# User Guide
## CREO - 3D PDF

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THEOREM HOUSE
MARSTON PARK
BONEHILL RD
TAMWORTH
B78 3HU
UNITED KINGDOM

+44(0)1827 305 350

THEOREM SOLUTIONS INC.
100 WEST BIG BEAVER
TROY
MICHIGAN
48084
USA

+(513) 576 1100

THEOREM.COM

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# Overview of PUBLISH

## About Theorem



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

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

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

## Theorem's Product Suite

Theorem have 3 main Product brands. These are:

### CAD**Translate**

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

See our website for more detail.

### CAD**Publish**

The creation of documents enriched with 3D content

See our website for more detail.

### Theorem**XR**

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

See our website for more detail.

# 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.

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



Review our website here for a short video explaining 3D PDF.

## Primary Product benefits?

- Being a direct database converter all pre and post processing is eliminated, saving time.

- Reduce costs due to processing time and increase overall conversion success levels by filtering input data and focusing the conversion to only those elements required.

- Reduce costs and risks associated to accessing the wrong version of data by integrating the conversion process into a related business processes.

- With over 20 years of industrial use Theorem translation products robustness and quality is well proven, reducing your business risk.

This document will focus specifically on guidance for the use of the Visualize 3D for CATIA V5 – JT product. For information regarding any of Theorem's product ranges please contact sales@theorem.com

# Getting Started

## Documentation & Installation Media

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

http://www.theorem.com/Documentation

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

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

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

## Installation

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

## License Configuration

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

## Using the Product

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

# Using the Product

## PUBLISH On-Demand

By default the PUBLISH for Creo 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 predefined document format.

More complex template examples are provided in the following the locations:

### *<installation_directory>\data\publish_3dpdf\*

- default_Manifest – Illustrates various Adobe capabilities ☐    publishBlank – Illustrates publishing to a blank template

### *<installation_directory>\data\publish_3dpdf\templates\*

- 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](mailto:sales@theorem.com) quoting "3D PDF Template Design".

## Default Publishing

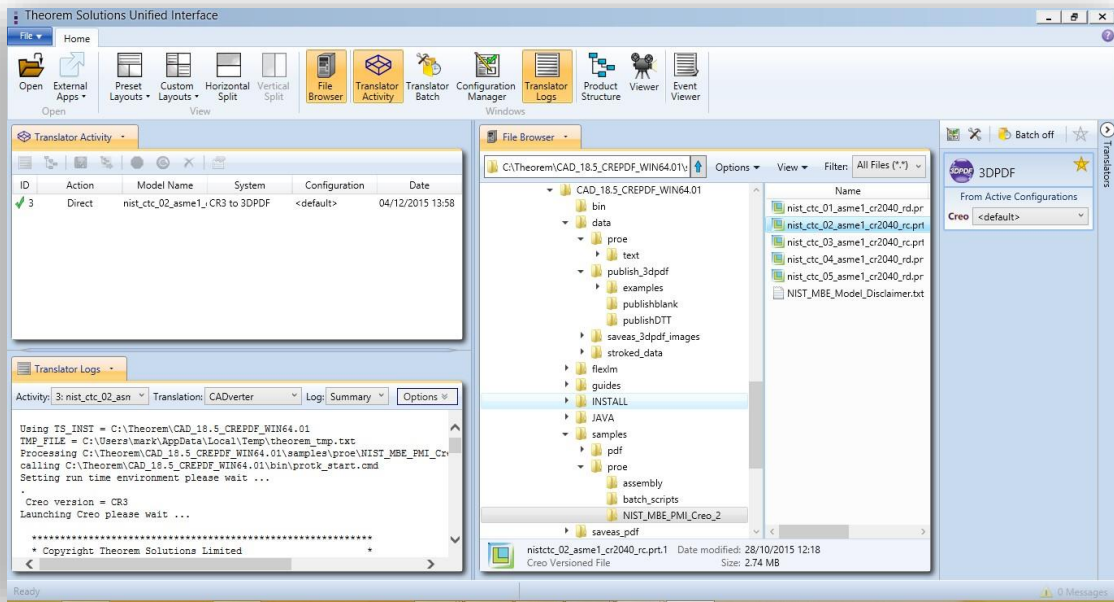*Default Publishing – via the Unified Interface*
The Unified Interface can be started via the Start Menu – if a shortcut was added during installation.

Alternatively, the Unified Interface can be run via a Windows Explorer selection in:
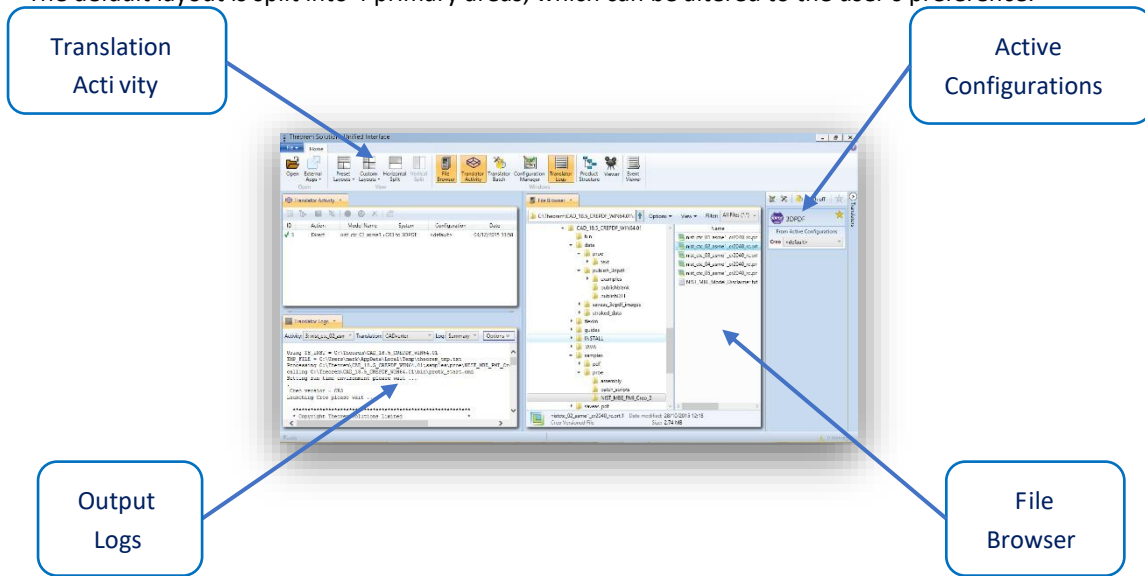
### *<UI_installation_directory>\bin\Unified_Interface.cmd*

The following interface will be launched:

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

Translation
Activity

Active
Configurations



Output
Logs

File
Browser

The simplest way to publish a Creo 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.
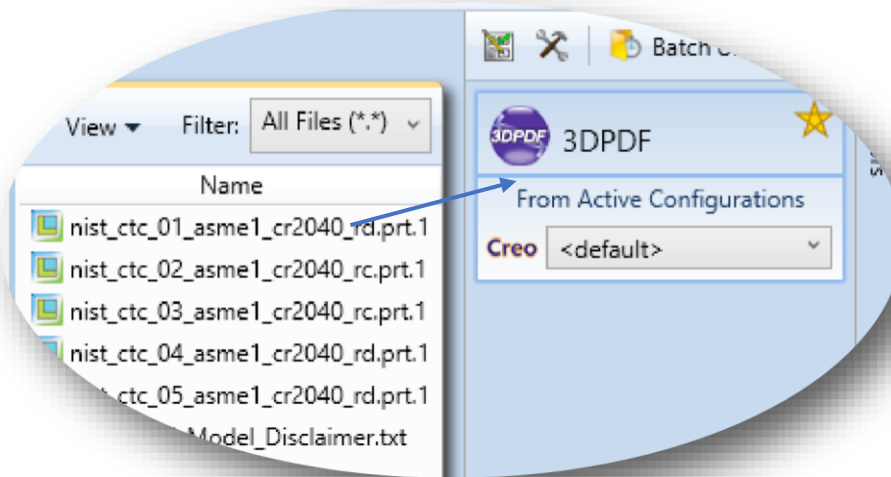


On completion, the Unified Interface will display the activity information and details from the log file created during the translation, if requested, in the Translation Activity and Output Log panes, respectively.
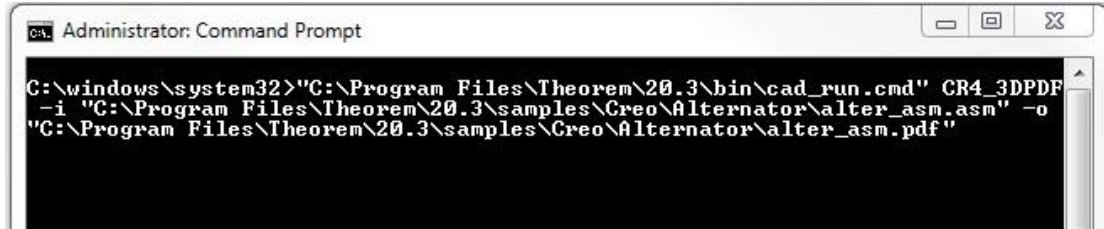
The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:
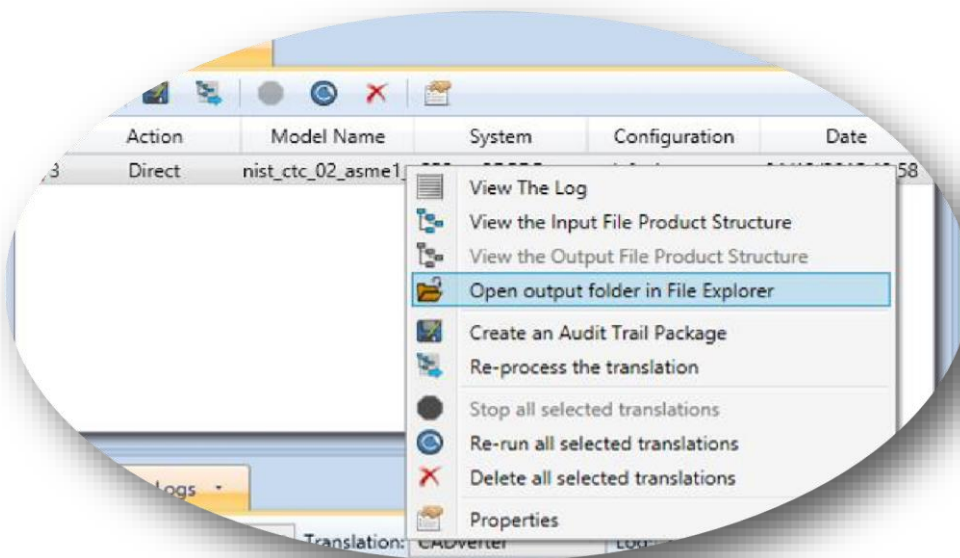


*Default Publishing – via the Command Line*

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 (Note! Replace the [version] from the example with your version of Creo, e.g. for Creo 3.0 use CR3 and Creo 4.0 use CR4):

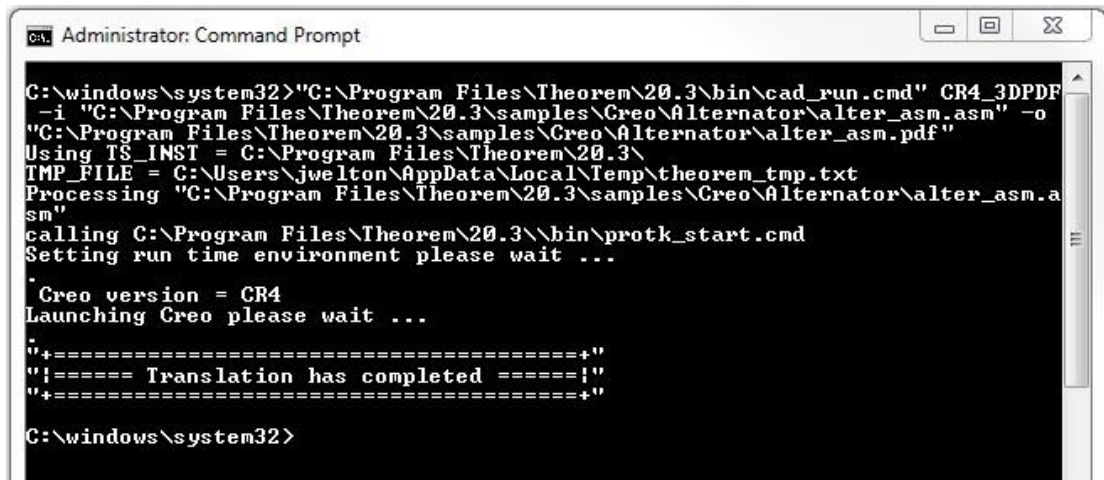**<Translator_installation_directory>\bin\cad_run.cmd CR[version]_3DPDF –i <input_file> -o <output_file>**

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



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

**C:\Program Files\Theorem\20.3\samples\Creo\Alternator\alter_asm.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:
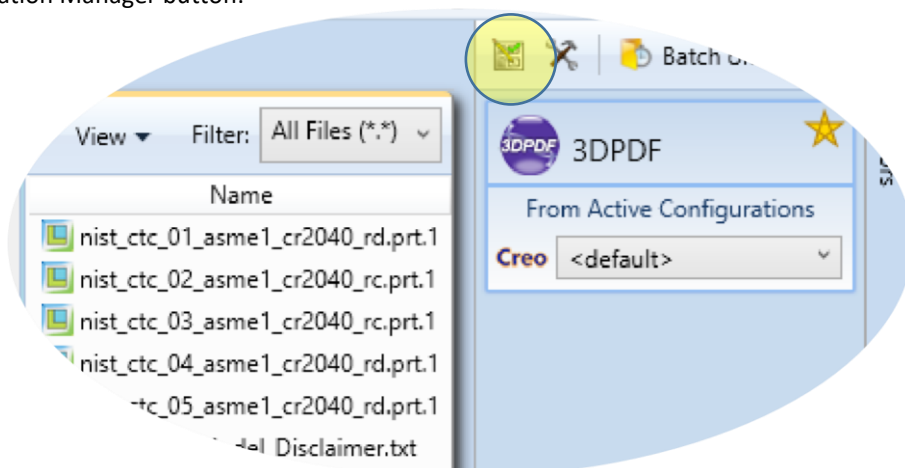
OR



