Alias® | StudioTools™

Data Transfer

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StudioTools 13

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Not all features described are available in all products.

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Contents

Data transfer 1

Introduction 3

Introduction to Data Transfer 4
Learn how Solid Modeling Theory works 6
Learn the Solid Modeling workflow 8
Learn about the tolerance requirements for Solid Modeling 9
Learn how to get the topology right before transferring data 11

Requirements and workflows for CAD packages 12

Pro/ENGINEER 13
CATIA V4 21
CATIA V5 24
I-deas NX series 27
Unigraphics 31
Solid Imaging 34
TC VisProducts 37

How do I? 39

Manage files through a WebDAV interface 40

Manage your digital assets in WebDAV 41

Import CAD data files 47

Import a data file 48
Import Illustrator files 49
Open a Binary STL file 50
Import Cloud data 51
Import a PTC Granite or a Pro/ENGINEER part file 52
Import from Pro/ENGINEER into StudioTools via Pro/RENDER 53
Pro/ENGINEER options for export and import of STEP or IGES files 55
Import CATIA V4 into StudioTools 56
Import a CATIA V5 file into StudioTools 57
Import a Unigraphics file into StudioTools 61

Export CAD data files 63

Export a data file 64
Add a description to an export file 65
Check for gaps before exporting to a solid model 66
Export an Illustrator file 67
Use the tolerances required by a specific CAD package 68
Export STEP/IGES/PTC Granite to Pro/ENGINEER 70
Export STL files 74
Export SDL files 77
Export to CATIA V4 79
Export to CATIA V5 80

Troubleshoot 83

Overview 84

Edges do not match 85

View and interpret data transfer log files 86

Log file names 87
View STEP log files 89
Interpret CATIA log files 90
Viewing IGES log files 92
Identifying IGES supported entities in log files 94
Unigraphics log files 96

Evaluation tools 99

Appendix 101

Details on File Format types 102
Wire format 105
IGES format 106
VDAFS format 107
VDAIS format 109
C4 format 110
Unigraphics proprietary format 111
JAMAIS format 115
PTC Granite format (Windows Only) 116
DES format 119
DXF format 120
STEP format 121
CAI format for CATIA V4 files 125
CATIA V5 format 135
OBJ format 138
Encapsulated Postscript format 140
Inventor format (IRIX and Windows) 141

Illustrator format 143
VRML format 144

Index 147

Data transfer

How to import and export data from external software packages.

Introduction

Learn the theory behind CAD data transfer and how it works within StudioTools.

It is not necessary to read this information to complete a data transfer. However, it may help you understand how data can be transferred successfully.

Introduction to Data Transfer

Explains the general workflow of data transfer used in StudioTools.

StudioTools provides translators of industry standard data exchange formats as well as DirectConnect's file formats.

Data transfer workflow

1. Set your model environment to match your CAD system.
   Preferences > Construction options > Construction Preset
2. Create your model.
3. Use evaluation tools to verify that the geometry is suitable for data transfer.

- **Evaluate > Check model**

  Use this tool to analyze a model (or portions of a model) for geometry that has particular characteristics. Depending on the options you choose, a report is generated describing the contents of a model and the results of the checks performed.

  See *Prepare a model for import into CAD systems* (page 402) for a sample workflow.

- **Evaluate > Continuity > Surface continuity**

  This tool checks the position, tangent and curvature continuity between and within surfaces.

- **Locators > Deviation**

  Use these tools to check the maximum distance between surface boundaries in StudioTools to confirm the integrity of the model before transferring it to the target CAD system.

- **Surface Edit > Stitch > Shell stitch**

  This tool enables you to create a valid solid model topology within StudioTools. Stitching surfaces within StudioTools creates a shell. When the shell is exported to a downstream (CAD) system, it includes an extra layer of information.

The stitching process also identifies surface boundaries that exceed the prescribed tolerances. These problems can then be corrected by the designer prior to the translation of the data.

Save the original model before stitching.

Stitching is not required prior to transferring Unigraphics , Pro/ENGINEER, or CATIA files. If the geometry is stitched, it comes into the target system with topology information. If it is not stitched prior to transfer, the geometry comes into the target system as NURBS geometry.

Learn how Solid Modeling Theory works

Learn how geometric and topological data works together to form a solid model.

You should be familiar with the concepts of solid model data types to understand how geometric and topological data work together to form a complete model.

- Geometric data

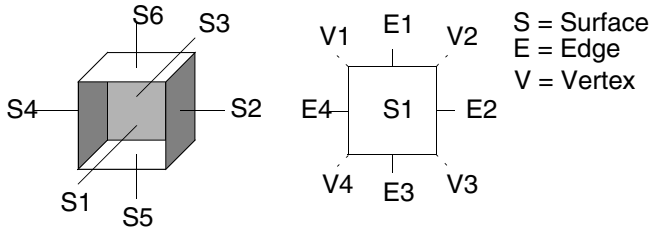
  Surfaces contain the geometric data of a solid model. The geometric data describes the basic shape of an object and can be represented using NURBS (Non-Uniform Rational B-Splines).

- Topological data

  Loops, edges, and vertices contain the topological relationships between the individual surfaces that form the solid model. Topological data describes how the geometric components are connected together. In solid modeling terminology, surfaces are called *faces*, and each face is made up of loops, edges, and vertices.

How does it work

The following is an example of a solid model of a cube in the which has six surfaces (faces) labeled S1 to S6 which form the geometrical and topological information required to define it as a solid model. Each of the surfaces has a loopset, but in this case each surface has only one loop. The loop for S1 has four edges and four vertices.



The edges are used to connect two loops from adjacent surfaces. The vertices are used to connect two or more edges.

This solid box consists of six surfaces, twelve edges, and eight vertices that form the geometrical and topological information required to define it as a solid model. For example, if a hole is placed in the box through S1 and S6, S1 and S6 would each have a loopset containing two loops.

Learn the Solid Modeling workflow

Learn what requirements are expected in a solid modeler.

Creating solid geometry from imported StudioTools models is a common workflow when integrating StudioTools and CAD systems. When exporting the model from StudioTools, you must ensure that the geometry is built to the correct tolerance and that it can also be stitched. Stitching in StudioTools identifies gaps between surfaces so that you can repair the geometry before exporting the file.

During stitching, the surfaces are twinned. This means that the surface boundaries may be split to accommodate adjacent surfaces, and periodic geometry is detached into multiple surfaces. For this reason, you should save the StudioTools wire file before stitching so that if further modifications to the StudioTools model are required, the construction history will be intact.

A stitched geometry saved to an StudioTools wire file cannot be unstitched to its original state.

What happens when you import data into a solid modeler

When you import a StudioTools model into a solid modeler, you provide the geometric and topological information of the model. When creating a solid model, the solid modeler system creates a valid data base from the supplied data, and the supplied data must satisfy the solid modeler's rules for topological and geometric data.

The topology of a model defines how each surface relates to all other surfaces in the model. The important element of getting the topology right for data exchange to solid modelers is that an edge on one surface must have a “twin” edge on the adjacent surface. Edges are defined by natural surface boundaries or trimmed surface boundaries.

You can transfer surfaces and complete the stitch procedure in a CAD system or first stitch them in StudioTools and then export tha data.

Learn about the tolerance requirements for Solid Modeling

Learn how to achieve the tolerances required in solid modeling.

To achieve the tolerance required by solid modeling, it is important to manage the modeling units and tolerances when creating your model. The millimeter (mm) or inch is generally used as the base linear unit. Standards for tolerances have been developed as they apply to engineering-based CAD systems.

If you are not sure of the standards your companies or clients use, ask your CAD system manager. Set up your units and tolerances at the beginning of your modeling session and save them as a preset in the **Construction Options** box. The next time StudioTools is opened, the preset that was in use when StudioTools was last exited, will be in effect.

To successfully join or align surfaces in the target system, the gap between the surfaces of your model must be less than the accuracy defined within the solid modeler.

To specify various tolerances choose **Preferences > Constructions options.**

Rational and non-rational geometry concerns for data transfer

In the **Preferences > Constructions options** window, you can specify whether or not the new geometry being created will contain the rational or non-rational component.

Rational geometry contains CVs that do not have a uniform weight, while the CVs of non-rational geometry all have the same weight. Some CAD systems that do not support rational geometry will rebuild the rational element of geometry upon import. This will change the intended design and therefore the user should know ahead of time whether rational geometry is supported by the target CAD system.

Rational fillets are created with fewer isoparametric curves and the tangency to the adjacent surface can be up to ten times more accurate. While this is an advantage in StudioTools, it is

even more apparent when the geometry has been transferred to a solid modeler. The closer adjacent surfaces are to exact tangency, the more usable the imported StudioTools data is in downstream operations. For example, the further the geometry can be offset during the thickening operation.

Once the above conditions have been met, you should try several sample translations to verify that the geometry is being passed from StudioTools successfully. Before modeling a project in StudioTools that is intended for export, you should model several sample pieces of geometry in mock modeling situations, then transfer them and attempt the stitching operation in the target CADS system. This will confirm that the model, when completed, will transfer successfully.

Whether you are creating a model, verifying a model, or debugging a translation, there are a number of tools in StudioTools you can use to check the quality of the geometry you have created. The most useful tools are the surface continuity checker (**Evaluate > Continuity > Surface continuity**) and the **Min/Max** measurement tools (**Locators > Deviation**). Use these tools to check the maximum distance between surface boundaries in StudioTools to confirm the integrity of the model before transferring it to the target CAD system.

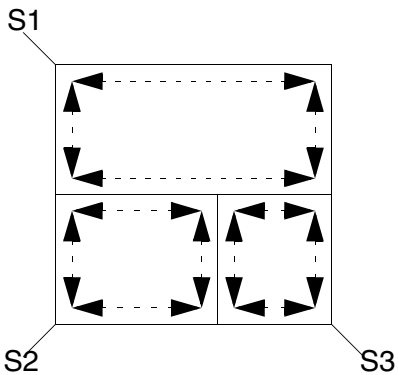
Learn how to get the topology right before transferring data

Learn how to get the topology correct for solid modeling.

The **Surface Edit > Stitch** tools in StudioTools creates a valid solid model topology.

- Stitching surfaces can greatly improve the data transfer to a solid modeler.
- The stitching process also identifies surface boundary gaps that exceed your tolerances.
- The stitching process identifies duplicate surfaces in the model and unifies the direction of the surface normals of the completed shell.

The following illustration shows three individual surfaces (labeled S1, S2 and S3). The edges of each surface are represented by dashed lines.



When models are constructed using the StudioTools advanced surface tools (**Swept, Rail Surface** and **Square**), it is common to create a number of smaller surfaces along the edge of one larger surface. This modeling technique does not create the twin edges required for a solid model. Stitching adds this information.

Some modeling techniques, such as **Trim**, **Intersect** and **Round**, create twin edges.

Requirements and workflows for CAD packages

Review the requirements and data transfer workflows.

PTC Granite™, IGES or STEP file formats can be used to transfer StudioTools models to or from Pro/ENGINEER Wildfire.

Pro/ENGINEER Requirements

- Pro/ENGINEER Wildfire 1.0 or 2.0, IGES, STEP, or Granite translators.
- SurfaceStudio, AutoStudio, Studio or DesignStudio.

Model Preparation

Units

When working in Pro/ENGINEER set the Units to be the same as what was used in the StudioTools model.

Geometry/Topology

When using StudioTools two types of model information can be sent to and read by Pro/ENGINEER. Those two types are geometry information and topology information.

The Studio-created IGES file includes only the description of the geometry information. The Studio-created STEP and Granite file supports both the geometry information as well as the topology information.

The geometric data describes the basic shape of the object and in both StudioTools and Pro/ENGINEER, geometric data is represented using NURBS.

Topological data describes how the geometric components are connected together to form a solid. The StudioTools STEP file format has advantages over IGES when transferring StudioTools models to Pro/ENGINEER because there is more information describing the model that is being transferred.

Tolerances

- In the Construction Settings window:

  Set **Preferences > Construction options** - **Construction Preset** to **Pro/ENGINEER**.

Information specific to IGES

- StudioTools sets and IGES levels.

  StudioTools set information is only exported in files for IGES, VDAIS, or JAMA-IS, if the option Level Mapping is set to SET. If a StudioTools set is given a name of the form of LEVEL<n>, where <n> is an IGES level number (and greater than 0), then the corresponding IGES entity for each member of the StudioTools set is assigned to level<n> in the IGES file. For example, the IGES entities corresponding to each member of the set LEVEL245 is assigned to level 245 in the IGES file.

  If a StudioTools object is a member of several multisets that conform to this naming convention, then the IGES file contains a Property Entity 406 form 1 (Definition Levels) listing the IGES levels to which the corresponding entity belongs.

Information specific to Granite

The translation time of rational geometry (for example, exact radius surfaces) is longer than the translation of non-rational geometry. In StudioTools you can create non-rational surfaces and translate them into PTC Granite.

Curve Fit Distance is the key to translation quality. The recommended tolerance in the **Preferences > Construction options - Construction Preset - Tolerances > Fitting** is based on testing done translating models between StudioTools and Pro/ENGINEER.

Maximum Gap Distance is the value that is used to check if the adjacent boundaries are built closely enough to each other.They should never be smaller than the Curve Fit Distance.

Workflow

Before transferring geometry between StudioTools and Pro/ENGINEER, you should consider the purpose of the

transfer to plan an appropriate workflow. When you import your StudioTools model into Pro/ENGINEER, you provide the geometric and topological information of the model. When creating a solid model, the Pro/ENGINEER system must create a valid Pro/ENGINEER data base from the StudioTools supplied data. The StudioTools supplied data must satisfy the Pro/ENGINEER's rules for topological and geometric data.

Get the geometry right

To achieve the tolerance required by solid modeling it's important to manage the modeling units and tolerances when creating your model. Most engineering organizations use the millimeter or inches units as the base linear unit and have developed standards for tolerances that they apply to their CAD systems.

If you are not clear as to which standards your companies or clients use, seek out your CAD system manager. Set up your units and tolerances at the beginning of your modeling session and save them in your usr_options file.

The maximum distance or gap between the surfaces of your model must be less then the accuracy defined within Pro/ENGINEER for successful joining of surfaces. The Pro/ENGINEER system defines accuracy as a value less then the ratio of the length of the smallest edge of a part divided by the length of the largest side of a part.

You can lower the part accuracy to successfully join surfaces when the gap exceeds the required tolerance. However, we recommend that your StudioTools models are constructed to within the accuracy defined by the engineering requirements of your organization.

A recommended tolerance to maintain during transfer from StudioTools to Pro/ENGINEER is dependent on the size of the part being described. StudioTools uses an absolute tolerance system to describe geometry which means that every piece of geometry in a particular wire file is built to plus or minus a given value (tolerance). Pro/ENGINEER uses a system of relative tolerance, referring to the fact that the acceptable gap between pieces of geometry is based on the relative size of the geometry.

The default accuracy in Pro/ENGINEER is set to .0012 and the range available is .01 to .0001. Using the default accuracy, the maximum allowable distance between two surfaces when the longest edge of the surface is five inches would be less than 5 * .0012 = .006 inches. You must create surfaces in StudioTools that adhere to this accuracy to be successful in creating a Pro/ENGINEER solid model.

Whether you are creating a model, verifying a model, or debugging a translation, there are a number of tools within StudioTools to check the quality of the geometry you have created. The most useful tool is the min/max measurement tool in the Locators menu. Use this tool to check the maximum distance between any two surface boundaries.

Get the topology right

The StudioTools stitching operation is recommended to be done on geometry being prepared for transfer to Pro/ENGINEER.

The **Surface Edit > Stitch> Shell Stitch** feature within StudioTools creates a valid solid model topology within the StudioTools modeling environment. The stitching of surfaces within StudioTools greatly improves the robustness of the interface to Pro/ENGINEER. The stitching process also identifies surface boundaries that exceed the prescribed tolerances. These problems can then be corrected by the designer prior to the translation of the data to Pro/ENGINEER.

In addition, the stitching process also identifies duplicate surfaces in the model and orients the surface normals of the completed shell.

When models are constructed using the advance surface tools (swept, Rail Surface and square) it is quite common to create a number of smaller surfaces along the edge of one larger surface. This modeling technique does not create the twin edges required for a solid model. The stitching feature will automatically create the twin edge topology required by Pro/ENGINEER.

