User Guide

CATIA - NX

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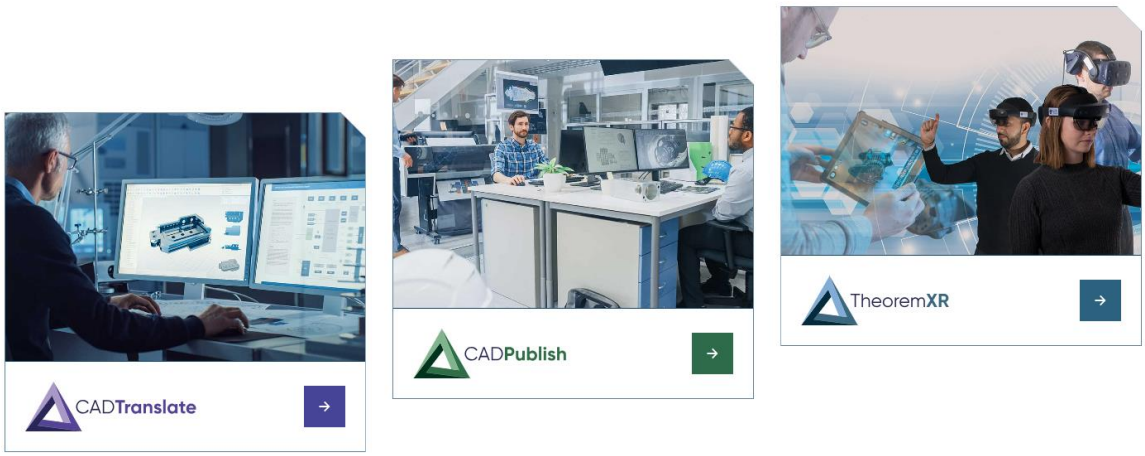
Overview of TRANSLATE

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:

CADTranslate

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

See our website for more detail.

CADPublish

The creation of documents enriched with 3D content

See our website for more detail.

TheoremXR

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 & 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

