



CADverter for CATIA V5 – CATIA ICEM Surf

Product Release Version 25.1



USER GUIDE

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Overview of CADverter

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 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 ICEM Surf Bi-directional CATIA V5 Translator

The Translator may be installed on a number of machines each accessing a central networkfloating license.

Theorem's CADverter product for CATIA ICEM Surf to CATIA V5 is a direct database converter between Dassault Systemes CATIA ICEM Surf and CATIA V5. It enables the user to convert all forms of mechanical design geometry, as well as assembly and attribute information, between these two systems.

The CATIA ICEM Surf - CATIA V5 CADverter product is a bi-directional product. It can be used interactively, from the command line or in a batch mode, from a standard GUI Interface, offering combined viewing, data filtering and translation capabilities.

The CADverter directly accesses native CATIA ICEM Surf files using the Dassault Systemes supported programming interface. Structure details and geometry colour information is retained during translation.

The relevant CAD products will be referred to as Surf and V5 throughout the rest of this document.

Primary Product Features

- Converts all types of wire frame, surfaces, trimmed surfaces (faces)
- Converts structure between the systems.
- Converts attribute data including colour and layer information
- The conversion process can be run Interactively or in Batch mode
- Data can be filtered by layer and entity type
- Geometry can be filtered and selectively processed

Primary Product benefits?

- Direct conversion between Surf and V5 reduces processing time, simplifies integration and retains accuracy of the model
- The integrated viewing capability enables visually verification, pre and post translation
- The integrated data filtering options allows selected data ONLY to be processed, enabling optimisation of translations and time savings
- By converting all forms of geometry no data is lost, eliminating the time required to recreate missing data

- With over 20 years industrial use Theorem's product robustness and quality is well proven, reducing your business risk

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

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 msi.

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

Installation

The installation is run from the MicroSoft Installer package(s) provided.

Currently, there are 2 distinct installation stages that are required.



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



In addition, the Theorem Unified Interface will also need to be installed. The installation process is the same as for the Translator. 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.

All specific Surf environment configuration details are documented in [Appendix A](#) of this document.

There are 3 distinct ways of running the translator:

- Via the Theorem Unified Interface



- The Unified Interface offers a Desktop Environment that allows CAD and Visualization data to be viewed pre and post translation.
- For a full guide to this process, please see our 'How to Translate Using the Surf – V5 CADverter via the User Interface' demonstration video located [here](#).

- 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 Surf – V5 CADverter via the Command Line' demonstration video located [here](#).

- Interactively from within Surf



- The Interactive Interface provides a direct method of Translating Surf data to V5 from within Surf itself.
- For a full guide to this process, please see our 'How to Translate Using the Surf – V5 CADverter Interactive Mode' demonstration video located [here](#).

Using the Product

Default Translations

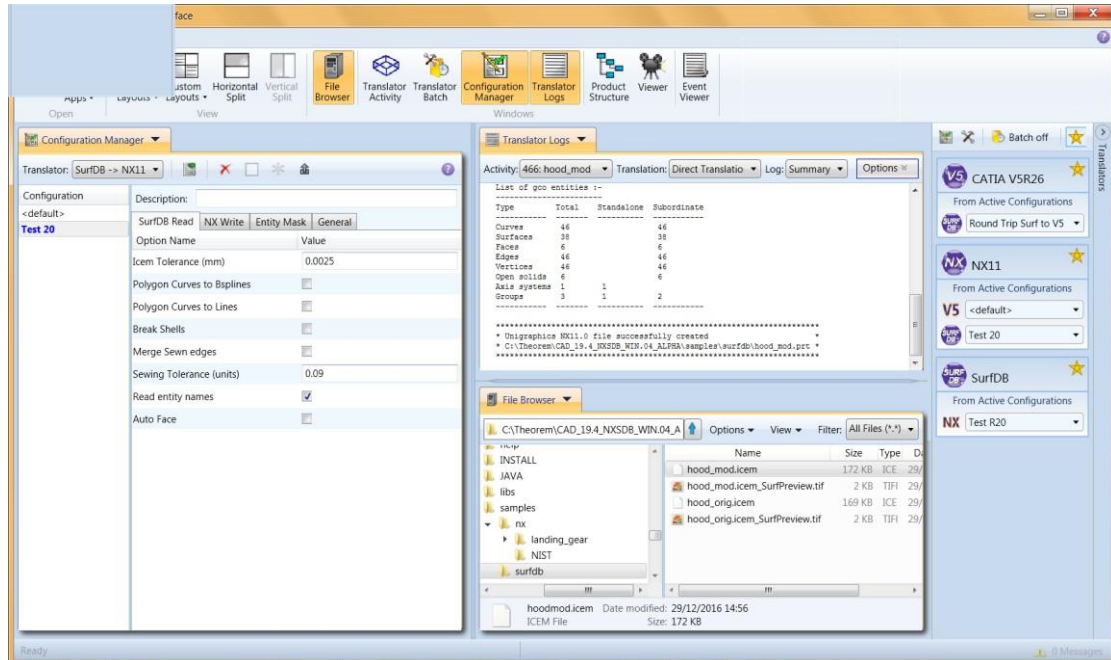
Default Translation – via the Unified Interface

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

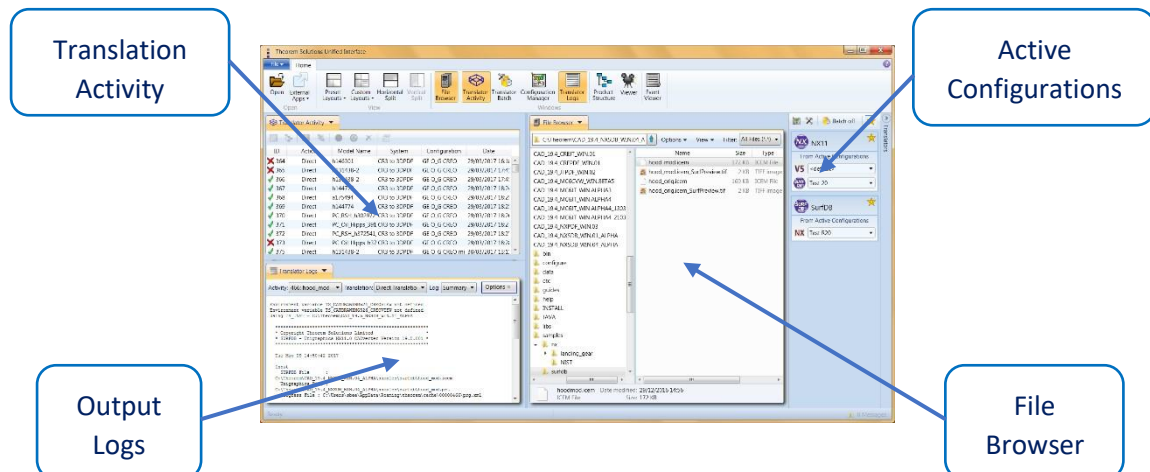
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

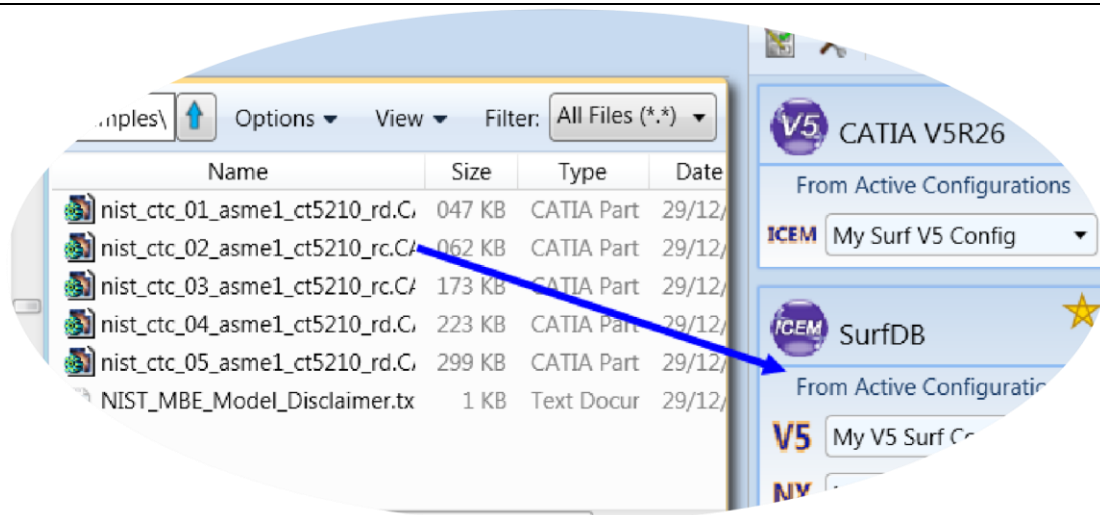
The following interface will be launched:



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

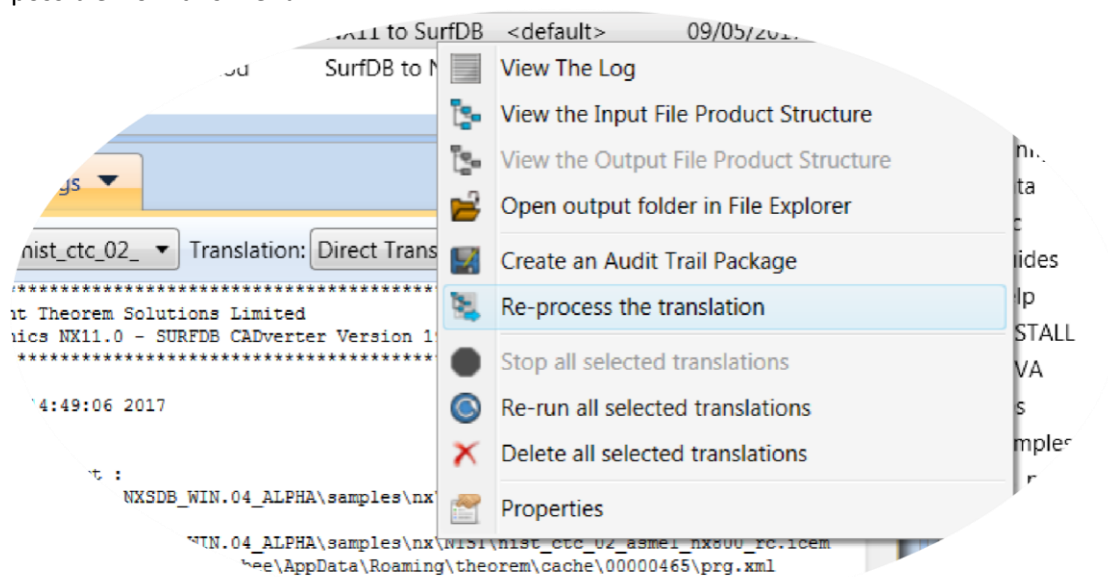


The simplest way to translate from Surf or V5 is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



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. Other tasks such as Re-processing the translation are also possible from this menu:



Default Translation – via the Command Line

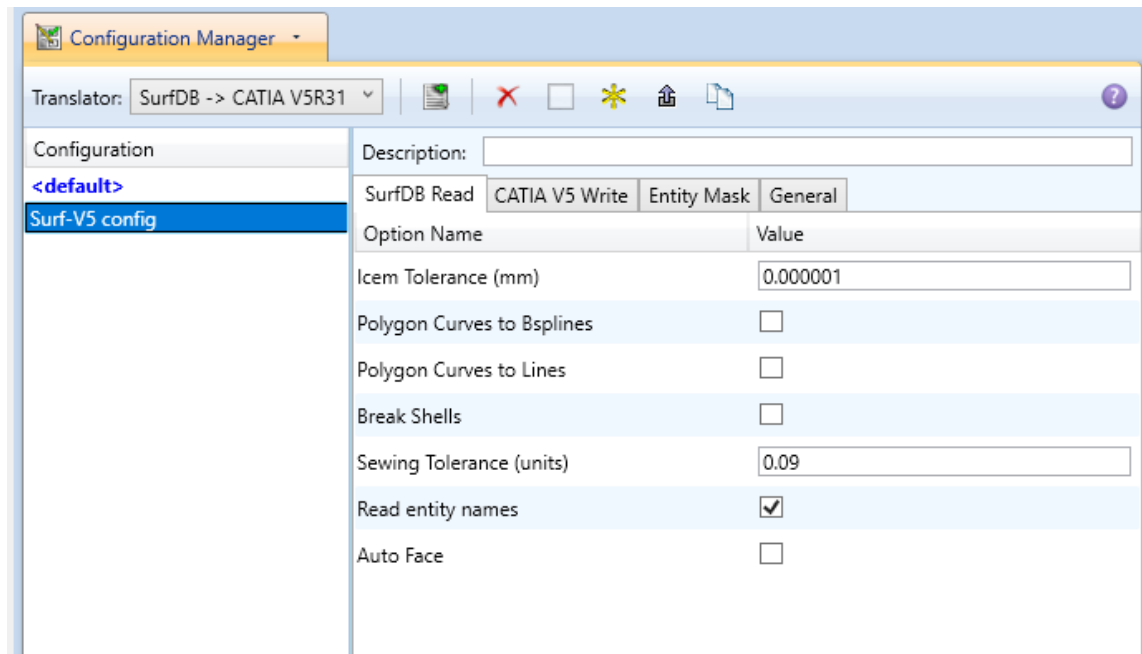
Running a translation via the command line can be carried out via the **cad_run_UI.cmd** file located in the **<installation_directory>\bin** directory. The format of the command is as follows when translating from Surf to V5:

```
<Translator_installation_directory>\bin\cad_run_UI.cmd SurfDB_CATIA5[XX] -i
<input_file> -o <output_file> -c "<configuration name>"
```

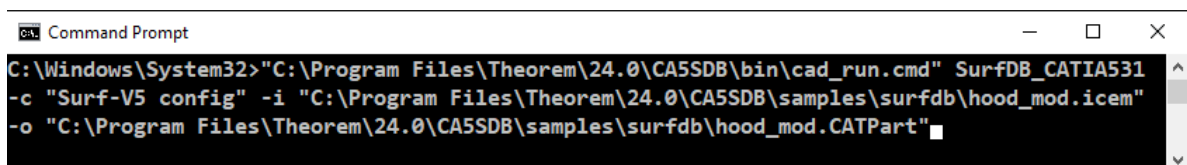
The format of the command is as follows when translating from V5 to Surf:

```
<Translator_installation_directory>\bin\cad_run_UI.cmd SurfDB_CATIA5[XX] -i
<input_file> -o <output_file> -c "<configuration name>"
```

Where **<configuration name>** is the name of a Configuration e.g. **"My SURF V5 Config"** (see below)



(Note! Replace the [XX] seen in the example with the version of V5 that you are using. E.g. for V5R31, change to CATIA531):



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

```

Administrator: Command Prompt

*****
* Copyright Theorem Solutions Limited *
* SURFDB - CATIA5 CADverter Version 20.0.002 *
*****

Wed Jun 07 11:49:36 2017

Input
SURFDB File : C:\Program Files\Theorem 20.0\samples\surfdb\hood_mod.ice
CATIA5 Part : C:\Program Files\Theorem 20.0\samples\surfdb\hood_mod.CATPart
Progress File : C:\Program Files\Theorem 20.0\samples\surfdb\hood_mod.proguy.log

CATIA U5 Library Ver. : 20.0.002
CATIA U5 Build Release: B26.SP0 64

List of SURFDB entities read:-
-----
Entity Type      Total    Standalone  Subordinate  Failed
-----
Groups           3         3           0            0
2D Splines       46         0          46            0
Spline Surfaces  38        32           6            0
Faces            6         6           0            0
Loops            6         0           6            0
Diredges        46         0          46            0
Work Planes      1         1           0            0
-----

CATIA U5 Cache Mode : Disabled
*****
* CATIA5 file successfully created *
* C:\Program Files\Theorem 20.0\samples\surfdb\hood_mod.CATPart *
*****

```

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

C:\Program Files\Theorem 20.0\samples\surfdb\NIST\hood_mod.CATPart

CADverter Customization

CADverter 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.