One case that cannot be solved topologically is the closed or periodic surface (a primitive sphere is an example of a closed surface). The reason for this is that in most solid modelers, a

face cannot be joined to itself. The presence of closed or periodic geometry in StudioTools (not true for Granite) is another reason that geometry intended for transfer to Pro/ENGINEER must be stitched before export. Using stitch has the same effect as detaching the geometry to create two surfaces before writing the IGES or STEP file for Pro/ENGINEER.

To import StudioTools models into Pro/ENGINEER

Define Absolute Tolerance Process (STEP and IGES only)

Before importing any foreign geometry (such as models created in StudioTools) the Pro/ENGINEER user can change from the default **Relative Tolerancing process** to the **Absolute Tolerance process**. Before a foreign model (that is created anywhere other than Pro/ENGINEER) is imported into Pro/ENGINEER, the desired absolute tolerance can be set to the value that the incoming model was built to.

For example if the Curve Fit Distance in StudioTools was set to 0.002mm, then the Absolute Tolerance in the Pro/ENGINEER work session should be set to 0.002mm.

This option can be enabled by writing the line:`enable_absolute_accuracy yes` into the `config.pro` file of the working directory.

Once the option is enabled you must go to the Setup section of the Pro/ENGINEER application and choose Absolute Accuracy, and set the units and numerical value of the tolerance you wish to work at, each time a new part is created.

This is important to ensure that the StudioTools-created model can be used in further downstream operations in Pro/ENGINEER.

Advanced data sharing techniques

The following are some suggestions for StudioTools modeling that provide enhanced inter-operability with Pro/ENGINEER.

Export individual “Features” from StudioTools

Because Pro/ENGINEER creates each element of a model as a feature, it can be very useful to import components of the StudioTools model as individual export files that can be manipulated in Pro/ENGINEER as individual import Features. Major components of your StudioTools model can be transferred separately so that they can be used to construct individual features within Pro/ENGINEER.

The advantage of this technique is that individual features can be “reordered” in Pro/ENGINEER to give added flexibility to the engineer. The Feature > Reorder command allows the user of Pro/ENGINEER to modify the sequence feature construction. This is useful during the engineering process. Additional “mechanical” features are added to the industrial design model and the result is based on geometry previously created.

Surface replacement

The surface replacement technique can be very useful when the model is a mix of mechanical elements defined by an engineer and styling elements defined by an industrial designer. By replacing the styling elements of a Pro/ENGINEER model all of the parametric/feature information is retained for the mechanical elements. This allows for continued parametric editing, automatic dimensioning, and so on.

Exporting assemblies from Pro/ENGINEER in IGES format

When exporting assemblies from Pro/ENGINEER, there are four types of IGES output available: Flat, One Level, All Levels and All Parts.

Flat

With the Flat option, Pro/ENGINEER exports the entire geometry of the assembly in a single IGES file. All components are transformed into model space before exporting, but there will be no hierarchy contained in the IGES file. All geometry of the assembly exists in the Flat IGES file, and it will all be correctly positioned in model space.

To help organize these files that have the potential of being very large, place each instance of a part within an assembly into its own layer in Pro/ENGINEER before exporting the assembly as IGES Flat. The layers will be transferred as “levels” along with the assembly in the IGES file. The IGES levels are translated into ALIAS Sets. This means that the members of one set are all of the surfaces that comprise an instance of a part in the assembly.

The interface ID that was specified for each layer is the means by which Pro/ENGINEER layer information is transferred via IGES. IGES does not support names for layers. Layers in IGES are called “levels” and a level is identified by a number, not a name. This is why Pro/ENGINEER asks you to assign a number as well as a name to a created layer. The name is more useful within Pro/ENGINEER, but the number is important for data transfer.

When the IGES file is imported into StudioTools, the IGES level information is created as StudioTools Sets. To display the level information, go the Set Lister (Pick > from lister > SETS). You will notice there exists sets whose names have the format LEVEL#n, where n is the "interface id" that was specified in Pro/ENGINEER.

One level

Outputs an assembly IGES file with external references pointing to the IGES files of its components. This contains only top-level geometry.

All levels

With the All Levels option, Pro/ENGINEER outputs n+ 1 IGES files, assuming that there are n parts or sub-assemblies in the assembly. There will be one IGES file for each part or sub-assembly (for a total of n IGES files) and one master IGES files that contains external references (IGES entity 416 form 1) to the n-component IGES files. Each external reference to a component IGES file within the master IGES file is contained within an IGES Subfigure Definition (entity 308), which is instanced once by an IGES Subfigure Instance (entity 408). The model of each component referenced by the master IGES file is in definition space; that is, placed at the origin. Each component is transformed into model space via the

transformation contained in the Subfigure Instance (entity 408) in the master IGES file.

If the n-component IGES files are individually imported into StudioTools, the resulting model will be incorrect, since each component will be placed at the origin, rather than the correct spot in model space.

If the master IGES file is imported into StudioTools, there will be no model at all! This is because StudioTools does not support the IGES External Reference (416 form 1) entity. This entity is generally frowned upon because it contains the filename of the component IGES file, and filenames are generally not portable between operating systems (for example, Unix > DOS).

All parts

Outputs an assembly to IGES as multiple files containing geometry information of its components and assembly features. These parts use the same reference coordinate system to ease reassembly in the receiving system.

Detailed file format information

PTC Granite format (Windows Only) (page 116)

STEP format (page 121)

IGES format (page 106)

CATIA V4

StudioTools CATV4 DirectConnect is a stand-alone utility that allows the exchange of 3D model data between StudioTools and CATIA using the CATIA/StudioTools neutral format CAI.

CATIA Requirements

- Version 4.2n of CATIA
- SurfaceStudio, AutoStudio, Studio or DesignStudio.

Use the following summarized list of modeling practices discussed in this section as a quick reference guide if problems arise.

Before you create the model

- Units should be set to mm.

In the **Construction Settings** window:

- The **Rational** geometry flags should be toggled **OFF**.

Tolerances should be set as follows:

- **Curve Fit Distance** =.01 mm (lower as necessary)
- **Curve Fit Checkpoints** = 10
- **Max Gap Distance** = .01 mm (this value should remain the same as **Curve Fit Distance**)
- **Trim Curve Fit** =.005 mm (lower as necessary)

While you create the model

- Use degree 5 curves and surfaces to achieve curvature continuity between surfaces.
- Models should be transferred periodically from StudioTools to CATIA during construction to manage the quality of the model being created.
- Periodically stitch the geometry once it is in CATIA to ensure that the model meets all tolerance requirements.
- Avoid using **Object edit > Attach > Attach** since this function creates multiknots in StudioTools geometry.

- Avoid using **Surfaces > Skin** between trimmed surface boundaries, since excessive amounts of data are created in the resulting surface. If Skin is used between trim boundaries, the resulting surfaces should be checked for multiknots before export.
- Use surface building tools such as **Surfaces > Boundary Surfaces > Square** and **Surfaces > Swept surfaces > Rail surface** to ensure and control curvature continuity between surfaces.

Workflow

Before transferring geometry between StudioTools and CATIA, you should consider the purpose of the transfer to plan an appropriate workflow. Two common workflows are:

- Geometry (describing mechanical components) is transferred from CATIA to StudioTools as reference data for concept design surfacing, then the StudioTools model is transferred back to CATIA.
- A StudioTools model is transferred to CATIA, and both StudioTools and CATIA databases are developed independently. Later, the modified StudioTools model is transferred again to CATIA, replacing the StudioTools geometry from the first transfer. In this scenario, all work done in CATIA on the original StudioTools model (ribs, thickness) will be applied to the new modified StudioTools model.

There are many variations on these two examples. Whatever the transfer scenario, you should carefully plan the transfer process, to ensure that the appropriate data is written out and is useful.

What happens when you replace StudioTools geometry

A common workflow using StudioTools and CATIA together is one where you replace existing StudioTools geometry in a CATIA model file with updated StudioTools geometry. This workflow allows you to continue working in StudioTools, modifying a model that has already been passed over to CATIA.

When you want to update the CATIA database with the completed changes, the surfaces that have been modified are

passed to CATIA. You import the new StudioTools geometry and then redefine the skin that includes the faces in question using the **Limit2 > Skin > Create/Modify** tool.

If you want to make changes to a face or surface using StudioTools and then include that modified surface in the CATIA model, you only have to redefine the *skin* to its members. That is, this time you leave out the original face and include the new StudioTools-modified face. This way StudioTools geometry can be used to modify CATIA models at any point throughout the development cycle.

What are the curve to fit distance tolerances in StudioTools

The **Curve Fit Distance** is the tolerance to which trim boundaries are rebuilt to (or approximated). The default positional tolerance in CATIA is .1 mm, and the StudioTools **Curve Fit Distance** setting should be set to 0.01mm.

This **Curve Fit Distance** setting should normally be accurate. If you find that it is not resulting in StudioTools geometry that can be successfully used in CATIA, then experiment with the **Curve Fit Distance**—it can set to as low as 0.005 mm. This setting will enhance the success of post transfer processes, such as skinning, that are to be carried out once the geometry is in CATIA.

The **Curve Fit Distance** tolerance in StudioTools should not be set at less than 0.001 mm. Lower than this will impact processing time.

Whether you are creating a model, verifying a model, or debugging a translation, there are a number of tools in StudioTools you can use to check the quality of the geometry you have created. The most useful tool is the **Locators > Deviation** Min/max measurement tools. Use this tool to check the maximum distance between any two surface boundaries.

Detailed file format information

CAI format for CATIA V4 files (page 125)

CATIA V5

StudioTools CATIA V5 DirectConnect is a stand-alone utility that allows the exchange of 3D model data between StudioTools and CATIA V5 using the native CATIA part (.CATPart) and product (.CATProduct) documents. Please follow installation instructions provided in the README.txt file.

CATIA Requirements

- Version 5 of CATIA (Release 10, 11, 12, 13, 14, or 15) and an appropriate CATIA license.
- Version 13 of SurfaceStudio, AutoStudio, Studio or DesignStudio, and a CATIA V5 DirectConnect license.
- IRIX or Windows operating system.
- On IRIX, ftn_eoe *must* be installed for CATIA V5 DirectConnect to work:
  - ftn_eoe Standard Execution Environment (Fortran Headers and Libraries, 7.4)
  - ftn_eoe.sw.lib Standard Execution Libraries (N32bit)

See *Before you create the model* (page 24) for a list of modeling practices to use as a quick reference guide if problems arise.

If you have a network installation of CATIA V5, please see the installation notes (README.TXT) for important information about environment variables that must be set.

Before you create the model

- Units should be set to mm.

Optimal tolerances should be set as follows, as recommended by Dassault Systemes:

- **Curve Fit Distance** = 0.001 mm (lower as necessary)
- **Curve Fit Checkpoints** = 10
- **Max Gap Distance** = 0.01 mm
- **Trim Curve Fit** = 0.005 mm (lower as necessary)
- Topology Distance = 0.02 mm

While you create the model

- Models should be transferred periodically from StudioTools to CATIA during construction to manage the quality of the model being created.
- The StudioTools model should be capable of being successfully stitched before export. If you periodically stitch the geometry to ensure that the model meets all tolerance requirements, you'll have a good indication of whether the final model will stitch correctly.
- Avoid using **Object edit > Attach > Attach** since this function creates multiknots in StudioTools geometry that may result in unusable geometry in CATIA.
- Use surface building tools such as Square and Rail Surface, taking advantage of the **Boundary Rebuild** option to control curvature continuity between surfaces and ensure surfaces do not contain multi-knots.
- Use **Evaluate > Check model** to be alerted to potential problems: it's another good practice.

Workflow

Before transferring geometry between StudioTools and CATIA, you should consider the purpose of the transfer to plan an appropriate workflow. Two common workflows are:

- Geometry (describing mechanical components) is transferred from CATIA to StudioTools for concept design surfacing, then those Studio surfaces are transferred back to CATIA.
- A StudioTools model is transferred to CATIA, and both StudioTools and CATIA databases are developed independently. Later, the modified StudioTools model is transferred again to CATIA, replacing the StudioTools geometry from the first transfer. In this scenario, all work done in CATIA on the first StudioTools model transfer will affect the new, modified geometry.

There are many variations on these two examples. Whatever the transfer scenario, you should carefully plan the transfer process, to ensure that the appropriate data is written out and is useful.

What are the curve fit distance tolerances in StudioTools

The **Curve Fit Distance** is the tolerance to which trim boundaries are rebuilt to (or approximated). The default positional tolerance in CATIA V5 is 0.001 mm, and the StudioTools **Curve Fit Distance** setting should be set to 0.001mm.

This **Curve Fit Distance** setting should normally be accurate. If you find that it is not resulting in StudioTools geometry that can be successfully used in CATIA, then experiment with the **Curve Fit Distance**—it can set to as low as 0.001 mm. This setting will enhance the success of post transfer processes, such as skinning, that are to be carried out once the geometry is in CATIA.

The **Curve Fit Distance** tolerance in StudioTools should not be set at less than 0.001 mm. Lower than this will impact processing time.

Whether you are creating a model, verifying a model, or debugging a translation, there are a number of tools in StudioTools you can use to check the quality of the geometry you have created. The most useful tool is the **Locators > Deviation** Min/max measurement tools. Use this tool to check the maximum distance between any two surface boundaries.

I-deas NX series

You can intergrate StudioTools models into I-deas NX (UGS PLM solutions). Often the workflow will require that the designer send geometry over to the engineer using a CAD system. The CAD operator will then use the model to describe a solid part and perform other engineering process such as describing ribs, bosses and other mechanical details.

At any point the designer may need to update the information in the CAD system by re-exporting changes made to the original model in StudioTools so that those changes can be integrated into the CAD database.

MasterSeries Requirements

- I-deas NX 9, 10, or 11 series.
- I-deas DirectConnect
- SurfaceStudio, AutoStudio, Studio or DesignStudio.

Use the following summarized list of modeling practices as a quick reference guide to avoid data transfer problems.

Before you create the model

- **Units** should be set to mm.
- In the Construction Options window:
  - The **Modeling Modes** should be set to **NURBS**.
- Tolerances should be set to the following values
  - **Curve Fit Distance** = 0.005 mm
  - **Maximum Gap Distance** = 0.005 mm
  - **Curve Fit Checkpoints** = 10
  - **Trim Curve Fit** = 0.001 mm

While you create the model

- Use degree 5 curves and surfaces to achieve curvature continuity between surfaces.

  I-deas does not support degree 7 geometry. Geometry created in StudioTools which is degree 7 will be rebuilt to degree 3 (cubic) upon import to I-DEAS. Adjacent

StudioTools surfaces which had been built with continuity between them may no longer have that continuity after they are rebuilt in I-DEAS.

- Surfaces can be overbuilt and trimmed back before exporting. This will result in a greater success rate when the surface geometry is offset in I-DEAS.
- Models should be transferred periodically from StudioTools to Master Series during construction to manage the quality of the model being created.
- Avoid using **Surfaces > Skin** and **Object Edit > Patch** between trimmed surface boundaries since excessive amounts of data are created in the resulting surface.
- Use surface building tools such as Square and Rail Surface, taking advantage of the **Boundary Rebuild** option to control curvature continuity between surfaces and ensure surfaces do not contain multi-knots.

Geometry types exported to I-deas

The following geometry types can be exported to I-deas using I-deas DirectConnect:

- Single CVs (points)
- All curves (with or without attributes: lines, polylines, etc.)
- Faces
- Curves on Surface
- Surfaces
- Target Surfaces
- Trimmed Surfaces
- Trimmed Surfaces with multiple trim regions
- Target Trimmed Surfaces
- Shells

Polysets are the only geometry entities that are not supported by Master Series.

Workflow

The workflow of transferring data into I-DEAS NX series requires that the designer send geometry over to the engineer. The CAD operator will then use the model to describe a solid part and perform other engineering process such as describing ribs, bosses and other mechanical details.

At any point the designer may need to update the information in the CAD system by re-exporting changes made to the original model in StudioTools so that those changes can be integrated into the CAD database.

Shell imported geometry

Shelling, or creating a topological description from StudioTools models, is a common workflow in StudioTools and I-deas NX series. When exporting a model from StudioTools, you must ensure that the geometry is built to the correct tolerance and that it can be stitched. The stitching process in StudioTools identifies gaps between surfaces so that you can repair the appropriate geometry before exporting to I-deas NX series.

