AUTODESK UNIVERSITY 2014

Complex and Organic Shapes Using Surfacing and Free-form Tools in Inventor

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Inventor 2015 software offers a new modeling approach that gives users more alternatives, flexibility, and freedom to design models with smooth and flowing shapes commonly found in consumer products. Attend this class and learn how to start using this new modeling technique to create very complex and stylized designs. Learn how to select the suitable technique for your design, choosing between conventional and free-form modeling methods. You will also learn how to combine both modeling techniques when needed.

Learning Objectives

At the end of this class, you will be able to:

- Discover the new flexible modeling method in Inventor 2015 software
- Learn how to define the differences between surfacing and free-form tools
- Discover how to select the most appropriate modeling method for your designs
- Discover how to combine surfacing and free-form tools in a design

About the Speaker

Sergio is a Mechanical Engineer, accomplished designer and an Autodesk Certified Instructor (ACI-07451217) with more than 8 years of experience working with Autodesk Manufacturing Solutions. He has extensive experience and knowledge in CAD, design automation, simulation (CAE), visualization as well as product data management (PDM). He has a proven success in training students and clients on CAD, CAE and PDM applications. Sergio has a strong ability to assess business issues and understand the client's current processes in an effort to formulate implementation plans. He has demonstrated success in implementing data management solutions (PDM) that suit clients' needs and in advising them on the best practices. He assists clients in design solutions, optimization and efficient workflows.

Currently Sergio works as an Applications Specialist for SolidCAD Solutions, providing professional consulting, implementation, training and support services.

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Introduction

In general, ideation and concept modeling have been a task for industrial designers. When designing, they want to have complete freedom in order to achieve aesthetic and ergonomic products without having any type of restriction or limitation.

Inventor has always been an engineering tool, but now the latest release is overcoming the barriers of conventional modeling techniques by including a new alternate approach. This new flexible and very intuitive method, named Free-form, enables you to create very complex and organic shapes breaking free from conventional rigid surface and parametric modeling workflows. Save time and become more productive when exploring and presenting multiple design variations.

Now, Inventor offers the traditional hybrid parametric workflow (parametric surfaces and solids) along with this new freeform technique bringing industrial and mechanical design together in a single powerful application. Therefore, users can learn this new modeling tools and add them to their Inventor skillset. In doing so, they will avoid importing and exporting geometry from other applications (such as Autodesk Alias) to Inventor when is not required. Thus, designers and engineers can focus on the aesthetics, ergonomics and functionality of the digital prototype within the same CAD application.

In this document, you will learn how to use the new freeform tools alone or in combination with the conventional surface modeling. You will be able to differentiate the pros and cons of both modeling approaches to select the most appropriate workflow for your designs. Keep in mind that you will end up using one of them or both together when needed.

New flexible modeling method in Inventor 2015 software

Overview

Today Inventor presents a new modeling method that enable users to very rapidly explore new concepts and generate ideas. Consequently, they can be quickly presented to prospects and clients meeting their requests in a very short time. In a collaboration environment, this new modeling approach allows the design team to greatly accelerate the design process since sharing and evaluating concepts is far faster than before.

Sometimes it is really hard or even impossible to use parametric tools to achieve the desired design shape. The new Free-form modeling technique gives users more alternatives, flexibility, and freedom to design models with smooth and flowing shapes commonly found in the automotive, marine and aerospace industries, as well as the consumer products.

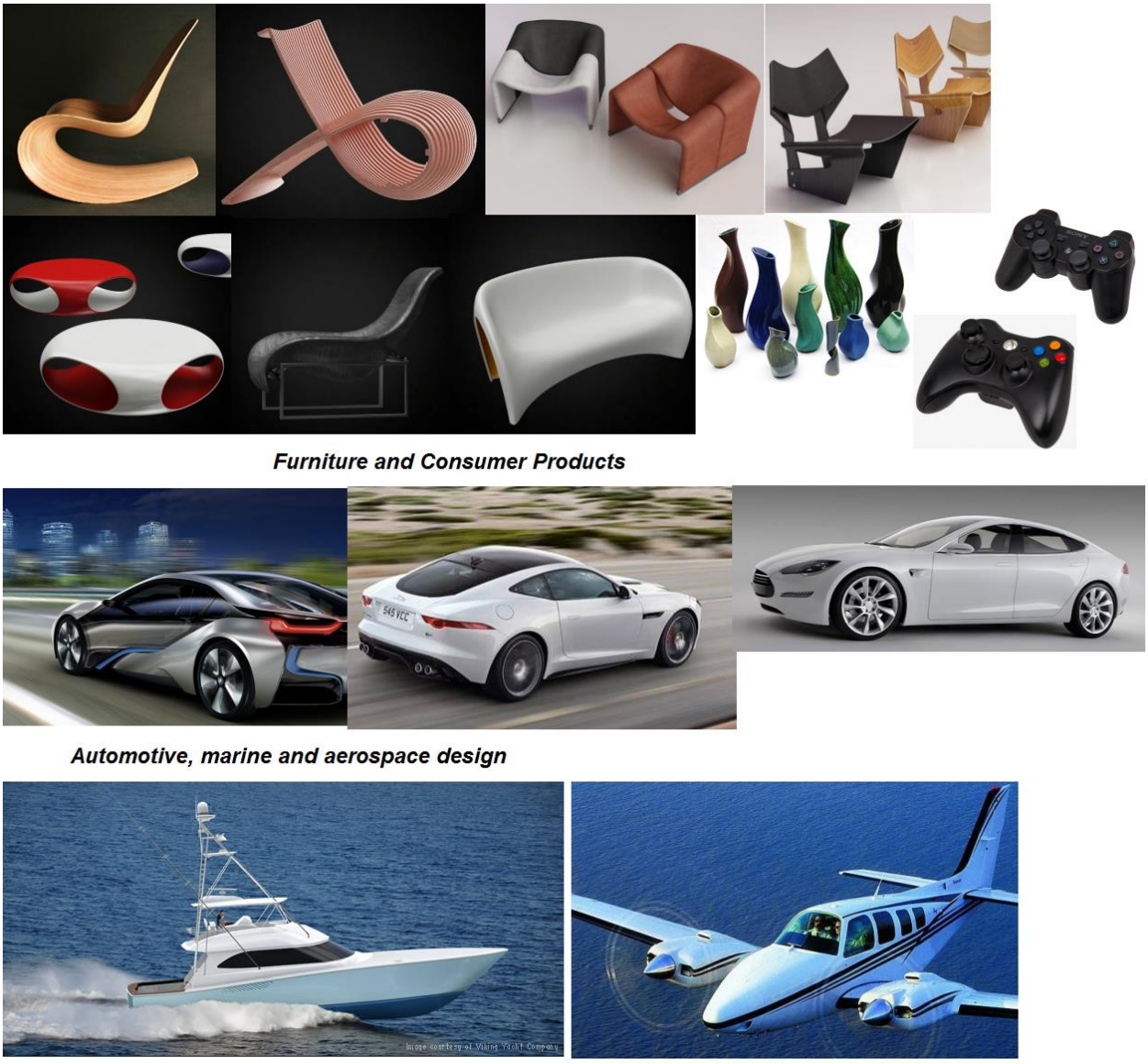

Figure 1 – Examples of complex and organic shapes

This new modeling method introduced in Inventor 2015 software uses the T-Splines technology acquired by Autodesk in 2011. First incorporated in Autodesk® Fusion 360™ with a powerful set of tools and now in the latest Inventor release, the new alternate modeling approach perfectly blends with the conventional parametric modeling method. In comparison to Fusion 360, Inventor has fewer freeform tools. However, more commands and workflows, to create and edit complex shapes, are expected to be added to this Inventor environment in the near future.

The Freeform modeling is basically a direct manipulation method in which the user explores new shapes by pushing and pulling points, edges, faces as well as the entire body. The method allows the user to

make decisions right away since the model updates smoothly in real time. It is like working with digital clay.

