

Design and Engineering Collaboration with Autodesk Alias and Autodesk Inventor

Collaborate effectively on 3D design data

Reliable data exchange with CAD software is important throughout the design process. Autodesk Alias helps solve CAD integration issues by providing fast, high-quality data exchange with Autodesk® Inventor® software. By working closely together, designers and engineers can effectively address both aesthetic and functional requirements to develop the best design solution.



Style, form and function all influence a customer's choice of products. These days, your customers want innovative designs for less, requiring you to be as efficient and productive as possible. It's not enough to just use the latest 3D tools anymore. In today's highly competitive market, collaboration between design and engineering teams is essential. By working closely together, designers and engineers can effectively address both aesthetic and functional requirements to develop the best design solution.

The Autodesk solution for Digital Prototyping, which includes the Autodesk® Alias® family of products and the Autodesk® Inventor® product line brings together design data from all phases of the product development process to create a single digital model, enabling industrial designers and engineers to design, visualize and simulate their products.

This white paper introduces the interoperability between Autodesk® Alias® and Autodesk® Inventor®. The interoperability between Alias and Inventor helps industrial design and engineering professionals communicate more effectively by directly reading native Alias WIRE files into Autodesk Inventor. Alias files can now be associatively or non-associatively opened in Inventor, which mitigates potential translation errors into generic 3D file formats (i.e. STEP or IGES). In addition, the associative link between Alias and Inventor is very powerful; industrial designers are able to maintain their original design intent throughout the entire development process and engineering is able to leverage the industrial design surfaces without rebuilding the model.

Building Alias Geometry for Inventor

To achieve the tolerance required by solid modeling, it is important to manage the construction units and tolerances when creating your model. Most engineering organizations have developed standards for tolerances that they apply to their CAD systems. If you are not clear as to which standards your company or clients use, seek out your CAD system manager for more information.

For efficient data transfer from Autodesk Alias to Autodesk Inventor software, it is important to set the correct Construction Options before the modeling begins.

The Inventor preset in the Construction Options (**Preferences > Construction Options**) is a good starting point and should suffice for most users. There are times that might require the user to fine tune these settings such as changing between imperial and metric units. By Creating a Copy of the default Inventor Preset, the user will be able to modify the settings.



Evaluating the Geometry in Alias

Before exchanging data with the engineering team it is essential to validate the quality of the model to find any potential problems that might appear downstream in the development process. Within Alias, users have the ability to check the model globally or per-surface. The following tools are the most common for validating data quality prior to data transfer.

Check Model

Palette > Evaluate > Check Model

The Check Model tool allows you to check for potential problematic conditions - chose either “all of the visible” or “selected” geometry. Additional Check Model options are offered in Alias Surface and Alias Automotive software products.



Autodesk Alias Family of Products

Autodesk® Alias® Design

Tailored for consumer products designers, it provides all the tools you need to explore concepts, refine concept models, create realistic imagery, develop high-quality surfaces, and share data with engineering teams.

Autodesk® Alias® Surface

Evolve concept models and scan data into high quality production surfaces for consumer product designers. Create Class-A surfaces for automotive design and styling with Autodesk® Alias® Surface modeling software. Designed for virtual modelers, the software offers dynamic modeling capabilities for developing, refining, and controlling shapes.

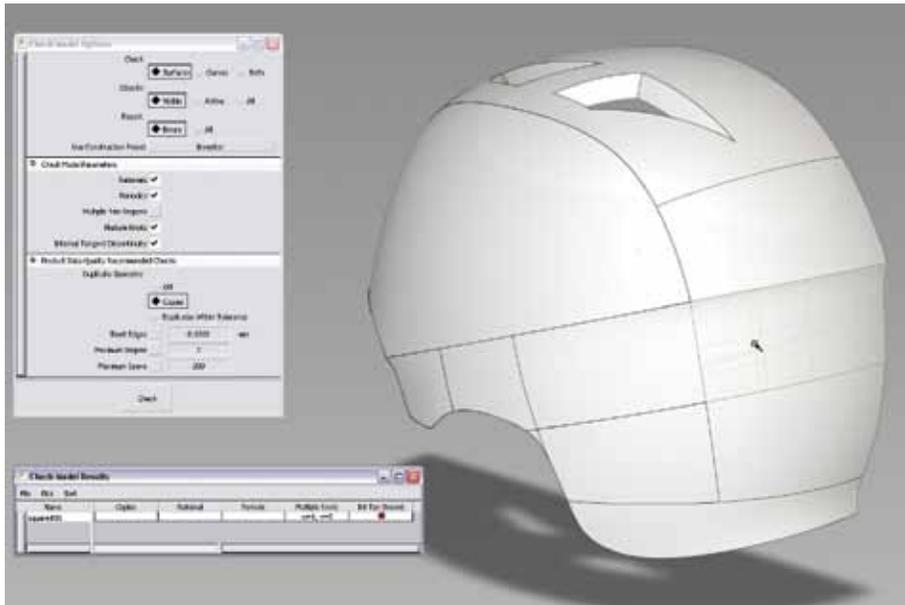
Autodesk® Alias® Automotive

Autodesk® Alias® Automotive is an industry-leading application for automotive design and styling, and the choice of the majority of automotive styling studios throughout the world. The software provides a comprehensive set of visualization and analysis tools for the entire shape-definition process, from concept sketches through Class-A surfacing.