During stitching, the surfaces are twinned. This means that the surface boundaries may be split to accommodate adjacent surfaces, and periodic geometry is detached into multiple surfaces.

For this reason, you should save the StudioTools wire file before stitching so that if further modifications are required to be made to the StudioTools model, the construction history will be intact.

Stitched geometry saved to an StudioTools wire file cannot be unstitched to its original state.

Export features from StudioTools

The term *part* in I-deas NX series refers to geometry that has been saved out as a part. The term *feature* refers to any attribute that augments the basic shape of a part and distinguishes it from other parts that could be derived from the same basic shape. More precisely, features are objects whose key dimensions and orientations have been controlled,

thus allowing you to control your design. You can bring in parts or features to replace existing features on the workbench.

Since I-deas NX series creates each element of a model as a part or feature, it is useful to import components of the StudioTools model as individual StudioTools wire files that can be manipulated in I-deas NX series as individual features. Major components of the StudioTools model can be transferred separately so that they can be used to construct individual features. The advantage of this technique is that individual features can be “replaced” to give added flexibility.

Unigraphics

StudioTools Unigraphics DirectConnect is a stand-alone utility that allows the exchange of 3D model data between StudioTools and Unigraphics.

- **See movie:** unigraphics.rm

Unigraphics Requirements

- In order to use Alias UG DirectConnect, the Unigraphics installation must include one of the following licenced options:
  - Unigraphics NX Open API Execute or
  - Unigraphics NX Open Package Execute
- You must have access to Unigraphics software for the same platform on which you are running StudioTools.
- Unigraphics utilities supported now include versions 16-20, NX, and NX2 for AlToUG and UGToAl.
- SurfaceStudio, AutoStudio, Studio or DesignStudio.

Before you create the model

- Units should be set to mm.

In the **Construction Settings** window:

- The **Rational** geometry flag can be toggled **OFF**.

Tolerances should be set as follows:

- **Curve Fit Distance** = .01 mm (lower as necessary)
- **Curve Fit Checkpoints** = 10
- **Max Gap Distance** = .01 mm (this value should remain the same as **Curve Fit Distance**)
- **Trim Curve Fit** =.005 mm (lower as necessary)

While you create the model

- Use degree 5 curves and surfaces to achieve curvature continuity between surfaces and successful data transfer.
- Models should be transferred periodically from StudioTools to Unigraphics during construction to manage the quality of the model being created.

- The StudioTools model should be successfully stitched before export, but you should also periodically stitch the geometry to ensure that the model meets all tolerance requirements.
- Avoid using **Object edit > Attach > Attach** since this tool creates multiknots in StudioTools geometry.
- Avoid using **Surfaces > Skin** between trimmed surface boundaries, since excessive amounts of data are created in the resulting surface. If **Skin** is used between trim boundaries, the resulting surfaces should be checked for multiknots before export.
- Use surface building tools such as **Surfaces > Boundary Surfaces > Square** and **Surfaces > Swept surfaces > Rail surface** to ensure and control curvature continuity between surfaces.

Workflow

Before transferring geometry between StudioTools and Unigraphics, you should consider the purpose of the transfer to plan an appropriate workflow.

Two common workflows are:

- Geometry (describing mechanical components) is transferred from Unigraphics to StudioTools to be used as reference data for concept design surfacing, then the StudioTools surface model is transferred back to Unigraphics.
- A StudioTools model is transferred to Unigraphics, and both StudioTools and Unigraphics Databases are developed independently. Later, the modified StudioTools model is transferred again to Unigraphics, replacing the StudioTools geometry from the first transfer. In this scenario, all work done in Unigraphics on the first StudioTools model transfer will affect the new, modified geometry.

There are many variations on these two examples. Whatever the transfer scenario, you should carefully plan the transfer process, to ensure that the appropriate data is written out and is useful.

Detailed file format information

Unigraphics proprietary format (page 111)

Solid Imaging

Solid Imaging is a component of Rapid Prototyping which uses a database to translate three-dimensional geometry into physical models or parts using a variety of resins and other materials. The file formats used by StudioTools to output files for Rapid Prototyping are the STL and SLC.

Solid Imaging Requirements

NURBS surfaces must be translated into either the .stl or .slc format before reading the file into the solid imaging machine software.

Workflow

StudioTools wire files exist as NURBS data. To use that data to create physical models using solid imaging technologies, you must translate the NURBS to either the `.stl` or .slc format so that the geometry can be read by the solid imaging machine's software. Included in the list of solid imaging technologies is SLA (Stereolithography), SLS (Selective Laser Sintering), LOM (Laminated Object Manufacturing), SGC (Solid Ground Curing), FDM (Fused Deposition Modeling) and others.

Converting the StudioTools geometry to the .stl format or the `.slc` format can be done from within StudioTools.

Consult with the operator of the solid imaging machine to optimize the transfer of data.

STL Format

An.stl file is a tessellated file (binary or ASCII), which means the NURBS surface is described by a series of triangles. The resolution of this polygonized data base is defined in StudioTools by the subdivision characteristics of the original NURBS surface. Once the tessellated geometry is sent to the Solid Imaging technology, the geometry is sliced, and then those slices are used to describe the physical model that will be produced.

With STL as the transfer format, you can send geometry to most Solid Imaging technologies while controlling the resolution of the finished model.

The STL file exported from StudioTools conforms to 3D systems file format version 2.0. When you export a model as an STL file, StudioTools displays:

- A "solid check" is run-on the model and the results are displayed at the prompt line. This tessellation check determines if it is a valid solid watertight model or if it has any gaps indicating topological errors. This allows the StudioTools user to determine if the data being transferred can be used by the operator of the solid imaging (for example SLA) machine to build the part.

  If gaps are found, the user receives a warning indicating that it is an illegal solid and the number of free edges in the model. When you view the model, edges with gaps are highlighted in red so that you can easily identify where gaps are and then repair the surface model.
- Stitch and tesselation tolerances options, allow you to set the merge vertices tolerance, the maximum distance at which two vertices will be merged together into one.
- During tessellation, degenerate triangles (with two or more equal vertices) are removed and the normals of the triangles are recalculated.

SLC format

An.slc file (StereoLithography Contour) cuts 2D contours of the 3D data base. These contour lines are polylines. The advantage to using this file format is that the NURBs geometry description in StudioTools is directly sliced and therefore fewer iterations are required between the original geometry and the data sent to the Solid Imaging machine to be built.

SLC header information

The header section of the .slc file is an ASCII character string (up to 2048 bytes) containing global information about the model.

The output in the header provides the following information:

- SLC file format version number (-SLCVER2.0)
- Output units (-UNITS<INCH/MM>
- Type of model (-TYPE<PART/SUPPORT/WEB>)
- Vendor package and version number (which produce the SLC file (-PACKAGE ALIAS STUDIO V12.0)
- Calculations and sets from SLC output x,y,z extends of the model (-EXTENTS mx,Mx,my,My, mz,Mz)

Header keywords (CHORDDEV, ARCRES, SURFTOL, GAPTOL, MAXGAPFOUNS, EXTLWC, STHICK, STARD and ENDD) are set to 0.0.

TC VisProducts

TC Direct Connect is a real-time rendering solution based on OpenGL. It is a file-based translator used to bidirectionally convert StudioTools native format wire files to UG's native formatted DirectModel files in the Jupiter (.jt) file format.

Functionality of TC VisProducts

The following sections provide detailed descriptions of the functionality available in StudioTools's TC Direct Connect translator. At a high level, there are two interfaces available in TC Direct Connect. The software is offered either as a stand-alone command-line driven tool, or it can be launched from within StudioTools via the **File > Save As** ❐ > **TC VisProducts** menu. The TC software is designed to operate in one direction, converting StudioTools data to TC DirectModel files. (For import into StudioTools, please see the JT import translator.)

How the TC translator works

Hierarchy in StudioTools is represented by DAG node objects. The basic parent/child relationship is reflected through the group node, which refers to a list of child DAG nodes. This type of DAG node allows the hierarchical grouping of DAG nodes. A group node can share its list of children with another sibling group node.

The translator works by converting each DAG node object into the equivalent DirectModel node object. DirectModel can represent many of the same types of nodes that StudioTools supports. For example, StudioTools surfaces are individually tessellated into unique DirectModel parts. There should be a 1:1 correspondence between StudioTools surface names and DirectModel part names. If surfaces are grouped in StudioTools, they will also be grouped in DirectModel.

The TC translator is controlled through several possible methods. You have the choice of either an StudioTools plug-in, accessed within the **File > Save As** ❐ > **TC VisProducts** menu, or batch mode driven by a command-line. This gives you control over various tessellation parameters and other important options.

How do I?

How to import and export data from WebDAV environments.

Manage files through a WebDAV interface

WebDAV features in **File** operations, available on the Studio browser only.

Manage your digital assets in WebDAV

How to manage your files on a WebDAV server while working within the StudioTools environment.

Unlike local file storage, this function enables you to share and manage digital assets in a group environment. For more information about WebDAV and WebDAV standards, see http://www.webdav.org.

It allows you to:

- **Work in collaborative environments** giving access to existing Asset Management systems with WebDAV support.
- **Browse a WebDAV server** from within the StudioTools environment.
- **Access password protected directories** on a WebDAV server.
- **Create, Copy, Move,Delete and drag and drop** WebDAV resources from within the StudioTools environment.
- **Create and Rename** directories and files from within the StudioTools environment.
- **Drag and Drop** resources from a WebDAV server onto StudioTools to open the file.
- **Sort** resources by **File Type, Date modified, Size** or **Name**.
- Set up your **proxy server** workgroup for use from behind a firewall.
- In order to use WebDAV functionality, you must use the StudioTools browser (set in **Preferences > General Preferences**, in the **System** section).

To browse a WebDAV server

From within the StudioTools environment, browse resources on a WebDAV server.

1 Choose **Preferences > General Preferences**, go to the **System** section, and set **Type of File Browser** to **StudioTools**.

The Windows browser does not support WebDAV servers.

2 Choose **File > Open**.

3 Type the URL location of your server

`http:// WebDAV_server_name/`

or, if in a secure environment

`https://WebDAV__server_name/passwordProtected/`

Spaces in URLs are permitted in the StudioTools browser.

To access a password protected directory

Use HTTP calls to access secure files and launch the security detection system to scan a request from user's computer.

1 Locate the secure area on your WebDAV server.

`https://WebDAV__server_name/passwordProtected/`

2 Enter the required user authentication information.

User Name:

Password:

To copy a WebDAV resource

Download a resource to and from a WebDAV server to a client machine.

1 Highlight the file you wish to copy.

2 Choose **File > Copy.**

Authentication (for example, user name and password) must be the same on the source and destination URLs.

To move a WebDAV resource

Drag and drop the copied file into the new directory. From within the StudioTools browser move files around in a WYSIWYG environment.

1 Highlight the file and choose **File > Copy**.

2 With the ***left mouse button*** held down drag the file to a new directory.

3 Release the mouse button. The file is moved.

Authentication (for example, user name and password) must be the same on the source and destination URLs.

To delete a WebDAV resource

From within the StudioTools environment learn how to delete a resource.

1 Highlight the file.

2 Choose **File > Delete**.

To drag and drop a WebDAV resource

Drag and Drop resources from a WebDAV server onto StudioTools to open the file.

1 Locate and hightlight the wire file you wish to work in StudioTools

2 Drag and drop this file on StudioTools which is already running on your desktop.

StudioTools will then open that file.

To sort WebDAV resources

Sort files by **file type, date modified, size** and **name**.

1 Choose the directory you wish to sort and use the **Sort** pop-up menu.

2 Choose to sort your directory list by **File Type, Date modified, Size** or **Name**.

To save a file as a WebDAV resource

1 In StudioTools, choose **File > Save**.

2 Type or browse to the directory to which you want to save the resource.

3 Double-check the name of the wire file and click **Save**.

To save a new file as a WebDAV resource

1 In StudioTools, choose **File > Save As**.

2 Type or browse to the directory to which you want to save the resource.

3 Type a name for the wire file and click **Save**.

To save a WebDAV resource with another name

1 In StudioTools, choose **File > Save As**.

2 Type or browse to the directory to which you want to save the resource.

3 Backspace or ***Delete*** the text in the **Name** field, type a new name for the wire file and click **Save**.

To export a WebDAV resource to another file format

1 In StudioTools, choose a submenu item from the **File > Export** menu.

2 Type or browse to the directory in which you want to save the resource.

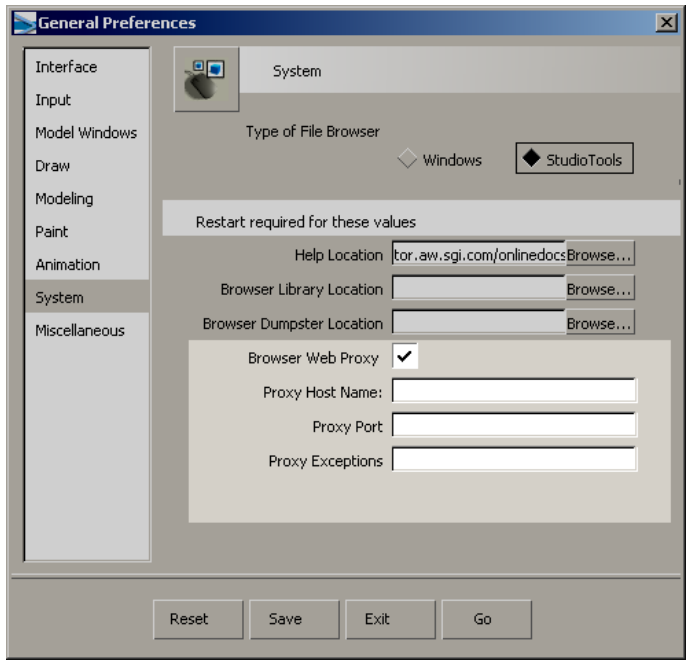
3 Either delete and type a new name for the file, or use the existing file name, and click **Save**.

To set up proxy servers for WebDAV

To access external servers which have different proxy settings.

Add proxy addresses, ports and exceptions listing internal servers to enable access to servers.

1 Select **Preferences > General Preferences**.



2 In the **File System** section, check **Browser Web Proxy**.

You may need to contact your system administrator for in formation to fill in the following fields.

3 Type the **Proxy Host Name** of the proxy server your company uses to allow users to browse outside their firewall.

4 Type the **Proxy Port** for this host name.

5 Type the **Proxy Exceptions** of internal servers used within your company. Separate entries with a semicolon.

6 Select **Save**.

Import CAD data files

How to import CAD data types into StudioTools.

Import a data file

How to import a number of different file formats.

- Use **File > Import > File** ❐

  or

- Use **File > Open** ❐

  Choose the appropriate file format type.

Import Illustrator files

How to import the geometry of an Illustrator file.

Only up to Illustrator version 8 is supported.

To import an illustrator file

1 Choose **File > Import > File** and set the **Illustrator options**.

By setting Illustrator options, you can enable StudioTools to import the geometry as a **Group** of curves or as individual curves. You can also **Scale** the image before importing it into StudioTools.

Open a Binary STL file

How to open a Binary STL file.

Choose **File > Open** to open a binary STL file. The STL file opens as a mesh in StudioTools.

The STL file is a unitless format. Set the units in **Preferences > Construction options** before importing the file.

Import Cloud data

Learn how to import point cloud data from a file or directly from EvalViewer.

Some cloud data is available from your own scanner, the network, DAT tape, or CD-ROM. StudioTools can import Cyberware point cloud files.

To import point clouds from storage

1 Choose **File > Import > Cloud** from the menus.

The file requester appears.

2 Type the path to the cloud point file you want to import.

or

Use the file list to browse through directories, then double-click the icon of the point cloud file you want to import.

To import point clouds directly from EvalViewer

1 In EvalViewer, choose **Edit > Copy** from the menus.

2 Switch to StudioTools, and choose **Edit > Paste** from the menus.

You may want to copy or link your scanner data directory to the relevant directory inside the **`cloud`** directory of your project.

For example:

- To copy Cyberware data to your StudioTools project directory from your current working directory, type the following in a Unix Shell window:

```
cp * ~/user_data/projdir/cloud/cyberware/
```

- To link Steinbichler data on a CD-ROM into your StudioTools project directory, type the following:

```
mv ~/user_data/projdir/cloud/steinbichler
~/save.sb
ln -s /CDROM ~/user_data/projdir/cloud/
steinbichler Options
```

Import a PTC Granite or a Pro/ENGINEER part file

How to import PTC Granite files or a Pro/ENGINEER part file.