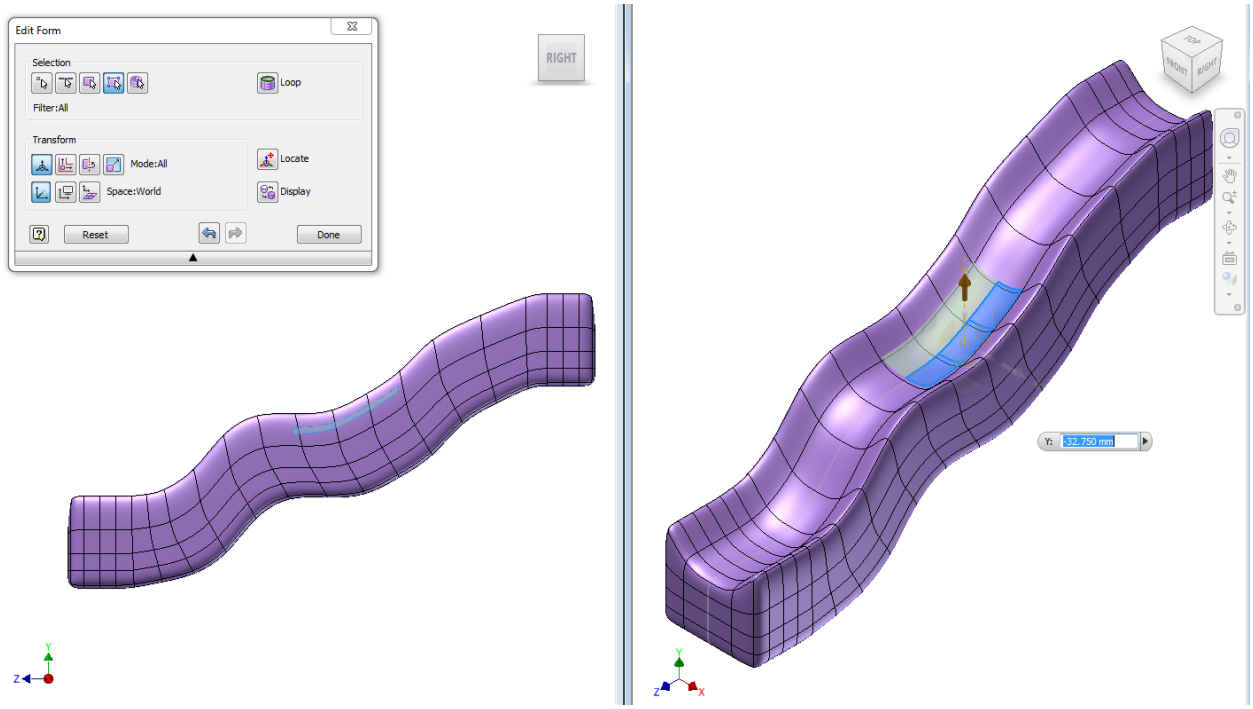

Figure 2 – Direct Manipulation

Freeform modeling uses T-Splines surfaces which are actually considered enhanced NURBS surfaces. T-Splines technology generates surfaces with fewer points in comparison to NURBS surfaces, since edges with control points are allowed to terminate without traversing the entire surface. Additionally, users can refine surfaces over specific areas by adding more control points when required. This local adjustments can be done by inserting more edges or subdividing the existing faces (see the image below). By requiring fewer control points (50-70% less), T-Splines surfaces allow users to quickly create and easily manipulate complex shaped models.

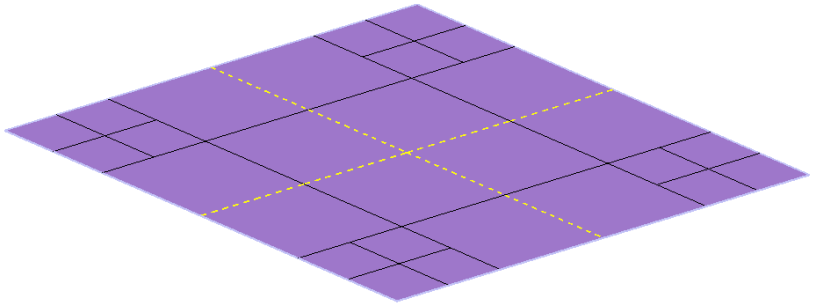

Figure 3 – Subdivided freeform plane with refined faces on the corners

Workflow

The workflow consists of two main steps:

a. Create the main body

Start with a subdivided freeform primitive and create a watertight volume (optionally assign symmetry). Select the primitive option that best represents the desired body.

Note: Do not assign symmetry on the freeform body if you will be using the Match Edge command right after the creation of the main body. This process automatically deletes the symmetry. However, once you have used the Match Edge tool, you can manually assign it if necessary.

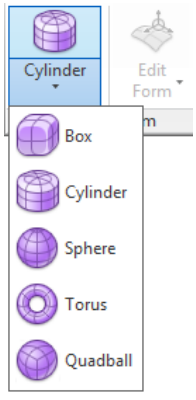

Figure 4 – Tools to create subdivided freeform primitives

b. Use the freeform edit tools

Once you have created the main body using one of the primitive shapes, then you can start using the eight different freeform edit tools in order to achieve a complex visually appealing design.

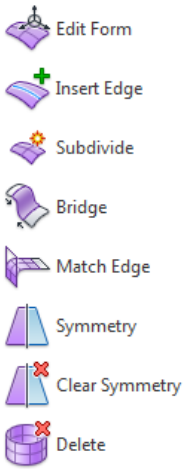

Figure 5 – Tools to edit subdivided freeform primitives

Inventor 2015 only allows you to start with a closed shape (watertight volume) and exit the freeform mode with a solid body (watertight volume) as well.

There might be a future release with a more flexible workflow like the following:

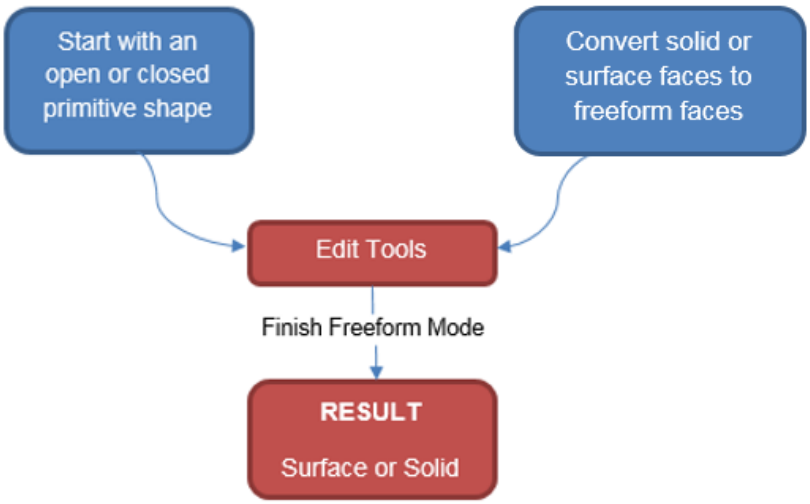

Figure 6 – Possible workflow in a future release of Inventor software

Tip: As mentioned above, there are eight edit tools and the marking menu has eight spots available. To speed up the edit process, I recommend creating a user-defined marking menu (Ctrl+right clik menu) where you can have all the freeform edit tools. Save time and forget the drop-down menu on the Ribbon.

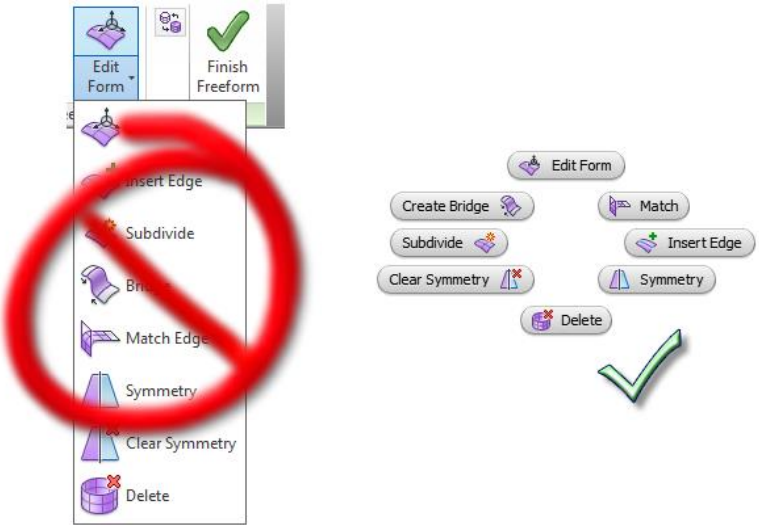

Figure 7 – User-defined marking menu for freeform edit tools

To create the user-defined marking menu, follow the next instructions: On the Ribbon > Tools Tab > Customize button > Marking Menu Tab > Environment Freeform > on the Sub Environment select Add Ctrl + Right Click Menu > select the location on the menu where you want to add the command > search the desired command > click on the command from the list.

