CADverter for NX – CATIA ICEM Surf

Product Release Version 26.0

USER GUIDE

Revision: 1.0

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Overview of CADverter

About Theorem

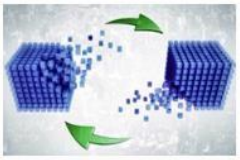
Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of 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 Siemens PLM Software. These relationships enable us to deliver best in class services and solutions to engineering companies worldwide.

What is CADverter?

CADverter is one of 5 core Theorem brands which consist of:



CADverter

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

Multi-CAD

Interactive integration of non-native 3D data formats into the native CAD system *Visualize 3D*

Direct translation of 3D data for the purpose of Visualization

Publish 3D

The creation of documents enriched with 3D content

Process Automation

Applications to automate any Data Exchange and collaboration processes

The CATIA ICEM Surf Bi-directional NX 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 NX is a direct database converter between Dassault Systemes CATIA ICEM Surf and Siemens NX. It enables the user to convert all forms of mechanical design geometry and attribute information, between these two systems.

The CATIA ICEM Surf - NX 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 NX 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 NX 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 – NX 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 – NX 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 – NX 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 NX from within Surf itself.
  - For a full guide to this process, please see our 'How to Translate Using the Surf – NX CADverter Interactive Mode' demonstration video located here.

Using the Product

Default Translations

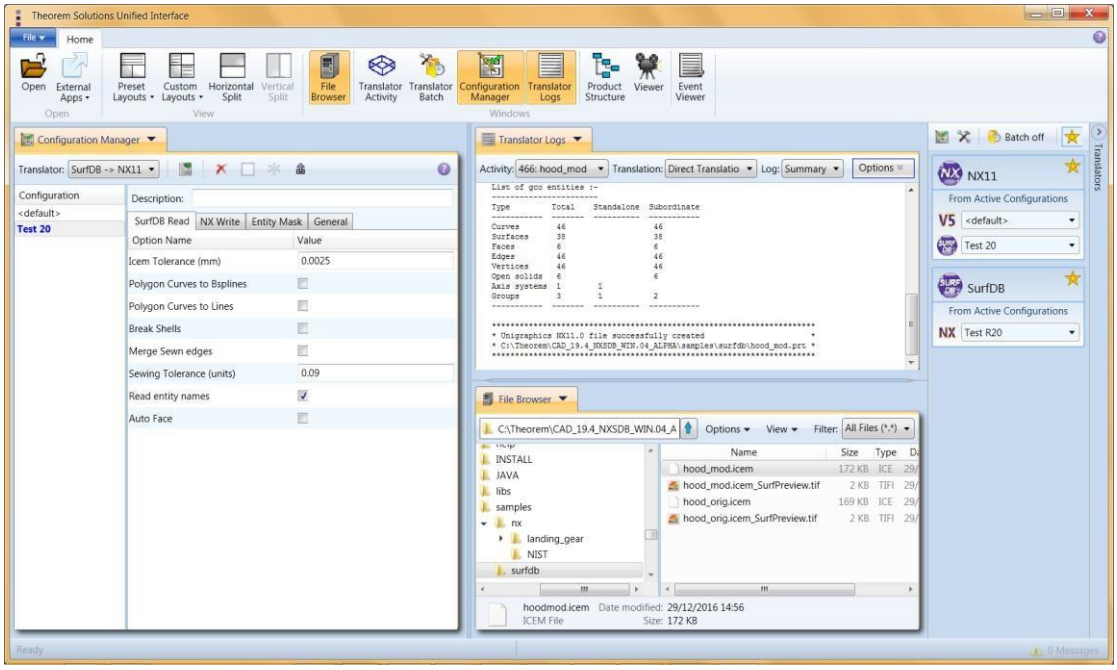
Default Translation – via the Unified Interface

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

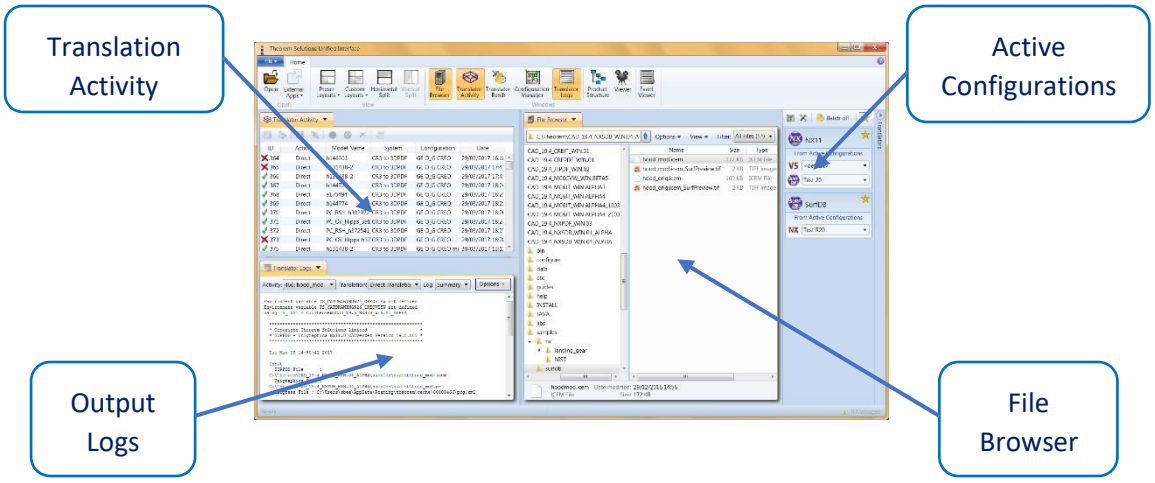
Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

<UI_installation_directory>\bin\Unified_Interface.cmd

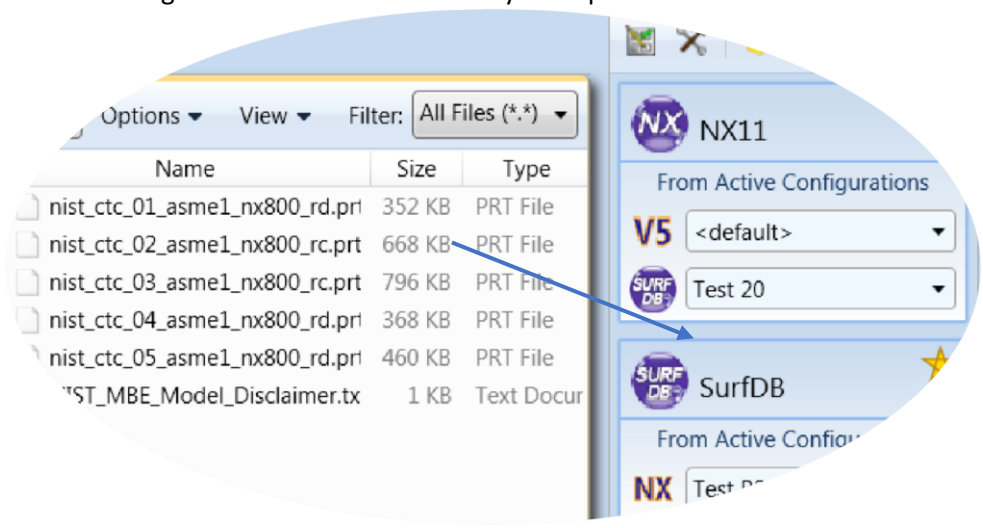
The following interface will be launched:



