



Advanced Consumer Product Customization Techniques with Autodesk® Fusion 360™ and Autodesk® Inventor®

Matt Harris – Principal and Co-Founder



MA2491 Let your customers unleash their inner rock star. "Engineer to order" has been done, but what about "design to order"? Learn how to create flexible complex designs driven from customer inputs. Increase efficiencies and margins using the sculpting tools in the Autodesk Fusion 360 3D CAD design app and the automation capabilities of Autodesk Inventor software. We use our startup, Orphanage Guitars, as a case study to illustrate how our customers are able to design their own custom carved-top electric guitars to their specifications in a matter of minutes. We let you behind the curtain, show our process, and discuss the pros and cons of alternative mass customization methods.

Learning Objectives

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

- Harness the power and flexibility of the sculpting tools inside Fusion 360
- Apply best practices when transferring data from Autodesk Fusion 360 to Autodesk Inventor software
- List the pros and cons of different customization techniques with Autodesk Inventor
- Apply best practices for creating flexible organic models with Autodesk Fusion 360

About the Speaker

Matt is currently Principal and cofounder of Redpoint Studios in Manchester, NH. Redpoint is a mechanical and industrial design consultancy primarily serving the medical and consumer product industries.

Matt was formally an Autodesk certified instructor of both the mechanical (Inventor) and industrial design (Alias) suites of products while working as an applications engineer at M2 Technologies, a Gold Partner and ATC Autodesk reseller. Prior to M2, Matt held the position of Lead Industrial Designer at Deka Research and Development, a world renowned R&D firm founded by the inventor Dean Kamen.

Matt has experience designing and building marketable products around proprietary technologies. He specializes in product usability, ergonomics and aesthetics. He provided design expertise on projects in the fields of dialysis, diabetes care, surgical devices, diagnostics, water purification, power generation, action sports, lighting design, ground penetrating radar, robotics and guitars.

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Document Scope

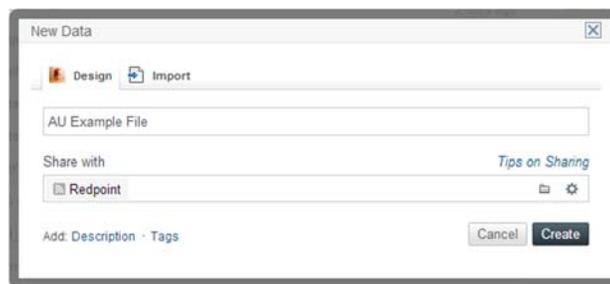
This handout is intended to capture the basic principles that will be presented in live demo format at AU 2013. This document may be used as a general guideline when creating a new configurable product originating in Fusion 360.

Learning Objective 1 - Harness the power and flexibility of the sculpting tools inside Fusion 360

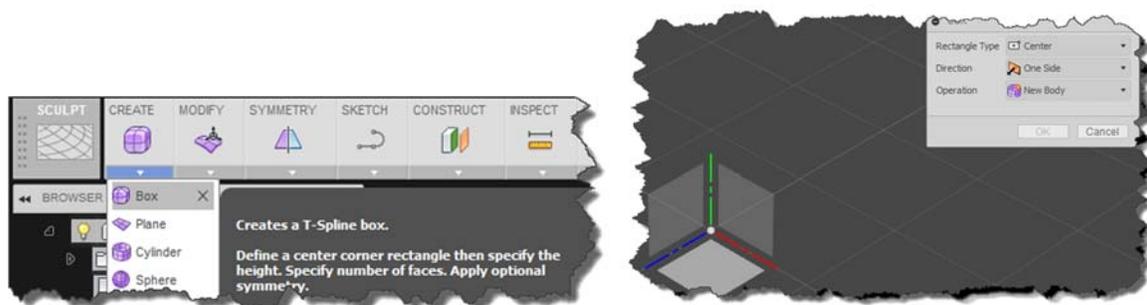
Create the basic overall form of your “product”

- Only focus on overall volume and avoid adding too much detail
- Save details such as parting lines, bosses, ribs, landings, fillets and chamfers for Inventor

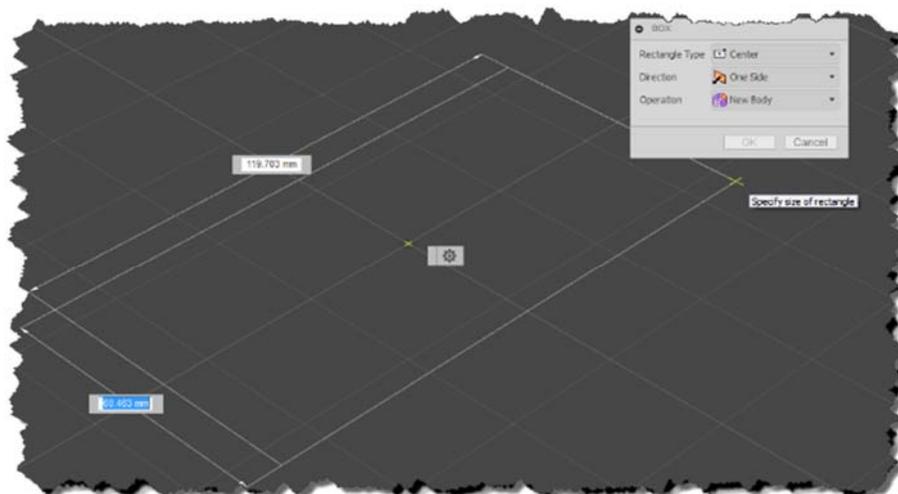
1. Open Fusion 360
2. When the dashboard appears create “New Data”, type a file name and click “Create”



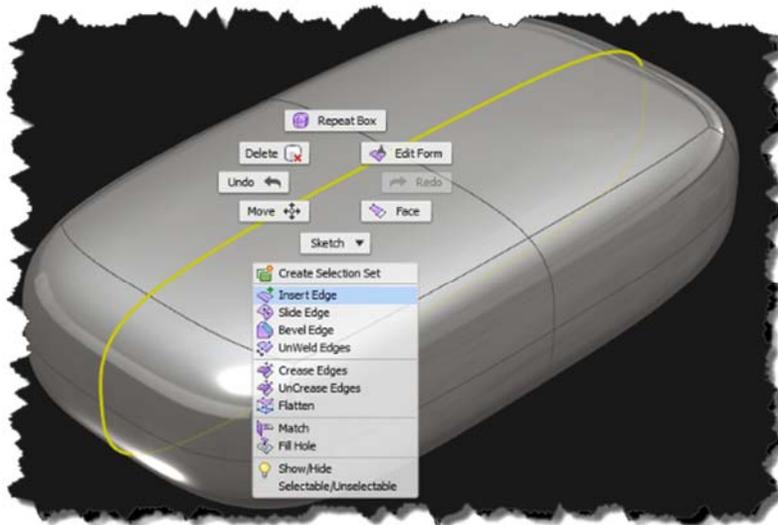
3. Create a T-Spline body by choosing “Box” under the “Create” drop down menu
4. Choose an origin plane from the graphics window on which you would like to sketch your 2D profile



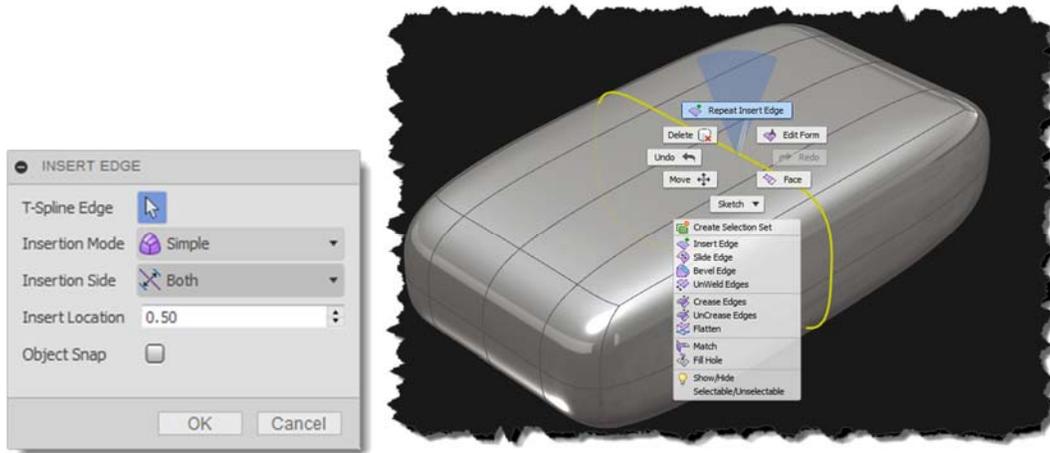
5. Sketch a rectangular profile by placing two points and add thickness by placing a third



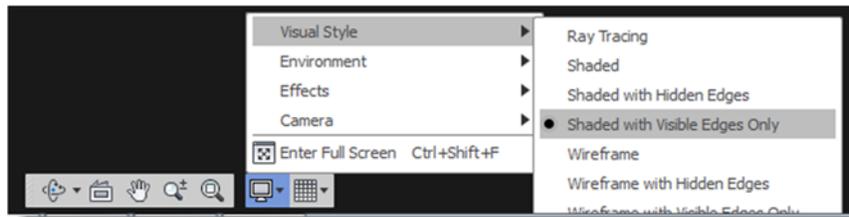
6. Once the T-Spline body is created double click the center line. Once it's highlighted right click to bring up the marking menu
7. Click "Insert Edge" to add additional points of control to the T-Spline body



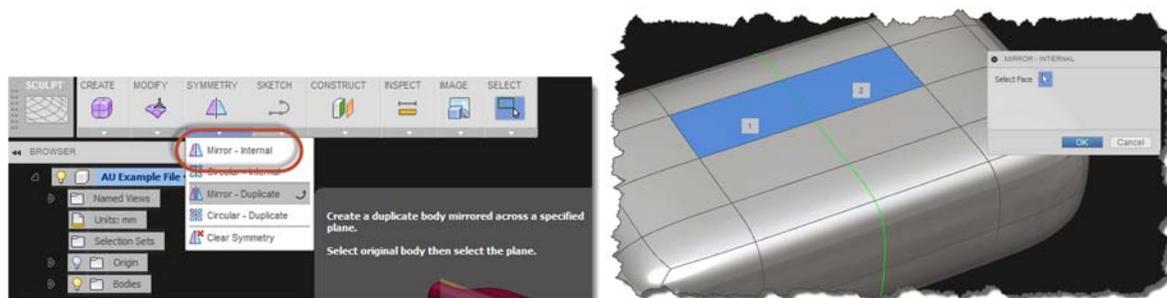
8. For "Insertion Side" change the selection to "Both" to place points of control to both sides of the chosen centerline
9. Repeat the process by selecting the centerline in the other direction and inserting an edge to both sides



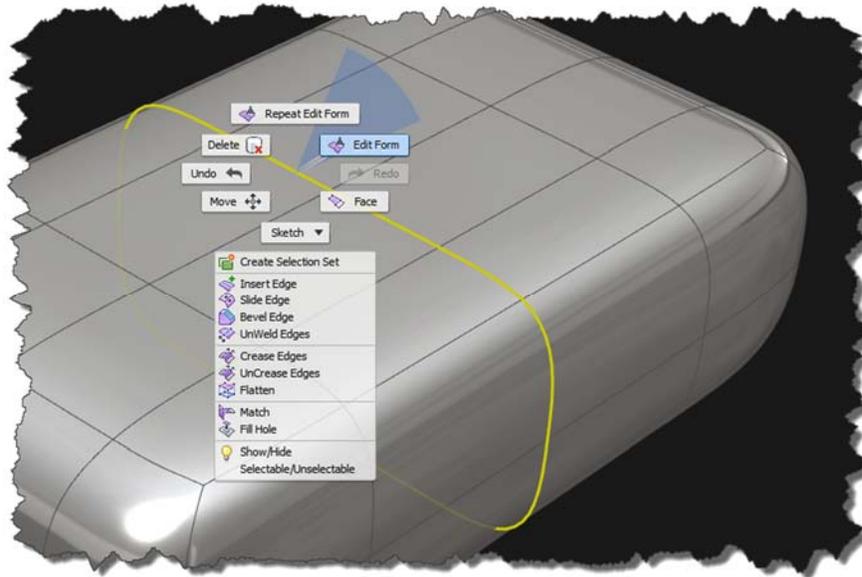
Note: At any time if the control lines are not appearing on the body you may turn them on by adjusting the “Visual Style” at the bottom of the graphics window