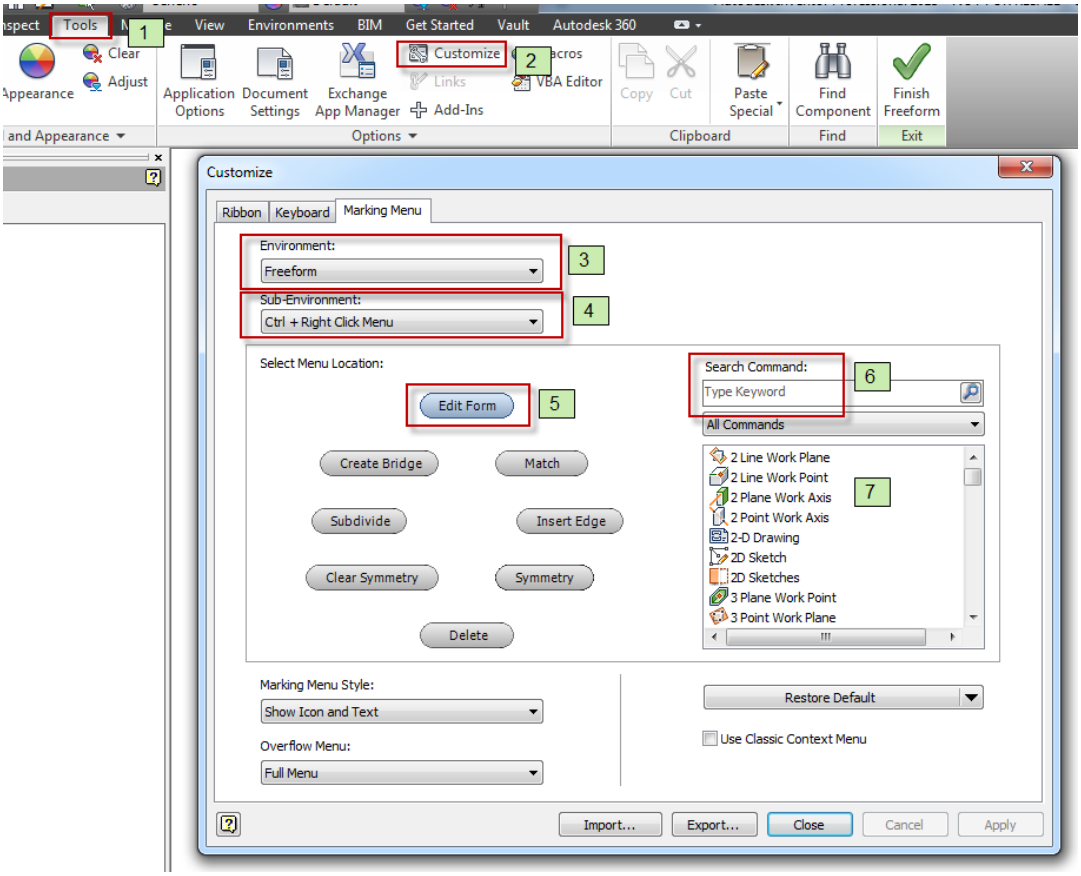

Figure 8 – Customizing the user-defined marking menu

Editing the freeform shape

As previously mentioned, creation of the main body is the first step of the workflow. One of the best and quickest ways to start defining the main body shape is by combining parametric geometry with freeform tools. For instance, you can create sketch geometry that defines sections/profiles and/or rails; then, create a freeform primitive; finally, use the Match Edge command to align the freeform body edges with the sketch geometry (see the image below). You can also use Match Edge to align freeform edges with edges from surfaces or solid bodies which is a pretty handy option when dealing with open freeform shapes (perhaps, it will be possible in the coming release). This procedure described in the previous example is quite similar to what Loft and Sweep commands do. The difference is that you create a complex freeform body that can be modified using direct manipulation at a later time without any restriction.

Note: You can only use Match Edge command between two open loops or two closed loops. You will get an error when trying to align an open loop with a closed loop or the other way around.

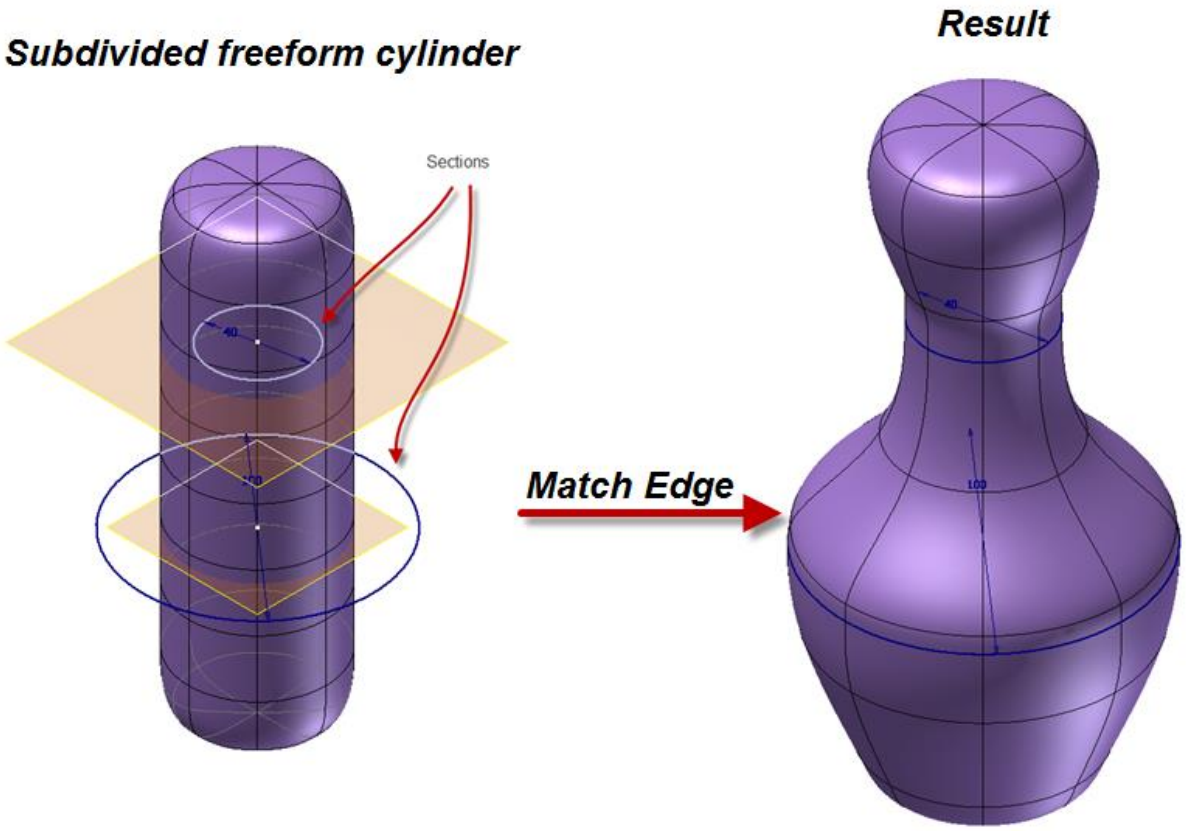

Figure 9 – Aligning freeform edges with sketch geometry

There are multiple ways to select a loop that will be used with some of the freeform edit tools (Insert Edge, Match Edge and Edit Form commands).

- Double click on an edge.
- Select an edge, then right click and select “Select Loop” on the marking menu (see Figure 10).
- When using Edit Form, select an edge and then press the Loop button on the Edit Form dialog box.

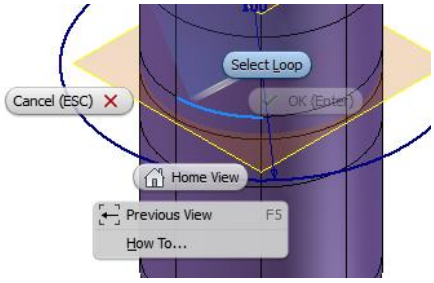

Figure 10 – Selecting a loop

Best practices for Match Edge

- Do not assign symmetry when creating a freeform primitive if Match Edge will be used right after that.
- If required, create symmetry manually after having used the Match Edge command.
- Only use Match Edge command between two open loops or two closed loops.
- Before using Match Edge command, try to place the freeform edges very close or on the same plane of the geometry that will be aligned with. As a result, the freeform body will get a more accurate shape.
- To follow the previous recommendation, you can move or insert edges and/or subdivide faces as needed.
- Use Match Edge first and then edit the shape where needed. Match Edge only aligns geometry but does not create an actual constraint. Therefore, you can drag the same edges you used for Match Edge to tweak or change the shape at any time in the design process.
- As Match Edge command does not create a real constraint, you need to reapply Match Edge on the same edges once you have finished your design. This should be done if you have manipulated the freeform edges assigned on the alignment. By doing so, you ensure the alignment between freeform edges and the selected geometry if required.
- Change the requested tolerance value based on what you need.

    - If you need to get a very accurate alignment, then use a very low value. However, the downside is that you will end up with a highly refined mesh. As a consequence, the direct manipulation and control will be more difficult and more time-consuming. In the two following examples, Match Edge command has been used with the default tolerance value (0.001mm). In both cases, the freeform primitive box got a very accurate shape but dramatically increased the number of faces and edges.