TIP Model Verification

Test geometry created in Alias to predict and diagnose problems when transferring to Autodesk Inventor

In the screenshot below, Check Model has red-flagged a potential problem that needs to be addressed.



Key Benefits of Design and Engineering Collaboration

1. Create production-quality geometry that can move back and forth between Alias Design and Autodesk Inventor for collaborative product development.
2. Design and engineering teams can work together so both aesthetic and functional requirements are addressed when developing design solutions.
3. Create sketches at real-world scale, and draw over imported Inventor data to ensure design feasibility.
4. Verify the quality of created surfaces, including curvature and radii analysis, to help ensure that geometry can be used by Autodesk Inventor.
5. Test geometry created in Alias Design to predict and diagnose problems when transferring to Autodesk Inventor

Surface Continuity

Palette > Evaluate > Continuity > Surface Continuity

The Surface Continuity evaluation allows users to check the relationship between two surfaces based on the position (Go), tangent (G1), and curvature (G2) continuity. Green indicates that the continuity is acceptable between surface patches and yellow would indicate that the continuity is not within tolerance. This feature can also be activated within the surfacing tools so adjustments can be made prior to exiting the tool.



Take your 3D models to the next level. Evolve concepts through a creative, iterative process. The concept development phase demands close collaboration between designers and engineers to produce concepts that are new, yet technically feasible.

Stitching

Palette > Surface Edit > Stitching

The Stitching tool typically is used to stitch surfaces together to create a solid topology shell. It can also be used as a quick way to verify if the data will easily stitch or shell in Autodesk Inventor. Gaps or edges will show up in yellow when selected.

In the image below, the red arrows are pointing to the open edges of the surface model. If the arrows and yellow lines are within the model, then the surfaces would need to be investigated and corrected to make a valid solid.



Using Alias Data within Inventor

Open > Import of Place Component

The Alias file format, WIRE can be selected when you use Open, Import or Place Component.

When a WIRE file type is selected, several options are available to determine how Inventor will import the data. The options can be found at the bottom of the file



Our Alias surfaces are so high-quality we can go straight to tooling. We're cutting our turnaround time on projects tremendously --which helps us get to market faster.

—Kurt Bender
Design and Engineering
Viking Yacht Company

Option Details

Import Type

- **Associative Import (recommended method)**
Select the check box to import the Alias file and maintain associativity between the Inventor file and the Alias source file. If changes are made to the Alias file after import, the Inventor file will allow for automatic updates. If Associative Import is selected, all other options in the dialog box are unavailable and do not apply.
- **Non-Associative Import**
Select the check box to import the Alias file without association to the Alias source file. This method is for users that want to manually redefine Alias surface features in the development cycle.



Save Options

- **Save Components during Load**
Select the check box to save the component files in Autodesk Inventor format during the import process. Choose where to save the components from the drop-down menu. If you choose to Select Save Locations, the Component Destination folder becomes available. This setting is not available when the Import command is used.
- **Component Destination Folder**
Sets the location for the component files created from the import operation. If Save in Workspace is selected, this folder is defined in the Edit Projects panel.

Entity Types to Import

- **Solids**
Select to import solid bodies and water tight stitched shells as individual solid bodies
- **Surfaces**
Select to import surface bodies. Water tight stitched shells are imported as solid bodies
- **WIRE Files**
Select to import WIRES

TIP

Specify file destinations that are included in the active project or add the paths to the project to assure that referenced files resolve when you open a file.

Data Organization

- **Create Surfaces As**
Select the surface types to create during the import. Choose from:
 - **Individual Surface Bodies**
To import each surface as a single surface body in the part environment.
 - **Single Composite Feature**
To import the surfaces as a single composite in the part environment.
 - **Multiple Composite Features**
To import the surfaces as multiple composites in the part environment.
 - **Single Construction Group**
To import the surfaces as a single group in the construction environment.
 - **Multiple Construction Groups**
To import the surfaces as multiple groups in the construction environment.
- **Group Name to Place Data**
Select a group name for the imported data. The group name will show in the browser.

Units

- **Import Units**
Converts the imported geometry and parameter values to the selected units.

