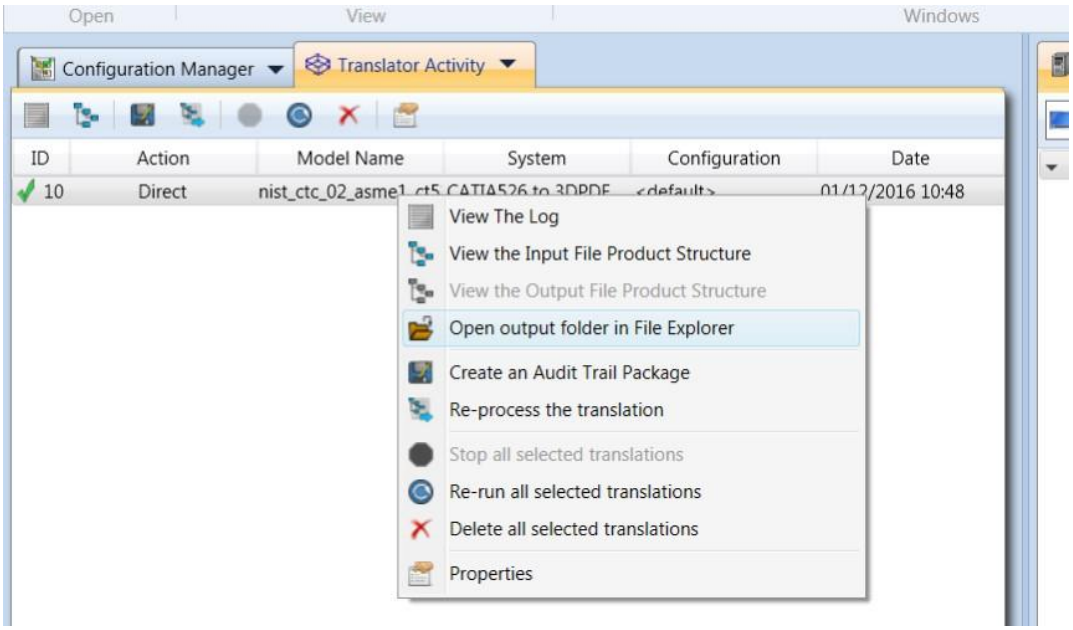
The simplest way to translate a CATIA V5 file to a 3D PDF is to drag the file from the file Browser tab on to the 3D PDF Active Configuration. This will create a PDF file based upon the default template.



On completion, the Unified Interface will display the activity information and details from the log file created during the translation, in the Translation Activity and Translator Logs tabs respectively.

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



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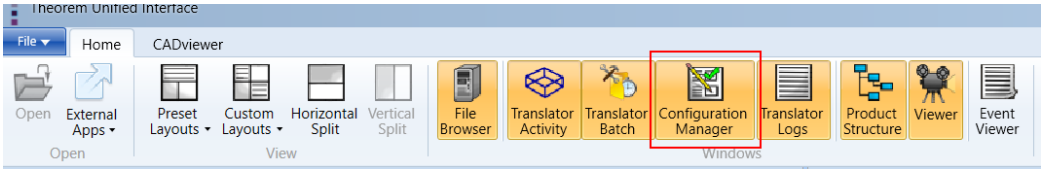
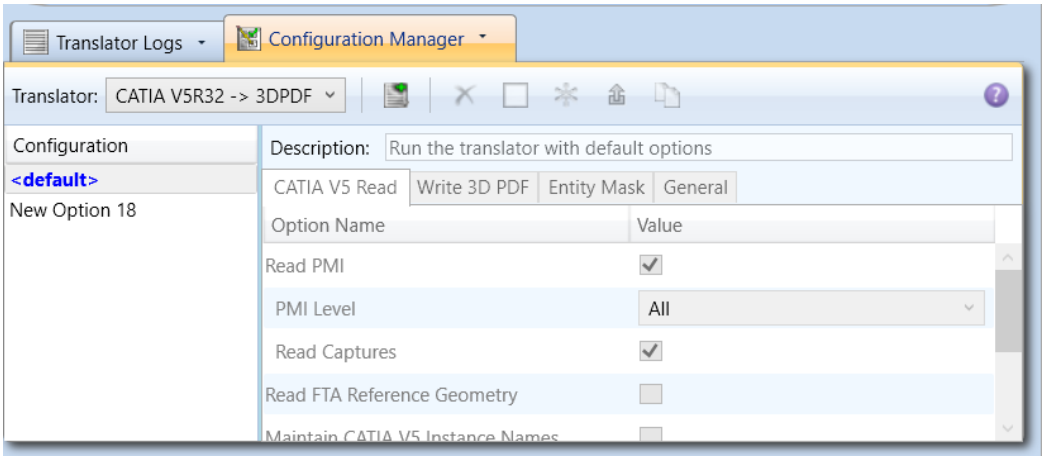
Accessing the Configuration Manager in the Unified Interface

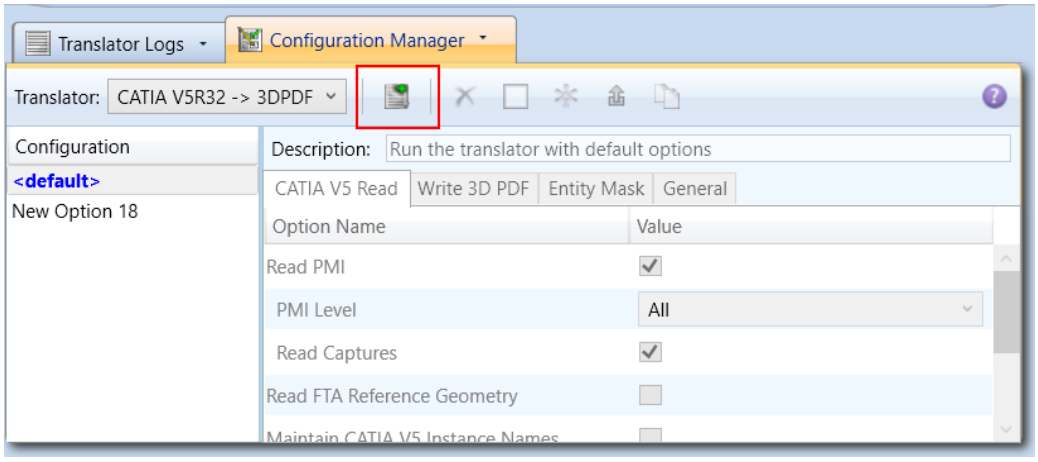
To access the configuration manager in the UI, ensure that the Configuration Manager option has been toggled on in the ribbon at the top. The configuration manager option is visible under the Home tab.



Once toggled on a Configuration Manager tab should be displayed as per the image below.



New configurations can then be created based upon all available options within the CATIA V5 to 3D PDF translator. To create a new configuration, select the Add a New Configuration icon In the configuration manager tab. Select the necessary options and rename as required. (See Configuration Manager section for more information on the different options available to select).



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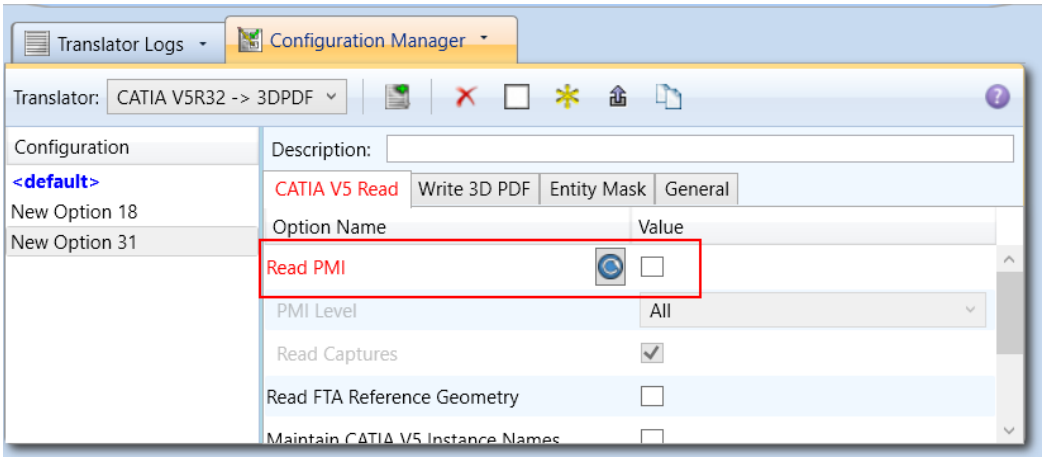