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

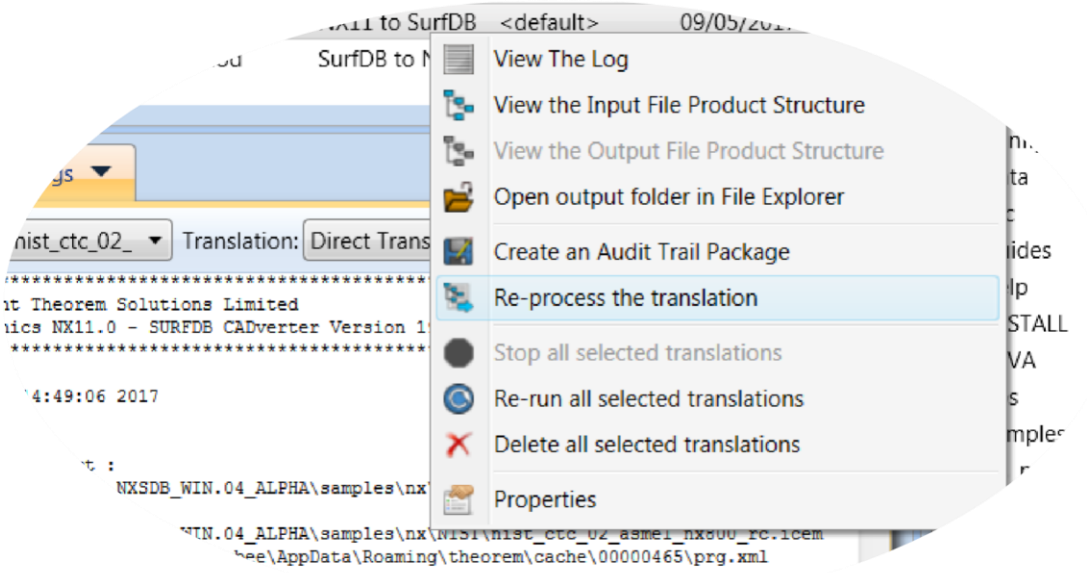


The simplest way to translate from Surf or NX is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.



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 NX:

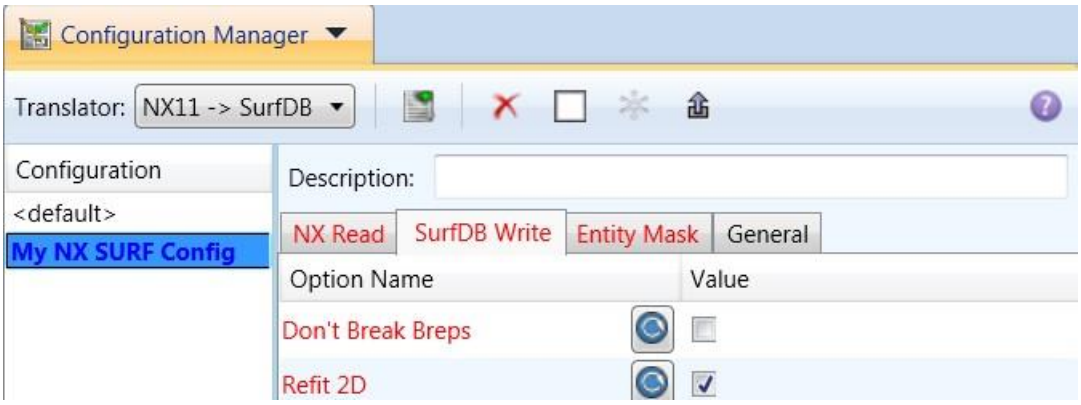
***<Translator_installation_directory>\bin\cad_run_UI.cmd SurfDB_NX[XX] –i <input_file> -o <output_file>* -c "<configuration name>"**

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

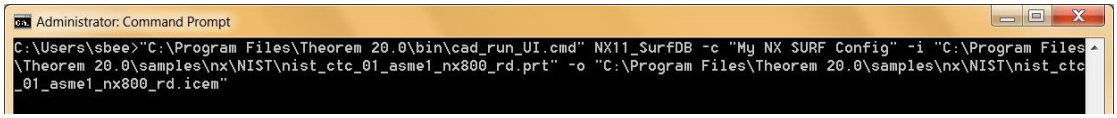
***<Translator_installation_directory>\bin\cad_run_UI.cmd NX[XX]_SurfDB –i <input_file> -o <output_file>* -c "<configuration name>"**

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

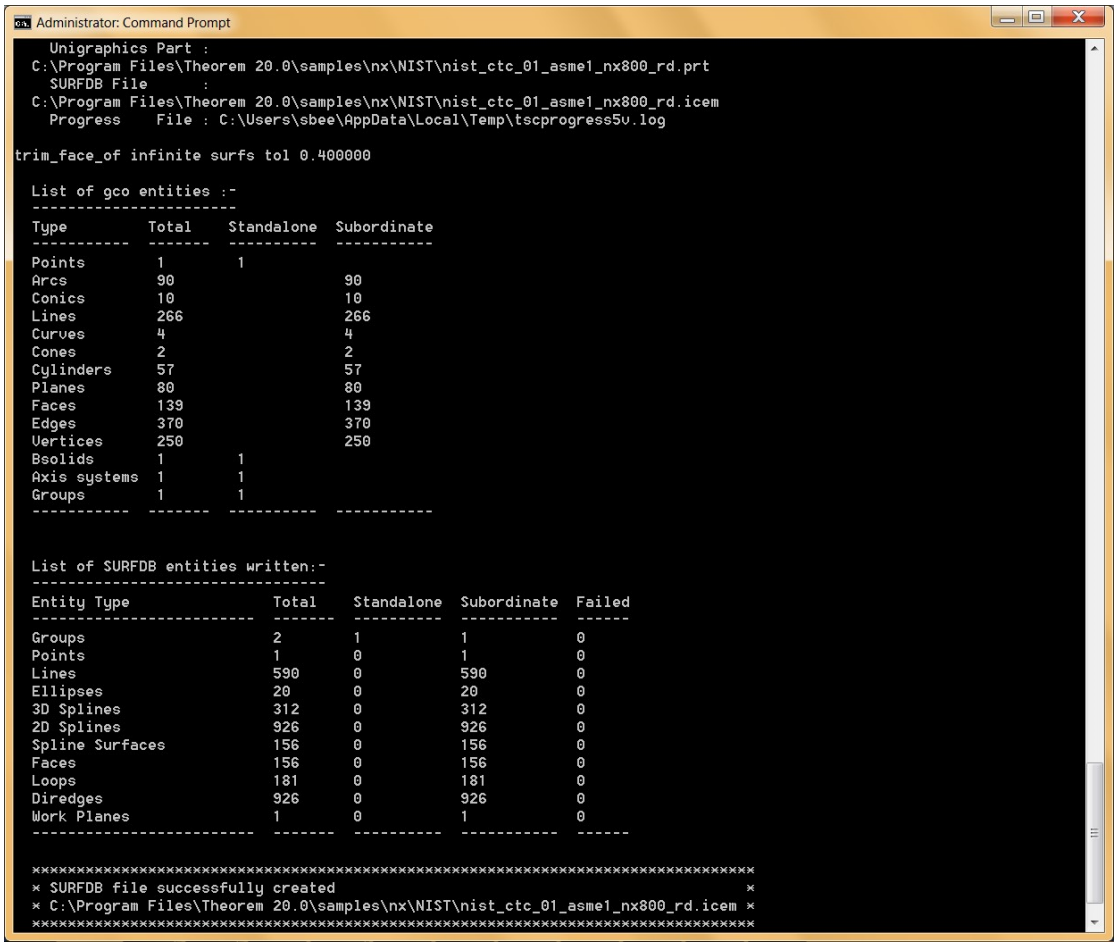
Note: If there is no valid configuration or you wish to translate using defaults, then omit the –c option (i.e. don't use a config)



(Note! Replace the [XX] seen in the example with the version of NX you are using. E.g. for NX 12.0, change to NX12):



The example above will translate an NX 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 22.2\samples\nx\NIST\nist_ctc_01_asme1_nx800_rd.icem

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, via the Command Line invocation or from inside Surf.