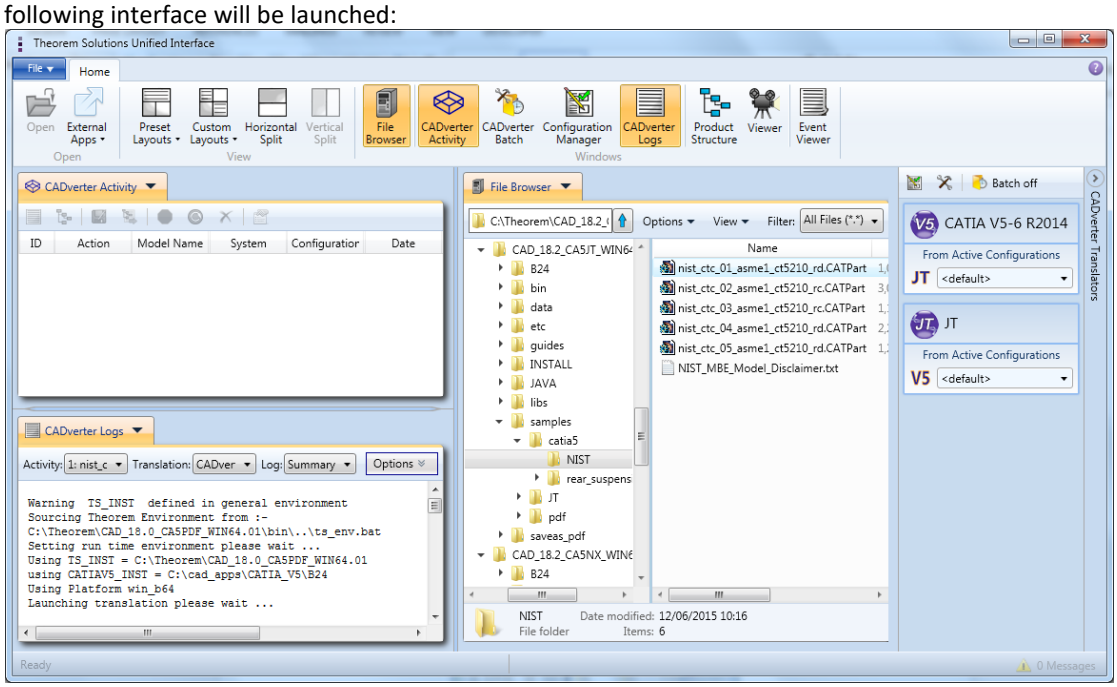
Using the Product

Default Translation – via the Unified Interface

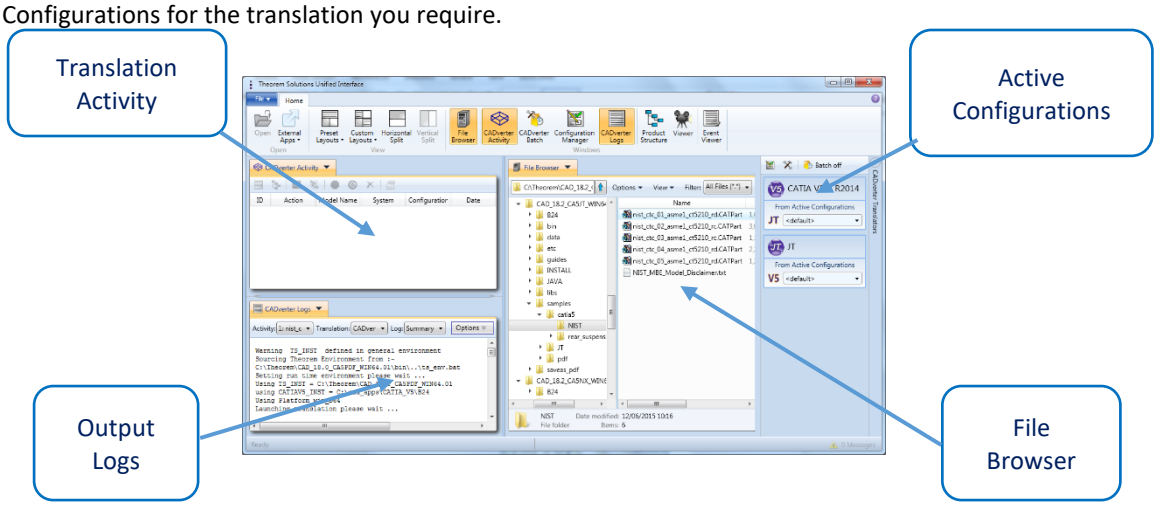
The Unified Interface can be started via the Start Menu – if a shortcut was added during installation.

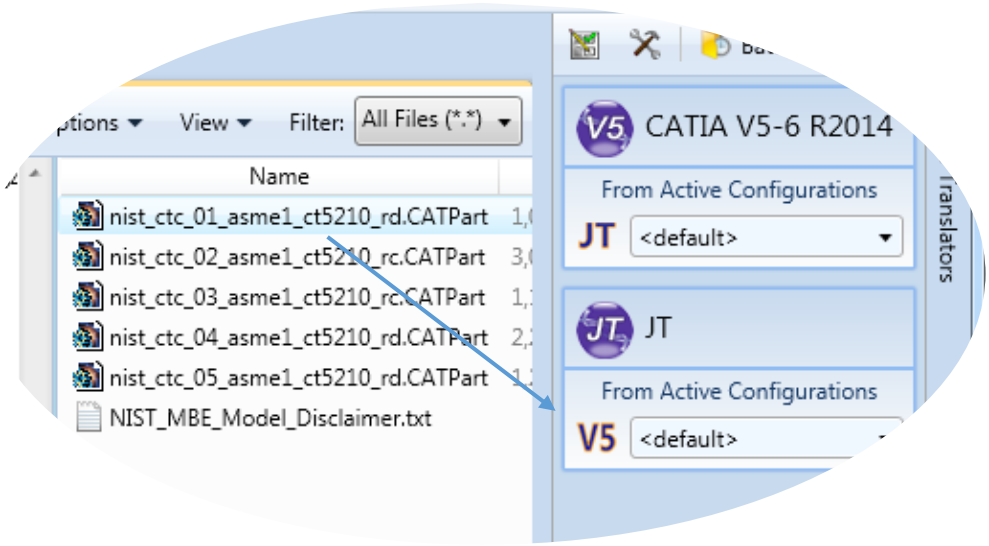
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:
<UI_installation_directory>\bin\Unified_Interface.cmd