Post Processes

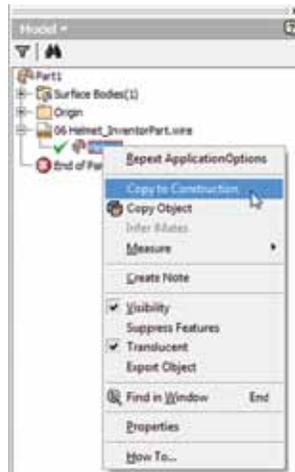
- **Check Parts during Load**
This quality check of the imported data is automatically performed when you select to create surfaces as composite features. It is not performed when you select to create surfaces as construction groups. If bad data is found, the composite is marked within the browser and the remaining bodies are not checked.
- **Auto Stitch and Promote**
When selected, Inventor attempts to stitch surfaces into a quilt or solid. If the surfaces are stitched into a single quilt or body, the resulting quilt or body is promoted to the Part Environment. Otherwise, the surfaces remain in the Construction Environment.
- **Enable Advanced Healing**
If selected, slight alterations in the surface geometry are allowed to stitch the surfaces.

Checking Surfaces in Inventor

Once you have successfully opened a WIRE file in Inventor you need to make sure that Inventor doesn't have any issues with the surfaces so you will need to perform a quality check in Construction Mode.

Copy the target surfaces into the Construction.

Expand the Alias WIRE file icon in the browser and right click on the surface icon and select "Copy to Construction"



We have chosen the right products from the Autodesk Digital Prototyping product portfolio. With them, we create 3D CAD models, high-end visualizations and animations, simulation results, and rapid prototyping models.

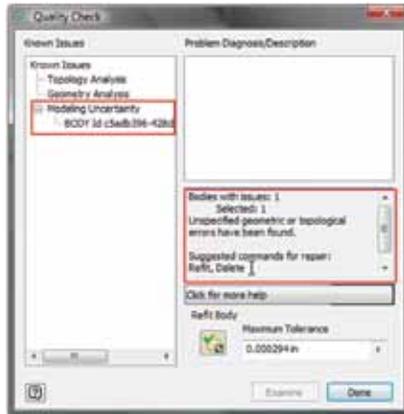
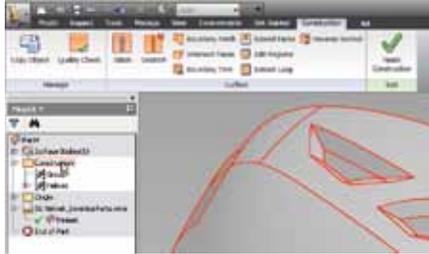
—Christian Sperka
CIO
Franke

Activate the Construction Mode.

Double click the Construction folder in the browser and it will activate a new Construction tab in the ribbon. You can now perform the quality check.

Click on Quality Check. Select your Alias surfaces and click examine.

The quality check will either show no issues (image on the left) or will show the “known issues” (image on the right). If there are errors, the software will indicate the type of error and provide suggested repair steps.



Click on Finish Construction

If there are no errors found in the Alias surface geometry you can return to the modeling environment and continue to work on creating the solid body. Clicking on “Finish Construction” in the ribbon will return you to the modeling environment.

TIP

- Construction Mode inside Inventor also has a number of surface editing capabilities such as: reverse normals, create boundary patches, extend faces, stitch, unstitch, etc.
- The Quality Check dialog box will tell you how many bodies are selected for examination.

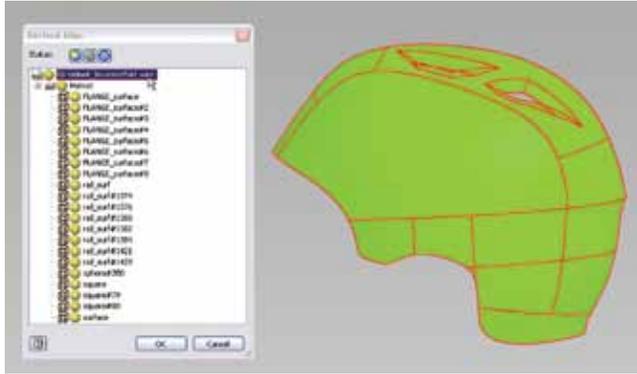
TIP

Once you exit the construction environment, turn off the visibility of any surface geometry. This will ensure that you do not have any surface artifacts on the display of your solid body once it has been created. You can also return to the construction environment if you need to copy or reference any of the original surfaces for solid features creation, such as extend “up to” or “terminate at”.

Converting Surfaces to a Solid

Select File to Open or Import. The Derive Part interface will appear allowing you to select only the geometry or layers that are required. Selecting the top item will open all available geometry.

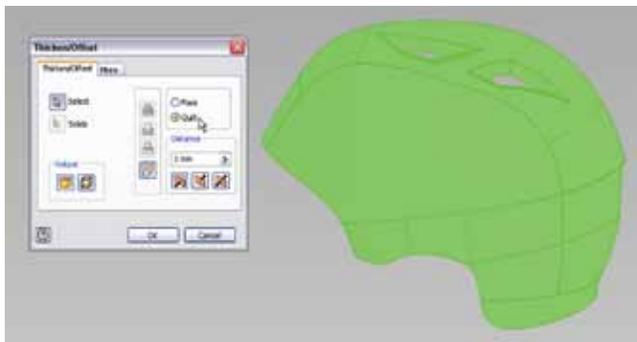
Only visible layers in Alias will be accessible by Inventor.