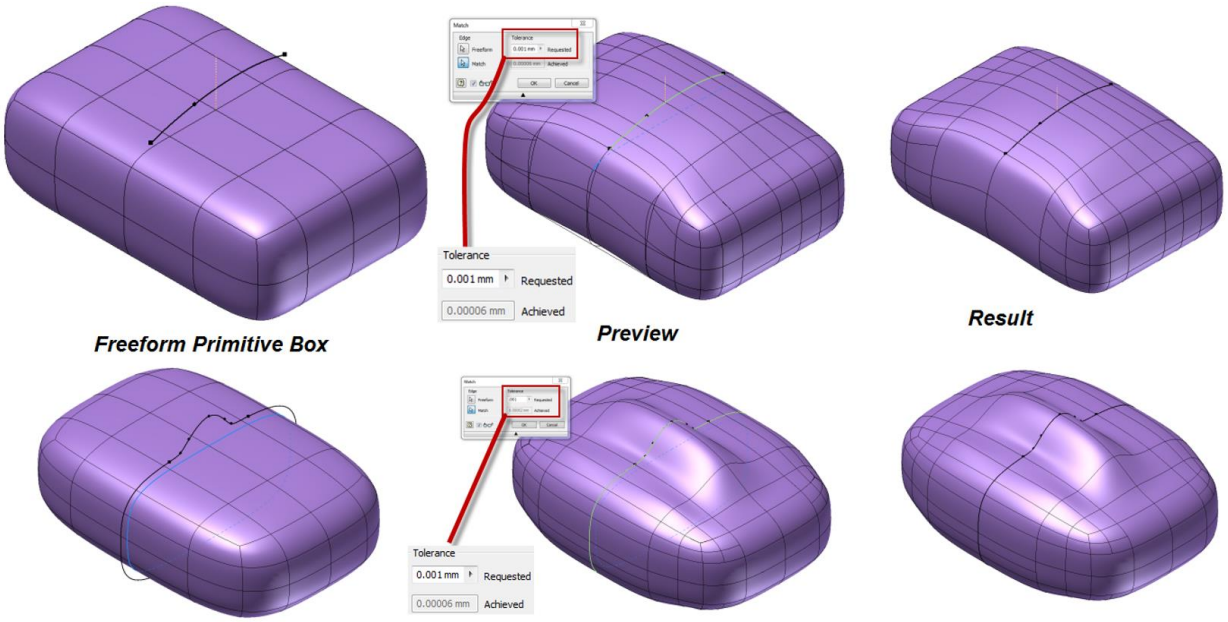

Figure 11 - Using Match Edge with the default tolerance value

- On the contrary, if your concern is not the accuracy and you only want to get a good approximation, then use a higher value. In the two following examples, Match Edge command has been used with 1 mm and 5 mm as requested tolerance values for the top and bottom examples respectively. In both cases, the freeform primitive box got a good approximated shape but the advantage is that it did not significantly increase the number of faces and edges. This is a more conservative result.

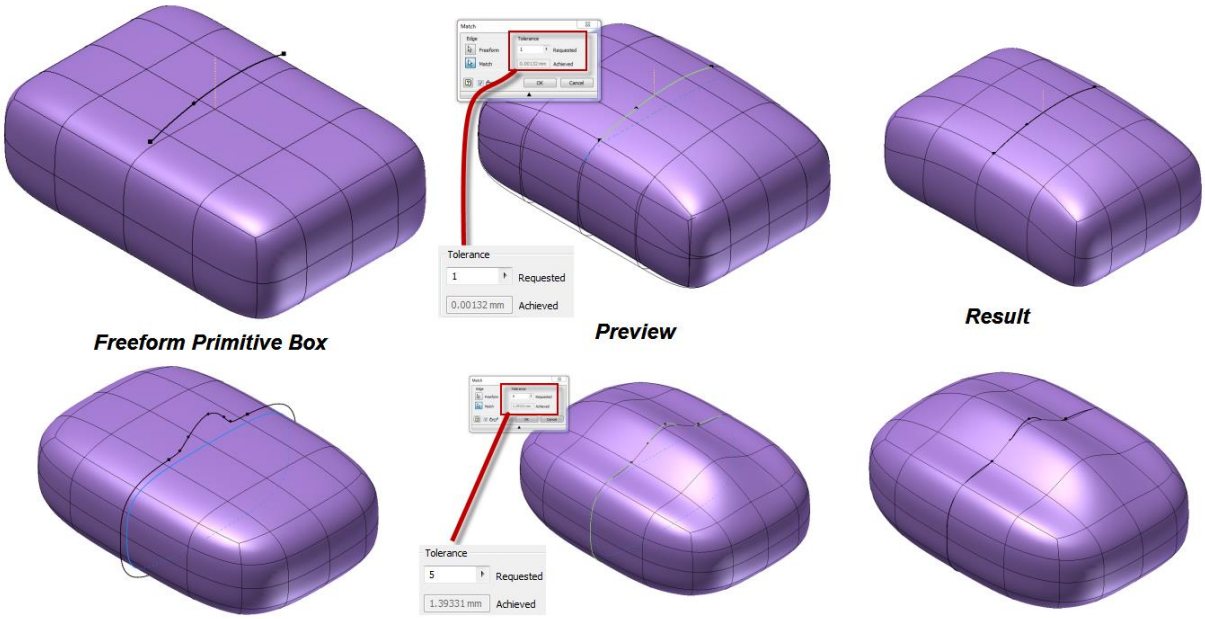

Figure 12 - Using Match Edge with higher requested tolerance values

Ways to flatten the surface ends

Once the freeform main body has been created, you proceed to edit the form using direct manipulation on points, edges and/or faces by pulling and pushing or by entering precise values.

The freeform body has an inherent continuity which is curvature (G2). For this reason, there are not flat faces in the model since the freeform body maintains smooth transition between faces. Freeform faces have the same speed (curvature) where they merge.

Sometimes the design requires flat or almost flat faces with rounded or sharp edges. The following will describe some techniques for those cases.

a. Flattening the surface ends by dragging edges

Taking the image below as a reference, you can see how the bottom faces become flat when dragging down the closed loop nearest to the end of the body. This procedure can be used on any surface end.

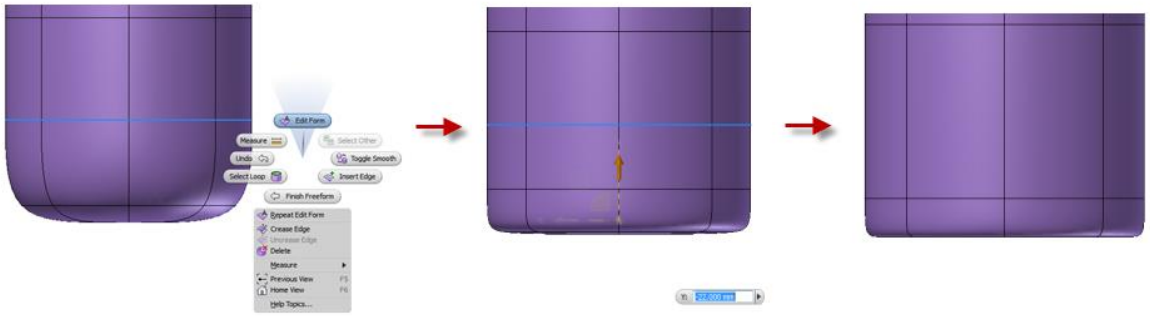

Figure 13 – Flattening the surface ends by dragging edges

b. Translate and extrude the surface end using value zero

Edit Form > Move/Translate (Extrude On) → Only Select Faces → Drag and return to value zero

To turn the Extrude option on, you can right click and select Extrude when the Edit Form command has been activated.

Tip: If you want to activate the Extrude option on the fly, all you need to do is hold down "Alt" key while dragging. This is better since you do not need to activate or deactivate it manually.

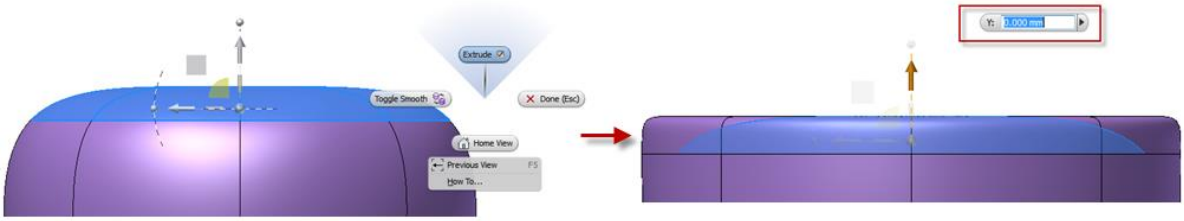

Figure 14 – Flattening the surface end by translating and extruding with value zero

c. Scale and extrude the surface end using scale factor one

Edit Form > Scale (Extrude On) → Only Select Faces → Drag and return to scale factor one

To turn the Extrude option on, you can use the same options mentioned before.

