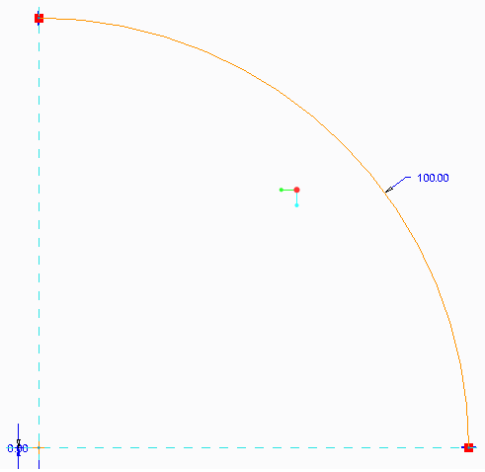
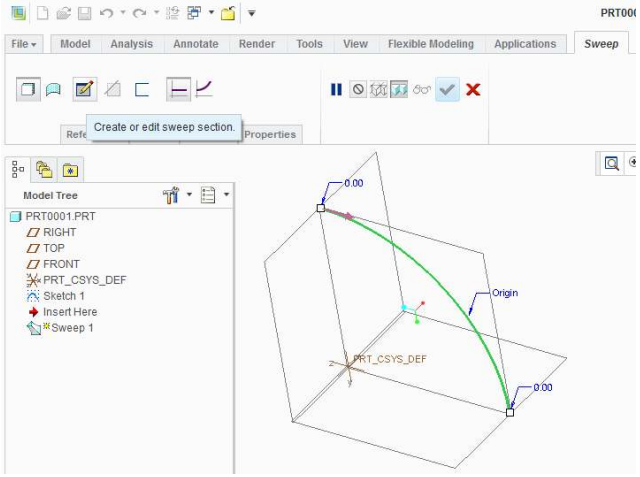
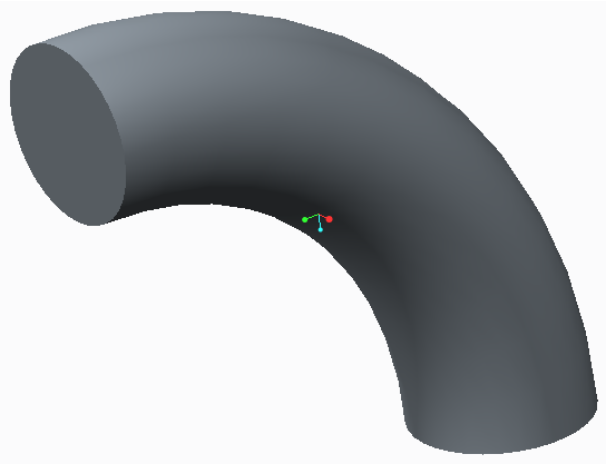
Συστήματα CAD – Sweep

Σάρωση κλειστής (στερεό) ή ανοικτής (επιφάνεια) διατομής κατά μήκος τροχιάς στο χώρο

1. Create a new part
2. Select FRONT and press SKETCHER to sketch a **100** radius **90** degrees arc, with its center aligned and the beginning of the axes. Exit SKETCHER.
3. Insert Sweep from the menu.
4. Select the arc, which will be the trajectory of the sweep.
5. Press the button shown, to create the sweep section.
6. Draw a **40** diameter circle at the beginning of the trajectory.
7. Press OK and OK again, to complete the feature.
8. Save the model.



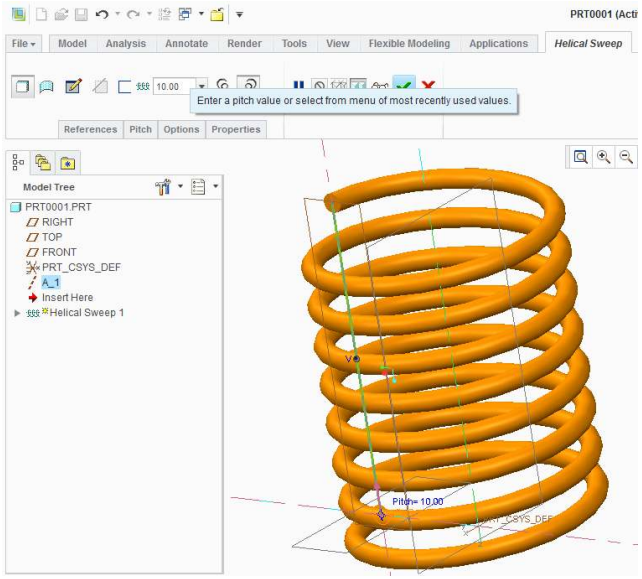




Συστήματα CAD – Helical Sweep

Σάρωση κλειστής (στερεό) ή ανοικτής (επιφάνεια) διατομής κατά ελικοειδούς τροχιάς στο χώρο