The Stitch feature can be found in the modeling section of the ribbon bar. This feature will quilt or stitch the surfaces together to be a single feature.



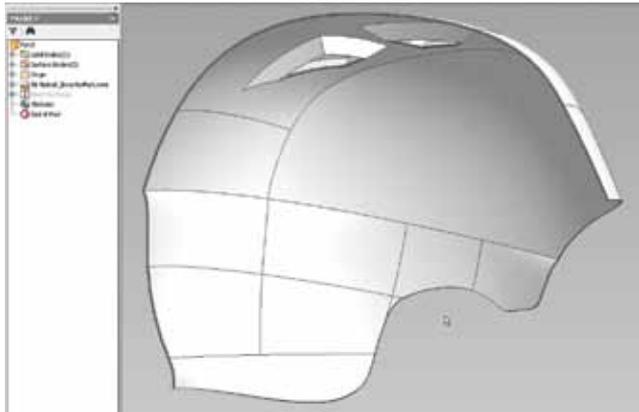
At this point in the design, the surfaces can now be used with many of the Inventor features. For example, you could change/add to the part thickness with the “Thicken/Offset” feature found in the Modeling section of the ribbon bar.



In the past, we would receive storyboards from external industrial design firms. Now, industrial designers send surface models designed in Alias software to us and we bring them directly into Inventor. Our mechanical engineers are actudesigning around these surface models.

—Dave Bosworth
 Manager, Enterprise Engineering Systems
 Parker Hannifin Corporation

Once your model is a Solid Part feature, it can be used with all Inventor tools and simulation functionality.



Alias Design helps us bring our products to the marketplace in less time. It is an essential part of our business and keeps us ahead of the competition.

—Marcus Ting
Founder and Designer
Sumajin Pte. Ltd.

Updating the Alias Association

Global Update

Use the following workflow to update your Inventor model when the original Alias model has changed.

Open the part to be updated in Inventor. Look for the global update notifications in the feature tree and in the title bar.

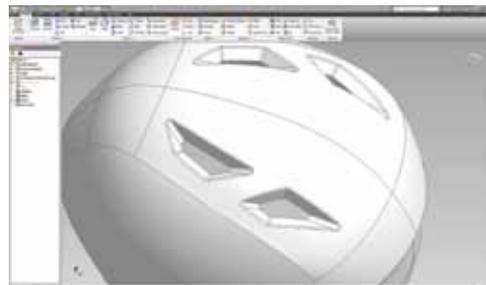


Original

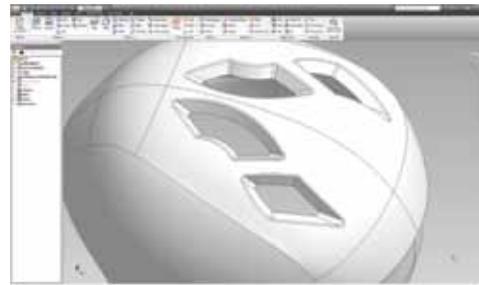


Updated

Selecting the Update Notification in the title bar will either update the associativity automatically or will allow you to manually update the association.



Original

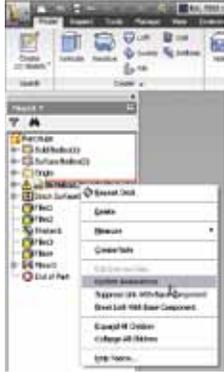


Updated

Using Update Association

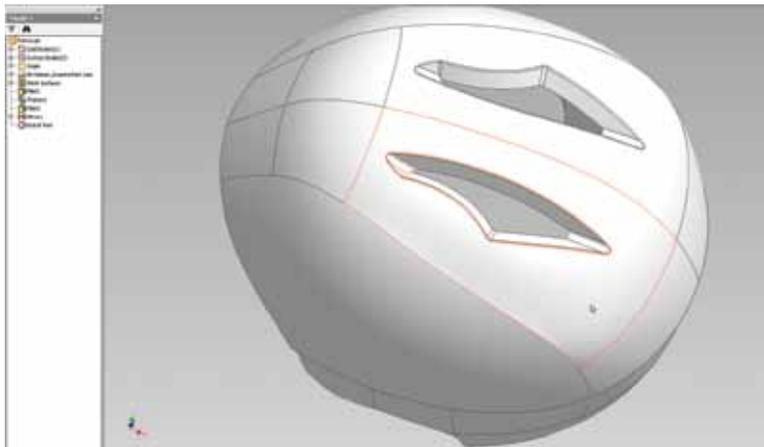
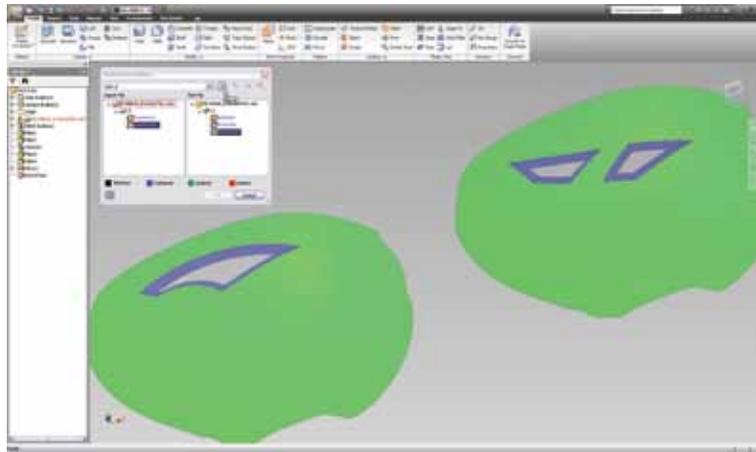
If a caution icon appears after performing an update, then there is a feature that cannot be automatically resolved. Often this is because the number of surfaces has changed.