General Notes for Surf to V5

1. The SURFDB Ellipse entity holds both the Arc and Ellipse definitions. The appropriate CATIA entity will be created.
2. Conversion of the Polygon Curve is dependant upon whether the 'mc_to_bsp' and 'mc_to_lin' options have been used. If none of these options are used then Polygon Curves are ignored. They can both be used together if required.
3. Standalone Spline Surfaces are always faced as CATIA V5 always has topology on geometry.
4. Shells will be written as separate Faces. If the 'break_shells' option is used or the Shell sewing process fails, the shell will be formed by a Surface Join.

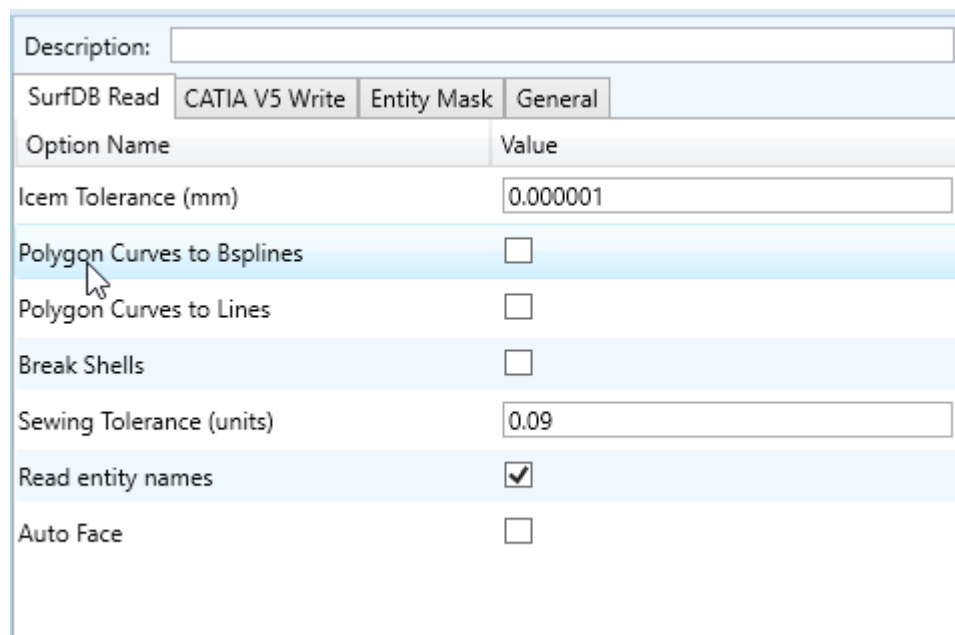
Common Options for Surf to V5

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing Surf data into V5 are grouped into 4 areas:

- Surf Read – Those arguments that affect how data is read from Surf
- V5 Write – Those arguments that affect how the data is written to V5
- Entity Mask – Those arguments that allow specific read entities to be masked
- General – Those arguments that are common to ALL Publishing activities regardless of source data

Surf Read Arguments

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



Option Name	Value
Icem Tolerance (mm)	0.000001
Polygon Curves to Bsplines	<input type="checkbox"/>
Polygon Curves to Lines	<input type="checkbox"/>
Break Shells	<input type="checkbox"/>
Sewing Tolerance (units)	0.09
Read entity names	<input checked="" type="checkbox"/>
Auto Face	<input type="checkbox"/>


Each of these options is described below:

Option	Description
Icem Tolerance	<p>This tolerance is used in the translation when checking Face base surfaces for degeneracy and when fitting 3D edge curves. This can be altered with this option if required (<i>Default is 0.000001m</i>) ○ Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <code>icem_tol0.000001</code>
Polygon Curves to B-splines	<p>Converts polygon curves to b-splines. (<i>Default is OFF</i>). ○ Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <code>mc_to_bsp – to turn on</code>

Polygon Curves to Lines	<p>Converts polygon curves to lines. <i>(Default is OFF)</i>. ○</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <code>mc_to_lin</code> – to turn on
Break Shells	<p>Any Shells encountered in the translation will be converted to UG Trimmed Surfaces (Solids). If this option is used then Shells will be broken into separate Faces. <i>(Default is OFF)</i>.</p> <p>○ Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <code>break_shells</code> – to turn on
Sewing Tolerance	<p>When a Shell is read it has to be sewn together (unless the break_shells option has been used). <i>(Default is 0.09/units)</i>. ○ Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <code>sew_tol<tolerance></code> – to change tolerance option.
Don't Read Names	<p>By default entity names are read from V5 <i>(Default is Off)</i> ○</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <code>no_read_names</code> – to turn on
Auto Face	<p>This option specifies that standalone Bezier patches should be automatically faced. <i>(Default is Off)</i> ○ Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <code>auto_face</code> – to turn on

V5 Write Arguments

The image below shows the Write V5 arguments that are available, with their default settings:

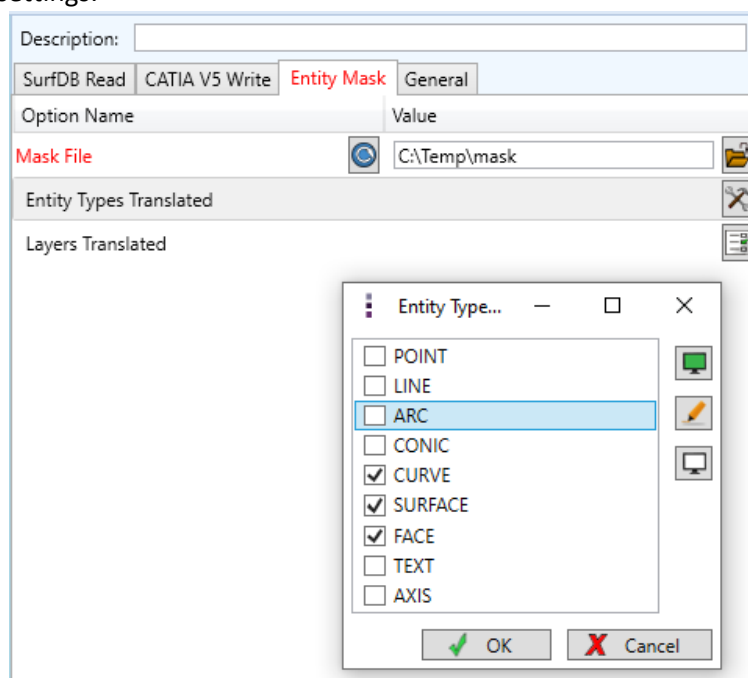
Description: <input type="text"/>	
SurfDB Read	CATIA V5 Write
Entity Mask	General
Option Name	Value
Output Geometry File Type	CATPart
Write Face Colours	<input type="checkbox"/>
Show Reference Planes	<input type="checkbox"/>
Retain Assembly Structure	<input checked="" type="checkbox"/>
Property Mapping File	<input type="text"/> 

Each of these options is described below:

Option	Description
Output Geometry Type	<p>Allows the user to define the file format to be created. Options are; CATPart, model, cgr, igs, CATShape, Tesselated. Default is CATPart</p> <ul style="list-style-type: none"> ○ Command Line Syntax <i>the corresponding settings in order are</i> <ul style="list-style-type: none"> ▪ <i>output_type model</i> ▪ <i>output_type cgr</i> ▪ <i>output_type igs</i> ▪ <i>output_type CATShape</i> <ul style="list-style-type: none"> ▪ <i>create_cgr</i>
Write Face Colours	<p>Reads face colour but this can lead to a performance hit. The alternative option is apply the colour from the solid. Default is on</p> <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>SOLID_COLOUR</i> to toggle from Face Colours
Show Reference Planes	<p>Shows planes. Default is Off.</p> <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>plank_planes</i> ▪ <i>dont_blank_planes</i> – to turn on

