SESSION 7
SHAPE DESIGN

Overall Customer Benefit

Customers can create cast and molded parts that meet design requirements for aesthetics, ergonomics, and manufacturing constraints because Inventor now delivers tools to create a wide range of complex geometry.

Customer Needs and Inventor Solutions

The primary needs of customers working with shape design are listed in the table below. For each Customer Need, the Solutions that Inventor offers are shown along with a top-level look at how those Solutions are accomplished in the column entitled How It's Done.

Customer Needs	Solutions	How It's Done
Create a wide variety of blended shapes with profiles and control curves	Enhanced Loft • Centerline rail • G2 edge continuity • Loft to a point	• Create two or more curves to control the cross section • Create a path or centerline to control the sweep direction
	Enhanced Sweep • Normal profile option • Two-rail control • Tangency control	• Create a profile to define a cross-section • Create a curve to control the path of the sweep • Optionally create a curve to use as a guide rail
Create shapes with sharp or rounded ends	• Loft to a point with sharp or rounded end conditions	• Create section profiles and 3D points where required • Select the profiles and point(s) to define the loft • Select the desired end condition
Place fillets on complex geometry	Enhanced Fillets • Face – Face Fillets • Full Round Fillets • G2 Edge Fillet	• Select the appropriate Fillet option • Full Round Fillets • Select side and top fillets and apply the Full Round fillet
Shape and manipulate	• Improved 3D sketch	• Project curves to surface

Customer Needs	Solutions	How It's Done
complex 3D curves		• Project 2 curves to create 3D curve • Edit tangency handles on 3D splines • Use Smooth (G2) constraint
Use surfaces to define or modify the shape of 3D parts	• New Sculpt Tool	• Create a set of surfaces that defines a fully enclosed volume and use the Sculpt tool to generate the corresponding 3D part • Create surfaces to define material to be removed from a part or use surfaces to define material that you want to add. Then use the Sculpt tool to modify the part.
Make sure that all ribs are created with correct draft angle	• Rib Tool with new Taper option	• Create sketch geometry to define the location of the ribs • Use the Rib tool and enter the required taper angle

Autodesk Advantage

Inventor delivers hybrid surface and solid design technology with innovative tools to sculpt solids with surfaces. For AutoCAD users, the advantage is the capability to move into 3D shape design with fast and easy-to-learn modeling tools.

Hands-On Exercises

To provide practice with the new Inventor functionality described in this section, the exercises listed below – along with their learning objectives – are detailed on the following pages with step-by-step instructions.

1. Use Sketch Enhancements
    - Use Curve Enhancements
    - Use Project to Surface
    - Import Excel Data Points
    - Create and Edit a 3D Spline

2. Use Loft Enhancements
   - Create a Center Line Loft
   - Create a Loft to a Sharp or Round Point
   - Create a Smooth G2 Loft

3. Use Sweep Enhancements
   - Use Guide Rail Sweep
   - Guide Surface Sweep

4 Use Sculpt Features
   - Use the Sculpt Feature

5. Use Rib Features
   - Use Rib

6. Use Surface Features
   - Extend Surfaces
   - Trim Surfaces
   - Create a Boundary Patch

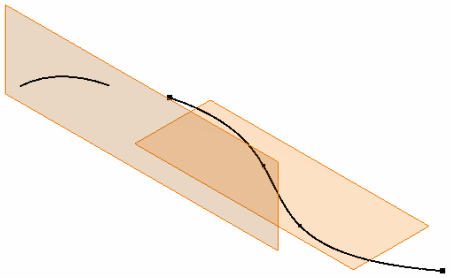
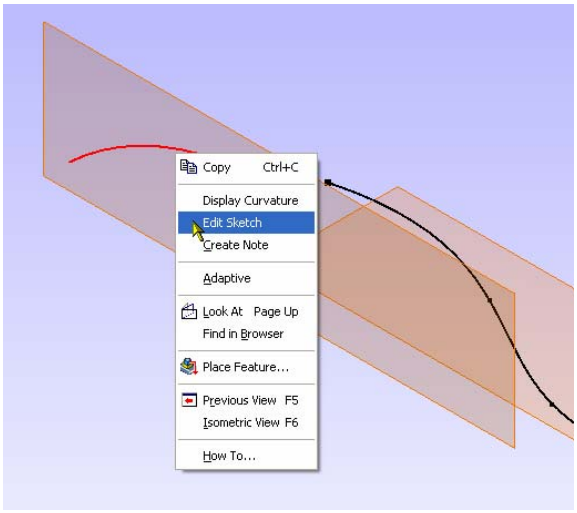
7. Use Analysis Tools
   - Create a Proximate Offset
   - Use Analysis Visibility
   - Measure Distance between Two Parts
   - Make Changes in Work Geometry

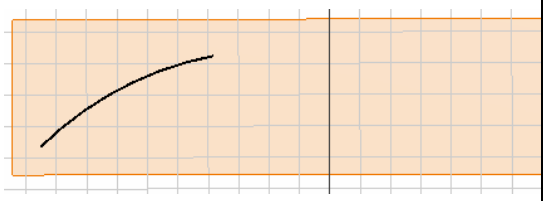
Exercise Datasets

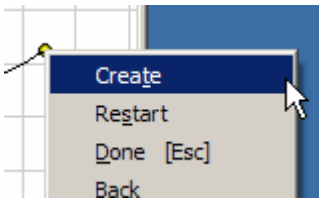
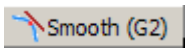
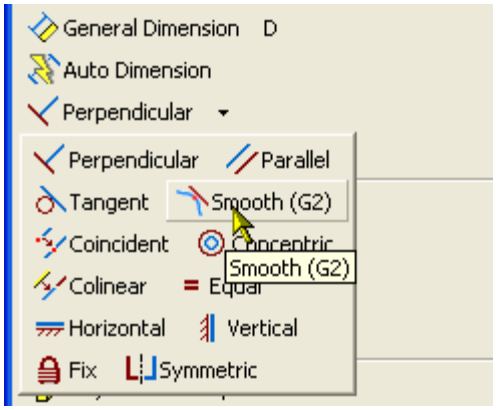
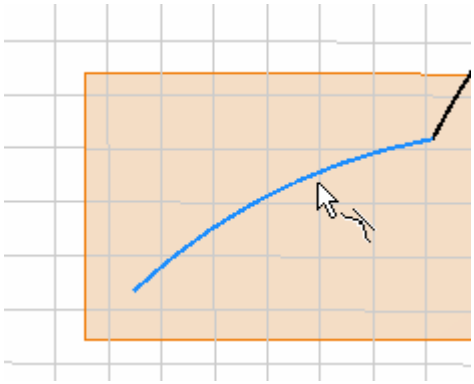
- 11 Loft_02 (Centerline).ipt
- 01 Sketch.ipt
- 14Sweep_01_(Hairdryer).ipt
- 17 Sculpt2.ipt
- 18 RibWithDraft.ipt
- 19 Trim,Extend,BP 2.ipt
- Trim,Extend,BO 2.ipt
- 21 Shell,Thicken,Offset.ipt

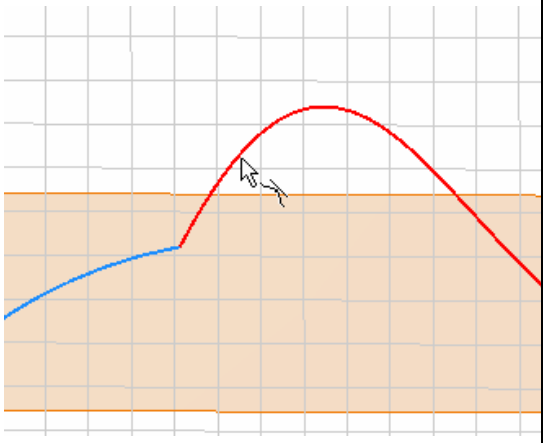
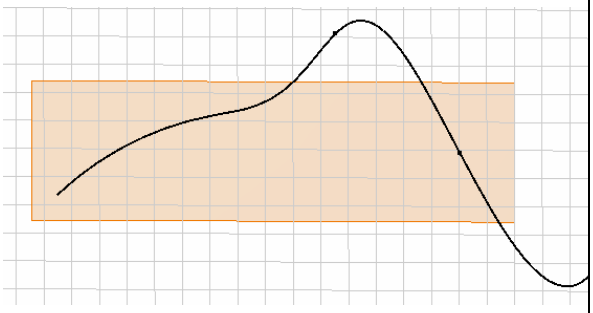
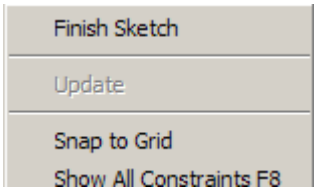
Exercise 1: Use Sketch Enhancements

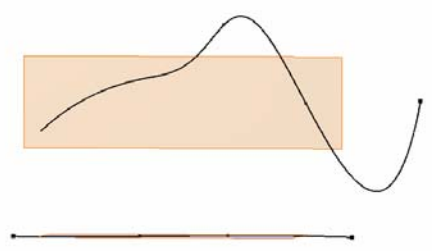
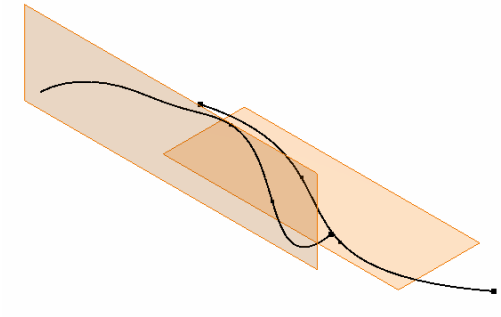
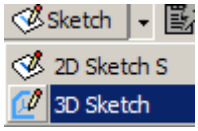
Task: Use Curve Enhancements

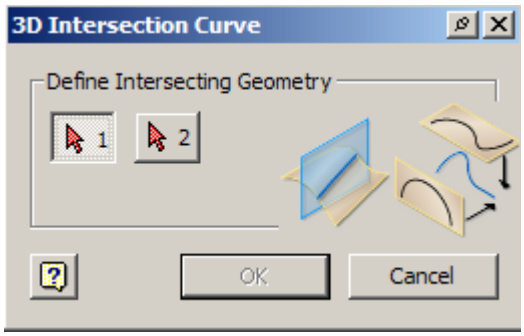
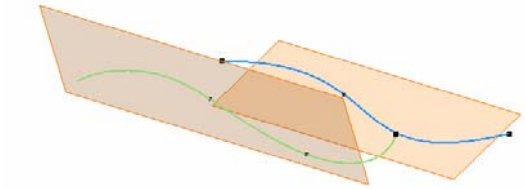
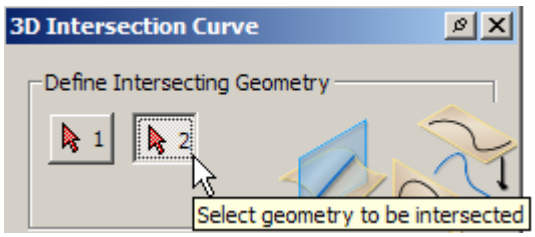
Step	Action	Result
1.	• Begin by launching Inventor 11. • Select **File** menu > **Open** > **01 Sketch.ipt**.	
2.	• Select **Open**.	Shape appears. 
3.	Right-click on the shape for menu. Select **Edit Sketch** to launch the 2D Sketch Panel. 	

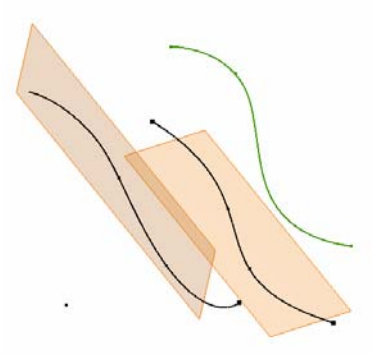
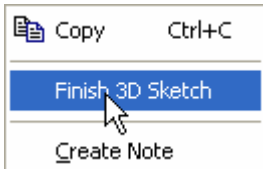
Step	Action	Result
4.	• Use the **Look At** tool and select the sketch to rotate the sketch normal to the viewing plane (optional). 	
5.	Create a spline. • Select **Spline** from the Sketch Panel. 	
6.	• Click end of line and drag to create a spline. 	
7.	• Click at three or four separate points to create the spline.	The spline should look like this. 

Step	Action	Result
8.	• When done, right-click and select **Create** to finish. 	
9.	Previously, there was the ability to apply a tangent constraint. Now a smooth or G2 constraint can be applied. • Select the **Smooth** constraint from the list of available 2D sketch constraints. 	
10.	• Select the arc. 	

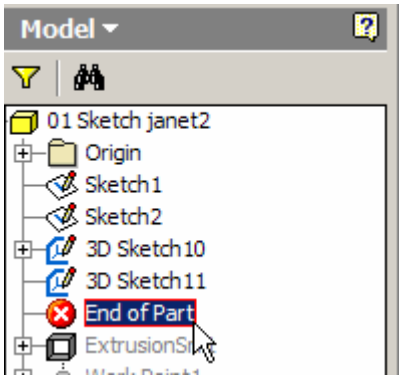
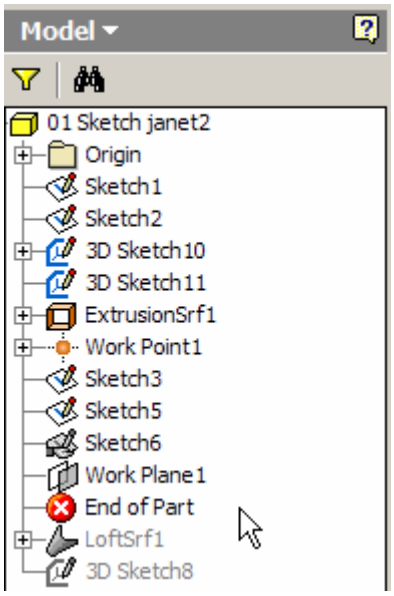
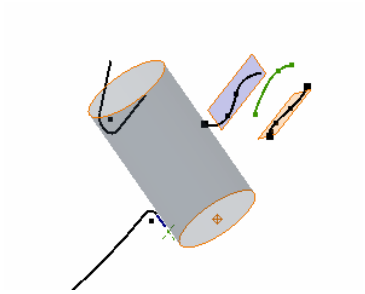
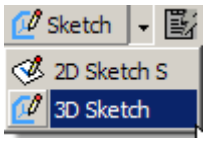
Step	Action	Result
11.	• Select the spline. 	Spline curvature is constrained to match that of the selected arc. 
12.	• Right-click in the graphics window for the menu.	
13.	• Select **Done**.	
14.	• Right-click again for menu.	

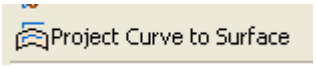
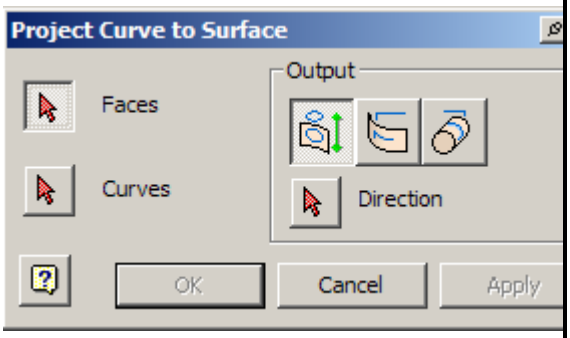
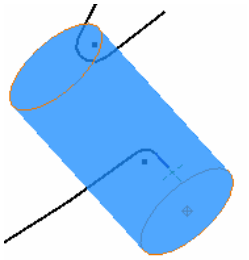
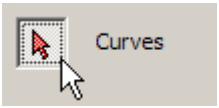
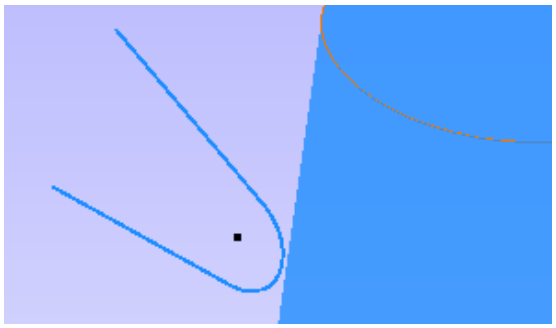
Step	Action	Result
15.	• Select **Finish Sketch**.	
16.	• Rotate the sketch until it looks like this. 	
17.	Go into 3D Sketch Mode • Select **Sketch drop-down** > **3D Sketch**. 	

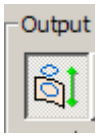
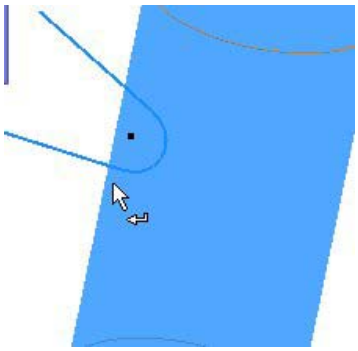
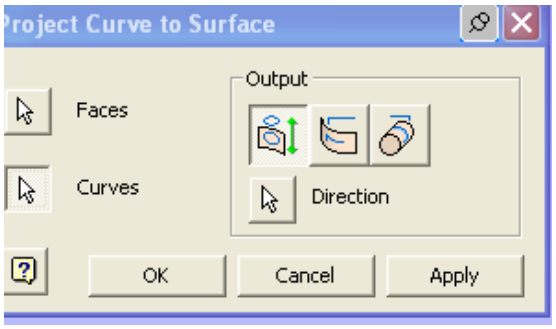
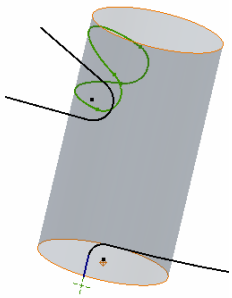
Step	Action	Result
18.	Click **3D Intersection**. • **New**: You can now select 2d sketch curves in addition to faces, as in R10. 	
19.	• Select the highlighted curve. 	
20.	• Select **2,** is now automatically highlighted. Select the second curve 	

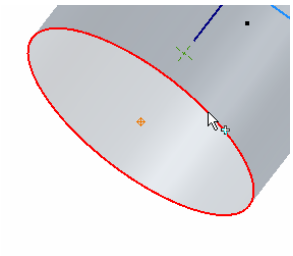
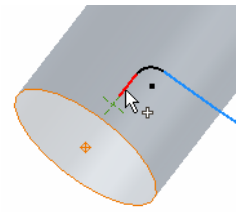
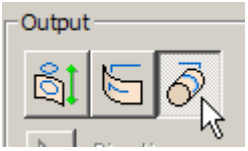
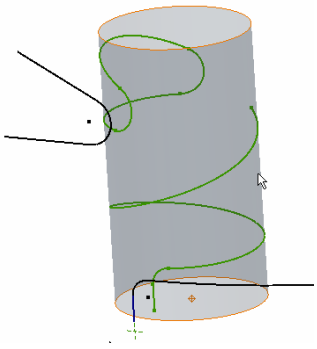
Step	Action	Result
21.	• Select **OK**.	The two 2d curves are projected, normal to their respective sketch planes, to generate a resulting intersection curve. 
22.	Complete sketch. • Right-click in the graphics window. • Select **Finish 3D Sketch**. 	The Part Features Panel appears.