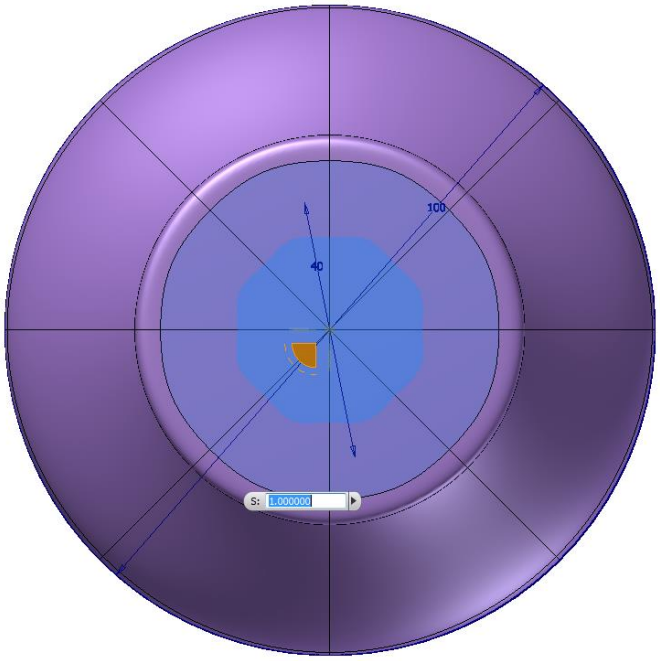

Figure 15 – Flattening the surface end by scaling and extruding with scale factor one

d. Crease Edges

Select edges > Right click menu > select the Crease Edge option

This option creates a flat surface end with sharp edges whereas all the previous alternatives keep rounded edges.

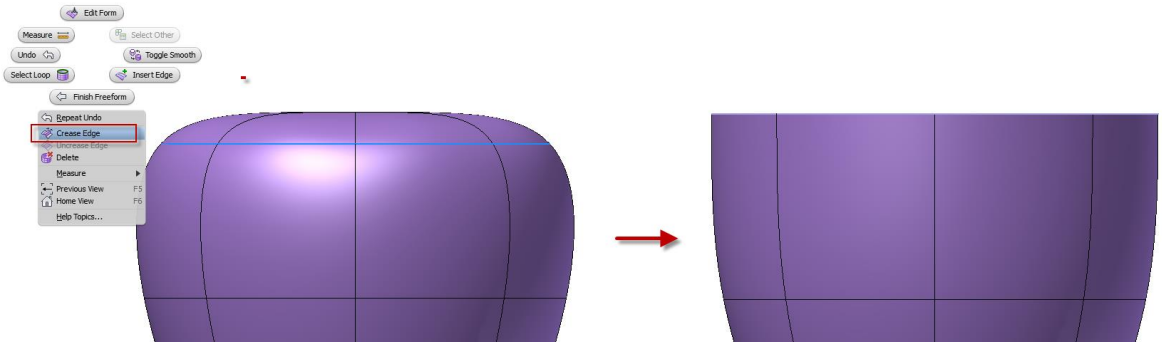

Figure 16 – Crease Edges

Working with multiple windows

When editing the freeform shape, sometimes it is better and easier to manipulate the body using multiple windows. In doing so, you do not need to constantly rotate the model using the orthographic views from the ViewCube. Furthermore, this is a great option since you can see how the model updates in real-time from different views.

To create multiple windows: On the Ribbon > View Tab > Windows Panel > New

You can create as many windows as you need. For instance, if you create three new windows, then you will have four windows in total where you can set up the three main orthographic views (top, front and right) and an isometric view (see image below). To set up these windows, you can do it with the “Tile All” button (On the Ribbon > View Tab > Windows Panel > Tile All)

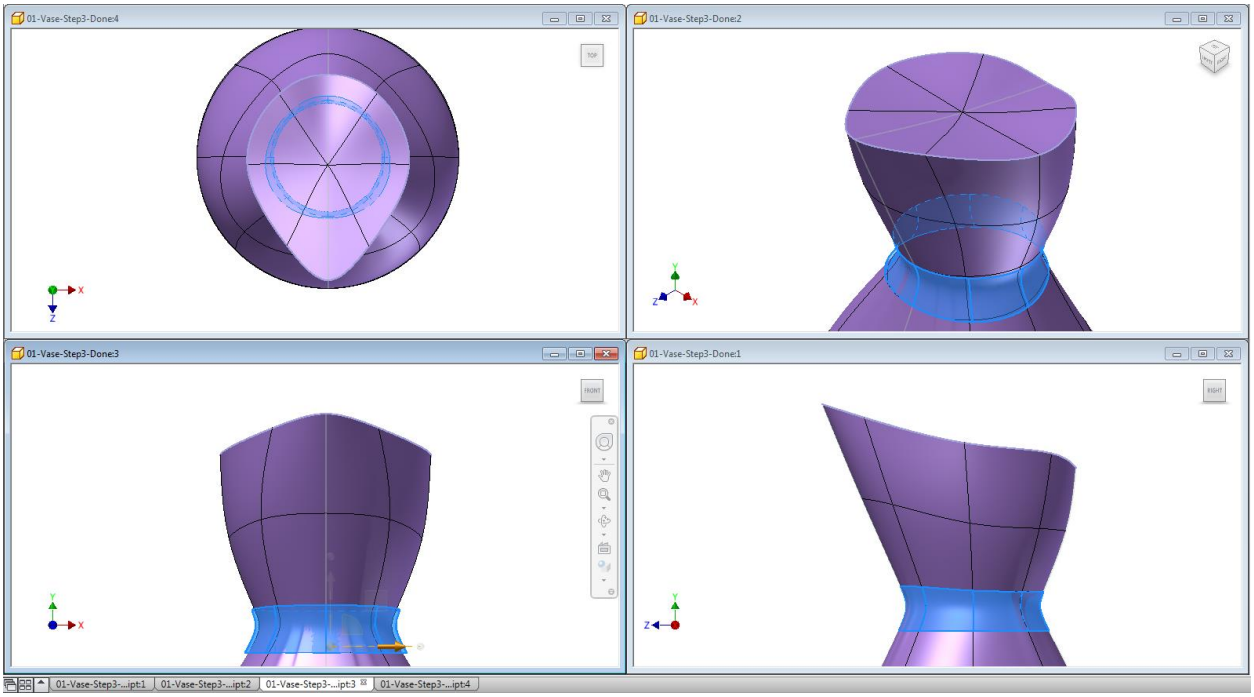

Figure 17 – Working with multiple windows

Nevertheless, often two windows are sufficient. One view where you can have an orthographic view and the other one to select an isometric view (see image below). If you work with two windows, you can quickly set the arrangement up with Tile Vertically or Tile Horizontally (On the Ribbon > View Tab > Windows Panel > Tile Horizontally / Tile Vertically).

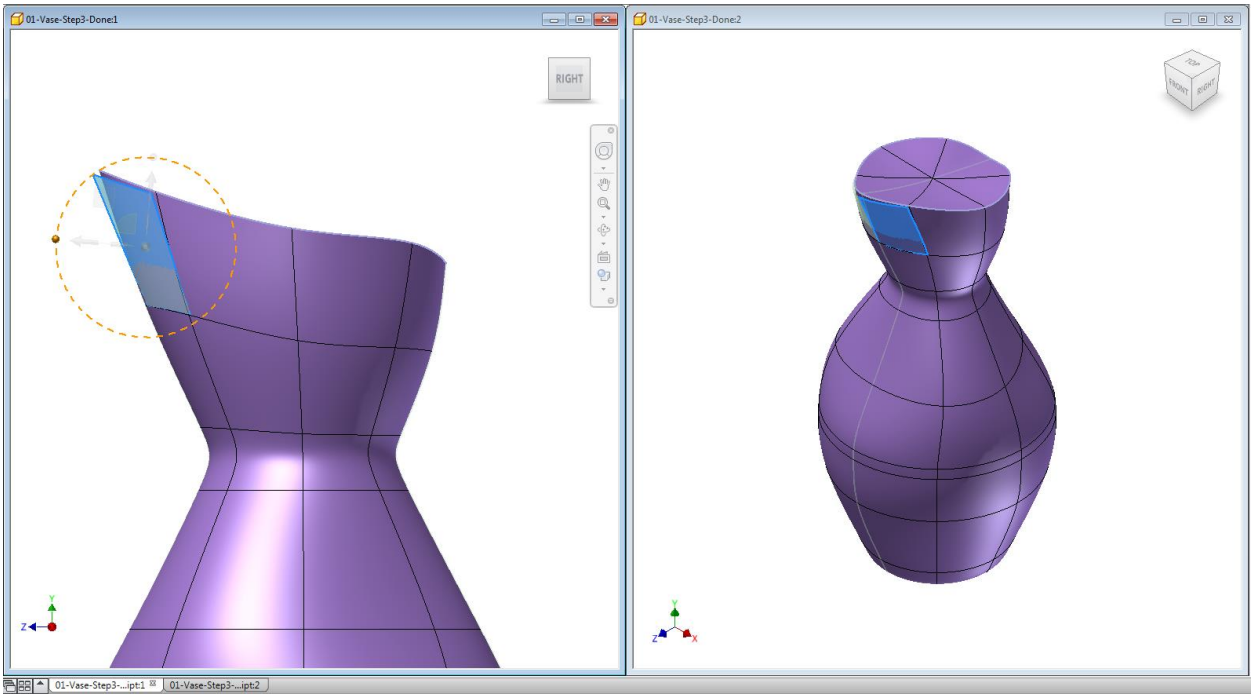

Figure 18 – Working with two views and arranged with Tile Vertically

Differences between surfacing and free-form tools

Next, freeform and surface modeling workflows will be compared against each other on different points.