The following interface will be launched:



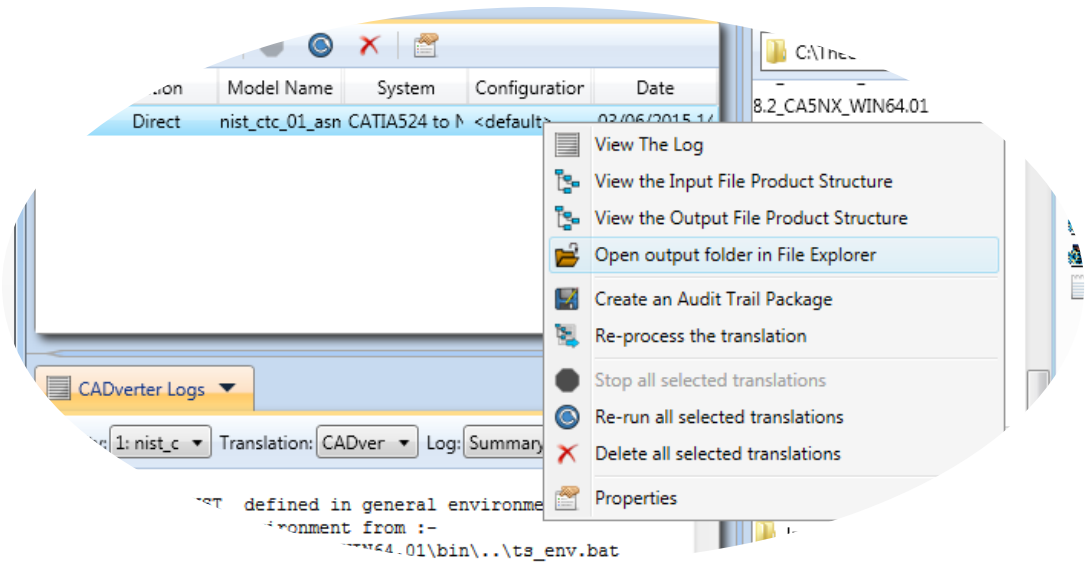
The default layout is split into 4 primary areas, which can be altered to the users prefer:
The simplest way to translate from CATIA V5 or JT is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.





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.

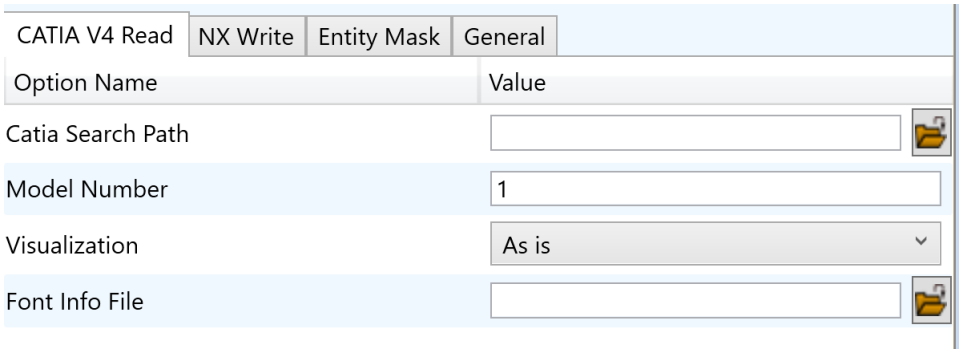
The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:



Common Options for CATIA to NX

CATIA Read Options

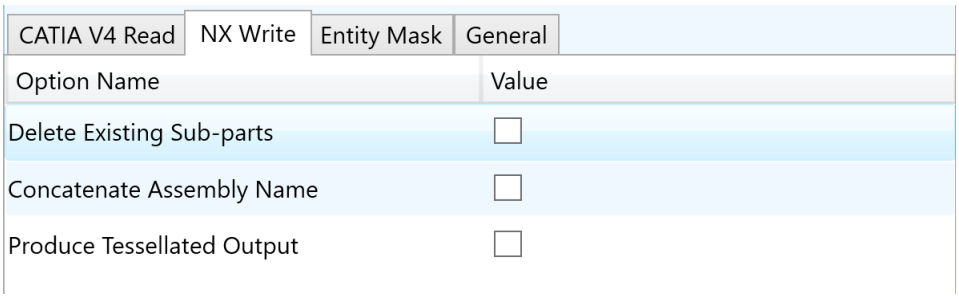
The image below shows the default CATIA Read options and their settings



