



## TRANSLATE for CATIA V4 - NX



# USER GUIDE

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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.

## Theorem's Product Suite

Theorem have 3 main Product brands. These are:



### *TRANSLATE*

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

See our [website](#) for more detail.



### *PUBLISH*

The creation of documents enriched with 3D content

See our [website](#) for more detail.



### *VISUALIZE*

Visualization for [Augmented \(AR\)](#), [Mixed \(MR\)](#) and [Virtual \(VR\)](#) Reality applications

See our [website](#) for more detail.

## The CATIA V4 Bi-directional NX Translator

This document provides outline information regarding the use of Theorem's CATIA V4 to NX Translator.

For further information please refer to the AVI's provided on our web site at:

<http://www.theorem.com/Documentation>

### Getting Started

#### Documentation

The latest copy of this documentation can be found on our web site at:

<http://www.theorem.com/Documentation>

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

### Installation Media

The latest copy of Theorem software can be found via our web site at:

<http://www.theorem.com/Product-Release-Notes>

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

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

### Installation

The installation is run from the CD or ZIP file download provided.



To install the translator, select the **Setup.exe** file and follow the installation process. For a full guide to the process, please see our 'Translator Installation Process' demonstration video located [here](#).

### License Configuration



In order for the translation to run successfully, the Theorem license file provided to you needs to be configured using FlexLM. For a full guide to this process, please see our 'FlexLM License Set Up and Configuration' demonstration video located [here](#).

## Running the Product

Once configured and licensed, the product is ready to be run.

- Via the Command Line



- The Command Line Interface provides a direct method of invoking the translator. It can be used via a DOS shell or called via a third party application as part of a wider process requirement.
- For a full guide to this process, please see our 'How to Translate Using the CATIA V4 – NX Translator via the Command Line' demonstration video located [here](#).

## Using the Product

### Default Translations

#### Default Translation – 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 when translating from CATIA V4 to NX:

```
<Translator_installation_directory>\bin\cad_run.cmd CATIA_NX[XX] <input_file>  
<output_file>
```

The format of the command is as follows when translating from NX to CATIA V4:

```
<Translator_installation_directory>\bin\cad_run.cmd NX[XX]_CATIA <input_file>  
<output_file>
```

(Note! Replace the [XX] seen in the example with the version of NX you are using. E.g. for NX11 change to UnigraphicsNX11):

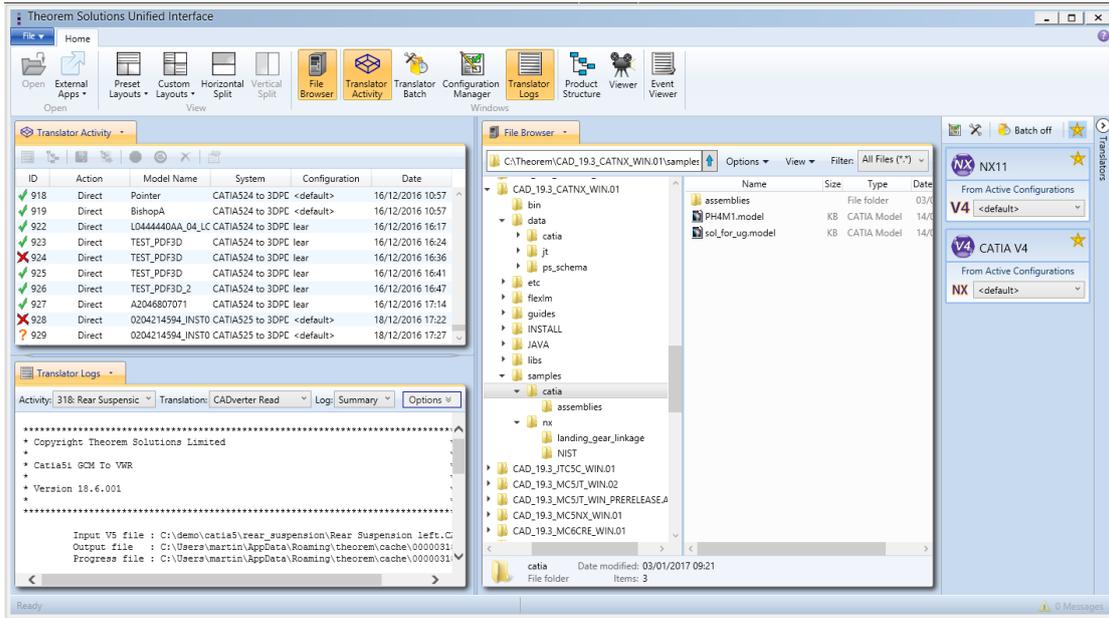
#### Default Translation – via the Unified Interface