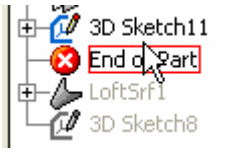
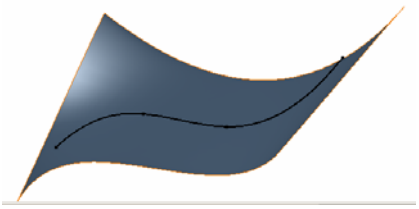
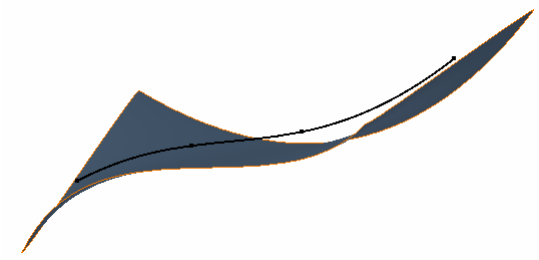
Task: Use Project to Surface

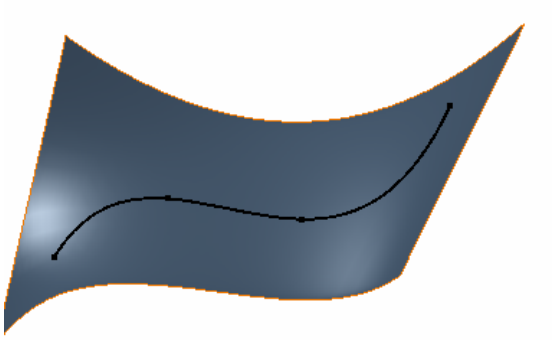
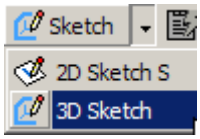
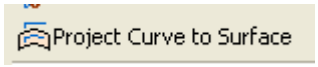
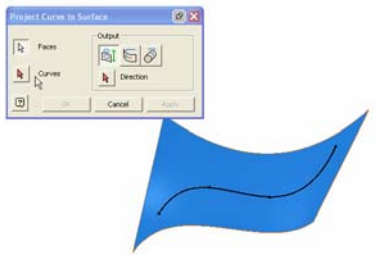
Step	Action	Result
23.	• Click **End of Part** and drag it to below **Work Plane 1.** 	End of Part shows here.  Screen shows shape. 
24.	• Select **Sketch 3D**. 	

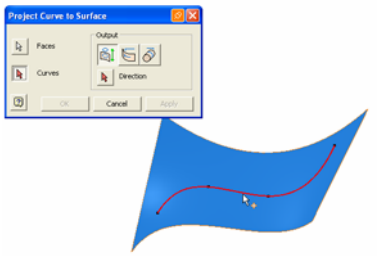
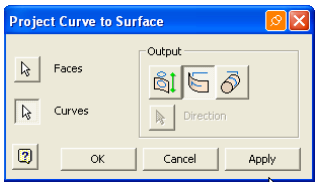
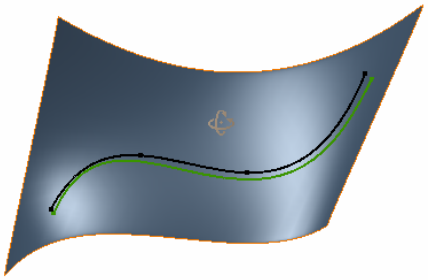
Step	Action	Result
25.	• Select **Project to Surface** from Panel. 	Project Curve to Surface dialog box appears. 
26.	• Select the cylindrical face. 	
27.	• Click the **Curves** icon from the dialog box 	
28.	• Select the three curves as shown. 	

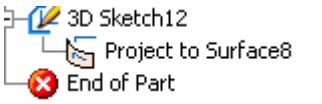
Step	Action	Result
29.	• Ensure the **Output as Project** along vector is selected from the dialog box  When selecting a 2d sketch, the projection direction is normal to the sketch plane by default. An alternative direction may be defined by selecting the **Direction** button, and then by selecting a work axis or linear edge. When selecting a 3d sketch, the projection direction must be defined by selecting a work axis or a linear edge.	
30.	• Select **Apply**. 	Curve is projected onto the entire cylinder, providing the “full” solution. 
31.	Repeat the same steps with a different set of curves.	

Step	Action	Result
32.	• Select the cylindrical face.  • Click the **Curves** icon from the dialog box and select the three curves as shown. 	
33.	• Select **Project Wrapped to surface** as Output. 	
34.	• Select **OK.**	Inventor will wrap only to cylindrical and conical faces, but not to spline-based surfaces. 

Step	Action	Result
35.	• Right-click **in the graphics area** > **Finish 3D Sketch** to leave this environment.	Part Features Panel appears.
36.	• Click and drag **End of Part** to below 3D Sketch8 in browser. 	End of Part shows here. 
37.	• Rotate and zoom until this surface is visible. 	
38.	• Rotate the model until it shows that curve is not laying on surface. This is imported data (for example) where curves and surfaces are not lining up as they should be. 	

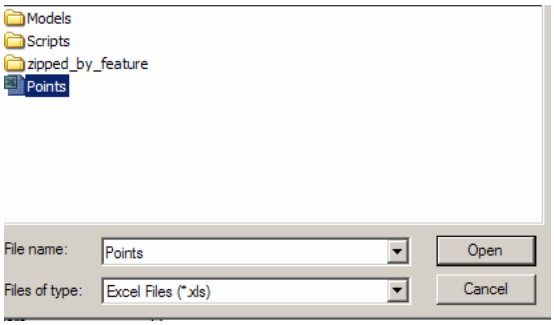
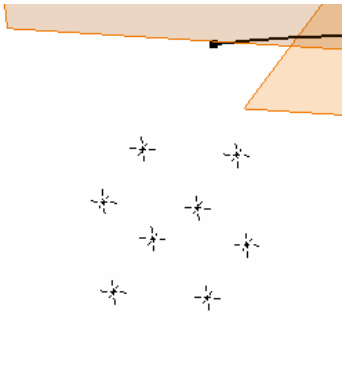
Step	Action	Result
39.	• Rotate the model back to this view. 	
40.	• Select Sketch > 3D Sketch. 	
41.	• Select **Project to Surface**. 	
42.	• With **Faces** selection active, select the surface. 	

Step	Action	Result
43.	• Click the **Curves** selection button from the dialog box, and then select the curve 	
44.	• Select **Project to Closest Point** so that it projects the closest point along the normal of that surface. 	
45.	• Select **OK**.	The curve is projected onto the surface based upon multiple vectors defined by the curve's proximity to the surface. The projection vectors are based upon a sampling of points along the curve. The result is a curve which has been translated the minimum distance required to allow it to lie upon the surface. 
46.	• Right-click on screen to select **Finish 3D Sketch**.	

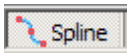
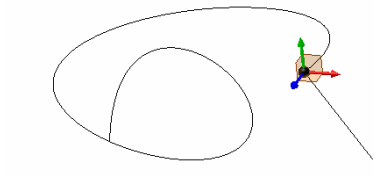
Step	Action	Result
47.	• Notice that projections live under 3D Sketch (in browser) so that they are editable. 	

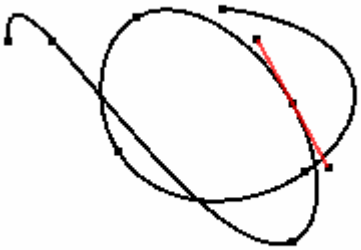
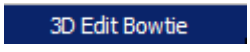
Task: Import Excel Data Points

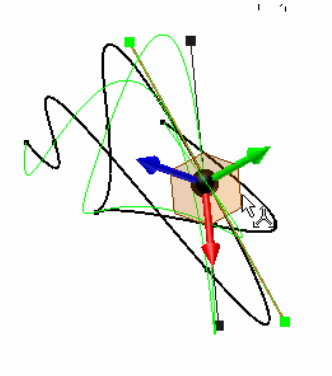
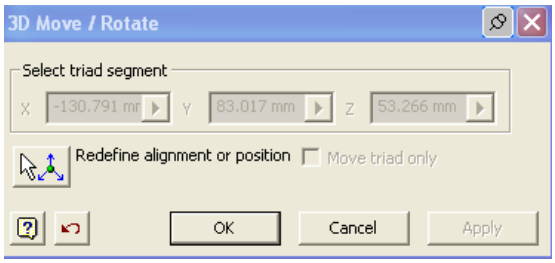
Step	Action	Result
48.	• Select **Sketch > 3D Sketch**.	
49.	Point data from Excel can now be imported into a 2d or 3d sketch. The Excel data must adhere to the following format: • Columns A, B, & C represent X, Y & Z coordinates, respectively. • Cell A1 is reserved to specify unit of measure. If this cell is left blank, Inventor will default to units of the active document. • Cells A2, B2, and C2 may be used for X, Y, and Z labels, respectively, or other strings (optional). mm x y z -200 0 0 -150 0 0 -150 50 0 -200 50 0 -200 0 60 -150 0 60 -150 50 60 -200 50 60	
50.	• Click **Import Points** from browser. 	

Step	Action	Result
51.	• On the browser, find the Excel document. 	
52.	• Select **Open**.	
53.	• Rotate and zoom until the points are shown, now that they are imported. • Notice the new 3D point object which is the same as the 2D Sketch center point except that it has three axes. Imported points maintain no associativity with the Excel document. Z data will be ignored when importing into a 2d sketch.	Points look like this: 

Task: Create and Edit a 3D Spline

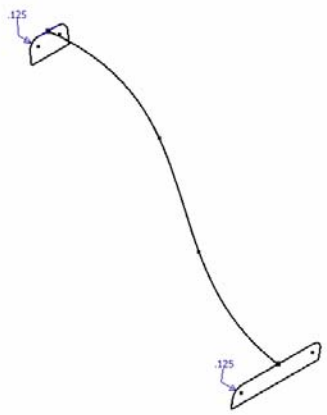
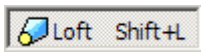
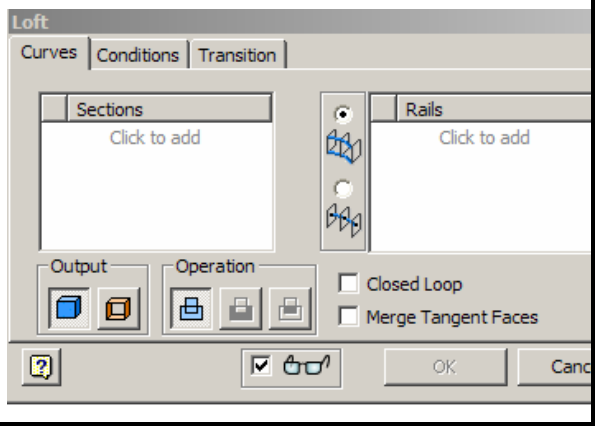
Step	Action	Result
54.	Create a 3D spline. • Select spline from 3D Sketch Panel. 	The Inventor Precise Input dialog box appears. 
55.	Create a 3D spline by selecting four points in the graphics window. 	
56.	When you've finished creating the spline, right-click and select **Create** to complete the spline.	A 3D spline is created. 

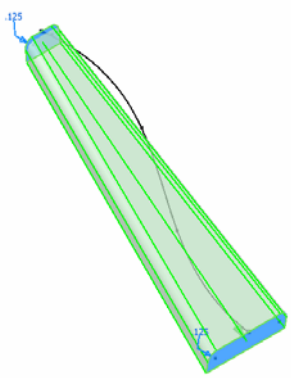
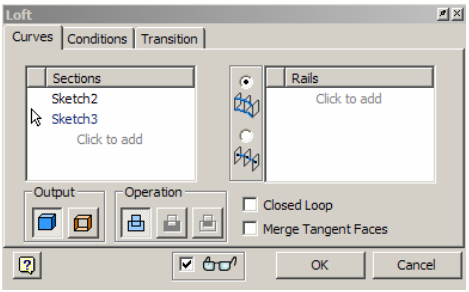
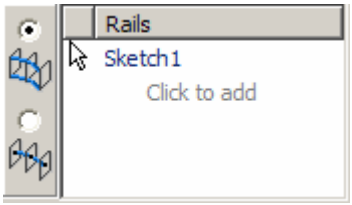
Step	Action	Result
57.	• Locate a control point on the spline. • Right-click the control point for the menu. • Select the **Bowtie** and **Handle** option to activate. 	Handle appears. This hasn't been in 3D before, only in 2D. 
58.	Change the magnitude of the spline handle: • Click on one of the ends of the bowtie handle and drag to see the spline change shape. 	
59.	Edit the direction/rotation of the spline handle: • Right-click on the handle point. • Select **3D Edit Bowtie**. 	

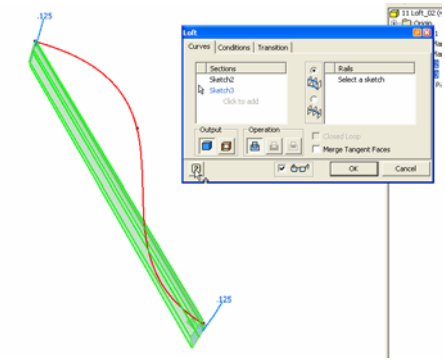
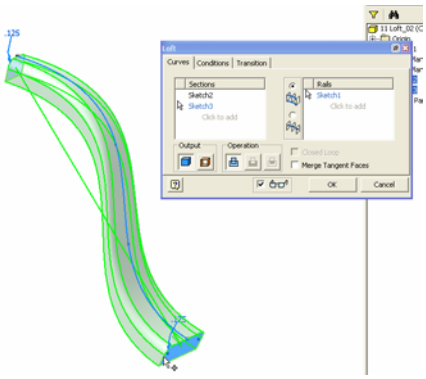
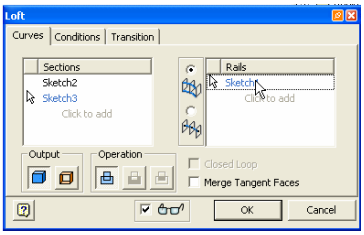
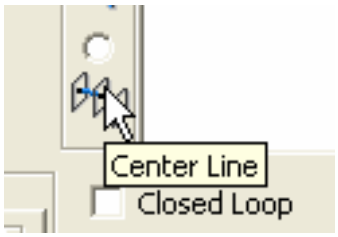
Step	Action	Result
60.	• Click on the 3D Rotate/Move Triad to move the position of the bowtie in 3D space or rotate the triad to modify the bowtie's position and influence over the spline. 	
61.	• Select **OK** from the 3D Move / Rotate dialog box when complete.	
62.	• Right-click on the graphics window for the menu. • Select **Finish 3D Sketch**.	

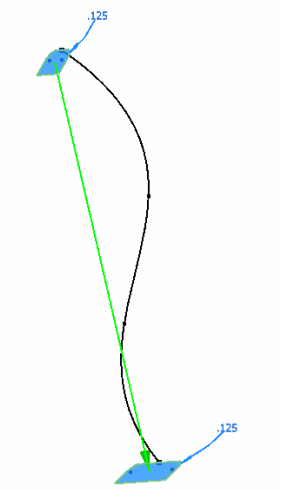
Exercise 2: Use Loft Enhancements

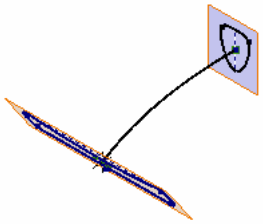
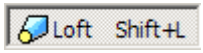
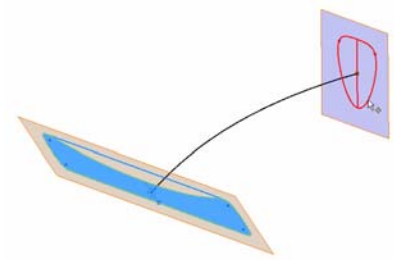
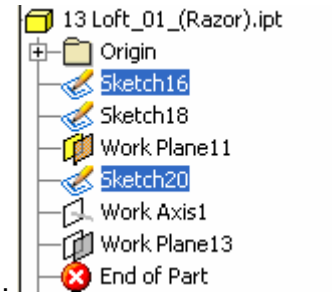
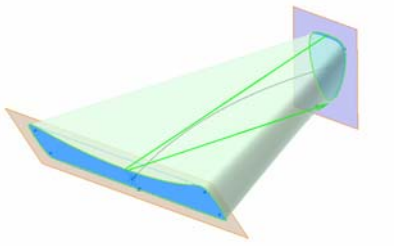
Task: Create a Center Line Loft

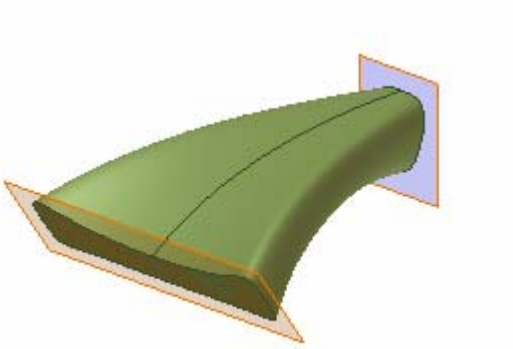
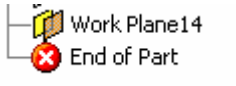
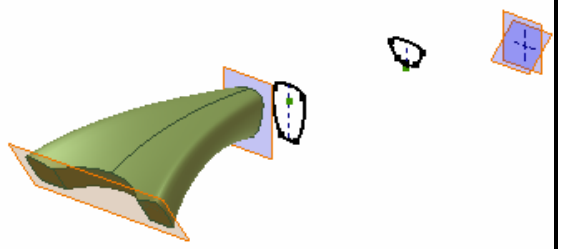
Step	Action	Result
63.	• Select **File Menu > Open > 11 Loft_02 (Centerline).ipt**.	Model appears. 
64.	• Select **Loft** from the Part Features Panel. 	

Step	Action	Result
65.	• Click the first sketch…  Followed by the second sketch.  	The Loft is now previewed.  Loft dialog box appears as follows:  Note the loft does not follow the guide rail and transitions directly from one profile to the other.
66.	• Under Rails, select **Click to add**.	Sketch 1 appears. 

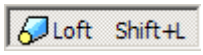
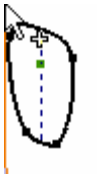
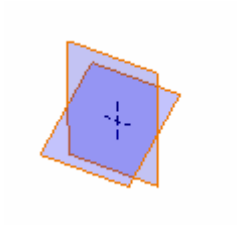
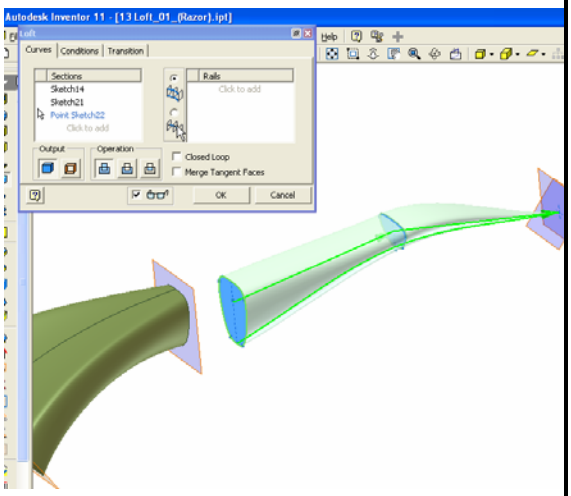
Step	Action	Result
67.	• Rotate the shape like this, and select the rail curve from the graphics window. 	The loft will now conform to the rail curve as shown below. Note how the loft becomes “thin” as it transitions along the rail. The sections are not held constant or normal to loft rails. 
68.	• Remove the initial rail selection from the Dialog box by highlighting sketch 1 from the rails section and pressing the Delete key.	
69.	• Select the **Center Line** option from the loft dialog box 	

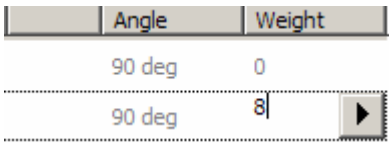
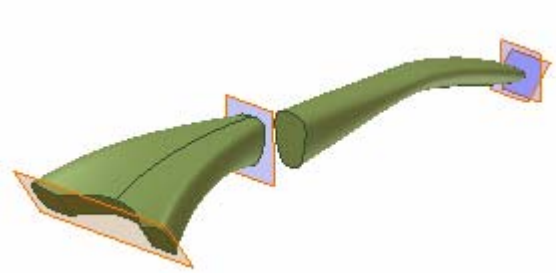
Step	Action	Result
70.	• Select the same loft rail as before  •	The new loft will be previewed as shown below. Note the difference between the resulting shape using Centerline vs. Rail. In a centerline loft, the sections are held in a constant relationship (in this case, normal) with the centerline, similar to a Sweep. The result is a better preservation of cross-sectional area between loft sections. 
71.	• Select **OK**.	Note the Final shape. 

