User Guide

CADPublish JT > PDF

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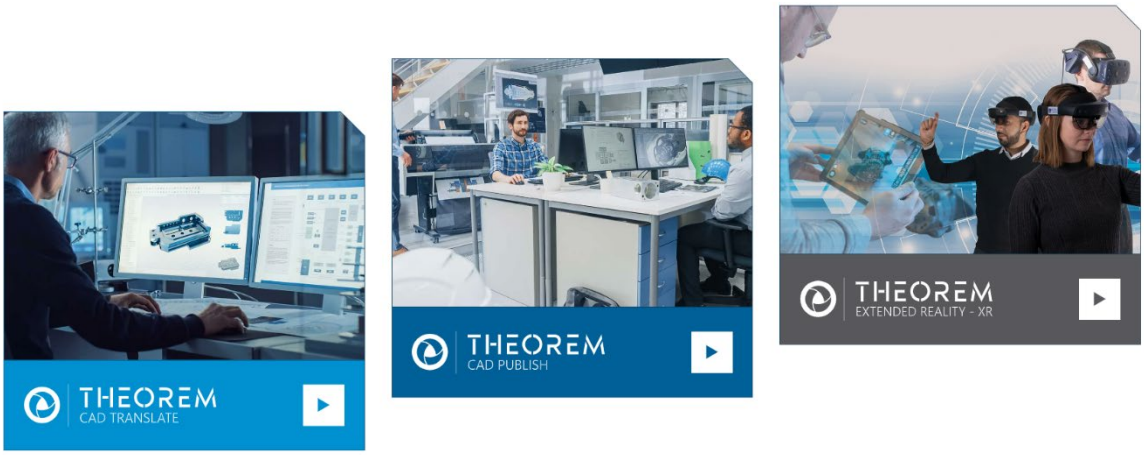
Overview of TRANSLATE

About Theorem



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

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

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

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Theorem's Product Suite

Theorem have 3 main Product brands. These are:

CAD**Translate**

CAD Data Exchange: Seamless and robust data translation between CAD and Visualization formats.

See our website for more detail.

CAD**Publish**

Interactive Documentation: 3D PDF Publisher and Composer. Making CAD data accessible to non-CAD users.

See our website for more detail.

Theorem**XR**

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

Extended Reality – XR: Augmented, Mixed and Virtual Reality for the Engineering Metaverse.

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.
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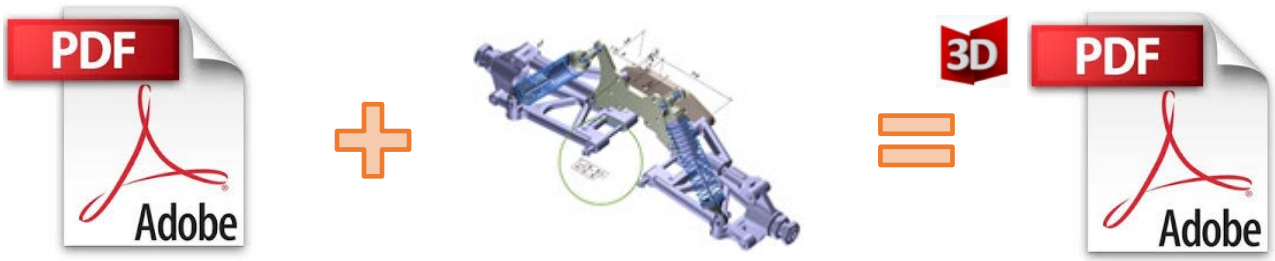
It's already used for documents which commonly relate to products e.g.; Request for Quotation, Engineering Change Requests, Bill of Material reports, Inspection Documents, Works instructions, Service manuals, Sales literature, etc.



Review our website here **for a guide explaining 3D PDF.**

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What Is PUBLISH?

Theorem's **PUBLISH** 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 in the form of 3DPDF.

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 organisations of all sizes, **PUBLISH** 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 in to the world of PDF.

Theorem's PUBLISH 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 please contact sales@theorem.com

What does PUBLISH provide?

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

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

For further information on other Theorem PUBLISH Products please contact sales@theorem.com

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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 and Tutorials.

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

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

Installation

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

License Configuration

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

Using the Product

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

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Using the Product

Default Translations

Publish On-Demand

By default the PUBLISH for JT translator will translate source data using a default template.
This template can be used as a basis for a customer's specific template design and illustrates fundamental capabilities of the translator with regard to publishing 3D data into a pre-defined document format.
More complex template examples are provided in the following the locations:

<installation_directory>\data\publish_3dpdf

The following examples are available:

- default_Manifest – Illustrates how to output an Assembly Bill of Materials
- publishBlank – Illustrates publishing to a blank template

<installation_directory>\data\publish_3dpdf\templates

The following examples are available:

- 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
- publishECR – Illustrates a simple Engineering Change Request
- publishRFQ – Illustrates a simple Request for Quotation
- publishAttachments – Illustrates adding attachments to a template

These templates are supplied as examples only and can be used by customers as a guide to 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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Default Publishing