Surf to V5 Entity Masking Arguments

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

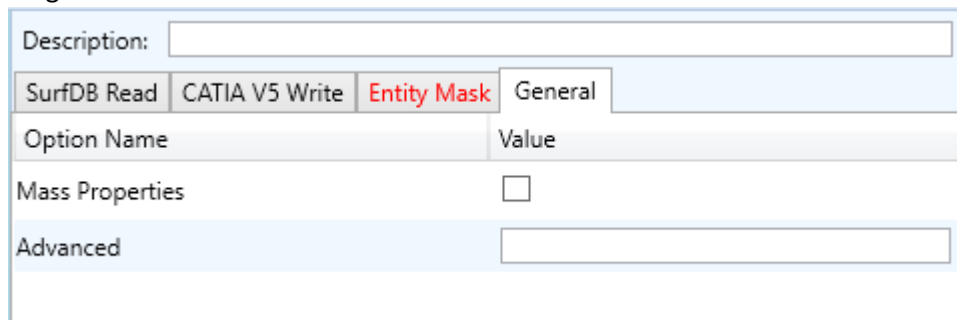


Each of these options is described below:

Option	Description
Mask File	<p>Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: ○ Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <i>Mask <filename></i>
Entity Types Translated	<p>Specifies a selection list from which to select which entity types are to be processed. <i>The following types are available: "POI", "LIN", "ARC", "CON", "CUR", "ISO", "SUR", "FAC", "TEX", "AXI"</i> ○</p> <p>Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <i>Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g.: ON POI to ensure they are considered in the translation</i>
Layers Translated	<p>Specifies a selection list from which to select which layers are to be processed. ○ Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <i>A single entry of ON ALL LAY Must precede any Layer Mask command.</i> ▪ <i>Add a list or range of numbers representing layer to be processed to the specified mask file to ensure they are NOT considered in the translation e.g.: OFF LAY 114,149,166,167,168</i>

Surf to V5 General Arguments

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



Option Name	Value
Mass Properties	<input checked="" type="checkbox"/>
Advanced	

Each of these options is described below:

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

There are a number of arguments that may be provided to the CADverter. They can be supplied in any order. Those marked in red italics are on by default when running from within the ICEM SURFDB GUI. These options are chosen to maintain the data quality with respect to surface and curve parameterization etc. rather than ensure greatest success on conversion.

Option	Description
Advanced	Allows any of the Command Line Advanced arguments documented below to be passed to the Unified Interface invocation
<i>icem_tol <val></i>	This tolerance is used in the translation when checking Face base surfaces for degeneracy and when fitting 3D edge curves. The default icem_tol is 0.0025 mm or 0.0001 inch. If required, this can be altered with this option if required.
mc_to_bsp	This specifies that Polygon Curves read from the SURFDB file should be converted to linear B-Splines. Both mc_to_bsp and mc_to_lin can be used in a translation. If neither of these options are used, then NO Raw Data is converted. Only Polygon Curves of type PolygonTypePoint3d and PolygonTypePoint3dPoint3d are supported.
<i>mc_to_lin</i>	This option specifies that Polygon Curves read from the SURFDB file should be converted to Line entities. Both of mc_to_bsp and mc_to_lin can be used in a translation. If neither of these options are used, then NO Raw Data is converted. Only curves of type PolygonTypePoint3d and PolygonTypePoint3dPoint3d are supported.

break_shells	Specifies that any Shells encountered will be broken into separate Faces in the translation.
<i>sew_skins</i> [tol]	When a Shell is read its subordinate faces have to be sewn together. This is achieved by the join operator which checks if all entities are within the defined tolerance. The tolerance by default is 0.001 mm. If the check fails the join are not successful.
skins_angle [ang]	When the join is created the angle between connecting edges can be checked to see if they are within a given angle. By default the angle is 0.5 degrees. If the check fails the join are not successful.
skins_tangency	When the join is created the tangency between adjacent edges can be checked. If the check fails the join are not successful.
skins_connexity	When the join is created the connexicity between connecting edges can be checked. If the check fails the join are not successful.
<i>sew_ccrvs</i> [tol]	When a composite curve is read its subordinate curves have to be sewn together. This is achieved by the join operator which checks if all entities are within the defined tolerance. The tolerance by default is 0.001 mm. If the check fails the join are not successful.
ccrvs_angle [ang]	When the join is created the angle between adjacent curves can be checked to see if they are within a given angle. By default the angle is 0.5 degrees. If the check fails the join are not successful.
ccrvs_tangency	When the join is created the tangency between connecting edges can be checked. If the check fails the join are not successful.
ccrvs_connexity	When the join is created the connexicity between connecting edges can be checked. If the check fails the join are not successful.
<i>dont_print_info</i>	By default entity conversion messages are output to the progress file, this option restricts the output to only errors and warnings.
<i>create_2d_edge_curves</i>	By default, 2D edge curves for faces are not written directly to CATIA V5. They are created in CATIAV5 by projecting the 3D edge curve onto the surface. This option allows the parameterized NURBS representation to be maintained as long as the underlying surface was originally of BEZIER or NURBS type.
<i>read_names</i>	By default, entity names will not be read. If this option is used the names will be read and passed to CATIA V5.

<i>dont_trim_face_surfaces</i>	By default face supporting surfaces are trimmed back to the face boundary. This option will cause these surfaces not to be trimmed during the translation .
blank_axis_systems	By default when an axis system is created it is shown. This option will allow the axis system to be hidden.
<i>groups_on</i>	By default groups are not read. This option allows the groups to be read and converted to bodies in CATIA V5 to preserve the hierarchy.
auto_face	By default standalone surfaces have topology added to them in CATIA5. However there is no control on the curve types created. This option adds the topology on the read leg and therefore the edge curves are of a known type. Needs to be used in conjunction with create_2d_edge_curves option.
no_plane_create	By default, if the CATIA V5 write encounters faces which are based on order 2x2 B-Spline Surfaces, it checks the surfaces to see if they are planar and if so will convert them to Planes to save space. If such surfaces should be kept as B-Spline Surfaces then the no_plane_create option should be used.
small_curves	The default behaviour of reporting an error if an attempt was made to create a curve whose length is less than the CATIA V5 resolution values has been altered. This option reinstates this error reporting to the progress file.
blank_construction_plane	The default behaviour is to leave standalone planes in show mode. This option cause them to be hidden
[keep_parameterization / reparameterize_both / reparameterize_curves / reparameterize_surfaces]	The default behaviour is to maintain the NURBS surface and curve parameterization from the input but this may not be ideal for CATIA V5 this option allows CATIA V5 to optimize the parameterization on NURBS surfaces, curves or both.
[ignore_failed_faces / process_failed_faces]	The default behavior is to split edges that pass through surface degeneracy's. This option stops this.
[dont_trim_face_surfaces / dont_trim_back_face_surf / trim_face_surf] [extend_nurb / no_extend_nurb]	To improve the success rate of face creation and limit the amount of data created the underlying surfaces are trimmed back to the face extents and for NURBS surfaces a margin of error is allowed to improved the success in projecting the 3D edge curves onto the surface to create the 3D edge curves. These option alter this behavior./
[simplify_crvs]	Causes NURBS curves to be simplified to analytics if possible
[convert_surface]	Causes analytic surfaces to be created as NURBS.
[dont_remove_double_edges]	Stops the removal of double edge curves i.e. two adjacent edge curves that run between the same vertices.
[dont_fix_3d_edges]	Stops the fixing up of 3D edges curves including trimming back to vertices

General Notes for V5 to Surf

1. Ellipses with $h_{maj}/h_{min} > 1000$ are converted to 3D Splines due to loss of precision during conversion.
2. SURFDB does not support the Parabola or Hyperbola so they are converted to 3D Splines.
3. Standalone Planes are ignored unless the 'pln_to_wpln' option is used in which case they are converted to Work Planes.
4. Planes that are base surfaces of Faces will be converted to Spline sur-faces.
5. Face 3D edges that are Circular Arcs will be converted to 3D Splines.
6. Since the SURFDB file does not support Solid Entities, any Solids encountered will be written to the SURFDB file as a collection of Faces.
7. If a mirrored Axis is encountered it will NOT be converted as the SURFDB file does not support mirrored axes.
8. Axis names are converted to upper case.