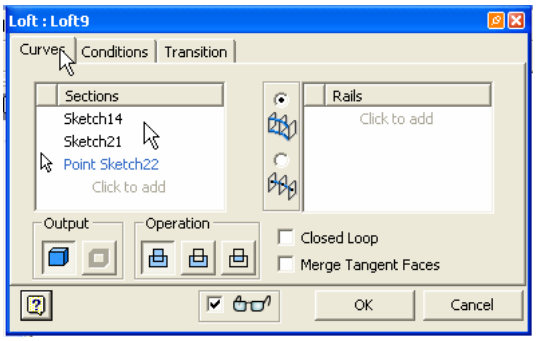
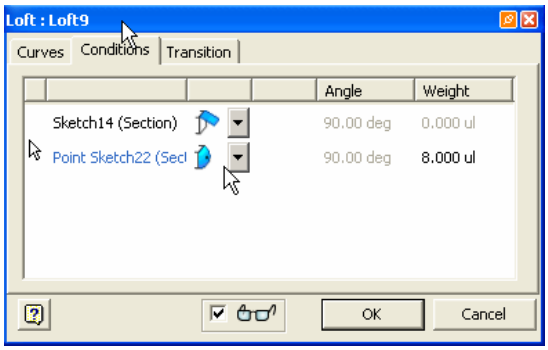
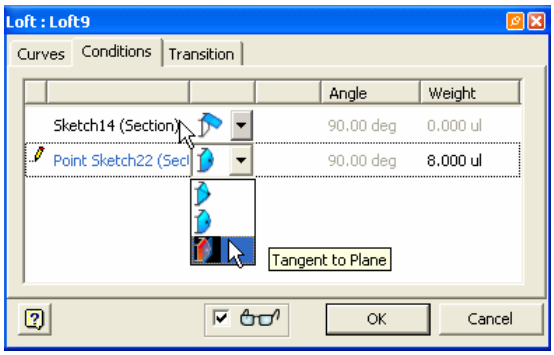
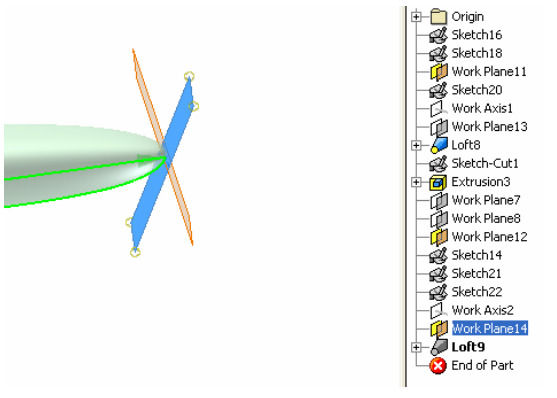
Step	Action	Result
72.	• Select **File** menu > **Open** > **13 Loft_01 (Razor).ipt**.	Model appears. 
73.	• Select **Loft** from the Part Features Panel**.** 	
74.	• Select sketch 20 and sketch 16 for the sections. 	
75.	• Select **Center Line Option**. 	The Loft is previewed 

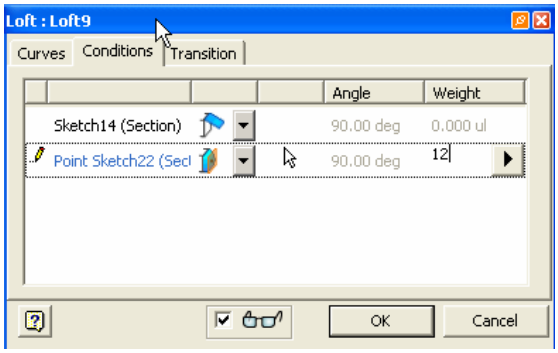
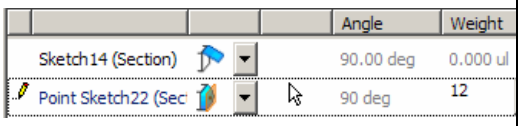
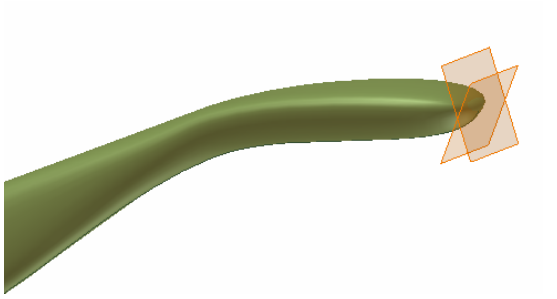
Step	Action	Result
76.	Select **OK**.	
77.	Move the End of Part marker to underneath Work Plane14. 	The remaining sketches and workplanes appear. 

Task: Create a Loft to a Sharp or Round Point

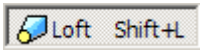
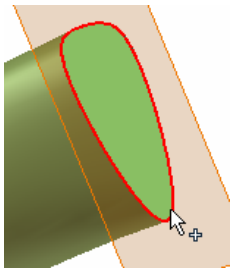
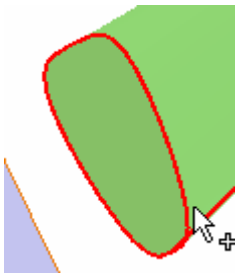
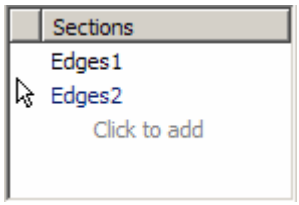
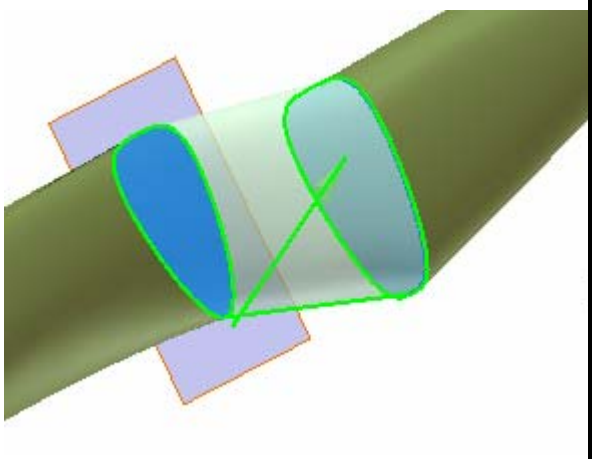
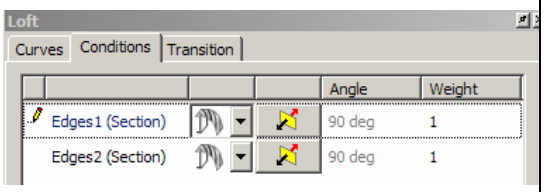
Step	Action	Result
78.	• Select **t**he **Loft** command from the tool pallet.  • Add the following sketches to the sections portion of the dialog box: Sketch 14  Sketch 21  and Point Sketch 22 	

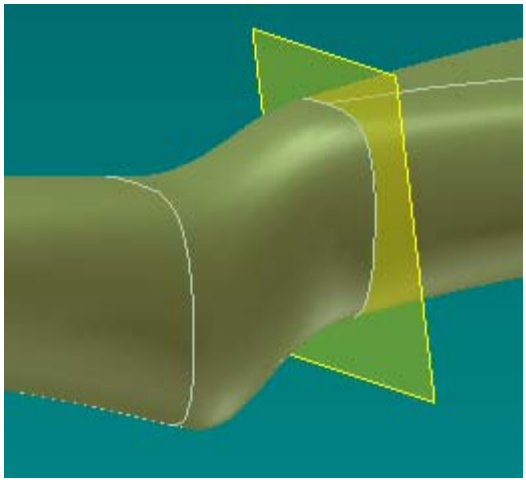
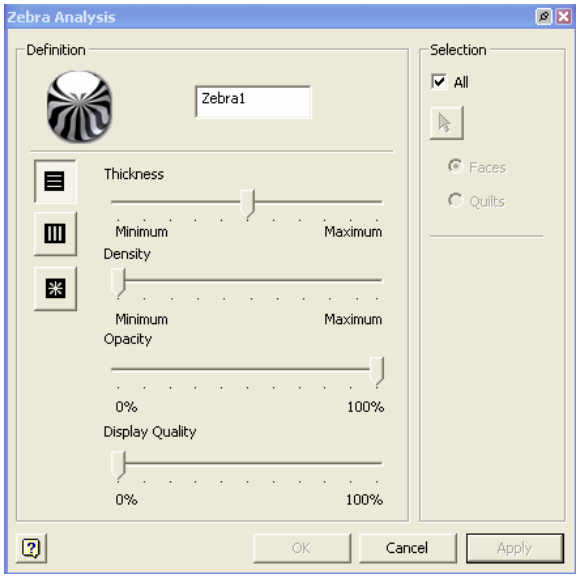
Step	Action	Result
79.	• Select the **Conditions Tab** from the Loft dialog box box. • Notice types of conditions possible : sharp, tangent, tangent to plane  • Select the **Tangent option for Point Sketch 22 (Section)**	• Note Sketch 14 (section) and Point Sketch 22 (Section) are already listed. Conditions apply to the start and end sections.
80.	Add a weight to see it better exaggerate the tangency influence over the end condition of the loft. • Enter **8** under Weight for Point Sketch 22. 	
81.	• Select **OK**.	

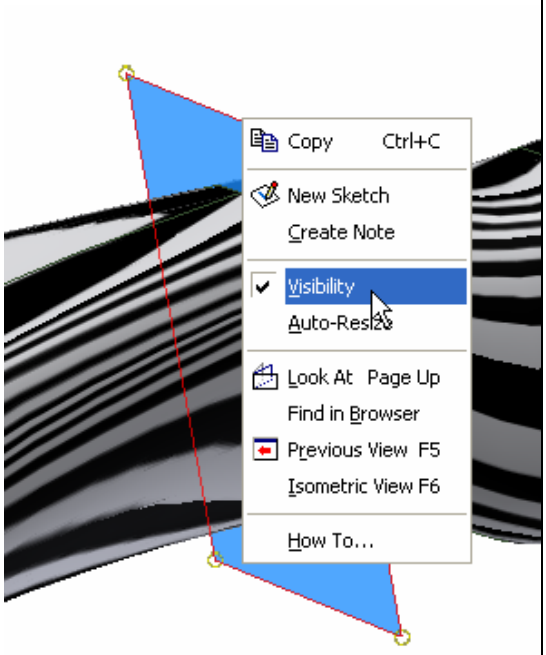
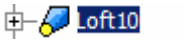
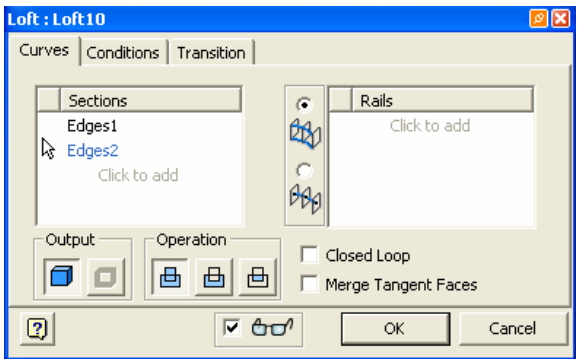
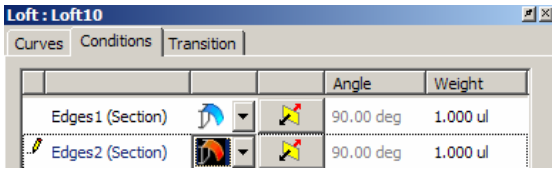
Step	Action	Result
82.	**Loft Editing** Right-click **Loft 9** from the Browser. • Loft9 • Select **Edit Feature**. Edit Feature	The Loft Dialog box appears 
83.	Select the **Conditions** tab.	
84.	Select the **Tangent to Plane** for Point Sketch 22.	
85.	The Tangent to Plane option allows the selection of a work plane to better control the shape of the loft. Select **WorkPlane 14** from the graphics window.	

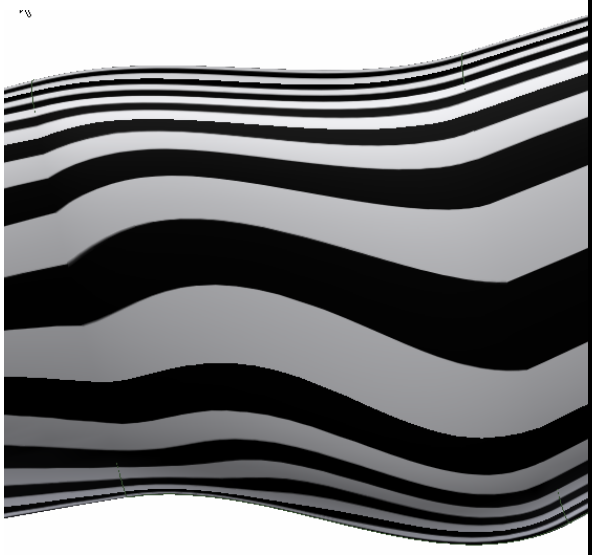
Step	Action	Result
86.	Adjust the weight for Point Sketch 22 to **12.**	
87.	• Select **OK**. Enter weight to 12. • 	
88.	• Select **OK**.	Note how the final shape changes. The resulting loft passes through the selected point section, and is tangent to the selected plane. 

Task: Create a Smooth G2 Loft

Step	Action	Result
89.	• Select **Loft** from the Feature Pallet.  • Select this edge…  and this edge. 	 
90.	• Select the **Conditions** tab from the Loft dialog box.	
91.	• Select **Smooth (G2) Condition** for both Edges 1 and 2**.** 	
92.	• Select **OK**.	

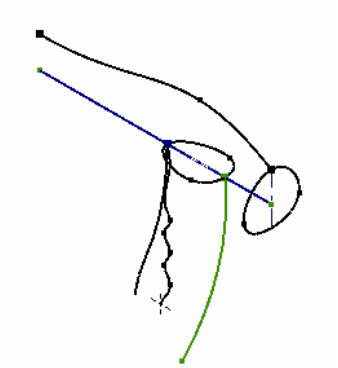
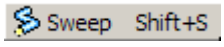
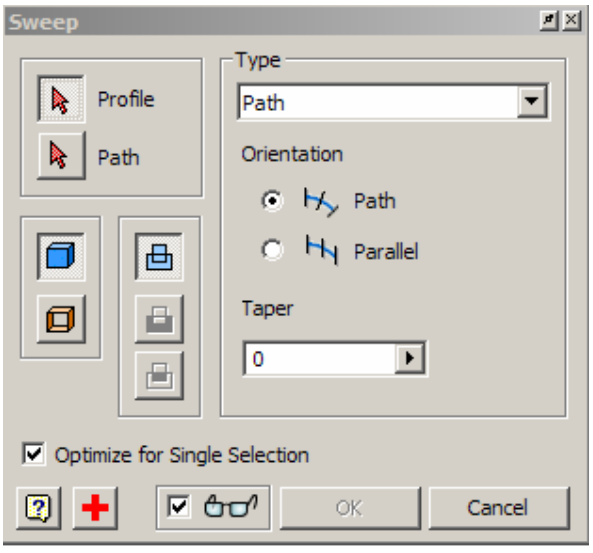
Step	Action	Result
93.	• Notice Smooth Condition of area.	
94.	• Turn on the Zebra striping so that the effects of the G2 transitions on the Loft can be seen. • Click the **Zebra Analysis** icon in the main menu bar. 	

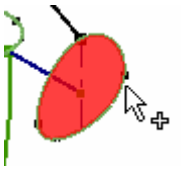
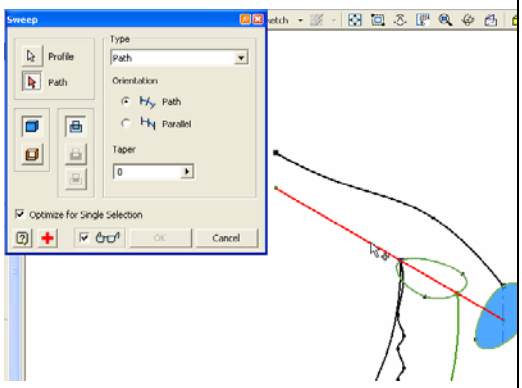
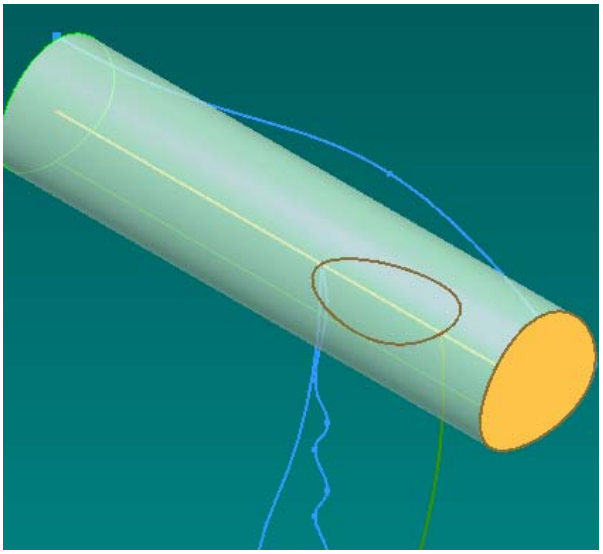
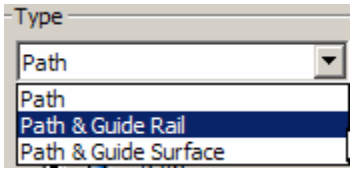
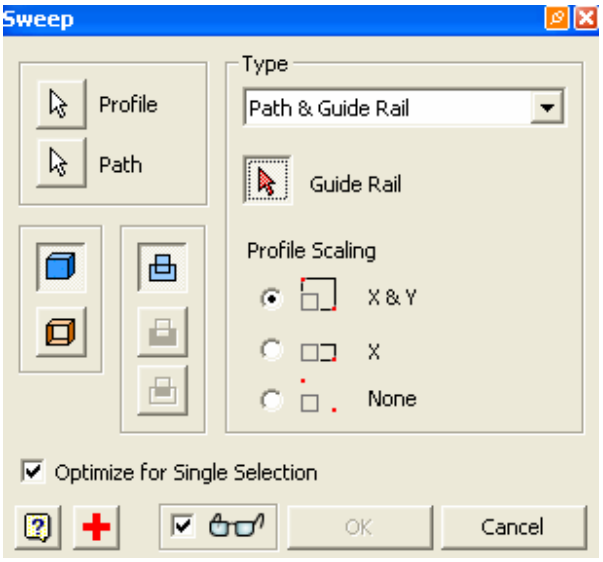
Step	Action	Result
95.	Turn the visibility of the Work Plane off to better investigate the Zebra Analysis. Note that the zebra stripes maintain their width as they pass over model edges, and appear smooth – this indicates a Smooth or G2 loft. 	
96.	Edit the loft. • Right-click **Loft10** from the browser and select **Edit Feature**. 	
97.	• Change both conditions to **Tangent** to see the difference in the resulting loft.	
98.	• Select **OK**.	