Select the 'Configuration Manager' option from the ribbon menu at t      he top of the Unfied 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 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 selected only one template):
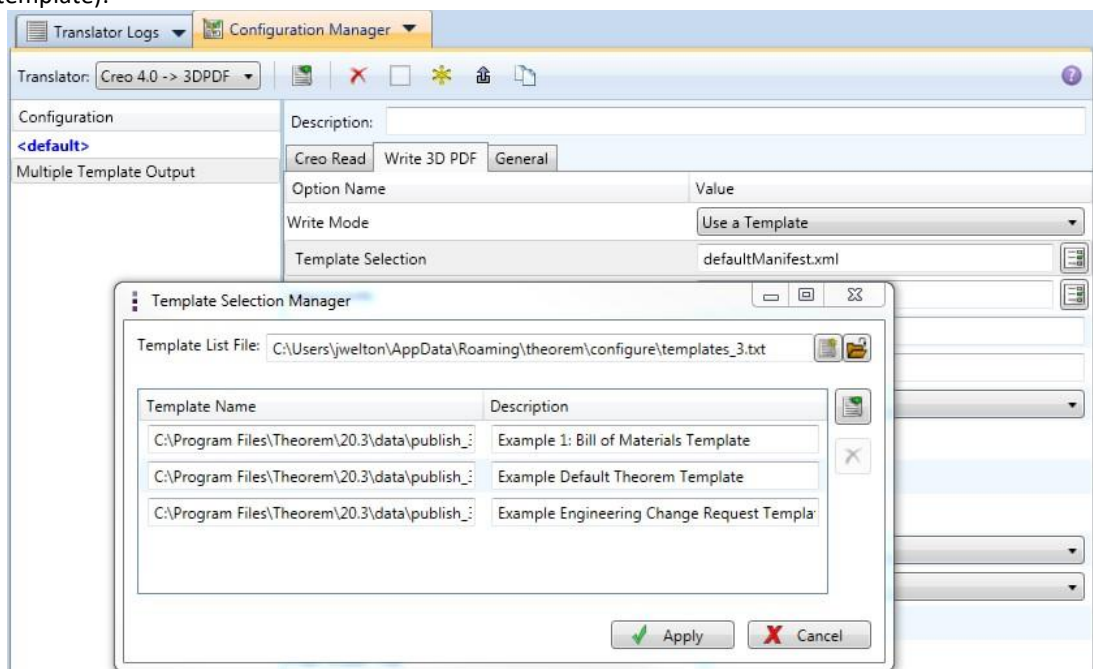


In order to run a translation using this configuration the Active Configuration setting for Creo to 3D PDF must be changed to The "Multiple Template Output" Configuration:
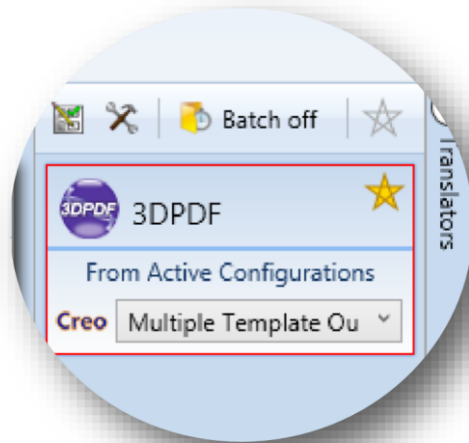
Once this has been done the translation can be invoked in the same manner as for the Default publishing mechanism

*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.

First a list of target Manifest files that describe how to use a template 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\examples\publishBOM.xml*
> *<Translator_installation_directory>\data\publish_3dpdf\examples\publishDTT.xml*
> *<Translator_installation_directory>\data\publish_3dpdf\examples\publishECR.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 (Note! Replace the [version] from the example with your version of Creo, e.g. Creo 2.0 use CR2 and Creo 3.0 use CR3):

> *<Translator_installation_directory>\bin\cad_run.cmd CR[version]_3DPDF  -i <input_file> -o*
> *<output_file> publish <manifest list>*
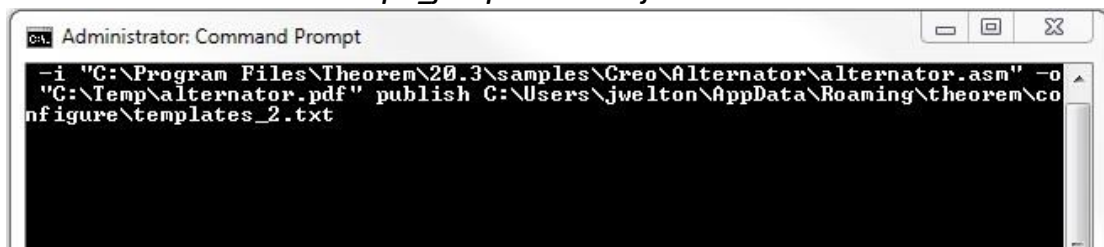


The example above will use the referenced template manifest list and will publish an output to each of the templates referenced via the manifest files provided, whether there be one or many in the list.  The following screen output should be expected:

## Publishing without a Template

*Publishing without a Template – via the Unified Interface*

To run PUBLISH without a template in 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'. Save this configuration as previously described, you are now ready to publish without using a template.
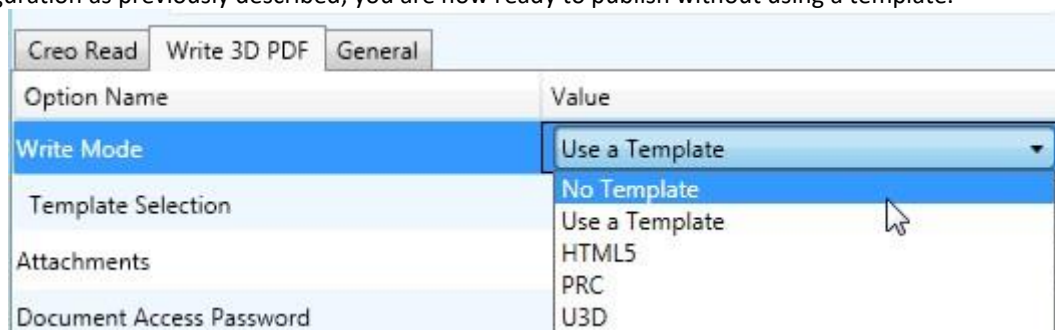