When importing geometry from Pro/ENGINEER to StudioTools, you should:

- Use menu **File > Open** to read the Pro/ENGINEER part file (.prt) or the Granite file (.g). The translator will be automatically launched.
- Review the **File > Open** **PTC Granite** import options.

Import from Pro/ENGINEER into StudioTools via Pro/RENDER

How to import Pro/ENGINEER render format files.

To convert Pro/ENGINEER Render format files into StudioTools wire files.

1 Open a Unix shell.

2 Type utility the help command:

```
PRenderToAlias -h
```

3 Choose from the following options to write your command.

```
PRenderToAlias

Arguments: [<options>] [<infile> [<outfile>]]

Options:

-s

scale Input scale factor (for example. -s2.0)

-u

Input units. Acceptable values are
MI,FT,IN,MIL,UIN,KM,M,CM,MM,UM]. The default is
-uIN

-g

Group the geometry

-n

Do not merge vertices

-r

Recalculate vertex normals

-p

Merge vertices according to xyz position only

-t pos_tol

Specify merge vertices position tolerance value in
input units. The default is 0.0001.

-a nrm_tol.

Specify merge vertices normal tolerance value in
degrees. The default is 1 degree.

Where:
```

<infile> is a Pro/ENGINEER Render file. If it is absent, input comes from stdin.

<outfile> is an Alias wire file. If it is absent, output goes to stdout.

By default all vertices are merged according to their positions and normals.

Examples

```
PRenderToAlias crankshaft.slp crankshaft.wire
```

```
PRenderToAlias -g -t0.001 hammer.slp hammer.wire
```

Pro/ENGINEER options for export and import of STEP or IGES files

To make the import and export of IGES and STEP files easier for StudioTools to read.

1 In Pro/ENGINEER, select **Utilities > Options**

2 Add the following data to the configuration file.

```
IGES_OUT_ALL_SRFS_AS 128
IGES_OUT_SPL_CRVS_AS_126 YES
IGES_OUT_SPL_SRFS_AS_128 YES
IGES_OUT_MIL_D_28000 NO
IGES_OUT_TRM_SRFS_AS_143 NO
IGES_OUT_TRIM_CURVE_DEVIATION DEFAULT
INTF_OUT_BLANKED_ENTITIES NO
INTF3D_OUT_EXTEND_SURFACE YES
INTF3D_OUT_FORCE_SURF_NORMALS YES
IGES_IN_106_F2_AS_SPLINE NO
IGES_IN_DWG_LINE_FONT YES
IGES_IN_DWG_PNT_ENT YES
IGES_IN_DWG_COLOR YES
FIX_BOUNDARIES_ON_IMPORT YES
```

Import CATIA V4 into StudioTools

How to import a CATIA V4 file and review the mapping process used for geometry types and non-geometric data.

1 Choose **File > Import > File** ❐to display the basic file format options.

2 Choose File format **CATIA V4**

What happens on import of a CATIA file into StudioTools?

The model's surface geometry is converted in StudioTools into a *skin* by joining surfaces and trimmed surfaces or faces. The skin can then be used for a variety of downstream processes in CATIA.

If the StudioTools geometry is exported as a StudioTools *shell*, CATIA automatically creates a *skin* from it. In other words, the successfully imported CATIA geometry is made up of surfaces and faces, as well as a CATIA skin. This significantly reduces the amount of time the CATIA operator must spend preprocessing the StudioTools model to be used in CATIA.

Import a CATIA V5 file into StudioTools

To open a CATIA V5 file, choose **File > Open** from within StudioTools.

You can open either part (.CATPart) or product (.CATProduct) documents in StudioTools simply by clicking on the file name in the file browser.

If you get “File not recognised” errors on your CATIA part file, you can either use drag or drop with Studio or use the stand-alone utility to convert the file.

One option is available for importing CATIA V5 document files:

Auto Stitch

Click this option on to convert a CATIA V5 Solid or closed Shell to a shell in StudioTools; if this option is off, a grouped set of trimmed surfaces are created instead.

The following CATIA V5 entities are brought into StudioTools:

CATIA V5 entity	brought into StudioTools as
Curves	NURBS curve of appropriate degree.
SplineCurve	
Circle	
Ellipse	
Conic	
Hyperbola	
Parabola	
Line	
IntCurve	
BsplineCurve	
MergedCurve	
SimCurve	
HelixCurve	
PLine	
PSpline	
PCircle	
PEllipse	
PParabola	
PHyperbola	
PNurbs	
Surfaces	NURBS surfaces of appropriate degree
Products	Groups and instanced groups.
Solids	Set of faces that have shared edges at the boundaries where the faces meet within tolerance. Brought in as NURBS surfaces. Closed shell surfaces are stiched in a shell (if Auto Stitch is ON); otherwise, a group of trimmed surfaces is created.

CATIA V5 entity	brought into StudioTools as
Color RGB, Opacity, Layer, Show (Visibility), Pick (Template)	Shader Color RGB, Transparency, Layer, Visibility, Template.
Layer	By default, Layer. Caution: no names are used, just layer numbers. If the option “Import by Layer” (-a option for the stand-alone utility) is ON, a layer is created for each surfacic body (GSMTool feature) with the appropriate name.
Point	SpacePoint or curve, according to the option import by construction entity (-c option)
Construction plane	Construction plane or planar surface, according to the option import by construction entity (-c option).
GSMTool (feature)	A open body or geometrical set is imported as a layer in StudioTools if the -a option is ON.

CATIA V5 entity	brought into StudioTools as
Surface Patch	NURBS Surface of appropriate degree
Surfaces	
Plane	
NurbsSurface	
Cylinder	
Sphere	
Torus	
Cone	
FilletSurface	
ChamferSurface	
DraftSurface	
OffsetSurface	
SweepSurface	
GenericFillet	
GenericRuledSurface	
Tabulated Cylinder	
RevolutionSurface	
BsplineSurface	
CircularSweep	
RuledSurface	
Pickable	template
Invisible entities Noshow	Invisible entities, unless the option -v is used when importing the file.
Invisible datum entities	Ignored by default. A geometric element with no parents that is autonomous and does not depend on other specifications is a datum. Invisible datum objects are processed and converted if the datum option -d is ON.

Import a Unigraphics file into StudioTools

How to import Unigraphics files.

1 Choose **File > Import > File** ❐

2 Choose Unigraphics file format options.

What happens on import of a Unigraphics file into StudioTools?

When importing a Unigraphics file into StudioTools watch out for the following:

- You must have access to Unigraphics software for the same platform on which you are running StudioTools.
- StudioTools supports Unigraphics NX 2.0 and 3.0.

Export CAD data files

How to export CAD data types from StudioTools.

Export a data file

How to export a number of different file formats.

To export a file

- Use **File > Export > Active as** ❐

  or

- Use **File > Save as** ❐

  Choose the appropriate file format type.

Add a description to an export file

How to attach a comment to your export file.

To include a comment to your file

1 Choose **Include Comments** from the **Save Options** window.

2 Choose **Edit File Comments**, a shell will be displayed in which you can create or update the comment.

There are no limits to the number of characters per comment line because **Edit Comment** uses a user-defined editor.

The file comment that is exported from StudioTools will be altered to conform to these restrictions. That is, lines over 70 characters in length will be wrapped, and the comment will be truncated at 500 lines.

Check for gaps before exporting to a solid model

How to stitch the surfaces before exporting your model to a wire file and replace surfaces in I-DEAS MasterSeries.

To check stitching of surfaces

1. Stitch the surfaces using **Surface Edit > Stitch > Shell stitch** (with **Keep Originals** on) and carry out a visual check to confirm there are no gaps. If the model that has been stitched is intended to describe a closed volume, check the stitched geometries using **Object edit > Query edit**.

2. Delete the shell and repair the NURBS model.

3. If required, stitch the surfaces again and export the stitched object to a wire file. Also, save the NURBS model for further modeling operations.

For more information see *Edges do not match* (page 85).

Export an Illustrator file

How to export an Illustrator file from StudioTools.


Only up to Illustrator version 8 is supported.

To save an Illustrator file

1 Choose **File > Print setup.**

2 Set **Output** to **File.**

3 Set **Output Style** to **Illustrator**.

The output transfers the paint and shapes information. The results of opening this file in Illustrator is a display of two objects:

- an image plane (rendered image).
- the shapes of the model which can be individually selected and manipulated.

4 Close the Print Setup window and print the file.

Use the tolerances required by a specific CAD package

How to use tolerance presets.

Use **Preferences > Construction options** to set the tolerances enforced by the modeling tools. The requirements for transfer to specific CAD packages are available as presets.

To set up tolerances for transfer to a specific CAD package

1. Choose **Preferences > Construction options**.
2. Open the **Construction Presets** section.
3. Choose one of the settings profiles from the list:

- Click the CAD software you want to model for. This sets the construction options to values needed for maximum compatibility with that CAD package.
- Click General CAD Settings if the CAD package that you want to model for is not on the list. This sets the construction options to generic CAD values that will be compatible with most CAD packages.
- Click User Defined to set the construction options manually.

You cannot edit the presets of the CAD packages in the list directly. Choose the name of the CAD package and click Copy to create a new profile you can edit.

To create a set of custom tolerance setting

1. Open the **Construction Presets** section of the **Construction Options** window.
2. Choose one of the settings profiles from the list:

   If the settings you want to add are very similar to one of the preset CAD packages, click the name of the CAD package.

   Otherwise, click **User Defined**.
3. Click **Copy**.

4 Double click the new profile to rename it.

Export STEP/IGES/PTC Granite to Pro/ENGINEER

StudioTools surface models can be transferred to Pro/ENGINEER in either IGES, STEP or PTC Granite formats.

By using StudioTools, there are two types of model information that can be set to and read by Pro/ENGINEER: geometry information and topological information.

The StudioTools IGES file includes only the description of the geometry information. The STEP entities that StudioTools supports means that both the geometry information as well as the topology information can be transferred and read into Pro/ENGINEER. The geometric data describes the basic shape of the object and in both StudioTools and Pro/ENGINEER, geometric data is represented using NURBS.

Topological data describes how the geometric components are connected together to form a solid. The STEP file format has advantages over IGES when transferring StudioTools models to Pro/ENGINEER because there is more information describing the model that is being transferred.

Workflow for exporting PTC Granite files

When exporting PTC Granite geometry from StudioTools to Pro/ENGINEER, you should:

- Set **Preferences > Construction options Construction Presets** to **Pro/ENGINEER** *before* creating a model in StudioTools for Pro/ENGINEER.
- Use StudioTools, choose **File > Save as,** with **File Format Option** set to **PTC Granite**, to save a model,

  or

  Use StudioTools choose **File > Export- > Active as**, with **File Format Option** set to **PTC Granite**, to save selected parts of the model.
- Make sure the units system's settings in both Pro/ENGINEER and StudioTools are set to **millimeters** as the default.

- Make sure the model part files do not have short edges (shorter than curve-fit tolerance) before importing the PTC Granite file into Pro/ENGINEER. To check, use **Evaluate > Check model** and select the option **Model Parameters > short edges**.
- Try to remove all surface multi-knots from your StudioTools model before saving as a PTC Granite file. Multi-knot PTC Granite files may not be read into Pro/ENGINEER with proper trimmed edges.
- Save translation time and make the transfer to PTC Granite format faster by splitting closed surfaces, such as cylinder, torus, sphere and cone, into two or four parts before saving. The Pro/ENGINEER translator will split closed surfaces automatically but if this procedure is done before it will save time.

To create a PTC Granite file for Pro/ENGINEER

1. Choose **Surface Edit > Stitch > Shell stitch** to ensure you have created a closed volume before exporting it in PTC Granite format.
2. Choose the object to be exported so that it is the “active” object.
3. Choose **File > Export > Active as**.
4. Choose **PTC Granite** from the File Format pop-down menu.
5. Choose the **Save** icon and specify the PTC Granite file name. StudioTools automatically appends the `.g` extension to the file name. This file is now ready to be read into Pro/ENGINEER.

For more information refer to *Workflow* (page 22).

To export PTC Granite files

1. Choose the object to be exported so that it is the "active" object.
2. From the File menu, choose **File > Export > Active as**.

3 Choose **PTC Granite** from the **Basic Save Options File Format** pop-down menu.

To create an IGES format file for Pro/ENGINEER

1 Pick the object to be exported so that it is the "active" object.

2 Choose **File > Export > Active as**.

3 Choose **IGES** from the File Format pop-down menu.

4 Choose **pro_engineer from the IGES** by Vendor browser.

The configuration of the output variables for Pro/ENGINEER have been determined through data exchange testing to provide the best conditions for the creation of a solid model within Pro/ENGINEER. You can alter the output variables to achieve different results.

5 Choose the **Save** icon and specify the IGES file name. StudioTools automatically appends the `.igs` extension to the file name.

The file is now ready to be read into Pro/ENGINEER.

To create a STEP file in StudioTools for Pro/ENGINEER

Follow these steps to create a STEP format file for Pro/ENGINEER:

1 Select the object to be exported so that it is the “active” object.

2 From the File menu, choose **File > Export > Active as**.

3 Choose **STEP** from the File Format pop-down menu.

4 Choose **AP214** from the Application Protocol.

5 The StudioTools model should have already been stitched into a solid shell (volume) or a shell (group of stitched surfaces that do not describe a volume.

6 Choose under the **Model Type** either:

Manifold Shells- A collection of stitched surfaces that do not describe a volume is saved out as a G3 Manifold Shell.

or

Brep Solids-Stitched geometry that describes a closed volume is written out as a G5 Brep Solid.

7 Choose the **Save** icon and specify the STEP file name. StudioTools automatically appends the `.stp` extension to the file name. This file is now ready to be read into Pro/ENGINEER.

To set options in Pro/ENGINEER for export and import

To make the import and export of IGES and STEP files easier for StudioTools to read.

1 In Pro/ENGINEER, select **Utilities > Options**

2 Add the following data to the configuration file.

```
IGES_OUT_ALL_SRFS_AS 128
IGES_OUT_SPL_CRVS_AS_126 YES
IGES_OUT_SPL_SRFS_AS_128 YES
IGES_OUT_MIL_D_28000 NO
IGES_OUT_TRM_SRFS_AS_143 NO
IGES_OUT_TRIM_CURVE_DEVIATION DEFAULT
INTF_OUT_BLANKED_ENTITIES NO
INTF3D_OUT_EXTEND_SURFACE YES
INTF3D_OUT_FORCE_SURF_NORMALS YES
IGES_IN_106_F2_AS_SPLINE NO
IGES_IN_DWG_LINE_FONT YES
IGES_IN_DWG_PNT_ENT YES
IGES_IN_DWG_COLOR YES
FIX_BOUNDARIES_ON_IMPORT YES
```

Export STL files

How to create a solid model for STL export and save it in an STL file.

To export a model to STL format, it must be either a mesh object, or a stitched shell.

- See *Meshes* on page 56.

To stitch your model

1 Stitch the StudioTools model to create a closed volume shell before exporting an `.stl` file. Use **Surface Edit > Stitch > Shell stitch** with **Keep Originals** on.

For best results in creating an `.stl` file, ensure that the geometry is stitched to create a closed volume. If there are gaps in the shell, displayed in the geometry window indicated by yellow lines, the user can potentially still use that shell to successfully create a solid volume `.stl` file, but trying to use an open shell to create an .stl file is not recommended.

The closer the stitched NURBS representation is to defining a closed volume, the more likely that model can be used to define a closed volume (`.stl` file).

To check for a closed volume

1 Choose **Object edit > Query edit** and use the ***right mouse button*** to display the status of the shell. If the Close Volume equals true, the geometry can be used to create an STL file.

Shell Tessellation Parameters

The two **File > Export > STL** tolerance settings are used for exporting shells (*not* meshes) and are intended to be adjusted in order: **Tessellation Tolerance** first, then **Merge Vertices Tolerance**. In each case, if gaps are found, the user receives a warning indicating that it is an illegal solid. The number of free edges found in your model is displayed in this warning. When you view your model, boundary edges with gaps are highlighted in red to allow you to easily identify where they are located. This will enable you to repair your surface model quickly.