Level of difficulty

Freeform modelling approach is a very intuitive method and presents a shallow learning curve being suitable for beginners. On the contrary, the conventional surface modeling is considered an intermediate or advanced technique due to the complex work required and the skillset. To work with this traditional technique, users need to be familiar with sketch constraints, 2D and 3D splines, work features (planes, axes and points), and more.

Amount of time to acquire skills

As previously mentioned, freeform modeling has a shallow learning curve allowing users to get the knowledge and skills in a shorter time in comparison to surface parametric modeling. Surface modeling has a difficult initial learning process what involves longer time to be proficient with this method.

Modeling time and amount of work

A design concept can be easily and quickly created with freeform tools. Freeform modeling is much faster and more flexible than surfacing modeling. This gives you the advantage of exploring more design concepts and generate new ideas in a very short time. Surface modeling requires curves, work features, parameters and constraints which implies that you will spend more time on modeling.

Precision

Although freeform tools allow users to enter values when creating and editing the form, this method is not as precise as the surface modeling. The freeform tools that have been incorporated into Inventor 2015 do not give enough options to get a very exact shape. Therefore, this new modeling approach has been thought as an alternate method to create rough shapes in which size and shape are not yet established. It is a perfect technique for the ideation and conceptual design stage to present multiple design variations in less time. Thus, the design process is being accelerated and once the final design has been selected, the design and engineering teams can proceed with detailed design using a parametric workflow or by combining freeform modeling with conventional surfacing tools.

One of the greatest advantages of using the surfacing modeling is the precision you can achieve in your model. 2D and 3D sketches are the foundation of basic and advanced surfacing tools. Next you will find different examples that demonstrate the precision you can get when working with surface modeling:

Conditions and dimensions

Dimensions and geometric constraints allow you to be very specific on 2D and 3D sketches to determine exact shape, size and orientation. The dimensional and geometric constraints are maintained throughout the entire design process. You can easily modify any dimension by using the parameters dialog box or by finding the parameter on the feature or sketch geometry. This is not possible when using freeform modeling since values are not saved.

Control and Continuity

2D and 3D curves in Inventor are created by using either interpolation splines or CV splines. The transition and the smoothness between curves are controlled by using constraints. Next you will see the types of continuity that you can apply to two curves

- Position (G0): two endpoints are joined with coincident constraint. They only touch at the join.
- Tangent (G1): curves are tangent and their endpoints are coincident.
- Curvature (G2): smooth constraint is applied to the curves and their endpoints are coincident. These two curves present the same speed at the join.
- Acceleration (G3): also known as curvature with constant rate of change, these two curves require the same constraints as G2 (coincident and smooth) plus the manipulation of spline points or vertices.

All these continuity conditions cannot be applied when using the freeform tools that Inventor has so far. Only G2 and G0 are possible. G2 is inherent when creating and editing the freeform shape. G0 is applied when using the “Crease Edge” option over edges.

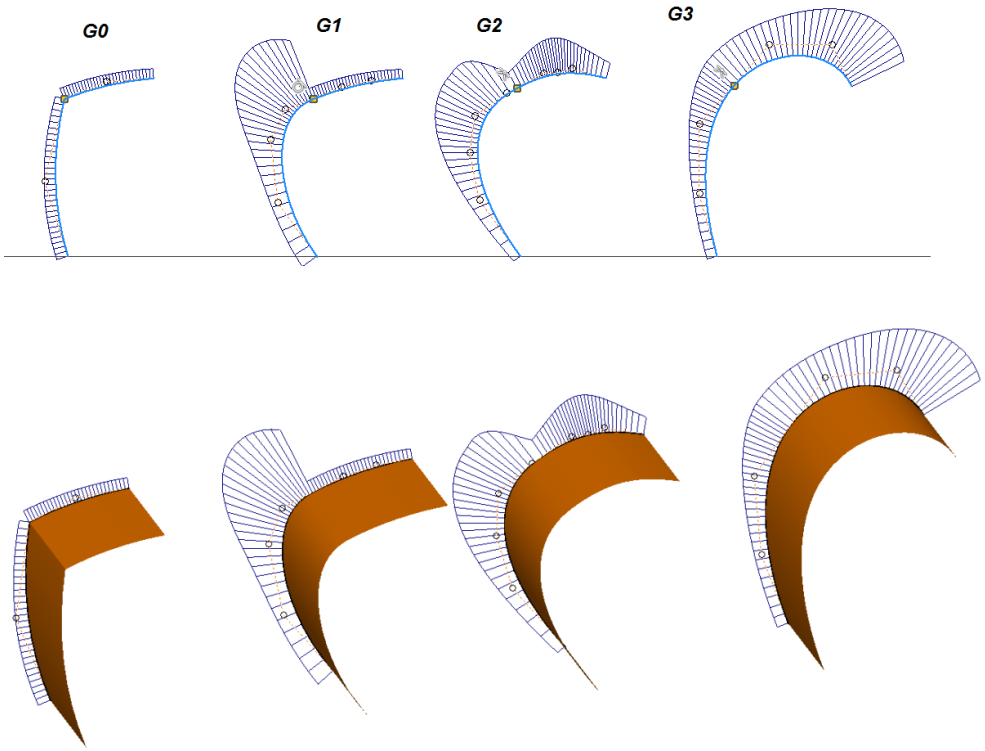

Figure 19 - Curvature and smoothness

Creation of curves by using imported points

2D and 3D interpolation splines can be created from imported points. An interpolation spline is a curve created through points which can be specifically placed at precise locations using Cartesian coordinates specified in an Excel spreadsheet.

Equation Curves

There are some forms that require great precision based on mathematical equations like airfoils (see image 21).

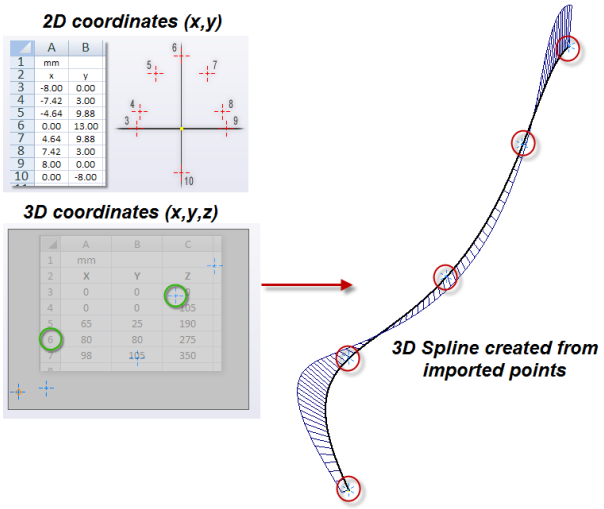

Figure 20 – Curves created with imported points

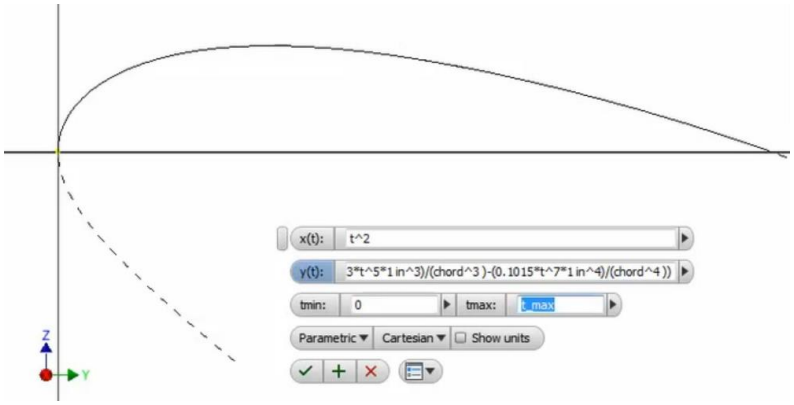

Figure 21 – Equation Curve

Transitions

Creating rigid and smooth transitions from sections with different type of geometry (e.g. a transition from square/rectangle to ellipse/circle) is quicker and easier with conventional surfacing modeling.