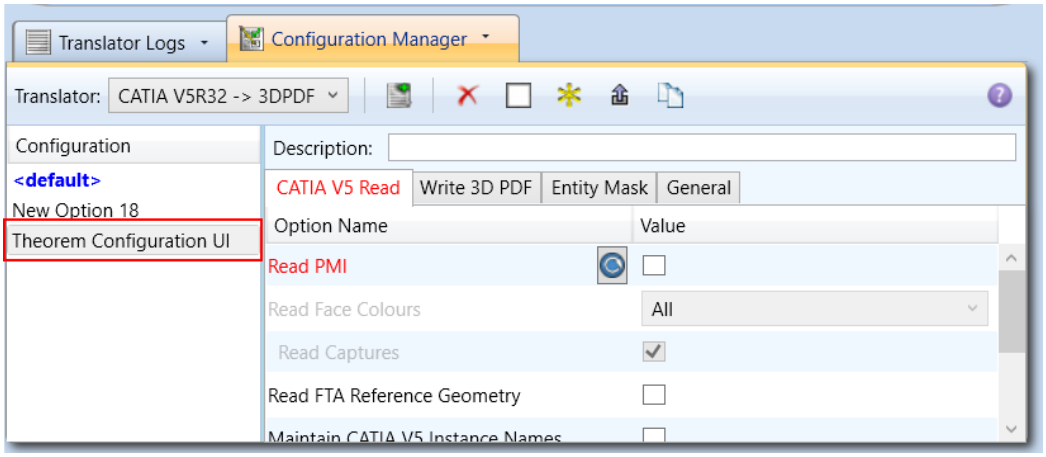

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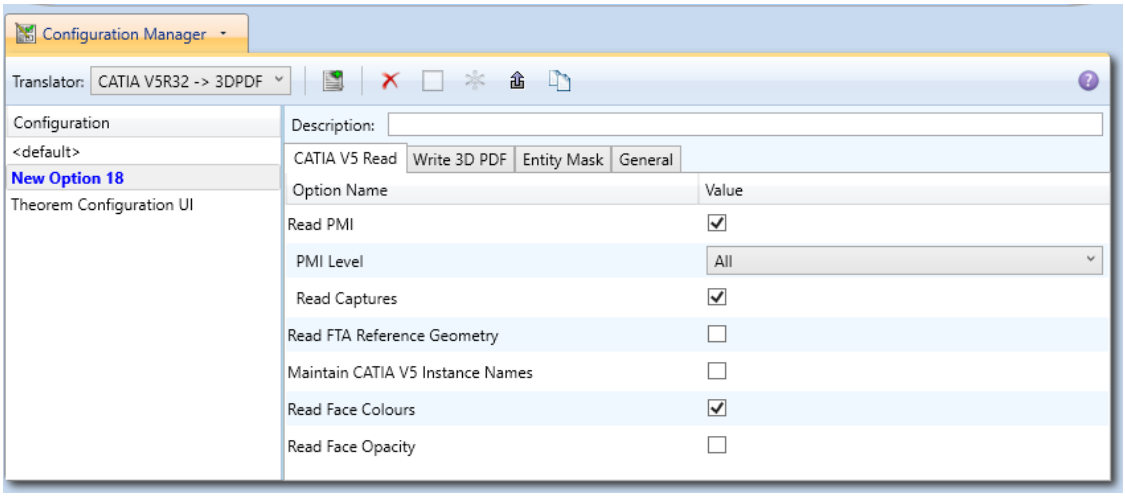
Configuration Manager

Publish 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 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 the Command Line invocation.

Within the Configuration Manager, the options that are available to use are grouped into the following 4 areas when translating data from 3DExperience to 3DPDF.

- CATIA V5 Read – Those options that affect how data is read from CATIA V5
- Write 3D PDF – Those options that affect how the data is written to 3D PDF
- Entity Masking – Those arguments that allow specific read entities to be masked
- General – Those options that are common to ALL Publishing activities regardless of source data. Advanced arguments are added in this tab.

CATIA V5 Read



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

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Option	Description
Read PMI	Enables PMI data read. *(Default is on).* Command Line Syntax: • *read_pmi – default* • *dont_read_pmi – 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'***
PMI Level	A secondary argument to 'Read PMI' and allows control of the level of PMI to be read. Default is ALL when 'Read PMI' is marked as ON. Options Available (command line syntax in italics and square brackets next to the option) • All - [*read_pmi*] • Part Level - [*read_part_pmi*] • Assembly Level - [*read_assy_pmi*] • Assembly Set (From CATPart) - [*read_part_assy_pmi*] • Assembly Set (All) - [*read_all_assy_pmi*]
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* • *don't_read_captures – to turn off*
Read FTA Reference Geometry	Enables 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*

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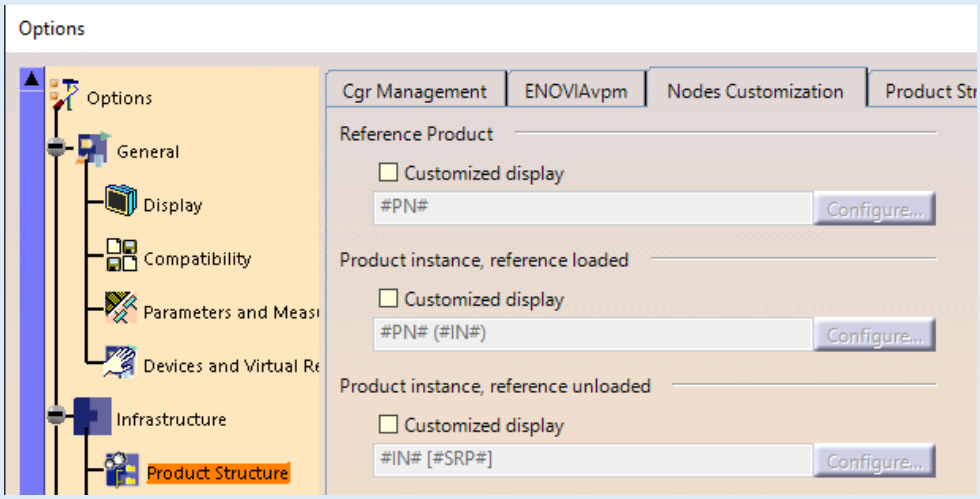
Maintain CATIA V5 Instance Names	Honours CATIA V5 *Tools->Options->Infrastructure->Product Structure>Nodes Customization* panel settings. *(Default is Off)*  Command Line Syntax: • *ditto_naming V5 – to turn on*
Read Face Colours	Read the face colours. *(Default is off)* Command Line Syntax: • *read_face_colours – to turn on*
Read Face Opacity	Read the opacity of the face. *(Default is off)* Command Line Syntax: • *read_materials – to turn on*

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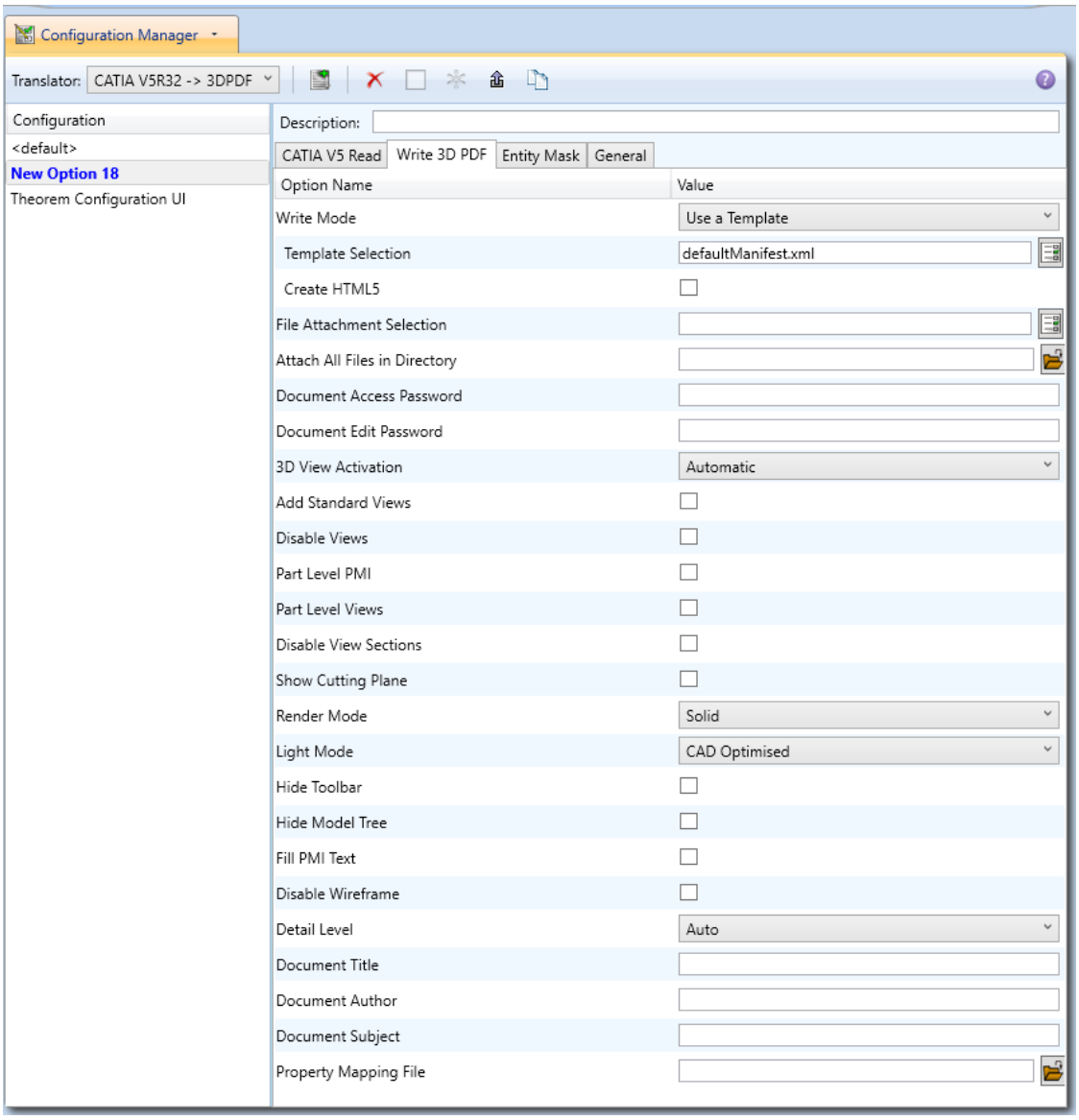
Write 3D PDF