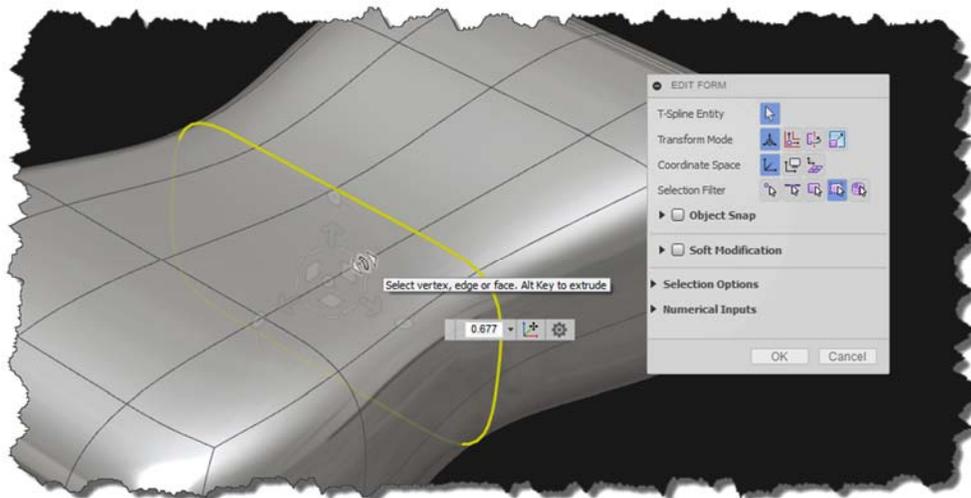
10. To add symmetry to the model choose “Mirror – Internal” from the symmetry drop down menu.
11. Choose one face on either side of the line you wish to become the symmetry plane. Now any edit to one side of the symmetry line will be recreated on the other in real time



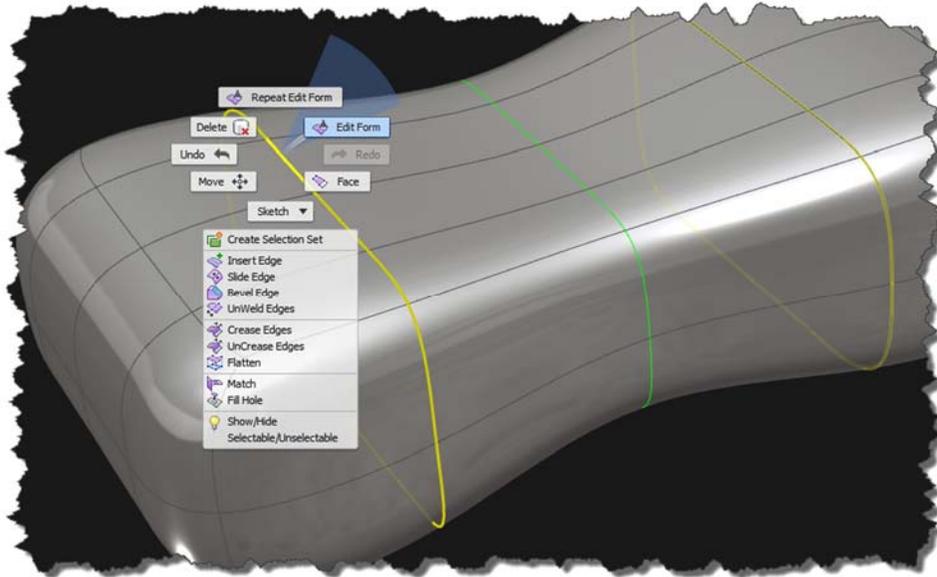
12. Next double click to highlight the entire center line and right click to bring up the marking menu. Choose “Edit Form” to bring up the transform glyph



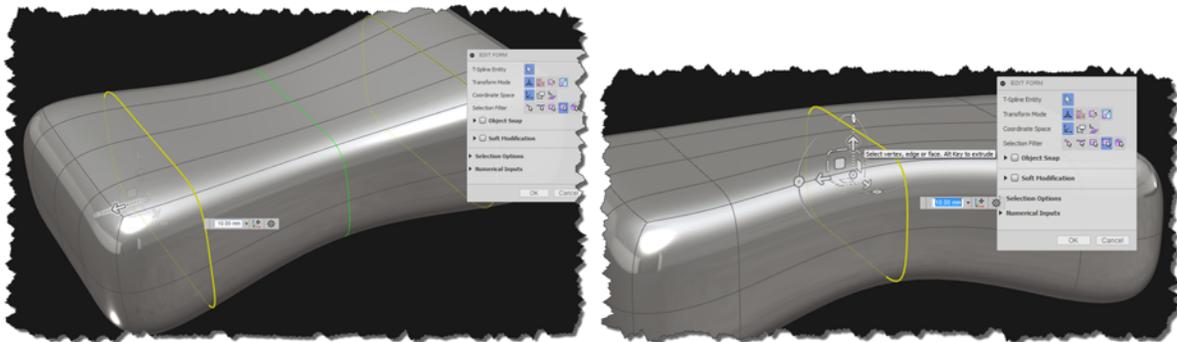
13. My clicking and dragging one of the corners of the glyph you are able to 2D proportionally scale the center line to modify the shape of the body



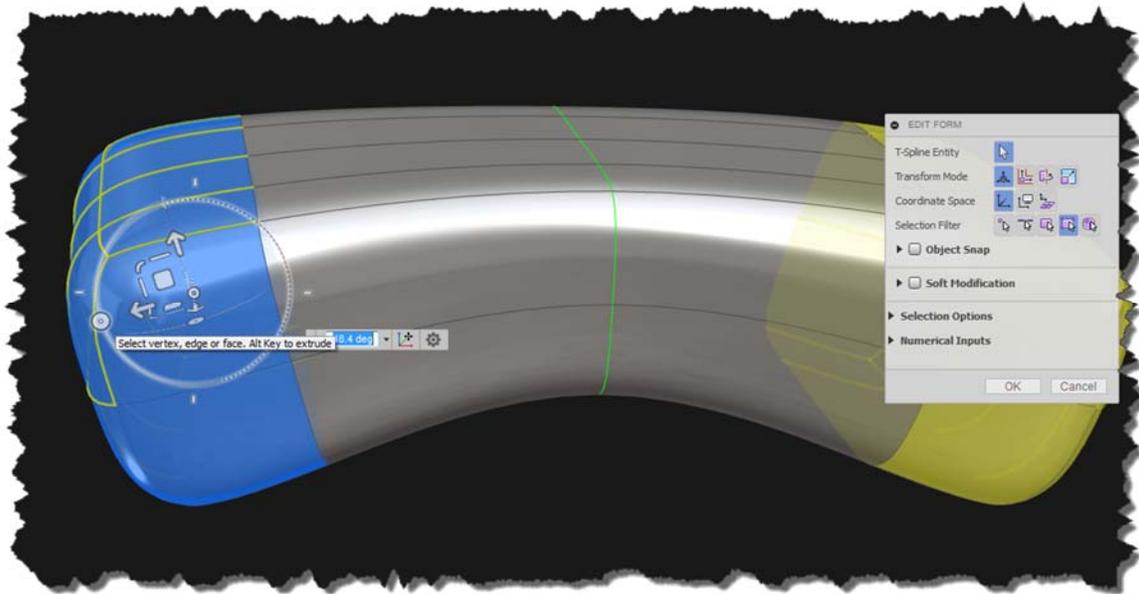
14. Double click one of the edges to either side of the symmetry line to select the entire edge. Right click and choose "Edit Form" from the marking menu



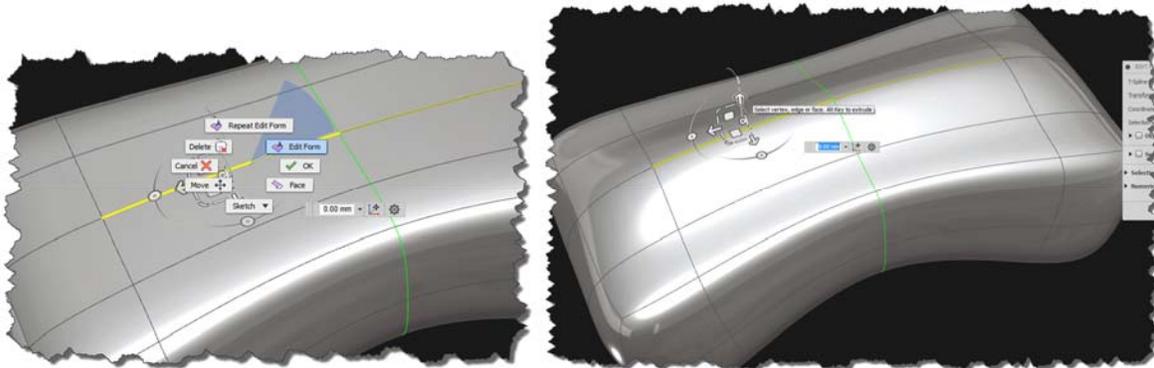
- 15. This time use click and drag one of the arrows to drag the selected edges orthographically. Notice how both sides of the form are modified at the same time.
- 16. Edit the form of the center line again and this time drag in one direction to add interest to the form



- 17. Select the faces of one of the ends of the T-Spline body. Both sides will highlight indicating symmetry is still active
- 18. Again “Edit Form” and this time pick and drag one of the axis of rotations of the glyph to rotate the position of the faces

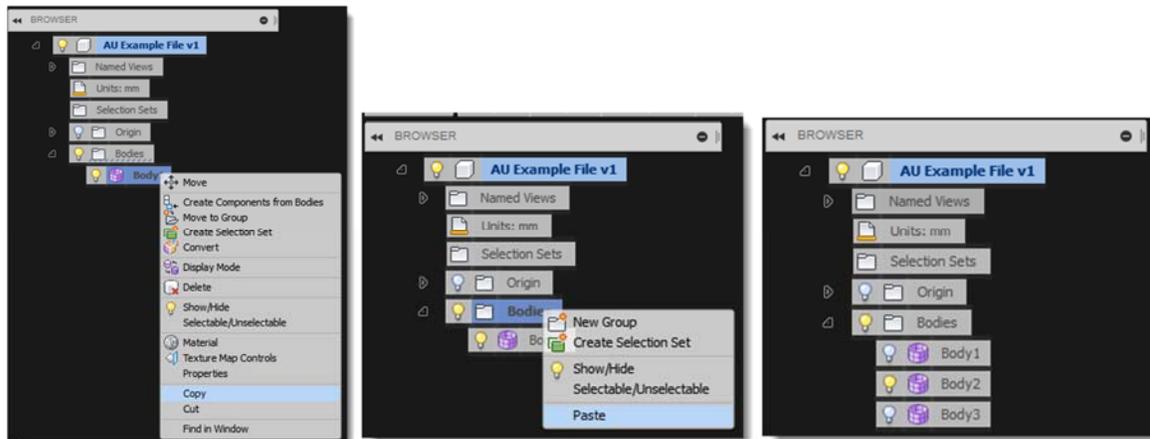


19. You may also select individual faces and edges to modify the form. Select a single edge and edit the form to see a more localized effect



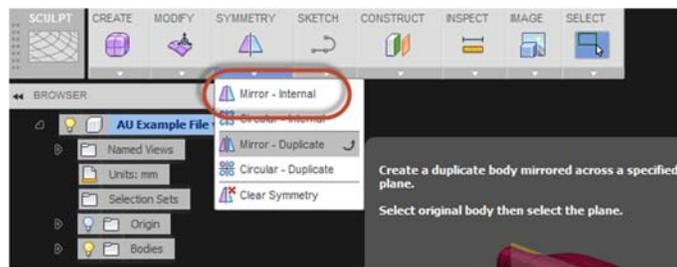
Create duplicate versions of your “product”

- Once your first form is completed it's time to duplicate your design to create more versions. Think of these versions as custom orders for your customers
 - We'll create two subtly different versions and one a bit more drastic
1. Right click the T-Spline body you created in the browser and choose “Copy”
 2. Right click on the “Bodies” folder and click “Paste”
 3. Paste one more time to create a third version of the product
 4. Toggle visibility by clicking the light bulb icon next to each T-Spline body

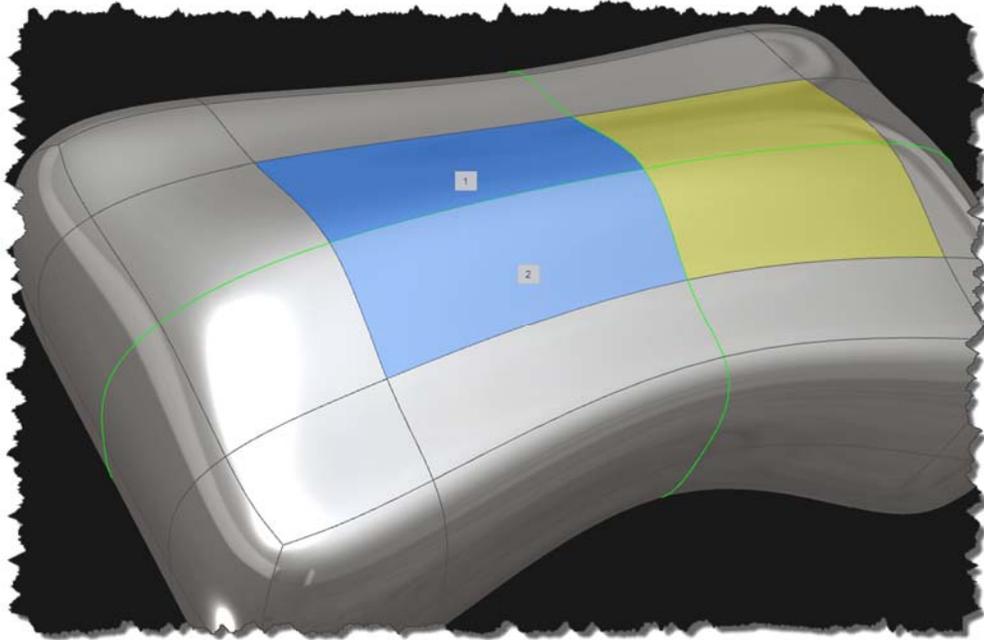


- For “Body2” add an additional symmetry plane
- Simplify the form by removing control points