Now, in the 'Active Configurations' pane select your configuration from the menu and then 'drag and drop' your data to publish it to a PDF document.
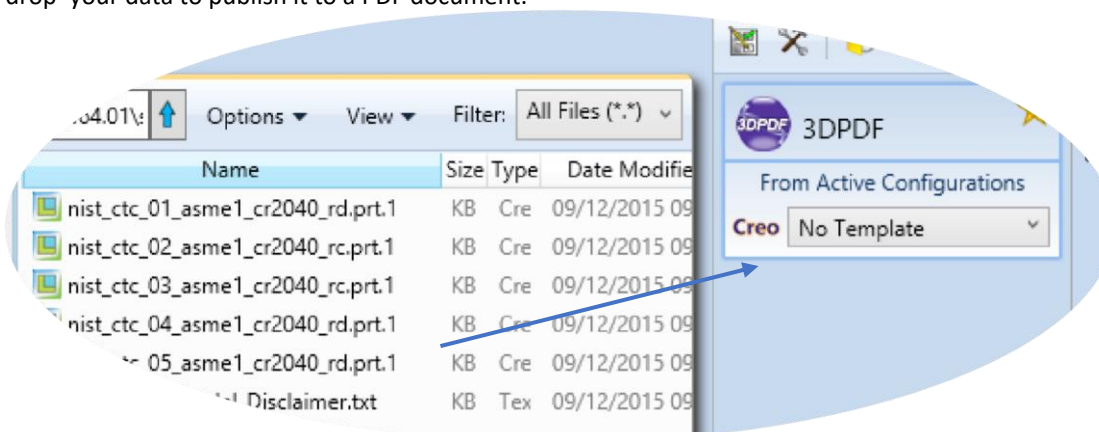


On completion, the Unified Interface will display the activity information and details from the log file created during the translation, if requested, in the Translation Activity and Output Log panes, respectively. The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:
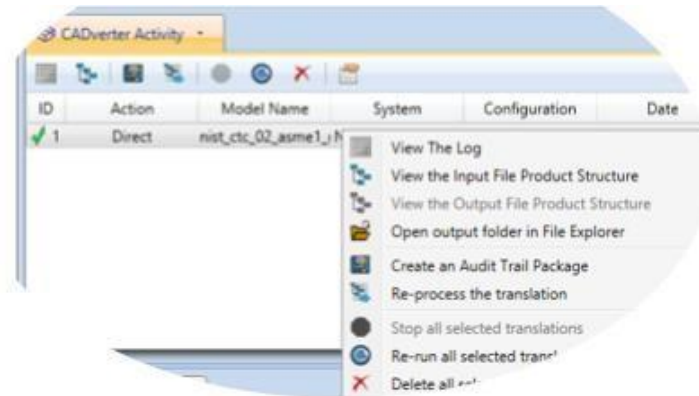
*Publishing without a Template – via the Command Line*

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

The syntax for running a command to publish without a template is as follows (Note! Replace the [version] from the example with your version of Creo, e.g. for Wildfire 5 use WF5; Creo 1.0 use CR1 and Creo 3.0 use CR3): ***<Translator_installation_directory>\bin\cad_run.cmd CR[version]_3DPDF -i <input_file> -o <output_file> publish none***
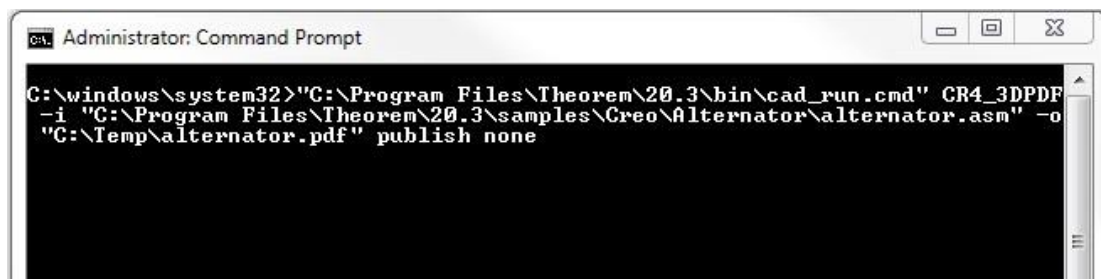


The example above will publish a PDF file containing the model data without any formatting. The following screen output should be expected:
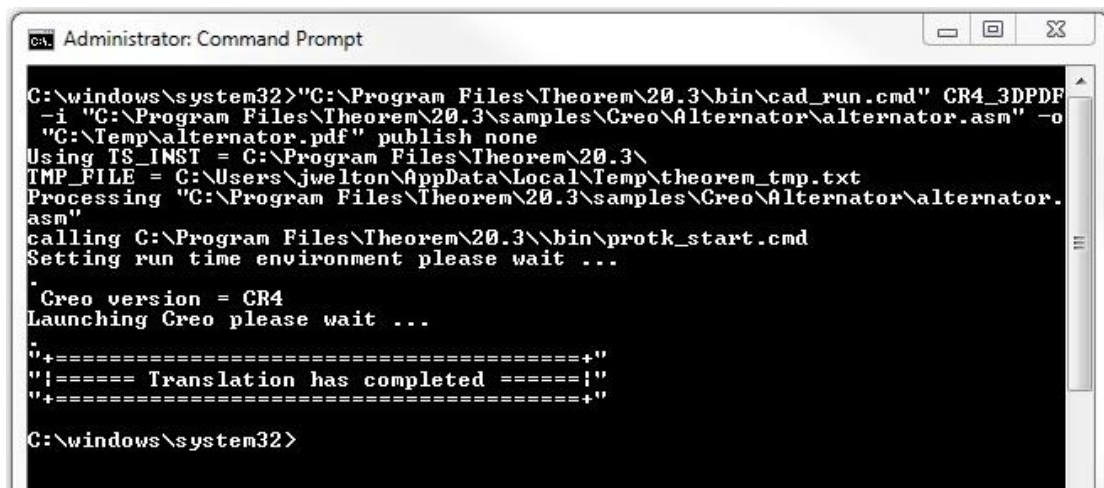


# PUBLISH Customization

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 Creo Publishing

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

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

- Write 3D PDF – Those arguments that affect how the data is written to 3D PDF

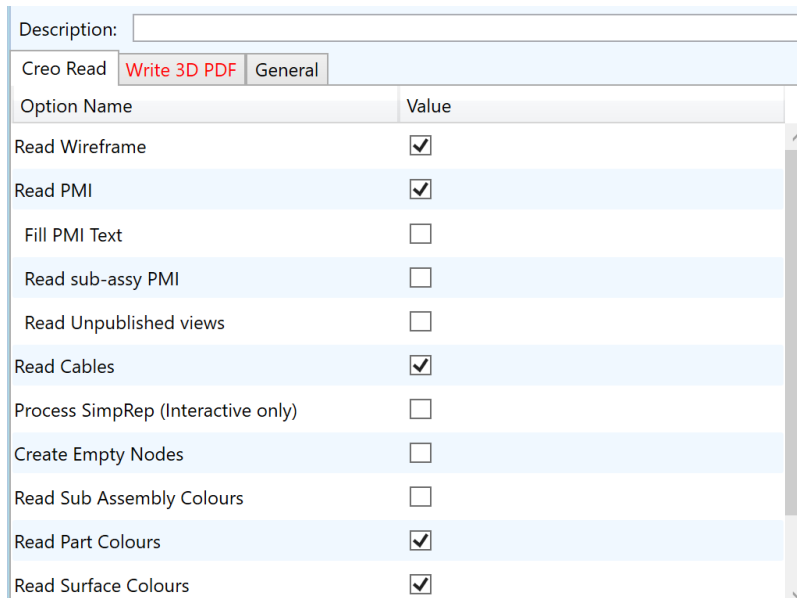- General – Those arguments that are common to ALL Publishing activities regardless of source data