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

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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 list of full path names to file 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>*

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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*

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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 visualization. *(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

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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*
Fill PMI Text	Create filled fonts for PMI. *(Default is off)* Command Line Syntax: • *pmi_filled_text on*
Disable Wireframe	Disable Wireframe Processing. *(Default is to Enable Wireframe processing)* Command Line Syntax: • *process_wf off*

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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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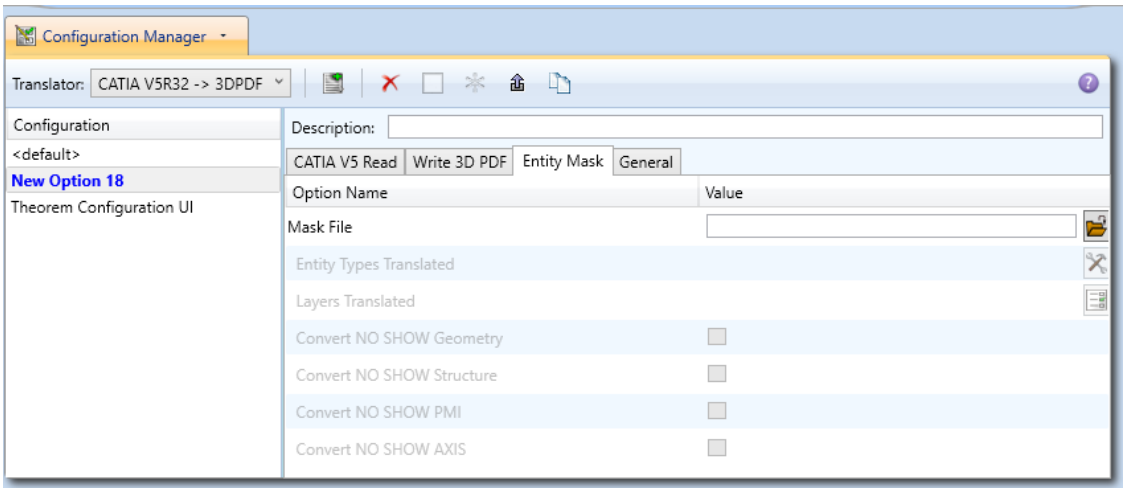
Entity Mask



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

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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 the entity types that are to be processed. The following types are available: "SOL" – Masks any 3D entity "SKIN" – Masks any 2D entity “CUR” – Masks and 1D entity "POI" – Masks any 0D entity "AXIS" – Masks Axis Systems "ISOL" – Masks Isolated facetted solids "CCRV" – If on creates a CCRV curve for wire frame edges that have more than one supporting curve "TEXT" – Masks PMI Text 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***
Layers Translated	Specifies a selection list from which to select the layers that 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 the layers 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***
Convert NO SHOW Geometry	Enables Hidden geometry to be processed. *(Default is Off)* Command Line Syntax: • ***ON NOSHOW GEO –*** *Add to the Mask file*
Convert NO SHOW Structure	Enables Hidden Assembly Structure to be processed. *(Default is Off)* Command Line Syntax: • ***ON NOSHOW STR –*** *Add to the Mask file*

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Convert NO SHOW PMI	Enables Hidden PMI to be processed. *(Default is Off)* Command Line Syntax: • ***ON NOSHOW PMI –*** *Add to the Mask file*
Convert NO SHOW AXIS	Enables Hidden Axis Systems to be processed. *(Default is Off)* Command Line Syntax: • ***ON NOSHOW AXI –*** *Add to the Mask file*

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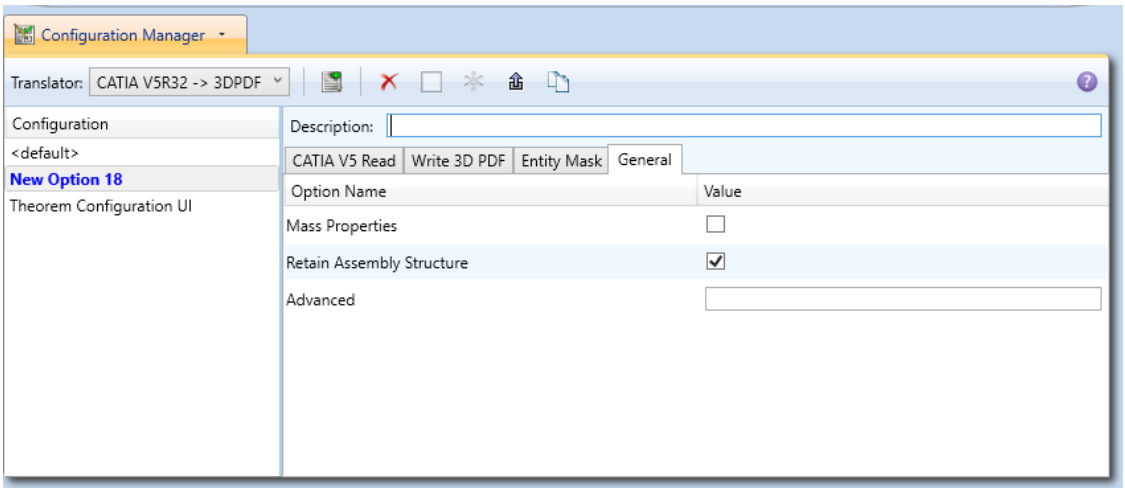
General



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 translation

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Advanced Arguments

CATIA V5 Advanced Arguments

Argument	Description
axis_systems on / value	The **axis_systems on** option controls the reading of CATIA V5 axis system definitions. The axis system is recreated using 3 wireframe lines. The lines will be coloured RED, GREEN & BLUE to represent the X,Y &Z direction and will extend from the X0Y0Z0 of the axis system. The default length of the lines will be 1" or 25.4 mm depending upon the units of the file. The length of the output graphical representation can be controlled by using a value for the axis_systems option e.g. **axis_systems 100** would create the lines at 100 units in length of the units of the input file.
convert_curves	Converts curves to NURBS form
convert_surfaces	Converts surfaces to NURBS form
dont_fill_pmi_arrows	Disables the read of filled arrow information. *(Default for 'read_pmi')*
dont_fill_pmi_text	Disables the read of filled text information. *(Default for 'read_pmi')*
noshow ***noshow_geom*** ***noshow_struct*** ***noshow_pmi***	Reads hidden geometry / structure / PMI
output_mbd	Allows sub-part specification tree information to be read and presented to 3D PDF as product structure information. This option also enables the *'part_level_views'* and *'part_level_pmi'* 3D PDF options
read_geometry_edges	CATIA V5 has a display mode that allows the display of shaded surfaces and edges. This option allows the translator to mimic this for FTA construction geometry by promoting the edge curves to standalone wireframe.

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3DPDF Advanced Arguments

Argument	Description
use_part_names <on/off>	When 'on' the 3DGeom node names are replaced by the TAG name from the detail (part name). *Default is off.*
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 override 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, clearly this is a single override colour for all PMI.

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wireframe_RGB <r g b>	Where r g b are values 1 to 255, this controls the override PMI 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, clearly this is a single override colour for all PMI.
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 process, 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.

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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.
hide_empty_node	When assembly nodes contain no geometry, (could be due to missing part files) this option results in the model tree with 'un-ticked' nodes where no geometry was read. Where the **3DGeom** node would be, there will be a **NoGeom** node instead. **Please node in order to activate this behaviour it was necessary to add a point at 0,0,0 which is contained by the NoGeom node.**
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

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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 <val>	When section cutting planes are enabled, the cutting plane opacity can be specified by setting 'val' 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.**

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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.
invisible <on or off>	Wireframe entities that are marked as invisible are by default **NOT** processed. This option allows them to be processed.
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.

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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...

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	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.

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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>/<size in mm>	Enables the output of axis systems to the PDF. Default is 25mm
ts_cfile <text file>	This option allows command line arguments to be placed in a text file, with one command or option line per line, for example: read_pmi dont_fill_pmi_text disable_points pmi_RGB 255 1 1 wireframe_RGB 255 1 1 **Please note!** that the progress_file <file> option cannot be added into the text file because the progress file is opened before the command file is read.

