



CATIA V5 Direct Connect

StudioTools 11.0.2.0001

© 2001-2004 Alias Systems, a division of Silicon Graphics Limited.

The following are trademarks of Alias Systems:

Alias®	Alias PowerTracer™	Spider™
Alias Natural Phenomena™	Alias RayCasting™	StudioPaint™
Alias OpenAlias™	Alias RayTracing™	SurfaceStudio™
Alias OpenModel™	Alias SDL™	StudioTools®
Alias PowerCaster™	DesignStudio®	

Maya is a registered trademark of Silicon Graphics, Inc. exclusively used by Alias Systems.
SGI is a trademark, and IRIX is a registered trademark, of Silicon Graphics, Inc.

Microsoft and Windows® operating system are registered trademarks of Microsoft Corporation in the United States and/or other countries. Solaris is a trademark of Sun Microsystems Inc. All other product names mentioned are trademarks or registered trademarks of their respective holders.



All PTC Technology logos are used under license from Parametric Technology Corporation, Needham, MA, USA. Alias Systems, a division of Silicon Graphics Limited, is an independent software vendor.

All digital documentation ("Documentation") is copyright © 2001-2004 Alias Systems, a division of Silicon Graphics Limited ("Alias Systems") and contains proprietary and confidential information of Alias Systems. The Documentation is protected by national and international intellectual property laws and treaties. All rights reserved. Use of the Documentation is subject to the terms of the license agreement that governs the use of the software product to which the documentation pertains (the "Software"). The authorized licensee of the Software is hereby authorized to print no more than one (1) hardcopy of the Documentation per valid license of the Software held by such licensee. Except for the foregoing, the Documentation may not be translated, copied or duplicated in any form (physically or electronically), in whole or in part, without the express written permission of Alias Systems, a division of Silicon Graphics Limited.

The information contained in this document is subject to change without notice. Neither Alias Systems nor its employees shall be responsible for incidental or consequential damages resulting from the use of this material or liable for technical or editorial omissions made herein.

Not all features described are available in all products.



Alias Systems, 210 King Street East, Toronto, Canada M5A 1J7

Contents

About CATIA V5 Direct Connect 1

CATIA V5 Direct Connect 5

- Opening CATIA V5 files in StudioTools 5
- Saving CATIA V5 files from within StudioTools 8
- Converting CATIA V5 files to Alias format from the command line 11
- Converting Alias files to CATIA V5 format from the command line 13

About CATIA V5 Direct Connect

StudioTools CATIA 5 Direct Connect is a stand-alone utility that allows the exchange of 3D model data between StudioTools and CATIA 5 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 (R10, R11, or R12) and an appropriate CATIA license.
- Version 11.0.2 of SurfaceStudio, AutoStudio, Studio or DesignStudio, and a CATIA V5 Direct Connect license.
- IRIX or Windows operating system.
- On IRIX, `ftn_eoe` *must* be installed for CATIA V5 Direct Connect 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 1) 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** and **Curve edit > Join Curves** since these functions create multiknots in StudioTools geometry that may result in unusable geometry in CATIA.
- It's generally good practice in StudioTools to use surface building tools such as **Surfaces > Boundary surfaces > Square** and **Surfaces > Swept surfaces > Rail Surface** to ensure and control curvature continuity between surfaces.
- 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 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 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.

Tip 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.

CATIA V5 Direct Connect

Opening CATIA V5 files in 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 standalone 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 SplineCurve Circle Ellipse Conic Hyperbola Parabola Line IntCurve BsplineCurve MergedCurve SimCurve HelixCurve PLine PSpline PCircle PEllipse PParabola PHyperbola PNurbs	NURBS Curve of appropriate degree.
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.
Color RGB, Opacity, Layer, Show (Visibility), Pick (Template)	Shader Color RGB, Transparency, Layer, Visibility, Template.
Layer	Layer. Caution: no names are used, just layer numbers
Point	SpacePoint
Construction plane	Planar surface

CATIA V5 entity	brought into StudioTools as
CATIA Library Material	Not imported
Group	Not imported
Set (Group)	Not imported
Surface Patch Surfaces Plane NurbsSurface Cylinder Sphere Torus Cone FilletSurface ChamferSurface DraftSurface OffsetSurface SweepSurface GenericFillet GenericRuledSurface Tabulated Cylinder RevolutionSurface BspineSurface CircularSweep RuledSurface	NURBS Surface of appropriate degree
Template	pickable

CATIA V5 entity	brought into StudioTools as
Invisible entities	Invisible entities, unless the option -v is used when importing the file, or the environment variable ALIAS_CATIAV5_INVISIBLE is set to 1, in which case the CATIA invisible entities are ignored and not processed.
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, or the environment variable ALIAS_CATIAV5_DATUM is set to 1.