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

| Description: | |
| --- | --- |
| **Creo Read**  Write 3D PDF  General | |
| **Option Name** | **Value** |
| Read Wireframe | ☑ |
| Read PMI | ☑ |
| Fill PMI Text | ☐ |
| Read sub-assy PMI | ☐ |
| Read Unpublished views | ☐ |
| Read Cables | ☑ |
| Process SimpRep (Interactive only) | ☐ |
| Create Empty Nodes | ☐ |
| Read Sub Assembly Colours | ☐ |
| Read Part Colours | ☑ |
| Read Surface Colours | ☑ |

Each of these options is described below:

| Option | Description |
| --- | --- |
| **Read Unpublished Views** | Unpublished views are no longer read by default.  New option **read_unpublished_views** has been added to allow them to be read. |
| **Don't read Wireframe** | Wireframe reading can now be turned off |
| **Read PMI** | Enables Datum Surface processing. *(Default is on).*  ◻ Command Line Syntax: ○ *no_read_pmi* – *to Disable* |
| **Fill PMI Text** | this will make the text very bold and clear, but will increases file size and run time when parts with a lot of notes are run  ◻ Command Line Syntax: ○ |

*fill_pmi_text*

| | |
|---|---|
| **Read Cables** | Enables Electrical data to be read 
 Command Line Syntax: ○ read_cables |
| **Process Simprep (Interactive only)** | Switches on the processing on Simplified reps |
| **Instance Processing** | If a list of Instance names has been generated (see above), or the specific Instance is known, this can be entered and the translation can be processed. <br> . Off <br> . List all instances in log file <br> . Process specified instance |
| **Instance Name** | Enter the name of the instance to process |

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



Each of these options is described below:

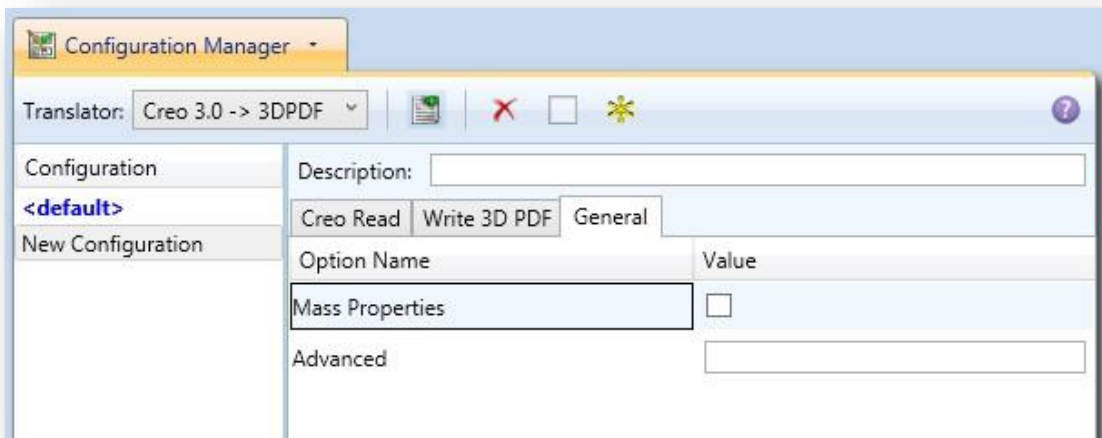| Option | Description |
|---|---|
| **Write Mode** | Contains two options:<br><br>• Use a Template – Allows data to be published to a given template *(Default)* ○ Command Line Syntax:<br> ▪ *Publish std <Path manifest list>*<br><br>• No Template – No template will be used during publishing. If this option is selected, 'Template Selection' is excluded as an option.<br> ○ Command Line Syntax:<br> ▪ *Publish none* |
| **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* |
| **Attachments** | 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)*<br><br>⬜ Command Line Syntax: ○ *Attach_file*<br><br>*<Attachment List>*<br><br>*Where Attachment_List is a list of full path names to file followed by an optional description line: e.g.*<br>**C:\TEMP\my_image.jpg**<br>**This is a jpg Image**<br>**C:\TEMP\report.txt**<br>**Report Document**<br>**C:\TEMP\related.pdf**<br>**C:\TEMP\another.pdf**<br>**This PDF has a description**<br>*Note in this example the 'Related.pdf' attached file doesn't have the optional comment.*<br>*The file path may contain environment variables which will be resolved. e.g. %TEMP%/myFile.pdf* |
| **Document Access Password** | Contains two options selectable via a checkbox:<br><br>• 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.<br> ○ Command Line Syntax:<br> ▪ *No entry required*<br><br>• 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:<br> ▪ *password <password>*<br><br>***Note:*** *If using in conjunction with the "Document Edit Password", these 2 passwords must be different.* |

| | |
|---|---|
| **Password** | Specify what the password is to be used. |
| | ☐ Command Line Syntax: ○ *See Document Access Password - This is a secondary argument to this command.* |
| **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>* |
| | ***Note:*** *If using in conjunction with the "Document Access Password", these 2 passwords must be different.* |
| **Password** | Specify what the password is to be used. Note this needs to be different to the Document Access Password |
| | ☐ Command Line Syntax: |
| |     ▪ *See Document Edit Password - This is a secondary argument to this command.* |
| **3D View Activation** | Specifies how the 3D view in the resultant 3D PDF file is activated *(Default is On Selection) Options are:* ○ *Automatic* ○ *On Selection* ○ *Page Visible* |
| | ☐ Command Line Syntax: ○ *activate_mode PAGE_OPEN (for Automatic)* ○ *activate_mode CLICK (for On Selection)* ○ *activate_mode PAGE_VISIBLE (for Page Visible)* |
| **Disable Views** | Switch off any Captures/Views from being written into the PDF. *(Default is off)* |
| | ☐ 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 cut plane visualization (Default is off) ☐ Command Line Syntax: ○ *cutplane_on* |

| | |
|---|---|
| **Render Mode** | Controls which Rendering Mode to use for the model.  Options are:<br>• Solid *(Default)*<br>• Transparent<br>• Wire<br>• Illustrated<br>• Outline<br>• Shaded ○ Command Line Syntax:<br>    ▪ *Render_mode <mode>*<br>    Where mode is set to one of the above values |
| **Light Mode** | Controls which Light Mode to use. Options are:<br>• Default<br>• Day<br>• Bright<br>• Prim<br>• Night<br>• Blue<br>• Red<br>• Cube<br>• Head ○ Command Line Syntax: |
| | ▪ *light_mode <mode>*<br>Where mode is set to one of the above values.<br>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)* ⬜<br>    Command Line Syntax: ○<br>    *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: ○<br>*hide_model_tree* |