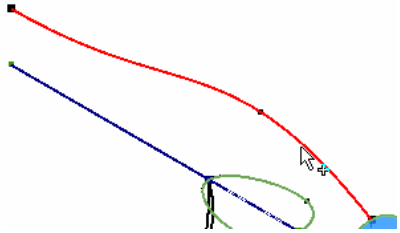
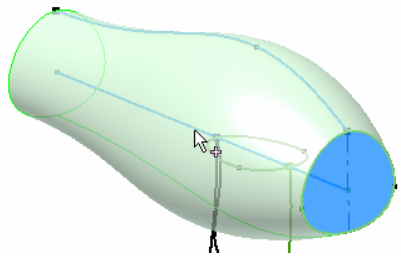
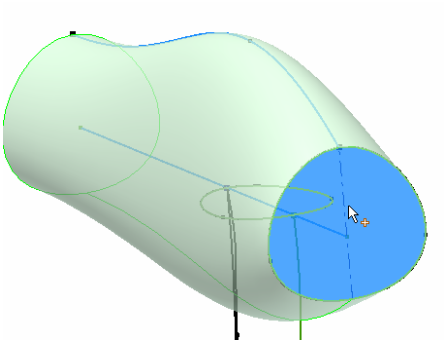
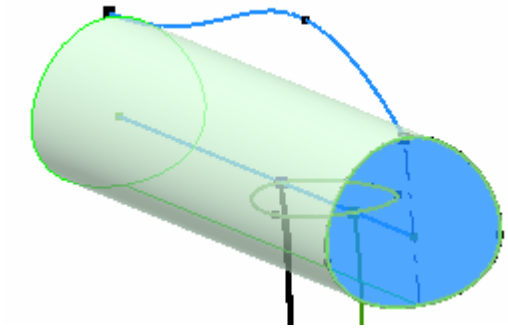
Step	Action	Result
99.	• Zoom in close-up. • Notice the differences in the line smoothness. The zebra stripes still have the same width as they pass over model edges, but don't appear continuous – this indicates a tangent loft. • Select **Undo** and see the difference back to G2. 	
100.	• Select **File menu > Close** (and close any other open files.).	

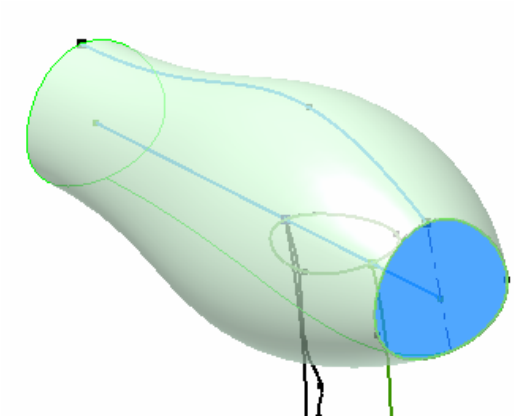
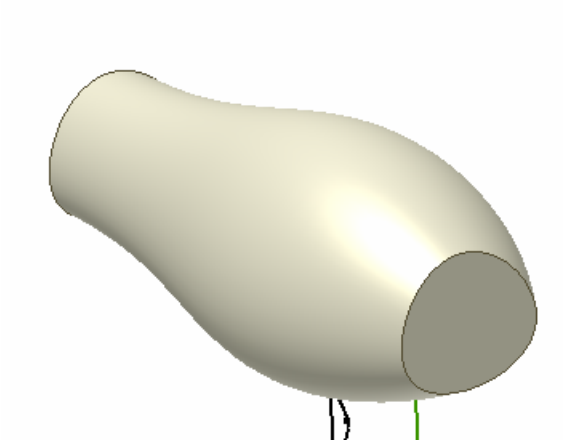
Exercise 3: Use Sweep Enhancements

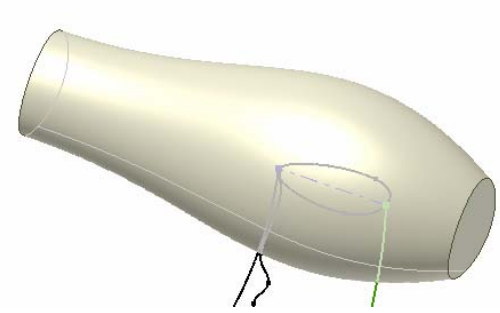
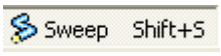
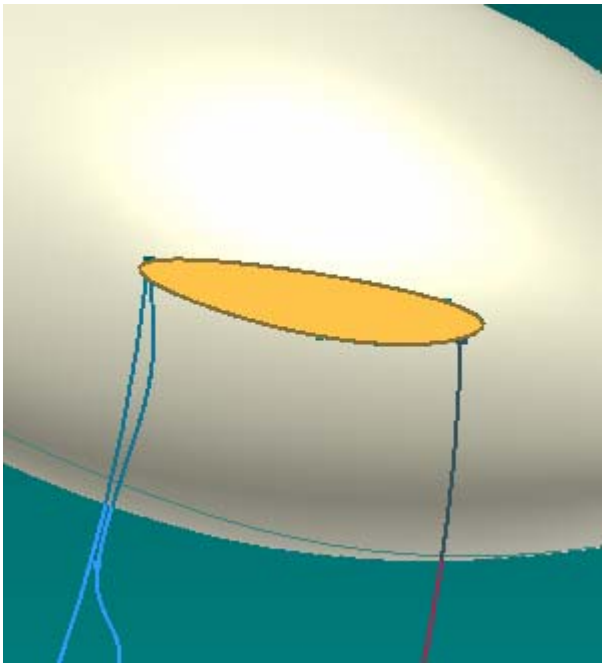
Task: Use Guide Rail Sweep

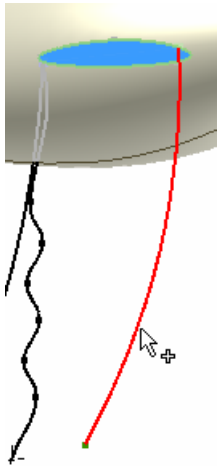
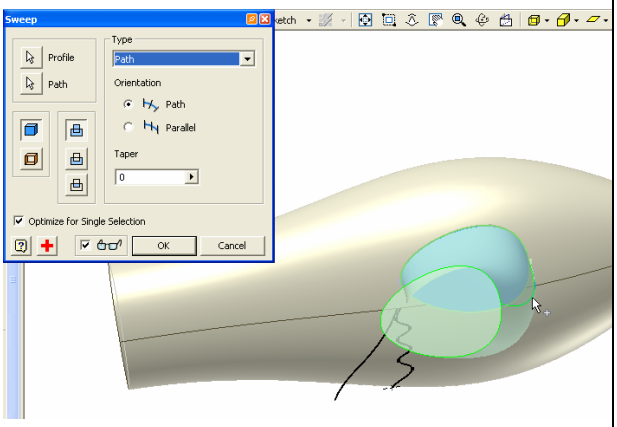
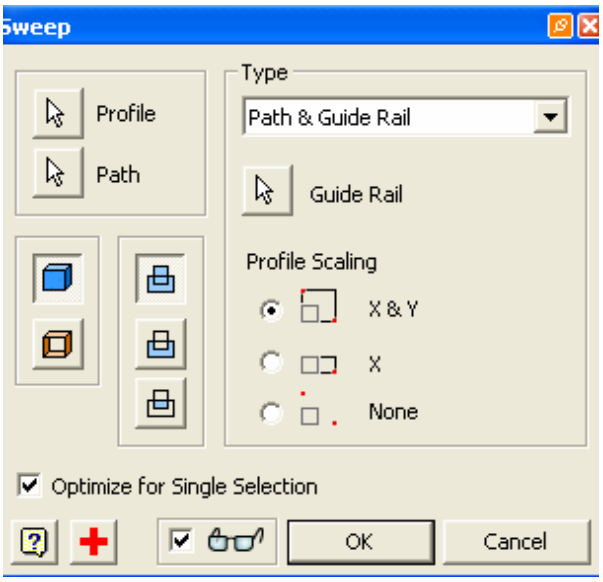
Step	Action	Result
101.	• Select **File** menu > **Open** > **14 Sweep_01 (Hairdryer).ipt**	
102.	• Select **Open**.	Model appears. 
103.	• Click **Sweep** in the Part Features Panel.  Notice that dialog has changed considerably from previous releases.	

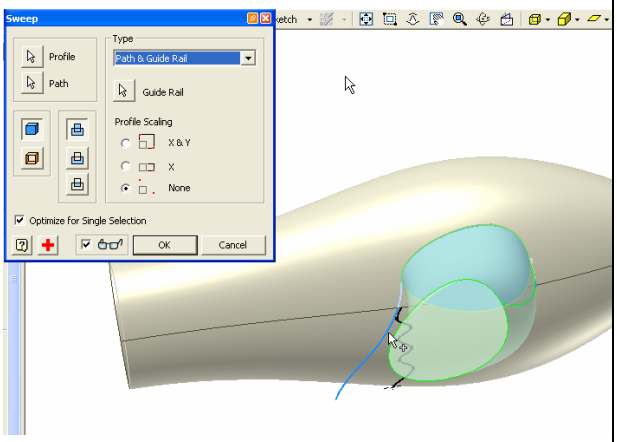
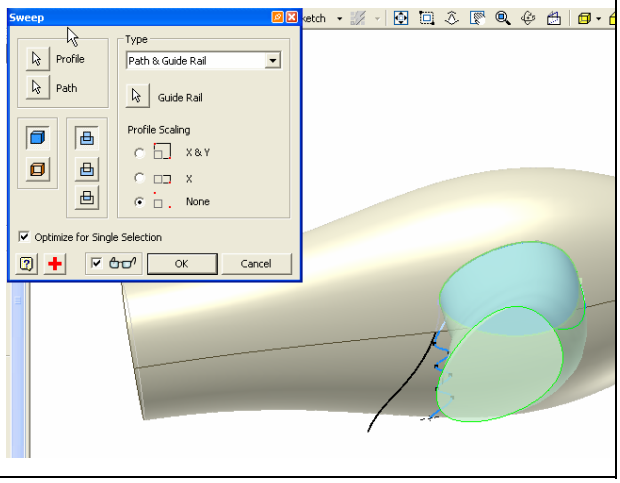
Step	Action	Result
104.	• Click **Closed Profile** as shown. 	
105.	• Select the line shown for the Sweep path. 	
106.	• Select **Path & Guide Rail** option.  Note how the dialog box changes depending on the sweep type.	

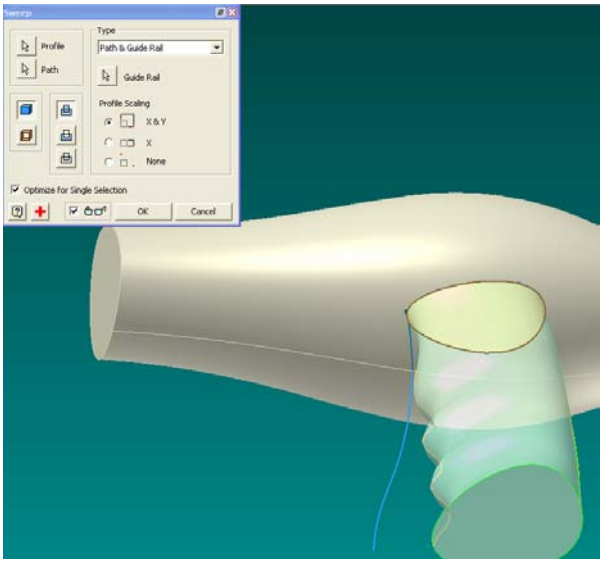
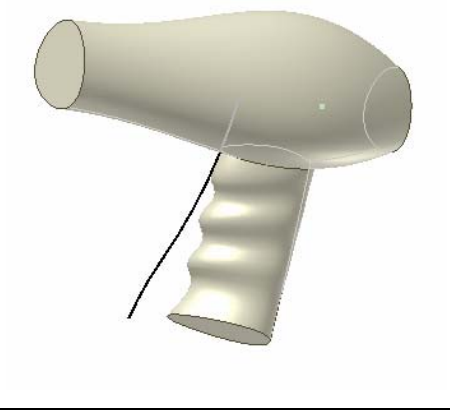
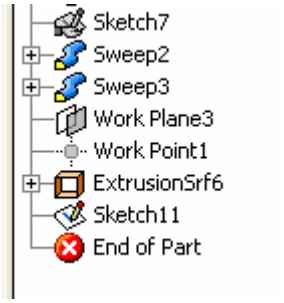
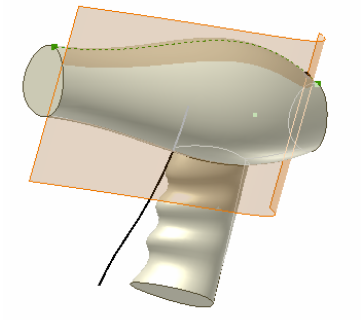
Step	Action	Result
107.	Select curved path as shown.  The selection of the second (curved line) for a rail will control both the scaling and twist of the profile along the previously selected path.	The Profile is swept along the path (center), but is scaling to meet the rail (top). 
108.	• Rotate the object to view the axes. • Notice Profile Scaling. Default is X & Y, but it can be changed to X so that the height scales, but the width doesn't. • Select the **X only** option • The X and Y directions are established where the profile meets the rail and where the profile meets the path.	
109.	• Select **None for Path** to experiment. This means only twist is being controlled as the profile is swept along the path. (**Note**: in this example, the rail does not introduce twist.)	

Step	Action	Result
110.	• Select **X & Y Scaling** again to achieve the desired shape. 	
111.	• Select **OK**.	
112.	Select **Edge Display > Hidden Edge** from main menu. 	

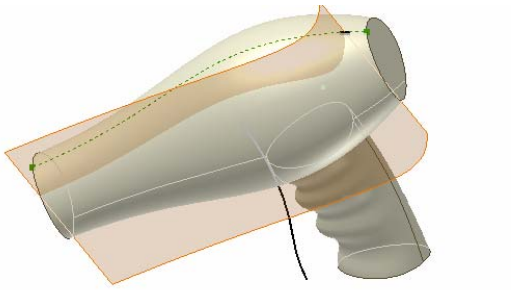
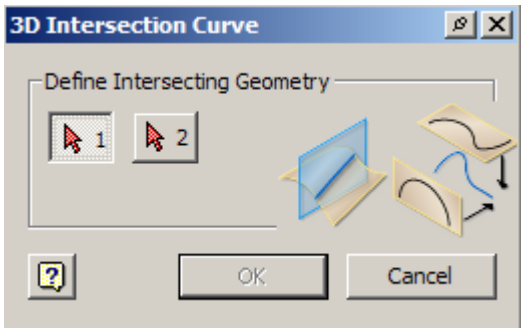
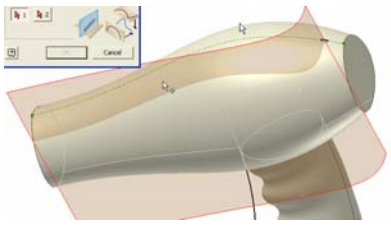
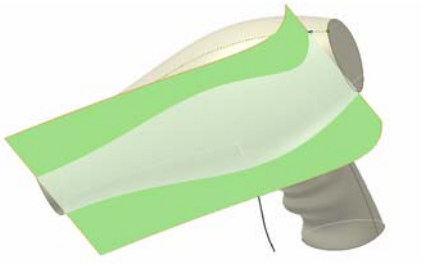
Step	Action	Result
113.	• Rotate as shown. 	
114.	• Select **Sweep**.  • Notice the Profile is automatically selected.	

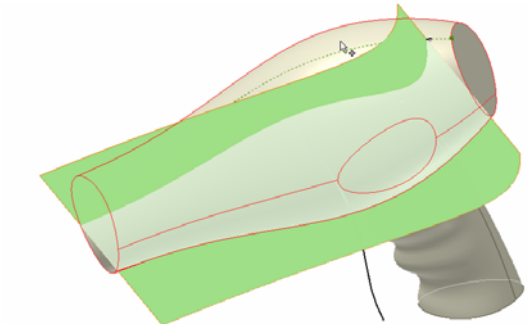
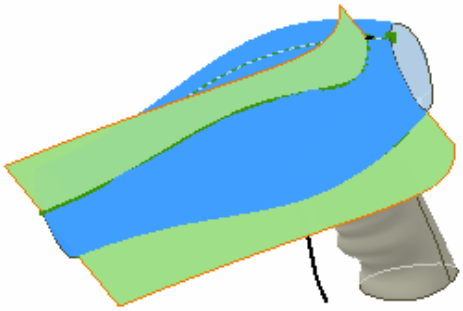
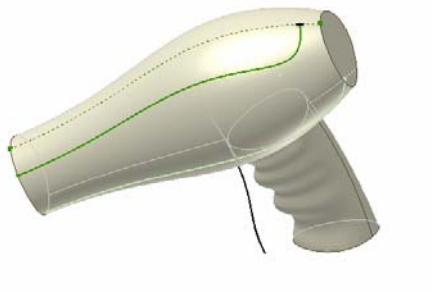
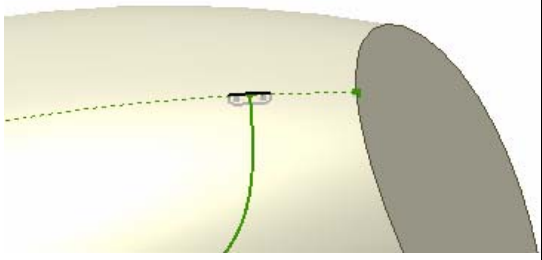
Step	Action	Result
115.	• Select the **Path** type. • Select the line as shown.  •	Note how the profile follows the path along with no influence from any rail. 
116.	• Select **Path & Guide Rail** type.	

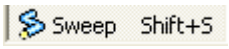
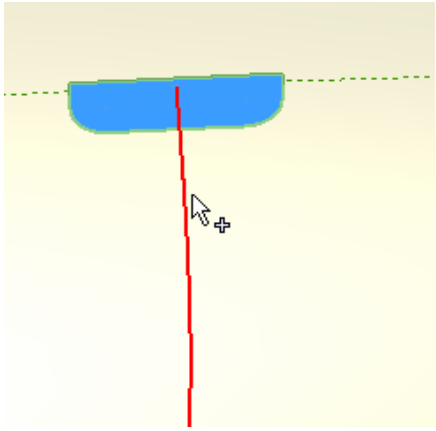
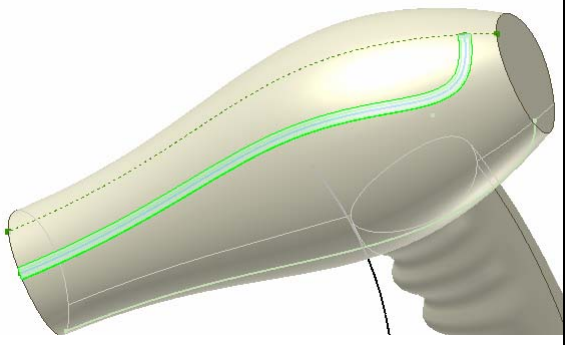
Step	Action	Result
117.	• Select the line as shown highlighted in **blue**. Note how the handle changes in twist but not scale or shape when the **None** scaling option is selected with this path selection. In this case the path is controlling the twist of the profile along the path.	
118.	• Select the **Guide Rail** icon from the Sweep dialog box. • Press Ctrl & to select the rail used in the previous step to unselect it, and then select the curved line.	Note the newly selected path is highlighted in blue. 