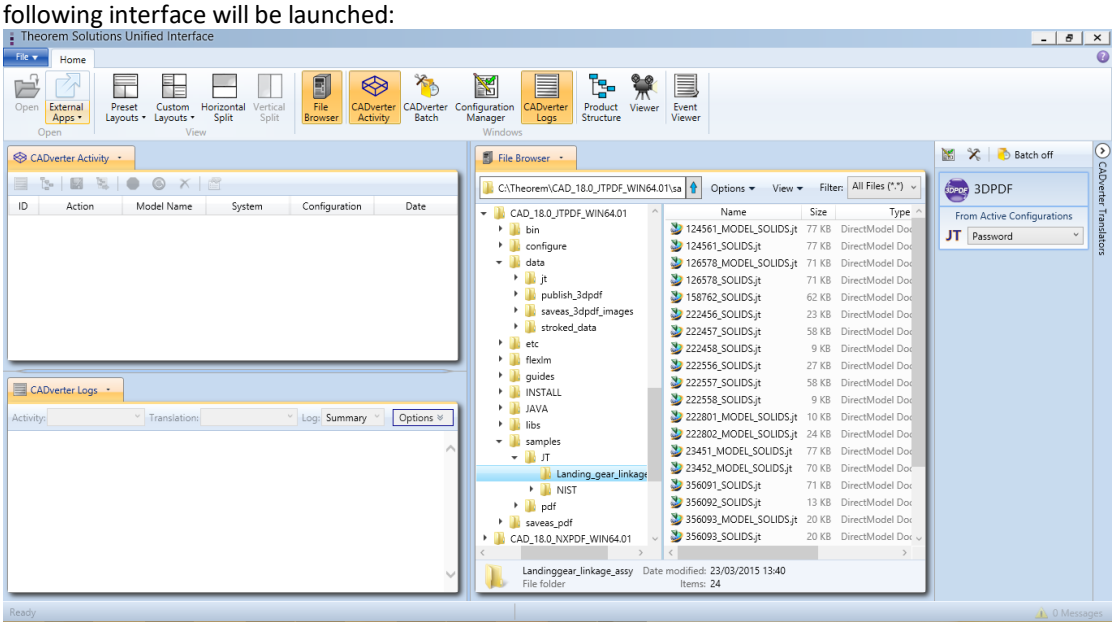
Default Publishing – via the Unified Interface

The Unified Interface can be started via the **Start Menu** under the *Theorem Solutions* folder – if a shortcut was added during installation.

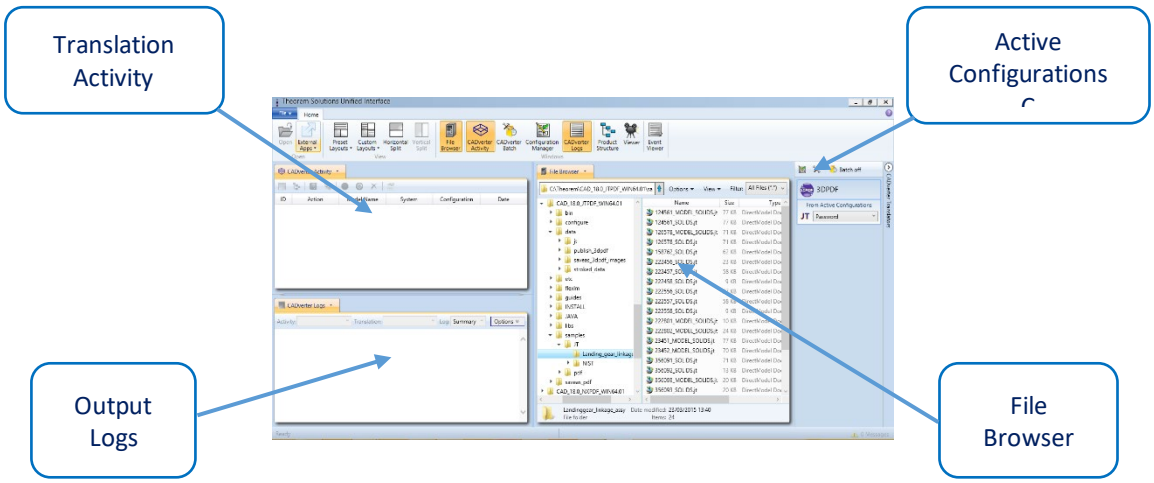
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

The following interface will be launched:



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



The simplest way to publish a JT file in 3D PDF is to drag a file from the file Browser Pane on to the 3D PDF Active Configuration. This will create a PDF file based upon the default template.