- Update Association
Right Mouse Button (RMB) on the WIRE Feature and choose “Update Association”.



- Match the Surfaces
The surfaces must be matched or removed from the feature associations in order to proceed. For example, in the image below, let’s say the number of vents has changed so the extra surfaces of the vent in the original will need to be removed from the associations.

Use the “Match” option to link the remaining surfaces.



By embracing Autodesk Inventor and using it to create digital prototypes, we were able to take the development timeline from roughly 18 months to 9 months. In addition to the time savings, there’s a tremendous amount of money saved and benefits to be gained just simply getting to market sooner.

—Jason Faircloth
Product Manager
Marin Bikws

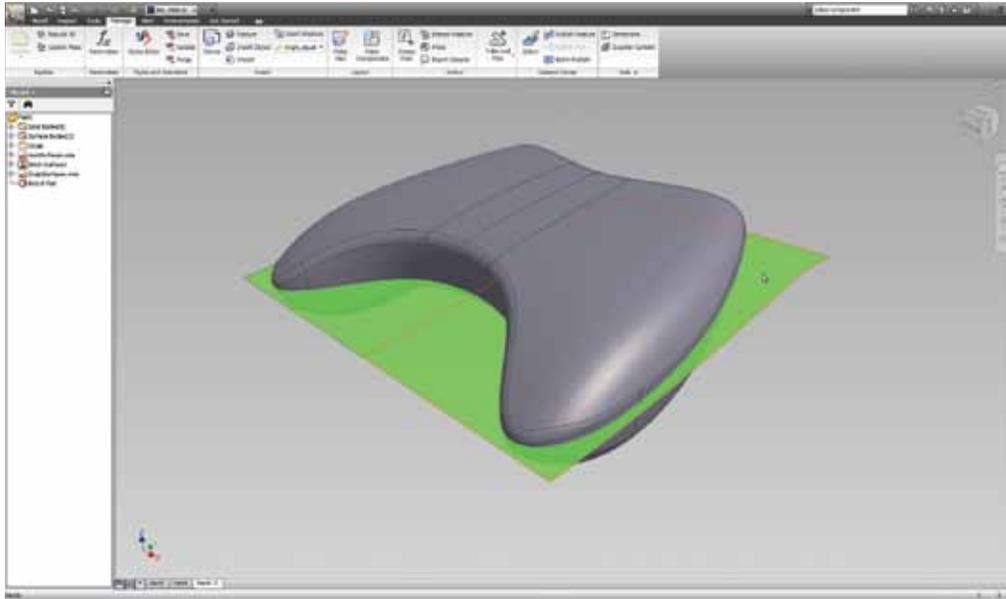
TIP

Often there will be features that remain in the browser that no longer are required. These can be “deleted”.

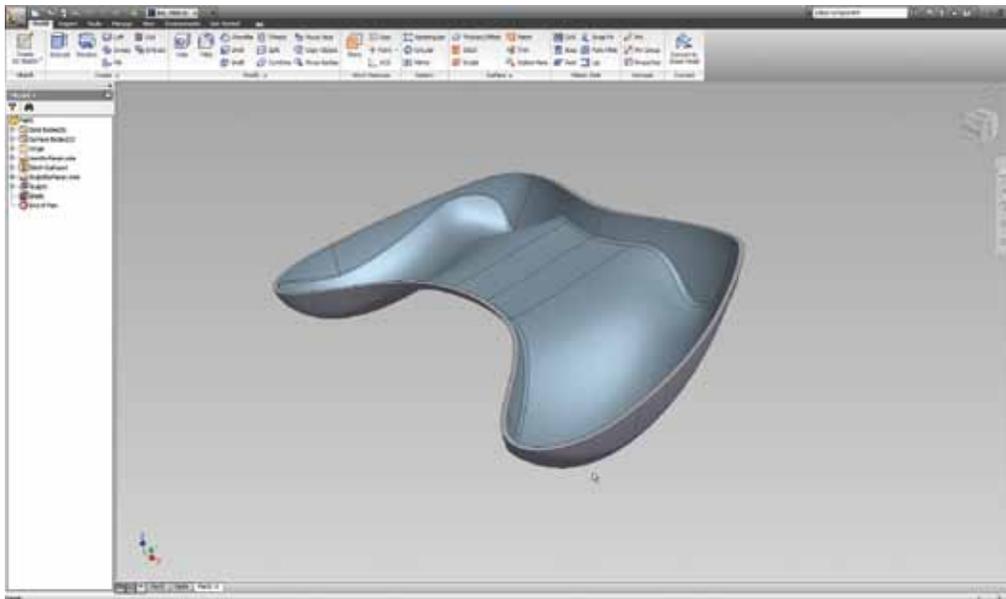
Building Sculpt Geometry with Alias Surfaces

A surface developed in Autodesk Alias can be used to cut a volume in Autodesk Inventor. This capability is useful to add complex characteristics, such as creating organic shaped parting surfaces for Inventor.