Saving CATIA V5 files from within StudioTools

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.

Studio entity	saved to CATIA V5 format as
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.
InstanceCopy	NonRationalSurface. No history.
Stitch	Solid. Caution: Alias closed shell becomes a CATIA Solid. If -s option is specified, the shell's faces will becomes 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.
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.
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

Studio entity	saved to CATIA V5 format as
Shader Color RGB, Transparency	Color RGB, Opacity.
CurvePoints	Not converted
Layer	Layer (numbered). Layer name is ignored.
Invisible	Imported as Invisible (hidden)
Group	Not converted
Stage	Not converted
Locator	Not converted
Vector	Not converted
Template	Pickable
Construction planes	Not converted
Symmetry	Not converted

Converting CATIA V5 files to Alias format from the command line

Alias provides a stand-alone utility, C5ToAl, to enable you to convert CATIA Part (.CATPart) and CATIA product (.CATProduct) document files to Alias Wire files without running the StudioTools application. You may find this useful for creating a batch script if you have many files that need to be translated from the CATIA format.

Usage:

C5ToAl [<options>] -i <input CATIA file> -o <output Wire file>

where

Argument	Value
-i	input CATIA filename (either .CATPart or .CATProduct)
-o	output Wire filename (.wire)
-l	logfile option: 0=No logfile output (default) 1=Output a logfile (-l1) 2=Output an extended logfile (-l2)
-s	do not stitch surfaces of solid bodies (stitch by default)
-d	convert invisible datum (autonomous) geometric entities
-v	do not convert invisible (hidden) geometric entities
-h	display this help information, then exit

Using stand-alone utilities on Windows

- 1 Choose **Programs > Accessories > Command Prompt** from the Start menu.
- 2 Click the cursor in the window.
You can see what directory you are in by typing
dir
A list of the directory contents will be provided; at the top of the list is your current location.

- 3 Navigate to the directory containing the files to be converted. If you have installed StudioTools in its default location, the path will be similar to
`C:\aw\[UserID]\Alias\user_data\[Project]\wire`
 where [UserID] is replaced by your login userid (and no square brackets) and [Project] is replaced by the name you have assigned to the project: the default project is demo (again, no square brackets). So to navigate to the directory, type
`cd C:\aw\[UserID]\Alias\user_data\[Project]\wire`
- 4 To use the stand-alone utility to convert a CATIA file named "headlight.CATPart" to a StudioTools wirefile named "headlight.wire" without autostitching, and with a logfile created, type
`C5ToAl -l1 -s -i headlight.CATPart -o headlight.wire`

Using stand-alone utilities on IRIX

- 1 Open a UNIX shell through the Toolchest or select UNIX shell from the Studio Utilities menu.
- 2 Click the cursor in the window.
 You can see what directory you are in by typing
`pwd`
- 3 Navigate to the directory containing the files to be converted. If you have installed StudioTools in its default location, the path will be similar to
`/usr/u/[UserID]/Alias/user_data/[Project]/wire`
 where [UserID] is replaced by your login userid (and no square brackets) and [Project] is replaced by the name you have assigned to the project: the default project is demo (again, no square brackets). So to navigate to the directory, type
`cd /usr/u/[UserID]/Alias/user_data/[Project]/wire`
- 4 To use the stand-alone utility to convert a CATIA CATPart file named "headlight.CATPart" to a StudioTools wirefile named "headlight.wire" without autostitching, and with a logfile created, type
`C5ToAl -l1 -s -i headlight.CATPart -o headlight.wire`

Converting Alias files to CATIA V5 format from the command line

AlToC5 - Convert Alias Wire files to CATIA Part (.CATPart) files.

Usage:

AlToC5 [<options>] -i <input Wire file> -o <output CATIA file>

where

Argument	Value
-i	input Wire filename (.wire)
-o	output CATIA filename (.CATPart)
-l	logfile option: 0=No logfile output (default) 1=Output a logfile (-l1) 2=Output an extended logfile (-l2)
Surface (divide/split) options:	
-c	divide closed/periodic surface in multiple surfaces
-d	divide surface with multi-knots in multiple surfaces
Shell stitch (join/close) options:	
-t <tol>	merging tolerance for stitching shell faces (default 0.001 cm)
-k	keep shell faces after stitching
-g	stitch open shell faces
-s	do not stitch open/closed shell faces
-f	confirmation for overwriting an existing CATIA Part (.CATPart) file
-v	do not convert invisible geometric entities
-h	display this help information, then exit