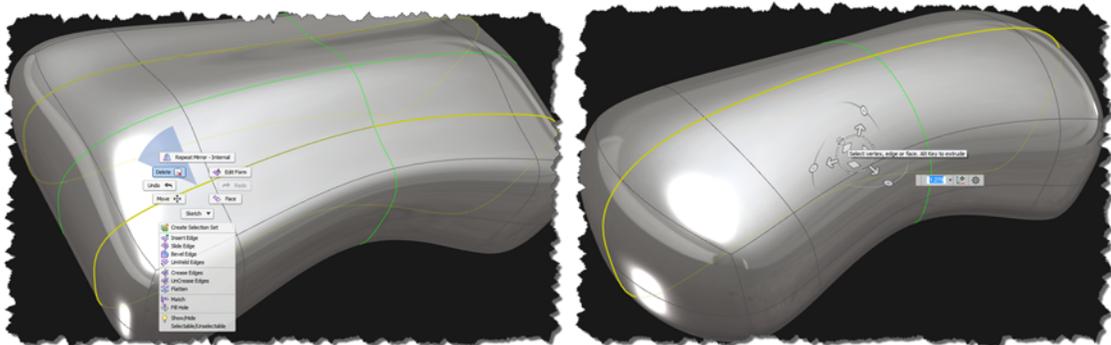
5. Click “Mirror – Internal” again from the “Symmetry” drop down menu



6. Select a face from either side of the other centerline which already had symmetry.

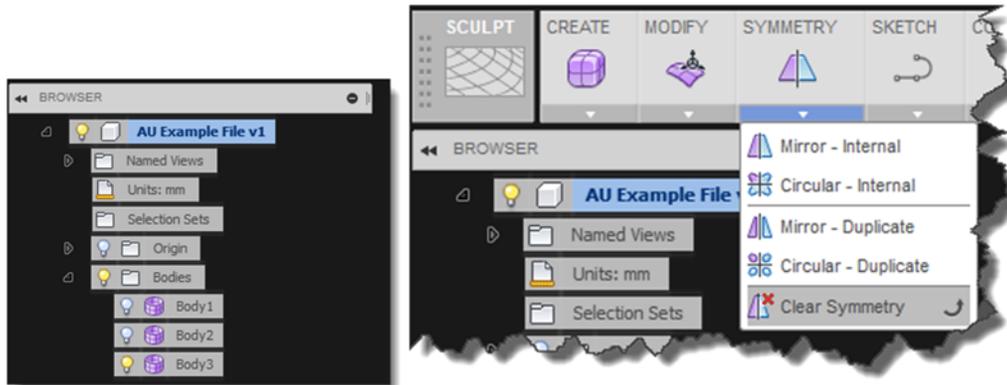


7. Double click one of the surface edges parallel to the symmetry plane you just created to select the entire edge loop (note the edge loop on the other side of the midline is highlighted as well)
8. Right click to bring up the marking menu and click “Delete” to remove the edges and simplify the form

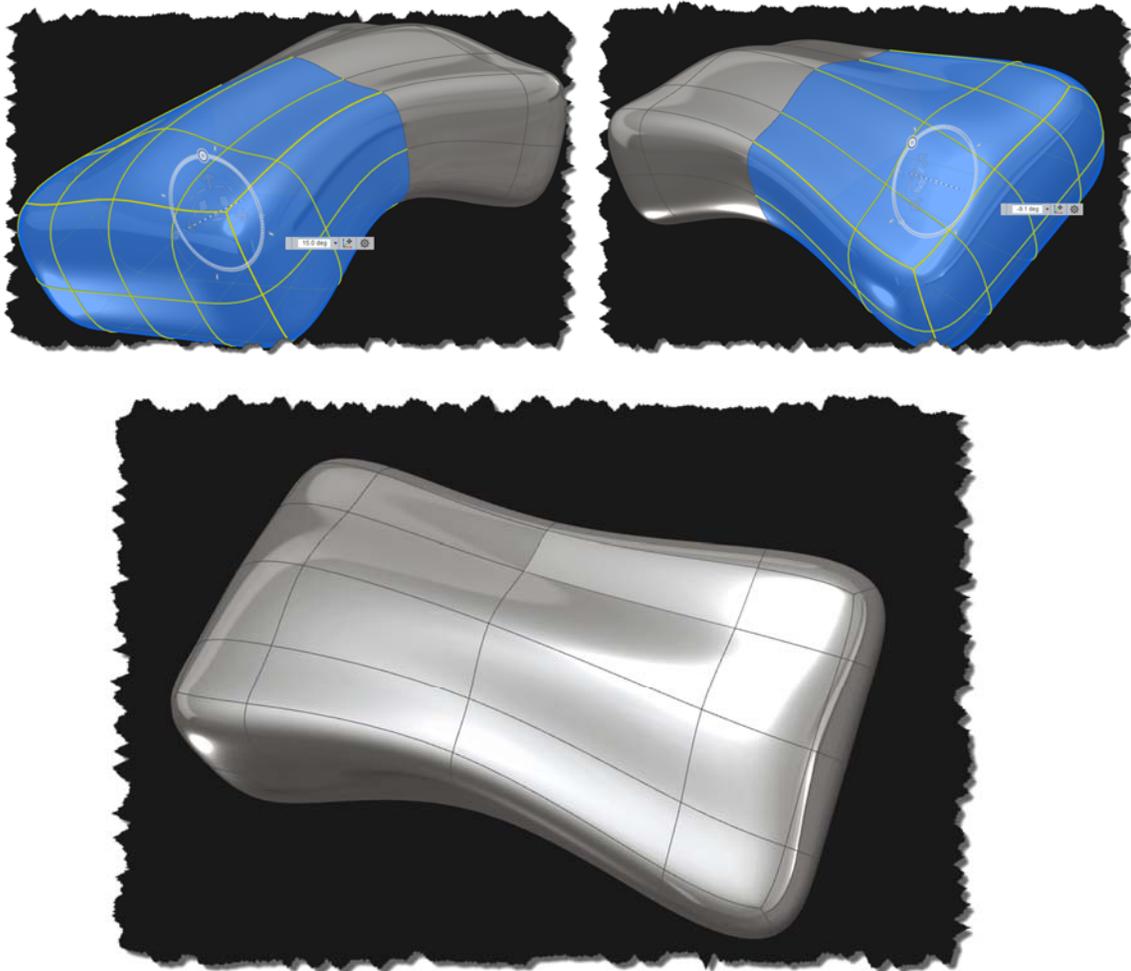


- For the third version you will remove any symmetry and modify the form to be more drastic than the first two

9. Turn off the visibility of Body2 and turn on the visibility of Body3 in the browser
10. Select “Clear Symmetry” from the Symmetry drop down menu and select Body3 from the graphics window to remove any symmetry



11. Once symmetry is removed “Edit Form” on each side separately to create an interesting form



Learning Objective 2 - Apply best practices when transferring data from Autodesk Fusion 360 to Autodesk Inventor software

Prepping data for export to Inventor

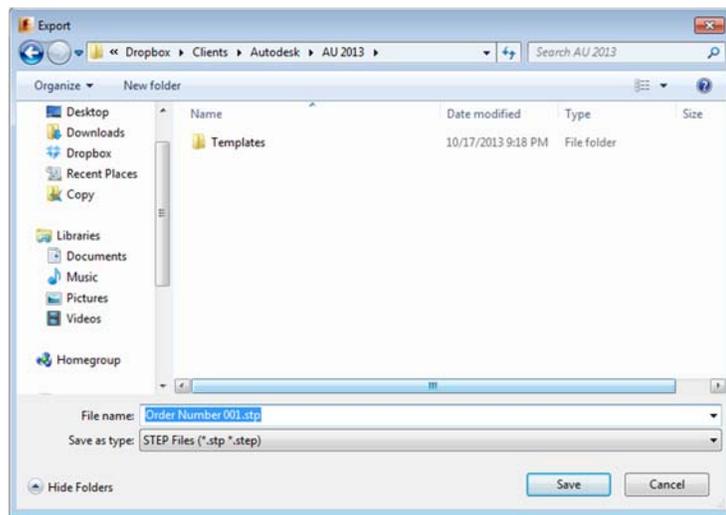
- The T-Spline bodies must be converted to NURB surfaces for consumption in to Inventor
1. Right click Body1 in the browser and click “Convert” to create a NURB surface version of your design.

Note: that the T-Spline body is kept with its visibility turned off

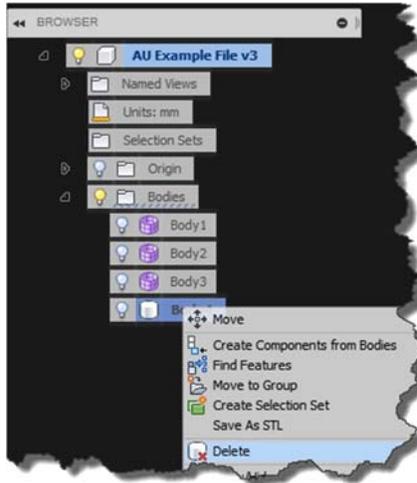
2. From the file dropdown menu click “Export”



3. Save the file as a .stp file



- Right click the newly created NURB surface and “Delete” it. All converted surfaces will become exported therefore it’s important to only export one surface at a time.

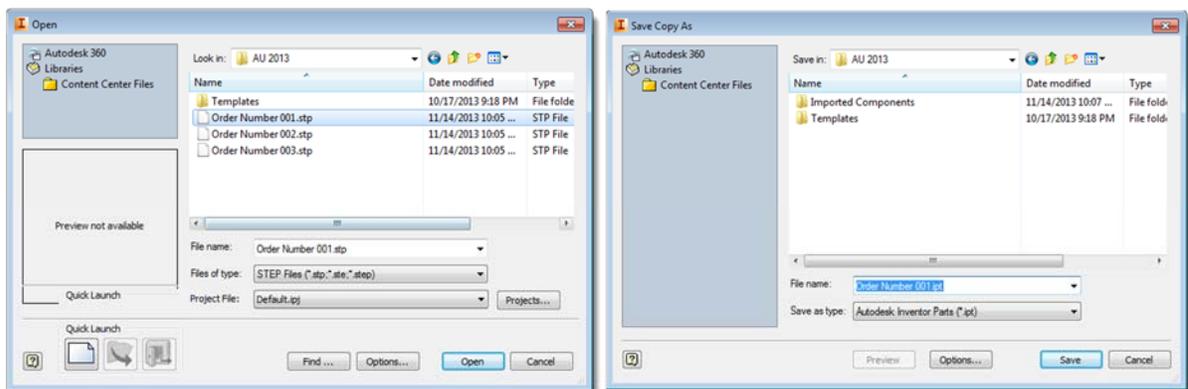


- Repeat the process for the other two bodies

Importing Fusion data in to Inventor

- Open each of the .stp files in Inventor and save them as native .ipt files

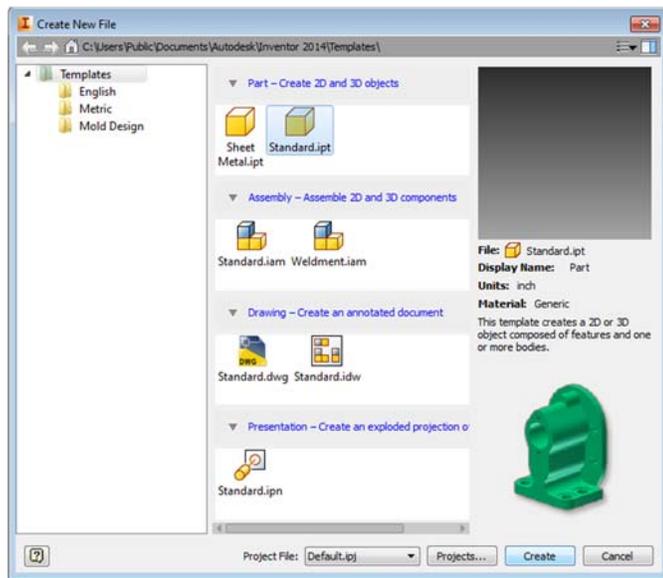
Tip: By default inventor creates a folder named “Imported Components” when converting non-native file formats. Inventor will attempt to save the converted file in to that directory if a new one is not chosen during the initial save.