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

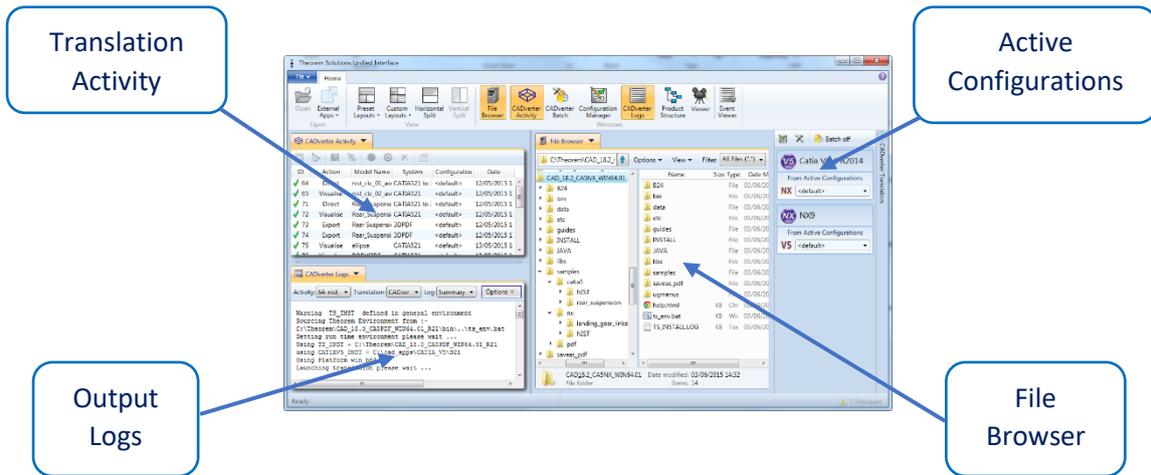
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

```
<UI_installation_directory>\bin\Unified_Interface.cmd
```

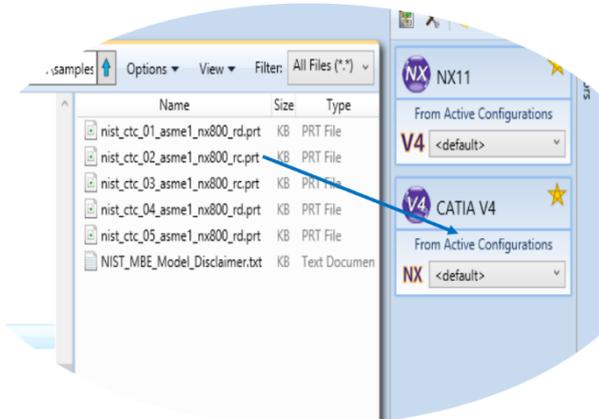
The following interface will be launched:



The default layout is split into 4 primary areas, which can be altered to the users prefer:



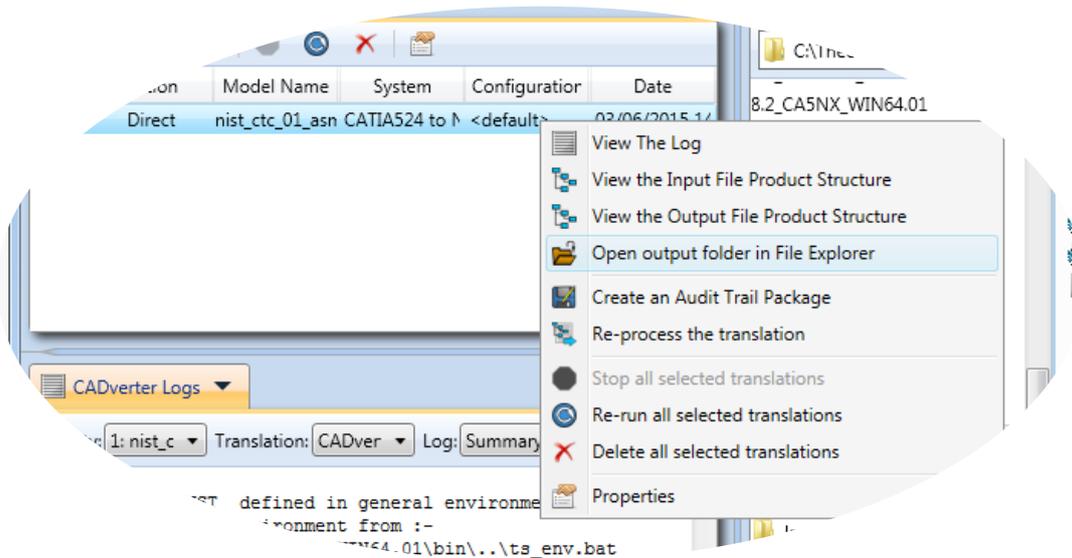
The simplest way to translate from CATIA V4 or NX is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



## TRANSLATE for CATIAV4 to NX

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:



## Customizing Translation Output

The following sections describe, in outline, available command line arguments to customize the output of the CATIA V4 to NX translator.

Some of these arguments are available via the Unified Interface. All can be used as additional arguments on the default command line:

## CATIA V4 to NX Arguments List

### CATIA Read Arguments List

<b>CMD LINE Option</b>	<b>Purpose</b>	<b>Data Type</b>	<b>Default</b>
<b>model</b>	Model number in export file	Int	1
<b>mvs</b>	Mainframe real conversion	Flag	Off
<b>cnode</b>	Read Entity Tag Names	Flag	Off
<b>ppoint</b>	Read Entity Tag Names	Flag	Off
<b>prop</b>	Unknown	Flag	Off
<b>cont</b>	Continuity required	Tolerance	Off
<b>prim_surf</b>	Primitive surface redefinition required	Flag	Off
<b>offditto</b>	Explode assembly structure during read	Flag	Off
<b>cvm &lt;set layer etype view&gt;</b>	Colour visualization mode as per set, layer, entity type or view	Choice set layer etype view	Off
<b>dim_view_realm</b>	Dimensions are view realm	Flag	Off
<b>dim_draft_realm</b>	Dimensions are draft realm	Flag	On
<b>set_read</b>	Read associated set name and store in attribute	Flag	Off
<b>heal_degen_surf</b>	Heal partially degenerate surfaces	Flag	Off
<b>show_info</b>	Displays information panel in progress file	Flag	On
<b>read_set &lt;set_name&gt;</b>	Read entities in named sets	Char *	All