| | |
|---|---|
| **Detail Level** | The Detail Level option can be used to set the render quality of the resultant 3D PDF output.<br>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.<br><br>The following options are available:<br>    &bull;   Auto       (Corresponds approximately to High)<br>    &bull;   Very High  (relative = 0.0001%, absolute = 0.001mm)<br>    &bull;   High       (relative = 0.001%, absolute = 0.01mm)<br>    &bull;   Medium   (relative = 0.01%, absolute = 0.1mm)<br>    &bull;   Low        (relative = 0.1%, absolute = 1mm)<br>    &bull;   Very Low  (relative = 0.5%, absolute = 5mm)<br>    &bull;   Ultra_Low  (relative = 2.0%, absolute = 25mm)<br>           &bull;   for very large plant data<br><br>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.<br><br>A setting of Medium is seen as a good compromise, between quality and size. This value is the default value.<br><br>    &#9633;   Command Line Syntax:<br>        o *lod_level \<value\>*<br><br>   *Where value =* ultra_low, *v_low, low, med, high, v_high, off* |
| **Document Title** | Set the Document Properties 'title' field.<br>    &#9633;   Command Line Syntax: o<br>    *Doc_description \<text\>*<br>    The text should be quoted |
| **Document** | Set the Document Properties 'author' field &#9633;<br>    Command Line Syntax: |
| **Author** |     o *doc_description_author \<text\>*  The text<br>    should be quoted |
| **Document Subject** | Set the Document Properties 'subject' field &#9633;<br>    Command Line Syntax: o<br>    *doc_description_subject \<text\>*  The text should be quoted |

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

# Command Line Advanced Arguments

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

## Important Note

It is important to understand the interaction/hierarchy which exists between template options files, config file and command line arguments will define the result. In essence the config file is a set of command line arguments and therefore will always over-rule the template options. Also if any commands specified in the 'Advanced Options' contradict the standard config file settings then they will take precedence.

## Creo Advanced Arguments

| Argument | Description |
|---|---|
| **pmi_pcurves** | Store non planar PMI graphics<br>(leaders not in the plane of the annotation) |
| **Transfer Solids** | Enables solid processing. (Default is on).<br>   • Command Line Syntax:  no_solids – to Disable |

| | |
|---|---|
| **Transfer Quilts** | Enables quilt processing. (Default is on).<br>• Command Line Syntax: no_quilts  – to Disable |
| **Transfer Datum Curves** | Enables Datum Curve processing. (Default is on).<br>• Command Line Syntax: no_datum_curves – to Disable |
| **Transfer Datum Surfaces** | Enables Datum Surface processing. (Default is on).<br>• Command Line Syntax: no_datum_surfaces – to Disable |
| **unique_occ** | Read multiple occurrences, eg support for occurrence PMI associations |
| **read_assy_pmi** | reads PMI in lower level assembly parts |
| **ignore_std_views** | Disable reading views with standard names (TOP, LEFT, etc). |
| **ignore_view_list <file>** | supply a list of view names to be ignored |
| **no_exploded_views** | Disables exploded views. |
| **part_level_views** | *Default: **off***<br>Enable the processing of Part Level Views within an assembly. |
| **part_level_views_moved** | Process part level views and move into assembly space, such that only one part instance's views are displayed. |
| **part_level_pmi** | *Default: **off***<br>Enable the processing of PMI on parts within an assembly. |
| **view_part_name** | Uses the part name in the view names to help identify the views when selected in PDF |
| **views_geom_exploded <on \| off \| hybrid \| hybrid2>** | *Default: **hybrid***<br>Geometry grouped into assembly nodes for views is by default done on a view bases, which re-uses geometry where possible. This option creates a node for every item of geometry so that the views can hide/show them in the views. This can (depending upon the data and views) reduce the resulting file size and shorten the translation time, BUT |

| | in some cases the PDF is slow to respond due to the VIEW limitations. |
|---|---|
| | *exploded - off* - create a node per view with all geometry / wire etc for that view |
| | *explode - on* - create nodes for very solid / wireframe etc that can be referenced by views |
| | *hybrid (default)* - mixture of explode on/off - solids being exploded and wireframe grouped (best compromise) |
| | *hybrind2* - as hybrid with points also grouped |
| *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. |
| *hide_merged_quilts* | This hides any 'Merged Features' that are greyed out in the model tree to overcome the fact that the Creo APIs do not return the status as (Hidden/Not Active/NotVisible etc.) |
| *process_simprep* | Allows the user to specify the Simplified Representation that they wish to process. NOTE: This is currently only available in Interactive (Save As) mode. |

## 3DPDF 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***<br><br>***<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.<br><br>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.<br><br>This option allows the PMI colour to be fixed to any colour, clearly this is a single override colour for all PMI.<br><br>**NOTE:** It is recommended that this value is not set if colour coded PMI values are required |
| ***wireframe_RGB***<br><br>***<r g b>*** | Where r g b are values 1 to 255, this controls the override Wireframe colour.<br><br>By default wireframe is displayed in the same colour(s) as the source CAD system.<br><br>This option allows the wireframe colour to be fixed to any colour, clearly this is a single override colour for all wireframe. |

| 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_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.<br><br>**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**<br><br>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 retain.  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` 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.<br><br>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/off>* | Wireframe entities that are marked as invisible are by default NOT processed. This option allows them to be processed. |
| *no_nx_camera_fixup* | By default extra checks are made to prevent views with clipped geometry.  This flag disables these checks. |
| *pmi_assoc_limit <off/on>* | 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.<br><br>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.<br><br>(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)<br><br>0x01 (1) - **DocumentPermOpen** The user can open and decrypt the document.<br><br>0x02 (2) - **DocumentPermSecure** The user can change the document's security settings.<br><br>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.<br><br>0x08 (8) - **DocumentPermEdit** The user can edit the document more than adding or modifying text notes (see also DocumentPermEditNotes). In the Document Security dialog, |

this corresponds to the Changing the Document entry.

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

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

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

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

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

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

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

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

0x10000 (65536) - **DocumentPermFormSubmit** This should be set if the user can submit forms outside of the browser.

This bit is a supplement to DocumentPermFillandSign.

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

0xFFFFFFFF - All permissions.

| | |
|---|---|
| *expand_part* | This option creates a pseudo assembly structure below a part, such that each solid is modelled by a separate node. The default is to combine all solids that belong to the same part.<br><br>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.<br><br>Values < 1.0 zoom out and values > 1.0 zoom in.<br><br>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.<br><br>Every effort has been made to set a good value by default. |
| *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. |
| *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<br><br>'on' allows for FLAT TO TEXT types to be processed<br><br>'all' allows all FLAT TO SCREEN types to be process, however only the text and NOT the frames are processed |
| *attach_files_in_dir <dir>* | Attach ALL file found in specified directory |
| *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 |

# PUBLISH Interactive

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

In order to publish from within Creo, the Creo application must be started from within a Theorem environment, so that the appropriate Creo menus are loaded.