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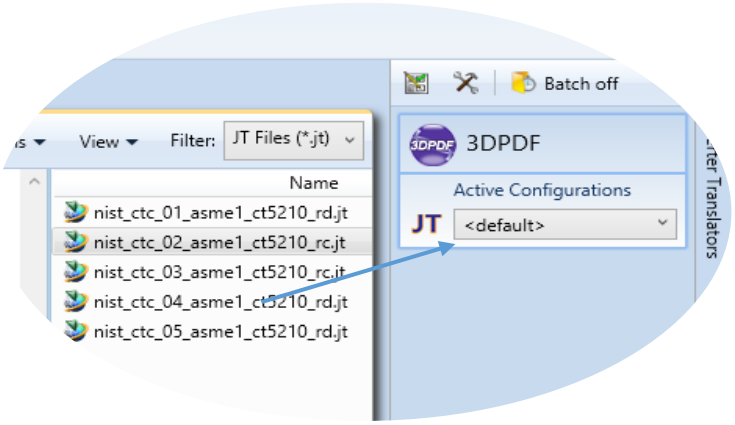
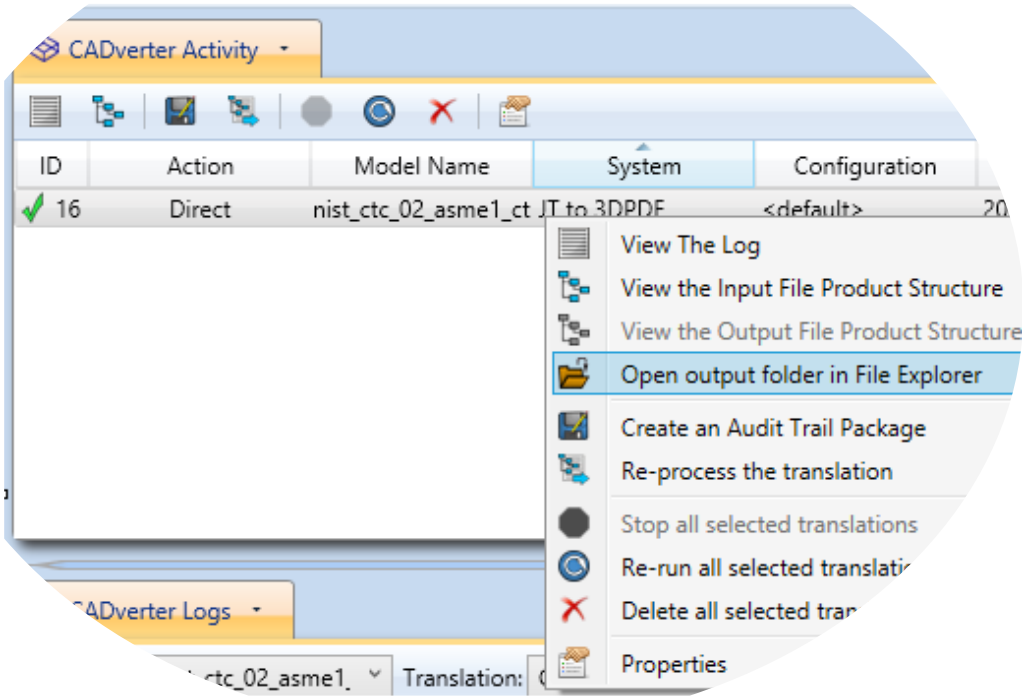
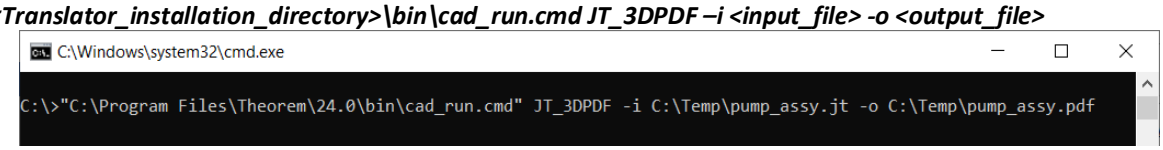


On completion, the Unified Interface will display the activity information and details from the log file created during the translation, if requested, in the Translation Activity Output Log panes, respectively.
The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:



Default Publishing – via the Command Line

Running a translation via the command line can be carried out via the ***cad_run.cmd*** file located in the ***<installation_directory>\bin*** directory. The format of the command is as follows:
<Translator_installation_directory>\bin\cad_run.cmd JT_3DPDF –i <input_file> -o <output_file>