Common Options for V5 to Surf

Within the Configuration Manager pane of the Unified Interface, arguments that can be specified when publishing Surf data into V5 are grouped into 4 areas:

- Surf Write – Those arguments that affect how the data is written to Surf
- Entity Mask – Those arguments that allow specific read entities to be masked
- General – Those arguments that are common to ALL Publishing activities regardless of source data

Each of these options is described below.

Surf Write Arguments

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

Description: <input type="text"/>	
SurfDB Write	Entity Mask General
Option Name	Value
Break Breps	<input checked="" type="checkbox"/>
Refit 2D	<input type="checkbox"/>
Edge Tolerance (mm)	<input type="text" value="0.0025"/>
Trim Face Surfs	<input type="checkbox"/>
ASCII	<input type="checkbox"/>
Write Names	<input checked="" type="checkbox"/>
Ensure Bezier Curves	<input type="checkbox"/>
Ensure Bezier Surfaces	<input type="checkbox"/>
Fixup	<input checked="" type="checkbox"/>
Polygon Curves	<input type="checkbox"/>
Planes to Work Planes	<input type="checkbox"/>
Planes to Surfaces	<input type="checkbox"/>
Side Length	<input type="text"/>
Unface	<input type="checkbox"/>
Tolerance	<input type="text"/>
Check Rational	<input type="checkbox"/>

Each of these options is described below:

Option	Description
--------	-------------

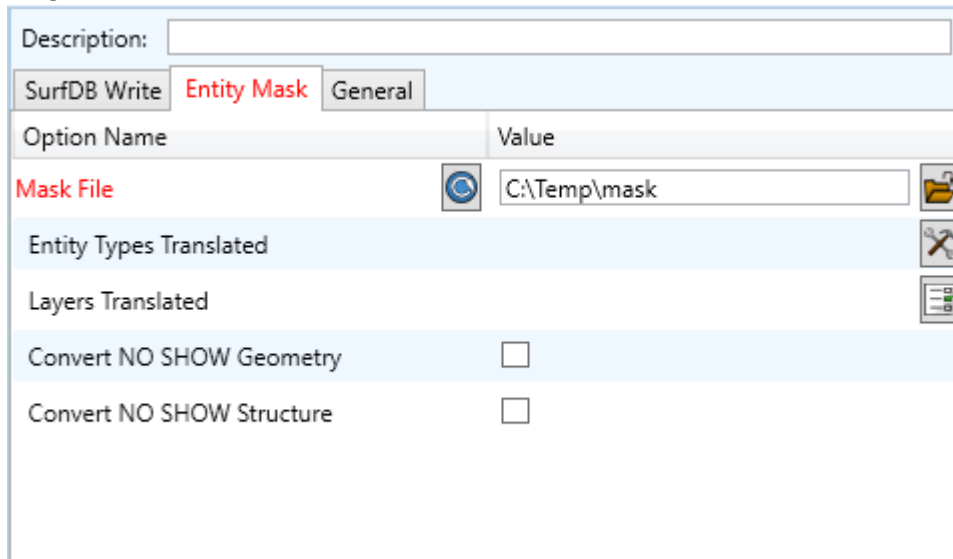
Break Breps	<p>By default, Brep Solids (as opposed to Open Solids) will be broken down into their constituent Faces during conversion. By using the break_breps option, Brep Solids will be converted to Shell entities. If the Brep has voids then each void will be converted to a Shell entity and the resulting Shells will become members of a Group entity which will be given the name of the originating Brep (unless no_write_names has been used or the Brep has no name). The outermost shell of the Brep will be the first member of the Group. Default is OFF. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>no_break_breps</i> – to turn off
--------------------	--

Refit 2D	<p>Optional. Specifies that faces read from UG should not have their 2D parametric edges refitted before being written to SURF but as far as possible, should be kept as they are from UG. This argument also increases the speed of processing because the translator does not have to perform the refitting process. If this option is not used then Face edges will be checked to see if splitting is required and whether the parametric loops are closed correctly. Default is OFF. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>read2d – to turn on</i>
Edge Tolerance (mm)	<p>There are 4 conditions for which 2D edge curves will be refitted for a particular Face during translation :-</p> <ol style="list-style-type: none"> 1. If the read2d option has not been used; 2. The read leg has failed to provide 2D edges for a Face; 3. The base surface for a Face has been converted to NURBS from some other form; 4. The trim_face_surfs option has been used and a Face has had its surface reduced; Default is 0.0025. ○ <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>edge_tol <tol></i>
Trim Face Surfaces	<p>Faces can be based on surfaces that are significantly larger than the face requires. By default, the translator will leave such surfaces as they are (which can lead to tolerance and extent problems in the receiving system). By using the trim_face_surfs option only the portion of the surface which is required by the Face will be translated thus reducing the amount of data stored and also reducing the risk of tolerance and extent problems in the receiving system. Default is OFF. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>trim_face_surfs – to turn on</i>
Ascii	<p>Creates reference planes. Default is OFF. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>Ascii – to turn on</i>
Write Names	<p>By default, each entity will be written to the output file with its name (if it has one). If the no_write_names option is used then all entities will be given an automatically generated name. Default is ON. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>no_write_names – to turn off</i>
Ensure Bezier Curves	<p>All spline curves that are single segment will be checked to ensure that they are in Bezier form, if possible. Default is OFF.</p> <p>○ Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>ensure_bezier – to turn on</i>

Ensure Bezier Surfaces	<p>All single patch surfaces will be checked to ensure that they are in Bezier form, if possible. Default is OFF.</p> <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>ensure_bezier_surfs – to turn on</i>
Fixup	<p>This option specifies that the check for effectively zero length parametric segments in B-Spline Curves should NOT be performed. If the option is not used then by default the check IS performed and any segments encountered which are too short are removed. Default is ON.</p> <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>no_fixup – to turn off</i>
Polygon Curves	<p>This option will check for degree 1 B-Spline curves that are not part of a Face and convert them to Polygon Curves. The default is to convert such curves to BSplines in the normal way. Default is OFF. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>bsp_to_mc – to turn on</i>
Planes to Work Planes	<p><i>Independent Planes are written to Work Planes in SURF.</i> Default is ON.</p> <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>no_pln_to_wpln – to turn off</i>
Planes to Surfaces	<p>By default, the translator will convert free planes in V5 to Work planes in SURF unless the argument 'no_pln_to_wpln' is specified, in which case they are ignored. If turned on, then Bsurfs will be created. Default is OFF.</p> <ul style="list-style-type: none"> ○ Command Line Syntax <ul style="list-style-type: none"> ▪ <i>pln_to_srf – to turn on</i>
Side Length (mm)	<p>Related to pln_to_srf Default is 1. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>pln_to_srf <value></i>
Unface	<p>Trims the surface back to the face boundary, if the value set is in the tolerance of the surface isoparms. Default is OFF. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>unface – to turn on</i>
Unface Tolerance (mm)	<p>Related to unface. Default is 0.00001. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>unface <value></i>
Check Rational	<p>Check to ensure incoming surfaces are rational. NOTE this can take will take a longer time to process if switched on. Default is OFF. ○</p> <p>Command Line Syntax</p> <ul style="list-style-type: none"> ▪ <i>check_rational – to turn on</i>





V5 to Surf Entity Masking Arguments

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



Description:

SurfDB Write **Entity Mask** General

Option Name	Value
Mask File	 C:\Temp\mask 
Entity Types Translated	
Layers Translated	
Convert NO SHOW Geometry	<input type="checkbox"/>
Convert NO SHOW Structure	<input type="checkbox"/>