Creo can be started from a shortcut, if requested at installation time. Alternatively, it can be started via the script provided in the PUBLISH installation at:

***<installation_directory>\bin\protk_start[version].cmd***

(Where [version] should be substituted _cr5 for Creo 5.0 (e.g. protk_start_cr5.cmd), _cr6 for Creo 6.0, and _cr7 for Creo 7.0):
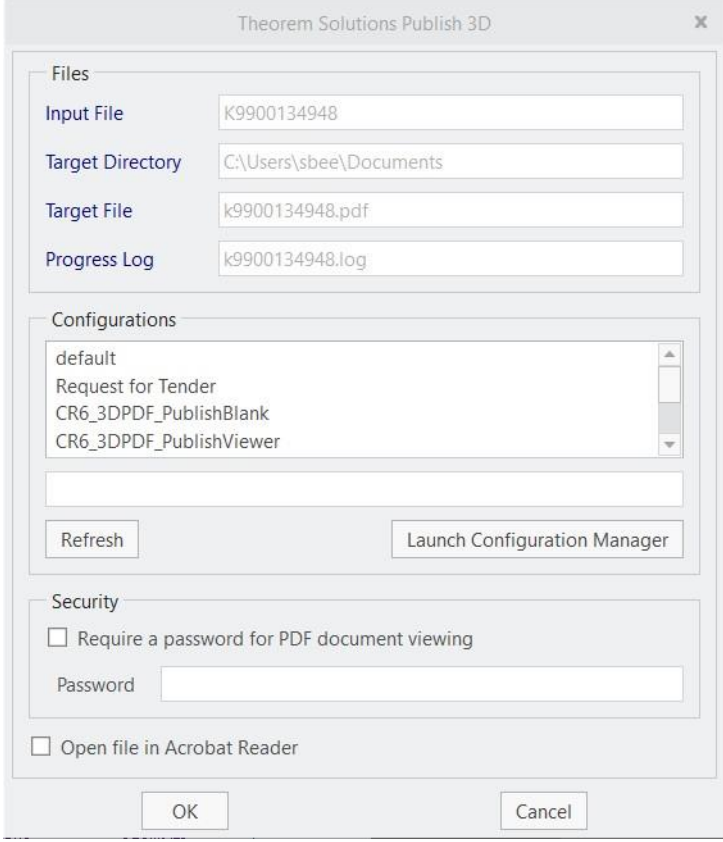
Having selected an output folder and a model loaded, the active Part or Assembly can be exported by selecting the File->Save As->Save a Copy menu:
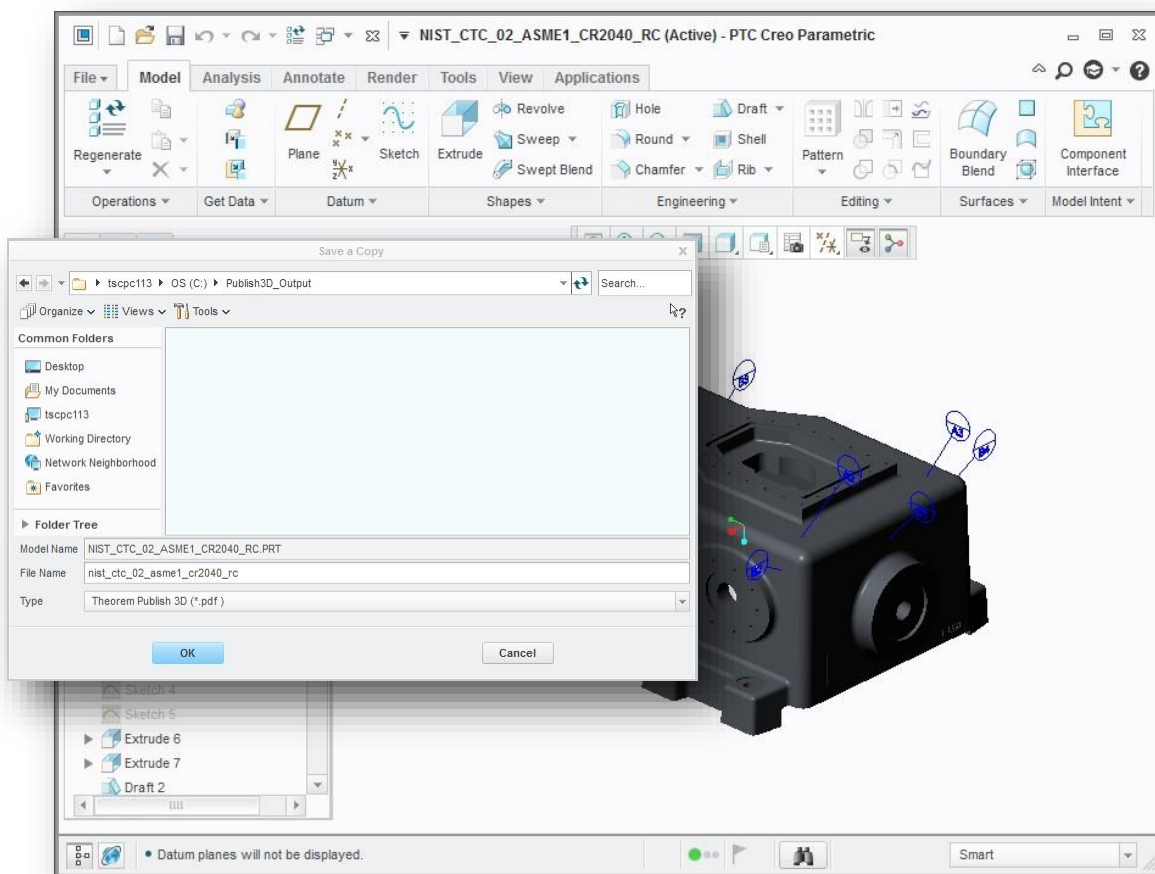


The user will be able to specify:

- A target output directory
- A target file name
- The log file (progress file) will be named the same as the target file name with a .log suffix

Having selected the output type as Theorem PUBLISH (*.pdf) from the pull-down menu the panel below will be presented to the user:



The Publish Interactive interface allows the user to specify:

- A predefined Configuration (a custom set of translator options)