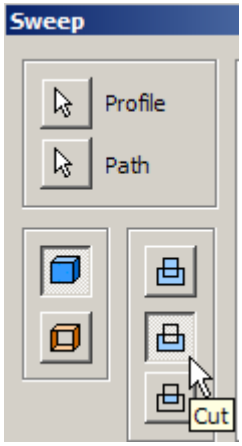
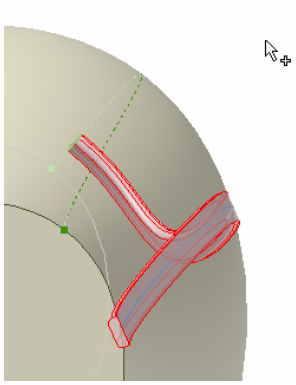
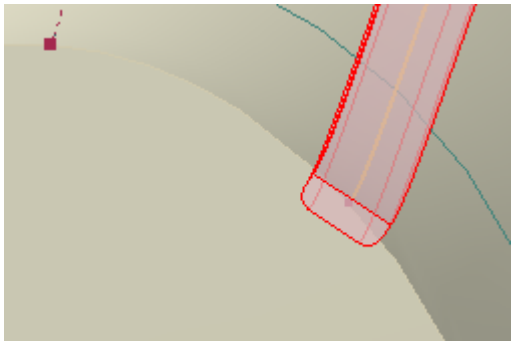
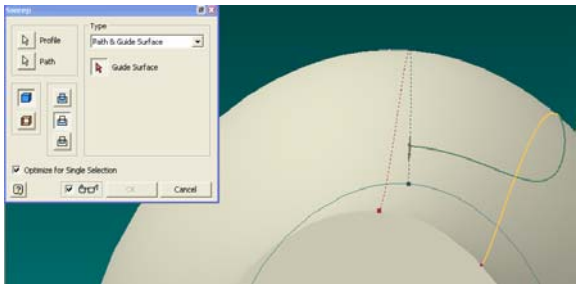
Step	Action	Result
119.	• Select **X & Y Scaling** to enable a nicely grooved handle.	
120.	• Select **OK**. • Rotate and zoom the hairdryer to get a good look.	
121.	• Click **End of Part Marker** and drag to underneath Sketch 11. 	

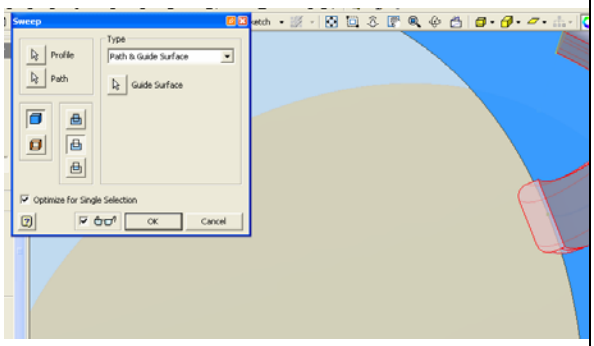
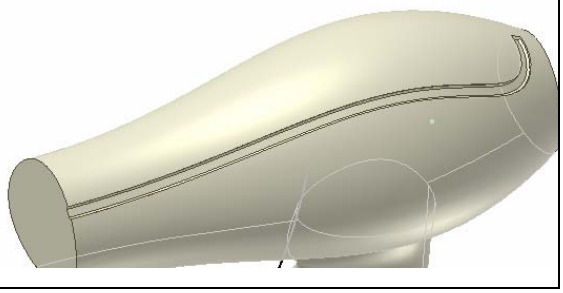
Task: Guide Surface Sweep

Step	Action	Result
122.	Next we need to add a detail stripe to the hairdryer. To begin, we will create a path using some 3D Sketch tools. • Select **Sketch** > **3D Sketch** from main menu.	
123.	• Rotate and zoom hairdryer into this orientation.	
124.	• Select **3D Intersection Curve**. 	
125.	• For the first selection set, click to select the surface. 	

Step	Action	Result
126.	• For the second selection set, click the body to select. • Click **OK** to accept command. 	
127.	• Right-click on screen for menu. • Select Finish 3D Sketch.	
128.	• In browser, right-click **ExtrusionSrf6** > **Visibility** to disable the visibility of the surface.  •	
129.	• Zoom in on profile. 	

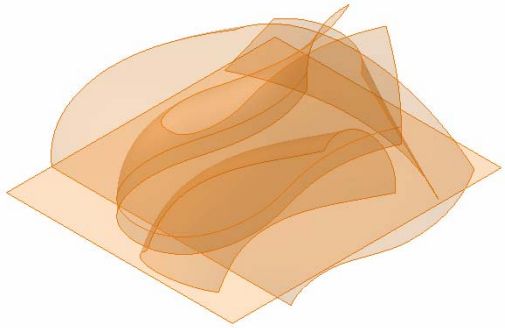
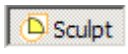
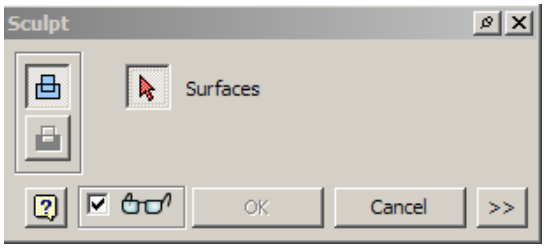
Step	Action	Result
130.	• Sweep the profile. • Select the **Sweep** command. •  • The **Profile** is automatically selected.	
131.	• Click **Path**. • Select the spline created from the intersection command. 	Preview occurs and shows final. 

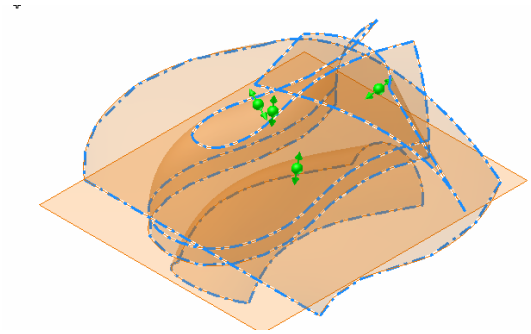
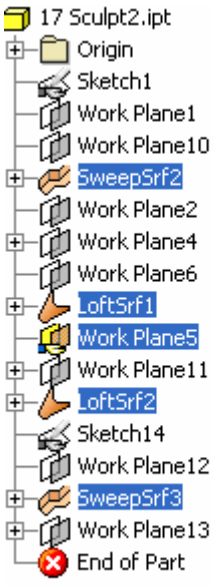
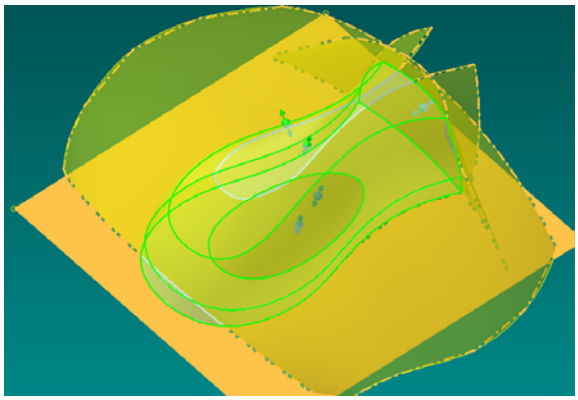
Step	Action	Result
132.	Change the operation type to Cut. • Select the **Cut**. option from the dialog box. 	 **Note**: This is the same type of sweep that was generated in R10. Up until R11, the orientation of the profile was controlled by the path, but the twist of the profile could not be controlled; thus if there was a 3D path, only circles could be swept accurately, because it doesn't matter if they spin on the path.
133.	• Zoom in on the edge of the cut. Notice that the cut loses its alignment with the surface as it is swept because profile twist is not controlled.	
134.	• Select **Path & Guide Surface** as the sweep type.	

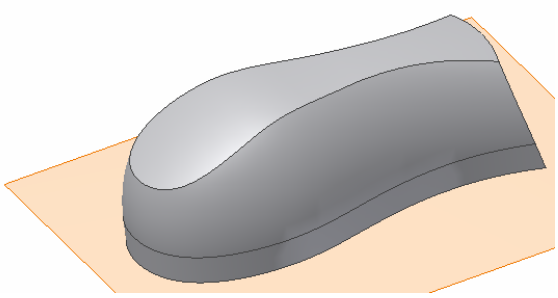
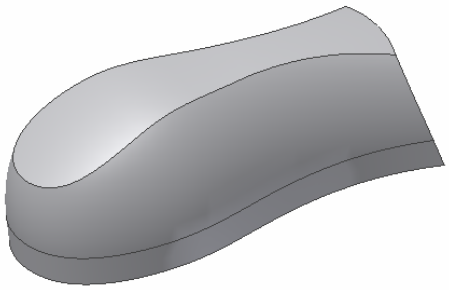
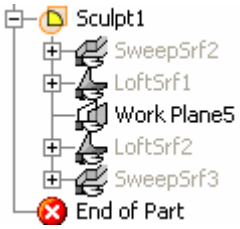
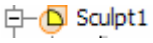
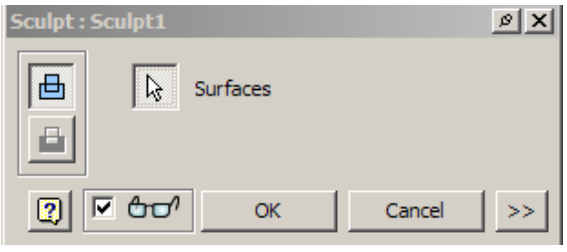
Step	Action	Result
135.	• Click the surface area and wait for preview, a computationally intense process.	 Note how the twist of the profile is now controlled by the surface.
136.	Select **OK** from the dialog box. Mirror the sweep to complete the detail stripe (optional).	Finished sweep. 

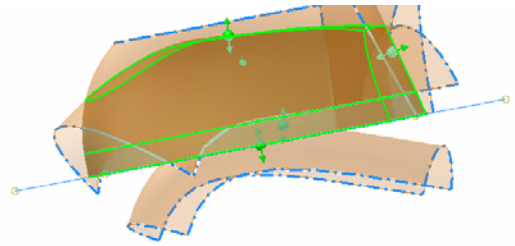
Exercise 4: Use Sculpt Features

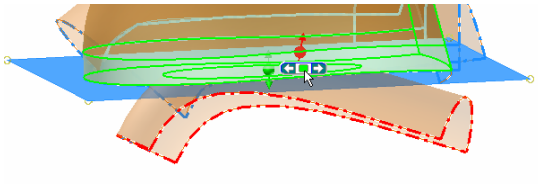
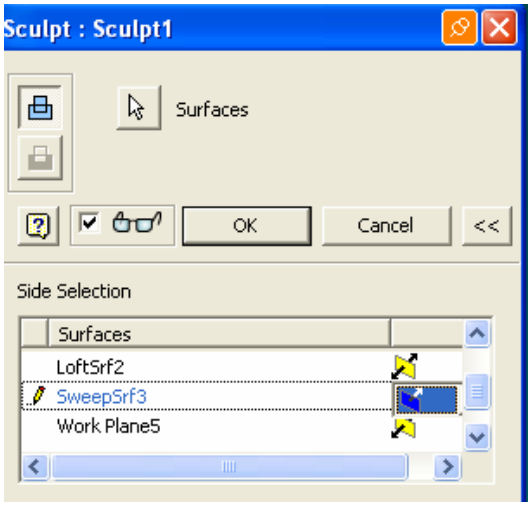
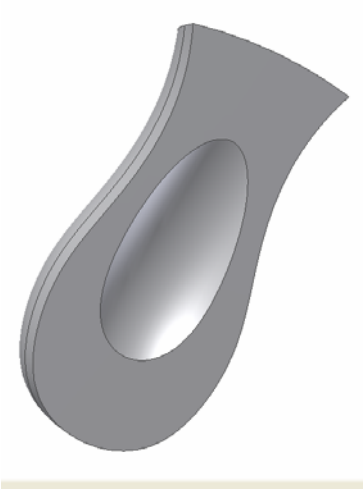
Task: Use the Sculpt Feature

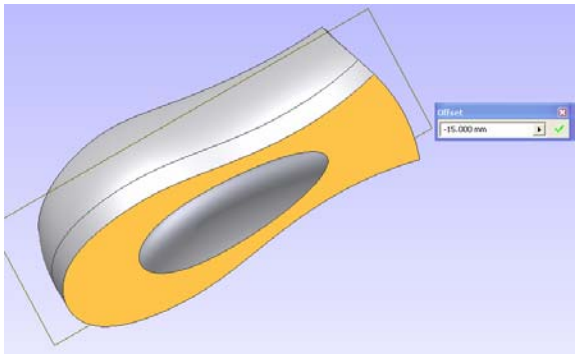
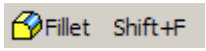
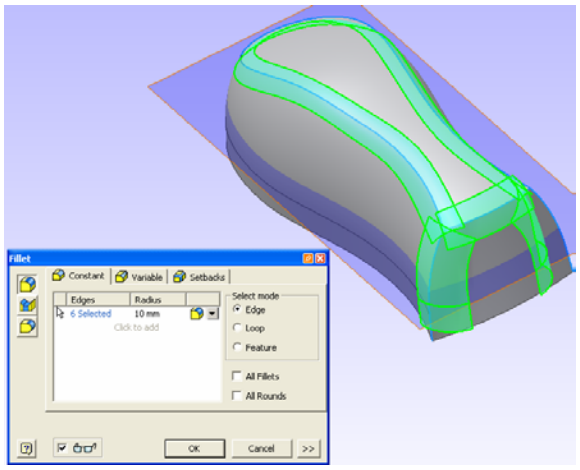
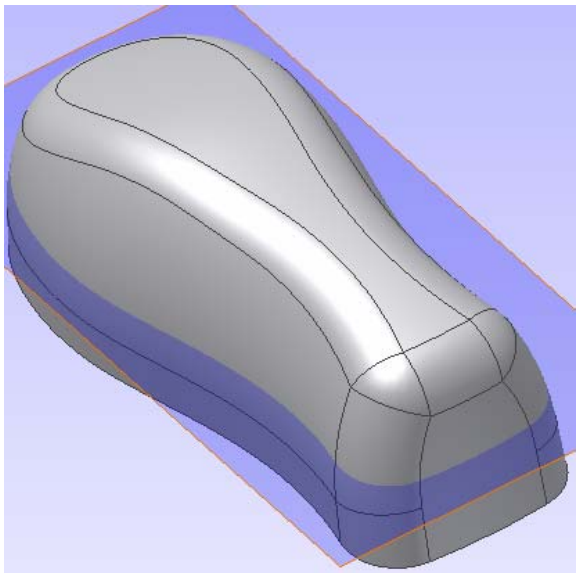
Step	Action	Result
137.	• Select **File** menu > **Open** > **17 Sculpt2.ipt.**	
138.	• Select **Open**. The new sculpt command enables you to add or remove material from untrimmed, unstitched surfaces.	Model appears. 
139.	• Click **Sculpt** in the Part Features Panel. 	

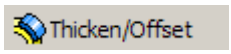
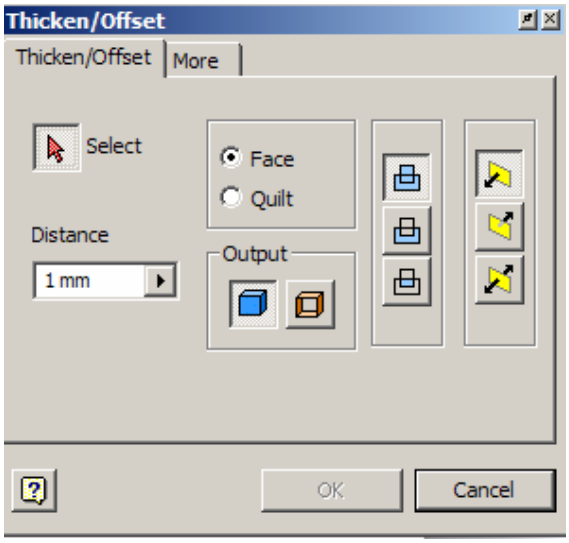
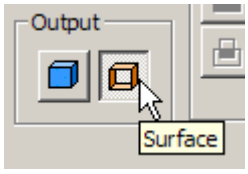
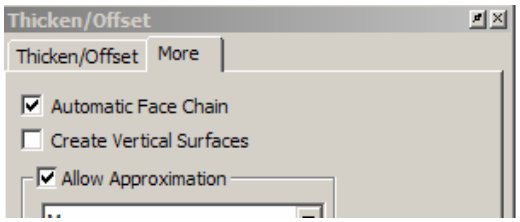
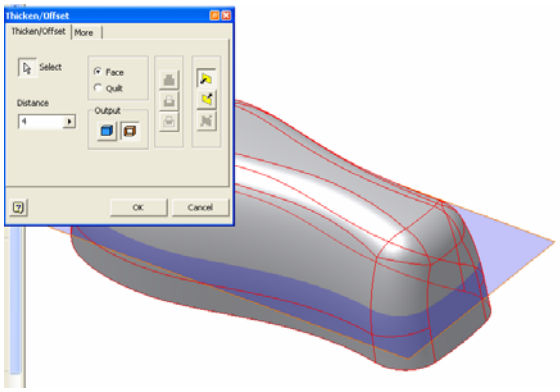
Step	Action	Result
140.	• Click to select surfaces. Rotate if necessary to get surfaces underneath.  	Once you've selected enough surfaces to define a closed volume, the preview demonstrates the resulting body.  Notice the glyphs that appear on each surface, indicating the side of the surface to be filled with material. By default, the arrow points both directions from each surface. As the feature knows to fill only closed volumes (and will not fill infinite space), this default option will likely provide the desired results.

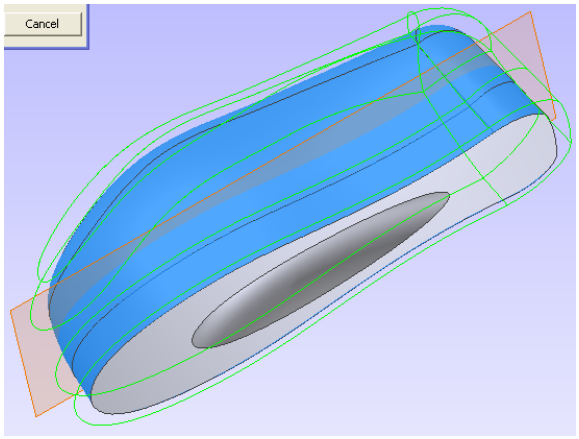
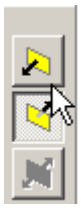
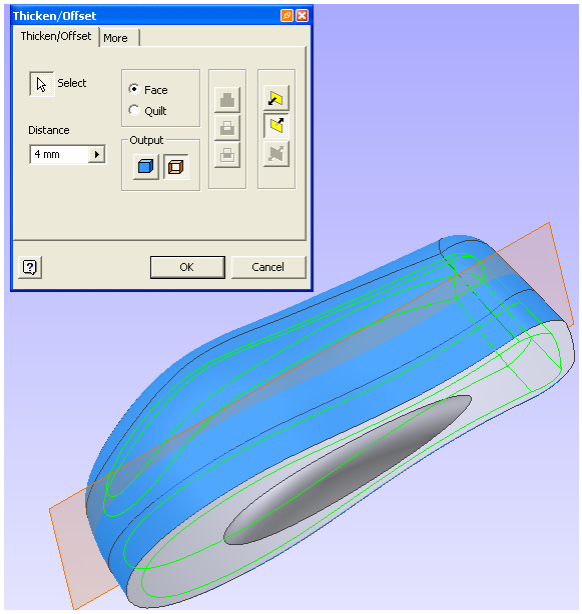
Step	Action	Result
141.	• Select **OK**.	
142.	• To turn off visibility of the work plane, right-click **Work Plane5 > Visibility**. • 	
143.	• Notice the new dependency-based browser. Sculpt is now consuming all the items below. These are actually being shared and are shown in the browser the same way sketches are done.	
144.	Edit sculpt shape by taking a bottom part that is not needed off of the shape. • In the browser, right-click **Sculpt1** > **Edit Feature**. 	