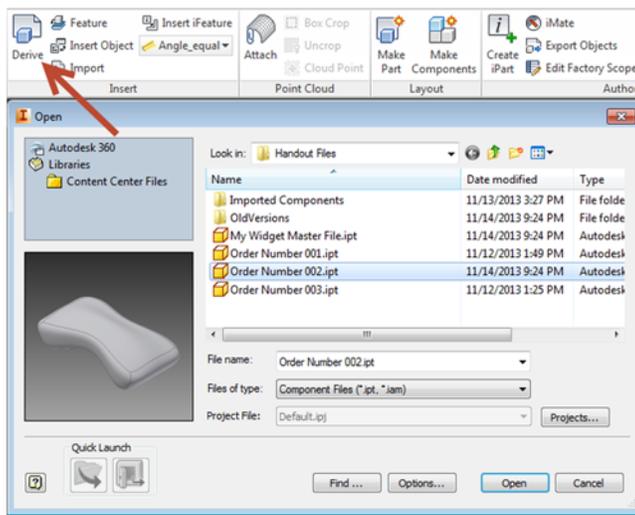
- Create a new .ipt file

- This file will be the master configuration file

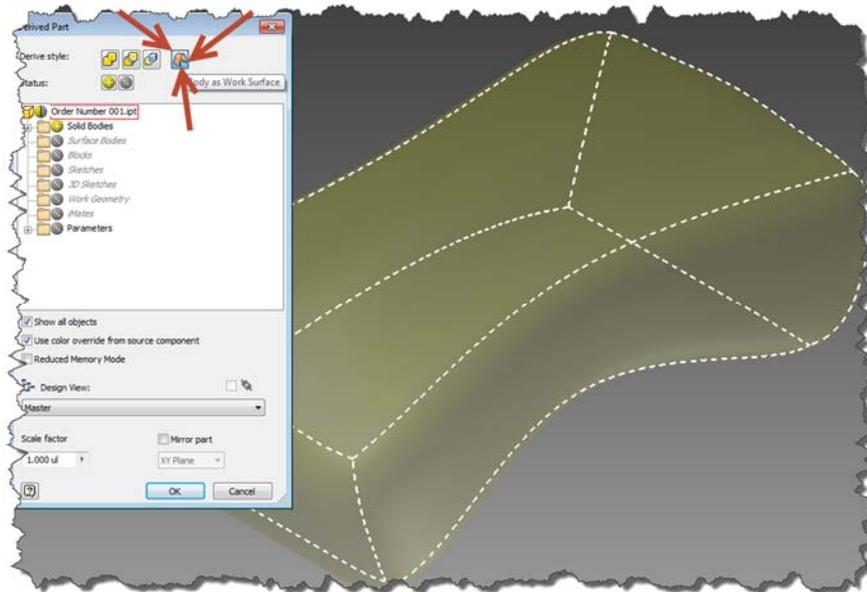
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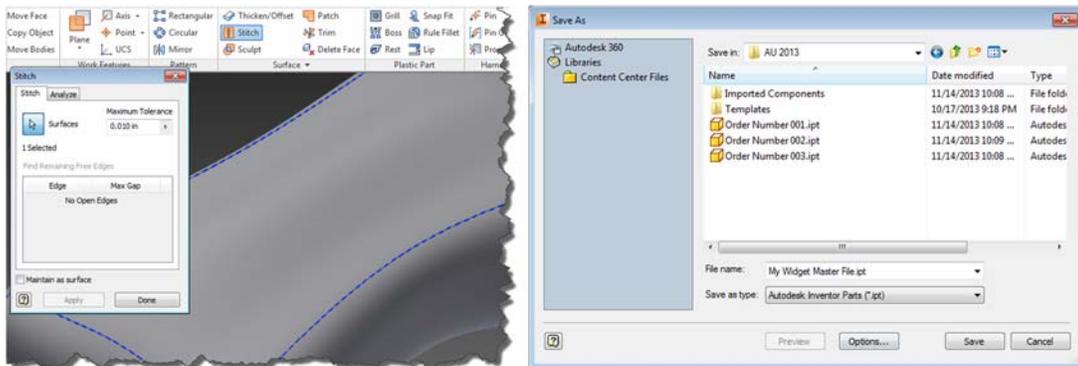
3. Click “Derive” from the “Insert” panel and browse to version one of your product



4. When the “Edit Derived Part” dialog box appears you must click the surface icon and bring the data in to the master file as surfaces

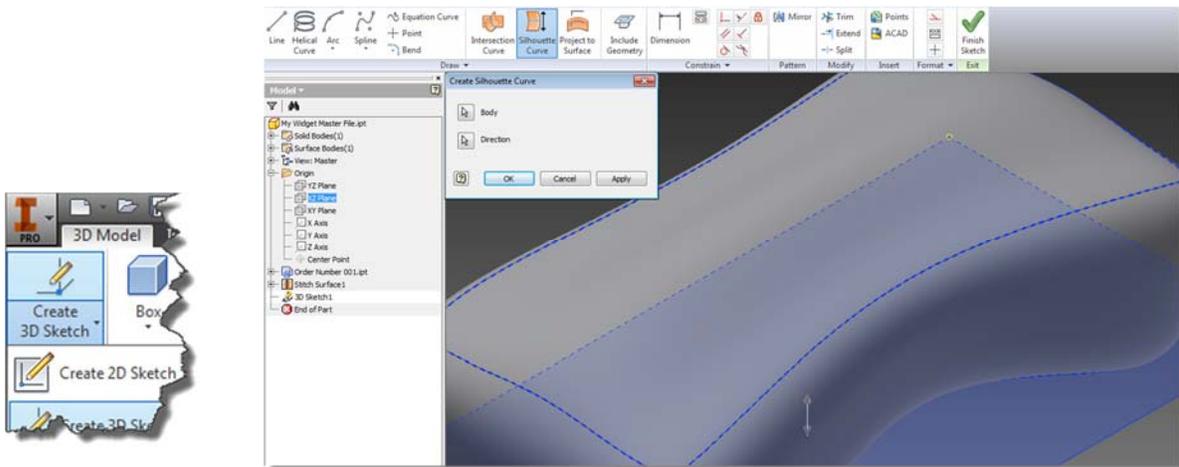


5. Next use the “Stitch” command to turn the surface in to a solid
6. Save the file as your master configuration file

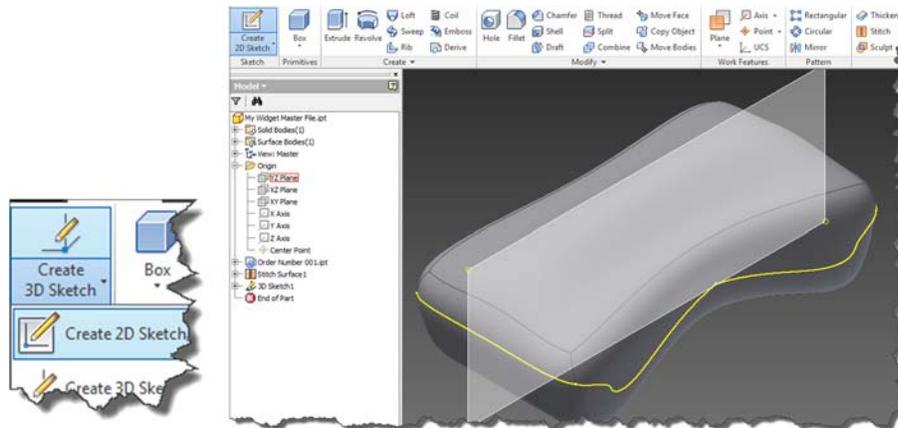


- Add the detail to your design that would exist across all versions of the product
- Ensure the coordinate system is set up properly, if not consider using a move body command
- Always use reference geometry when possible
- Only reference model geometry if you absolutely have to
- Solid body modeling is your friend
- Always plan for future geometry changes, i.e. extrude past what you think is necessary and in both directions

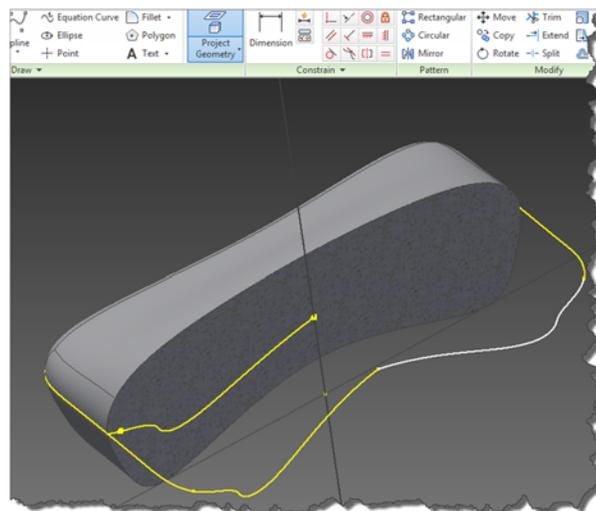
7. Create a 3D sketch
8. Create a silhouette curve to determine a potential parting line



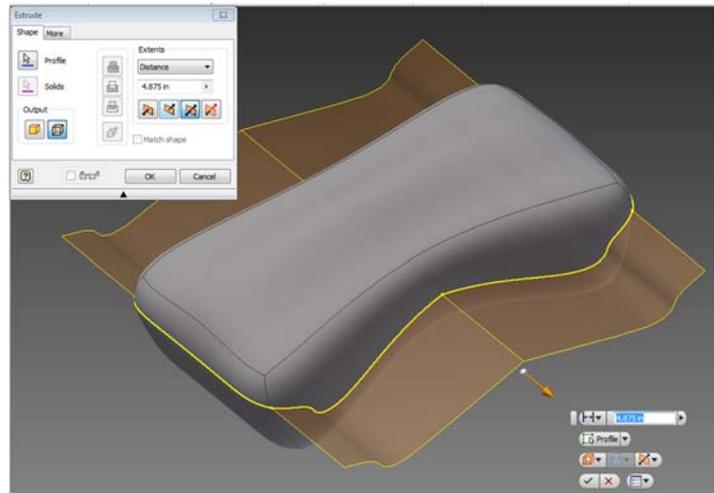
9. Create a 2D Sketch on the bisecting origin plane



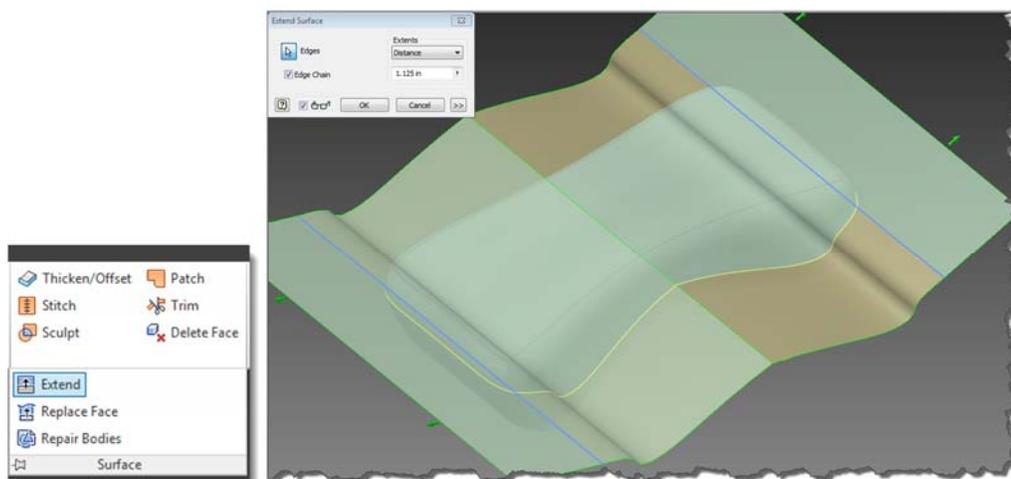
10. Project the silhouette curve to the sketch plane



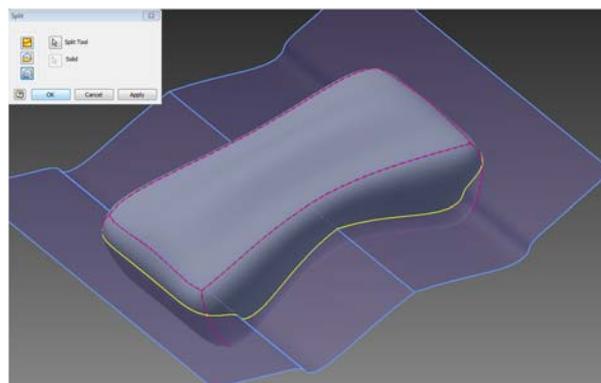
11. Extrude the projected curve as a surface in both directions. Note: Be sure to extrude past the width of the largest version of the product



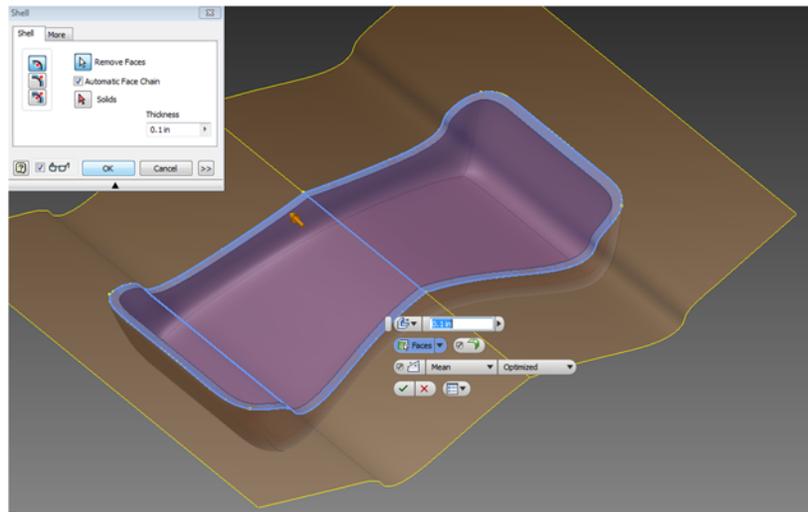
12. Use the “Extend” command to extend the surfaces enough to fully intersect the solid body