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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**

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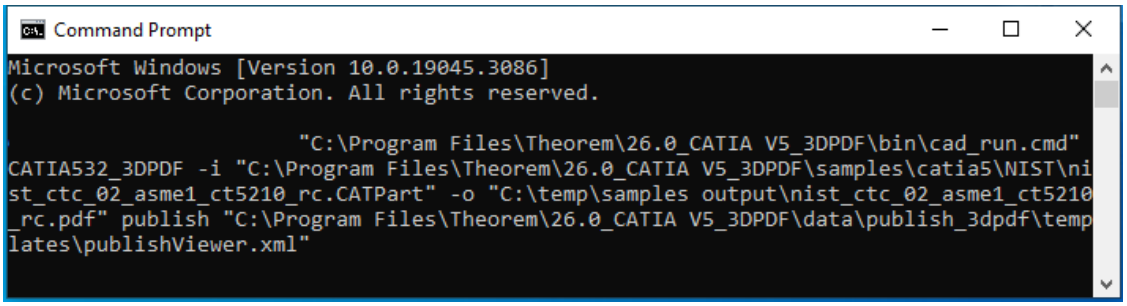
Templates

Publishing to specified Templates – via the Command Line

Publishing to an individual template can be achieved via the command line interface. In order to publish to an individual template a manifest XML file will need to be added to the command in the command prompt window.

The format of the command is as follows (Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for V5 R32, change to CATIA532):

<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_3DPDF -i <input_file> -o <output_file> publish <manifest XML file>



The example above will translate the file using the specified template to the given output location, In this case:

C:\temp\samples output\nist_ctc_02_asme1_ct5210_rc.pdf

Publishing to multiple templates can also be achieved via the command line interface.

First a text file containing a list of manifest files must be created. (Note! For further information regarding our template design services please contact sales@theorem.com quoting “3D PDF Template Design”.)

The manifest files should be listed in the following format in the text file:

<Translator_installation_directory>\data\publish_3dpdf\templates\publishViewer.xml
<Translator_installation_directory>\data\publish_3dpdf\templates\publishRFQ.xml
<Translator_installation_directory>\data\publish_3dpdf\templates\publishDTT.xml

Note – Each line points to an individual XML manifest file, examples of which are provided in the translator installation directory.

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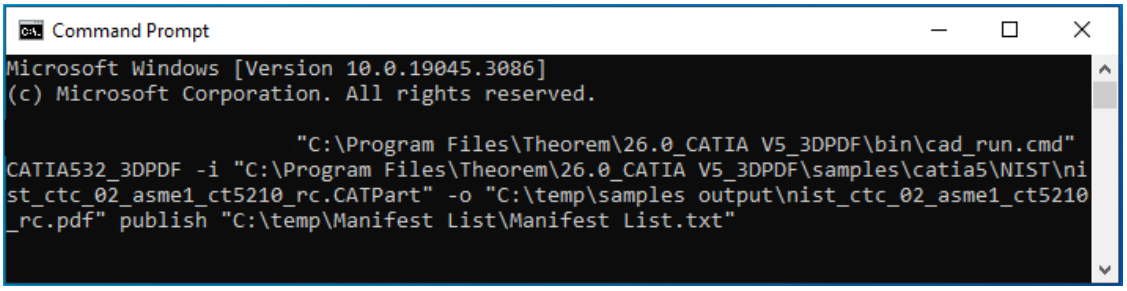
Once the text file has been created the format for running a command to publish to multiple templates is as follows (Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for V5 R32, change to CATIA532):

<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_3DPDF -i <input_file> -o <output_file> publish <manifest list>



The example above will translate the file to each template specified in the referenced manifest list to the given output location, in this case:

C:\temp\samples output\nist_ctc_02_asme1_ct5210_rc.pdf

Note – These PDF files will be created within a single PDF and will be accessible from within attachments.

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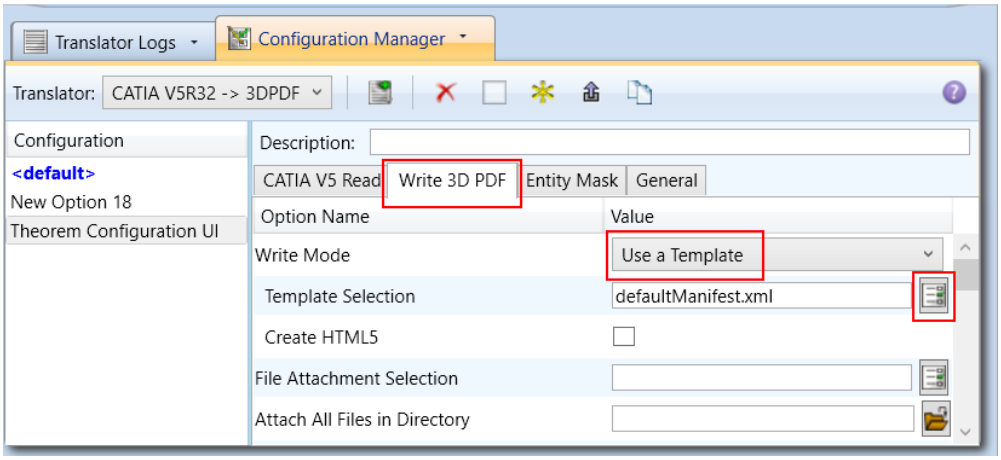
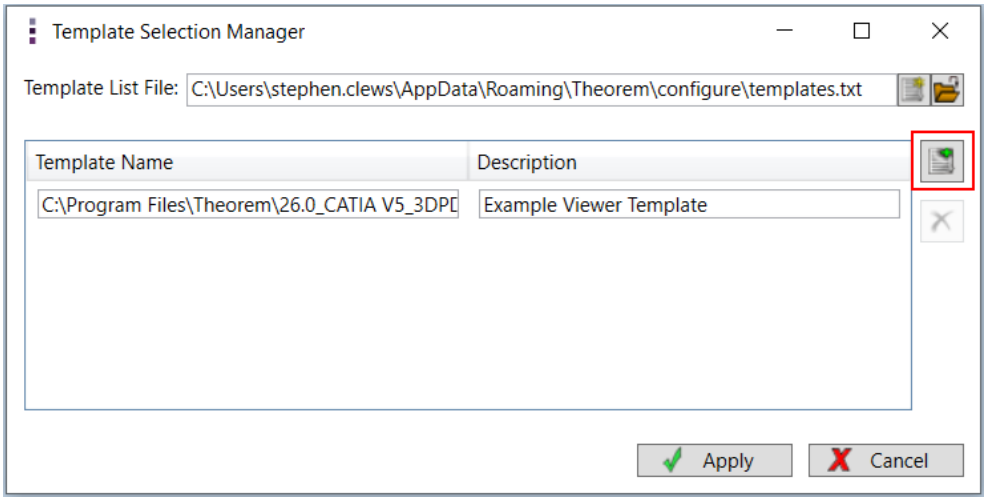
Publishing to specified Templates – via the Unified Interface

It is possible to publish to an individual template or multiple templates via one translation run using the Configuration Manager. To do this create a new configuration as described earlier or use an existing configuration if one has been created previously. Having a user generated configuration will enable the user to change any of the available options displayed 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 Add File icon. Go to the Required directory and select the template XML file to add it to the list. Click appy to confirm the selection. (Note – This will mean that the data will be translated to this single template)



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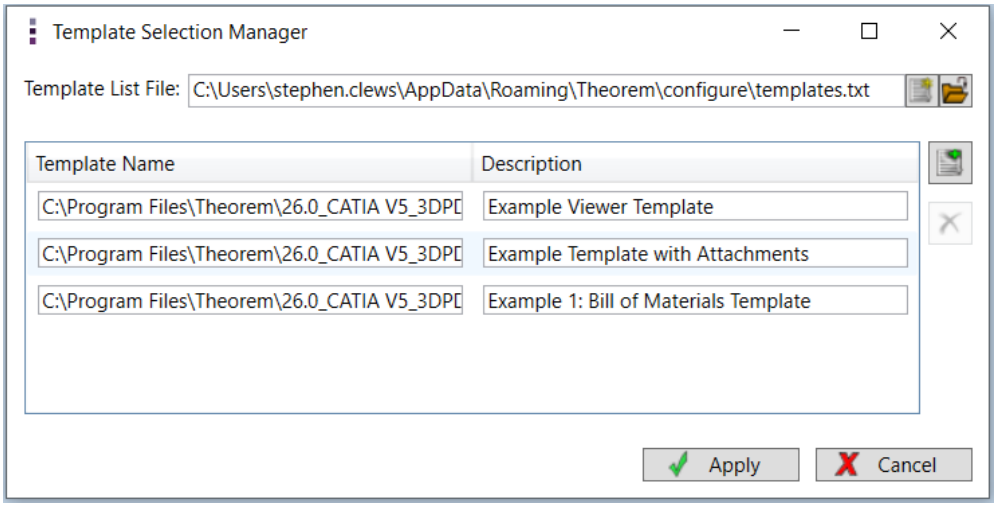
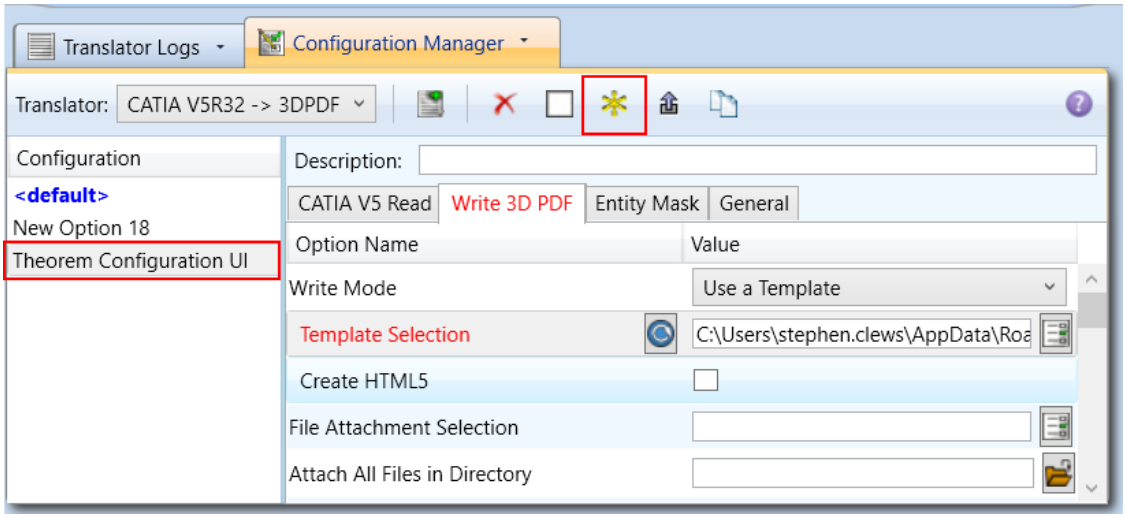
Repeat the process to add more templates if required. Click apply to update the Template Selection. (Note – This will mean that the data will be translated to multiple templates)