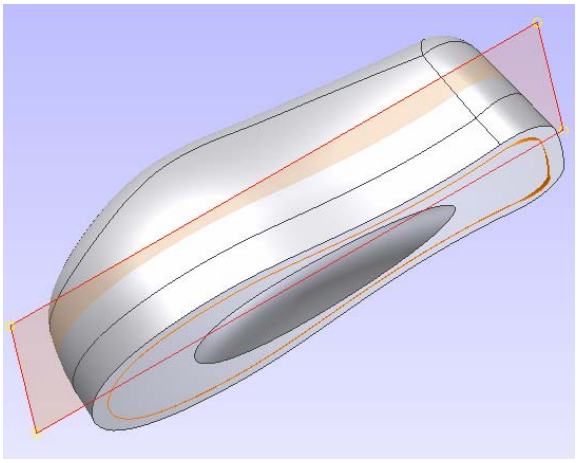
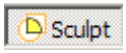
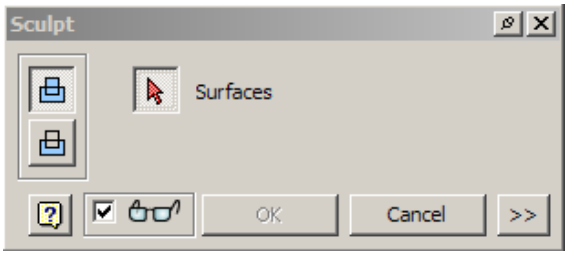
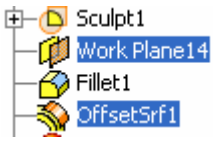
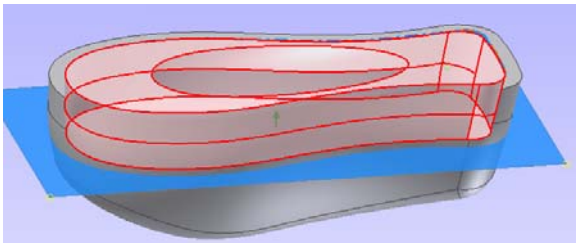
Step	Action	Result
145.	• Rotate shape to see underside. 	

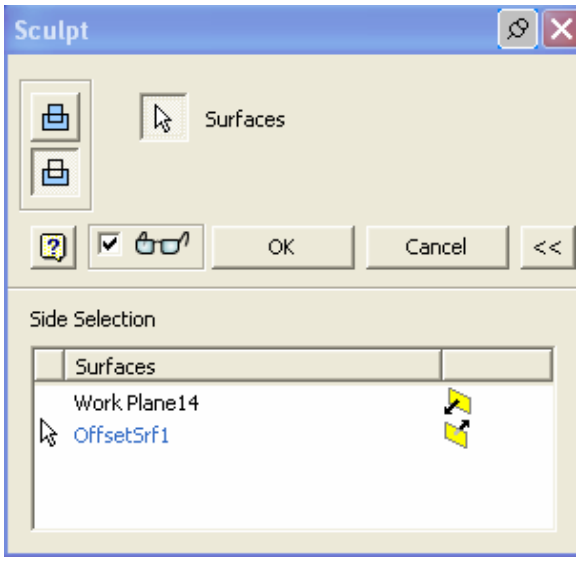
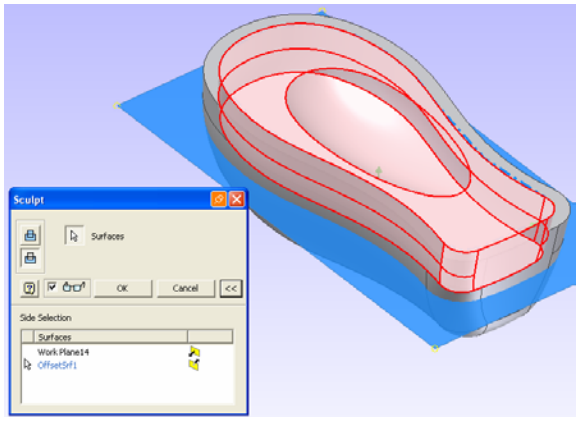
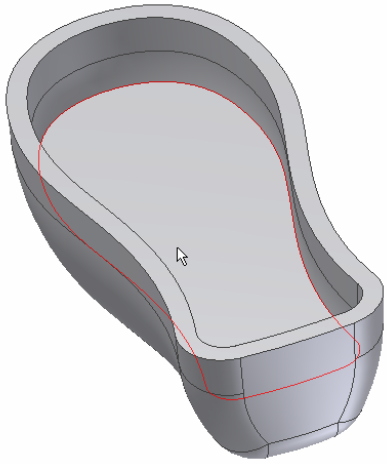
Step	Action	Result
146.	• Notice that SweepSRF3 has had no effect on the Sculpt feature. This is because, by default, we've specified to add material to both sides. • Select either the glyph that corresponds with SweepSRF3 to specify a different material direction…  or • Select **>>** to expand the dialog box, locate SweepSRF3 in the list control, and specify a different material direction using the direction drop-down menu. **Note**: Modifying directions in the dialog box provides the same results as modifying directions using the glyphs in the graphics area.  Select OK when done.	Result: T 

Step	Action	Result
147.	• Select to create a **Work Plane**. 	
148.	• Select Bottom Face of Part. • Create an Offset Work Plane. • Enter **-15 mm** for Offset.	
149.	Create a 10 mm Fillet. • Click **Fillet** from Panel.  • Select **Edges** as shown.	
150.	• Select **OK**.	

Step	Action	Result
151.	• Select **Thicken/Offset** from Panel. 	
152.	• Select **Surface**. 	
153.	• Click **More** > **Automatic Face Chain**. 	
154.	• Return to the **Thicken/Offset** tab and depress the **Select** Icon. Now select the outer face of the part. Note how all the surfaces are selected.	

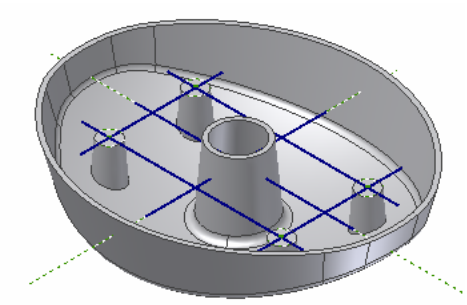
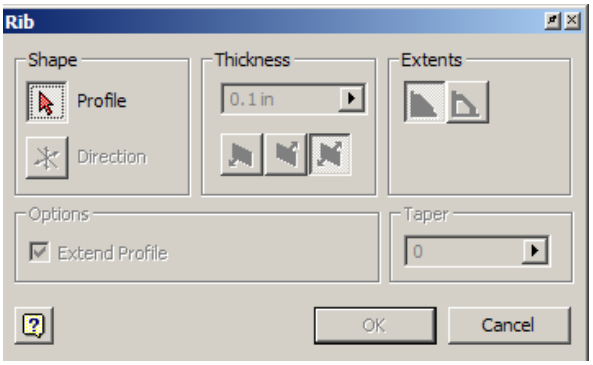
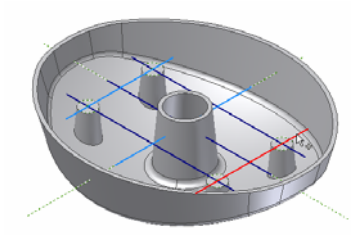
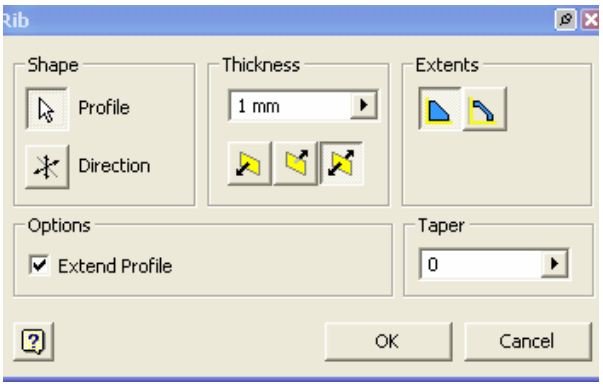
Step	Action	Result
155.	• Rotate part to this orientation. • Take notice of the **offset.**	
156.	• Modify the offset direction to inside the part 	
157.	• Set **Distance** to **4 mm**.	

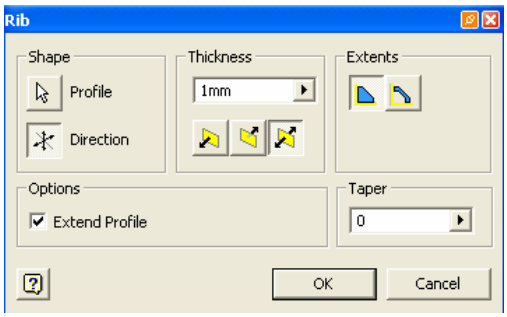
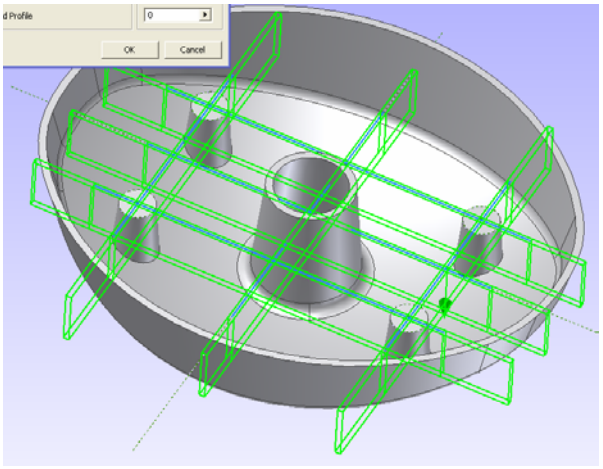
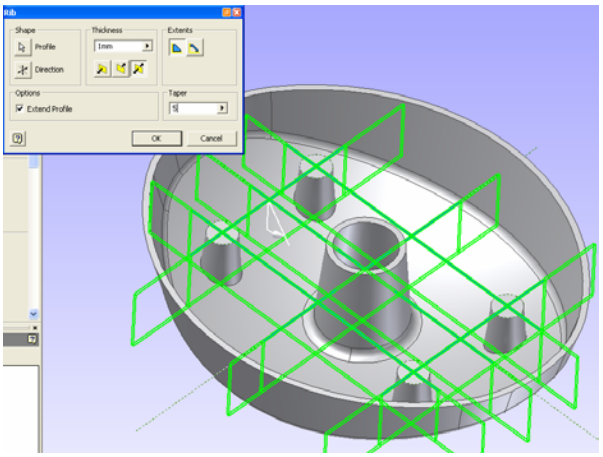
Step	Action	Result
158.	• Select **OK**.	
159.	Return to **Sculpt** feature. • Click **Sculpt**. 	
160.	• Select **Remove**. 	
161.	• Select **Work Plane 14** to remove and… • Select **OffsetSrf1** 	

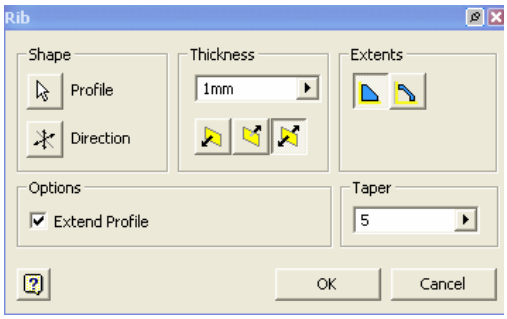
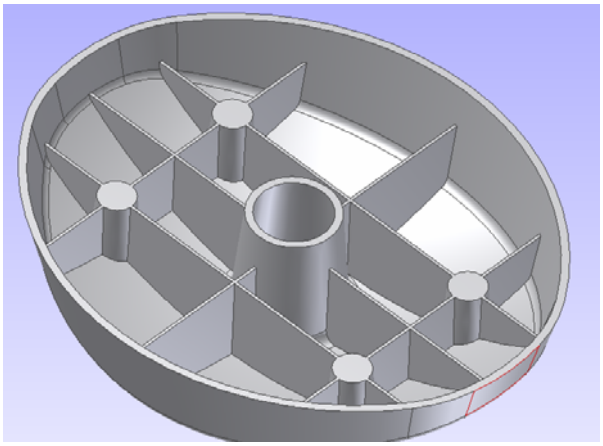
Step	Action	Result
162.	• Select **>>** on Dialog Box for further options.	
163.	• Set as below. 	
164.	• Select **OK**. **Note**: In a similar way, Sculpt2 can be selected and edited. It is also important to note this cannot be done with the shell command, due to the resulting non-uniform wall thickness.	Piece is hollowed out. 

Exercise 5: Use Rib Features

Task: Use Rib

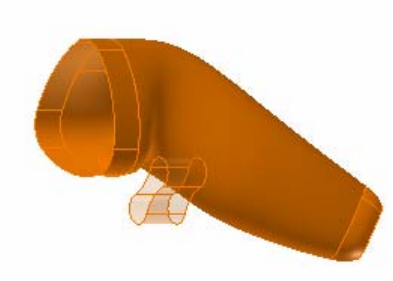
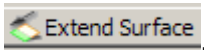
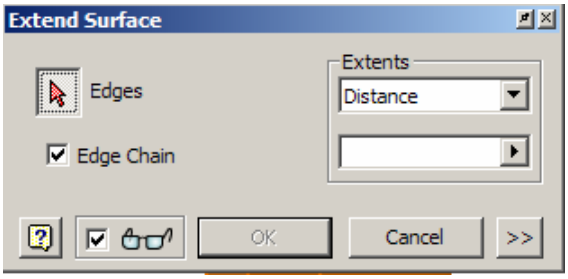
Step	Action	Result
165.	• Select **File** menu > **Open** > **18 RibWithDraft.ipt.**	
166.	• Select **Open**.	Model appears. 
167.	• Select the **Rib** command. 	
168.	• Click on all six lines – vertical and horizontal – to create a web. 	

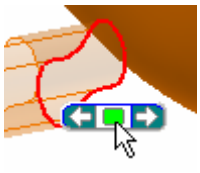
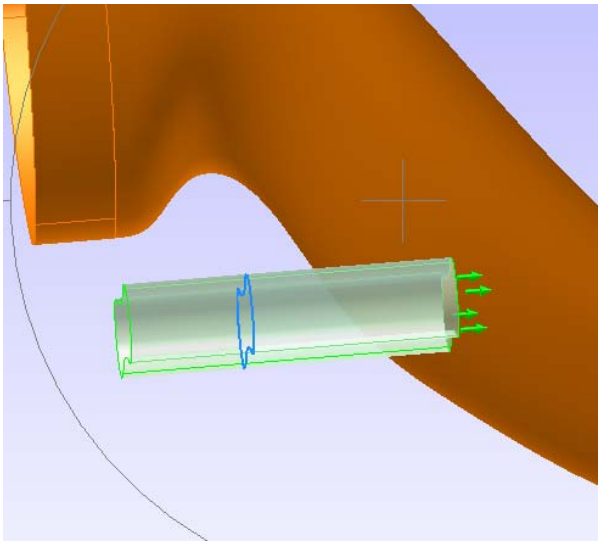
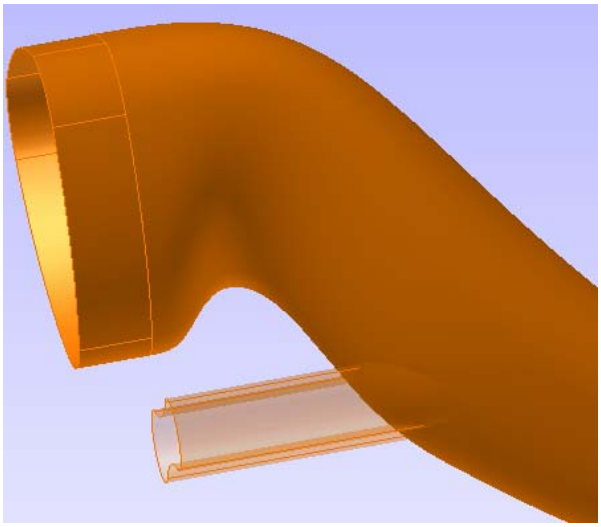
Step	Action	Result
169.	• Select **Direction**.  • Move the mouse over the selected sketch to preview potential web results. Move the mouse beneath the part to specify the direction. • Notice the preview arrow in the image to the right.	
170.	• Apply taper to the lateral faces of the web by setting the **Taper** to a value of **5** degrees. **Note**: The Taper control is new to Rib in Inventor 11. In previous releases, faces would have to have been selected individually to be tapered using the Face Draft command. • Notice Draft **Indicator**.	

Step	Action	Result
171.	• Enter **5** for **Taper** 	
172.	• Select **OK**.	**Note**: This will only add draft in the direction normal to the sketch plane. If you created a rib by sketching the side profile, draft cannot be applied.

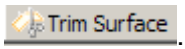
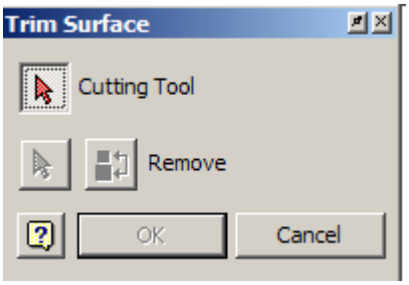
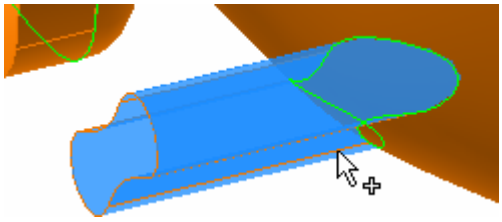
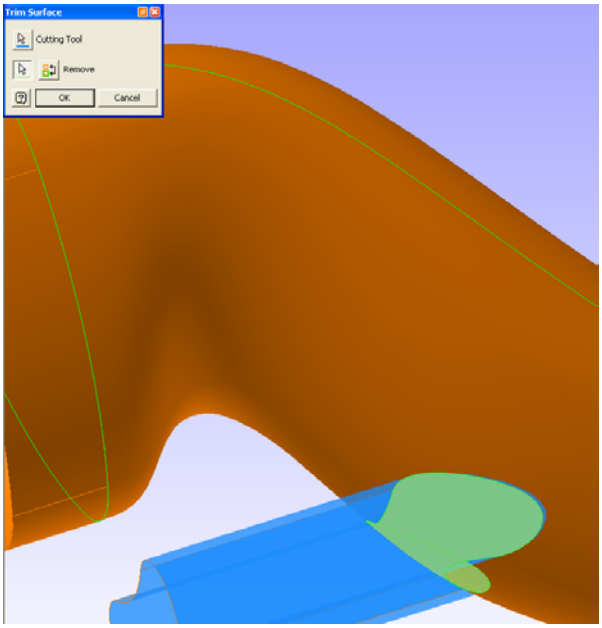
Exercise 6: Use Surface Features

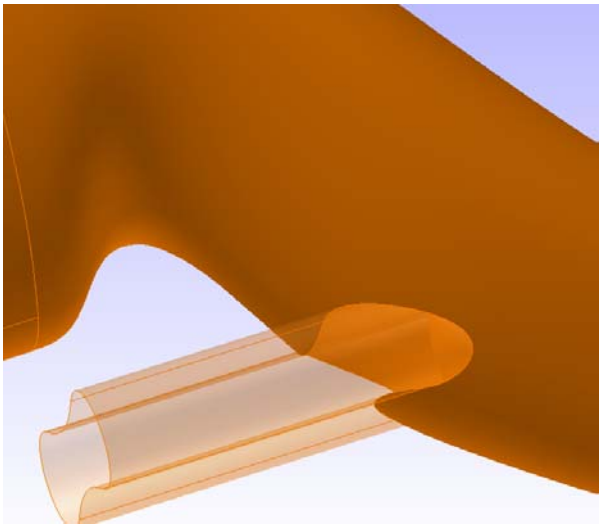
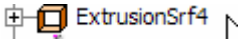
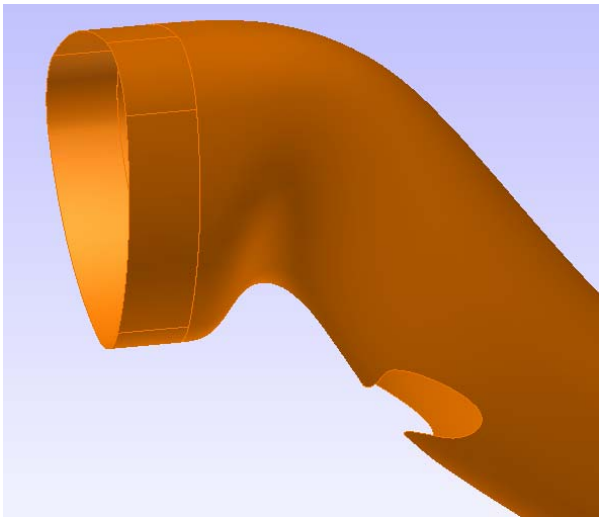
Task: Extend Surfaces

Step	Action	Result
173.	• Select **File** menu > **Open** > **19 Trim, Extend, BP 2.ipt.**	
174.	• Select **Open**.	Model appears. 
175.	• Click **Extend Surface**. 	