General Notes for Surf to NX

1. If the **auto_face** option is used then all standalone Spline Surfaces which are of Bezier type will automatically be faced.
2. 2D edges which have discontinuities in them are split at each discontinuity when read. This means the number of edges for a face may increase.
3. Shells will be written as separate Faces if the **break_shells** option is used or the Shell sewing process fails.

4. A 2x2 surface will get converted to a plane, unless 'no_plane_create' is specified (see 'Input Arguments' below).
5. Conversion of the Polygon Curve is dependent upon whether the **mc_to_bsp** or **mc_to_lin** options have been used. If none of these options is used then Polygon Curves are ignored.

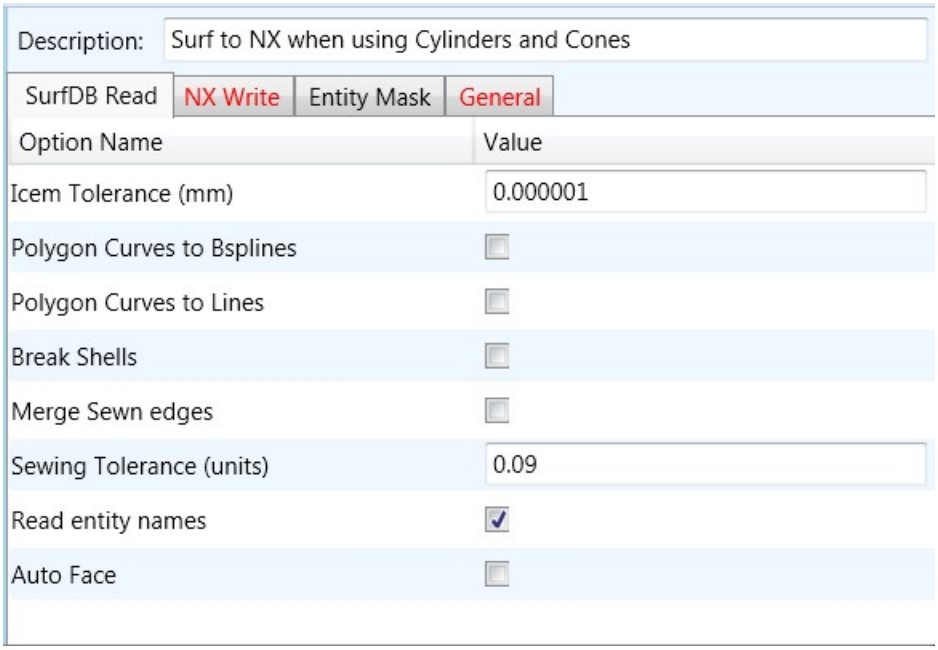
Common Options for Surf to NX

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

- Surf Read – Those arguments that affect how data is read from Surf
- NX Write – Those arguments that affect how the data is written to NX
- 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:



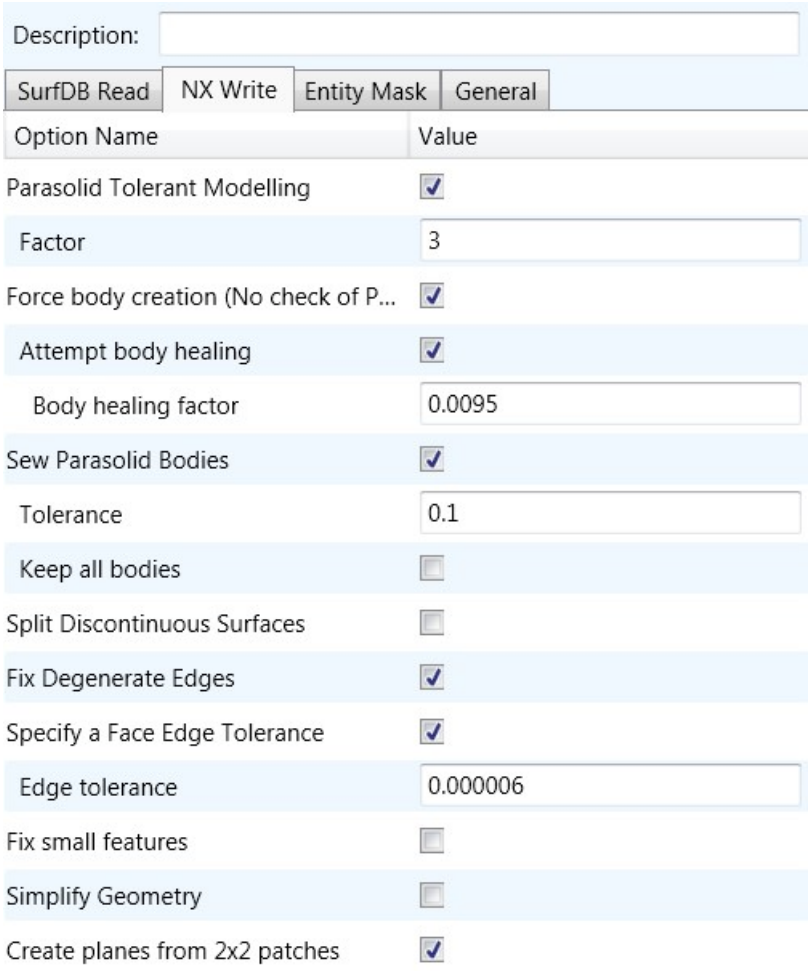
Each of these options is described below:

Option **Description**

Icem Tolerance	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 (*Default is 0.000001m*) ○ Command Line Syntax: ▪ *icem_tol 0.000001*
Polygon Curves to B-splines	Converts polygon curves to b-splines. *(Default is OFF).* ○ Command Line Syntax: ▪ *mc_to_bsp – to turn on*
Polygon Curves to Lines	Converts polygon curves to lines. *(Default is OFF).* ○ Command Line Syntax: ▪ *mc_to_lin – to turn on*
Break Shells	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. *(Default is OFF).* ○ Command Line Syntax: ▪ *break_shells – to turn on*
Merge Sewn Edges	By default in surfdb_ug, whilst reading a Shell there will be no merging of edges performed during the sewing function. If merging is required, then the merge_sewn_edges option can be used. This will look for pairs of edges that join with sufficient degree of continuity, and are the only two edges that join at that point, and merge them into a single edge, thus simplifying the model. *(Default is Off)* ○ Command Line Syntax: ▪ merge_sewn_edges – *to turn on*
Sewing Tolerance	When a Shell is read it has to be sewn together (unless the **break_shells** option has been used). *(Default is* 0.09/units). ○ Command Line Syntax: ▪ sew_tol<tolerance> – *to change tolerance option*.
Read Entity Names	Reads the entity names from NX *(Default is On)* ○ Command Line Syntax: ▪ no_read_names – *to turn off*
Auto Face	This option specifies that standalone Bezier patches should be automatically faced. *(Default is Off)* ○ Command Line Syntax: ▪ auto_face – *to turn on*

NX Write Arguments

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



Each of these options is described below:

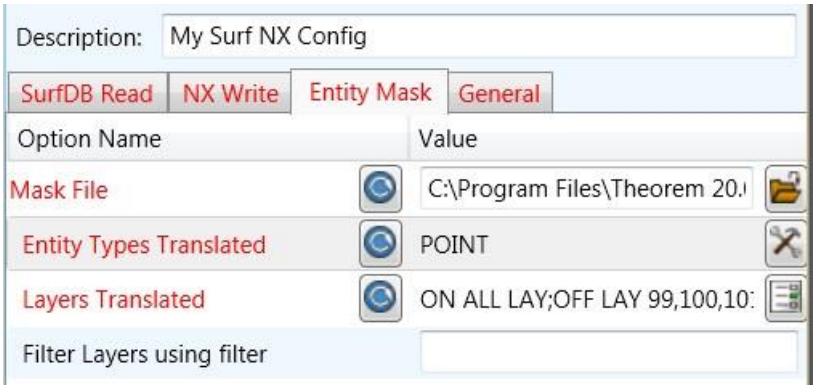
Option	Description
Parasolid Tolerant Modelling	Enables Parasolid tolerant modelling. Default is ON. ○ Command Line Syntax ▪ *nopstolmodel – to turn off*
Factor	A secondary option used with Parasolid Tolerant Modelling. Allows a factor to be defined. Default is 3. ○ Command Line Syntax ▪ *pstolmodel 3*