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

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```
Create Artwork ...
**********************************
* 3DPDF file successfully created *
* C:\Temp\pump_assy.pdf           *
**********************************
```

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

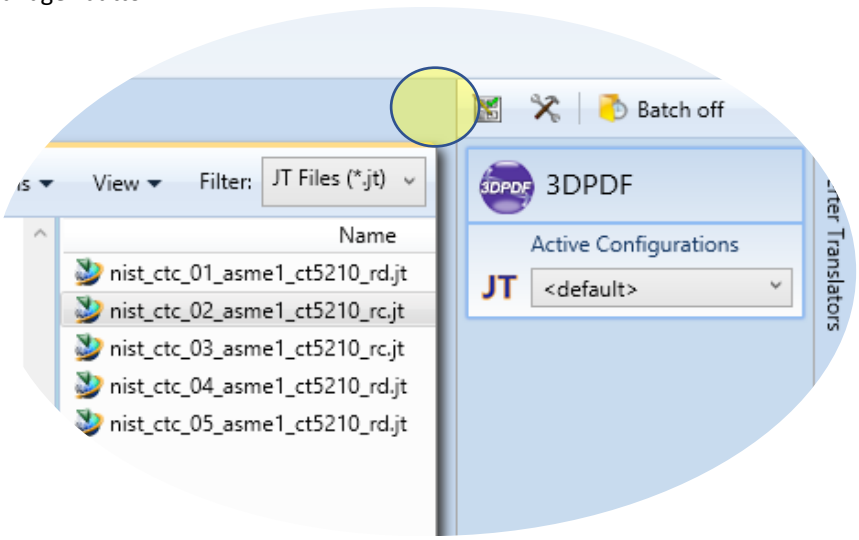
C:\temp\pump_assy.pdf

Publishing to Specific List of Templates

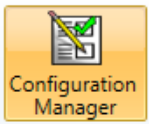
Publishing to Targeted Templates – via the Unified Interface

It is possible to publish to an individual or multiple templates via one translation run using the Unified Interface. In order to do this, a new configuration will be required.

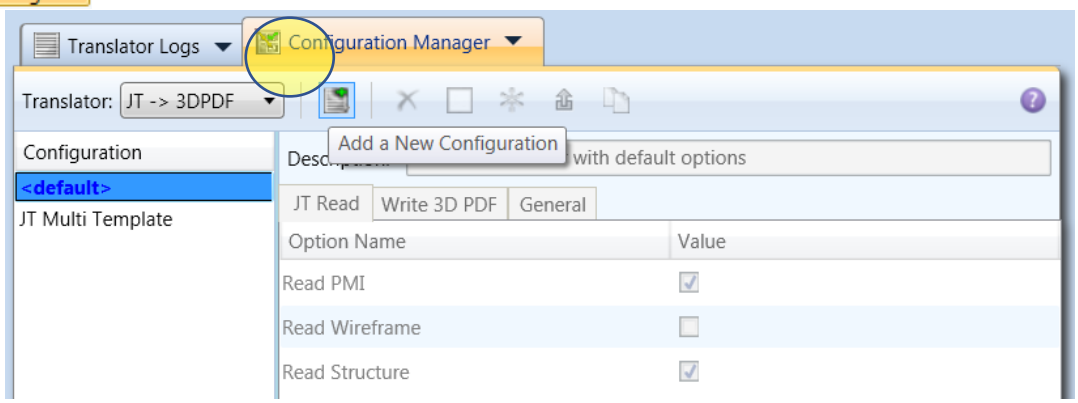
The Configuration Manager pane can be launched via the Active Configuration Pane or from the Ribbon Configuration Manager button:



OR



Select the 'Configuration Manager' option from the ribbon menu at the top of the Unified Interface. This will then display the 'Configuration Manager' pane.



The Configuration Pane allows new configurations to be created based upon ALL available arguments within the PUBLISH application. Selecting the New Configuration icon (highlighted above) will allow the user to change any PUBLISH argument.

The panels below show the required selections to create a Configuration to create output based upon Multiple Templates (Note! The same method is used to target an individual template, by selecting only one template):

Select the icon to the right of the 'Template Selection' option.

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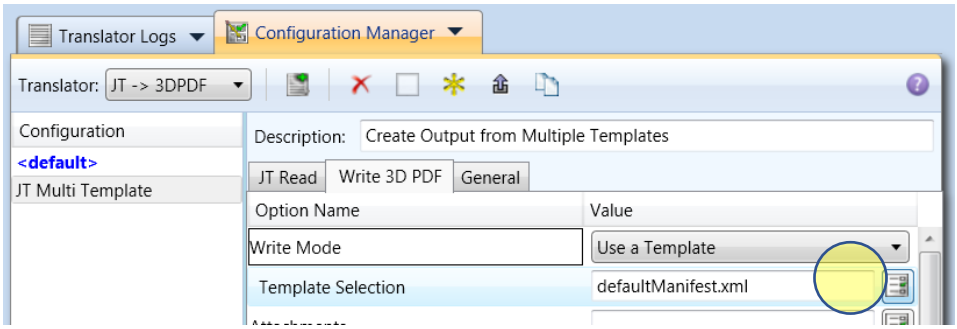
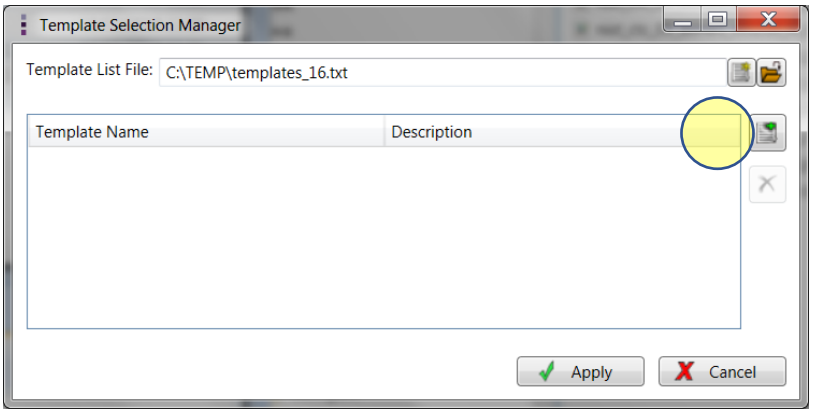
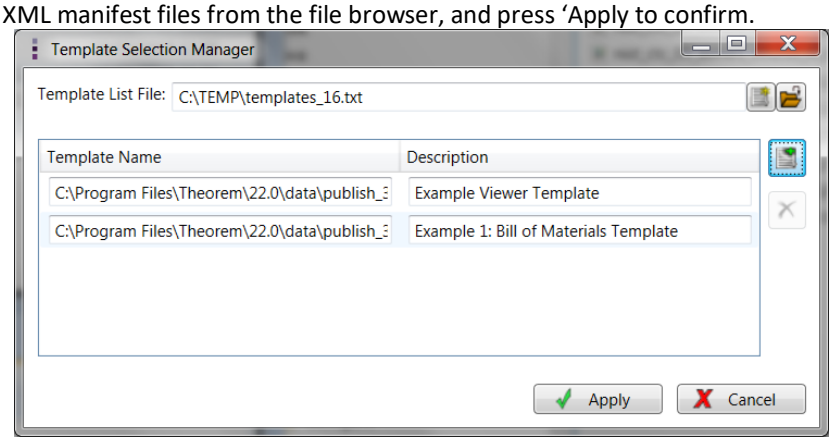


This will open a new selection window. Use the 'Add File' button on the right hand side of the window to select your templates



Now select your XML manifest files from the file browser, and press 'Apply to confirm.



In order to run a translation using this configuration the Active Configuration setting for JT to 3D PDF must be changed to the "Multiple Template Output" Configuration:

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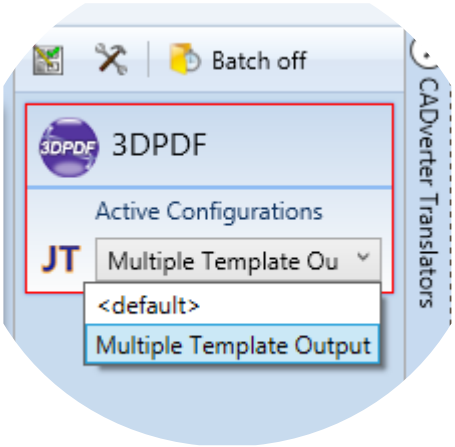


Once this has been done the translation can be invoked in the same manner as for the Default publishing mechanism

Publishing to Targeted Templates – via the Command Line

Publishing to an individual template or multiple templates can also be achieved via the command line interface.

The syntax for running a command to publish to a single template is as follows:

<Translator_installation_directory>\bin\cad_run.cmd JT_3DPDF -i <input_file> -o <output_file> PUBLISH <template_xml_file>

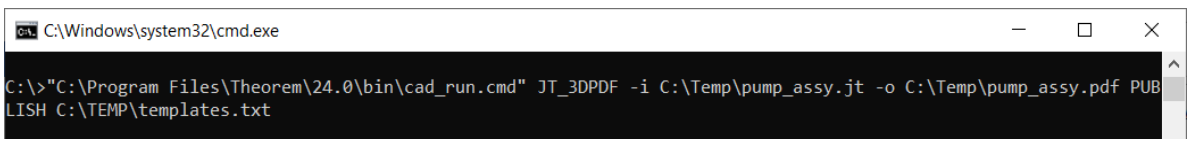
To publish to a list of target Template files a template list file must be created. (Note! For further information regarding our template design services please contact sales@theorem.com quoting “3D PDF Template Design”.)

The list is provided via a text file in the following format:

<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 XML manifest file, examples of which are provided in the PUBLISH installation. Once this file has been created the syntax for running a command to publish to multiple templates is as follows:

<Translator_installation_directory>\bin\cad_run.cmd JT_3DPDF -i <input_file> -o <output_file> PUBLISH <manifest list>



The example above will use the referenced template manifest list and will publish an output to each of the templates referenced via the manifest files provided, whether there be one or many in the list.

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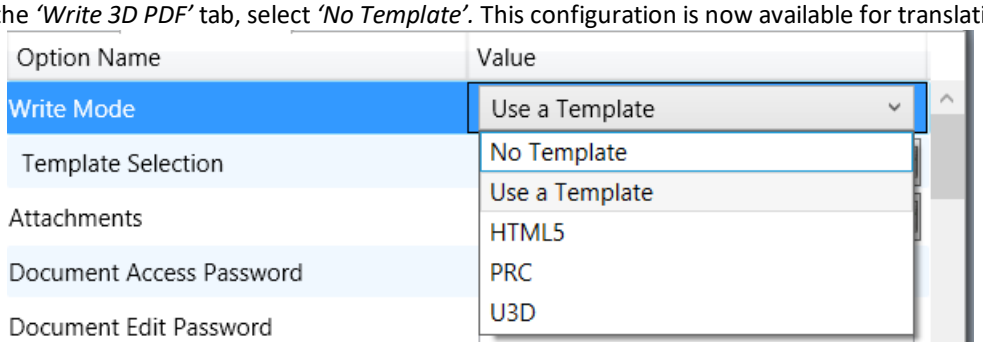
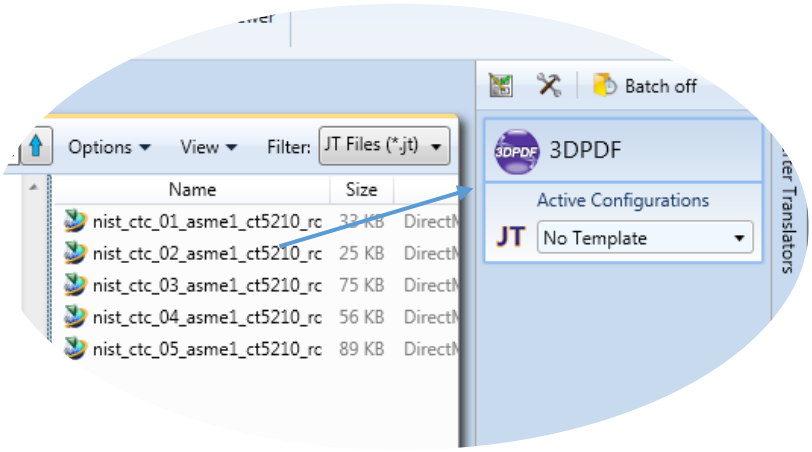
Publishing without a Template

Publishing without a Template – via the Unified Interface

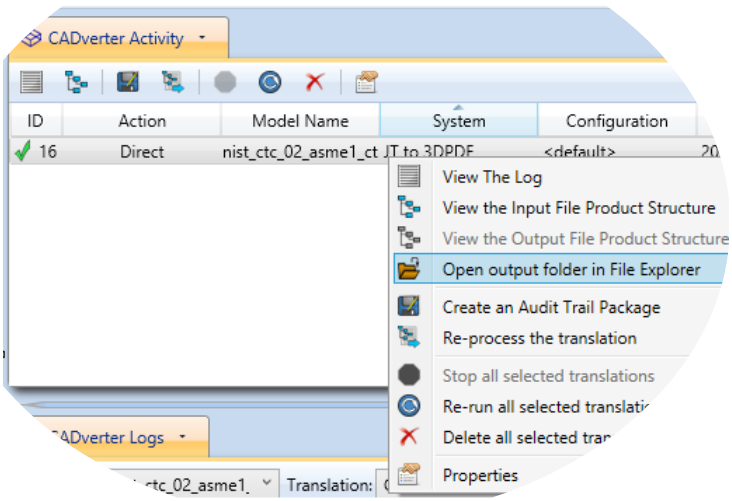
Running without a template will create a 3D PDF output file with no containing document. In order to do this via the *Unified Interface,* navigate to the configuration manager as described, and from the *'Write Mode'* option in the *'Write 3D PDF'* tab, select *'No Template'*. This configuration is now available for translation.



Now, in the 'Active Configurations' pane select your configuration from the menu and then 'drag and drop' your data to publish it to a PDF document.



On completion, the Unified Interface will display the activity information and details from the log file created during the translation, if requested, in the Translation Activity and Output Log panes, respectively.
The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:



Publishing without a Template – via the Command Line

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

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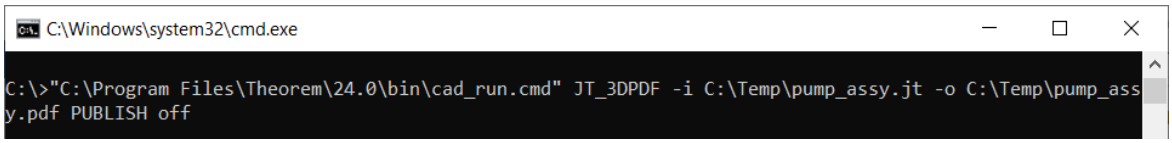
The syntax for running a command to publish without a template is as follows:

<Translator_installation_directory>\bin\cad_run.cmd JT_3DPDF -i <input_file> -o <output_file> PUBLISH off



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Publish Customization