<b>read_tag</b> <entity_name>	Read named entities	Char *	All
<b>surf_check_file</b> <file_name>	Creates a surf check file	Char *	
<b>catia_fonts</b> <file_name>	Catia graphism font mapping file	Char *	
<b>search_path</b> <search_path>	Search path used to file model files whilst reading asm or session files	Char *	
<b>only_use_search_path</b>	only_use_search_path	Flag	Off
<b>face_colours</b>	Read colours as applied to solid faces	Flag	Off
<b>solid_colours</b>	Read colour applied to solid	Flag	On
<b>pdegen &lt;tol&gt;</b>	Process pdegen surfs	Flag / Tol	Off
<b>group_pipes</b>	Group pipes in a detail	Flag	Off
<b>group_pipelines</b>	Group pipelines in a detail	Flag	Off
<b>ungroup_pipelines</b>	Dont group pipelines in a detail	Flag	Off
<b>use_axs_txt_name</b>	Names axis as per associated text	Flag	Off
<b>use_axs_name</b>	Names axis as per tag name i.e. *AXS1	Flag	On
<b>pipeline_layer</b>	Specify layer on which to put pipe line	Int 1-254	Use pipe segment layer
<b>facet_colour</b>	Read facet colours for solidm	Flag	Off
<b>no_facet_colour</b>	Dont read facet colours for solidm	Flag	On
<b>gvp</b>	Read validation properties	Flag	Off
<b>gvp_dont_subtract</b>	Calculate validation properties on individual volumns	Flag	Off
<b>gvp_absolute</b>	Calculate absolute validation properties		
<b>gvp_noshow</b>	Calculate validation properties if hidden		



<b>gvp_reprofile</b> <file_name>	validation properties report file name	Char *	
<b>gvp_outfile</b> <file_name>	validation properties output file name	Char *	
<b>noshow</b>	Read noshowed entities	Flag	Off
<b>draft</b>	Read 2D draftings entities	Flag	Off

NX Write Arguments

<b>CMD LINE Option</b>	<b>Purpose</b>	<b>Data Type</b>	<b>Default</b>
<b>poly_sol/no_poly_sol</b>	For gco Fsolids produce Facetted bodies (else attempt brep)	Flag	off
<b>heal_ug &lt;tol&gt; [def tol = 0.0095/units]</b>	attempt a UG heal on the created body (if nocheck on)	Flag	off
<b>keep_all_bodies/no_keep_all_bodies</b>	If input solid gets created as a solid after sewing, plus one or more tiny sheet bodies, keep or delete these	Flag	on (keep all)
<b>nocheck</b>	Don't check created Parasolid geometric entities	Flag	off
<b>no_brep_prep</b>	Prepare solids switched off	Flag	on (surfs read as nurbs+prep)
<b>pstolmodel &lt;num&gt;/nopstolmodel [def num = 3]</b>	Enable Parasolid tolerant modeling	Flag	on
<b>pssew &lt;tol&gt;/nosew</b>	Sew failed breps and opensols	Flag	on
<b>csg_prep &lt;tol&gt; [def tol = 0.000001*scale]</b>	Prepare CSG Primitives	Flag	off



<b>csg_shift &lt;tol&gt; [def tol = 0.000001*scale]</b>	Change CSG Shift Distance	Flag	off
<b>csgfix</b>	Fix CSG Primitives	Flag	off
<b>ps_fix_small/no_fix_ps_small</b>	Remove small edges, sliver and spike faces in breps	Flag	off
<b>ps_fix_osol/no_ps_fix_osol</b>	Remove small edges, sliver and spike faces in opensolids	Flag	off



## NX to CATIA V4 Arguments List

## NX Read Arguments List

<b>CMD LINE Option</b>	<b>Purpose</b>	<b>Data Type</b>	<b>Default</b>
<b>read_name no_read_name</b>	Read UG entity names (if they exist)	Flag	off
<b>part_layer</b>	Process As Saved part layers, else All	Flag	ALL
<b>read_pmi</b>	Read PMI as stroked data	Flag	off
<b>noprep/prepsol</b>	Prepare solids switched off / on	Flag	on (surfs read as nurbs+prep)
<b>rd_native_edge/no_read_native_edge</b>	Read native edge curves	Flag	off (read as nurbs curves)
<b>trim_face_surfs/no_trim_face_surfs</b>	Trim surface to face	Flag	off (don't trim)
<b>ugdiags</b>	Switch on validate read to progress file	Flag	off
<b>read_diags</b>	Switch on read diagnostics to progress file	Flag	off
<b>no_mergen</b>	No Parasolid merging of entities	Flag	on (merge)
<b>checksol/nochecksol</b>	Check Parasolid entities before read	Flag	off (don't check)
<b>noprep/prepsol</b>	Prepare solids switched off / on	Flag	on (surfs read as nurbs+prep)
<b>mprops</b>	Read Mass Props	Flag	off
<b>draft</b>	Process 2D drawings	Flag	off