To export shells in STL files

1 Choose **File > Export > STL** ❐

The **Export STL Settings** window is displayed and a solid volume check routine is done. If the shell is not a closed volume, the prompt line will display the number of gaps that exist. In the geometry window, the model will display red lines highlighting the boundaries where the gaps occur to allow the user to quickly identify where the problems are. If desired, the export `.stl` command can be aborted and the user can repair the surface model before starting the operation again

2 Adjust the **Tesselation Tolerance** value. This value controls the number of triangles on the tessellated shell. The **Tesselation Tolerance** describes the maximum difference (in centimeters) between the original shell and the tessellated shell output to the STL file. The lower the tolerance value, the greater the number of triangles. It is important to consider that the **Tesselation Tolerance** will be reflected in the resolution of the physical model that will be produced by the solid imaging technology.

The slider range is 0.01 to 1. The default value is 1.00.

3 Adjust the **Merge Vertices Tolerance** value. This value controls the tolerances within which the vertices of the tessellated mesh will be considered to be coincident. By raising this value, the user is broadening the distance within which vertices will be considered coincident. The slider range is 0.0001 to 10. The default value is 0.001.

The Merge Vertices Tolerance should always be small compared to the Tessellation Tolerance to avoid collapsing triangles.

4 Changing the values in the option box causes the solid volume check to be run, and the results of this calculation are displayed in the Prompt History line. If a closed volume .stl file has been defined successfully, the user will choose the **Accept** button and save the .stl file to the appropriate directory.

If your tessellated geometry still has gaps and you choose **Accept**, a warning will appear again telling you how many free edges were found.

To export meshes in STL files

1. Select one or more mesh objects.
2. Choose **File > Export > STL**.

   The **Accept** button appears in the lower right corner.
3. Click **Accept**.

   The **Export STL** browser is displayed.
4. Choose a name for the file and click **Save**. Only the selected meshes are saved.

Export SDL files

To save out a Scene Description Language (SDL) file.

To export an SDL file

1 Choose **File > Export > SDL**.

The File Requestor appears.

2 Type the full path and file name for the SDL file in the File Requestor, or click **Show List** and select the file using the File Lister. If you select a file using the **File Lister**, the SDL file will overwrite that file.

3 Click **Save SDL**.

To cancel the process, press Esc.

When exporting a SDL file

- Delete all non-referenced shaders before exporting an SDL file, because all shaders, whether they are actually assigned to surfaces or not, are written out to the SDL file.
- When you export an SDL file, each perspective window will generate an image (or series of images) when the SDL file is rendered, because each perspective window has an associated camera.

  The file name you use for the SDL file is also used within the SDL file to specify the output image file name. For example, if the SDL file is named Planet, then the camera section of the SDL file will contain:

  pix = "pix/Planet",

  If there is a second perspective camera named camera2, the SDL file will also contain:

  pix ="pix/Planet_camera2",

  If you have more than one perspective window, but only want to render an image (or series of images) from one of them, either edit the SDL file, or pick all objects, lights, and only one camera, and then choose **File > Export > Active as**, and use that file to export an SDL file from.
- By default, comments are not included in the SDL file. If you want comments included, put the following line in your shell before starting StudioTools:

setenv ALIAS_SDL_LONGFORM 1

To edit an SDL file

To edit an SDL file, you must first convert it to ASCII format (text), using the Alias Binary SDL command-line utility (**bsdl**). This utility allows you to extract the text component from a binary SDL file, and replace the text component in a binary SDL file. This is useful if you need to hand-edit an SDL file.

SDL files now contain a binary component for large mesh files, because writing the data out as text would increase the size of the SDL file to an extent that parsing time by the renderer would also increase by a very large amount.

bsdl usage:

```
bsdl extract [-f] <text> <binary>
```

Extract ASCII SDL to <text> from a binary SDL file called <binary>. The file <text> will not be overwritten unless the option `-f` is specified.

Any text editor can then be used to edit the <text> ASCII SDL file.

```
bsdl replace <text> <binary>
```

Insert the ASCII SDL file <text> into the binary SDL file <binary>, replacing the ASCII SDL data.

For more information on the formatting of SDL files, see *SDL.html* on the *Technical* reference page.

Export to CATIA V4

How to write out to the CATIA V4 file format.

Workflow

Create a ‘skin’ and ’offsetting’ to create solid geometry from imported StudioTools models.

Stitched geometry saved to a StudioTools wire file cannot be unstitched to its original state.

1. Stitch the surfaces to perform a visual check to confirm there are no gaps.
2. Undo the stitch operation.
3. Save the StudioTools wire file.
4. Stitch the surfaces again.
5. Export the stitched object to a CAI file.

Geometry not stitched prior to export from StudioTools will not automatically create a *skin* upon import into CATIA.

Review the summary of modeling practices as a quick reference guide if problems arise.
See *CATIA Requirements* (page 21)

Export to CATIA V5

Choose **File > Save As** ❑. In the **Save As** option window, choose **CATIA V5** from the list of file formats.

The following Studio entities are saved into CATIA V5:

Studio entity	saved to CATIA V5 format as
Curve (created with either CVs or Edit Points)	NonRationalNURBSCurve of appropriate degree
Line	Line
Arc	RationalNURBSCurve
Circular	RationalNURBSCurve. Note that 360 degree periodical objects will be divided into two if the -c option is used.
Blend curves	NonRationalNURBSCurve of appropriate degree.
Multiknot curve	NonRationalNURBSCurve
Curve on surface	NURBSCurves
Surface	NonRationalNURBSSurface.
Surface Revolve	RationalNURBSSurface. 360 degree periodical objects will be divided into two if the -c option is used.
Multiknot Surface	Surfaces divided in multiple surfaces if -d option is used.
Shell	Solid. Caution: Alias closed shell becomes a CATIA Solid. If -s option is specified, the shell's faces will become surfaces.
sphere, cone, cube, etc.	NonRationalNURBSSurface. 360 degree periodical objects are divided into two if -c option is used.

Studio entity	saved to CATIA V5 format as
Rational sphere, cone, etc.	RationalNURBSSurface. 360 degree periodical objects are divided into two if -c option is used.
Shell	CATIA solid body, if shell is closed, or open if Join or Heal is ON.
Surface with multiple disjoint trimmed regions	Multiple trimmed surfaces
Polyset, mesh	Each polygonal face is converted to a surface.
Closed periodic curve	2 curves each
Shader Color RGB, Transparency	Color RGB, Opacity.
Layer	Layer (numbered). Layer name is ignored. If -a option is ON (Export by Layer) a GSMTool (surfacic) body is created for each layer with its name.
Invisible	Imported as Invisible (hidden)
Construction vector	Line
Template	Pickable
Construction planes	Planes
Symmetry	Converted if -b option for layer symmetry is ON.
Space point, Curve point, Surface point, Curve-on-surface point	Point

Troubleshoot

How to interpret data transfer log files.

Overview

Log files contains information on entity mappings, entity counts and the errors and warnings given when transferring data. Use these files as tools to assist in troubleshooting the data transfer.

- Translation log files are produced during both import and export procedures.
- Translation log files are saved in the same directory as your model file.
- Translation log files contain entity mappings, entity counts, error and warning messages and other helpful information.

StudioTools provides utilities which are also useful in troubleshooting. For more details please refer to our *Utilities* online manual.

Edges do not match

All surface edges must be topologically matched before the geometry meets the requirements of topologically described geometry in a CAD system.

Edges fail to match because of a mismatch in the topology of two or more adjacent faces, or because the maximum distance between two adjacent edges is greater than the current accuracy defined in the CAD system. There may be a gap existing between the adjacent surface edges, or the edges may overlap to the extent that they can not be stitched.

- You can fix this problem by lowering tolerances in StudioTools and recreating the StudioTools geometry before re-importing the file into the CAD system.

View and interpret data transfer log files

Open a log file and interpret the data. Review this data to resolve problems in data transfer.

Log file names

Learn the default file naming convention used in generating data transfer log files.

In most cases translators automatically create log files using a user-defined filename with a default filename extension. When storing format files the default file name extensions are as follows:

For IGES

Import translation

```
.alias2iges (filename.alias2iges)
```

Export translation

```
.iges2alias (filename.iges2alias)
```

For VDAIS

Import translation

```
.alias2vdais (filename.alias2vdais)
```

Export translation

```
.vdais2alias (filename.vdais2alias)
```

For C4

Import translation

```
.alias2c4x (filename.alias2c4x)
```

Export translation

```
.c4x2alias  (filename .c4x2alias )
```

For Unigraphics

Import translation

```
.alias2unigraphics (filename
.alias2unigraphics)
```

Export translation

```
.unigraphics2alias (filename
.unigraphics2alias)
```

For JAMA-IS

Import translation

```
.alias2jamais (filename.alias2jamais)
```

Export translation

```
.jamais2alias (filename .jamais2alias)
```

For Granite

Import translation

```
.alias2granite  (filename .alias2granite )
```

Export translation

```
.granite2alias(filename .granite2alias)
```

For STEP

Import translation

```
.alias2step (filename.alias2step)
```

Export translation

```
.step2alias (filename step2alias)
```

For CAI

Import translation

```
.alias2catia (filename.alias2catia)
```

Export translation

```
.catia2alias (filename .catia2alias)
```

For TC VisProducts

Import translation(only)

```
.alias2jt (filename.alias2jt)
```

View STEP log files

How to view log files.

To view log files

1 Choose **Utilities > Errlog** to view these messages.

When retrieval is finished without errors, the following message is displayed:

```
STEP files retrieved successfully.
```

If there are errors in the retrieval, this message is displayed:

```
Problem Reading Step File, refer to log
file and/or errlog for details.
```

The StudioTools errlog file and STEP logfile contain a specific error message for each problem entity that is encountered, along with the entity's STEP ID and type.

Interpret CATIA log files

How to interpret a translation log file.

Translation log file

The translation log file is produced during both import and export of CATIA files. This log file contains entity mappings, entity counts, and Information, Error, and Warning messages.

Interpreting the import log file

On import of a CATIA file, a translation log file is created in the same directory as the CATIA file. The file name of the log file is the same as that of the CATIA file with the extension.catia2alias added to it.

Log file

The log file contains:

- a translation summary,
- the time of translation,
- conversion statistics, and
- translation time.

If an error or problem occurs while processing an entity, the CATIA entity's type name, user identifier name, and instance name (STEP file line number) are printed, followed by the error or warning message.

Extended log file

If you have chosen to have an extended log file produced, then the log file will contain the same information as the log file, with the addition of:

- CATIA entity name to Alias entity name mapping
- CATIA entity type to Alias entity type mapping

Interpreting the export log file

On export of a CATIA file, a translation log file is created in the same directory as the CATIA file. The file name of the log file is the same as the CATIA file with the extension .alias2catia added to it.

Log file

If you have chosen to have a log file via the interface options, the log file will contain:

- a translation summary,
- the time of translation,
- conversion statistics, and
- translation time.

Extended log file

If you have chosen to have an extended log file produced, then the extended log file will contain the same information as the log file with the addition of:

- product/part contents,
- number of solid bodies,
- number of surfacic bodies, and
- translation time.

Viewing IGES log files

Learn how the data of an IGES file is organized and locate the problem in the transfer.

IGES stands for *Initial Graphics Exchange Specification*. IGES was the first attempt to make a "neutral file format". This enables different systems to transfer data without the need of a specific translator for each system.

The first section of an IGES file typically describes the contents of a file. Each line of this section is displayed on the information line.

If there is more than one line in the section, you can browse the contents of the section after file retrieval is complete by using the prompt history window, or by viewing the comments section of the IGES logfile created by the retrieve process.

Open a IGES log file

1 Choose **File > Show image**

The StudioTools errlog file and IGES logfile contain a specific error message for each problem encountered.

Each error message includes the IGES entity type number and a **Directory Section** line number where the entity occurs. The errlog file is located at `/usr/a1demo/errlog`; you can browse it with any UNIX text editor (for example, vi or jot). The IGES logfile is located in the same directory as the retrieved IGES file.

Summary of messages in the IGES logfile

- `IGES file retrieved successfully`

  Indicates retrieval is finished, without errors or unsupported entities.

- `Errors detected during translation`

  Errors were encountered while processing the file.

- `Unsupported entities detected`

  Unsupported IGES entities were found in the file. (Displayed after errors were encountered.)

- `IGES file retrieved: Refer to errlog file and/or IGES logfile for details of errors.`

Errors or unsupported IGES entities were found in the file.

Identifying IGES supported entities in log files

Learn more about the IGES entities supported on import by StudioTools.

All other entities with an entity use flag value (Definition) are ignored by the input translator.

Type	Form	IGES Entity	StudioTools Entity
100	0	circular arc	B-spline curve
102	0	composite curve	B-spline curves (group)
104	0-3	conic arc	B-spline curve
106	1	copious data	B-spline curve
106	2	copious data	polyline
106	11	copious data	polyline
106	12	copious data	polyline
106	63	closed area	polygon of polyset
108*	0*	plane	B-spline surface, plane
110	0	line	line
112	0	parametric curve	B-spline curve
114	0	parametric surface	B-spline surface
116	0	point	control vertex
118	0-1	ruled surface	B-spline surface
120	0	surface of revolution	B-spline surface
122	0	tabulated cylinder	B-spline surface
124	0	transformation matrix	
126	0-5	rational B-spline curve	B-spline curve
128	0-9	rational B-spline surface	B-spline surface
130	0	offset curve	B-spline curve
140	0	offset surface	B-spline surface
141	0	boundary entity	trim curve

142	0	curve on surface	trim curve
143	0	bounded surface	trimmed surface
144	0	trimmed surface	trimmed surface
308	0	subfigure definition	instance geometry
402	7,9	associativity instance	group
406	1	property - definition levels	multi sets
408	0	singular subfigure instance	instance geometry
406	15	name	node name

IGES levels

All supported geometric IGES entities that are associated with IGES level <n> are added to an Alias exclusive set called LEVEL<n>. For example, if a 126 B-spline entity's directory entry indicates that it is on level 42, then it is added to StudioTools as either Set 42, Layer 42, Layer 42 and Set 42, or as Level 42, depending on the setting in the **Level Mapping** StudioToolssection of IGES.

An IGES 406 Form 1 Property Entity (**Definition Levels**) lists the IGES level numbers to which an associated entity belongs. An Alias multiset is created for each level number in the list, and the associated object is added to each of the sets. For example, if a **406 Definition Levels** entity indicates that a particular surface is on IGES levels 13, 26 and 39, then the multisets `LEVEL13`, `LEVEL26`, and `LEVEL39` are created and the surface is added to each of these sets.

IGES ASCII format

The IGES translator imports ASCII format IGES files with or without linefeed characters at the end of each record. Binary IGES files are not supported.

Translation log file

The translation log file is produced during both import and export of Unigraphics part files. This log file contains Entity Mappings, Entity Counts, and Information, Error, and Warning messages.

How to interpret an import log file

On import of a Unigraphics part file, a translation log file is created in the same directory as the Unigraphics part file. The file name of the log file is the same as that of the Unigraphics part file with the extension .unigraphics2alias added to it.

Short log file

If you have chosen to use a short log file via the interface options, the log file will contain:

- the pathname of the imported Unigraphics part file and the name of the current StudioTools stage into which the Unigraphics part file is being imported.
- the contents of the Unigraphics part file. This is a table that documents the number of each type of Unigraphics entity in the file. This table includes all entities in the Unigraphics part file.
- any information, warning, or error messages that may occur during the translation.

If an error or problem occurs while processing an entity, the Unigraphics' entity's type name, user identifier name, and instance name are printed, followed by the error or warning message.

Extended log file

If you have chosen to have an extended log file produced, then the log file will contain the same information as the short log file, with the addition of:

- the Detailed Entity Mapping Report. This report contains a mapping description line for each entity processed.

The mapping description line in an extended import log file contains:

- the Unigraphics entity type name (e.g. BsplineSurface)
- the Unigraphics entity's user id (e.g. WHEEL, or <NONE> if the entity does not have a name)
- the Unigraphics entity instance identifier
- the string converted to, indicating that the following information pertains to the Alias entity
- the Alias entity type name (e.g. Trimmed Surface)
- the Alias node name (e.g. WHEEL or node#50 in the case of a Unigraphics entity name of <NONE>).