Option Name	Description
Catia Search Path	Search path used to file model files whilst reading asm or session files
Model Number	Model number in export file
Visualization	Colour visualization mode as per set, layer, entity type or view
Font Info File	Use the specified file to obtain Font Information

NX Write Options

The image below shows the default NX Write options and their settings

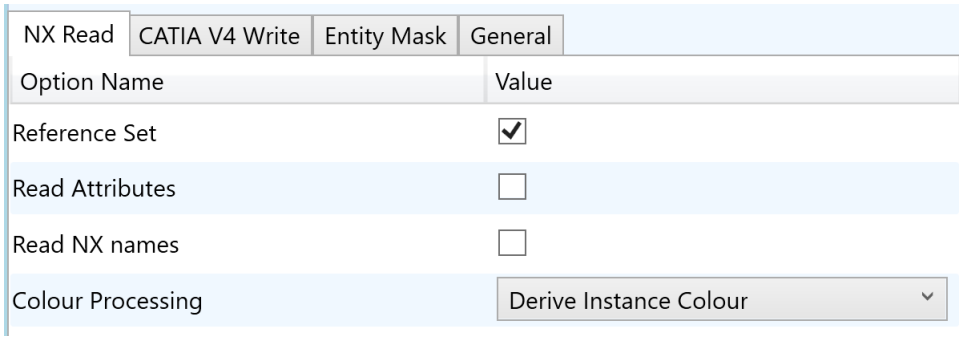


Option Name	Description
Delete Existing Sub-parts	Delete existing sub-parts
Concatenate Assembly Name	Concatenate Assembly Name
Produce Tessellated Output	Produce Tessellated NX file

Common Options for NX to CATIA

NX Read Options

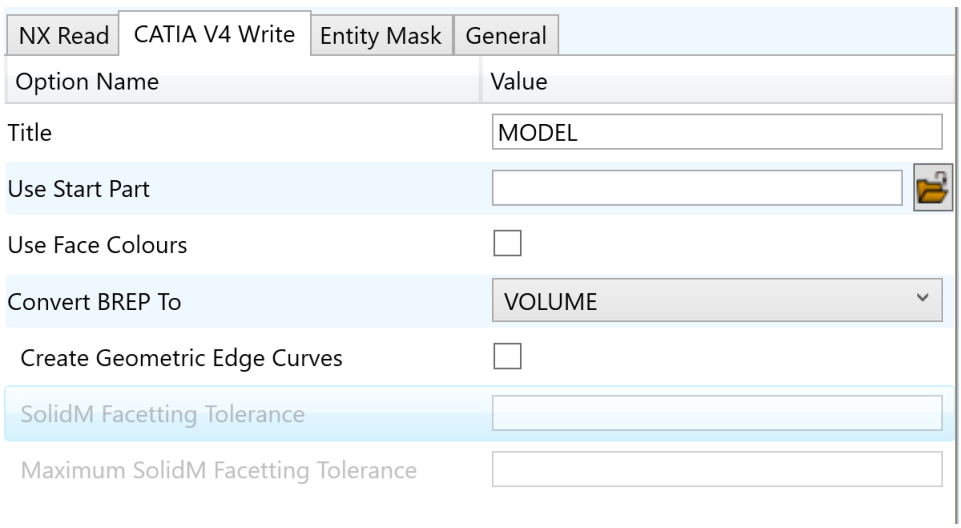
The image below shows the default NX Read options and their settings



Option Name	Description
Reference Set	Enabled reference set processing
Read Attributes	Read NX detail user attributes
Read NX Names	Read NX entity names, if they exist
Colour Processing	How to process colour

CATIA Write Options

The image below shows the default CATIA write options and their settings



Option Name	Description
Reference Set	Enabled reference set processing
Read Attributes	Read NX detail user attributes
Read NX Names	Read NX entity names, if they exist
Colour Processing	How to process colour

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 CATIA V4 you are using. E.g. for CATIA V4