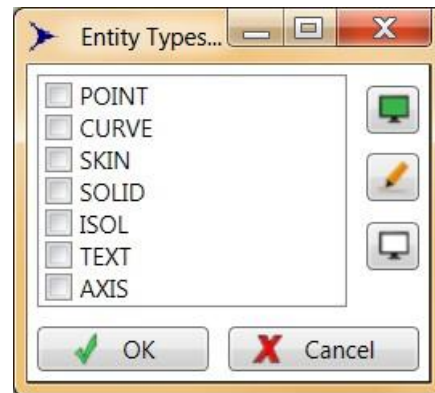
Each of these options is described below:

Option	Description
Mask File	<p>Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: o Command Line Syntax:</p> <ul style="list-style-type: none"> ▪ <i>Mask <filename></i>

**Entity Types
Translated**

Specifies a selection list from which to select which V5 entity types are to be processed. The following types are available:

"POI", "CUR", "SKI", "SOL", "ISO", "TEX", "AXI"



○ Command Line Syntax:

- Add any of the above to the specified mask file, one entry per line prefixed by the word ON, e.g.:

ON POI

to ensure they are considered in the translation

Layers Translated

Specifies a selection list from which to select which layers are to be processed.



Command Line Syntax:

- A single entry of **ON ALL LAY** must precede any Layer Mask command.
- Add a list or range of numbers representing layer to be processed to the specified mask file to ensure they are NOT considered in the translation e.g.:

OFF LAY 114,149,166,167,168

V5 to Surf General Arguments

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

Description:	
SurfDB Write	Entity Mask
Option Name	Value
Mass Properties	<input type="checkbox"/>
Advanced	<input type="text"/>

The option is described below:

Option	Description
Advanced	Allows any of the Command Line Advanced arguments to be passed to the conversion process

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.

Surf Advanced Arguments

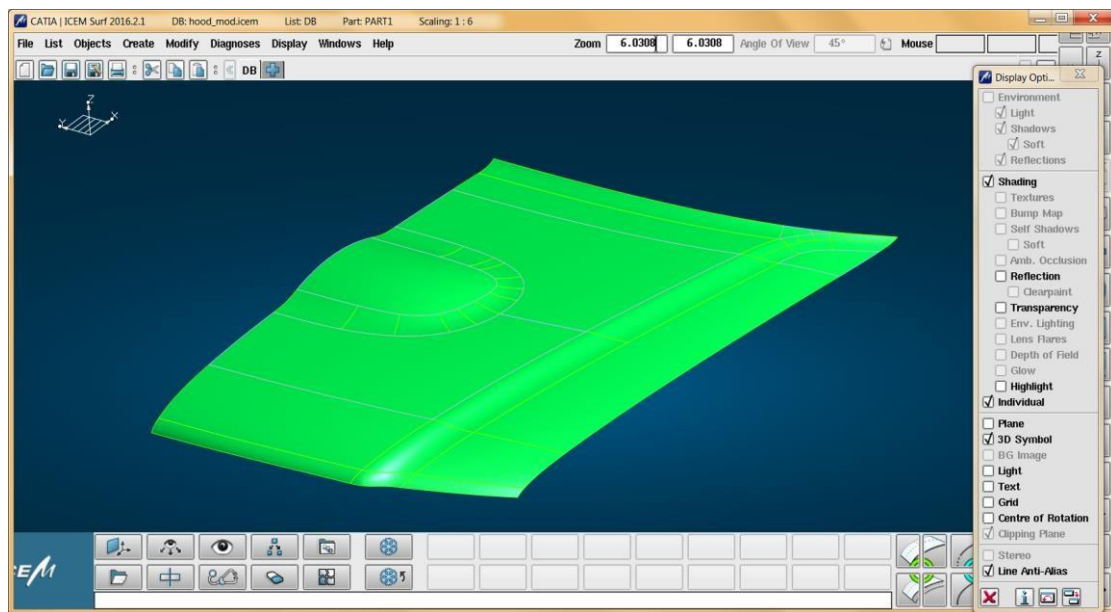
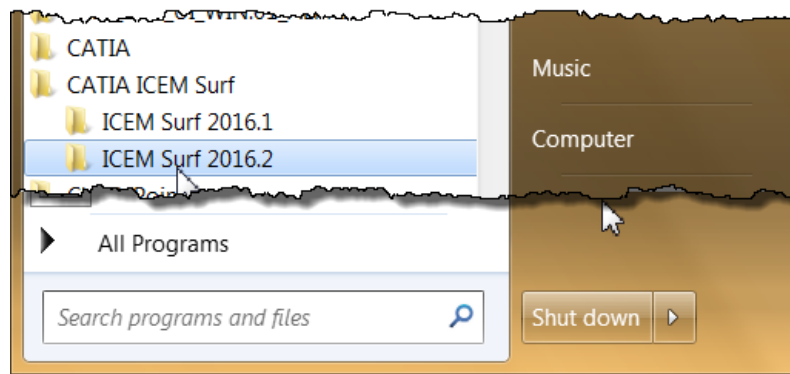
Argument	Description
convert_curves	<p>Converts curves to NURBS form. Default is <i>(Default is OFF)</i>.</p> <ul style="list-style-type: none"> ○ Command Line Syntax: <i>convert_curves</i>
convert_surfaces	<p>Converts surfaces to NURBS form. <i>(Default is OFF)</i>.</p> <ul style="list-style-type: none"> ○ Command Line Syntax: <i>convert_surfaces</i>
face_opacity	<p>Reads face opacity. This can lead to a performance degradation. <i>(Default is OFF)</i>.</p> <ul style="list-style-type: none"> ○ Command Line Syntax: <ul style="list-style-type: none"> ▪ <i>opacity_zero<tol></i> <p>Allows definition of tolerance below which an entity is considered to be totally transparent.</p>

Translating Interactively from within Surf

The Surf to V5 translator allows an active Surf Part or Assembly to be exported directly into V5 format or an V5 Part or assembly to be imported directly from within the Surf application.

In order to translate from within Surf, the Surf application must be started.

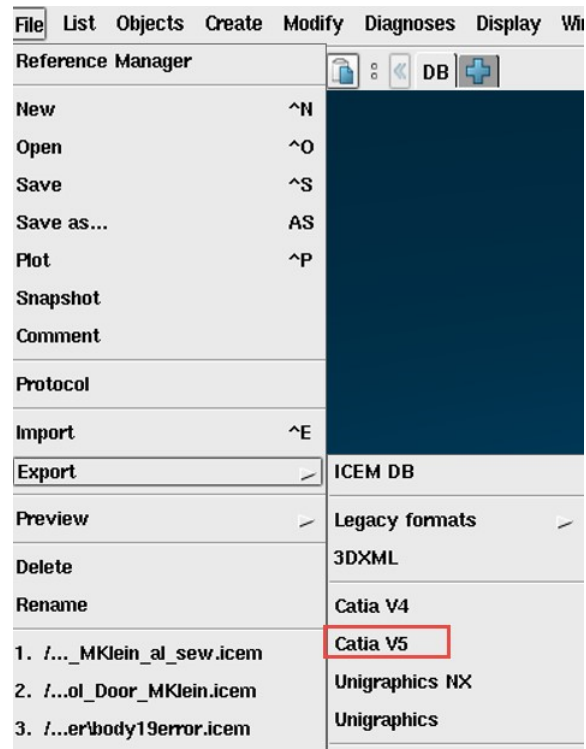
The appropriate KDdat files must be modified to set the environment up to successfully create good quality data depending on the translator use. The modification of these files is explained later in this document in the Appendix A – Configuring Surf Surf can then be started using the standard commands e.g.