Ensure that the correct configuration has been activated in the configuration manager tab. (Note – in order for the data to be translated to the template(s) specified, the configuration will need to be made active). When activated the configuration name will be displayed with blue lettering. If it isn't active, then select the configuration and click on the ***'Make the selected configuration the active configuration'*** icon.



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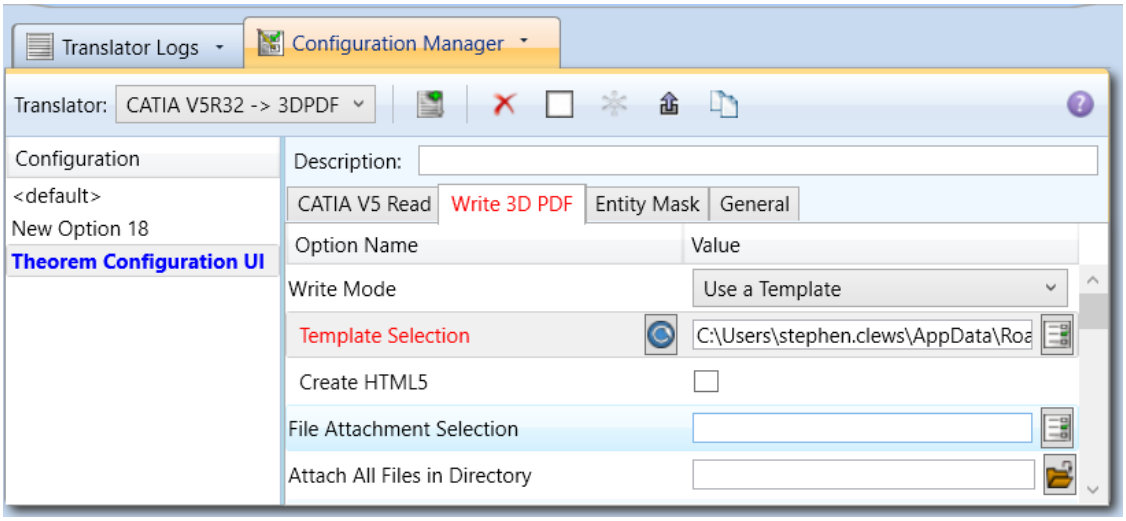
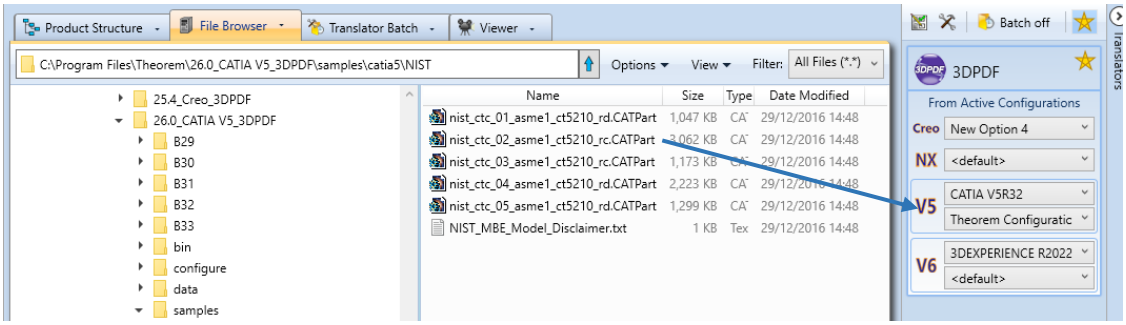


Drag and drop the file from the file Browser tab onto the 3D PDF Active Configuration. This will create PDF files based upon the template(s) specified.



On completion, the Unified Interface will display the activity information and details from the log file created during the translation, in the Translation Activity and Translator Logs tabs respectively.

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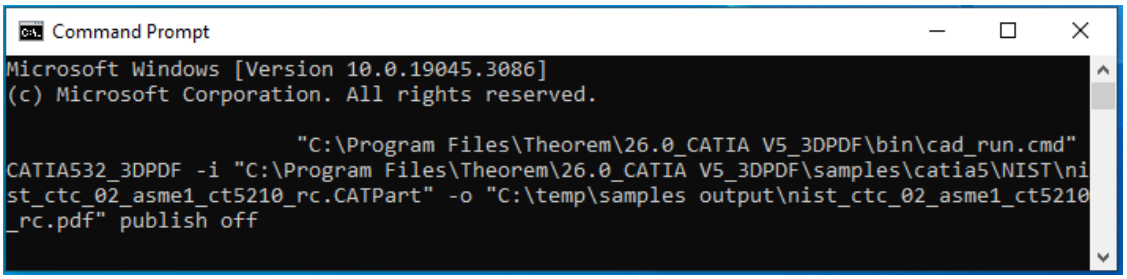
Publishing without a Template – via the Command Line

Publishing without a template can also be achieved via the command line interface.

The format for running a command to publish without a template is as follows (Note! Replace the [XX] seen in the example with the version of CATIA V5 you are using. E.g. for V5 R32, change to CATIA532):

<Translator_installation_directory>\bin\cad_run.cmd CATIA5[XX]_3DPDF -i <input_file> -o <output_file> publish off



The example above will translate the file without using a template to the given output location, In this case:

C:\temp\samples output\nist_ctc_02_asme1_ct5210_rc.pdf

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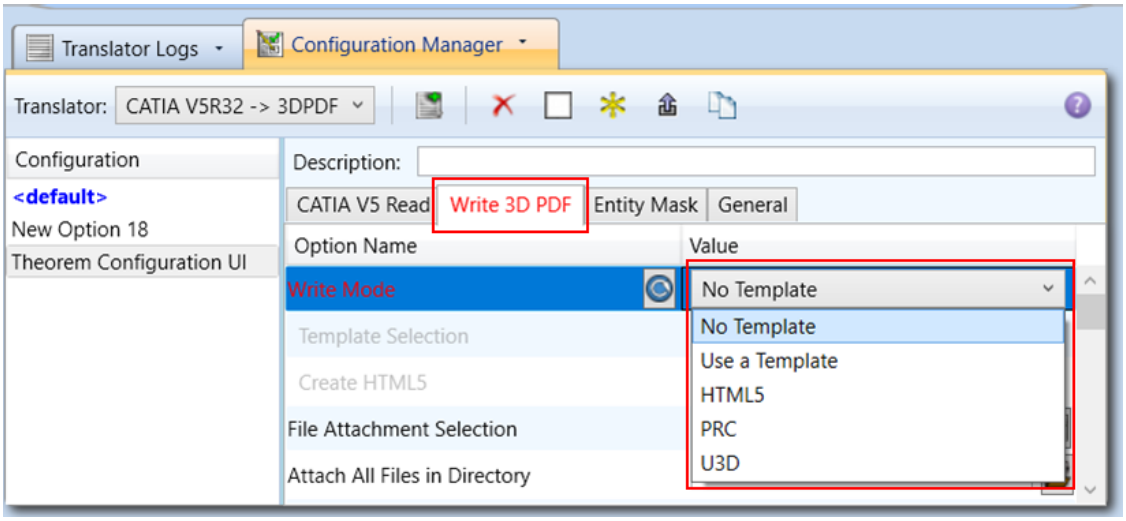
Publishing without a Template – via the Unified Interface

Running the translation without a template will create a 3DPDF output file with no containing document. To do this, click on the configuration manager then select the ***'Write 3D PDF'*** tab. From the ***'Write Mode'*** dropdown, select ***'No Template'***.