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

CMD LINE Option	Purpose	Data Type	Default
model	Model number in export file	Int	1
mvs	Mainframe real conversion	Flag	Off
cnode	Read Entity Tag Names	Flag	Off
ppoint	Read Entity Tag Names	Flag	Off
prop	Unknown	Flag	Off
cont	Continuity required	Tolerance	Off
prim_surf	Primitive surface redefinition required	Flag	Off
offditto	Explode assembly structure during read	Flag	Off
cvm <set layer etype view>	Colour visualization mode as per set, layer, entity type or view	Choice set layer etype view	Off
dim_view_realm	Dimensions are view realm	Flag	Off
dim_draft_realm	Dimensions are draft realm	Flag	On
set_read	Read associated set name and store in attribute	Flag	Off
heal_degen_surf	Heal partially degenerate surfaces	Flag	Off
show_info	Displays information panel in progress file	Flag	On
read_set <set_name>	Read entities in named sets	Char *	All
read_tag <entity_name>	Read named entities	Char *	All
surf_check_file <file_name>	Creates a surf check file	Char *	
catia_fonts <file_name>	Catia graphism font mapping file	Char *	
search_path <search_path>	Search path used to file model files whilst reading asm or session files	Char *	
only_use_search_path	only_use_search_path	Flag	Off
face_colours	Read colours as applied to solid faces	Flag	Off
solid_colours	Read colour applied to solid	Flag	On
pdegen <tol>	Process pdegen surfs	Flag / Tol	Off
group_pipes	Group pipes in a detail	Flag	Off
group_pipelines	Group pipelines in a detail	Flag	Off
ungroup_pipelines	Dont group pipelines in a detail	Flag	Off
use_axs_txt_name	Names axis as per associated text	Flag	Off
use_axs_name	Names axis as per tag name i.e. *AXS1	Flag	On

pipeline_layer	Specify layer on which to put pipe line	Int 1-254	Use pipe segment layer
facet_colour	Read facet colours for solidm	Flag	Off
no_facet_colour	Dont read facet colours for solidm	Flag	On
gvp	Read validation properties	Flag	Off
gvp_dont_subtract	Calculate validation properties on individual volumns	Flag	Off
gvp_absolute	Calculate absolute validation properties		
gvp_noshow	Calculate validation properties if hidden		
gvp_repfile <file_name>	validation properties report file name	Char *	
gvp_outfile <file_name>	validation properties output file name	Char *	
noshow	Read noshowed entities	Flag	Off
draft	Read 2D draftings enitities	Flag	Off

NX Write Arguments

CMD LINE Option	Purpose	Data Type	Default
poly_sol/no_poly_sol	For gco Fsolids produce Facetted bodies (else attempt brep)	Flag	off
heal_ug <tol> [def tol = 0.0095/units]	attempt a UG heal on the created body (if nocheck on)	Flag	off
keep_all_bodies/no_keep_all_bodies	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)
nocheck	Don't check created Parasolid geometric entities	Flag	off
no_brep_prep	Prepare solids switched off	Flag	on (surfs read as nurbs+prep)
pstolmodel <num>/nopstolmodel [def num = 3]	Enable Parasolid tolerant modeling	Flag	on
pssew <tol>/nosew	Sew failed breps and opensols	Flag	on
csg_prep <tol> [def tol = 0.000001*scale]	Prepare CSG Prinitives	Flag	off
csg_shift <tol> [def tol = 0.000001*scale]	Change CSG Shift Distance	Flag	off
csgfix	Fix CSG Primitives	Flag	off
ps_fix_small/no_fix_ps_small	Remove small edges, sliver and spike faces in breps	Flag	off
ps_fix_osol/no_ps_fix_osol	Remove small edges, sliver and spike faces in opensolids	Flag	off