Interpret a export log file

On export of a Unigraphics file, a translation log file is created in the same directory as the Unigraphics file. The file name of the log file is the same as the Unigraphics file with the extension `.alias2unigraphics` added to it.

Short log file

If you have chosen to have a short log file via the interface options, the log file will contain:

- the name of the current StudioTools stage from which the model was exported (or the name of the StudioTools Wire file if the command line interface was used) and the pathname of the exported Unigraphics file
- any **Information, Warning,** or **Error** messages that may occur during the translation
- a table that documents the number of each type of Alias entity converted or not converted to a Unigraphics entity.

If an error or problem occurs while processing an entity, the Alias entity's type name and node name are printed, followed by the error or warning message.

Extended log file

If you have chosen to have an extended log file produced, then the log file will contain the same information as the short log file with the addition of:

- the Detailed Entity Mapping Report which contains a mapping description line for each entity processed.

Interpret a Unigraphics translation log

If the log file identifies an error during translation between StudioTools and Unigraphics in either direction, you can trace the information in the error message back to the problem geometry. In some cases once the piece of geometry has been isolated, the nature of the problem will be visually obvious and you will need only to delete or manually fix the surface in question.

Evaluation tools

There are a number of useful tools to help you to complete a successful data transfer.

- **Evaluate > Check model**
- **Evaluate > Continuity > Surface continuity**
- **Evaluate > Surface Evaluate > Highlights**
- **Evaluate > Surface Evaluate > Curvature**
- **Evaluate > Surface Evaluate > Contour**
- **Evaluate > Surface Evaluate > Horizon**
- **Locators > Curve curvature**
- **Locators > Deviation > Closest Point**
- **Locators > Deviation > MinMax Curve-Curve deviation** (Min/Max CrvCrv deviation, Min/Max SrfSrf deviation, Min/Max CrvSrf deviation, Min/Max cloud >Deviation > Deviation)
- **Surface Edit > Stitch > Shell stitch**
- **Surface Edit > Stitch > Shell unstitch**

Appendix

Review background information on each file formats.

Details on File Format types

The following information provides some insight in how the different file format structures work.

.

Name	Extension	Polygon Import	Polygon Export	NURBS Import	NURBS Export
Wire format *(Alias Systems)*	.wire	✓	✓	✓	✓
C4 format *(Standard CAD)* IGES subset format.	.c4x	✓	✓	✓	✓
STEP format ISO standard industrial automation systems product data representation and exchange format.	.stp			✓	✓
Encapsulated Postscript format	.eps		n/a		n/a
PTC Granite format (Windows Only) (PRO/ENGNINEER or any Granite based system)	.g .prt			✓	✓
JAMAIS format *(Standard CAD)* Japan Automobile Association IGES subset.	.igs			✓	✓
VDAIS format *(Standard CAD)* Verband des Automobilindustrie - IGES Subset.	.iges	✓	✓	✓	✓

Name	Extension	Polygon Import	Polygon Export	NURBS Import	NURBS Export
VDAFS format *(Standard CAD)* Verband des Automobilindustrie - FlachenSchnittstelle.	.vda			✓	✓
DES format *(Standard CAD)* Data Exchange Standard. Contains polylines. Requires less space than equivalent IGES files.	.des			✓	✓
DXF format *(AutoDesk)* Drawing Exchange Format.	.dxf		✓		✓
SLC format *(Solid Imaging)* 3D Systems Slice format.	.slc				✓
STL Format *(Solid Imaging)* 3D Systems SLA file format. Used to send data to a stereo lithography machine. Binary and ASCII format.	.stl	✓ (imports as a mesh)	✓		✓ (uses a shell to convert to STL)
CAI format for CATIA V4 files *(CATIA)* CATIA Alias Inter operate ability file format.	.cai			✓	✓
CATIA V5 format	.cat		✓	✓	✓

Name	Extension	Polygon Import	Polygon Export	NURBS Import	NURBS Export
OBJ format *(Alias Systems)*	.obj	✓	✓	✓	✓
Encapsulated Postscript format *(Standard)*	.eps			✓	
VRML format *(3D Object Orientated Toolkit)*		✓	✓	✓	✓
Pro/ENGINEER Render file format	.slp	✓			
IGES format *(SolidDesigner)* Initial Graphics Exchange.	.iges, .igs	✓	✓	✓	✓
Inventor format (IRIX and Windows)	.iv	✓	✓	✓	✓
Illustrator format	.ai			✓	✓
Unigraphics proprietary format *(Unigraphics)*	.prt			✓	✓

Wire format

The compressed proprietary file format used by Alias Systems StudioTools products. Programmers can access wire file via the StudioTools OpenModel library. In StudioTools you can open wire files from previous versions. If you continue to use older versions of StudioTools and want to keep the older file format intact, be sure to copy the files before opening them, or rename the files after opening them.

IGES format

Initial Graphics Exchange Specification (IGES) is a neutral file format. This format enabled different systems to transfer data without the need of a specific translator on each system. StudioTools uses IGES version 5.3.

The basic file structure of IGES contains five sections:

- Start (S) section
- Global (G) section
- Directory (D) section
- Parameter (P) Data Section
- Termination (T) Section.

Pro/ENGINEER options for export and import

To make the import and export of IGES and STEP files easier for StudioTools to read.

1 In Pro/ENGINEER, select **Utilities > Options**

2 Add the following data to the configuration file.

```
IGES_OUT_ALL_SRFS_AS 128
IGES_OUT_SPL_CRVS_AS_126 YES
IGES_OUT_SPL_SRFS_AS_128 YES
IGES_OUT_MIL_D_28000 NO
IGES_OUT_TRM_SRFS_AS_143 NO
IGES_OUT_TRIM_CURVE_DEVIATION DEFAULT
INTF_OUT_BLANKED_ENTITIES NO
INTF3D_OUT_EXTEND_SURFACE YES
INTF3D_OUT_FORCE_SURF_NORMALS YES
IGES_IN_106_F2_AS_SPLINE NO
IGES_IN_DWG_LINE_FONT YES
IGES_IN_DWG_PNT_ENT YES
IGES_IN_DWG_COLOR YES
FIX_BOUNDARIES_ON_IMPORT YES
```

VDAFS format

Verband des Automobilindustrie - VDAFS is a standard used for the transfer of freeform shapes.

It was conceived for the transfer of freeform surfaces and their supporting objects. This means the dimensions, layers, 2D geometry and text are left out. This file format was logged as a National Standard in Germany (DIN 66301). StudioTools uses version 2.0 of VDAFS.

The basic file structure of the VDAFS format has a header section which gives the general information about where, when, by who and how the file was created. After the header, the entities are described one after the other.

The entity description has the following structure:

- Entity name
- Entity type
- Parameter information, separated by comma's
- Each line ends with a line number.

There are a limited number of entity types that can be used in the VDAFS file. These entities only deal with the geometry and topology of elements.

Entity type	Description
POINT	Point
PSET	Point Sequence
MDI	Point Vector Sequence (Master Dimension)
CIRCLE	Circle/Arc
CURVE	Curve
SURF	Surface
CONS	Curve on Surface

FACE	Bounded Surface
TOP	Topology of a Surface

The only non-geometrical entities that can be used are the following:

Entity type	Description
HEADER	Header
$$	Comment
BEGINSET	Set - begin ~ group
ENDSET	Set - end
GROUP	Group
TMAT	Transformation Matrix
TLIST	Transformation List

Because of this limited number of entities, compared to the 500+ of the IGES format, geometry is always defined using the same rules resulting in a lower number of errors. On the other hand, you lose a lot of information that was created in the originating system like layers but these can eventually be replaced by groups, colors, dimensions, etc.

VDAIS format

Verband des Automobilindustrie - IGES Subset (VDAIS) is a well defined collection of IGES entities, carefully selected for optimized exchange of geometry between manufacturers and subcontractors in the car industry.

C4 format

IGES subset file format.

Unigraphics proprietary format

Unigraphics is a solid modeling package based on the Parasolid kernel. The package contains many (mostly optional) modules, for example CAD, CAM, CAE, Sheet metal applications, knowledge bases, Quality control and Rapid prototyping. The file structure is binary.

The following table explains how common elements are called in both StudioTools and Unigraphics.

UG	StudioTools
Segment	Span (curve)
Point	Point
Patch	Span (surface)
Pole	CV (Control vertices)
Sew	Stitch
U/V Grid	Patch Precision
Parameters	History
Control polygon (display only)	Hull
Silhouette curve	Horizon curve
Blank	Invisible
Reference Set	Set
Category	Category
Custom view	Bookmark
Knot point	Edit point

Supported Unigraphics geometry and data mapping

The following sections describe the mapping process used for geometry types and non-geometric data that can be transferred between StudioTools and Unigraphics.

Supported StudioTools geometry types

The following StudioTools geometry types can be exported to Unigraphics. Non-geometry entities such as Lights, Cameras, Textures, Windows, Animation are not supported by the translator. The numbers in the table entries refer to *Notes for Unigraphics entities* below.

StudioTools Entity	Unigraphics Entity
Construction Plane	WCS
Polyset	Not supported at this time.
Conic	Rational Curve
Arc	RationalCurve
Circle	RationalCurve
Line	Line
Curve	BCurve
Surface	BSurface; **(1), (2)**
Trimmed Surface	Face**(1), (2)**
Plane	Bounded Plane**(1)**
Shader	Colour Attribute**(3)**
Shell (Open)	Sheet Body
Set	Reference Set
Group	Group
Layer	Layer**(4)**
Category	Category
Shell (Closed)	Solid Body

Notes for Unigraphics entities

1. Unigraphics can not have free standing Surfaces so all surfaces are mapped to faces which must be attached to a sheet body.

2. Surfaces having internal discontinuities will be split at those discontinuities.

3 Mapped as a Display Attribute of the mapped surface or shell.

4 Layer Name is not mapped

Supported Unigraphics Entity objects geometry types

The following Unigraphics geometry types can be imported to StudioTools. The letters and numbers in parentheses refer to *Notes for StudioTools Unigraphics entity object entities* below.

Unigraphics Entity Objects	StudioTools Entity
WCS	Construction Plane
BSurface	Surface
Bounded Plane	Curve
Cylindrical Surface	Curve
Conical Surface	Curve
Tabulated Cylinder	Surface
Ruled Surface	Surface
Blended Face Surface	Surface
Surface of Revolution	Surface
Offset Surface	Surface
Sculptured Surface	Surface
BCurve	Curve
Line	Line(Curve)
Arc	Curve
Conic	Curve
Point	Point (Curve)**(1)**
Sheet Body	Shell (Open)**(2)**
Assembly	Groups/Instance**(3)**
Group	Group
Layer	Layer
Category	Category

Unigraphics Entity Objects	StudioTools Entity
Reference Set	Set
Part Attribute	Blind Data**()**
Solid Body	Shell (Closed)

Notes for StudioTools Unigraphics entity object entities

1 A Unigraphics Point is converted to a degree 1 curve composed of two coincident points. On export to Unigraphics this construction is converted back to a Unigraphics point.

2 If the Sheet body only points to one face then this is converted to a trimmed surface in StudioTools.

3 This is a one way mapping, assemblies cannot be exported.

Added as blind data can be re-exported.

JAMAIS format

Japanese Automotive Manufacturers Association Iges Subset, JAMAIS is a file format subset to IGES. JAMAIS has the same.igs extension and has exactly the same file structure as the IGES format, it only supports less entities.

JAMA-IS:

- does not include Dimensions
- supports only a limited number of entities, optimized for the use in the car-manufacturing.
- does not have the Surface Type option
- does not have vendor tuned settings

The disadvantage of not supporting the above entities is that you can't put all the different elements from an IGES file into StudioTools. The advantage is that because of the limited elements, there are fewer errors made when translating, everything is optimized for the exchange of the specific models.

PTC Granite format (Windows Only)

The Granite format is a new design of the DirectConnect Product for StudioTools/Pro/ENGINEER. This translator allows you to open, save as, export and import Granite files.

Granite One is a CAD technology platform for the design collaboration of solid models.

When importing a Pro/ENGINEER model into StudioTools as template objects the StudioTools users can then modify the imported models. One StudioTools object corresponds with one Granite object. For example, if the StudioTools wire file has curves, surfaces, and shells, there will be different features created in the target Pro/ENGINEER file for those curves, surfaces and shells.

When using the Granite translator StudioTools models are converted to Granite simple geometry features without construction history.

.Supported PTC Granite geometry and data mapping

The following sections describe the mapping process used for geometry types and non-geometric data transferred between StudioTools and Pro/ENGINEER DirectConnect with PTC Granite support (Windows Only).

Exported supported StudioTools object geometry types

The following StudioTools geometry types can be exported to PTC Granite.

Non-geometry entities such as lights, cameras, textures, windows, animation, construction planes, color, and layers which are not supported by the translator.

StudioTools Entity	PTC Granite Entity
Construction Plane	WCS
Polyset	Not supported at this time.

StudioTools Entity	PTC Granite Entity
Point	Point
Conic	BSpline curve or bcurve
Arc	BSpline curve
Circle	BSpline curve
Line	Line
Curve	BCurve
Surface	Quilt of a face from a BSurface
Trimmed Surface Face	Quilt of faces of BSurface
Plane	Quilt of a face from BSurface
Shader	Not supported
Shell (Open)	Quilt of face
Set	Not supported
Group	Not supported
Layer	Not supported
Category	Not supported
Shell (Closed)	Solid body

Imported supported PTC Granite Entity object geometry types

The following PTC Granite geometry types can be imported to StudioTools.

PTC Granite Entity Objects	StudioTools Entity
WCS	Not supported
BSurface	Surface
Bounded Plane	Surface
Cylindrical Surface	Surface
Conical Surface	Surface
Tabulated Cylinder	Surface

PTC Granite Entity Objects	StudioTools Entity
Ruled Surface	Surface
Blended Face Surface	Surface
Surface of Revolution	Surface
Offset Surface	Surface
Sculptured Surface	Surface
BCurve	Curve
Line	Line(Curve)
Arc	Curve
Conic	Curve
Point	Point (Curve)
Assembly	Groups/instances
Group	Not supported
Layer	Not supported
Category	Not supported
Reference Set	Not supported
Part Attribute	Not supported

DES format

Design Exchange Specification (DES) format describes degree 1 curves Polylines in DES format require less disk space than the same data in IGES format.

The file structure is as follows:

- The START line indicates the linear units system that is used.
- The HEADER line contains a reference to the Userinfo.
- The SENDING SYSTEM line indicates the type of preprocessor used.
- The LINE indicates the start of a description of edit points for constructing a first degree line. The parameters are grouped by three and represent the X,Y and Z coordinates of the points that form the edit point of the lines. A new description of a line always starts with the LINE entity, followed by the number of lines, the number of edit points and a 0 to finish.
- The DES file concludes with a * TERMINATE section.
- Every line ends with a line number.

DXF format

The Drawing Exchange Format (DXF) from AutoDesk is a verbose ASCII geometry format capable of representing lines, simply defined surfaces, and polygons (3D faces).

The file structure of a DXF file is divided into 4 main sections:

- HEADER: identifying the system variables
- TABLES: providing general information such as line style, user-defined coordinate systems, etc.
- BLOCKS: defining instanced blocks in the model
- ENTITIES: describing the entities of the model


The above applies to the DXF format from Autocad 14

Each section is preceded by a group code 0, followed by the SECTION string, a group code 2 and the type of section. A section is concluded with the 0 groupcode and the ENDSEC string. In each section, there are different items, each consisting of 2 lines: the first to declare the nature of the item, the second to define it's value. Each item in a DXF file takes a new line, therefore even the description of a simple vertex takes up to 12 lines.

The group codes (GC) defines the nature of the value on the next line. There are general and section specific group codes.

STEP format

Standard for the Exchange of Product Data (STEP) is a ISO standard industrial automation systems product data representation and exchange format.

The file structure for a STEP file has a modular structure which makes it easier for developers to adapt the format to their own needs. The modules are called “classes” and are classified using numbers as follows:

- **11-->13:** *Description methods*: These classes describe the languages and methods that are used to create a STEP file. Right now, only two description method classes exit:

  Part 11: The EXPRESS language reference manual.

  Part 12: The EXPRESS-I language reference manual

  The EXPRESS language is a data definition language that is used to represent the structure of data and any constraints that may apply to it. The information models contained in STEP Integrated Resources and Application Protocols are defined using EXPRESS.