PUBLISH allows the information that is read from the source system and written to the target system to be tailored via a set of user specified arguments. Commonly used arguments are supported via the Unified Interface, with Advanced Arguments being described within this document for use in the Unified Interface or via the Command Line invocation.

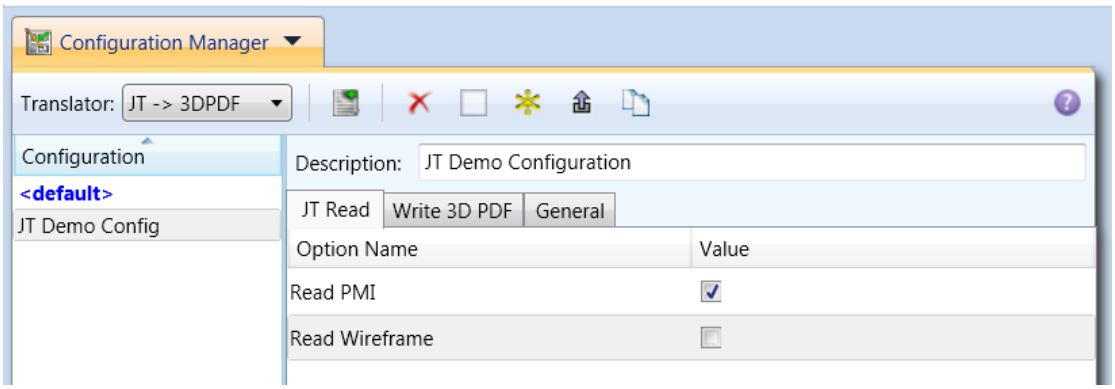
Common Options for JT Publishing

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

- JT Read – Those arguments that affect how data is read from JT
- Write 3D PDF – Those arguments that affect how the data is written to 3D PDF
- General – Those arguments that are common to ALL Publishing activities regardless of source data

JT Read

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



Each of these options is described below:

Option	Description
Read PMI	PMI data read from the JT file. *(Default is On)* ○ Command Line Syntax: ▪ *read_pmi*
Read Wireframe	Read Wireframe data from the JT file. *(Default is Off)* ○ Command Line Syntax: ▪ *read_wire_frame*

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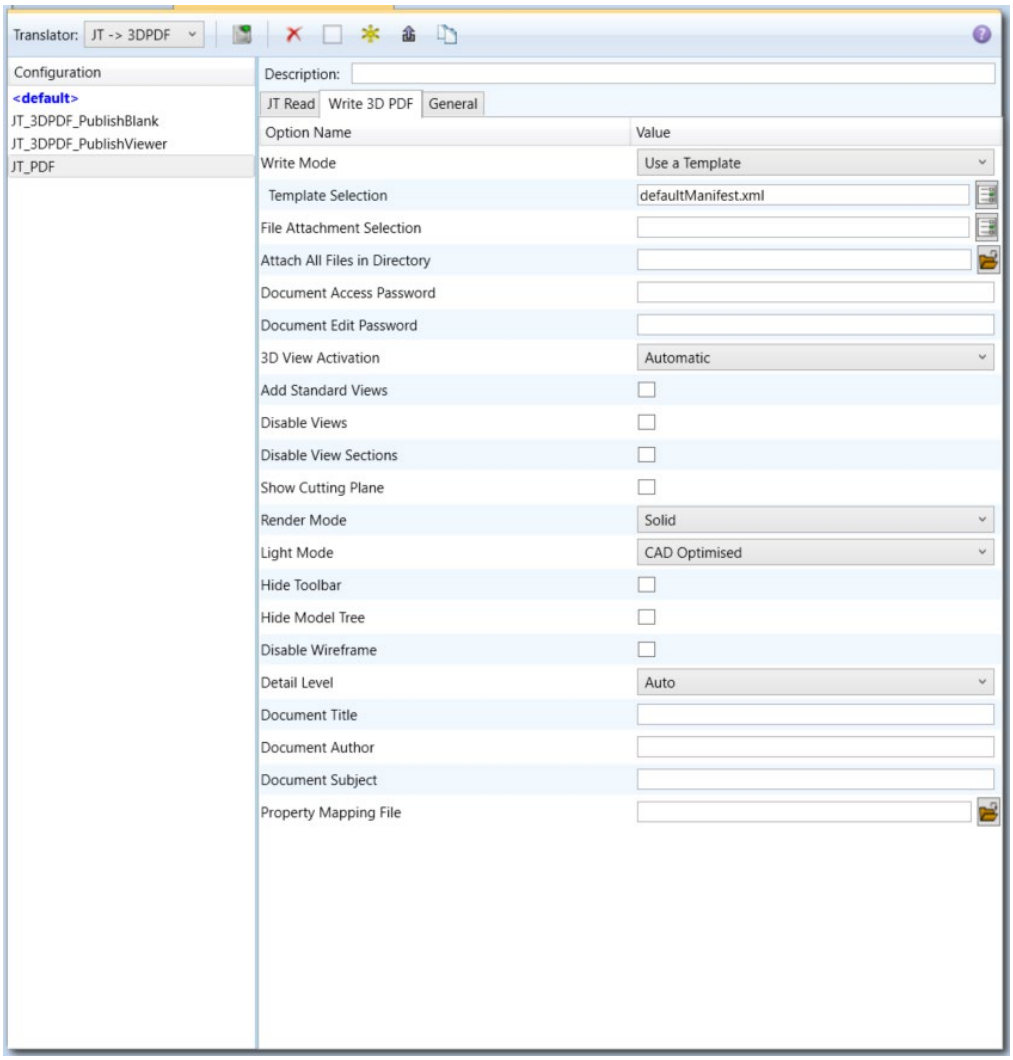
Write 3DPDF

The image below shows the Write 3D PDF arguments that are available, with their default settings:



Each of these options is described below:

Option	Description

Write Mode	Contains 5 options:
	• Use a Template – Allows data to be published to a given template *(Default)*
	  ○ Command Line Syntax:
	  ▪ *Publish std <list of Manifest files>*
	• 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*
	• HTML5 - This will create a HTML5 file which contains the 3D data
	  ○ Command Line syntax:
	  ▪ *Publish HTML5*
	• PRC – This will create a PRC file containing the 3D Data
	  ○ Command Line Syntax
	  ▪ *Publish PRC*
	• U3D – This will create a U3D file containing the 3D Data
	  ○ Command Line Syntax

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	▪ *Publish U3D*
Template Selection	Select which templates PUBLISH 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 to 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:* • *e.g.* ***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	This option allows the user to select a single directory and have all of the files in that directory to be attached to the output PDF. • Command Line Syntax ○ attach_files_in_dir <file directory>
Document Access Password	If a password is provided in the text field: • 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	If a password is provided in the text field *(Note! This password MUST be different from the Document Access Password to enable Document Edit Permissions to be honoured)*: • 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)*