NX to CATIA V4 Arguments List

NX Read Arguments List

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

CATIA Write Arguments List

CMD LINE Option	Purpose	Data Type	Default
mvs	Create a mainframe real conversion model file	Flag	Off
catia_v3, catiav3, v3	Produce a Catia V3 file	Flag	Off
surfopt l <tol>	Controls surface optimization tolerance	Double	Off / 0.001
maxtol <tol>, max_tol	Maximum solid faceting tolerance to be applied to solids which are found to be too large to go into CATIA	Double	Off
model_dimension l <tol> <units>, modeldimension, mdim	Catia Model Dimension. Followed by number, followed by blank or "inch" or "mm" specfies model dimension to be number of part units, inch or mm.	Double	Off
startpart, start_part, start-part	Name of model to be used as seed part to provide site specific information to the created CATIA model file	Char *	Off

startpartprefix	Prefix of Startpart file to use allows automatic switching dependent on units	Char *	Filename
solide	Create import solides rather than volumes	Flag	Off
solidv	Create complex volume solide rather than volume (doubles size of model file)	Flag	Off
solidm	Creates faceted solidm	Flag	Off
version	By default we create a model file version 4.1.5 this allows version to be set higher	Char *	Off
override_colour, overide_colour	Override default colour map	Flag	Off
use_default_colour	Use default entity colours	Flag	Off
use_default_ditto_colour	Use default ditto colours	Flag	Off
nurbs	Create standalone curve and surface entities in NURBS form	Flag	Off
create_edge_curves	Create 3D edge curves for faces on volumes	Flag	Off
Simplify <tol>	Simplifies BREP to tolerance may also define the tolerance	Flag / Double	Off / 0.01
vol_ids	Display volume ids as default	Flag	Off
skin_ids	Switch display of skin ids off by default	Flag	On
Face_ids, fac_ids	Display face ids as default	Flag	Off
face_colours	Set colour on BREP faces	Flag	Off
solid_colour	Set colour on BREP solid	Flag	On
solid_colour			
session <file_name>	Create session file by creating an IUA procedure	Flag	Off
export	Create export file	Flag	Off
export_header	Defines file to use as export header	Char *	
plane_ids	Display standalone plane ids as default	Flag	Off
plane_boundary	Display standalone plane boundary	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Off
surface_boundary	Display surface boundary	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Off
surface_boundary_pick	Allow surface boundary pick	Flag	Off
surface_isoparms	Display surface lines	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Off
surface_isoparms_nopick	Disallow surface iso-parametric line pick	Flag	Off
surface_isoparms_nu	Number of surface iso-parametric lines in U	Int (0-99)	1
surface_isoparms_nv	Number of surface iso-parametric lines in V	Int (0-99)	1

face_boundary	Display face boundary	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Solid
face_boundary_pick	Allow face boundary pick	Flag	Off
face_isoparms	Display face lines	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Off
face_isoparms_nopick	Disallow face line pick	Flag	Off
face_isoparms_nu	Number of face lines in U	Int (0-99)	1
face_isoparms_nv	Number of face lines in V	Int (0-99)	1
point_ids	Display point ids	Flag	Off
point_type	Display 3D point symbol	Choice {DOT, “.”}, {PLUS,” +”}, {CROSS, “x”}, {STAR, “*”},	DOT, .
line_type	Display 3D line font	Choice SOLID, DOTTED, DASHED, DOT-DASH, PHANTOM	Solid
Curve_type	Display 3D curve font	Choice SOLID, DOTTED, DASHED, DOT-DASH, PHANTOM	Solid
Thickness	3D Wireframe default thickness	Int 1-60 – 0.1-6.0	2 – 0.2
draw_point_type	Display 2D point symbol	Choice {DOT, “.”}, {PLUS,” +”}, {CROSS, “x”}, {STAR, “*”},	DOT, .
draw_line_type	Display 2D line font	Choice SOLID, DOTTED, DASHED, DOT-DASH, PHANTOM, BREAK	Solid
draw_curve_type	Display 2D curve font	Choice SOLID, DOTTED, DASHED, DOT-DASH, PHANTOM, BREAK	Solid
draw_thickness	2D Wireframe default thickness	Int 1-60 – 0.1-6.0	2 – 0.2