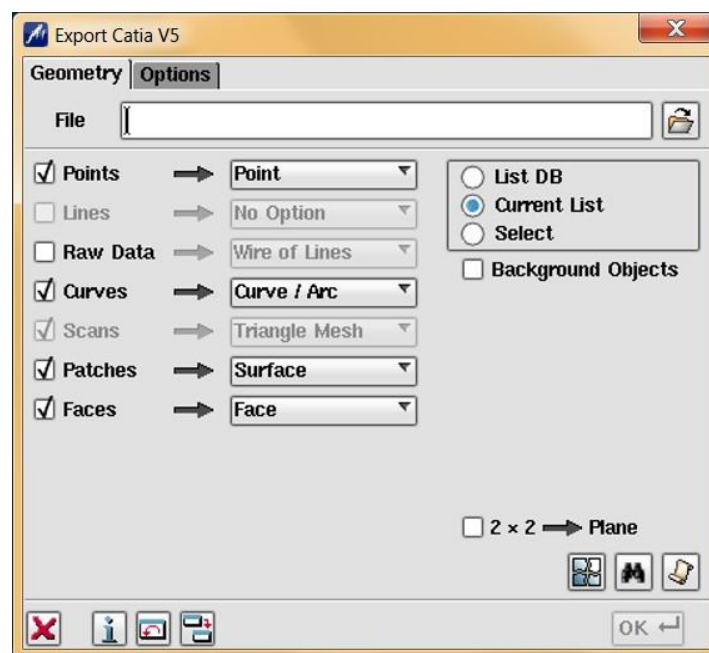
Export

Once Surf has been started and a model loaded, the active Part or Assembly can be exported to V5.

In order to export, the user selects the File -> Export Menu Option:

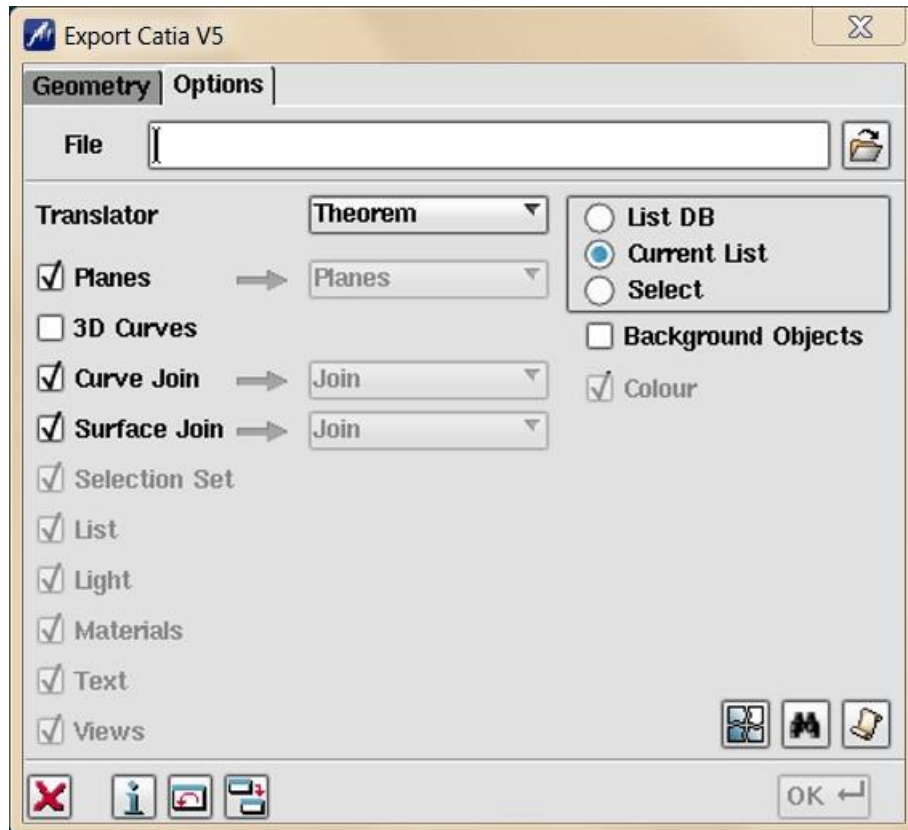


Which in turn launches the Export panel:



The **File** field may be prepopulated. However, this can be modified prior to selecting the **OK** button.

The **Options** tab is shown below, for further information please refer to the relevant section of the ICEM Surf documentation.

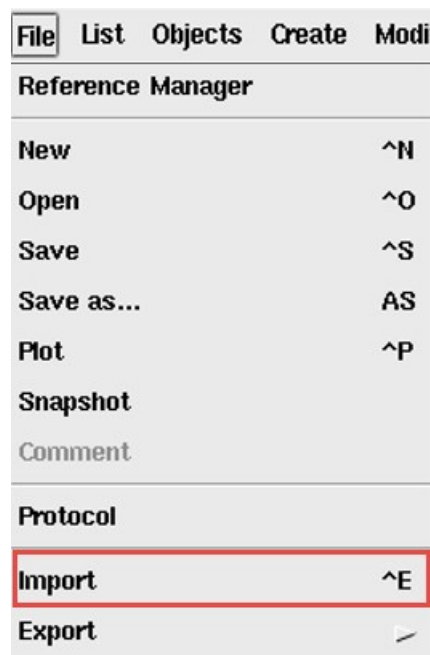


On selecting **OK** on the Export Panel, the active Part or Assembly will be written to CATIA V5 using the selected output directory.

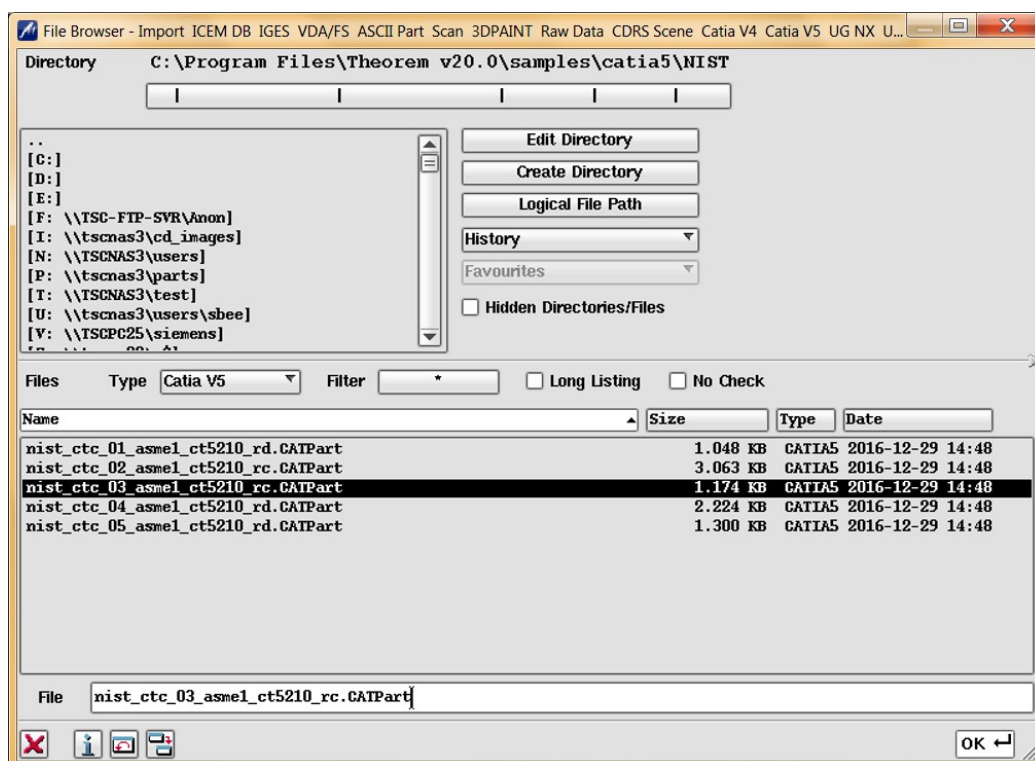
Import

Once Surf has been started, V5 data can be imported into Surf.