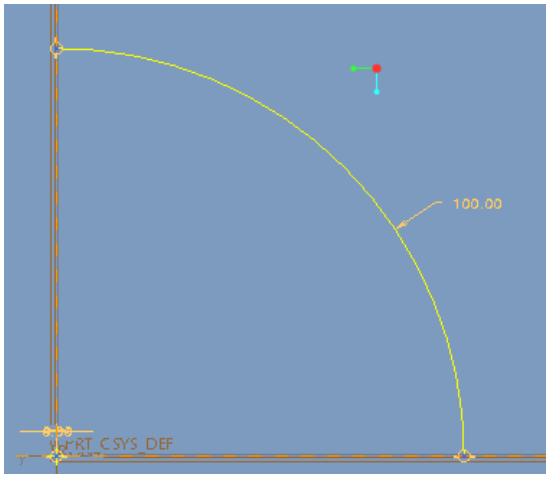
1. Create a new part
2. Insert Sweep > Helical Sweep from the menu.
3. Select References > Helical Sweep profile > Define
4. Select a sketching plane and select sketch, to sketch the helical sweep profile.
5. Draw a **80** length vertical line at a **30** distance from the middle plane.
6. Select Model and define a vertical Datum Axis at the intersection of the two vertical Datum Planes. This will be the axis of revolution of the helical sweep.
7. Press the button to define or edit the sweep section.
8. Draw a **5** diameter circle at the beginning of the sweep profile trajectory.
9. Exit sketcher and define the pitch at the appropriate field. Define 10 for the pitch value.
10. Complete the feature and save the model.

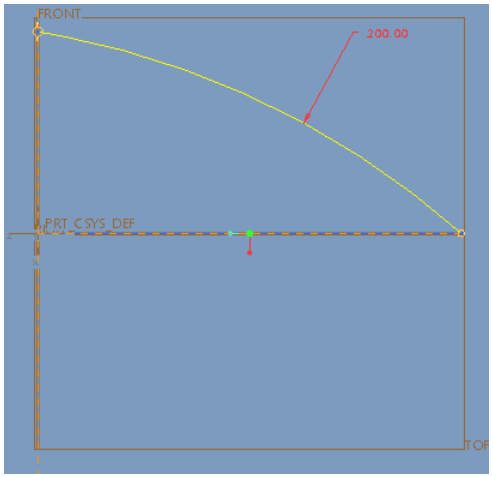


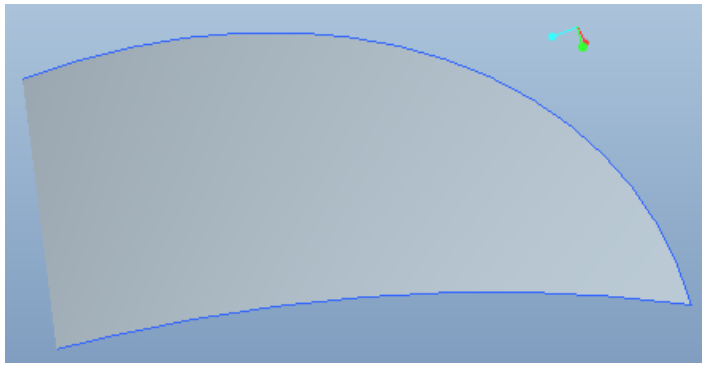
Συστήματα CAD – Boundary Blend

Δημιουργία επιφάνειας, ενώνοντας καμπύλες στο χώρο

1. Create a new part
2. Select FRONT and press SKETCHER to sketch a **100** radius **90** degrees arc, with its center aligned and the beginning of the axes. Exit SKETCHER.
3. Select RIGHT and press SKETCHER to sketch a **200** radius arc, with its one end on the front plane and the other aligned to the end of the previous arc. Exit SKETCHER.
4. Insert Boundary Blend.
5. Select the two arcs. Finish the feature.
6. Click Thicken and add a **2** thickness value to the feature to covert it from surface to solid.







Συστήματα CAD - Blend

(Expected time: 30 min)