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	○ Command Line Syntax: ▪ *no_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 cue 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 • Wire • 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 UI. 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)

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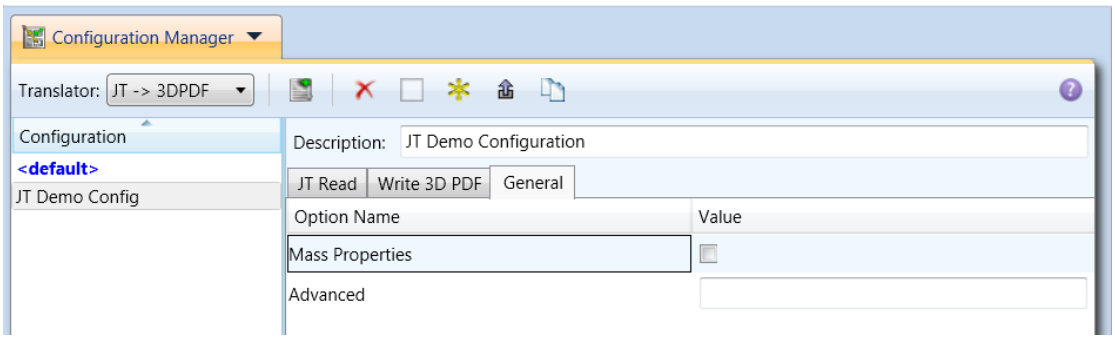
	• High (relative = 0.001%, absolute = 0.01mm) • Medium (relative = 0.01%, absolute = 0.1mm) • Low (relative = 0.1%, absolute = 1mm) • Very Low (relative = 0.5%, absolute = 5mm) • Ultra_Low (relative = 2.0%, absolute = 25mm) ▪ for very large plant data For very large (detailed) parts the use of this option (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* = ultra_low, *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>*

General

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



Each of these options is described below:

Option	Description
Mass Properties	Allows Mass Property information to be read from the source data and written as attributes to the PDF document *(Default is off)* ○ Command Line Syntax: ▪ *mprops*

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

Command Line Advanced Arguments

3D PDF Advanced Arguments

Argument	Description
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.
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.
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, all wireframe is contained in the **Wireframe** node.
attr_filter <filter file>	Selectively disable attributes from being written into PDF. e.g.EXCLUDE_ATTR_NAME attribute_name_to_exclude
attr_include <filter file>	This reverses the logic of **attr_filter** option, so attributes listed in the filter file are the ONLY attributes to be included; the filter file format is identical to the **attr_filter** option.

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attr_filter_file <filter file>	By default there is an attribute filter file installed and used - located 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,1,,, (Rename 'FILENAME' to 'F I L E')** **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. **Please note 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
copy_part_attrs <on/off>	Attributes on parts can be copied onto 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 options 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 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 to 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**