Force Body Creation	Force the creation of bodies. Default is ON. ○ Command Line Syntax ▪ *nocheck* ▪ *check – to turn off*
Attempt body healing	A secondary option used with Force Body Creation. Tries to heal the forced body. Default is ON. ○ Command Line Syntax ▪ *no_heal_ug – to turn off*
Body healing factor	The factor to be applied to Attempt Body Healing. Default is 0.0095. ○ Command Line Syntax ▪ heal_ug 0.0095
Sew Parasolid Bodies	Enabled Sewing of Parasolid Bodies. Default is ON. ○ Command Line Syntax ▪ *nosew – to turn off*
Tolerance	A secondary option for Sew Parasolid Bodies giving the tolerance level to use. Default is 0.1. ○ Command Line Syntax ▪ *pssew 0.1*
Keep all bodies	A secondary option used with Sew Parasolid bodies allowing all bodies to be kept (no matter how small) that may be created as a result of sewing Default is OFF. ○ Command Line Syntax ▪ *no_keep_all_bodies – default* ▪ *keep_all_bodies – to turn on*
Split Discontinuous Surfaces	Split Discontinuous Surfaces. Default is ON. ○ Command Line Syntax ▪ *brep_prep* ▪ *no_brep_prep – to turn off*
Fix Degenerative Edges	On face create failure, check and fix any degenerate edges. Default is ON. ○ Command Line Syntax ▪ *fix_degen* ▪ *no_fix_degen – to turn off*
Specify a Face Edge Tolerance	Specify an edge tolerance to be used when creating faces. Default is ON. ○ Command Line Syntax ▪ *Please see Edge Tolerance below*

Edge Tolerance	A secondary option used with Specify a Face Edge Tolerance where the tolerance value is assigned. Default is 0.000006. ○ Command Line Syntax ▪ *face_edge_tol 0.000006*
Fix small features in open solids	Remove small edges, sliver and spike faces from open solids. Default is OFF. ○ Command Line Syntax ▪ *ps_fix_osol – to turn on* ▪ *no_ps_fix_osol - default*
Simplify Geometry	Simplify Geometry. Default is OFF. ○ Command Line Syntax ▪ *simplify_solids – to turn on*
Create Planes from 2x2 patches	A 2x2 surface is converted to a plane in UG. Note: A standalone surface with no face will only get converted to a plane if the 'auto_face' arg is also specified. Default is ON ○ Command Line Syntax ▪ no_plane_create – *to turn off*

Surf to NX Entity Masking Arguments

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



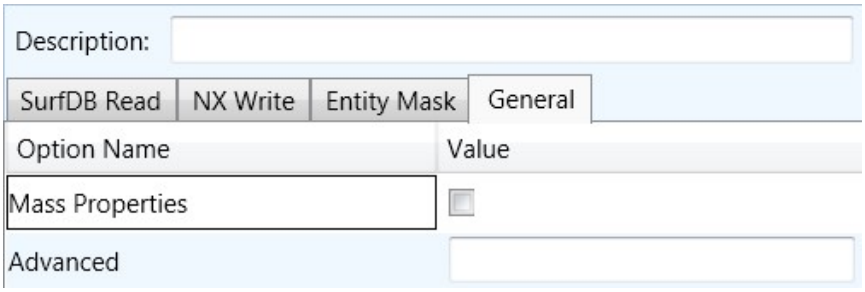
Each of these options is described below:

Option	Description
Mask File	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: ▪ *Mask <filename>*

Entity Types Translated	Specifies a selection list from which to select which entity types are to be processed. *The following types are available: "POI","CUR" ,"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***

Surf to NX General Arguments

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



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
ps_use2d	This option tells UG to use the 2d curves supplied by SURFDB, rather than recreating them from the 3D edges curves (the default is OFF) ○ Command Line Syntax: ▪ *Ps_use2d*

General Notes for NX to Surf

1. 3D edges for face that are Circular Arcs will be converted to 3D Splines.
2. Ellipses with hmaj/hmin >1000 are converted to 3D Splines due to loss of precision during conversion.
3. SURF does not support the Parabola or Hyperbola so they are converted to 3D Splines.
4. Planes that are base surfaces of Faces will be converted to Spline surfaces. Independent Planes are written (by default) to Work Planes in SURF, unless either of the the args [no_pln_to_wpln | pln_to_srf] is specified.
5. Since SURF does not support Solid Entities, any Solids encountered will be written to the SURF file as a collection of Faces, unless the no_break_breps option is used (see later).
6. If the bsp_to_mc option is used then degree 1 B-Splines are converted to Polygon Curves.
7. If there are only 2 points in the String then a Line is created, but, if more than 2 points then a Polygon Curve is created.