Ensure that the configuration has been activated in the configuration manager tab, then drag and drop the file from the file Browser tab on to the 3D PDF Active Configuration. This will create a PDF file using no template.

On completion, the Unified Interface will display the activity information and details from the log file created during the translation, in the Translation Activity and Translator Logs tabs respectively.

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Translating Interactively from within CATIA V5

Launching CATIA V5 with Theorem plug-ins

Publish 3D Interactive allows an active CATIA V5 Part or Assembly to be published directly into 3D PDF, directly from the CATIA V5 application.

In order to publish from within CATIA V5, the CATIA V5 application must be started using a Theorem environment, so that the appropriate Theorem partner plug-ins are available.

CATIA V5 can be started from a shortcut, if requested at installation time.

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

<installation_directory>\bin

The script name is:

catia5r[version]_start.cmd

(Where [version] should be substituted for the version of CATIA V5 that you have installed – e.g. 30 for V5-5R2020, 32 for V5-6R2022):

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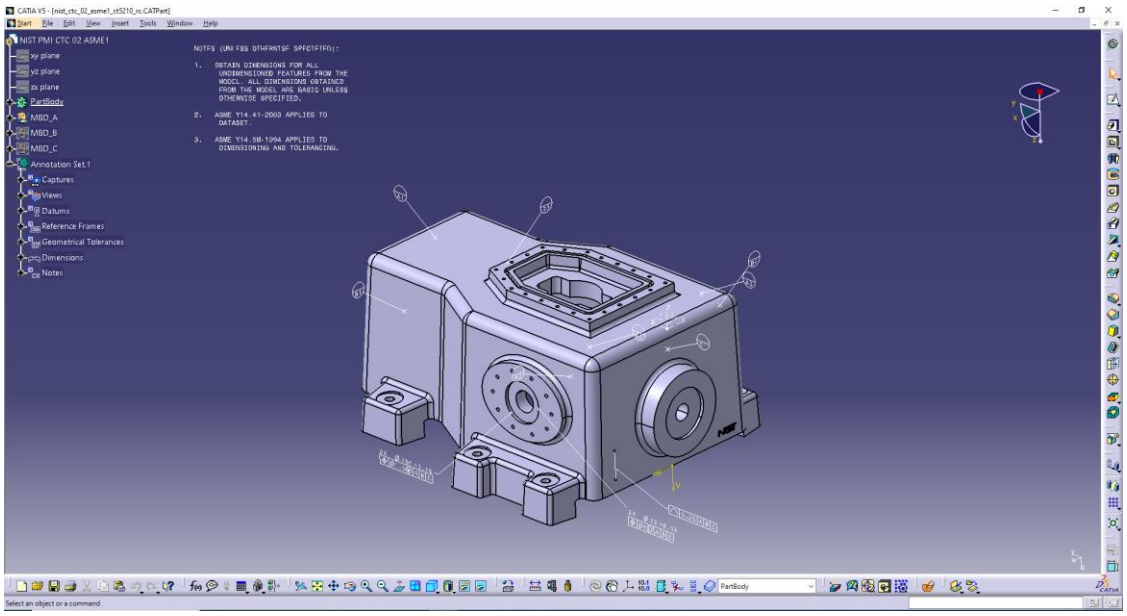
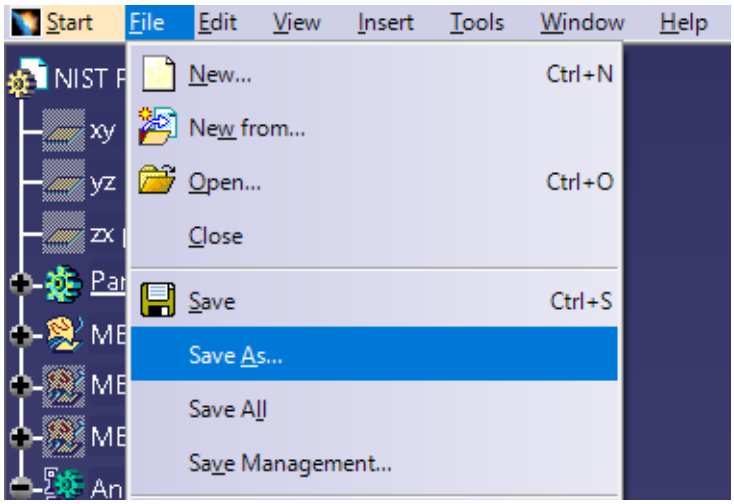
Default Translation from CATIA V5

Once CATIA V5 has been started, open the part or assembly that is going to be exported to 3DPDF.



Select File, then Save As.



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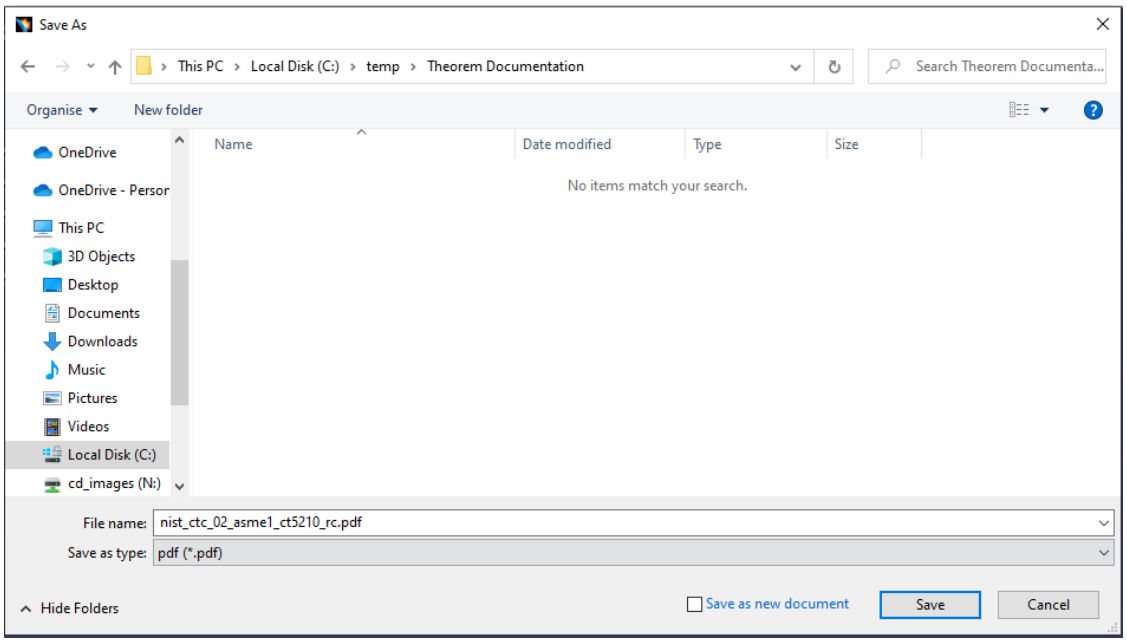
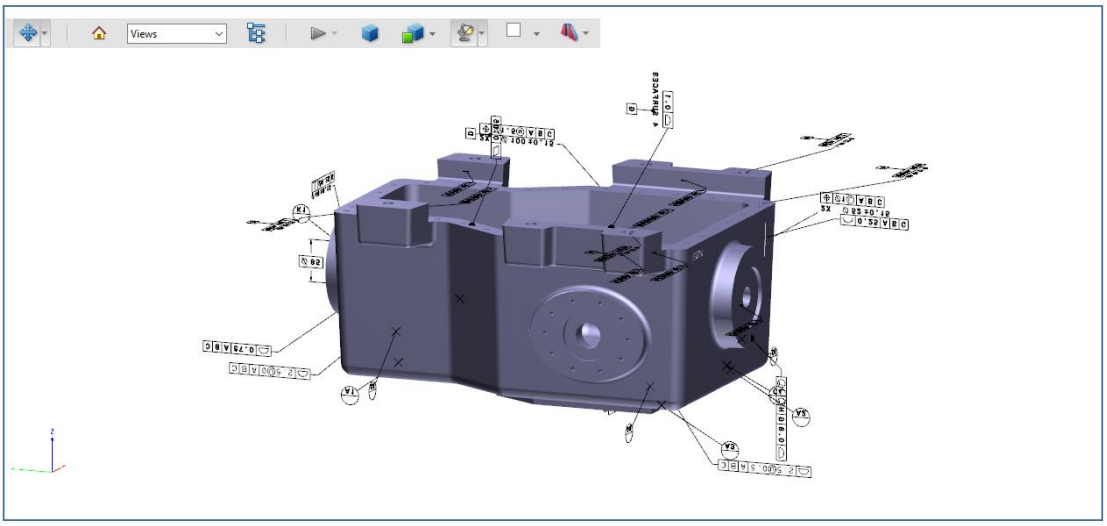
Select the required location for the file and in the Save As Type window select pdf (*.pdf)



Click save to initiate the translation and create the PDF file in the location specified.