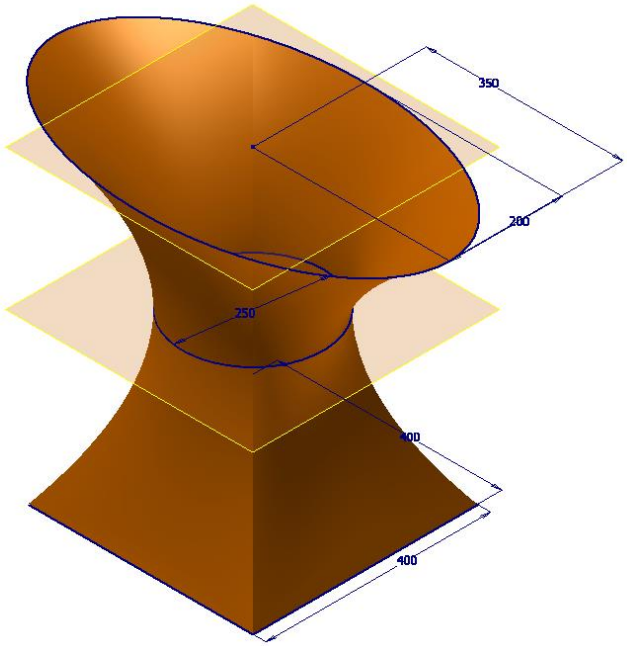

Figure 22 – Transitional shape between three different geometry sections

Relationships and feature history

The freeform modeling workflow only creates a feature per freeform body in the browser. The freeform subdivided primitive and all the changes made with edit tools are represented under one single feature.

This feature can be activated to modify the freeform body at any stage in the design. However all the freeform edits are not saved in the browser. Then, if you want to go back to a previous freeform shape, you can use the “undo” tool as long as you keep the model open. If you save the part, close it and reopen it, the undo tool does not work anymore. Then, you need to manipulate the model with freeform edit tools to retrieve a previous desired shape. On the other hand, the advantage of having a model without the whole edit history is that you can tweak it or modify it without getting errors or prompts due to the parent-child relationship. Furthermore, if you desire to make changes to a model a colleague has worked on, you will have the complete freedom to do so.

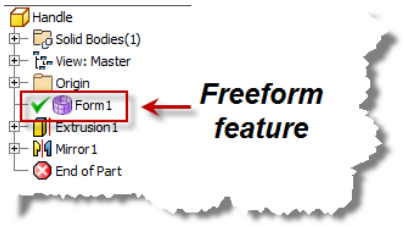

Figure 23 – Freeform feature

Surfacing modeling keeps the feature history and the parent-child relationships. For a simple change, all you need to do is find either a feature or a sketch and make the change. However, for some complex changes or to delete features, you might become overwhelmed with numerous dialog boxes, prompts and errors. Hence, you will end up spending a lot of time understanding the consequences of the changes and rebuilding features. When you delete parent features, child features need to be edited or recreated.

Design Intent

Design Intent in the CAD world is defined as the way you create your model and the way it behaves when making changes. In a collaboration environment, CAD users can model the same digital product using different features and techniques. However, when you modify these digital models, they might behave completely different even though they represent the same product. As a consequence, when working with surfacing modeling, sometimes you might get unexpected changes when you edit a model created by one of your colleagues since everyone uses different design intent. Therefore, some design and engineering teams try to implement modeling standards to get better results (e.g. positioning holes from the work features like center point, work planes and axes instead of dimensioning the hole from external or boundary edges).

In the freeform modeling, you always work with the final result since Inventor does not capture the design intent (parameters, features and relations). Hence, making changes in a design concept using freeform modeling is always quite simple regardless of the designer of the model.

Using references

You can establish relations between two components to easily create and modify a new part based on an existing model. The relation is basically created when you get information like sketches, parameters, surfaces and more. These objects and information are used as references to speed up the design process.

Users can take advantage of different tools like Derive and Copy Object to get surfaces as references. Once you have used Derive or Copy Object to get surfaces based on existing models, you can use the rest of the surfacing tools to get the desired shape. In Inventor 2015, surface modeling can use surfaces as references whereas freeform modeling cannot.

The image below illustrates how the temple covers update the shape when making changes on the glasses.

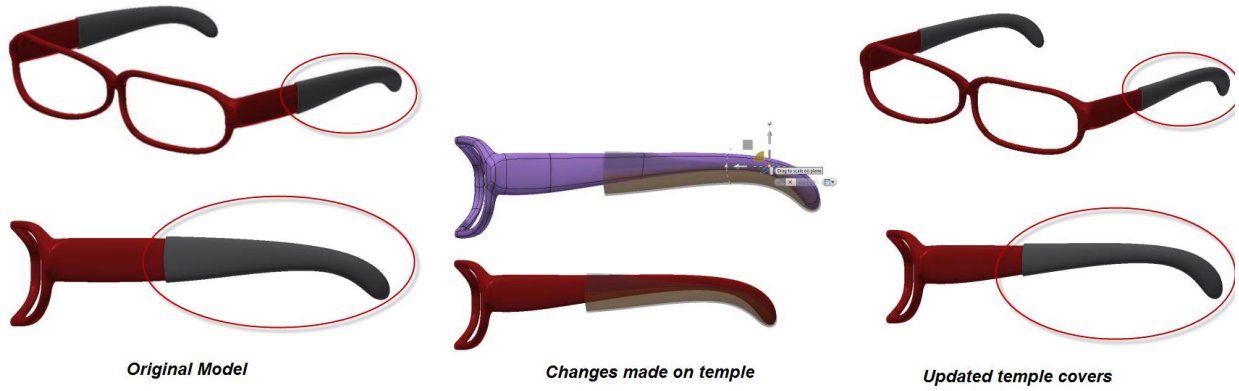

Figure 24 – Temple covers automatically adjust the shape when changing the glasses

The advantage of creating this relationship is that when you make changes on the base component, the new part that uses this one as a reference automatically adjusts to these design changes.

Repairing geometry

You can only use surface modeling to repair imported geometry. Gaps, overlapping faces and other types of geometry error can only be fixed using surfacing tools.

Most appropriate modeling method for your designs

The main differences between these two methods are: precision and modeling time. Therefore, in the ideation and conceptual design phase, you should use the freeform alone or combined with parametric modeling if some exact dimensions are required. Using the new alternate approach, you can quickly provide multiple design variations and speed up the design process.

If you will be designing a product in which the requirements and design criteria are already known, you should use the conventional surface modeling. With this method, you can be very precise when defining the shape and size. Additionally, this is the suitable approach if you need very precise curves to create surfaces where the shape is driven by equations or point with specific coordinates.

Finally, the conventional surfacing approach allows you to carefully analyze the curvature and be very accurate with the type of continuity (G0, G1, G2 or G3) you want to apply.

Combine surfacing and free-form tools in a design

For now, Autodesk Inventor only allows a simple workflow to combine surfacing and free-form tools in a product design. You can create a visually compelling design using freeform tools in which the shape will not be very precise, suitable for a design concept. Then, by using 2D and 3D sketches you can create very precise profiles/sections and rails. Match Edge can be used to enhance the freeform shape by aligning the freeform edges with sketch geometry or edges from solids or surfaces. Finally, you can use conventional surfacing techniques to add or cut material from the freeform body. For instance, by using basic commands such as extrude, revolve or sweep, you can create a surface that will interact with the freeform model, then by using the surfacing tools (trim, extend, patch, stitch and/or sculpt), you can add or remove material (see image below).

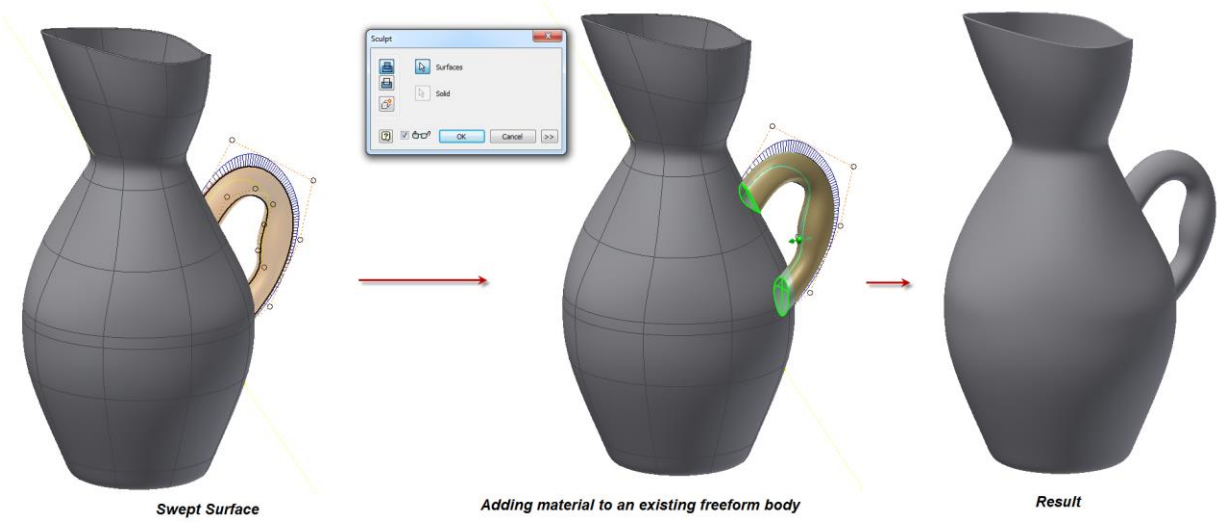

Figure 25 – Combining a freeform body along with conventional surfaces

By combining these two techniques you can create rough or very precise shapes when needed. In addition, you combine the freedom of freeform modeling with the constraints of surfacing parametric modeling.