- **21-->26:** *Implementation methods*: describe the correspondences between STEP and other formal languages. (text encoding, C++ binding, ...)
- **31-->35:** *Conformity tests*: used to check the degree of conformity of the software associated with the Application Protocol.
- **41-->49:** *Integrated generic resources*: the conceptual building blocks for STEP
- **101-->106:** *Integrated application resources*: contains the actual database, the building block of the file. divided into generic resources and Application resources
- **201-->233:** *Application protocols*: contain all the branch-specific classes.
- **301-->332:** *Abstract test suites*
- **501-->518:** *Application interpreted constructs Descriptions methods*

STEP Application protocols

- + 201: 2D explicit technical design

- + 202: 2D associative technical design
- + 203: configuration of mechanical parts and assemblies
- + 204: Brep 3D mechanical design
- + 205: Surface 3D mechanical design
- + 206: Wireframe 3D mechanical design
- + 207: Sheet Metal Die Planning
- + 208: Life Cycle Change Process
- + 209: Composite Structures
- + 210: PCA: Design & Manufacture
- + 211: Elect, Test, Diagnostics & Remfg
- + 212: Electrotechnical Plants
- + 213 : NC Process Plans
- + 214: Automotive design
- + 215: Ship Arrangement
- + 216: Ship Moulded Forms
- + 217: Ship Piping
- + 218: Ship Structures
- + 219: Inspection Process Plans
- + 220: PCA: Manufacturing Planning
- + 221: Functional Data & Schematic Rep. for Process Plants
- + 222: Design to Manufacturing for
- Composite Structures
- + 223: Exchange of Design, and Manufacturing Product Information for Cast Parts
- + 224: Mechanical Products Definition for Process Planning Using Form Features
- + 225: Strctrl. Blg. Elements Using Explicit Shape Rep.
- + 226: Ship Mechanical Systems/Moulded Forms
- + 227: Plant Spatial Configuration
- + 228: Building Services: Heating, Ventilation, and AC
- + 229: Exchange of Design and Manufacturing Info for Forged Parts

- + 230: Building Structural Frame: Steelwork
- + 231: Process engineering Data: Proc Design & Proc Specs of Major Equipment
- + 232: Technical Data Packaging

Each AP has its own subset of entities specific for the industry branch the AP was developed for StudioTools (supports the AP203 and the AP214.)

Application protocol support for STEP formats

The geometric descriptions contained within ISO10303-203 and ISO10303-214 are identical and comprises the core of the implementation of the translator. The following table shows the mappings made by StudioTools.

STEP Entity	StudioTools Entity
Cartesian Point	Point
Line	B-spline Curve
Circle	B-spline Curve
Ellipse	B-spline Curve
Parabola	B-spline Curve
Hyperbola	B-spline Curve
PolyLine	B-spline Curve
Composite Curve	B-spline Curve (Grouped)
Trimmed Curve	B-spline Curve
B-spline Curve	B-spline Curve
Plane	B-spline Surface
Cylindrical Surface	B-spline Surface
Conical Surface	B-spline Surface
Spherical Surface	B-spline Surface
Toroidal Surface	B-spline Surface
Surface of Linear Extrusion	B-spline Surface
Surface of Revolution	B-spline Surface

STEP Entity	StudioTools Entity
B-spline Surface	B-spline Surface
Rectangular Trimmed Surface	Trimmed Surface
Curve Bounded Surface	Trimmed Surface
Offset Surface	B-spline Surface
Manifold Solid Brep	Shell (Closed)
Shell Based Surface Model	Shell (Open/Closed)

Pro/ENGINEER options for export and import

To make the import and export of IGES and STEP files easier for StudioTools to read.

1 In Pro/ENGINEER, select **Utilities > Options**

2 Add the following data to the configuration file.

```
IGES_OUT_ALL_SRFS_AS 128
IGES_OUT_SPL_CRVS_AS_126 YES
IGES_OUT_SPL_SRFS_AS_128 YES
IGES_OUT_MIL_D_28000 NO
IGES_OUT_TRM_SRFS_AS_143 NO
IGES_OUT_TRIM_CURVE_DEVIATION DEFAULT
INTF_OUT_BLANKED_ENTITIES NO
INTF3D_OUT_EXTEND_SURFACE YES
INTF3D_OUT_FORCE_SURF_NORMALS YES
IGES_IN_106_F2_AS_SPLINE NO
IGES_IN_DWG_LINE_FONT YES
IGES_IN_DWG_PNT_ENT YES
IGES_IN_DWG_COLOR YES
FIX_BOUNDARIES_ON_IMPORT YES
```

CAI format for CATIA V4 files

CATIA is a CAD/CAM package. It contains modules for mechanical design, analysis, shape design & styling, equipment and systems engineering, product synthesis and infrastructure.

CAI is based on the STEP standard. The translator, CATIA DirectConnect, is created using the EXPRESS language from STEP. This file transfer system is very specific and completely optimized, the CAI scheme has 31 entities.

The file structure for CAI is an adapted version of the STEP format to the needs for converting CATIA and StudioTools specific entities. This is indicated by the file_schema ('CAI') declaration in the STEP file. For more information refer to the STEP options.

CATIA geometry and data mapping

Use the following sections to identify the mapping process used for geometry types and non-geometric data transferred between StudioTools and CATIA via the CAI file.

Supported StudioTools geometry types

The following StudioTools geometry types can be exported to CATIA. Non-geometry entities such as Lights, Cameras, Textures, Windows, Shaders and Animation are not supported by the translator. The numbers in the table entries refer to *Notes for table 1* below.

StudioTools Entity	CAI File Entity	CATIA Entity
Construction Plane	Axis(1)	8.1 3-axis system
Polyset	Not supported at this time.	N/A
Arc	RationalCurve	46.1 Rational B_spline curve
Circle	RationalCurve	46.1 Rational B_spline curve
Line	Line (partially supported) (2)	2.1 space line
Curve	BsplineCurve RationalCurve	3.2 B_spline polynomial space curve 46.1 Rational B_spline curve

StudioTools Entity	CAI File Entity	CATIA Entity
Face	FaceElement (3)	6.1 face
Surface	BsplineSurface RationalSurface	5.2 B_spline polynomial surface 47.1 Rational B_spline surface
Target Surface	CrvSrf (4)	12.1 edge (curve on surface)
Trimmed Surface	FaceElement (5)	6.1 face
Trimmed Target Surface	FaceElement and CrvSrf (l)	6.1 face and 12.1 edge (curve on surface)
Shell	OpenShell	13.1 skin

Notes for table 1

1. The scale component of the StudioTools Construction Plane is not transferred. That is, its orientation and position will remain the same, but its size will not. After a round trip translation (StudioTools > CATIA > StudioTools), an StudioTools Construction Plane with original scale values **Sx**, **Sy**, and **Sz**, will return with scale values of 1, 1, 1.

2. A StudioTools Line created by **Curves > Line** and related tools is not supported on export to CAI as a CAI Line primitive, but is supported as a CAI BsplineCurve. However, on import to StudioTools of a CAI Trimmed Curve whose basis curve is a CAI Line (this maps to a CATIA limited Line), an StudioTools Line is created. Such a Line will be exported as a CAI Line. This ensures the original CATIA limited Line is preserved.

3. A StudioTools Face element is converted to a Trimmed Surface temporarily and then exported as such (see note (5) below).

4. A StudioTools Target Surface contains one or more Curves-on-Surface. A CrvSrf entity is exported for each curve-on-surface. On import to StudioTools, multiple CrvSrf entities that reference the same base surface will be converted to one StudioTools Target Surface containing multiple curves-on-surface that correspond to these CrvSrf entities.

5 A StudioTools Trimmed Surface is exported as one or more FaceElements (one FaceElement for each region of the trimmed surface). On import to StudioTools, multiple FaceElement entities that reference the same base surface are converted to a StudioTools Trimmed Surface, with multiple regions corresponding to these FaceElements.

- A StudioTools Trimmed Target Surface also contains *curves-on-surface* that have not yet been used for trimming. Similar to the descriptions in Notes 2 and 3, it is exported as a combination of CrvSrf and FaceElement entities.

Identifying CATIA geometry and data mapping entities

To identify supported CATIA geometry and data mappings use the following tables to identify the mapping process used for geometry types and non-geometric data transferred between StudioTools and CATIA via the CAI file. The definitions for the letters and numbers in parentheses is provided below the table.

This information is valid for CATIA version 4.1 not version5.

CATIA Entity	CAI File Entity	StudioTools Entity
8.1 3-axis system (fixed)	Axis	Construction Plane
1.1 space point	Point	Curve and Locator **(1)**
2.1 space line **(1)**	Line	Line
3.1 polynomial space curve **(1)** 3.2 B_spline polynomial space curve **(1)** 20.1 space circle **(1)** 21.1 space ellipse **(1)** 22.1 space parabola **(1)** 23.1 space hyperbola (1)	BsplineCurve	Curve
46.1 Rational B_spline curve (1)	RationalCurve	Curve
4.1 plane 4.24 plane (2)	Plane	Surface **(2)**

CATIA Entity	CAI File Entity	StudioTools Entity
5.1 polynomial surface 0 unspecified 1 sphere 2 cylinder 3 cone 4 torus 5 unspecified surface of revolution 6 tabulated cylinder 7 ruled surface	BsplineSurface	Surface
5.2 B_spline polynomial surface	BsplineSurface	Surface
47.1 Rational B_spline surface	RationalSurface	Surface
12.1 edge (face boundary) **(3)**	OrientedEdge	Trim Curve **(3)**
12.1 edge (curve on surface)	CrvSrf	Target Surface **(4)**
6.1 face	FaceElement	Trimmed Surface **(5)**
13.1 skin	OpenShell	Shell
7.1 volume **(4)**	OpenShell(s)	Shell(s)
17.2 exact solid **(5)**	OpenShell(s)	Shell(s)
28.1 space ditto	Ditto	Instance **(6)**

Notes for CATIA entities in table 2

1. For all CATIA curves whose limits have been modified, a CAI Trimmed Curve instance may be created when exporting.

2. For CATIA planar faces, some 4.24 planes may be transferred as they are used as the base face surface.

3. The curve associated with the edge in CATIA is converted into a B-spline Curve, used by the Oriented Edge. The parametric definition of the CATIA edge is converted to a B-spline Pcurve used by the Oriented Edge.

4. The domains of the CATIA volume are converted to open shells.

5 The Boundary representation of the solid is extracted and converted to CAI elements (open shells, curves, etc.). Prior to exporting the exact solid, make sure that this solid is up-to-date.

Notes for StudioTools entities in table 2

1 A CAI Point is converted to a degree 1 curve composed of two coincident points and a Locator that identifies the location of this curve. On export to CAI, this construction is converted back to a CAI point.

2 A CAI Plane describes an infinite plane and is converted to a unit StudioTools Surface. This Surface will be re-exported as a CAI Plane.

3 The CAI Oriented Edge maps to a trim curve of a trimmed region of a StudioTools Trimmed surface.

4 A CAI CrvSrf entity maps to one StudioTools curve-on-surface for a particular surface. Multiple CAI CrvSrf entities in the CAI file referencing the same surface will cause multiple StudioTools *curves-on-surface* to be added to one StudioTools Target Surface.

5 A CAI FaceElement maps to one region of a StudioTools Trimmed Surface. If multiple CAI FaceElements in the CAI file reference the same surface, then a StudioTools Trimmed Surface will be created with multiple regions.

6 A CAI Ditto is imported as a StudioTools instance node, with the CAI Ditto transformation, of the StudioTools group representing the CAI Detail (see *Supported Non-Geometric Data, Groups/Instances* below).

However, if at least two CAI Dittos of the same CAI Detail each belong to different parent CAI details, then those dittos are “exploded”. That is, each geometric entity of the detail is copied and transferred by the transformation of the CAI Ditto.

Supported non-geometric CATIA data

The following non-geometric data can be imported and exported between StudioTools and CATIA.

Groups/instances

How a StudioTools Group is exported depends on whether it is instanced or not. A StudioToolsGroup that is not instanced has its component geometry exported to the appropriate independent CAI geometric entities. A StudioTools Group node that is instanced is exported as a CAI Detail entity (that is, the grouped StudioTools geometric entities become members of the CAI Detail); a StudioTools Instance node is exported as a CAI Ditto entity. CAI Dittos and Details are subsequently imported into CATIA as CATIA Dittos and Workspaces, respectively.

A StudioTools Instance that contains a non-proportional (non-p) scaling factor in its transformation is "exploded" on export (that is, each geometric entity that belongs to the StudioTools Group being instanced is copied and transformed to independent CAI geometric entities). Only instances that contain a non-p scale component are exploded. Other instances of the same group that do not have a non-p scale component will export as CAI Dittos of a CAI Detail (as noted above).

CATIA Workspaces and Dittos can feature a situation that is not supported by the StudioTools dag structure and instancing mechanism. This situation arises when Dittos of the same Workspace belong to different parent Workspaces (see note (6) in *Notes for StudioTools entities in table 2* above). The CATIA Ditto/Workspace structure is preserved as best as possible on CAI export from CATIA and subsequent import back to CATIA.

Names

StudioTools DAG node names are exported as an attribute of the CAI entity to which the node's geometry maps. StudioTools Instance node names are exported as an attribute of the CAI Ditto; the corresponding CAI Detail will have the name of the instanced StudioTools Group node. The name attribute of a CAI entity is mapped to the CATIA User Identifier for that entity in CATIA.

On CAI import to StudioTools, it is possible for multiple CAI entities to map to a single StudioTools object (see notes (4) and (5) in *Notes for StudioTools entities in table 2* above). Each of these CAI entities may have a unique name (CATIA User

Identifier), but the StudioTools object can only have one name available for viewing and editing. The CATIA User Identifiers that are not mapped to a StudioTools object with a modifiable name are maintained by StudioTools for subsequent export to CATIA via CAI, assuming the objects to which the names are assigned still exist.

For example, a CATIA user can name individual Faces that all reference the same surface. These Faces map to CAI FaceElements which in turn map to trim regions of a single StudioTools Trimmed Surface. Only the trimmed surface node can have a name, and this is given the name of the underlying CATIA surface being trimmed. The CATIA User Identifiers of each of the faces that mapped to trim regions are maintained within StudioTools so they can be transferred back to CATIA. A similar situation exists for multiple CAI CrvSrf entities that map to one StudioTools Target Surface. The CAI CrvSrf name attribute is preserved for export back to CATIA.

StudioTools generated names of the form `node#<n>` (for example, `node#5`) are not exported to CAI. CATIA generated names of the form `*<OBJ><N>` (for example, `*FAC5`) are not exported to CAI. CAI entities that do not have names are given a system generated name on import to StudioTools or CATIA.

Layers

The StudioTools Layer assignment is preserved when exporting to CAI. As with names, the layer number becomes an attribute of CAI geometric entities and Dittos. The layer name and other StudioTools Layer attributes are not exported.

On CAI import to StudioTools, geometric entities are placed on the layer defined by the CAI layer number attribute. If the layer is created by the import process, it is given the name CATIA_<N> where <N> is the layer number. All layers created by the CAI import process are sorted in the layer bar by ascending number.

As with the name attribute, the layer number assignment for CAI FaceElement and CAI CrvSrf entities is preserved on CAI import so that this information can be returned to CATIA through CAI even though it cannot be modified in StudioTools.

In StudioTools, a geometric object and all of its ancestors in the dag hierarchy must be in the same layer. In CATIA, this is not necessarily the case, as a Ditto may be in a different layer than the components of the instanced Workspace. On CAI import to StudioTools, this feature of CATIA layers is maintained for subsequent export back to CATIA. If you change the layer assignment of an StudioTools Instance that was created from a CAI Ditto, you will change the layer assignment of all the geometric entities being instanced to this new layer number within StudioTools. However, on CAI export, the original CATIA layer number assignments for the component geometric entities and Ditto will be used.

CATIA Dittos that are not assigned to a layer will be preserved through a bi-directional CAI translation with StudioTools. While in StudioTools, these Dittos will be imported as StudioTools instances that reside in the Default Layer.

Object colors

The wireframe display color of CATIA objects is preserved in a bi-directional CAI translation with StudioTools, *providing the CATIA model uses the CATIA Standard Color Table.* If the model references a user-defined color table, then on CAI import to CATIA, the CAI color of each element is chosen from the standard color table as the closest match to the original color.