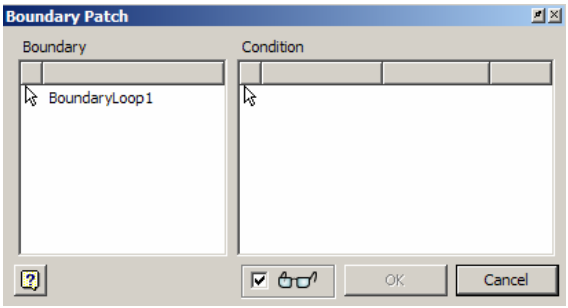
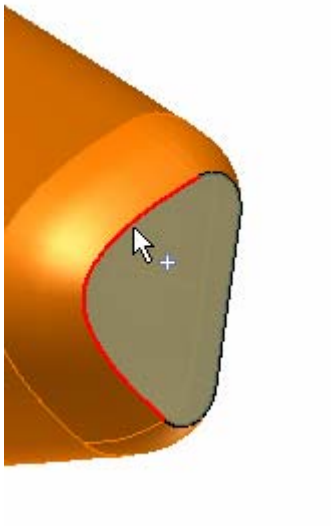
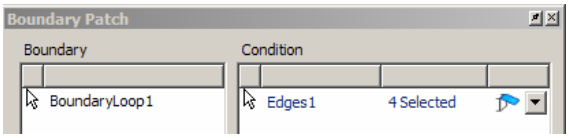
Step	Action	Result
176.	• Click edge of smaller surface and drag it toward larger surface using the **Arrows** or enter distance in field. 	
177.	• Select **OK**.	

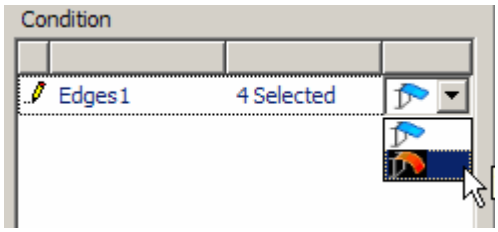
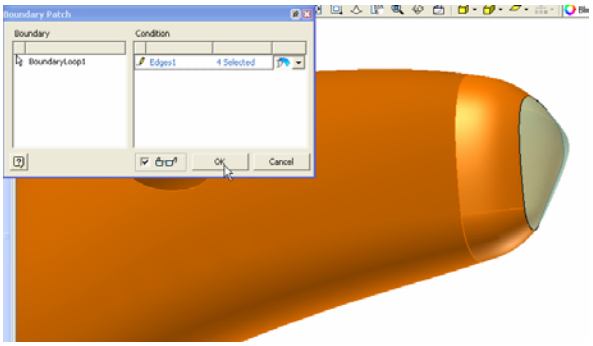
Task: Trim Surfaces

Step	Action	Result
178.	• Click **Trim Surface**. 	
179.	• Select the smaller surface as the Cutting Tool. 	
180.	• Select the portion of the surface to remove. • Optionally, use the **Invert Selection** button to toggle between keeping/removing selections.	

Step	Action	Result
181.	• Select **OK**.	
182.	• Right-click E**xtrusionSrf4** > **Visibility** to turn off. 	Now the part has a hole left to work with for buttons and so forth. 

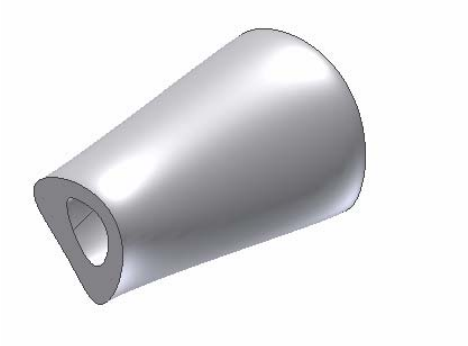
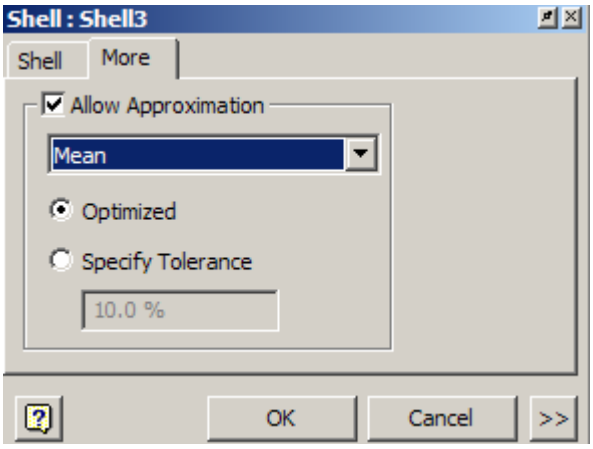
Task: Create a Boundary Patch

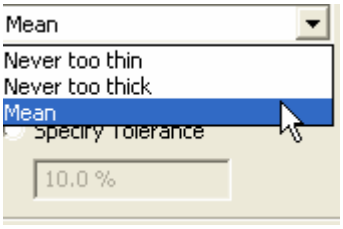
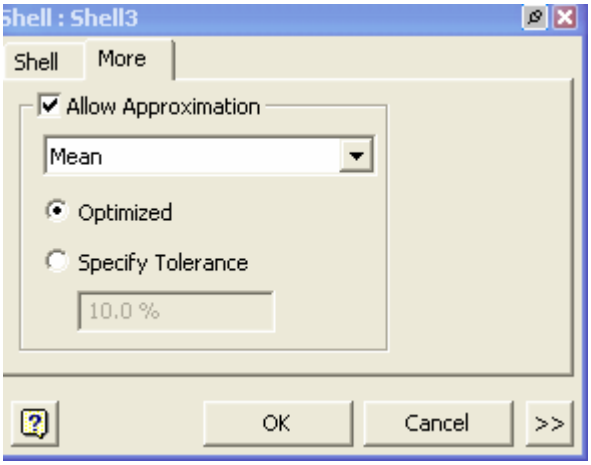
Step	Action	Result
183.	• Click **Boundary Patch**, which is now 3D.  .	
184.	• Select a loop on the part. Inside a loop are a number of edges. Four edges are inside this one. 	

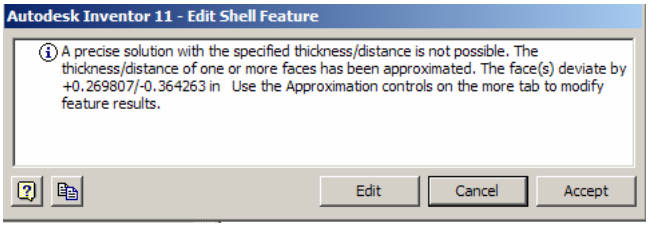
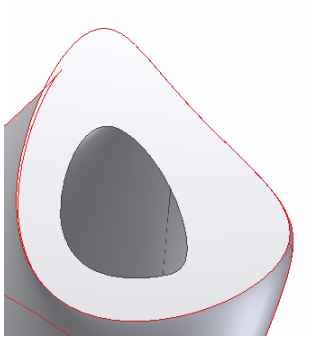
Step	Action	Result
185.	• Select **Tangent Condition**. 	The boundary patch will be tangent to adjacent surfaces.  **Note**: The Boundary patch will not create a G2 Condition when tangency is used.
186.	• Select **OK**.	

Exercise 7: Use Analysis Tools

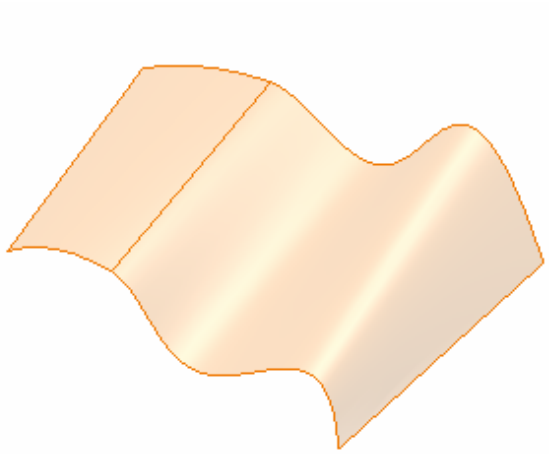
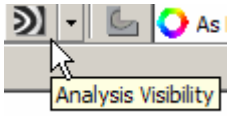
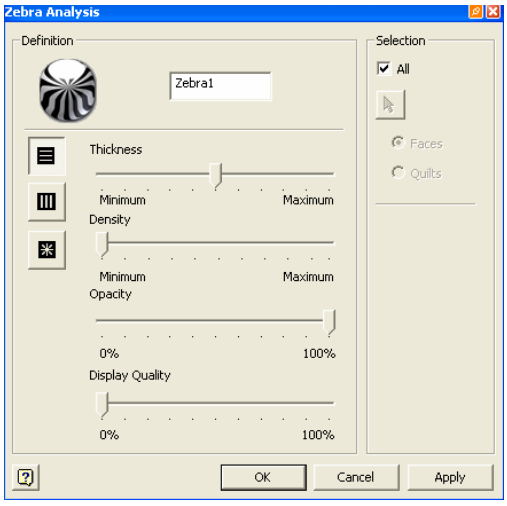
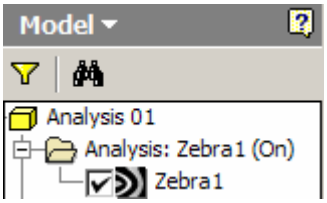
Task: Create a Proximate Offset

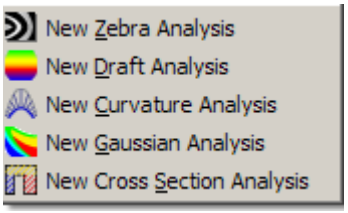
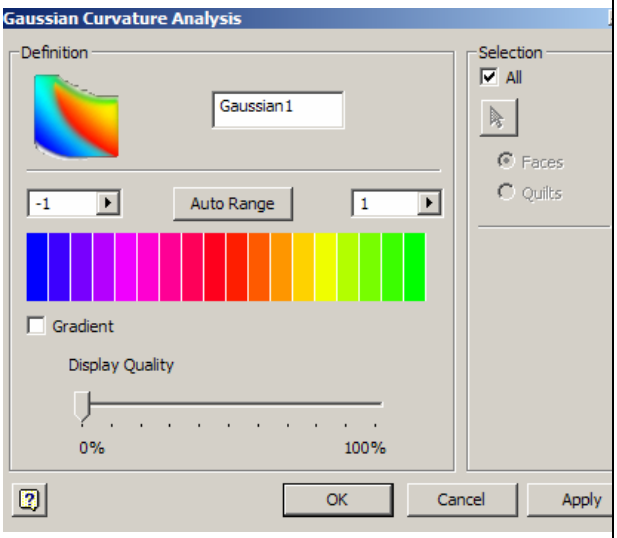
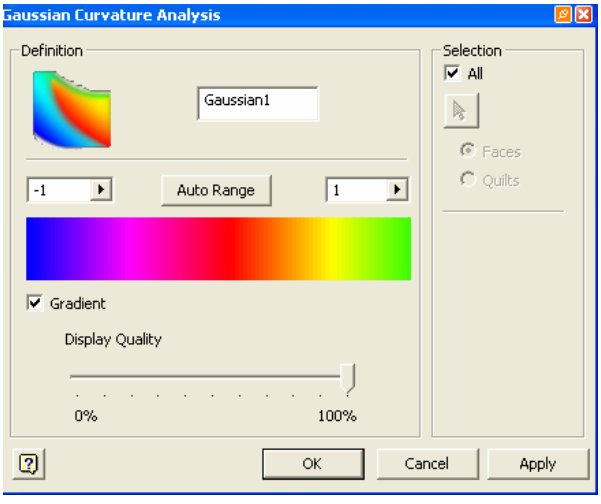
Step	Action	Result
187.	• Select **File** menu > **Open** > **21 Shell, Thicken, Offset.ipt**. .	
188.	• Select **Shell3** from browser > **Edit Feature** > **More** tab.	

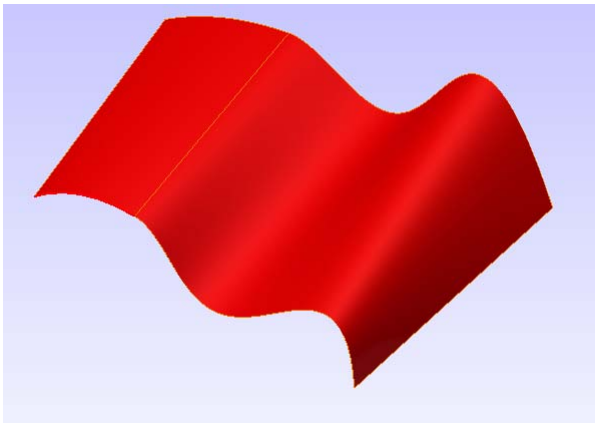
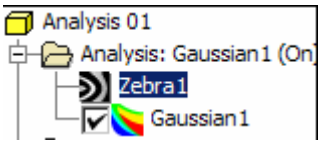
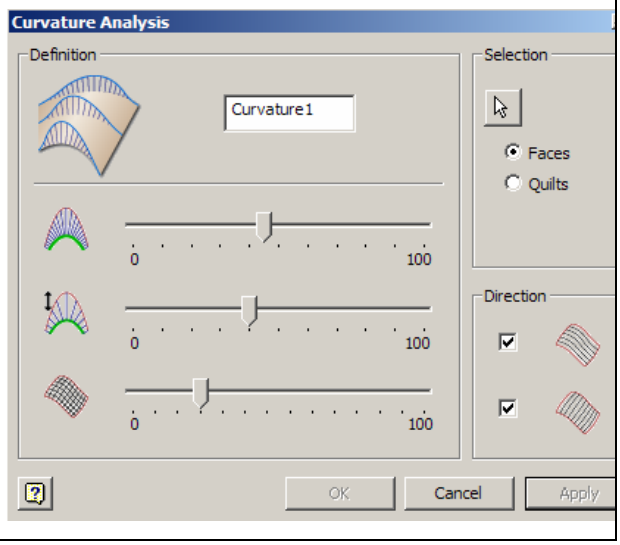
Step	Action	Result
189.	**New Approximation controls**. • When creating a Shell or Thicken/Offset feature in R11, Inventor first attempts to create the feature using a *precise* offset, in which surfaces are offset precisely with no deviation from the specified distance/thickness (legacy behavior). With **Allow Approximation** enabled, the feature will resort to a new *approximate* offset in the event that the precise offset fails. The approximate offset increases the likelihood of feature success by deviating slightly from the specified distance/thickness in order to handle complex geometry. • This drop-down controls the side on which the deviation occurs.  • **Optimized**: Tells approximation to do its best at approximating with performance in mind. It's a balance between performance and how tightly tolerances should be held. • **Specify Tolerance:** Allows users to specify a percentage by which the distance/thickness is allowed to deviate. For example, you might want to allow a one percent deviation for a thickness of a wall. **Note**: Smaller percentage values will result in slower feature performance.	

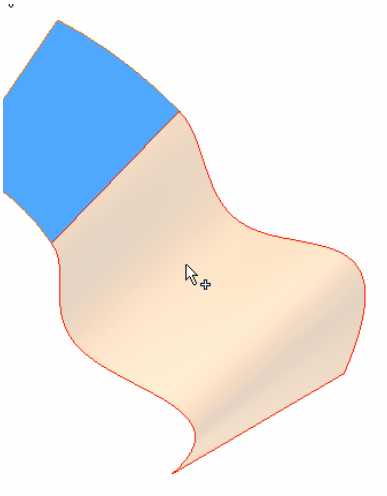
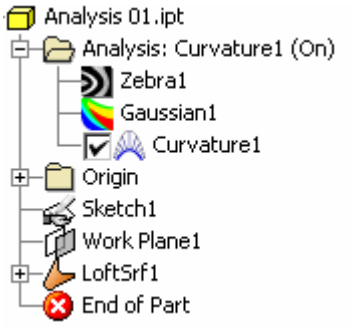
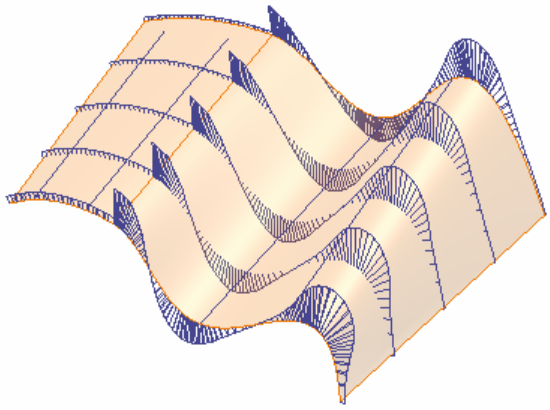
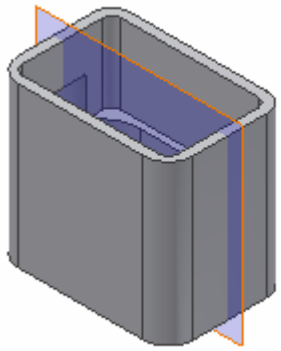
Step	Action	Result
190.	• Select **OK**. In the event that approximate offset is used, the feature provides an information dialog box noting the amount of deviation.	
191.	• In this case, select **Accept**.	
192.	The trouble area is on backside. • Rotate and zoom to see this. 	