catia_fonts	Defined file for mapping of catia graphisms to site specific intereger	Char *	Off
skins	Create a skin for an opensolid > 1 face	Flag	On
no_skins	Dont create a skin for an opensolid > 1 face	Flag	Off
mask_face_surfaces	Hide surfaces subordinate to a face	Flag	On
dont_mask_face_surfaces	Dont hide surfaces subordinate to a face	Flag	Off
shade_faces, shadefaces	Switches current face respect of view visualization mode (i.e. Shades)	Flag	Off
shade_vol, shadevol	Switches current volume respect of view visualization mode (i.e. Shades)	Flag	On
shade_skin, shadeskin	Switches current skin respect of view visualization mode (i.e. Shades)	Flag	On
shade_surf, shadesurf	Switches current standalone surface respect of view visualization mode (i.e. Shades)	Flag	Off
no_plane_create	Dont convert 2x2 NURBS surface surporting a face to a plane	Flag	Off
vol_edge	Display volume internal edges	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Solid
vol_edge_nopick	Make volume edges unpickable	Flag	Off
vol_isoparms	Display volume lines	Choice SOLID, DOTTED, DASHED, DOT-DASH	Solid
vol_isoparms_pick	Make volume lines pickable	Flag	Off
vol_isoparms_nu	Number of volume lines in U	Int (0-99)	1
vol_isoparms_nv	Number of volume lines in V	Int (0-99)	1
skin_boundary	Display skin boundary edges	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Solid
skin_boundary_pick	Make skin boundary pickable	Flag	Off
skin_edge	Display skin internal edges	Choice OFF, SOLID, DOTTED, DASHED, DOT-DASH	Solid
skin_edge_pick	Make skin internal edge pickable	Flag	Off
skin_isoparms	Display skin lines	Choice SOLID, DOTTED, DASHED, DOT-DASH	Solid
skin_isoparms_nopick	Make skin lines unpickable	Flag	Off
skin_isoparms_nu	Number of skin lines in U	Int (0-99)	1

skin_isoparms_nv	Number of skin lines in V	Int (0-99)	1
ps_assy	Create a CADDS PS structure file pointing at CATIA V4 model files	Char *	Off
tidy	If session file or CADDS PS file dont create empry CATIA model files	Flag	Off
leaf_nodes	If session file or CADDS PS fil		
pcurve_to_arc tol	Convert PCURVES to ARCS with tolerance	Double	Off / 0.00001
ident_curve_tol tol	Set the identical curve tolerance to <tol>	Double	Off
catia_fillet <tol>, cat_fillet	UNKNOWN		
trim_face_surfs	Trim supporting face surfaces to face boundary	Flag	On
no_trim_face_surfs	Trim supporting face surfaces to face boundary	Flag	Off
split_brep	Split BREP	Flag	Off
sew_check <tol>	Check skin / solid can sew to tolerance	Double	Off / mdim
use_tag	Name entity from tag as opposed to *FACXX form	Flag	Off
no_use_tag	Name entity using *FACXX form	Flag	On
nsmooth	Redfine face edges for surfaces which original patch boundary may contain C1 discontinuity	Flag	On
no_nsmooth	Use original face edge definition for surfaces which original patch boundary may contain C1 discontinuity	Flag	Off
no_routed	Dont change routed items to BREP definition	Flag	Off
no_routed_inner_radius	Dont include inner radius in changing routed items to BREP definition	Flag	Off
routed	Change routed items to BREP definition	Flag	On
no_conic_edges	Dont write conic solid edge curves as conies	Flag	Off
conic_centers <layer>	Create conics edge centers on layer specified else layer 254	Flag / Int	Off / 254
gvp	Produce a validation properties file which can be checked using a API program	Flag	Off
no_facet_colour	Dont change individual facet colours	Flag	Off
draw_symbols / no_draw_symbols"	Produce dimensions and multiline texts as draw symbols	Flag	Off
draw_symbol_texts	Produce simple texts as draw symbols	Flag	Off
draw_layer <layer>	Move all draw entities to layer specified (default 9)	Flag / Int	Off / 9
draw_lines	Process view dependent edits on 3D lines	Flag	Off
draw_2d_lines	Process view dependent edits on 2D & 3D lines	Flag	Off
draw_nfigs	Produces CADS NFIGS as draw symbols	Flag	Off

draw_wr_0_360	Produce text with 0 360 writing rule	Flag	Off
draw_wr_90_90	Produce text with -90 +90 writing rule	Flag	On
draw_wr_geo	Produce text with geo writing rule	Flag	Off
bae_options <catia_font_file>	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
noshow_ents	Allows a file to be defined similar to a mask file which allows entities to be created in NOSHOW	Char *	Off

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