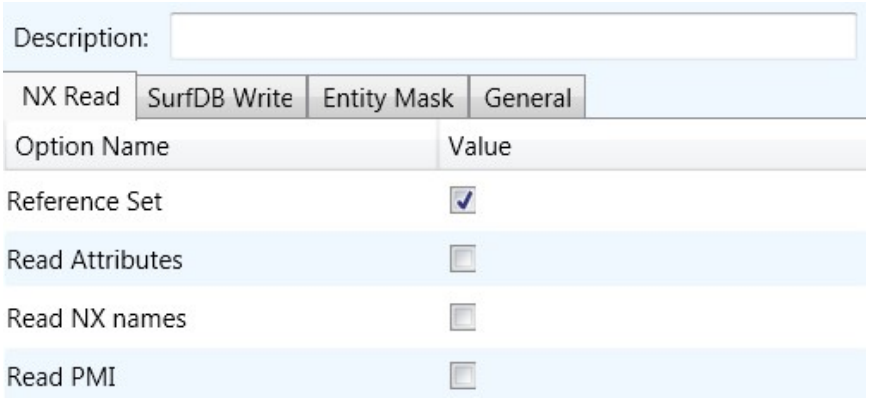
Common Options for NX to Surf

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

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

NX Read Arguments

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

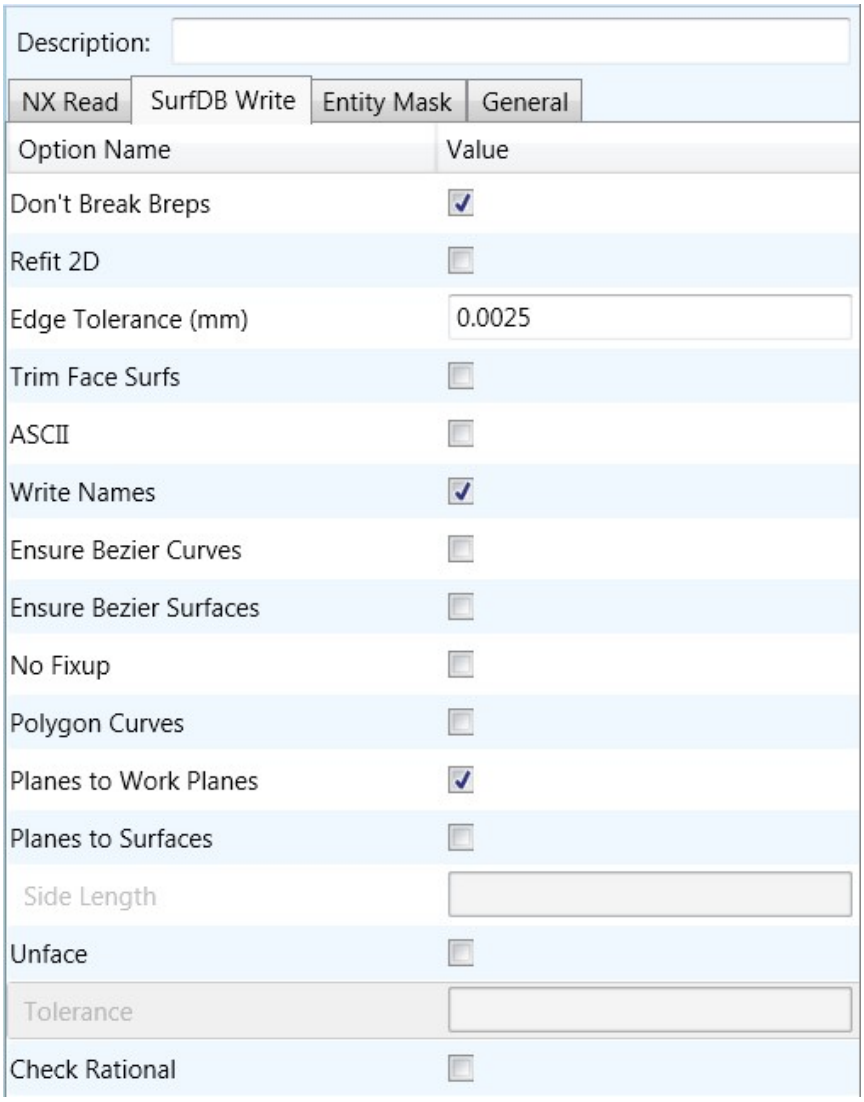


Each of these options is described below.

Option	Description
Reference Set	Enabled reference set processing. Default is ON ○ Command Line Syntax: ▪ *No_ref_set – to turn off*
Read NX Attributes	Read NX detail user attributes. Default is OFF. ○ Command Line Syntax: *read_attrs*
Read NX names	Read NX entity names, if they exist. Default is OFF. ○ Command Line Syntax: ▪ *no_read_name – default* ▪ *read_name – to turn on*

Surf Write Arguments

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



Each of these options is described below:

Option **Description**

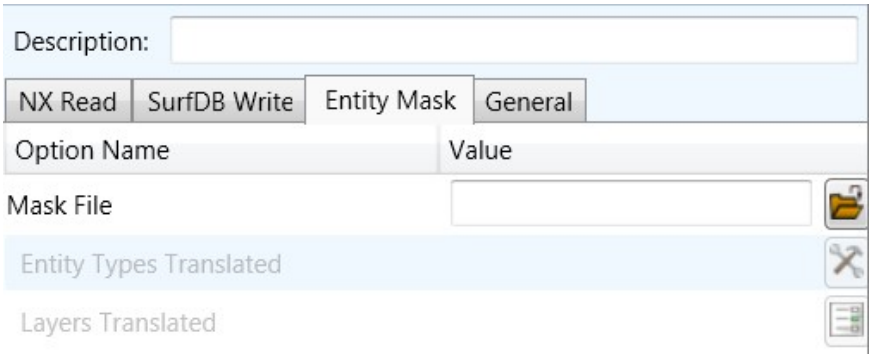
Don't Break Breps	By default, Brep Solids (as opposed to Open Solids) will be broken down into their constituent Faces during conversion. By using the **no_break_breps** option, Brep Solids will be converted to Shell entities. If the Brep has voids then each void will be converted to a Shell entity and 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. ○ Command Line Syntax ▪ *no_break_breps – to turn off*
Refit 2D Edge Curves	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. ○ Command Line Syntax ▪ *read2d – to turn on*
Edge Tolerance (mm)	There are 4 conditions for which 2D edge curves will be refitted for a particular Face during translation :- 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. ○ Command Line Syntax ▪ edge_tol <tol>
Trim Face Surfaces	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. ○ Command Line Syntax ▪ *trim_face_surfs – to turn on*
Ascii	By default, a binary output file will be created. With the use of this option an ascii output file will be created. Default is OFF. ○ Command Line Syntax ▪ *Ascii – to turn on*

Write Entity Names	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. ○ Command Line Syntax ▪ no_write_names – *to turn off*
Ensure Bezier Curves	All spline curves that are single segment will be checked to ensure that they are in Bezier form, if possible. Default is OFF. ○ Command Line Syntax ▪ ensure_bezier – *to turn on*
Ensure Bezier Surfaces	All single patch surfaces will be checked to ensure that they are in Bezier form, if possible. Default is OFF. ○ Command Line Syntax ▪ ensure_bezier_surfs – *to turn on*
No Fixup	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. ○ Command Line Syntax ▪ no_fixup – *to turn off*
Polygon Curves	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. ○ Command Line Syntax ▪ bsp_to_mc – *to turn on*
Planes to Work Planes	*Independent Planes are written to Work Planes in SURF*. Default is ON. ○ Command Line Syntax ▪ no_pln_to_wpln – *to turn off*
Planes to Surfaces	By default, the translator will convert free planes in NX 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. ○ Command Line Syntax ▪ pln_to_srf – *to turn on*
Side Length (mm)	Related to pln_to_srf Default is 1. ○ Command Line Syntax ▪ pln_to_srf <value>
Unface	Trims the surface back to the face boundary, if the value set is in the tolerance of the surface isoparms. Default is OFF. ○ Command Line Syntax ▪ *unface – to turn on*

Unface Tolerance (mm)	Related to unface. Default is 0.00001. ○ Command Line Syntax ▪ *unface* <value>
Check Rational	Check to ensure incoming surfaces are rational. NOTE this can take will take a longer time to process if switched on. Default is OFF. ○ Command Line Syntax ▪ *check_rational – to turn on*

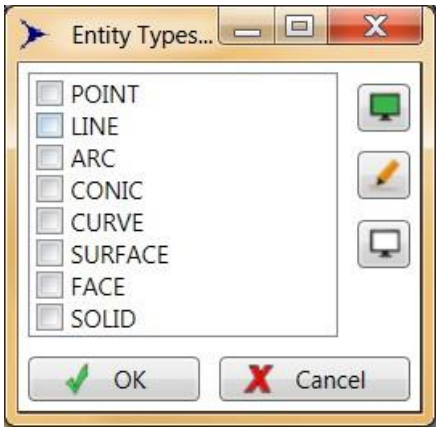
NX to Surf Entity Masking Arguments

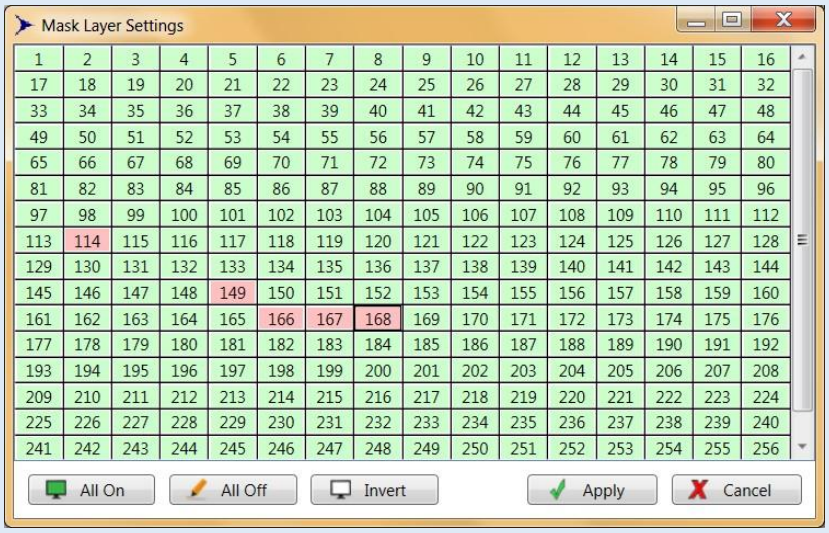
The image below shows the Masking arguments that are available, with their default settings:



Each of these options is described below:

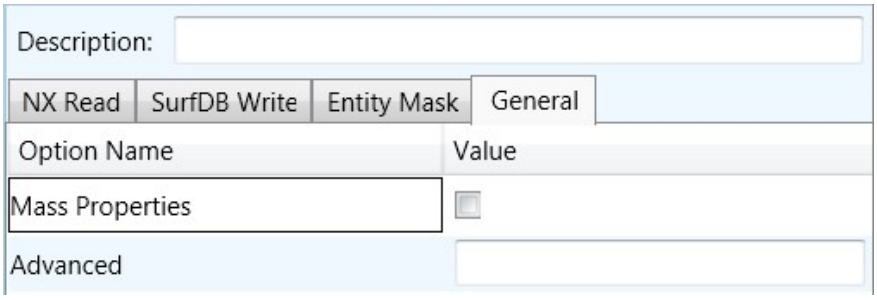
Option	Description
Mask File	Specifies the Mask File to be written to, that can be referenced by future translations. A Mask file MUST be specified if masking is required. The first line in this file is OFF ALL ENT: ○ Command Line Syntax: ▪ *Mask <filename>*

Entity Types Translated	Specifies a selection list from which to select which NX entity types are to be processed. The following types are available: "POI","LIN","ARC","CON","CUR","SUR","FAC","SOL"  ○ 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***

NX to Surf General Arguments

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



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	Converts curves to NURBS form. Default is *(Default is OFF).* ○ Command Line Syntax: ▪ *convert_curves*
convert_surfaces	Converts surfaces to NURBS form. *(Default is OFF).* ○ Command Line Syntax: ▪ *convert_surfaces*
face_opacity	Reads face opacity. This can lead to a performance degradation. *(Default is OFF).* ○ Command Line Syntax: ▪ *opacity_zero<tol>* Allows definition of tolerance below which an entity is considered to be totally transparent.

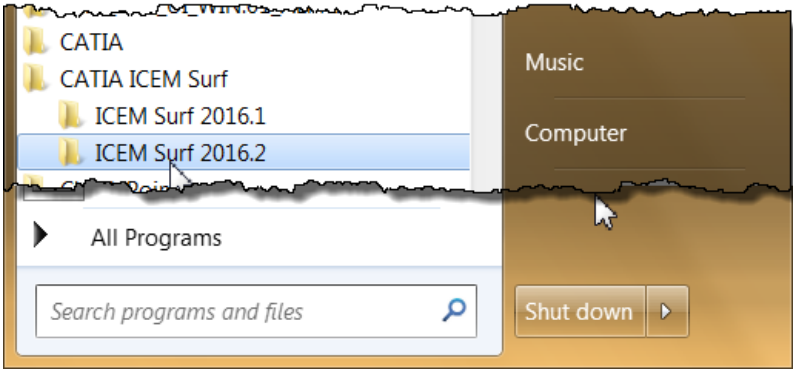
Translating Interactively from within Surf

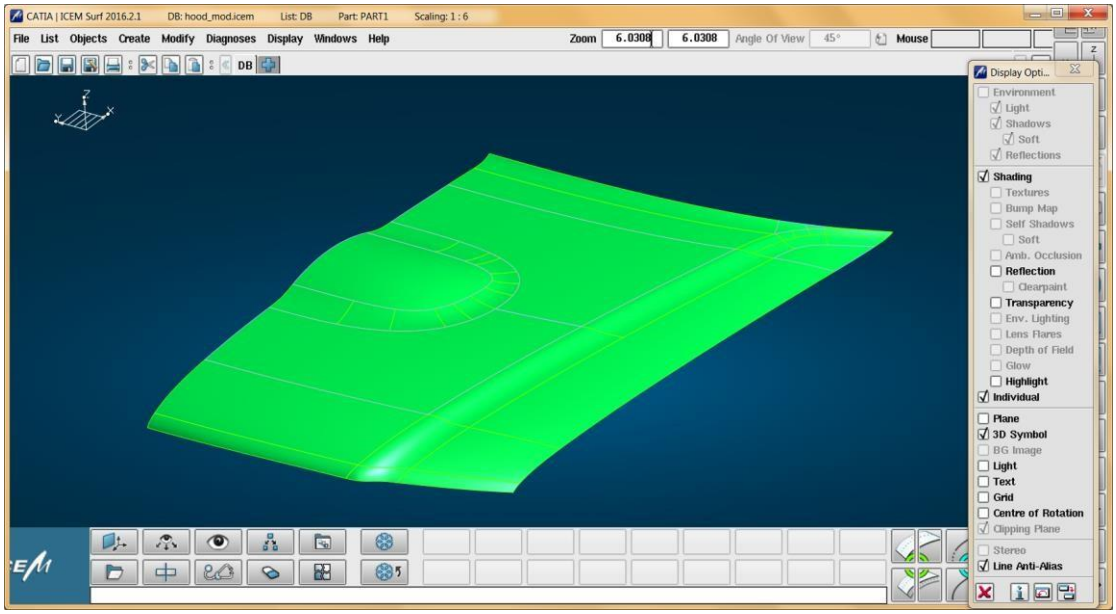
The Surf to NX translator allows an active Surf Part to be exported directly into NX format or an NX Part to be imported directly from within the Surf application.

NOTE: To run the translator interactively in Surf you will need an option license which is available from Dassault Systemes

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.