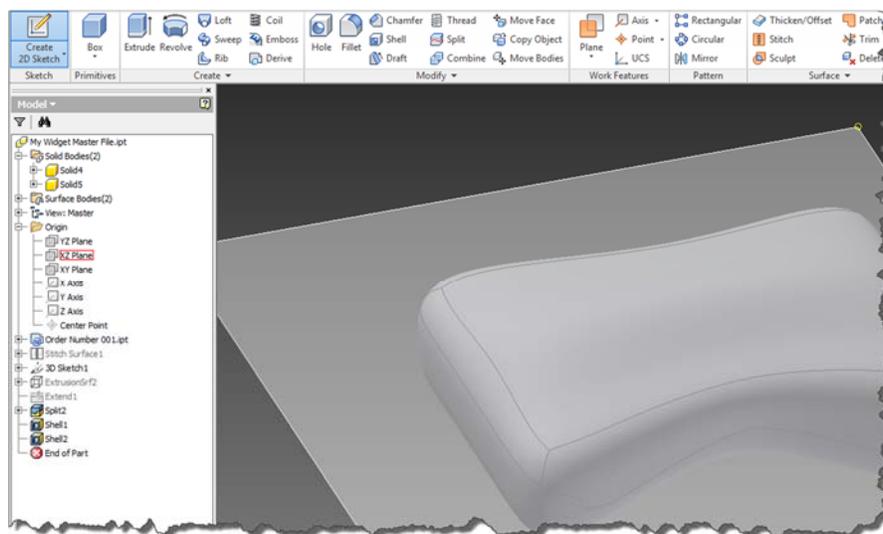
13. Using the split command split the body in to two separate solid bodies



14. Shell each half of the product



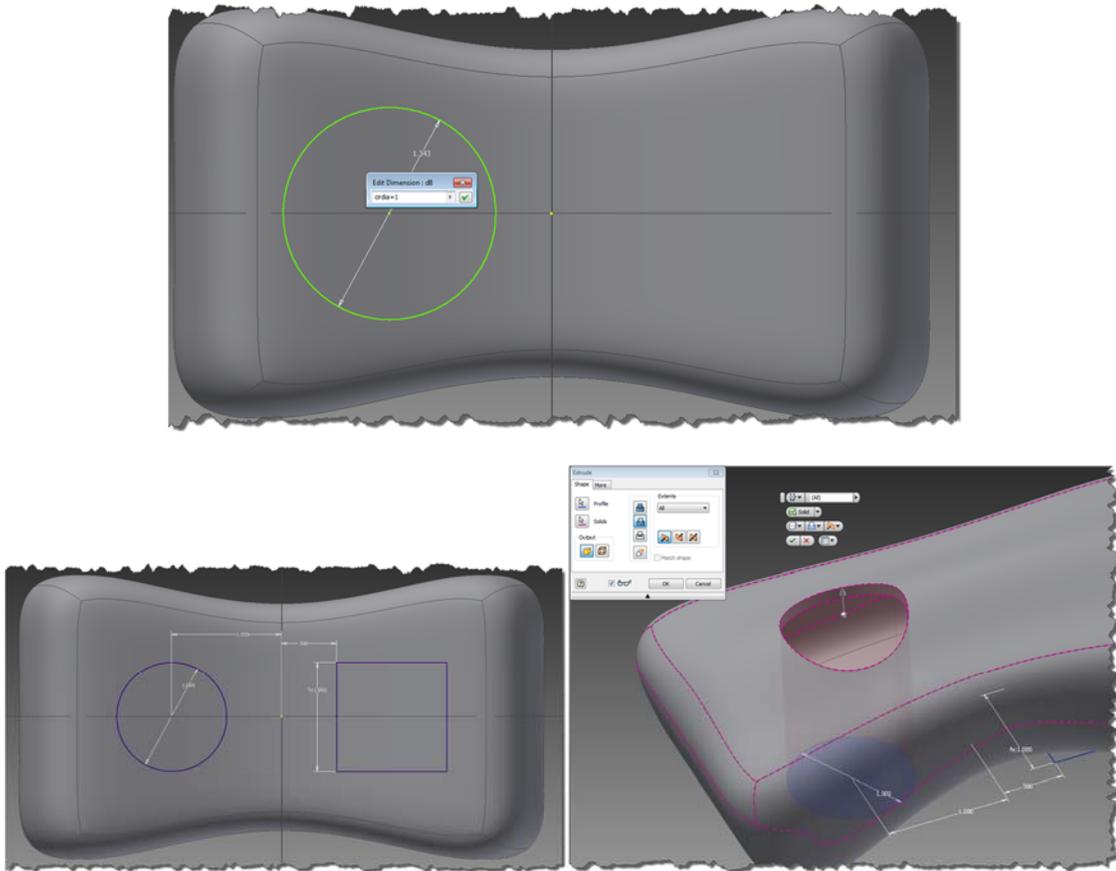
15. Create a sketch on the origin plane for additional features



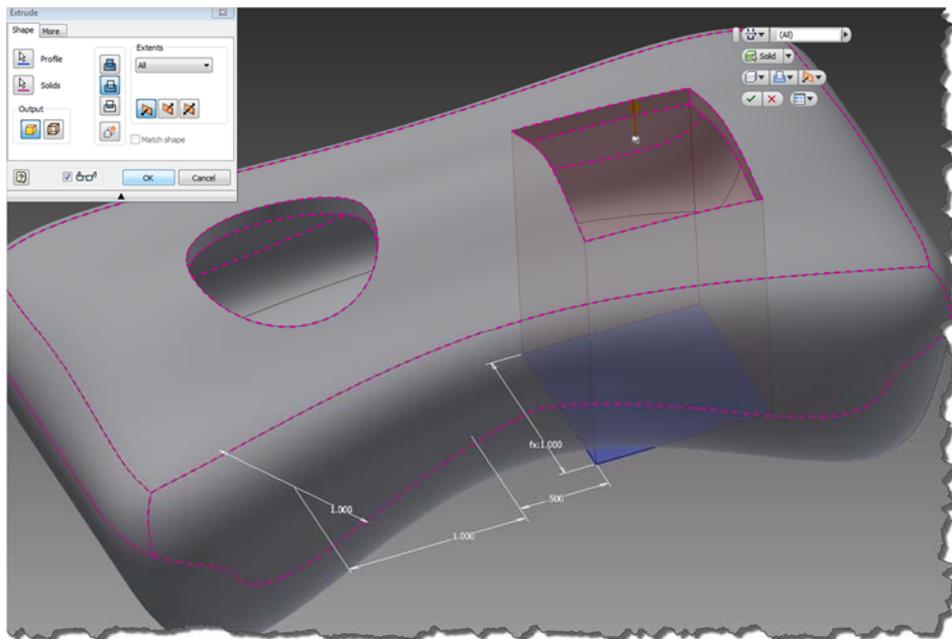
16. Create several profiles on the new sketch for the creation of several unique extrusions

17. Add parameter names to any dimensions that may be different from one product to the next

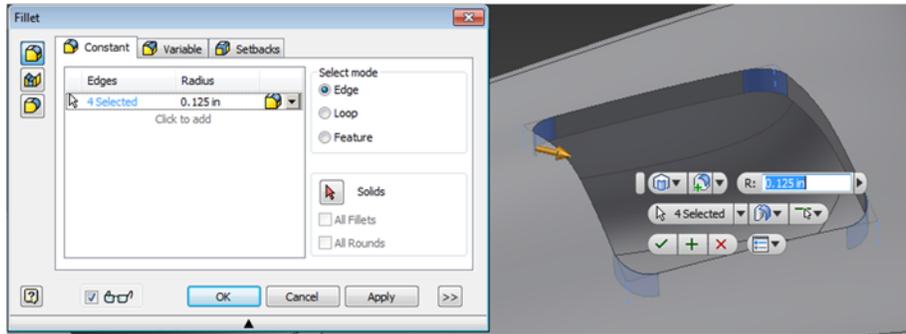
Tip: Turn on show expression under dimension display to easily identify parameter names



18. Extrude a cut for each of the profiles you just created through one of the bodies



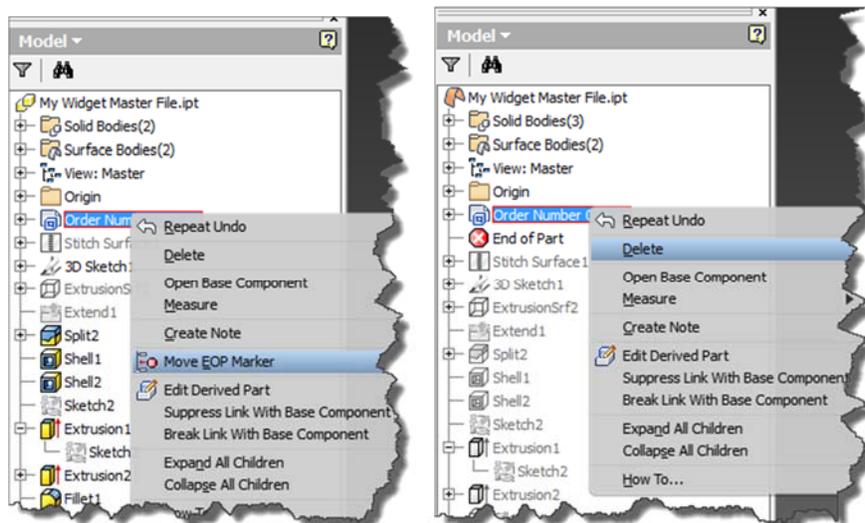
19. Add a secondary feature such as a fillet to one of the cuts



Learning Objective 3 - List the pros and cons of different customization techniques with Autodesk Inventor

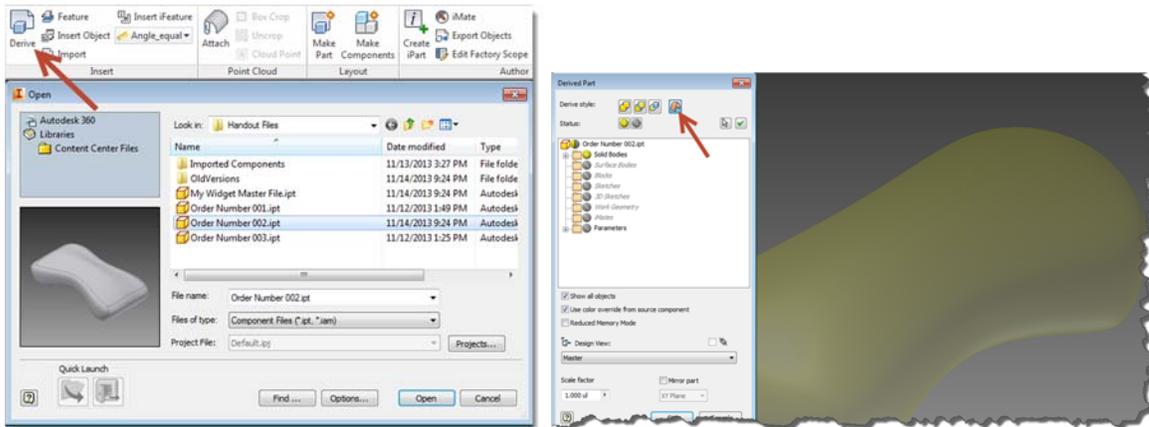
- There are multiple ways of setting up a configurable file with many different levels of complexity and control.
- The first method is how to import a new surface file from Fusion 360 while maintaining of the detail work that has been previously performed

1. Right click the name of the derived part in the browser and click “Move EOP Marker”
2. Right click the derived part and “Delete” it

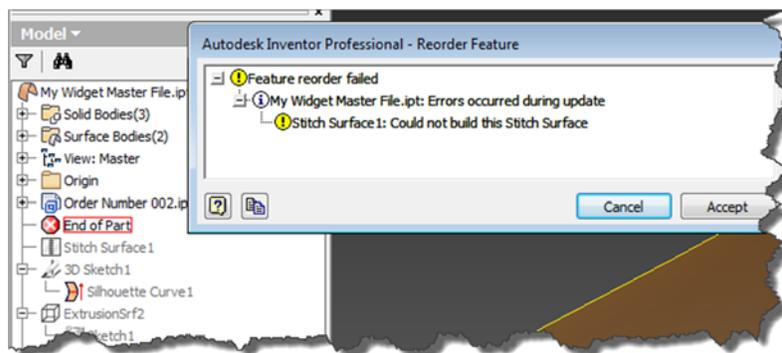


3. Click “Derive” from the insert panel and browse to the second version of your design
4. Be sure that the new .ipt file is brought in as a work surface as before

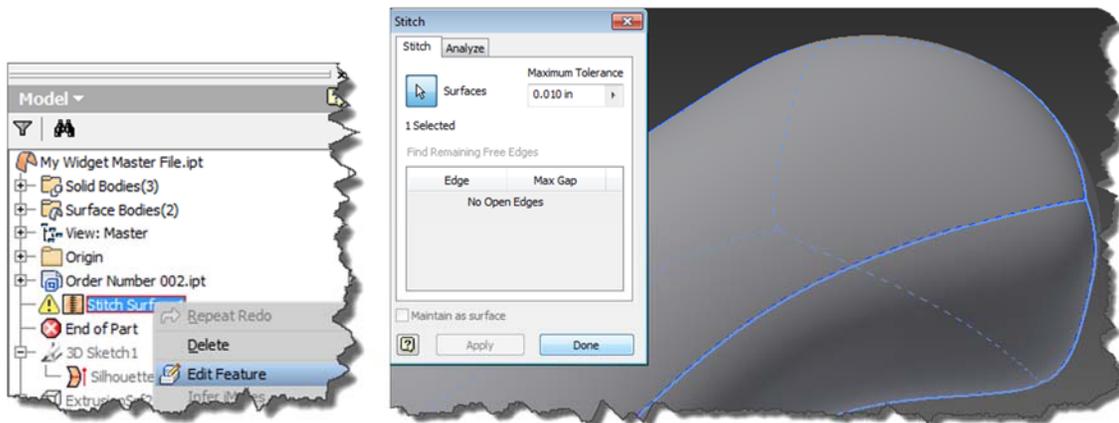
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5. Drag the “End of Part below the “Stitch Surface1” feature in the browser tree
6. When prompted by the “Reorder Failed” dialog box click “Accept”