After the surfaces have been imported into Autodesk Inventor, select the surface, and from the “Model” tab of the ribbon bar, choose Surface > Sculpt.



Surfaces are imported to trim the Controller Housing



Results from Sculpt surface feature

Alias reduces the gap between design and engineering. The early visualization allows us to explore more extreme shapes as well as to sell our best ideas to the marketing groups.

—Alan Hocknell
VP Innovation and Advanced Design
Callaway Golf Company

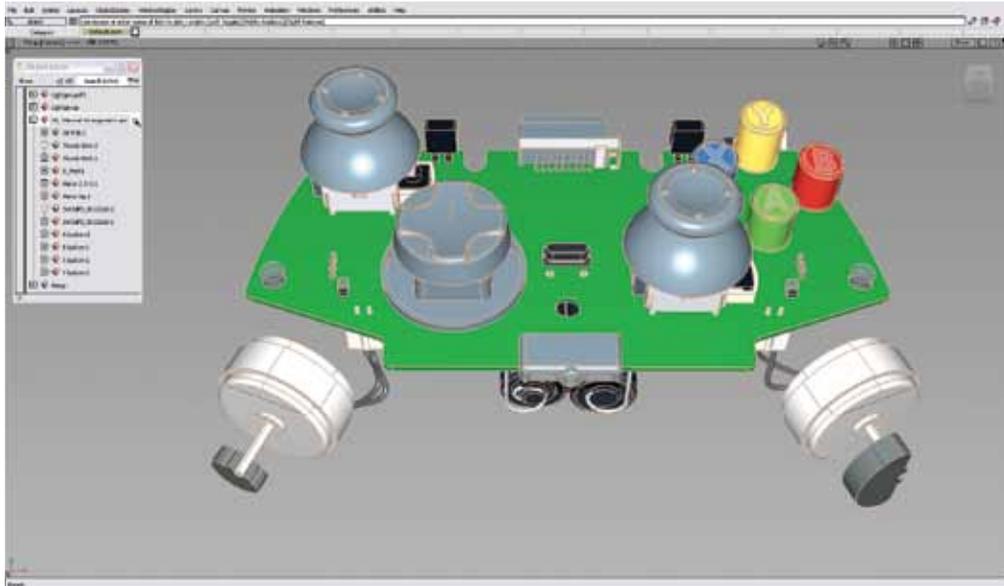
Importing Inventor Data into Alias

Inventor .IAM and .IPT files can be imported directly into Autodesk Alias.

The file can be opened or imported in several ways in Alias:

- Choose **File>Open** to open the file into a new scene or stage.
- Choose **File>Import>File** to add the data to an existing scene.
- Drag the file from Windows® Explorer to open the file to a new scene or stage.

Once you bring the solids in as stitched surfaces, the objects can be selected as a standard Alias object and assigned to layers as normal.



TIP

Warning: If the imported construction settings are different from the current construction settings, Autodesk Alias will prompt you to either “Keep Current Settings” or “Accept New Settings”. If your current settings are set up to your liking, then it is recommended to keep your current settings.

Referencing Inventor Data into Alias

Inventor .IAM and .IPT files can be translated into a reference file using the Alias Reference Manager. The Reference Manager, **File > Reference Manager**, converts the data into a separate file and allows it to be rapidly loaded and unloaded without adding to the host file size.

Benefits include

- Fast loading and unloading of large data sets
- Pre-Optimized files are only referenced into Alias so they will not add to the file size.
- Surfaces can be promoted or imported as needed.
- Variations can be easily managed