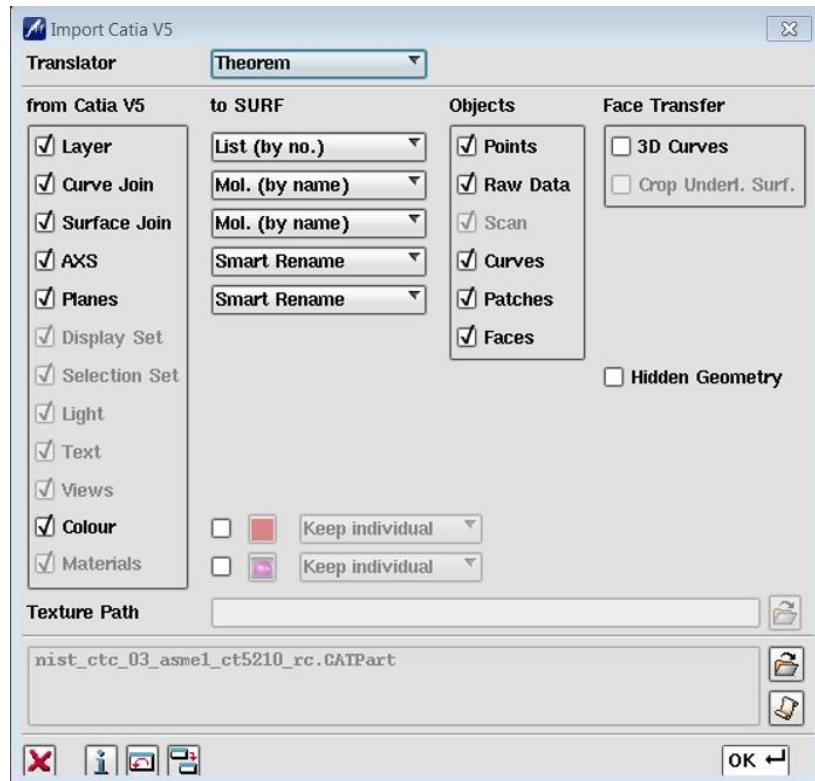
In order to import, the user selects the File -> Import Option:



Which in turn launches the Import File Browser panel:



The **Import From** option allows the user to select the V5 part or assembly to import. Selecting the OK button, brings up the Import dialog.



Selecting **OK** on the Import Panel import V5 data will be imported into Surf and the data saved using the input into the selected output directory. For more information please refer to the relevant section of the ICEM Surf documentation.

Appendix A – Surf Configuration

Introduction

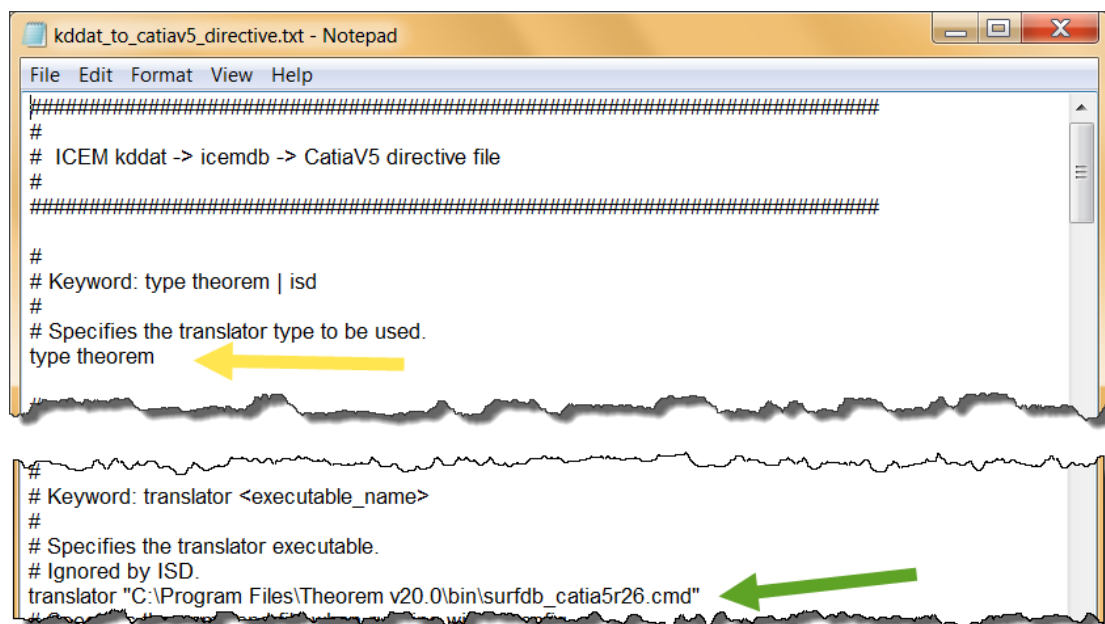
For the Surf product to work with a Theorem product for import/export it is necessary for the system administrator to make some modifications to directive files with the Surf product. These files are named...

- kddat_to_catia5_directive.txt
- catia5_to_kddat_directive.txt

For a default Surf 2017 installation their location is in

C:\Program Files\Dassault Systemes\ICEM Surf\icemsurf-2017.2\win_b64\reffiles

The partial contents of the catia5_to_kddat_directive.txt file are shown below



```
#####
#
# ICEM kddat -> icemdb -> CatiaV5 directive file
#
#####

#
# Keyword: type theorem | isd
#
# Specifies the translator type to be used.
type theorem

#####

#
# Keyword: translator <executable_name>
#
# Specifies the translator executable.
# Ignored by ISD.
translator "C:\Program Files\Theorem v20.0\bin\surfdb_catia5r26.cmd"
```

Two scripts control the launching of the Theorem

%TS_INST%\bin\catia5r[XX]_surfdb.cmd

%TS_INST%\bin\ surfdb_ catia5r[XX].cmd

As these scripts source the Theorem environment which includes the V5 ROOT DIR they can be used directly in the definition within the directives files. In the above diagram the translator type should be set to theorem (default is ISD) → The translator should point to the cmd file for the appropriate version of CATIA V5 → In the example shown this is R26.

Similar change are needed to the kddat_to_catia5r[XX]_directive.txt to enable the converter to point to the correct export translator type and cmd file e.g

Type theorem

C:\Program Files\Theorem\21.2\bin\surfdb_catia5r26.cmd

Default Values

The original products were written to support the premise that original Surface Design data would be written into V5 for final detailing. It was also seen to be important that these surfaces should not change, if possible, to keep the original design intent.

Current design practices of using scanned data or convergent modelling means that the data often starts off in the CAD system and passed to Surf to create the surface required.

If the surface then needs to be detailed, it may then need to be passed back to the CAD system. In an iterative Design process, this return trip may happen several times.

Because of the different modelling philosophies between CAD and Surface design, and exact and tolerant modelling techniques, a number of compromises may have to be made between success and accuracy during the translation

There are two sets of recommended defaults to suit the different design practices used. Please note that each customers data may differ in its requirements for translation, so the defaults used are advisory only.

Configuration files

From revision 21.1 the customer is able to use UI generated configurations on the command line, this functionality will be expanded in future revisions.

For printing a config file argument contents

The Configuration files are of xml format. If you want a list of the contents in text form, then the following command can be used

```
C:\Program Files\Theorem\21.1\ bin\cad_run.cmd -a "My V5 SURF Config" -o  
C:\temp\contents.txt
```

For running CAD_RUN with a config

```
C:\Program Files\Theorem\21.1\ bin\cad_run_UI.cmd CATIA526_SurfDB
```

```
-c "My V5 SURF Config"
```

-i C:\Program Files\Theorem\21.1\samples\catia5\NIST\nist_ctc_01_asme1_
ct5210_rd.CATPart

-o C:\Program Files\Theorem\21.1\samples\catia5\NIST\nist_ctc_01_asme1_
ct5210_rd.icem

Using Defaults

"C:\Program Files\Theorem 21.1\bin\cad_run_UI.cmd" CATIA526_SurfDB -i
"C:\PPC\Customers\XXXCo\R20 Tests_020617\Example.CATPart" -o
"C:\PPC\Customers\XXXCo\R20 Tests_020617\Example.icem"