CATIA Write Arguments List

<b>CMD LINE Option</b>	<b>Purpose</b>	<b>Data Type</b>	<b>Default</b>
<b>mvs</b>	Create a mainframe real conversion model file	Flag	Off
<b>catia_v3, catiav3, v3</b>	Produce a Catia V3 file	Flag	Off
<b>surfopt l &lt;tol&gt;</b>	Controls surface optimization tolerance	Double	Off / 0.001
<b>maxtol &lt;tol&gt;, max_tol</b>	Maximum solid faceting tolerance to be applied to solids which are found to be too large to go into CATIA	Double	Off
<b>model_dimension l &lt;tol&gt; &lt;units&gt;, modeldimension, mdim</b>	Catia Model Dimension. Followed by number, followed by blank or “inch” or “mm” specifies model dimension to be number of part units, inch or mm.	Double	Off
<b>startpart, start_part, start-part</b>	Name of model to be used as seed part to provide site specific information to the created CATIA model file	Char *	Off
<b>startpartprefix</b>	Prefix of Startpart file to use allows automatic switching dependent on units	Char *	Filename
<b>solide</b>	Create import solides rather than volumes	Flag	Off
<b>solidv</b>	Create complex volume solide rather than volume (doubles size of model file)	Flag	Off
<b>solidm</b>	Creates faceted solidm	Flag	Off
<b>version</b>	By default we create a model file version 4.1.5 this allows version to be set higher	Char *	Off
<b>override_colour, override_colour</b>	Override default colour map	Flag	Off
<b>use_default_colour</b>	Use default entity colours	Flag	Off
<b>use_default_ditto_colour</b>	Use default ditto colours	Flag	Off



<b>nurbs</b>	Create standalone curve and surface entities in NURBS form	Flag	Off
<b>create_edge_curves</b>	Create 3D edge curves for faces on volumes	Flag	Off
<b>Simplify &lt;tol&gt;</b>	Simplifies BREP to tolerance may also define the tolerance	Flag / Double	Off / 0.01
<b>vol_ids</b>	Display volume ids as default	Flag	Off
<b>skin_ids</b>	Switch display of skin ids off by default	Flag	On
<b>Face_ids, fac_ids</b>	Display face ids as default	Flag	Off
<b>face_colours</b>	Set colour on BREP faces	Flag	Off
<b>solid_colour</b>	Set colour on BREP solid	Flag	On
<b>solid_colour</b>			
<b>session &lt;file_name&gt;</b>	Create session file by creating an IUA procedure	Flag	Off
<b>export</b>	Create export file	Flag	Off
<b>export_header</b>	Defines file to use as export header	Char *	
<b>plane_ids</b>	Display standalone plane ids as default	Flag	Off
<b>plane_boundary</b>	Display standalone plane boundary	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Off
<b>surface_boundary</b>	Display surface boundary	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Off
<b>surface_boundary_pick</b>	Allow surface boundary pick	Flag	Off