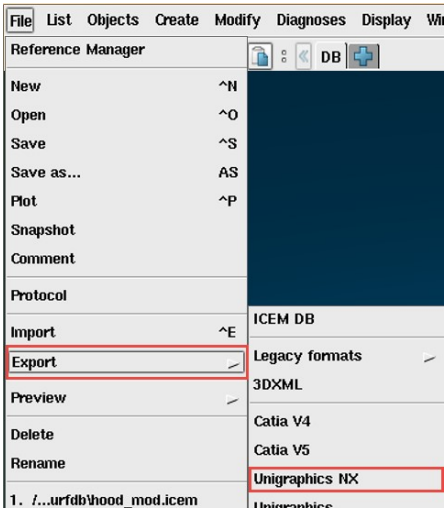




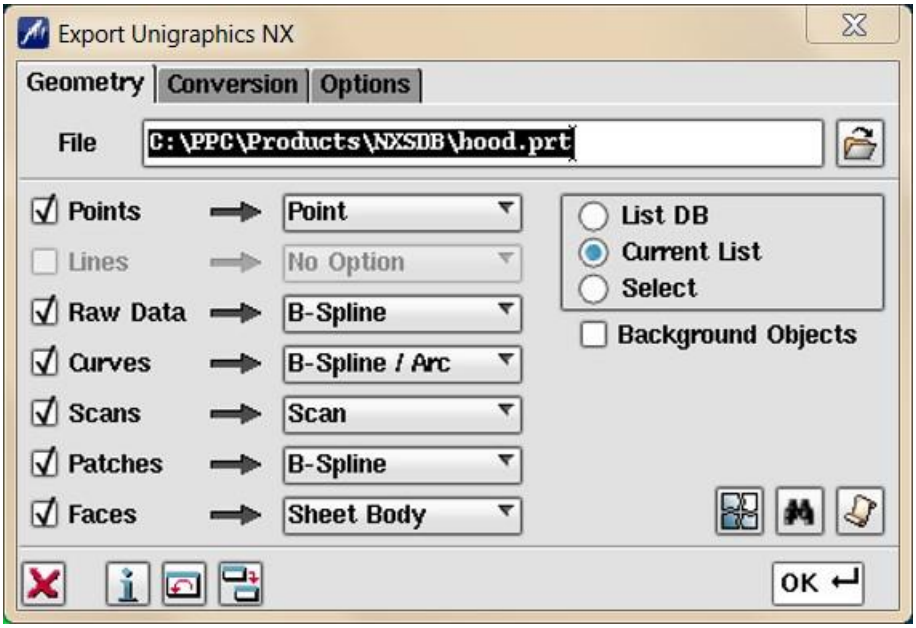
Export

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

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

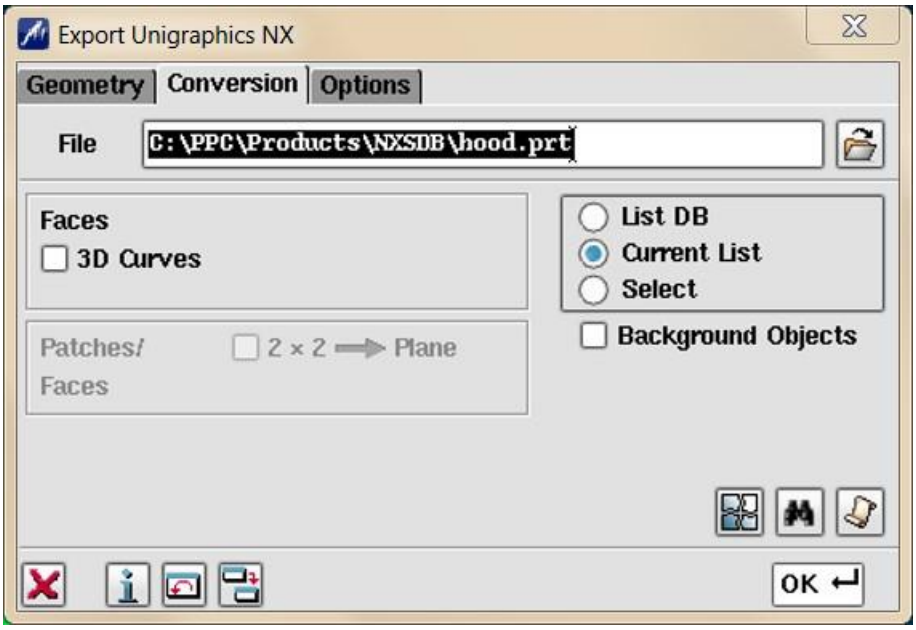


Which in turn launches the Export panel:

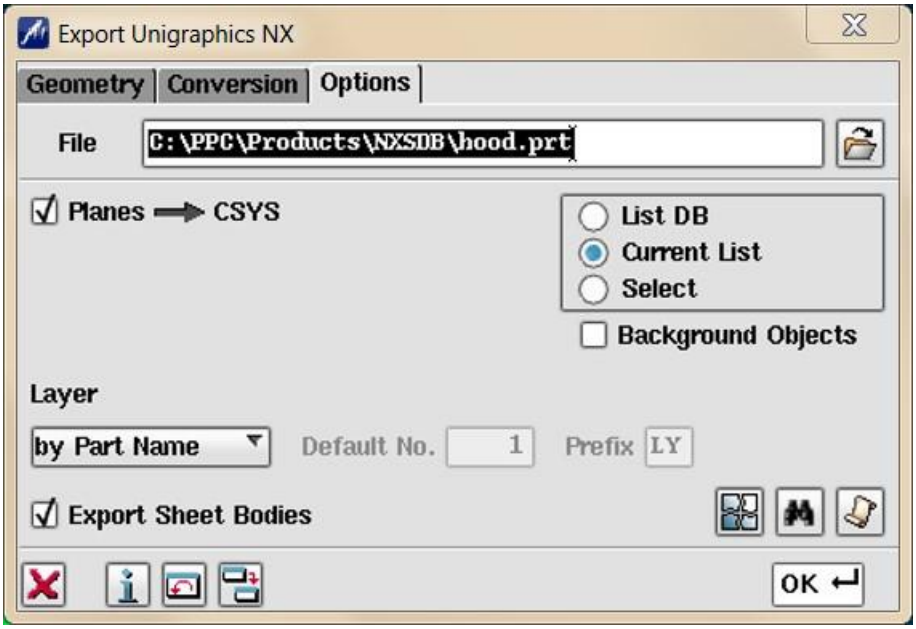


The ***File*** field may be prepopulated. However, this can be modified prior to selecting the ***OK*** button.

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



The ***Options*** tab



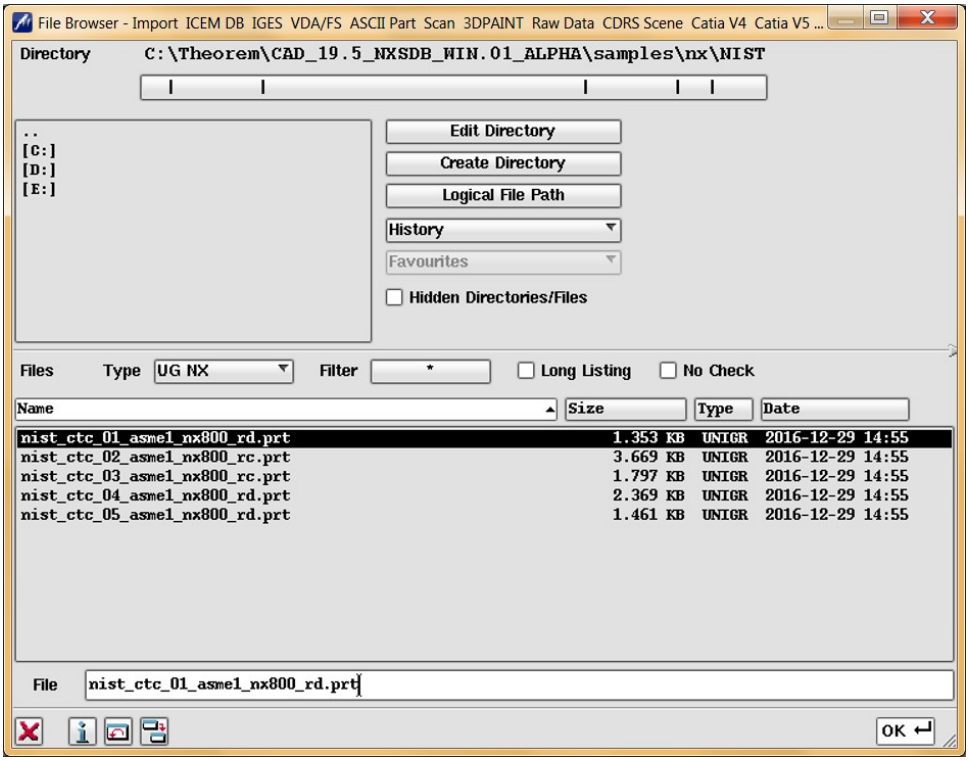
On selecting ***OK*** the on the Export Panel, the active Part will be written to NX using the selected output directory.