Create a Reference File

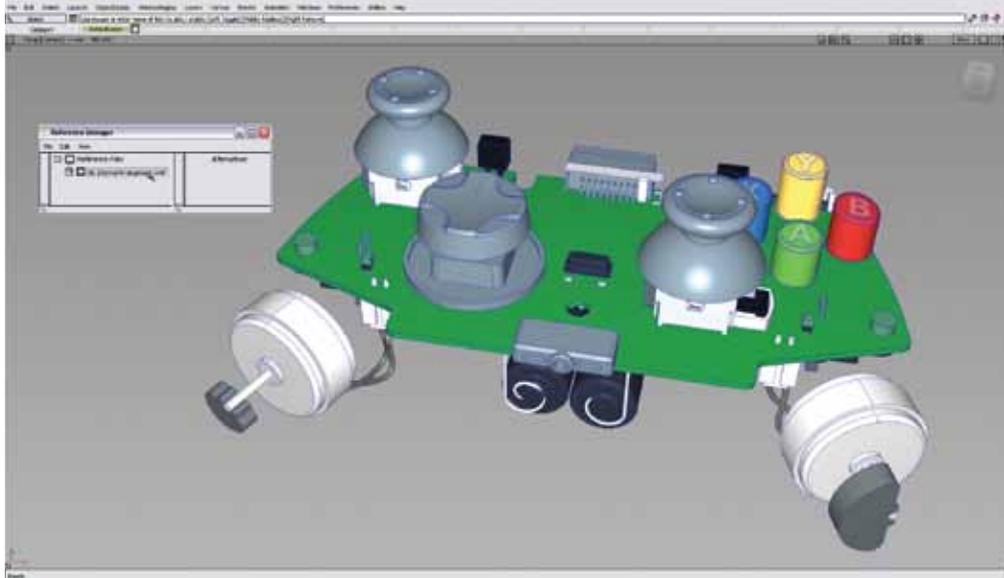
Reference Manager > File > Translate

Translate the Inventor file into a reference file prior to importing into Alias. This will generate a WREF file with the desired tessellation values. These settings affect the quality of the shading and display of the WIRE frame.

Import the Reference file.

Reference Manager > File > Import

Select the .WREF file to instantly display it in Alias.



TIP

- The Reference Data can only be selected with the “Pick Reference” data tool.



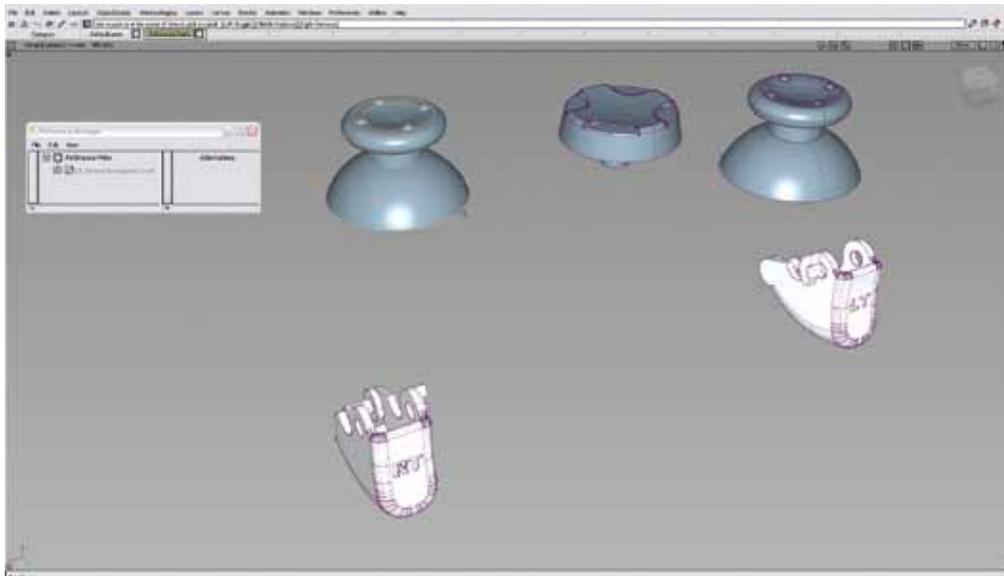
Selecting and Promoting Geometry

Palette > Pick > Reference

Reference Manager > Edit > Promote Geometry

Once the entire data set has been referenced into Alias, you will most likely need to work with a few of the reference surfaces or parts in order to build additional details from these elements. Your goal should be to only bring in the data you require to minimize the impact on file size and performance.

Once promoted into Alias, these can be assigned to Layers as normal. The rest of the reference data can be hidden until needed.



Using Illustrations from Alias in Inventor

Import Autodesk Alias sketches into Autodesk Inventor to use as guides for building models. In this example, we will focus on a top view orthographic illustration.



Autodesk Alias Design helps us bring our products to the marketplace in less time. It is an essential part of our business and keeps us ahead of the competition.

—Marcus Ting
 Founder and Designer
 Sumajin Pte. Ltd.

- Make sure that **Canvas_Top** is selected in the **Windows>Construction Plane Editor**.
- Export the Canvas Picture

File > Export > Make Canvas Picture > option box

Make Picture allows you to specify image size and preferred file format - JPEG, TIFF or BMP when exporting an image. The “Blend With Background Color” will use the color of the modeling background to fill any transparent areas of the illustration.



- In Inventor, open or create a new sketch and choose **Insert > Insert Image**. If you get a message at this point asking if you want to use Microsoft® Paint as your default editor, click Yes or the software will not import your image.
- The cursor is attached to the upper-left corner of the image. Click in the graphics window to place the image.

TIP

To create a TIFF with an Alpha Channel, the “Blend with background color” feature must be turned OFF.