From Revision 23.0, a customer having an 'On Demand' License can launch the Configuration Manager to create or modify configurations.
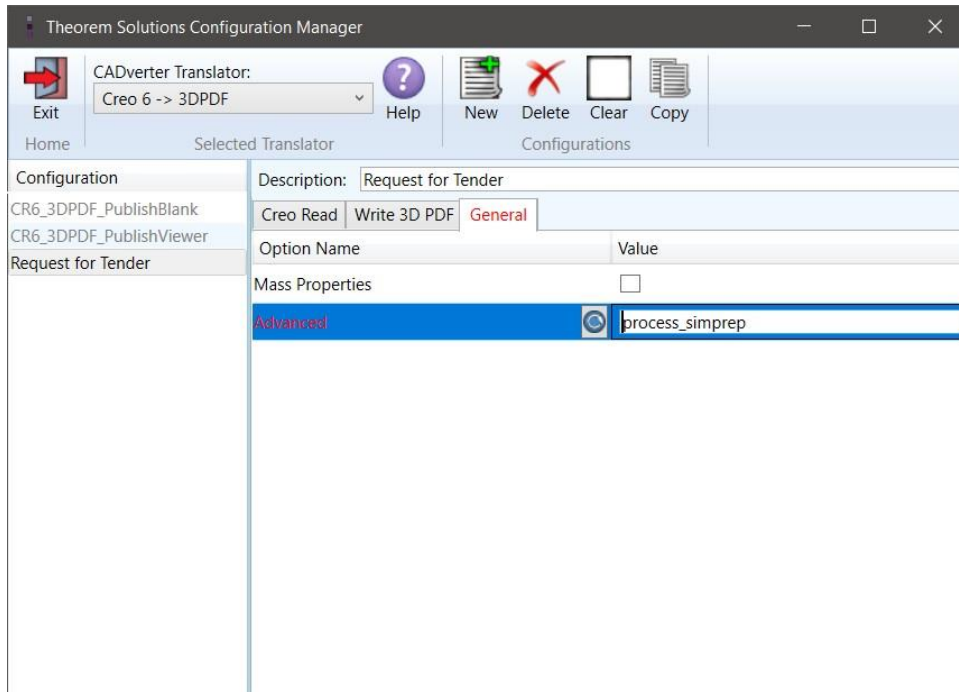
When this has been done the new/modified configuration can be immediately displayed in the list when the 'Refresh' button is clicked.

- 3D PDF Output Options, such as:
  o Adding a password
  o Auto Open the PDF file using Adobe Acrobat Reader

A new option can be set using the configuration manager. If 'process_simprep' is specified in the 'Advanced' text box on the 'General' page, then in Interactive mode a menu is displayed showing a list of available Reps. Select one then press 'Done' when you wish the translation to continue.
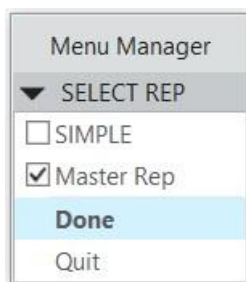


On selecting **'OK'** on the Publish Interactive interface the session will cycle through any 3D views and some flashing of pop-up windows may be seen as the translator processes PMI data.  On completion the "Translation Complete" status message will be displayed in the bottom left hand corner:
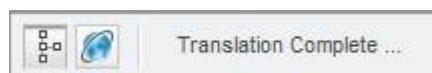


## Configuration Manager

The configurations listed under the Theorem Options panel are managed using the Configuration Manager from the Unified Interface for PUBLISH *On Demand* users (See 'Configuration Manager' on Page 12).

For PUBLISH Interactive only users, the configurations are managed using a standalone Configuration Manager that is launched by running the ConfigurationManager.cmd file from the \bin directory of your Theorem installation. The Configuration Manager is displayed below and is used in the same manner as the Configuration Manager used in the Unified Interface:
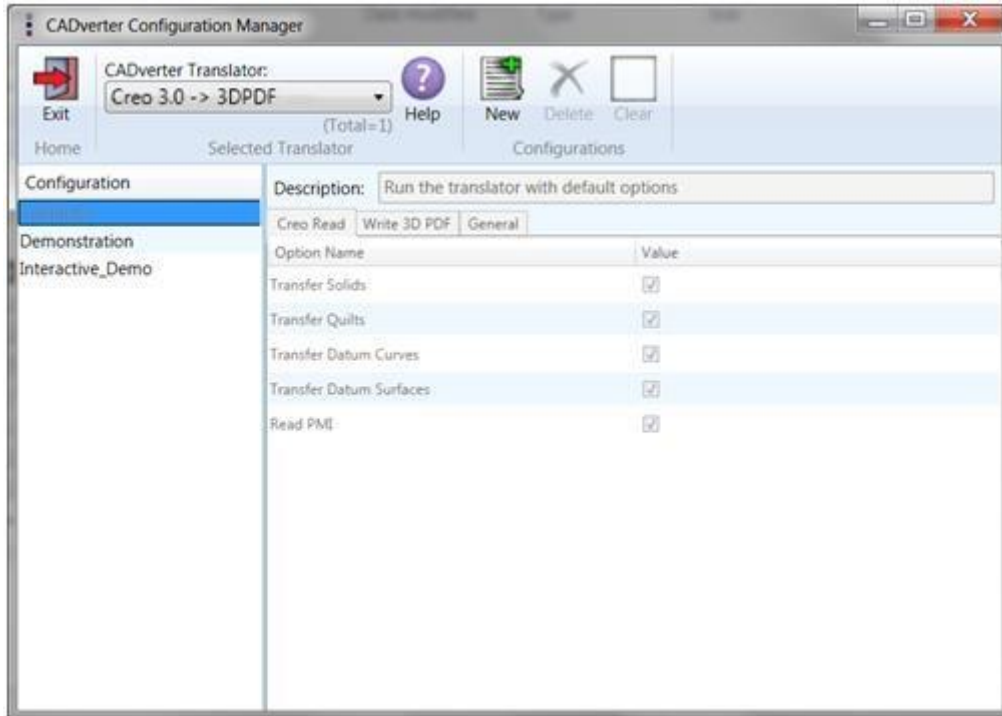


# PDF Documents

## Security Changes to Adobe Reader

Adobe have implemented new security options to disable the autoplay of 3D content in PDF files. The following banner will be displayed upon opening a PDF file that contains 3D content:
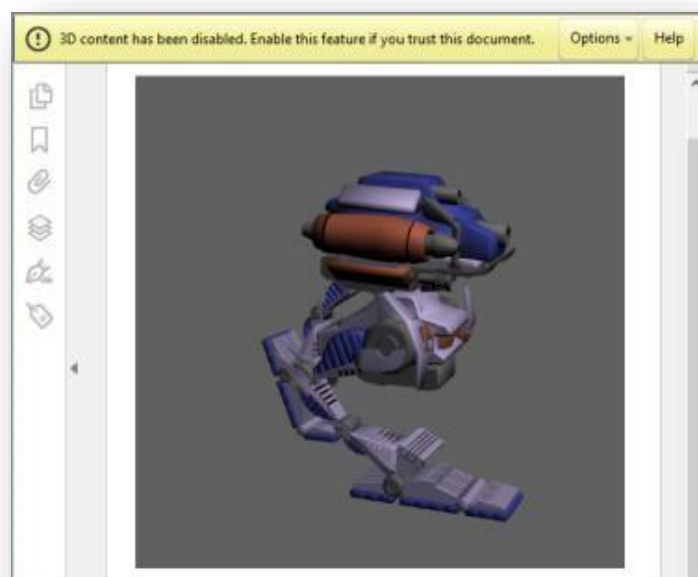
To enable 3D content in the current document:

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

# THEOREM
## SOLUTIONS