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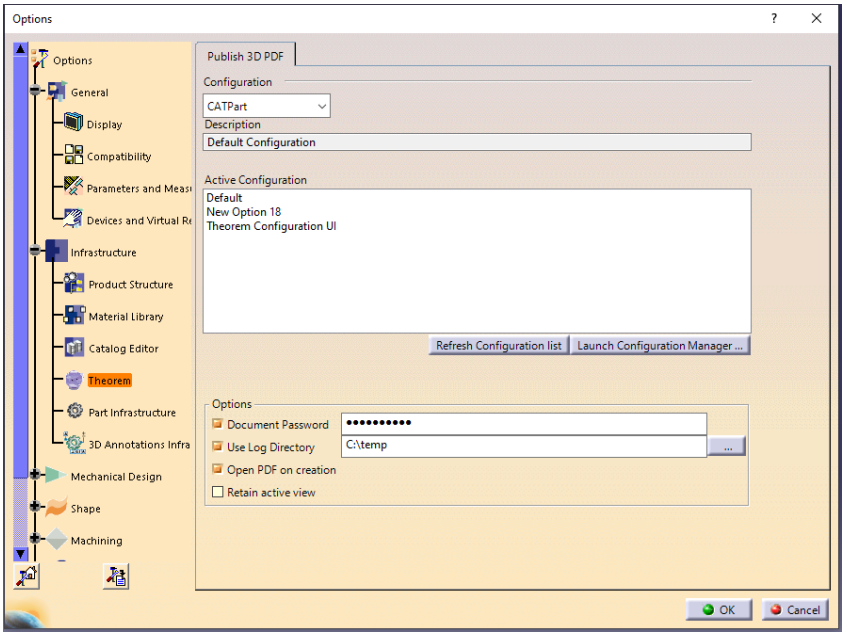
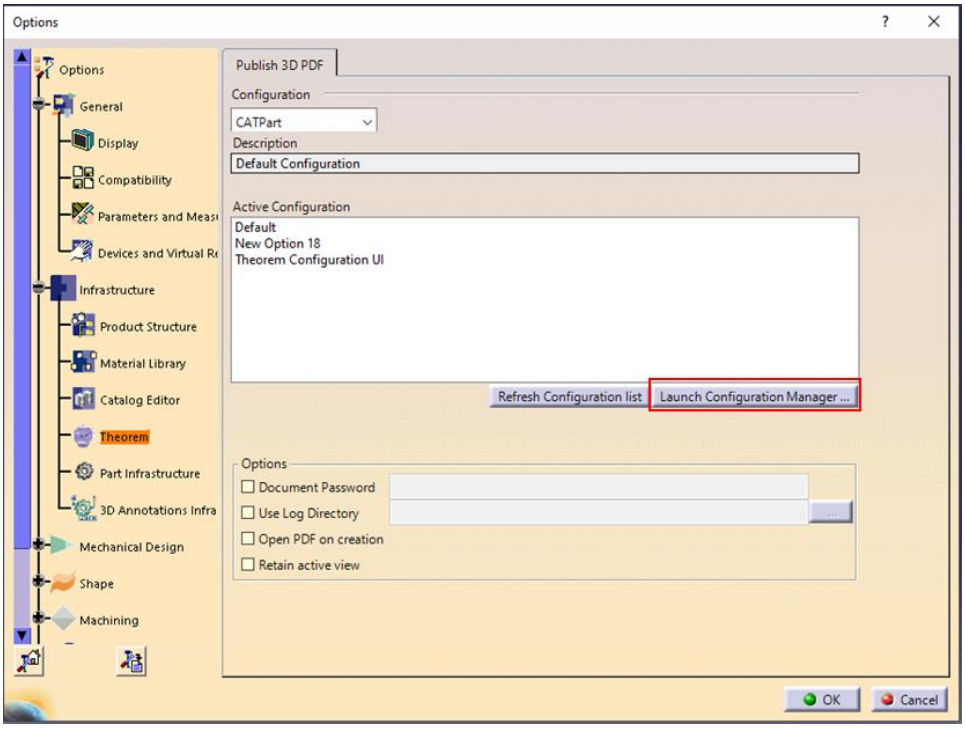
Accessing the Configuration Manager in CATIA V5

On Demand users have the option to create configurations, selecting a predefined configuration, whether a password should be set, specifying the log file directory and whether the PDF document is opened once created. These options are accessible from the Publish 3D tab via Tools->Options->Infrastructure->Theorem.



To create a new configuration, select the **'LaunchConfigurationManager'** command.



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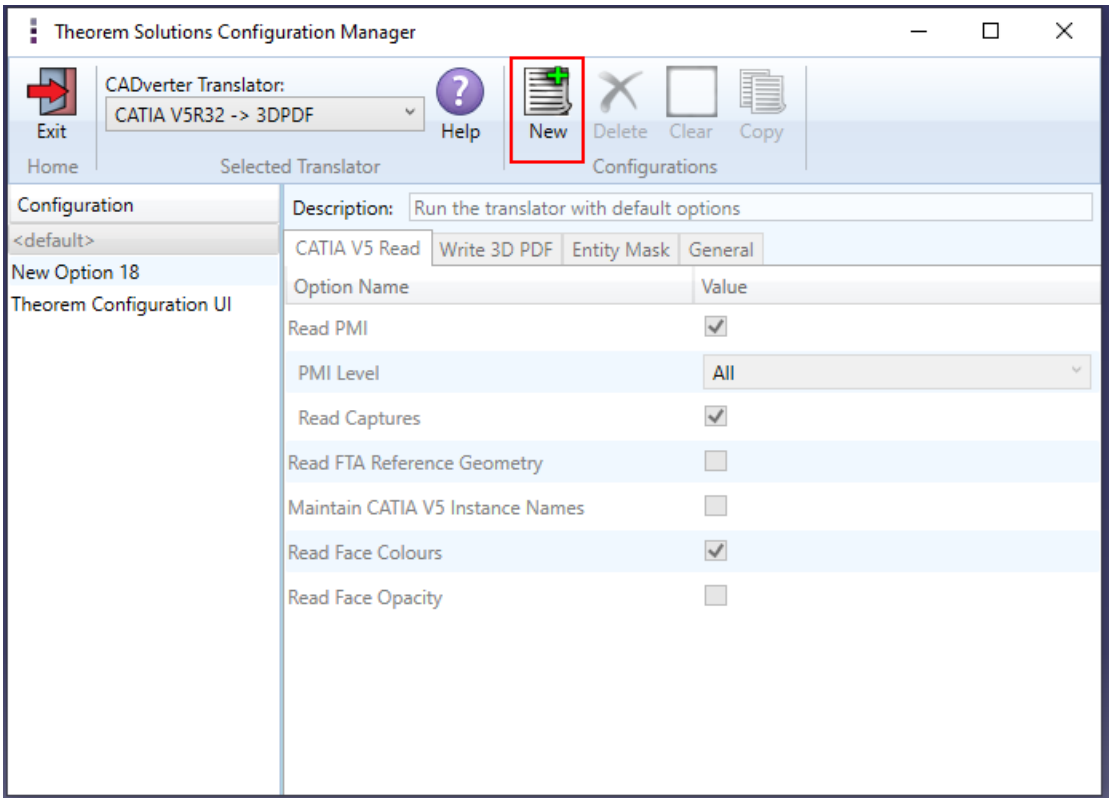
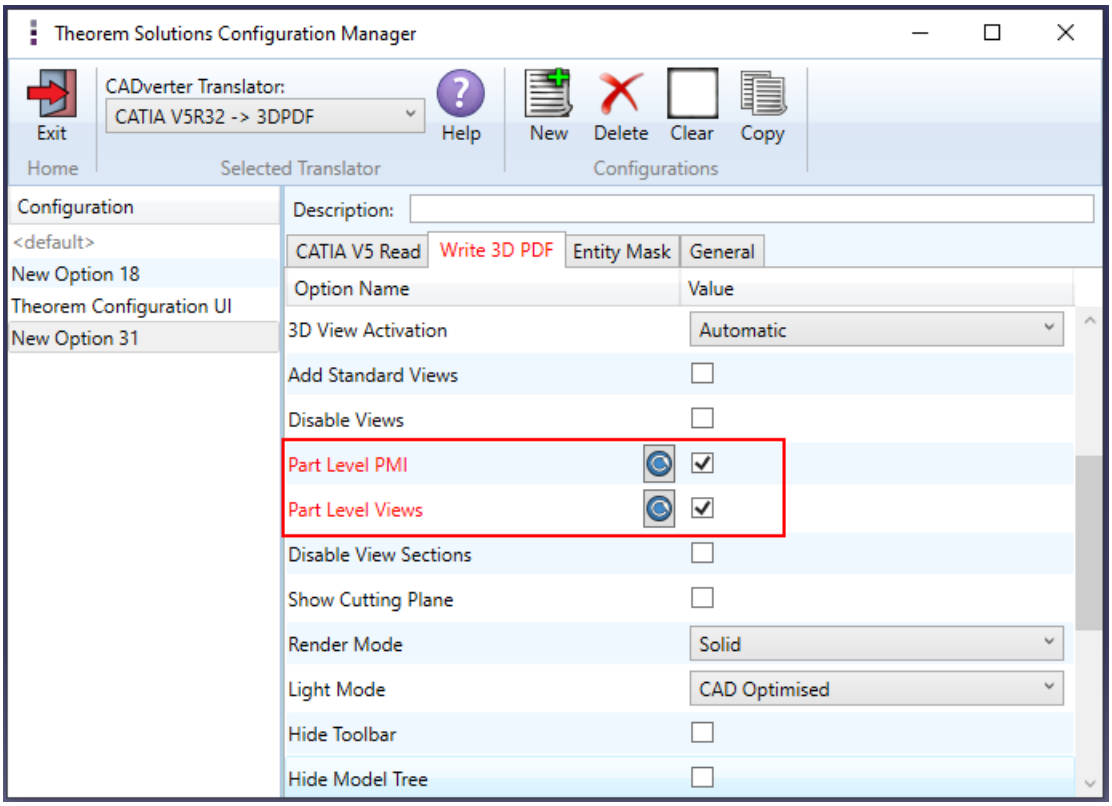
In the configuration manager window, select New to create a new configuration. Select the necessary options and rename as required. (See Configuration manager section for more information on the different options available to select). Click Exit to close the window.