2D Drawing Support in Alias

Autodesk Alias supports DWG™, DXF™ and exporting of DWF file formats.

Most of the drawing data is supported by Alias and will load as curves. This includes the Sheet Frame and Dialogs, Dimensions, and many more. Dimensions are not recognized as such in Alias so they do not display values.

In Inventor, use **File>Save Copy As**. In the Save as Type menu, choose either AutoCAD Drawings (*.dwg) or DXF Files (*.dxf) options.

Conclusion

Design and engineering teams no longer have to struggle to collaborate on 3D design data. Working collaboratively helps to solve integration problems by providing fast, high quality data exchange between Autodesk Alias and Autodesk Inventor software.

The Autodesk solution for Digital Prototyping enables manufacturing workgroups to create a single digital model that can be used in every stage of production, bridging the gaps that usually exist between conceptual design, engineering, mechanical design, electrical controls design, and manufacturing teams.

Design, visualize and simulate your design before producing a physical prototype to better enable collaboration and innovation. You'll save time and money, design better products, and beat your competition to market.

The digital prototype allows teams to easily validate their designs against the work of other teams. Designs progress more quickly and without collisions. With Digital Prototyping, our watch movements and cases are largely refined during design, so we produce just one physical prototype instead of three.

—Pierre Gyax
Executive Vice President of Production
Ulysse Nardin

TIP

DWG and DXF files load with cm default units (this means all the model's dimension will be interpreted as being in centimeters). Depending on the original file units, you might need to adjust the size of the model by selecting it and scaling it accordingly in Alias.

Appendix

How Solid Modeling Works

The following information provides more detail about the concepts of solid model data types. This information is helpful in understanding 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.
- **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.

Tolerance Requirements for Solid Modeling

To achieve the tolerances required by CAD applications, it is important to use the appropriate modeling units and tolerances when creating your model. Alias offers preset tolerances and settings for various engineering-based CAD systems which can be further modified if required. These settings should be selected prior to beginning work as they affect the creation and evaluation of surfaces, curves and trimming operations.

Get the Topology Right before Transferring Data

The **Surface Edit > Stitch** tools in Alias creates a valid solid model topology. Creating solid geometry from imported Alias models is a common workflow when integrating Alias and CAD systems. Prior to exporting the model from Alias, you must ensure that the geometry is built to the correct tolerance and that it can be stitched. Stitching in Alias identifies gaps between surfaces so that you can repair the geometry before exporting the file.

During the stitching process, surfaces are modified from their original state. Modifications include deleting of construction history, the splitting of boundaries to accommodate adjacent surfaces, and splitting periodic geometry into multiple surfaces. For this reason, you should either “Keep Originals” found in the Stitching tool or save the file before stitching so you have the option of keeping the construction history intact.

Importing Data into a Solid Modeler

When you import an Alias model into a solid modeler, the data must satisfy the solid modeler's rules for topological and geometric data in order to produce a valid part.

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 Alias and then export the data.

Rational and Non-Rational Geometry

In the Constructions Options dialog box, you can specify whether the new geometry being created will contain rational or non-rational components for primitives, fillets, and curves/surfaces.

Non-rational geometry assumes all CVs weight equals “1.0”. Rational geometry allows each CV to carry its own weight. The different weighting of the CVs allows the shape to be an exact radius value. It is acceptable to convert non-rational geometry to rational geometry, but not the other way around. The shape would change if the weights are all made to 1.0 when converting from rational to non-rational.

For example, if you build a circle with non-rational geometry with a radius of 100mm, when the radius is measured the actual value is allowed to float within a tolerance near 100mm. For rational geometry, the result would be exactly 100mm. However, some CAD systems may only allow non-rational geometry. So if Alias users create models in rational geometry, their results in those systems will change the shapes. Since Inventor supports rational geometry, there is not a concern for Alias users. They can create models rational or non-rational.



About Autodesk Alias

The Autodesk® Alias® product line — Autodesk Alias Design, Autodesk Alias Surface and Autodesk Alias Automotive — addresses the unique creative requirements of the entire industrial design workflow. Part of the Autodesk Solution for Digital Prototyping, the product line provides a comprehensive and powerful set of tools for conceptual design, surface definition and design realization. Optimize your design process with industry leading sketching, modeling, and visualization tools that quickly help you bring ideas to reality. For more information about the Autodesk Alias family of products go to www.autodesk.com/aliasfamily.

About Autodesk Inventor

The Autodesk® Inventor® product line provides a comprehensive and flexible set of software for 3D mechanical design, simulation, tooling creation, and design communication that helps users cost effectively take advantage of a digital prototyping workflow to design and build better products in less time. Autodesk® Inventor® gives engineers the ability to integrate AutoCAD and 3D data into a single digital model that can be used to validate the form, fit, and function of a design before it is built. By enabling engineers to design, visualize, and simulate their products digitally, Inventor helps teams communicate more effectively, reduce errors and deliver more innovative product designs. To learn how Autodesk Inventor takes you beyond 3D to Digital Prototyping, go to www.autodesk.com/inventor.