7. Next “fix” the failed Stitch feature by right clicking and picking “Edit Feature”
8. Select the newly imported surface and click “Apply”



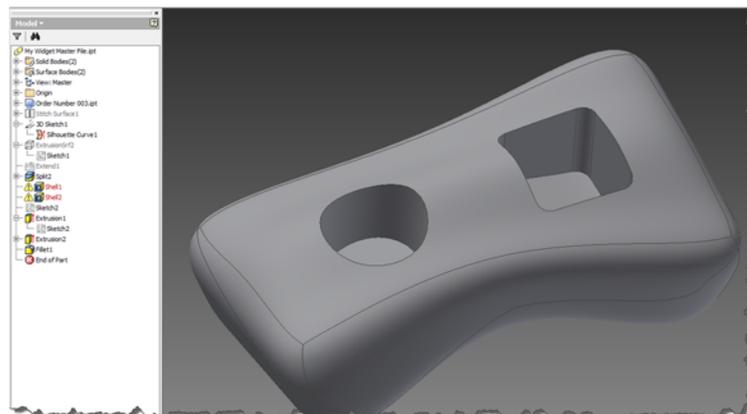
- Inventor will now recognize the surface as the original one and each of the following features will be properly mapped

9. Drag the “End of Part” to the end of the browser tree to see the result

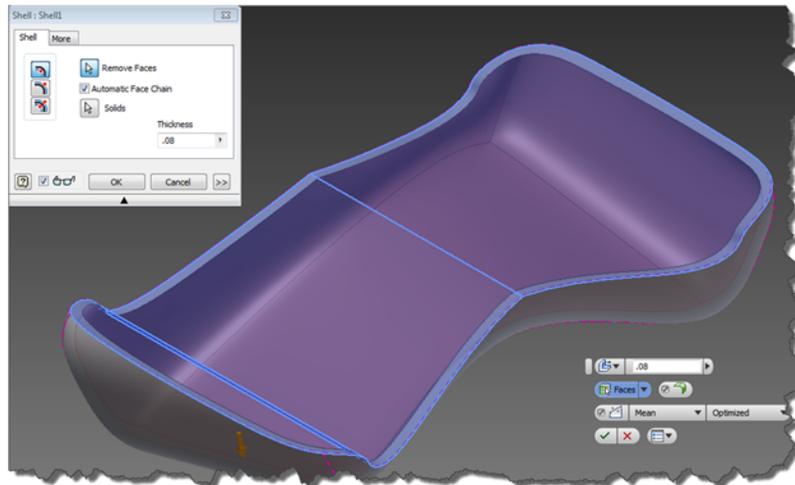


10. Repeat the process for the third version of the design

- Sometimes features will still fail for various reasons
 - Don't panic
 - Repairing the model is always easier than it looks
 - Start by repairing the model with the highest feature in the browser tree that failed first
 - The model will begin to sort itself out as you repair the features
 - Note: Always repair features by editing them vs. deleting and recreating them. If a feature depends on the feature you delete it will make fixing the model more difficult
-
- In this case the two “Shell” features failed



11. Each of the shelled solid bodies need a thinner wall thickness to compute



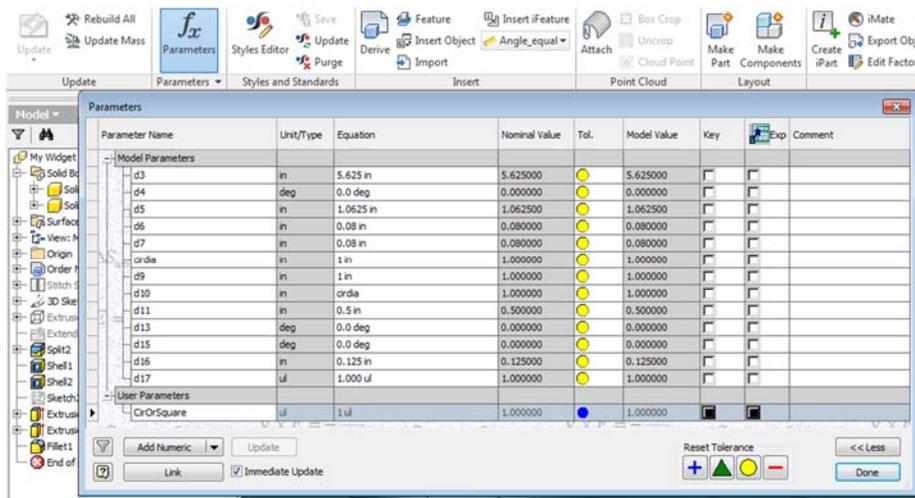
- One way to toggle features on and off without creating a table based part is to create a user parameter to be used for cycling through different configurations of a design

12. Click “Parameters” from the ribbon to bring up the Parameters dialog box

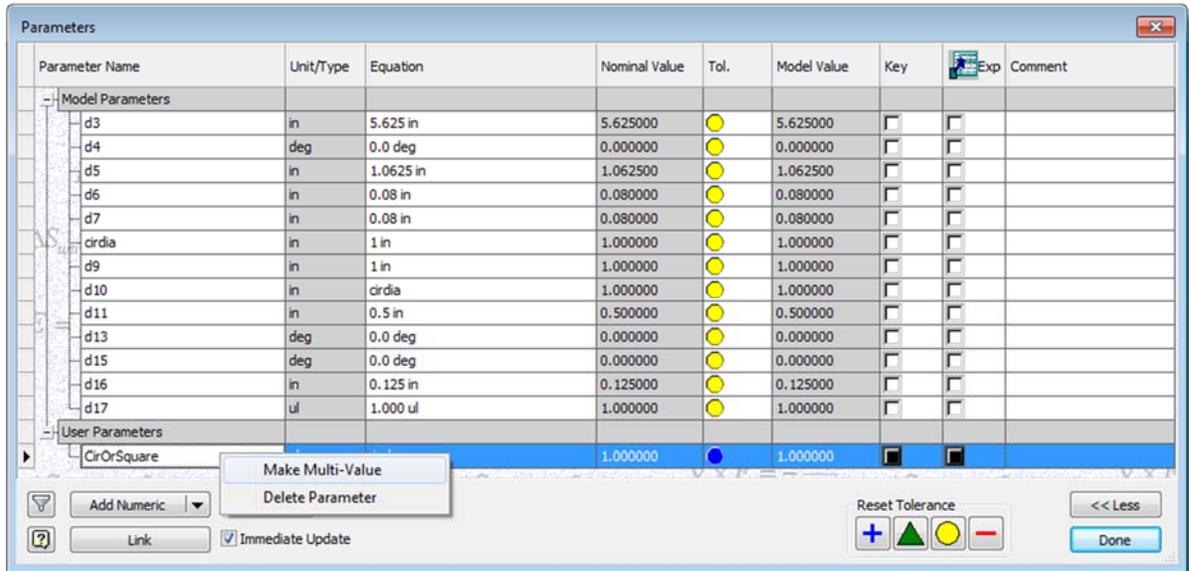
13. Click “Add Numeric” to create a new user parameter

14. Name the parameter CirOrSquare or something similar (be sure to not use spaces)

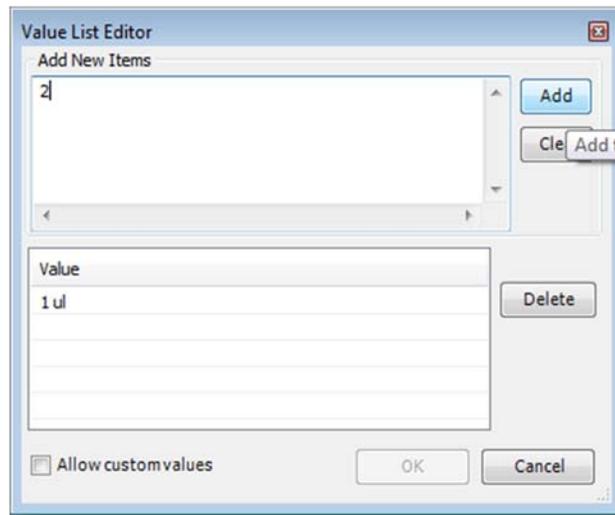
15. Assign the unit type to unitless (ul) and assign a value of 1



16. Optionally create a dropdown to select from predetermined values by right clicking the parameter and choosing “Make Multi-Value”



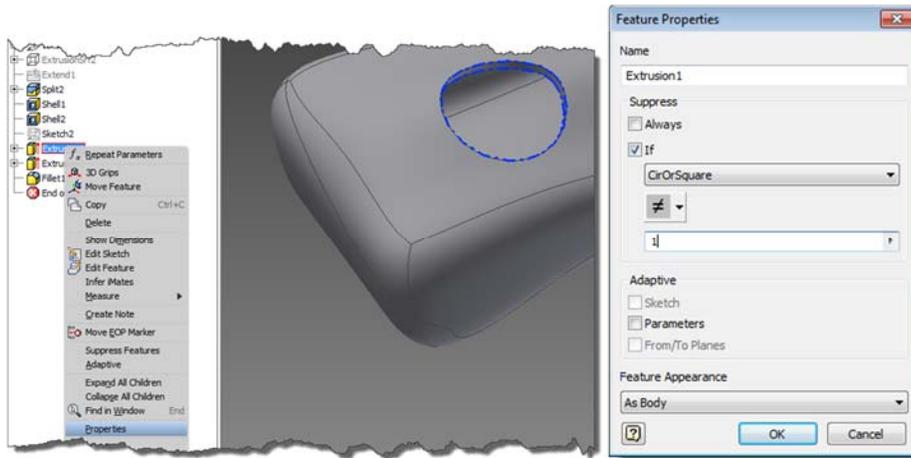
17. Add an additional value for each combo of features you would like to later cycle through. In this case there are only two (either the circle or the square cut)



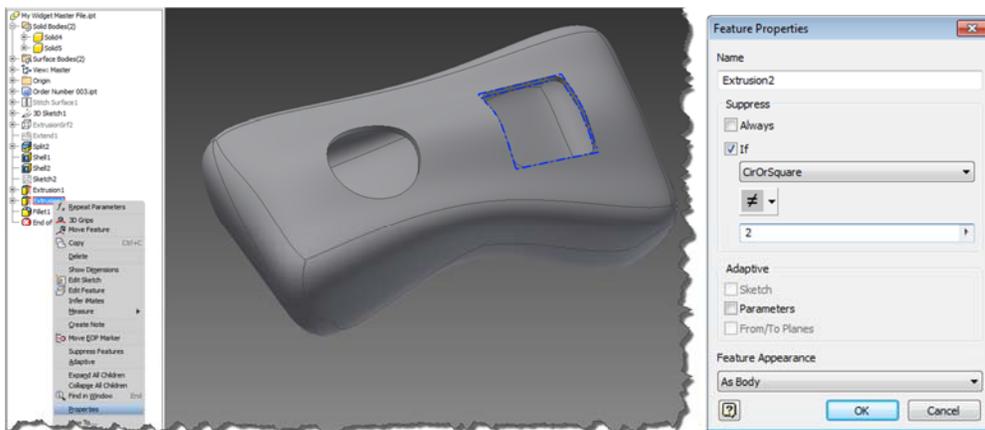
- Assign the rule to the feature to assign which value will suppress the feature

18. Starting with the circular extrusion right click the feature and click “Properties”
 19. From the Feature Properties dialog box check the “If” box, select the user parameter you just created and set it to suppress when the value is not equal to 1