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	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 n>	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 ensure 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.

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	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.
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 and 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 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.
use_part_names <on>/<off>/<all>	Changes the names used in the output structure. Default is off When 'on' the 3DGeom node names are replaced by the TAG name from the detail (part name). When 'all' all node name use the part name
pmi_flat_to_screen <on>/<off>/<all>	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.
attach_files_in_dir <dir>	Attach ALL files found in a specified directory
add_meta_data <default>/<file>	Option to instrument the attributes in PDF with useful data, like part count, which can be used by the templates.

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	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
process_parts	When processing assemblies this will create individual PRC output for each part
part_snap_shot <default>/<WWxHH>	Option to create a .jpg image of the part or assembly being processed Option can be used without <WWxHH> command WWxHH defines the Width and Height of the image
expand_parts	Option added to create a bode per point using the TAG name.
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.
mprops	Enables Mass Property values to be exported into the PDF.
axis_systems <on>/<size in mm> with default being 25mm	Enables the output of axis systems to the PDF.
views_sort	Sorts the views presented in the 3DPDF in alphabetical order

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JT to 3D PDF Advanced Arguments

Argument	Description
ts_cfile	This option allows command line arguments to be placed in a text (command) file, with one command or option line per line, for example : **attach_files** **C:\TEMP\attach_file_list.txt** **info** **no_pmi** ***Please note!*** The progress_file <file> option cannot be added into the command file, because the progress file is opened before the command file is read**.**

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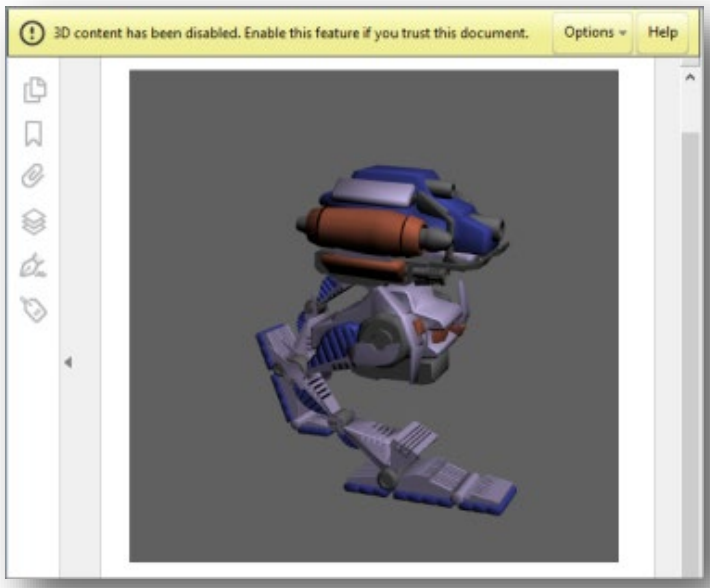
Security Changes to Adobe Reader PDF Documents

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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