Resources

- Autodesk Inventor 2015 Help
  http://help.autodesk.com/view/INVNTOR/2015/ENU/
- What's New in Autodesk Inventor 2015 – PDF file
- What's New in Autodesk Inventor 2015
  http://help.autodesk.com/view/INVNTOR/2015/ENU/?guid=GUID-6A22AD6D-7794-4969-8F0C-31CE55F1C5A8
- Autodesk® Inventor® 2015 Advanced Part Modeling - Autodesk Official Training Guide
- Autodesk Feedback Community for Inventor - Inventor Shelby Feedback Community site
- T-Splines technology
  http://www.tsplines.com/
- Images of complex and organic products retrieved from
  3D Furniture - http://www.3dfurniture.net/

Appendix

This lecture will be also focused on planned future functionality for freeform modeling in Autodesk Inventor. Next you will see some of the models that will be created or modified in this class using some freeform tools that will be coming in the next Inventor release.

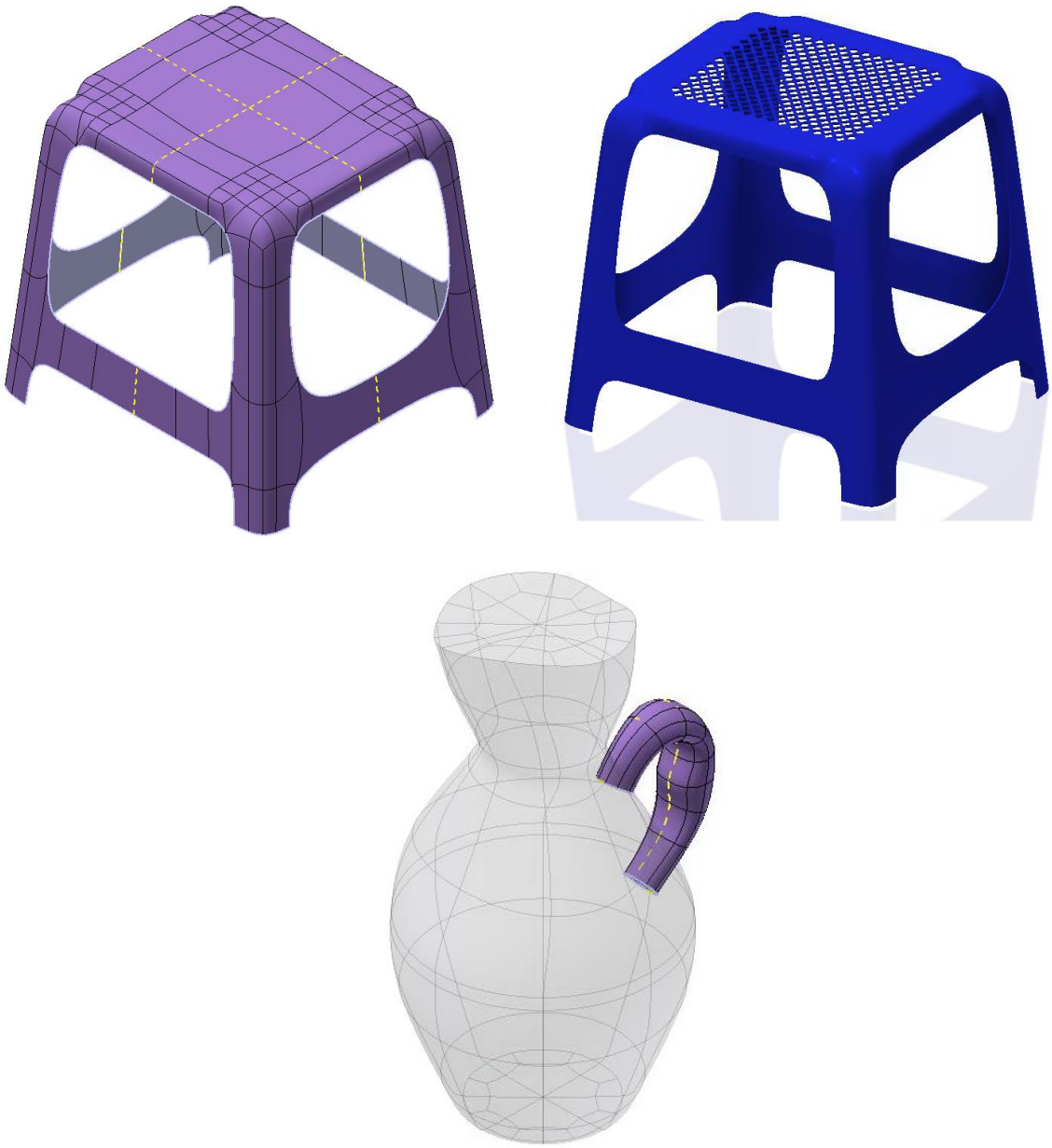

Figure 26 – 3D models created with the planned future functionality for freeform modeling

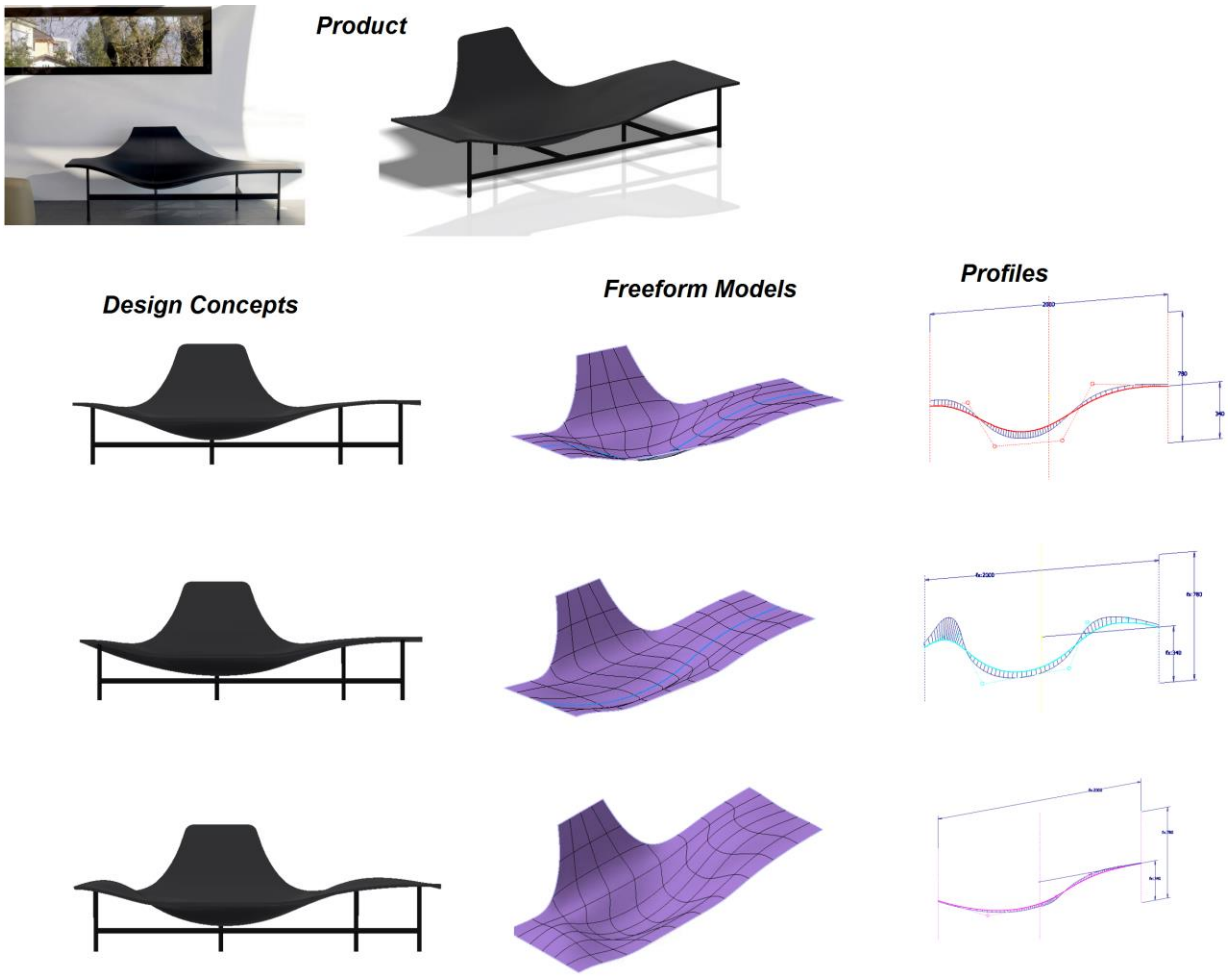

Figure 27 – 3D models created with the planned future functionality for freeform modeling

Figure 28 – 3D models created with the planned future functionality for freeform modeling