As with names and layers, the CATIA wireframe color index of an object is an attribute of the CAI geometric entity. On CAI import to StudioTools, the CAI geometric entity's color index is converted to an RGB color based on the CATIA Standard Color Table. This RGB color is used to create a simple shader that is assigned to the associated StudioTools object that is created, unless the object is a curve. Only for curves is the CATIA wireframe color index preserved. CAI surface geometry that has no CAI color attribute is assigned the StudioTools default shader.

On StudioTools export to CAI, the color index of a CAI geometric surface is determined by finding the index in the CATIA Standard Color table that represents the closest match to the RGB color of the associated StudioTools shader. For non-surface CAI entities, the preserved CATIA color index is exported if it exists. For original StudioTools curve geometry, the layer color is used to find a close match to a CATIA Standard Color index.

Layer colors

Layer colors are not transferred explicitly via the CAI format. However, layer colors are assigned to layers created as a result of CAI import by one of two possible methods.

In the first method (the default) a created layer is assigned one of the 15 available colors based on the remainder of the layer number divided by 15. This cycles through the available StudioTools layer colors as the layer number increases.

The second method uses the CATIA Standard Layer Color Table which is defined in the CATIA environment's startup initialization file CATINSTD.dcls. If this file exists in the /usr/aw/alias/.Alias/ directory of the StudioTools installation, or in the.Alias/Prefs.1/Color/ directory of the StudioTools user account, then the CATIA Standard Layer Color Table will be extracted from this file by the Alias CAI import process and used in the layer color assignment process.

The CATIA Standard Layer Color Table assigns to each of the 256 CATIA layers a default CATIA color index whose RGB color is defined in the CATIA Standard Color Table (see Object colors above). When the Alias CAI import process has access to this default layer color assignment, it assigns a color to a newly created StudioTools layer that closely matches the color for that layer number in the table.

To activate this second method copy the CATINSTD.dcls file from the CATIA installation to `/usr/aw/alias/.Alias/` or to the `.Alias/Prefs.1/Color` directory of your user account. Alternatively, create a symbolic link, in either of these StudioTools directories, to the `CATINSD.dcls` file in the CATIA installation.

For example, if StudioTools is installed in `/usr/aw` and CATIA is installed in `/usr/catia`, then the following command will create the necessary symbolic link:

```
ln -s  /usr/catia/cfg/dec/CATINSTD.dcls /usr/
aw/alias/.Alias/CATINSTD.dcls
```

What happens when creating a skin and offsetting for CATIA?

Creating a 'skin' and 'offsetting' (or creating solid geometry from imported StudioTools models) is a common workflow when integrating StudioTools and CATIA. When exporting the model from StudioTools, you must ensure that the geometry is built to the correct tolerance and that it can also be stitched. The stitching process in StudioTools identifies gaps between surfaces so that you can repair the appropriate geometry before writing the file out to CATIA.

During stitching, the surfaces are twinned. This means that the surface boundaries may be split to accommodate adjacent surfaces, and periodic geometry is detached into multiple surfaces. For this reason, you should save the StudioTools wire file before stitching so that if further modifications are required to be made to the StudioTools model, the construction history will be intact.

CATIA V5 format

The following entities are translated into CATIA V5 format.

Studio entity	saved to CATIA V5 format as
Curve (created with either CVs or Edit Points)	NonRationalNURBSCurve of appropriate degree
Line	Line
Arc	RationalNURBSCurve
Circular	RationalNURBSCurve. Note that 360 degree periodical objects will be divided into two if the -c option is used.
Blend curves	NonRationalNURBSCurve of appropriate degree.
Multiknot curve	NonRationalNURBSCurve
Curve on surface	NURBSCurves
Surface	NonRationalNURBSSurface.
Surface Revolve	RationalNURBSSurface. 360 degree periodical objects will be divided into two if the -c option is used.
Multiknot Surface	Surfaces divided in multiple surfaces if -d option is used.
Shell	Solid. Caution: Alias closed shell becomes a CATIA Solid. If -s option is specified, the shell's faces will become surfaces.
sphere, cone, cube, etc.	NonRationalNURBSSurface. 360 degree periodical objects are divided into two if -c option is used.
Rational sphere, cone, etc.	RationalNURBSSurface. 360 degree periodical objects are divided into two if -c option is used.

Studio entity	saved to CATIA V5 format as
Planar face	NonRationalNURBSSurface of degree 1 with a single span in U and V directions. Faces with holes will be transferred.
Shell	CATIA solid body, if shell is closed, or open if Join or Heal is ON.
Surface with multiple disjoint trimmed regions	Multiple trimmed surfaces
Polyset, mesh	Each polygonal face is converted to a surface.
Closed periodic curve	2 curves each
Shader Color RGB, Transparency	Color RGB, Opacity.
Layer	Layer (numbered). Layer name is ignored. If -a option is ON (Export by Layer) a GSMTool (surfacic) body is created for each layer with its name.
Invisible	Imported as Invisible (hidden)
Construction vector	Line
Template	Pickable
Construction planes	Planes
Symmetry	Converted if -b option for layer symmetry is ON.
Space point, Curve point, Surface point, Curve-on-surface point	Point

The standard document types provided by Dassault Systemes are:

- Part Document
- Product Document
- Drawing Document
- Material Document
- Catalog Document
- Analysis Document
- Process Document

A **Part Document** has a .CATPart suffix, and is mainly used during the design process. The Part Document is used as the default document type for the following workbenches:

- Mechanical Design: Part Design, Sketcher, Sheet Metal Design, Wireframe & Surface Design, Functional Dimensioning & Tolerancing.
- Shape: Free Style, GSM Debug, Digitized Shape Editor, Generative Shape Design.
- Equipment & Systems: Circuit Board Design.

A Product Document has a .CATProduct suffix, and is mainly used to assemble Product or Part documents in a structured, coherent system of sub-products and/or parts. The Product Document is used as the default document type for the following workbenches:

- Infrastructure: Product Structure, Knowledge Advisor, Knowledge Expert, Generative Knowledge, Product Engineering Optimizer, Rendering.
- Mechanical Design: Assembly Design, Structure Design.
- AEC Plant: Plant Layout.
- Digital Mockup: Navigator, Space Analysis, Kinematics, Fitting, Plant & Ship Review, Optimizer, Tolerancing Review.
- Equipment & Systems: Systems Space Reservation, Systems Routing, Electrical Wire Routing.
- DELMIA Infrastructure: Digital Manufacturing ReView.

OBJ format

The OBJ file format is a ASCII form. It supports a variety of geometry ranging from polygons to high degree NURBS surfaces.

The following types of data may be included in an OBJ file. In this list, the keyword (in parentheses) follows the data type.

Vertex data

- geometric vertices (**v**)
- texture vertices (**vt**)
- vertex normals (**vn**)
- parameter space vertices (**vp**)

Free-form curve/surface attributes

- rational or non-rational forms of curve or surface type.

Basis matrix, Bezier, B-spline, Cardinal, Taylor (cstype)

- degree (**deg**)
- basis matrix (**bmat**)
- step size (**step**)

Elements

- point (**p**)
- line (**l**)
- face (**f**)
- curve (**curv**)
- 2D curve (**curv2**)
- surface (surf)

Free-form curve/surface body statements

- parameter values (**parm**)
- outer trimming loop (**trim**)

- inner trimming loop (**hole**)
- special curve (**scrv**)
- special point (**sp**)
- end statement (**end**)

Connectivity between free-form surfaces

- connect (**con**)

Grouping

- group name (**g**)
- smoothing group (**s**)
- merging group (**mg**)
- object name (**o**)

Display/render attributes

- bevel interpolation (**bevel**)
- color interpolation (**c_interp**)
- dissolve interpolation (**d_interp**)
- level of detail (**lod**)
- material name (**usemtl**)
- material library (**mtllib**)
- shadow casting (**shadow_obj**)
- ray tracing (**trace_obj**)
- curve approximation technique (**ctech**)
- surface approximation technique (**stech**)

Encapsulated Postscript format

You can control what to retrieve from an EPS file.

You cannot import into StudioTools EPS files that were created using the SGI Showcase program.

Inventor format (IRIX and Windows)

Inventor is an object-oriented C++ based language that describes complete 3D-scenes which can be made interactive and that are optimized for OpenGL. It is an ASCII file format.

Inventor can be seen as a complete toolkit, consisting of:

- a library of (3D) objects
- a library of modules and utilities
- an API in C/C++ for programming

The Inventor language contains a large set of predefined objects called nodes such as geometry, properties, groups and manipulators.

These nodes are ordered using a scene graph which describes the hierarchy of the nodes. This means that you can have certain objects in the beginning of the file whose behaviour or properties influence other objects.

Inventor file that is exported from StudioTools does not contain a scene graph.

- Each nodes is defined by the following set of parameters:
- Nature: a geometrical object (for example,. cube, cylinder, and sphere), texture maps, cameras, lights.
- Fields: contains info on the size, and file for the texture maps.
- Name: Nodes can be referenced by a name. A name is not absolutely needed.
- Child nodes: every node can contain other nodes. This node is called a group node.

3D scenes can be constructed containing not only 3D geometry but also lighting, texturing, different camera settings.and animation (A node can also be a transformation). Nodes can also be references to files (for example, images) and external programs, like Midiplayers or WAV-players.

To have interactivity with the scene, special nodes like draggers, action nodes and manipulators are included.

Examples of nodes that can compose a scene and provide interactivity

Shape nodes: Cube, Cylinder, Cone, Sphere, Face set, Text2, Line, NURBS, Quad mesh.

Property nodes: Base color, Color index, Complexity, 3D and 4D coords, Environment (e.g. fog, haze, light), Font, Lighting model (Base color or Phong), Material, Normal, Pick Style, Texture, Texture Coordinates, Transform, Units.

Group nodes: Annotation, Array, File, Group, Level of detail, Multiple copy.

Actions nodes: Callback, GL render, Get bounding box, Handle event, Highlight render.

Engine and Sensor nodes: Blinker, Engine classes, Pendulum, Rotor, Sensor classes, Shuttle (oscillation).

Interactive 3D Manipulator nodes: Centerball (rotation), Drag point, Handle box (scaling, translation), Light draggers, Rotate draggers, Scale draggers.

Other nodes: Clip plane, Directional light, Label, Orthographic camera, Perspective camera, Point light, Spot light.

Now by using the Open Inventor file format and API, you can speed up the programming of OpenGL up to 10 times, and Inventor-viewers (like ivview) are designed specifically for use with OpenGL, resulting in very fast display of the scenes contained in the Inventor file.

Illustrator format

You can control what to retrieve from an Adobe Illustrator file.

Only up to Ilustrator version 8 is supported.

By setting **File > Import > File** Illustrator options, you can enable StudioTools to import the geometry as a **Group** of curves or as individual curves. You can also **Scale** the image before importing it into StudioTools.

When saving an Illustrator file choose **File > Print setup**, set **Output** to **File and** set the **Output Style** to Illustrator. The output results transfer the paint and shapes information. The results of opening this file in Illustrator is a display of three objects:

- the image plane displaying the final image (image plane).
- the curves that defines the shape.
- the shapes of the model which can be individually selected and manipulated.

If you want to change the geometry of the StudioTools model in Illustrator, export the model as a Post Script file from StudioTools. Open this file in Illustrator to manipulate the polygons.

VRML format

VRML (Virtual Reality Modeling Language) was developed with the intention of providing people with a standard tool that they could use to view and interact with 3D models on the Web.

The file structure of the VRML files is based on the Inventor format where there is a hierarchical arrangement of nodes.

VRML 2 resembles to the structure of VRML1 but uses slightly different keywords. E.g. vertex becomes point, DEF Default shader group becomes material DEF Default shader Material.

Features

- Cameras are converted to VRML2.0 Viewpoints.
- Ambient Lights are converted to a VRML point light with ambient properties.
- Per-vertex normals and texture coordinates (if desired).
- Material animation is converted.
- Animation is converted intelligently (only dynamic objects have animation information).
- Initial navigation mode can be specified in the translator.
- Double-sided/single-sided setting (in Render stats) is converted for groups.

- Does not support VRML2.0 embedded textures.
- The Inventor/VRML translator does not support annotation information.

These are the main differences between VRML 1 and VRML 2:

Version	Differences
VRML 1.0	Standard objects (cube, sphere, cone, cylinder, text)
	Arbitrary objects (surfaces, linesets, pointsets)
	Ability to fly through, walk through, examine scenes
	Lights
	Cameras (viewpoints)
	Textures on objects
	Clickable links
	Define and reuse objects
VRML 2.0 All VRML 1.0 features plus	Animated objects
	Switches
	Sensors
	Scripts (Java and JavaScript)
	Interpolators (colors, position, orientation, etc.)
	Extrusions
	Background colors and textures
	Sound (.wav and MIDI)
	Animated textures
	Event routing
	Define and reuse objects and behaviors and effectively add new nodes to the language with PROTO and EXTERNPROTO

To view and interact with VRML, you need a plug-in for your web browser which interprets the VRML language and makes the objects visible on the screen.

Examples of web browsers:

- Cosmoplayer
- Worldview
- Microsoft VRML 2.0 viewer
- Webscout (VRML1)

Index

Numerics

2D contours
cutting in 3D data (SLC) 35

A

accuracy
defining for solid modelers 9
ASCII
format for IGES files 95

B

browse
WebDAV resources 41

C

CAD data transfer
exporting files 114
rational and non-rational geometry 9
CAI formats
log file extensions 91
short log files 91
CATIA formats
DAG nodes names 130
data mapping 56, 125, 127
exchanging 3D data with 21, 24
exporting 130
exporting groups 130
exporting groups/ instances 130
layers 131
non-geometry data 129
object colors 132
summary of modeling practices 21, 24
using skins 23
colors
layer color 133
objects 132
preserving when exporting to CATIA 132
copy
WebDAV resources 42
creating
a solid model of a cube 6
model guidelines 21, 24

D

DAG node
definition 37
DAG nodes
exporting names to CATIA 130
data transfer
non-geometry for CATIA 129
debugging
Unigraphics translation 98
delete
WebDAV resources 43

E

exporting
Alias files to CAD 114
solid model of a cube 6
STL files 74
surfaces(faces) 6
tessellated files 34

F

feature
term in I-deas NX series 29
files
exporting to CAD 114
fillets
creating rational 9
Fused Deposition Modeling (FDM) 34

G

geometry
checking surface continuity 10
getting it right for data transfer 9
mapping process between Alias and CATIA 56, 125, 127
geometry types
groups/instances 129
non-geometric data 129
supported in Alias 125

H

HTTPS
WebDAV resources 42

I

I-deas NX series 27–28
modeling practices 27
importing
Unigraphics translation log files 96

L

Laminated Object Manufacturing (LOM) 34
layers
preserving when exporting to CATIA 131
linear units
base for data transfer 9
log files
importing into Unigraphics 96
interpreting importing files for Unigraphics 96
short for CAI 91

Unigraphics, interpreting
export log files 97

M

mapping
entities, STEP and Alias 123
Master Series 7&8
typical workflow for shelling 66
meshes
exporting in STL 76
modeling
before you create a model (CATIA) 21, 24
for CATIA 21, 24
for I-deas NX series 27
move
WebDAV resources 42
move command 42
multiset
creation for IGES 95

N

non-geometric data
supported for CATIA 129
NURBS
and tessellation 34

P

part files
term in I-deas NX series 29
PRenderToAlias program for Pro/ENGINEER 53

R

rational and non-rational geometry
for CAD 9

S

security
WebDAV resources 42
Selective Laser Sintering (SLS) 34
shelling
in I-deas NX series 29
typical workflow in MS 7&8 66
skins
using in CATIA 23
SLC formats
2D contours 35
format options 35–36
Solid Ground Curing (SGC) 34
Stereolithography (SLA) 34
stitching
in I-deas NX series 29
STL files
exporting 74
exporting meshes 76
exporting shells 75
surfaces
NURBS tessellation 34

T

TC DirectConnect 37
definition 37
how the translator works 37
tessellated files
exporting 34
tessellation parameters
for exporting to STL 74
tolerances
standards for CAD systems 9
translation log file
Unigraphics 96
translators
TC DirectConnect 37

U

Unigraphics formats
debugging 98
interpreting the export log files 97
translation log files 96

W

WebDAV
overview 41
WebDAV resources 42
browse 41
copy 42
delete 43
HTTPS 42