Import

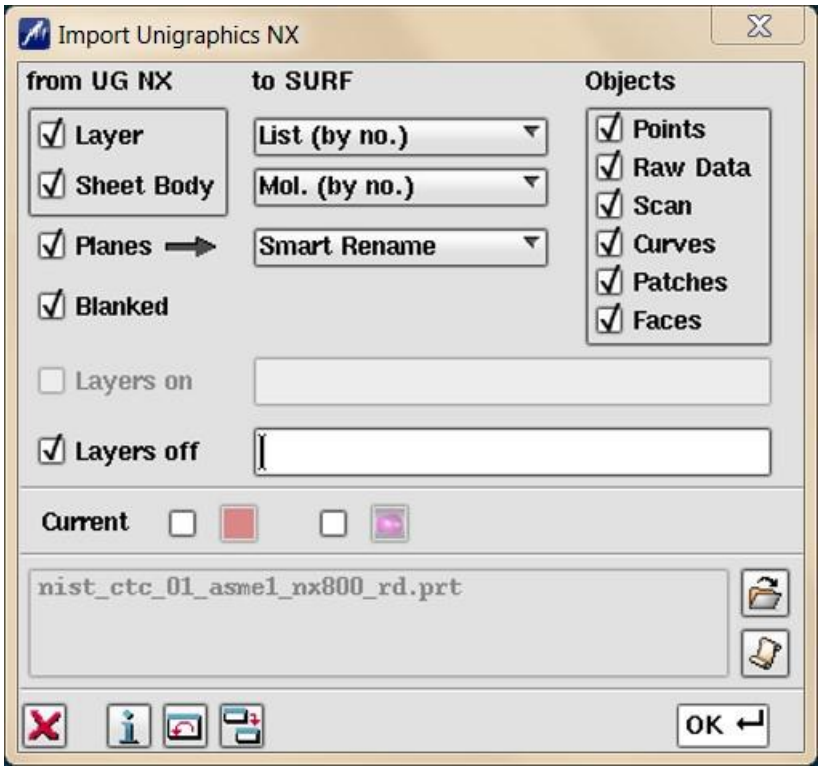
Once Surf has been started, NX data can be imported into Surf.

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 NX part to import. Selecting the OK button, brings up the Import dialog.



Selecting ***OK*** the on the Import Panel import NX 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

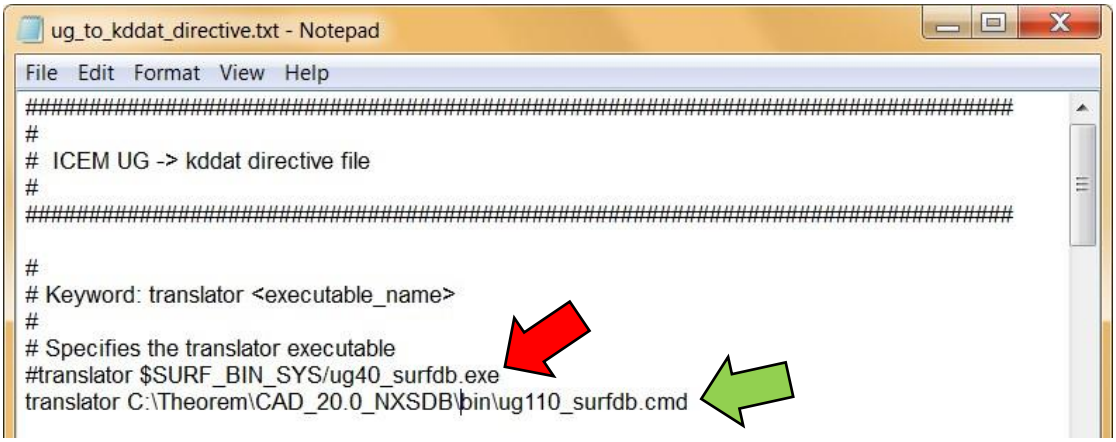
For the Surf product to successfully import/export data with a Theorem product it is necessary for the system administrator to make some modifications to directive files within the Surf product. These files are named…

- kddat_to_ug_directive.txt
- ug_to_kddat_directive.txt

For a default Surf 2016 installation their location is in

C:\Program Files\Dassault Systemes\ICEM Surf\icemsurf-201X\win_b64\reffiles

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



In the above diagram the supplied reference to NX 4.0 ← should be removed or commented out and the appropriate line for NX11 ← added.

A similar change is needed to the kddat_to_ug_directive.txt to enable the converter to point to the correct export translator e.g enter the line similar to that below

C: \Program Files\Theorem v22.2\bin\surfdb_ug110.cmd

At this revision there are two available methods with to which call the translation. The first is a new way of working which allows the user to specify a named config defined by the Configuration Manager within the UI as detailed earlier in the document. The advantage of this method is that the config, or its contents, can be changed without the need to close ICEM

The older method is still supported at this revision for those customers that wish to continue using this.

The two scripts which control the launching of the theorem translator when importing NX data

	%TS_INST%\bin\nx[XXX]_surf.cmd	**-New Method**
Or	**%TS_INST%\bin\ug[XXX]_surfdb.cmd**	**-Old Method**

Similarly for exporting data to NX

	%TS_INST%\bin\ surf_ nx[XXX].cmd	**-New Method**
Or	**%TS_INST%\bin\ surfdb_ ug[XXX].cmd**	**-Old Method**

As all of these scripts source the Theorem environment which includes the NX ROOT DIR they are used directly in the definition within the directives files.

If using the new method it is also necessary to supply the name of the config files to be used with the cmd files.

e.g.

surf_nx100.cmd

```
1  @echo off
2  SETLOCAL EnableDelayedExpansion ableDelayedExpansion
3
4  rem set My_SURF_NX_Config=-c "<CONFIGURATION NAME>"
5  set My_SURF_NX_Config="My_SURF_NX_Config"
6
7  set CWD=%CD%
8  set TSC_LOAD_LIB=%~dp0
9  cd /D %TSC_LOAD_LIB%..
10 set TS_INST=%CD%
   /D %
```

The configs can be created by the method shown earlier in this document

Default Values

The original products were written to support the premise that Surface Design data would be written into NX 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 four 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. Further examples will be added to a download area as they become available

Surf ➔ NX (single trip)

The default arguments are set for this option, i.e. a specific config file is not necessary

NX ➔ Surf (single trip)

The default arguments are set for this option, i.e. a specific config file is not necessary

Surf ➔ NX (when using cones and cylinders in Surf)

Data may need to be prepared to give an acceptable output in NX

no_brep_prep	NX write arg - don't prepare body
icem_tol 0.000001	SurfDB read arg used to set geometric read tolerance
surf_tol 0.0000001	NX write arg used to set surface and edge tol used
face_edge_tol 0.0000001	NX write arg to set edge tol
ps_use2d	use 2D curves if they exist

NX ➔ Surf (for NX parts using tolerant edges)

These arguments are suggested for use to maintain accuracy when converting NX parts which have been modelled with tolerant edges

noprep	NX read arg - don't prepare body
surf_tol 0.0000001	NX read arg used to set surface and edge tolerance used in read
edge_tol 0.000001	SurfDB write arg used to specify edge tolerance used in Surf

Sample Configs

A zip file containing some sample configurations can be found here. To use them they should be placed in the area where your existing config are stored. In a default installation for a single user this will be in

C:\Users*MyUserName*\AppData\Roaming\theorem\configure

For more information on how to make these available for all users see the section '*Common Configurations' in the UI User guide (UI_Doc.pdf)*

Three new configs have been added at 22.2 which should cover most eventualities :-

- If data is from Surf and translates to NX with missing faces, then try config 'SDBNX_prepare'
- If translated data has edge tolerances that are worse than the source data, use the appropriate '*_tight_tolerances' config

Further Examples

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\Theorem22.2\CAD_22.2_NXSDB_WIN.01\bin\cad_run.cmd -a "My NX SURF Config" -o C:\temp\contents.txt**

For running CAD_RUN with a config

C:\Theorem\CAD_19.5_NXSDB_WIN.01_ALPHA\bin\cad_run.cmd NX11_SurfDB -c "NXSDB"

-i
C:\Theorem\CAD_19.5_NXSDB_WIN.01_ALPHA\samples\nx\NIST\nist_ctc_01_asme1_nx800_rd.prt

-o
C:\Theorem\CAD_19.5_NXSDB_WIN.01_ALPHA\samples\nx\NIST\nist_ctc_01_asme1_nx800_rd_out.icem

Using Defaults

> "C:\Program Files\Theorem 22.2\bin\cad_run.cmd" NX11_SurfDB –i "C:\PPC\Tests\R20 Tests_020617\ErrorExample2.prt" -o "C:\PPC\Tests\R20 Tests_020617\ErrorExample2.icem