<b>surface_isoparms</b>	Display surface lines	Choice OFF, SOLID, DOTTED, DASHED, DOT- DASH	Off
<b>surface_isoparms_nopick</b>	Disallow surface iso-parametric line pick	Flag	Off
<b>surface_isoparms_nu</b>	Number of surface iso-parametric lines in U	Int (0-99)	1
<b>surface_isoparms_nv</b>	Number of surface iso-parametric lines in V	Int (0-99)	1
<b>face_boundary</b>	Display face boundary	Choice OFF, SOLID, DOTTED, DASHED, DOT- DASH	Solid
<b>face_boundary_pick</b>	Allow face boundary pick	Flag	Off
<b>face_isoparms</b>	Display face lines	Choice OFF, SOLID, DOTTED, DASHED, DOT- DASH	Off
<b>face_isoparms_nopick</b>	Disallow face line pick	Flag	Off
<b>face_isoparms_nu</b>	Number of face lines in U	Int (0-99)	1
<b>face_isoparms_nv</b>	Number of face lines in V	Int (0-99)	1
<b>point_ids</b>	Display point ids	Flag	Off
<b>point_type</b>	Display 3D point symbol	Choice {DOT, "."}, {PLUS,"	DOT, .



		+"}, {CROSS, "X"}, {STAR, "*"},	
<b>line_type</b>	Display 3D line font	Choice SOLID, DOTTED, DASHED, DOT- DASH, PHANTO M	Solid
<b>Curve_type</b>	Display 3D curve font	Choice SOLID, DOTTED, DASHED, DOT- DASH, PHANTO M	Solid
<b>Thickness</b>	3D Wireframe default thickness	Int 1-60 – 0.1-6.0	2 – 0.2
<b>draw_point_type</b>	Display 2D point symbol	Choice {DOT, "."}, {PLUS," +"}, {CROSS, "X"}, {STAR, "*"},	DOT, .
<b>draw_line_type</b>	Display 2D line font	Choice SOLID, DOTTED, DASHED, DOT- DASH, PHANTO M, BREAK	Solid
<b>draw_curve_type</b>	Display 2D curve font	Choice SOLID,	Solid



		DOTTED, DASHED, DOT- DASH, PHANTO M, BREAK	
<b>draw_thickness</b>	2D Wireframe default thickness	Int 1-60 – 0.1-6.0	2 – 0.2
<b>catia_fonts</b>	Defined file for mapping of catia graphisms to site specific intereger	Char *	Off
<b>skins</b>	Create a skin for an opensolid > 1 face	Flag	On
<b>no_skins</b>	Dont create a skin for an opensolid > 1 face	Flag	Off
<b>mask_face_surfaces</b>	Hide surfaces subordinate to a face	Flag	On
<b>dont_mask_face_surfaces</b>	Dont hide surfaces subordinate to a face	Flag	Off
<b>shade_faces, shadefaces</b>	Switches current face respect of view visualization mode (i.e. Shades)	Flag	Off
<b>shade_vol, shadevol</b>	Switches current volume respect of view visualization mode (i.e. Shades)	Flag	On
<b>shade_skin, shadeskin</b>	Switches current skin respect of view visualization mode (i.e. Shades)	Flag	On
<b>shade_surf, shadesurf</b>	Switches current standalone surface respect of view visualization mode (i.e. Shades)	Flag	Off
<b>no_plane_create</b>	Dont convert 2x2 NURBS surface surporting a face to a plane	Flag	Off
<b>vol_edge</b>	Display volume internal edges	Choice OFF, SOLID, DOTTED, DASHED, DOT- DASH	Solid
<b>vol_edge_nopick</b>	Make volume edges unpickable	Flag	Off



<b>vol_isoparms</b>	Display volume lines	Choice SOLID, DOTTED, DASHED, DOT- DASH	Solid
<b>vol_isoparms_pick</b>	Make volume lines pickable	Flag	Off
<b>vol_isoparms_nu</b>	Number of volume lines in U	Int (0-99)	1
<b>vol_isoparms_nv</b>	Number of volume lines in V	Int (0-99)	1
<b>skin_boundary</b>	Display skin boundary edges	Choice OFF, SOLID, DOTTED, DASHED, DOT- DASH	Solid
<b>skin_boundary_pick</b>	Make skin boundary pickable	Flag	Off
<b>skin_edge</b>	Display skin internal edges	Choice OFF, SOLID, DOTTED, DASHED, DOT- DASH	Solid
<b>skin_edge_pick</b>	Make skin internal edge pickable	Flag	Off
<b>skin_isoparms</b>	Display skin lines	Choice SOLID, DOTTED, DASHED, DOT- DASH	Solid
<b>skin_isoparms_no_pick</b>	Make skin lines unpickable	Flag	Off
<b>skin_isoparms_nu</b>	Number of skin lines in U	Int (0-99)	1
<b>skin_isoparms_nv</b>	Number of skin lines in V	Int (0-99)	1