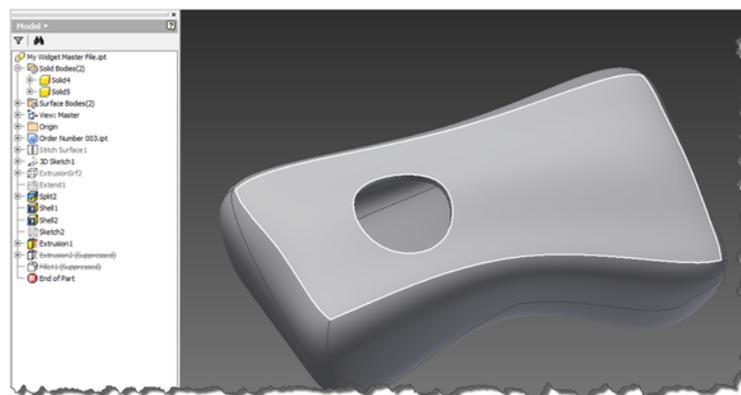
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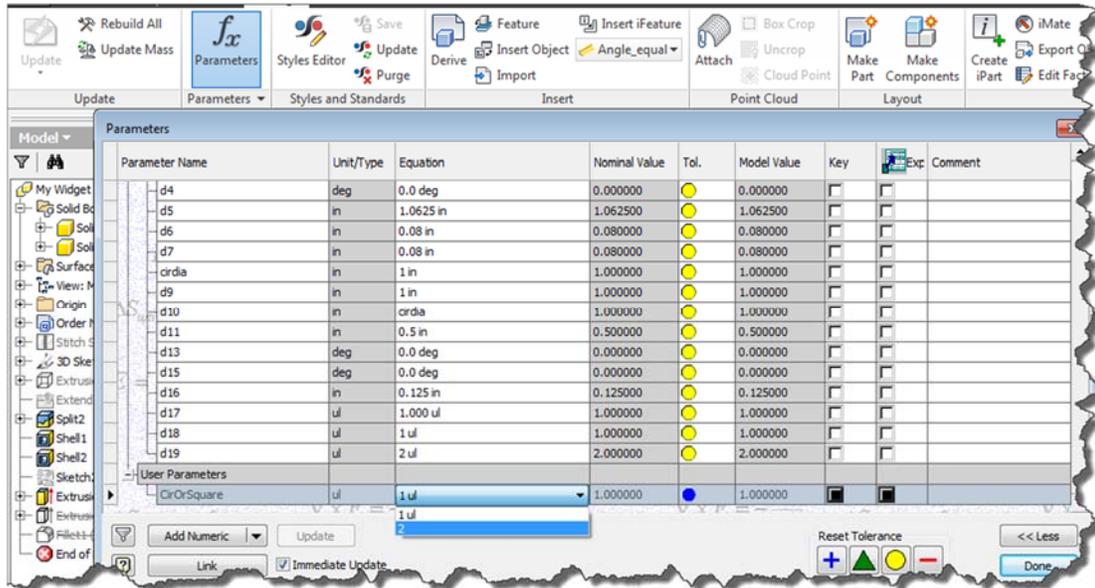
- 20. Repeat the process for the square extrusion but this time assign the value of 2
- 21. Click "Ok" to see the result



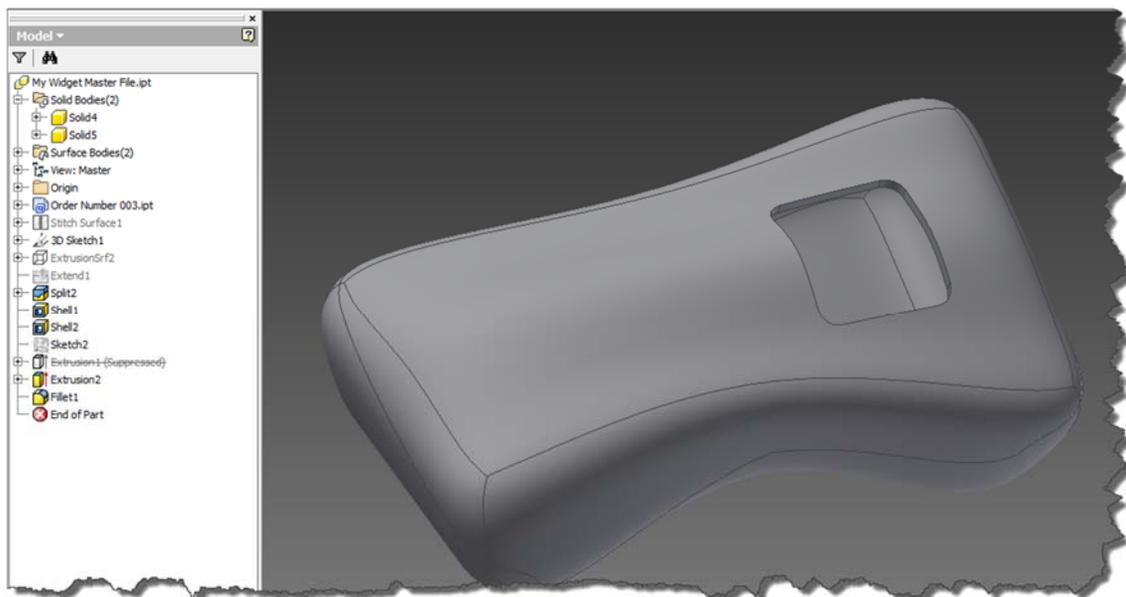
- Notice how only one feature will be computed at a time now



22. To toggle the square extrusion back on and suppress the circular extrusion in one step go back to the Parameters dialog box and select “2” from the dropdown list you created

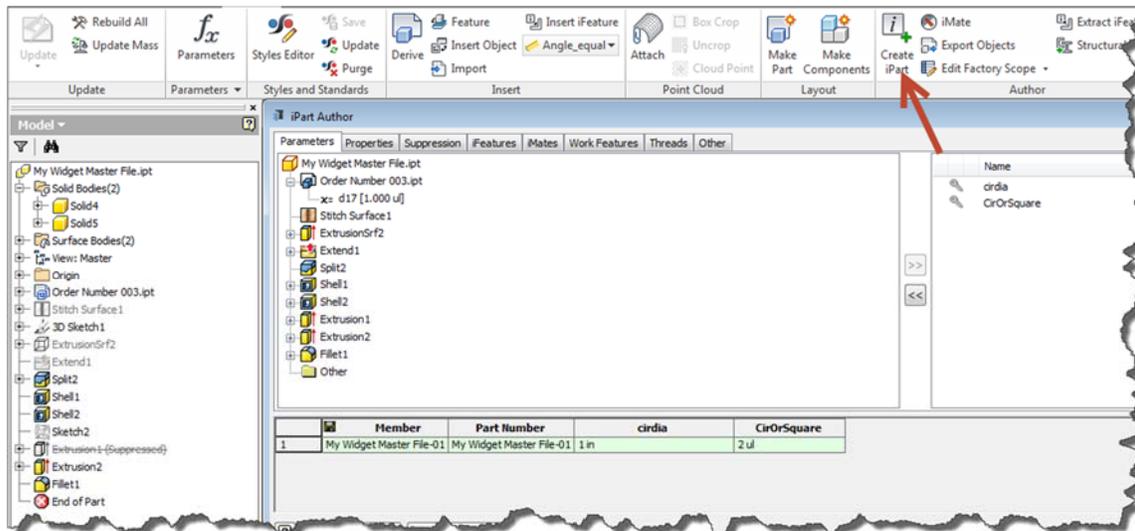


23. View the result

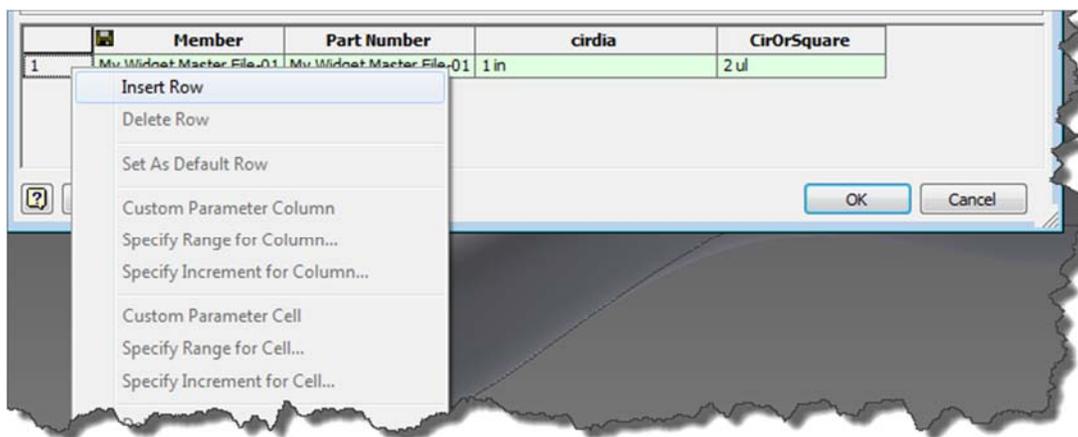


Alternatively you may also choose to create an iPart (table based part) of your design to more easily toggle and modify multiple parameters quickly

1. Click the “Create iPart” icon from the Author panel in the ribbon
2. An table will appear that will automatically have columns for each of the named and user parameters you created



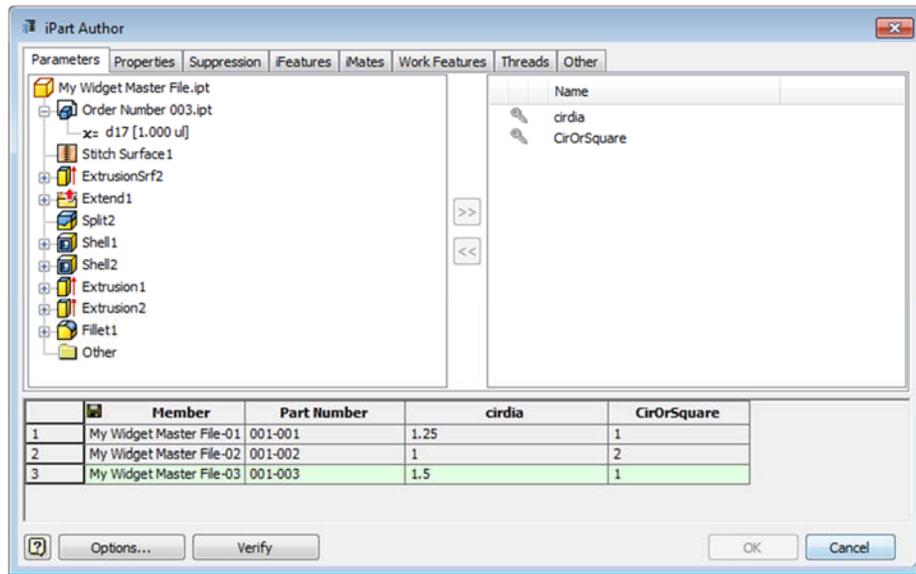
3. Right click the row in the table and click “Insert Row” to add additional versions of the design



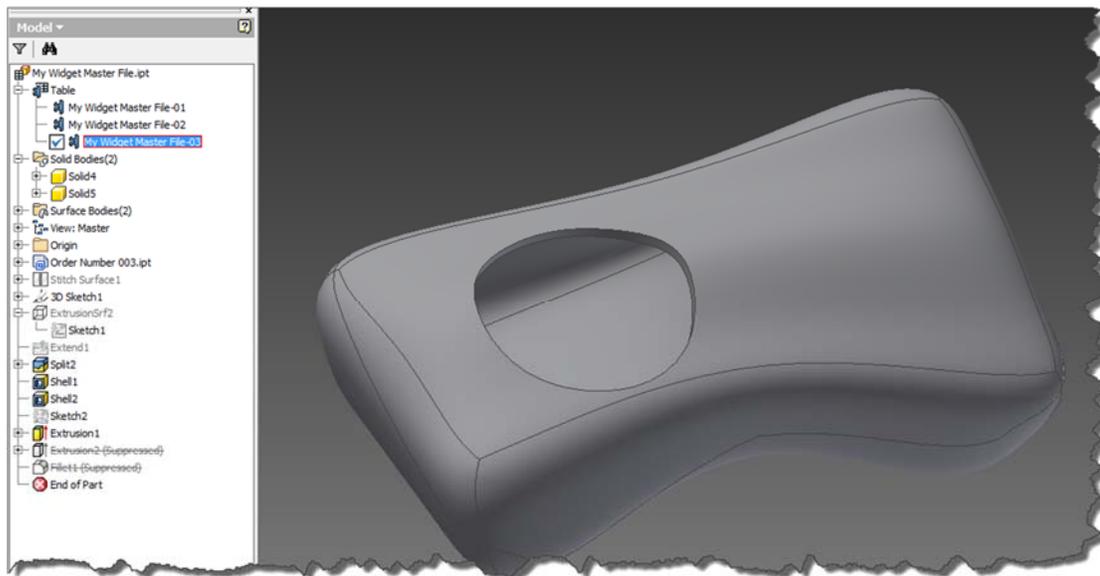
Tip: Now would be a good time to assign part numbers to each version of the design

4. Create multiple versions of your design to change the size of the circular extrusion and toggle off and on the extrusions

Note: the “cirdia” parameter is irreveleant in row 2 as shown as that extrusion will be toggled off



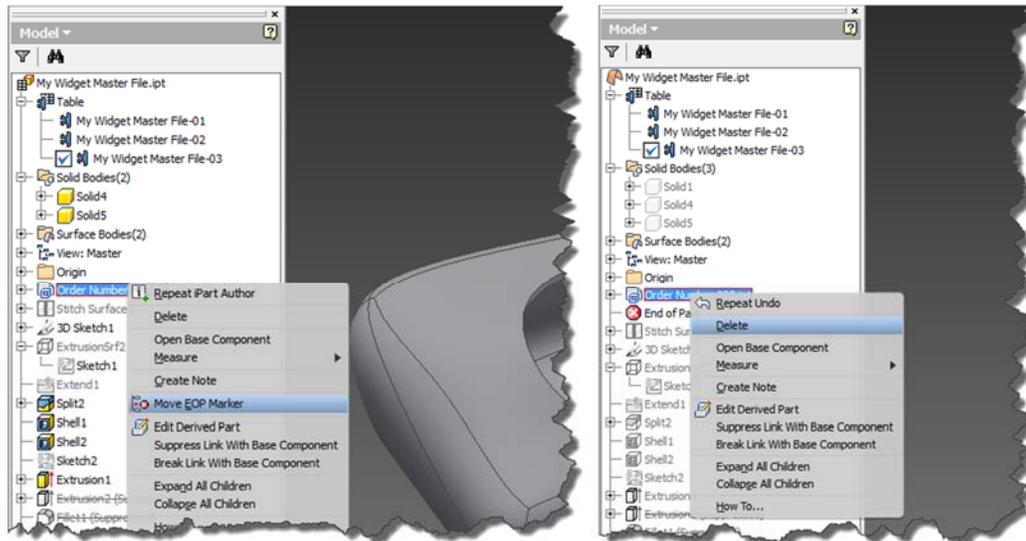
5. Click “Ok” and make notice of the “Table” section that is now in the browser
6. Double click each row of the tables to see the results and test your configuration