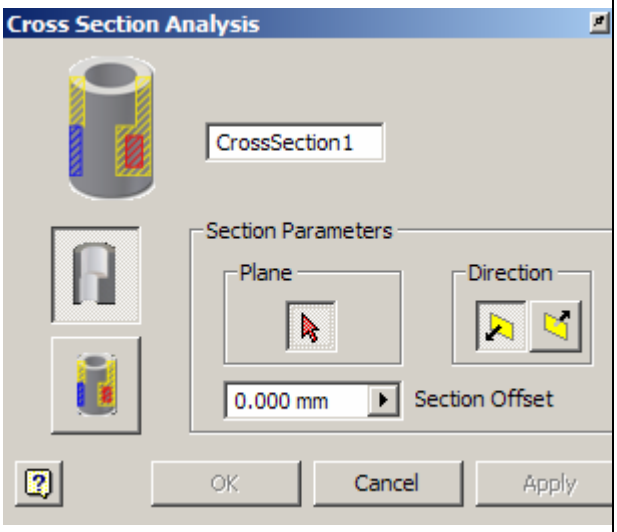
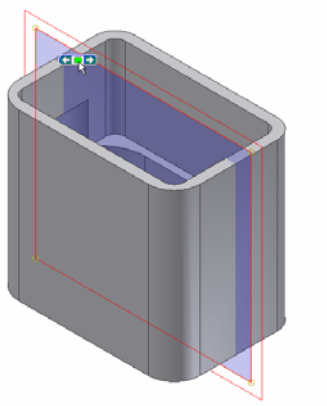
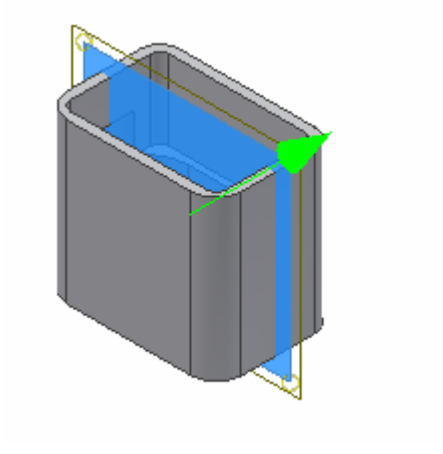
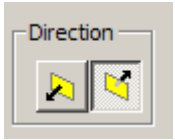
Task: Use Analysis Visibility

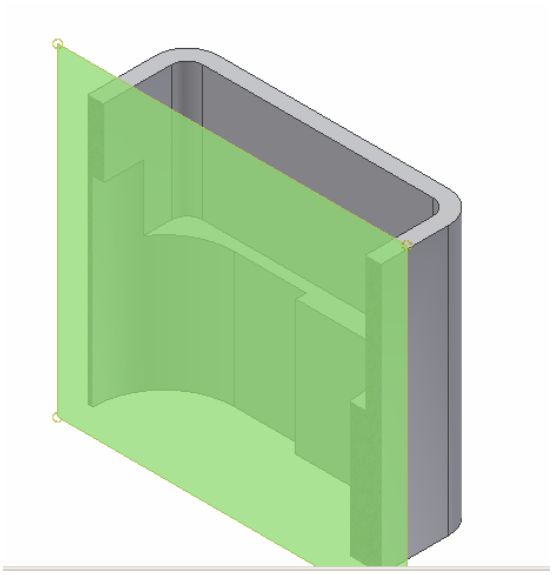
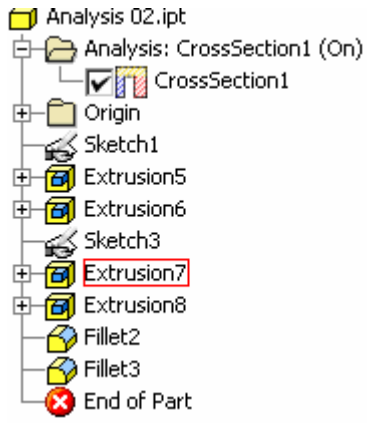
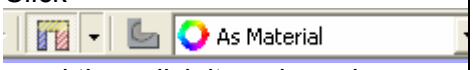
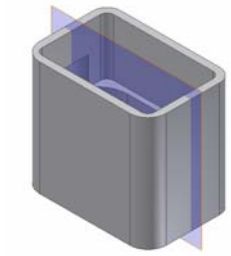
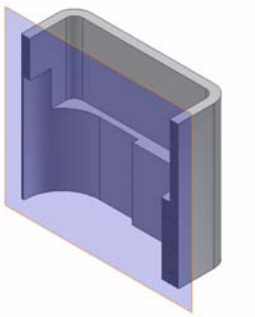
Step	Action	Result
193.	• Select **File** menu > **Open** > **Analysis 01.ipt**. .	
194.	• Select **Open**.	
195.	• Select **Zebra Analysis**.  • Select **OK**. 	Folder appears in browser.  Note Zebra Analysis 

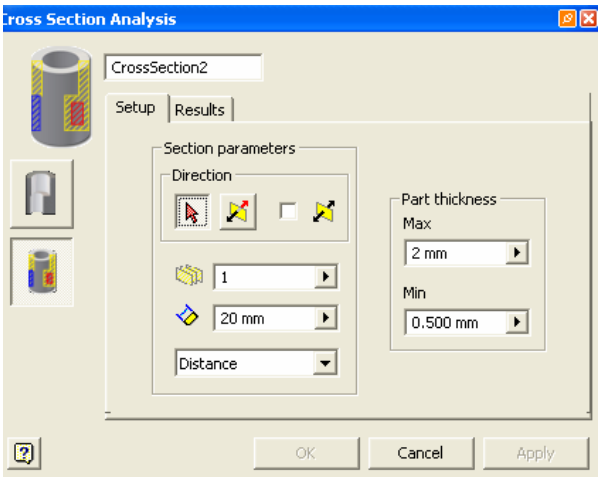
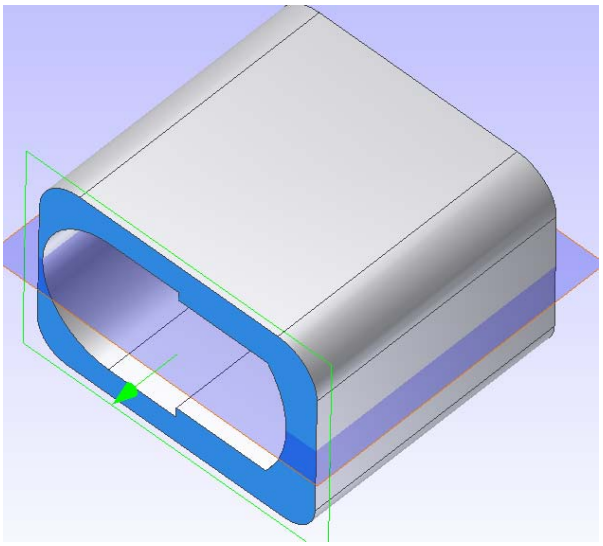
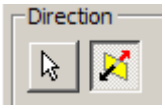
Step	Action	Result
196.	View other Analysis tools from drop-down.	
197.	• Select **New Gaussian Analysis**	
198.	• Select **Gradient, Auto Range** (which looks at part and sets a good range so there is a good result with the curvature), and turn up **Display Quality to 100 percent.**	

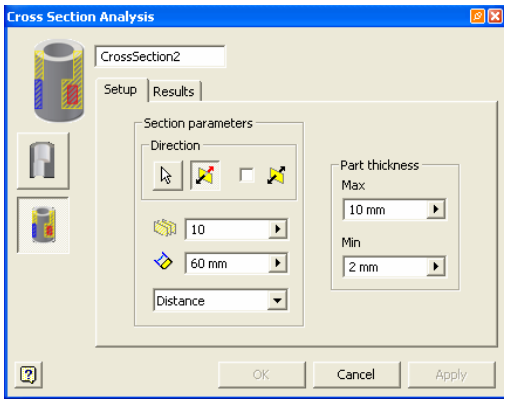
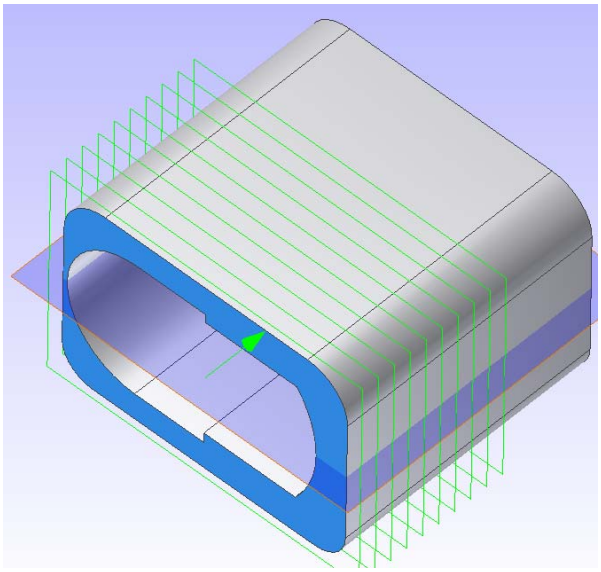
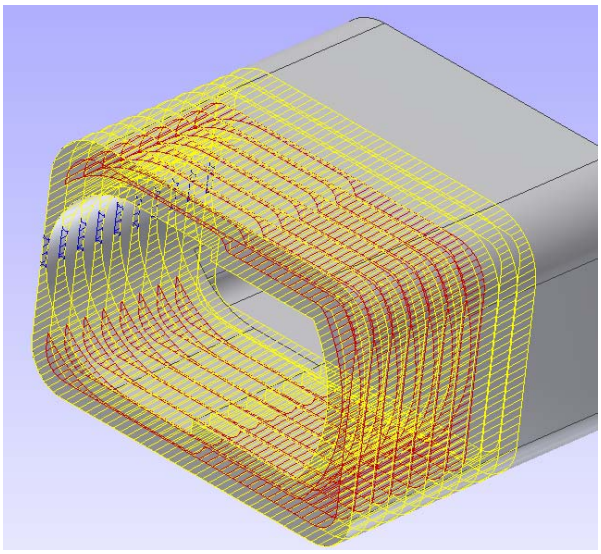
Step	Action	Result
199.	• Select **OK**. • This analysis is measuring changes in curvature along that surface.	
200.	• Notice that there are now two **Analysis Solutions**. Whichever one you click is active. • Click between **Zebra** and **Gaussian** to experiment.	
201.	• Select **Curvature Analysis**. 	

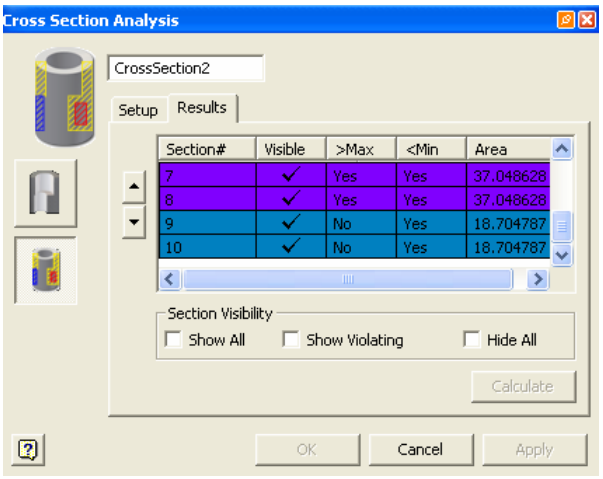
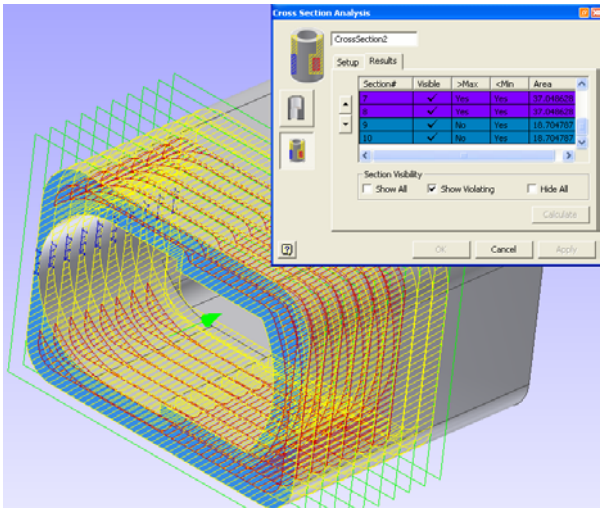
Step	Action	Result
202.	• Select both surfaces to be analyzed.	
203.	• Select **OK**. • Curvature Analysis Shows in browser. 	
204.	For cross-section analysis, open a different file. • Select **File** menu > **Open** > **Analysis 02.ipt**.	

Step	Action	Result
205.	• Select **New Cross Section Analysis**.	
206.	• Select **Simple** – the equivalent of a sliced graphic without having to burrow into a sketch to turn it on and off. 	
207.	• Select this plane. 	
208.	• Select **Direction**. 	

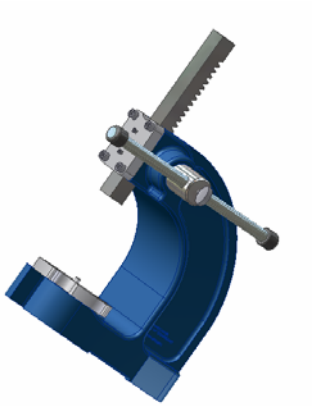
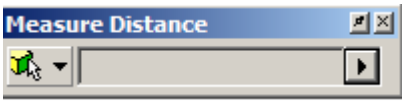
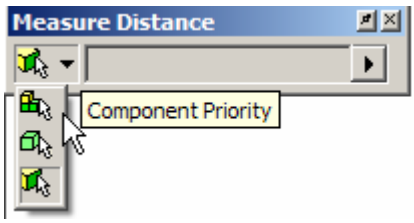
Step	Action	Result
209.	• Select **OK**.	
210.	With the sliced graphic checkmarked (indicating active), turn the slice on and off at will.  active • Click  and then click it again and so on – toggling the slice on and off.	 

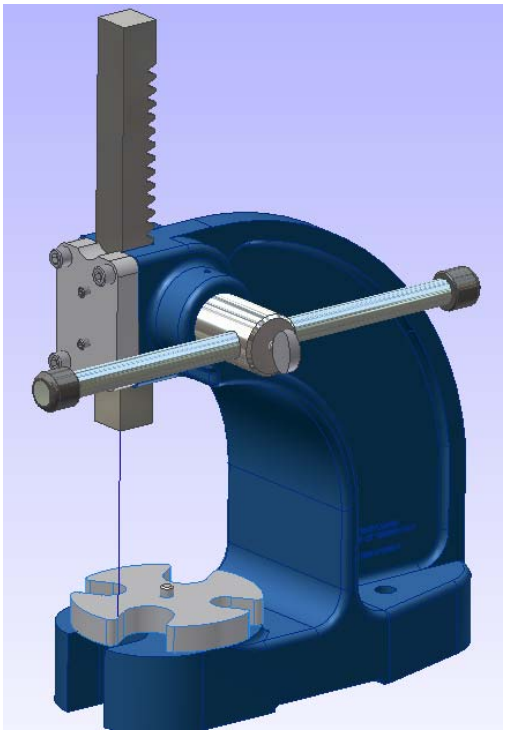
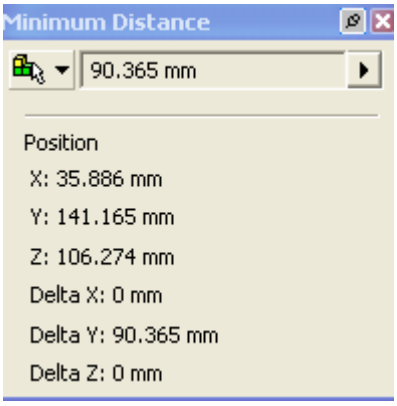
Step	Action	Result
211.	• Select **New Cross Section Analysis** to create another cross-section analysis from the drop-down. • This time, select **Advanced**.	
212.	• Select a plane.	
213.	• Select **Direction**. 	

Step	Action	Result
214.	• Enter the values as shown. 	
215.	• Select **OK**. The analysis creates slices or cross-sections at specified intervals. The cross-sections are colored based upon violation of the design requirements: • Yellow shows areas of the part which meet the design requirements (thicker than min, thinner than max) • Red shows areas where the part is too thick. • Blue shows areas where the part is too thin.	

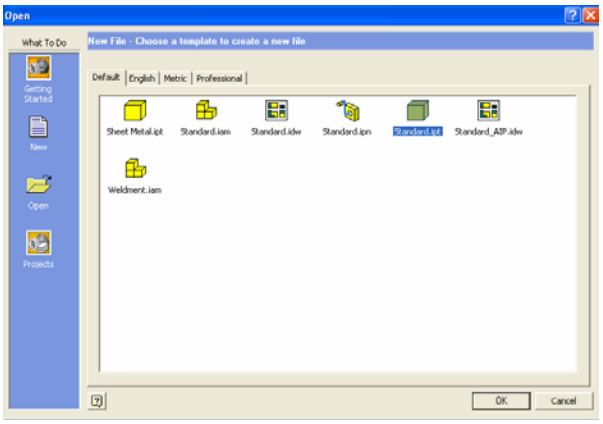
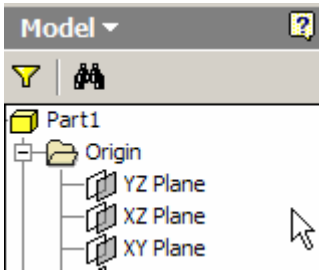
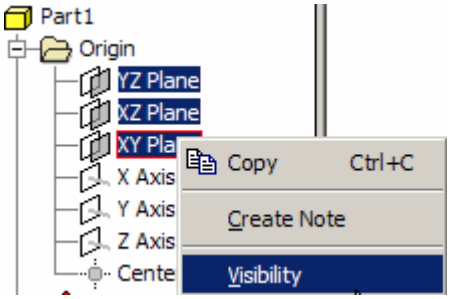
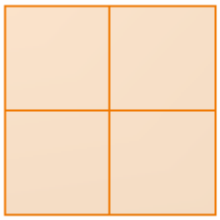
Step	Action	Result
216.	• Select **Results** tab.	
217.	• Select **Show All** and **Show Violating** to experiment.	

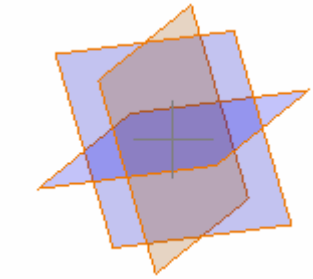
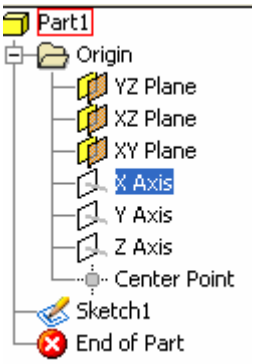
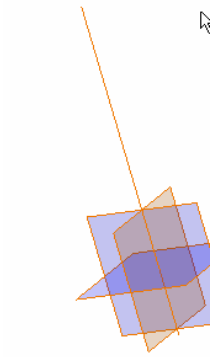
Task: Measure Distance between Two Parts

Step	Action	Result
218.	• Select **File** menu > **Open** > **Arbor Press. iam** .	
219.	• Select **Open**.	
220.	• Select **Tools** menu > **Measure Distance.**	
221.	• Select **Component Priority**. 	

Step	Action	Result
222.	• Select any two components. 	

Task: Make Changes in Work Geometry

Step	Action	Result
223.	• Select **File** menu > **New** > **Standard.ipt**	
224.	• Right-click on screen > **Finish Sketch**.	
225.	• Expand **Origin**. 	
226.	• Select three planes, right-click, select **Visibility**. 	

Step	Action	Result
227.	Notice the different color on normal versus anti-normal sides of planes. The color is changeable in registry settings. • Rotate and flip to experiment. 	
228.	• Work Axes are now resizable • Right-click on X Axis > **Auto-Resize turn off.**	
229.	• Click end of axis on the object to drag and release.	
230.	• Select **Close**.	

Check Your Understanding

1. What new tool is used to build part geometry from multiple surfaces?
2. Which fillet type enables G2 continuity to be applied?
3. What are the three control options for loft to a point?