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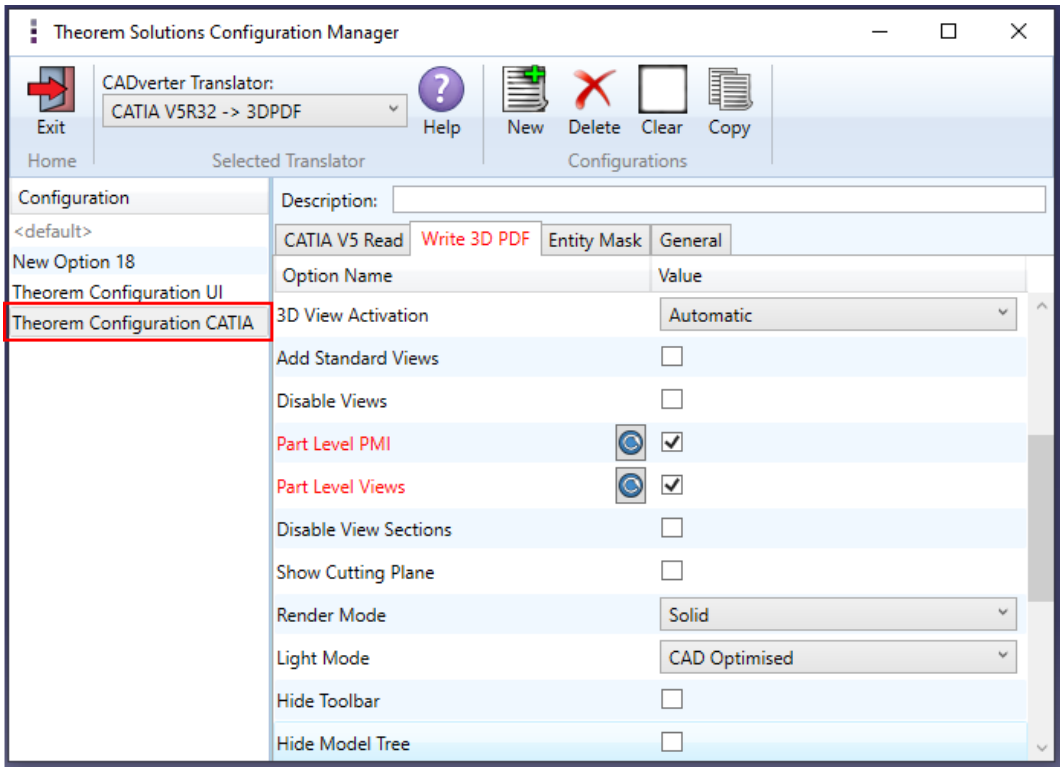
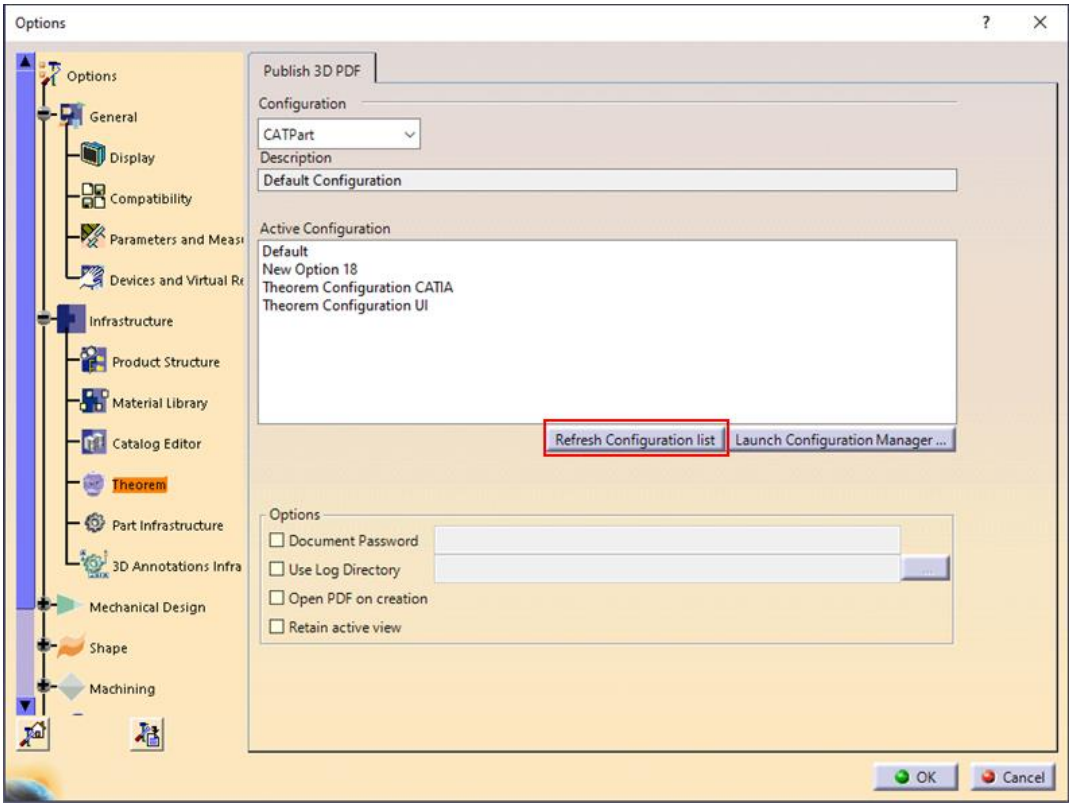


Any new configurations created will be displayed in the Active Configurations list once it has been refreshed. To do this select **'RefreshConfigurationList'.** Any predefined configurations will also be displayed in the Active Configuration list.



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Interactive users do not have access to the configuration manager or have the ability to create configurations with the standalone Interactive license. However the 3D Configuration Tool add on will allow interactive users access to the configuration manager to create configurations. This is licensed separately from the main CATIA V5 to 3DPDF product, if you would like more information on this feature please contact sales@theorem.com

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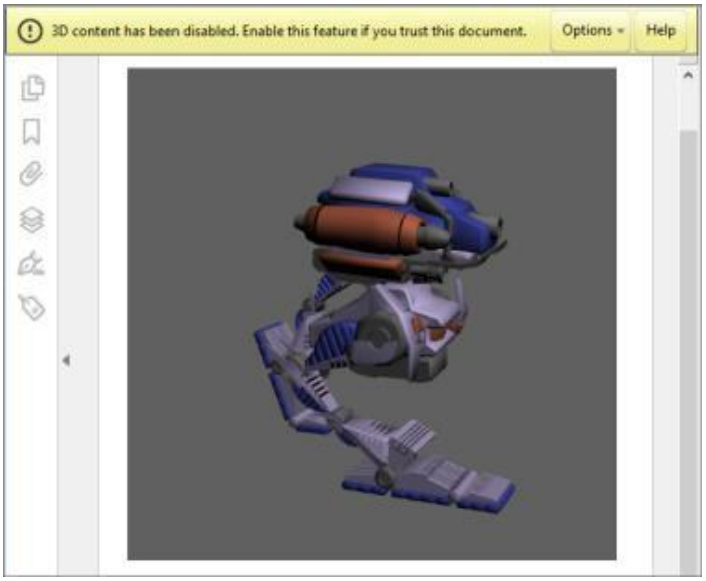
Appendix A – Security Changes to Adobe Reader PDF Documents

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:



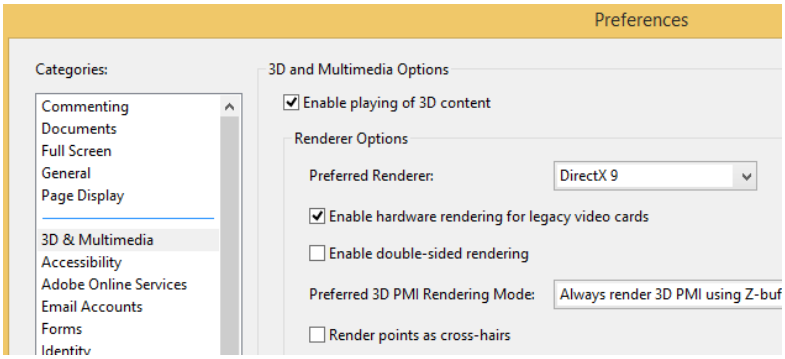
To enable 3D content in the current document:

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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