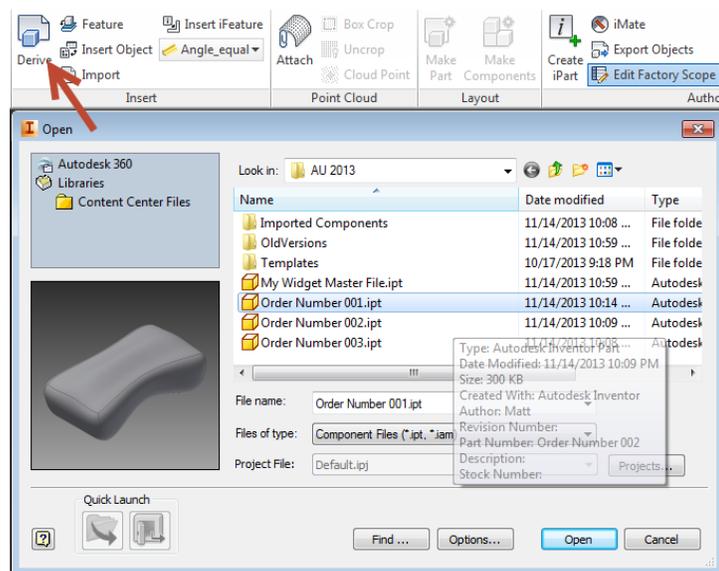
Advanced Consumer Product Customization Techniques with Autodesk® Fusion 360™ and Autodesk® Inventor

- The same method of bringing in multiple surface versions from Fusion 360 still apply and work with iParts

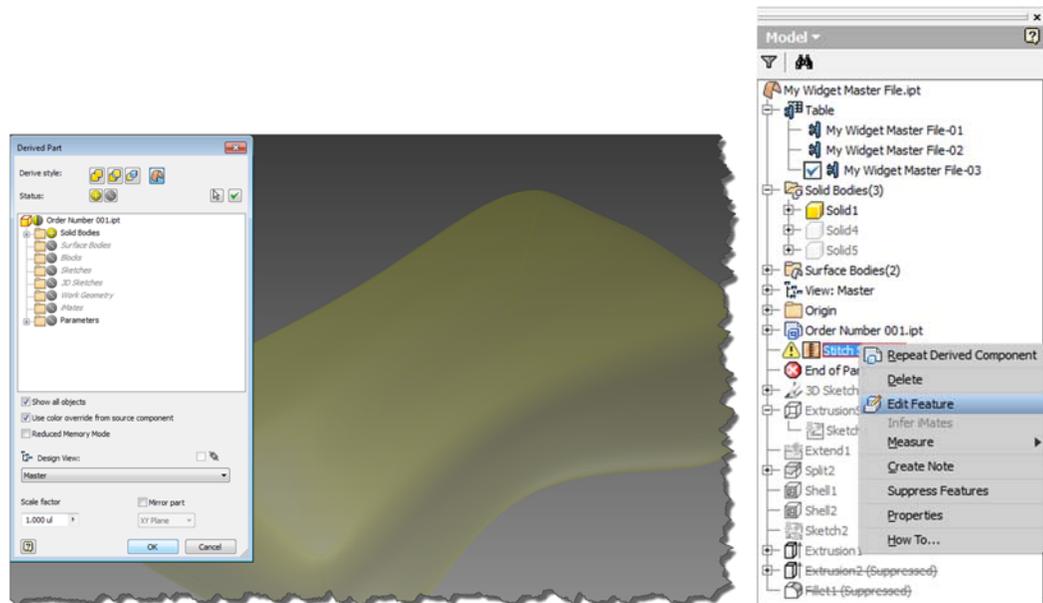
7. Move the End of Part just below the derived .ipt file as before
8. Delete the derived component



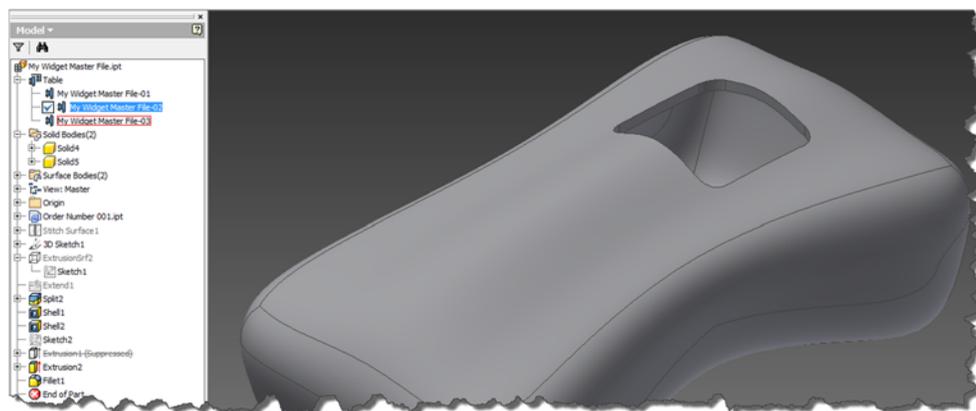
9. Derive in a different version of your design
 - Again be sure that the file is brought in as a work surface



10. Move the End of Part below the stitch feature and repair the feature



11. Click through the different versions of the iPart to see the results



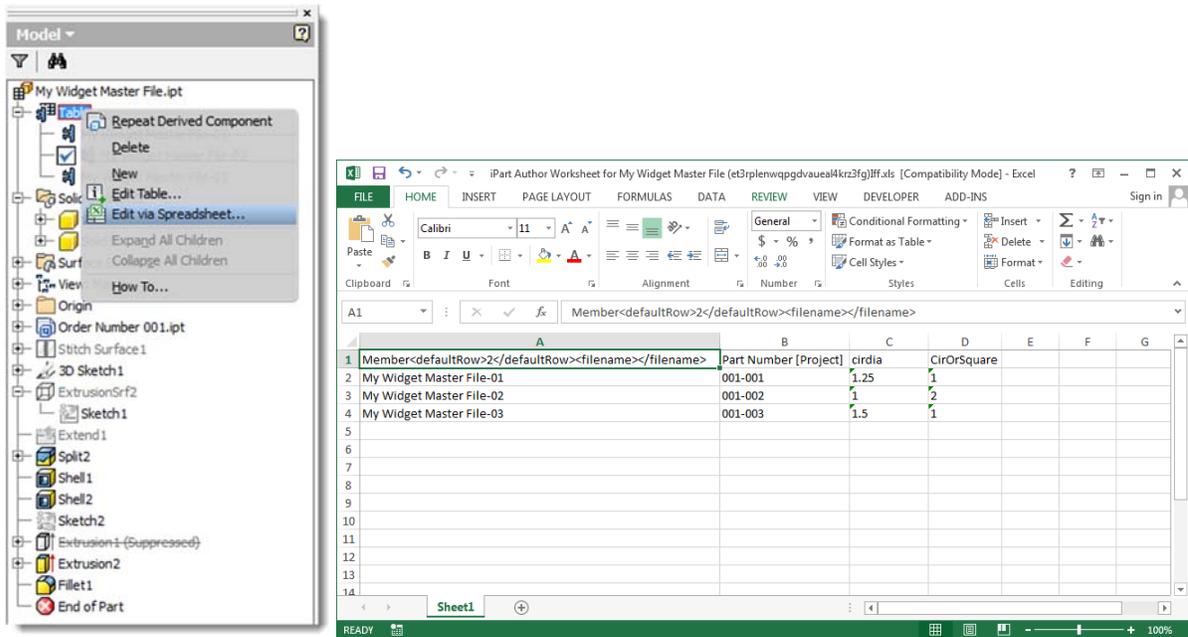
- Alternatively Excel can be used to edit the rows of the iPart table
- This may help with the sales process as the designer/engineer can receive an excel doc from sales and easily update the configuration with a copy/paste operation

12. Right click one of rows of the table in the browser and choose “Edit Via Spreadsheet”

13. If Excel is installed on your machine, it will automatically open

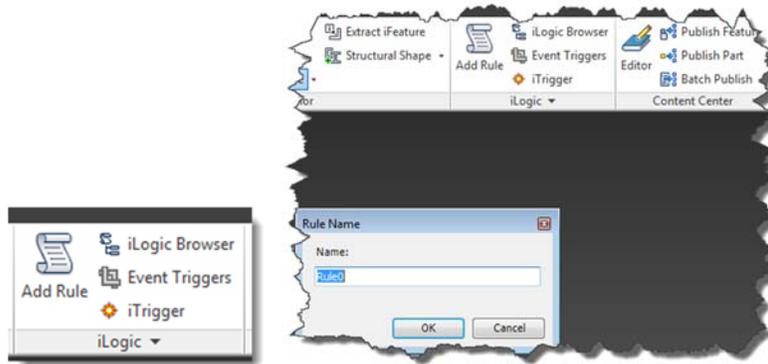
14. Simply modify the spreadsheet as desired and close it to see the updated result

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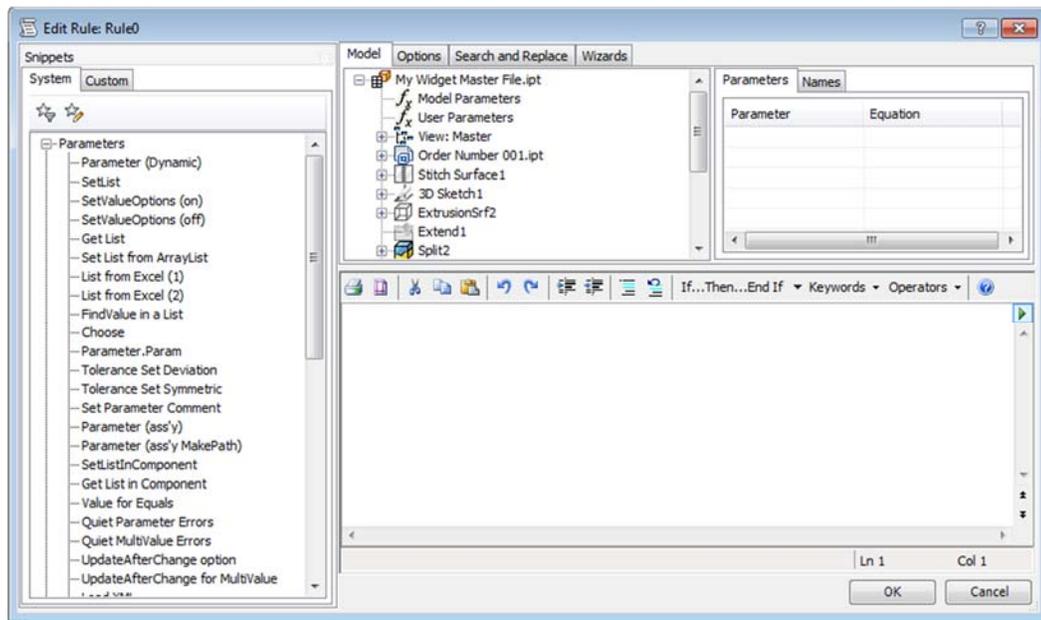


- iLogic may also be used to build in intelligence in to your master design file

15. Select “Add Rule” from the iLogic panel and click “Ok” to see the Edit Rule dialog box



Note: iLogic is extremely powerful and therefore may become complex. iLogic isn't for everyone but the point is there are no limits once a link has been established between your master config file and your new version of Fusion 360 geometry



Learning Objective 4 - Apply best practices for creating flexible organic models with Autodesk Fusion 360

In summary, creating multiple versions of complex geometry in Fusion 360 and importing the data in to Inventor can be a very valuable and time saving process for many businesses. Fusion 360 is a new product and therefore the functionality of Inventor in this process may begin to shift to Fusion 360 depending on future product enhancements.

The key points to be aware of when creating geometry in Fusion 360 are

- Work within a consistent coordinate system. As you develop your configuration file it may be helpful to pass data back and forth from Inventor to Fusion and not having to move the geometry to align it every time is beneficial
- Be careful to only export the data you need in to Inventor. Bringing multiple converted surfaces at the same time may complicate the process.
- Keep the Fusion geometry as clean as possible. Save all detail features for Inventor when possible
- Fusion geometry typically should be “water tight” to ensure the cleanest import in to Inventor. It’s not a requirement. For instance, you may choose to thicken the surface in Inventor as the first feature to reestablish a link vs. stitching. That’s fine.
- Have fun and experiment! Please share your results and findings with me so that I may continue to learn as well.