<b>ps_assy</b>	Create a CADDPS PS structure file pointing at CATIA V4 model files	Char *	Off
<b>tidy</b>	If session file or CADDPS PS file dont create empty CATIA model files	Flag	Off
<b>leaf_nodes</b>	If session file or CADDPS PS fil		
<b>pcurve_to_arc tol</b>	Convert PCURVES to ARCS with tolerance	Double	Off / 0.00001
<b>ident_curve_tol tol</b>	Set the identical curve tolerance to <tol>	Double	Off
<b>catia_fillet &lt;tol&gt;, cat_fillet</b>	UNKNOWN		
<b>trim_face_surfs</b>	Trim supporting face surfaces to face boundary	Flag	On
<b>no_trim_face_surfs</b>	Trim supporting face surfaces to face boundary	Flag	Off
<b>split_brep</b>	Split BREP	Flag	Off
<b>sew_check &lt;tol&gt;</b>	Check skin / solid can sew to tolerance	Double	Off / mdim
<b>use_tag</b>	Name entity from tag as opposed to *FACXX form	Flag	Off
<b>no_use_tag</b>	Name entity using *FACXX form	Flag	On
<b>nsmooth</b>	Redfine face edges for surfaces which original patch boundary may contain C1 discontinuity	Flag	On
<b>no_nsmooth</b>	Use original face edge definition for surfaces which original patch boundary may contain C1 discontinuity	Flag	Off
<b>no_routed</b>	Dont change routed items to BREP definition	Flag	Off
<b>no_routed_inner_radius</b>	Dont include inner radius in changing routed items to BREP definition	Flag	Off
<b>routed</b>	Change routed items to BREP definition	Flag	On



<b>no_conic_edges</b>	Dont write conic solid edge curves as conies	Flag	Off
<b>conic_centers &lt;layer&gt;</b>	Create conics edge centers on layer specified else layer 254	Flag / Int	Off / 254
<b>gvp</b>	Produce a validation properties file which can be checked using a API program	Flag	Off
<b>no_facet_colour</b>	Dont change individual facet colours	Flag	Off
<b>draw_symbols / no_draw_symbols"</b>	Produce dimensions and multiline texts as draw symbols	Flag	Off
<b>draw_symbol_text s</b>	Produce simple texts as draw symbols	Flag	Off
<b>draw_layer &lt;layer&gt;</b>	Move all draw entities to layer specified (default 9)	Flag / Int	Off / 9
<b>draw_lines</b>	Process view dependent edits on 3D lines	Flag	Off
<b>draw_2d_lines</b>	Process view dependent edits on 2D & 3D lines	Flag	Off
<b>draw_nfigs</b>	Produces CADs NFIGS as draw symbols	Flag	Off
<b>draw_wr_0_360</b>	Produce text with 0 360 writing rule	Flag	Off
<b>draw_wr_90_90</b>	Produce text with -90 +90 writing rule	Flag	On
<b>draw_wr_geo</b>	Produce text with geo writing rule	Flag	Off
<b>ignore_model_text</b>			
<b>bae_options &lt;catia_font_file &gt;</b>	Enables followings options draw_nfigs, draw_wr_geo, draw_2d_lines, draw_layer, draw_symbols , override_colour, draft. catia_fonts <catia_font_file>	Flag	Off
<b>noshow_ents</b>	Allows a file to be defined similar to a mask file which allows entities to be created in NOSHOW	Char *	Off