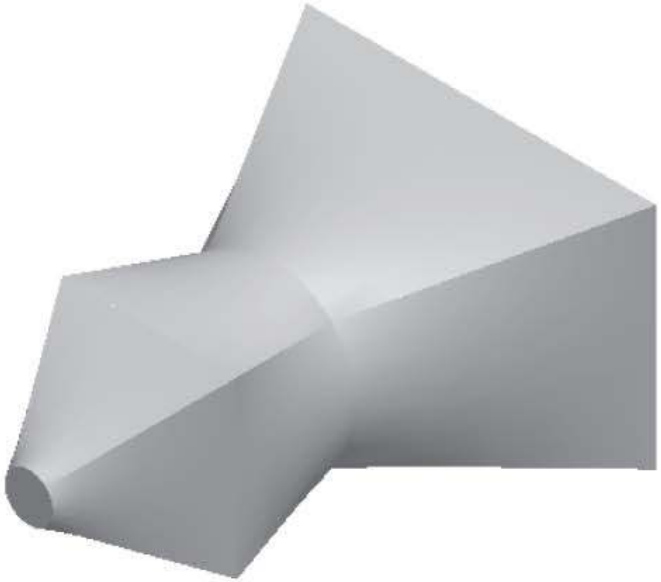


Figure 8-110 *Isometric view of the model*

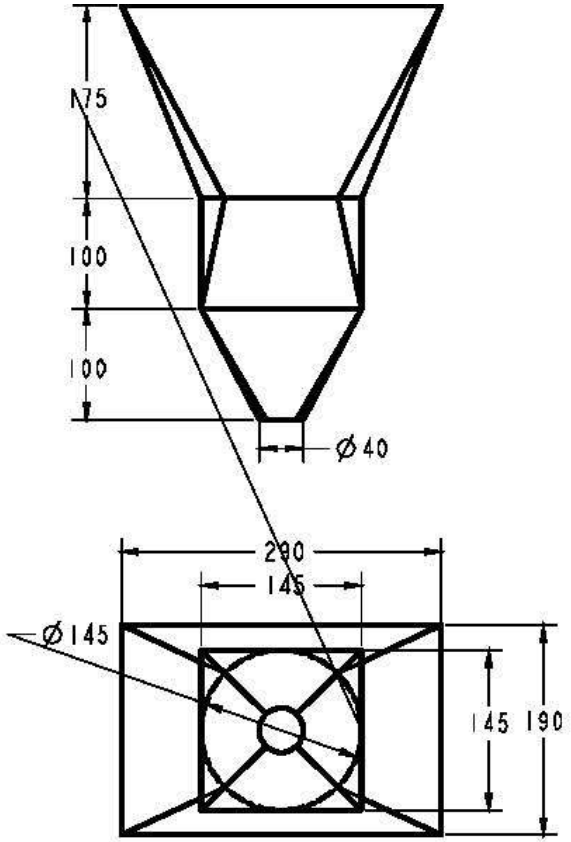


Figure 8-111 *Top and front views of the model*

In general, the following steps are required to complete this tutorial:
Create the sketch for the first section of the blend feature, refer to Figures 8-112.
1. Create the sketch for the second section of the blend feature, refer to Figures 8-113.
2. Create the sketch for the third section of the blend feature.
3. Create the sketch for the fourth section, refer to Figures 8-114. Give the depth between section numbers 1 and 2, 2 and 3, and 3 and 4.
4. Redefine the model to change the straight blending into a smooth blending, refer to Figure 8-116.

Detailed Description
Start a New Object File: The three default datum planes are displayed in the drawing area.
1. Choose Shapes > Blend from the menu bar.
2. Choose Straight option from the OPTIONS menu.
A Smooth blend will be created during the redefining of the model using the same sections that will be used to create the given model .

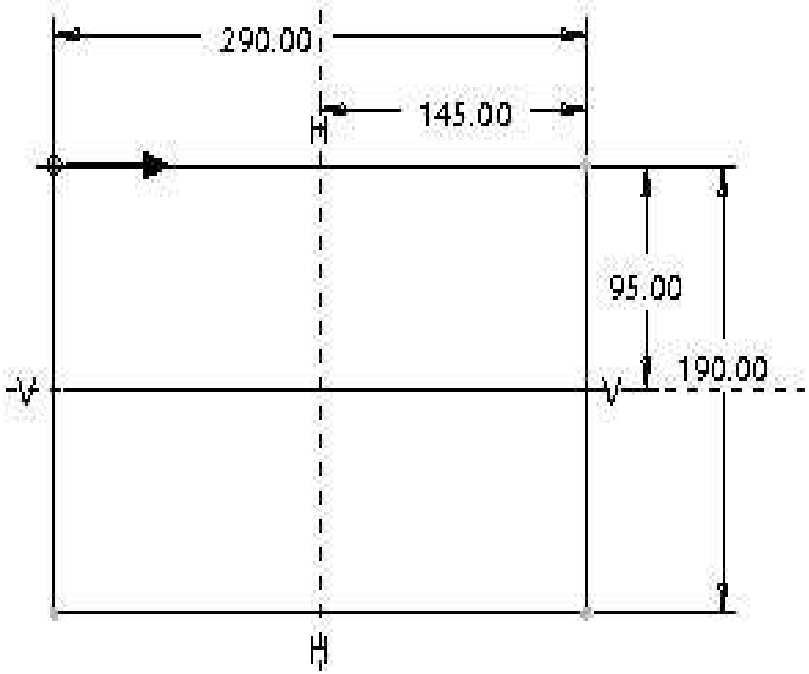
Selecting the Sketch Plane
1. Select Section > Define
2. On the menu Select the FRONT datum plane as the sketching plane.
3. Select Sketch

Drawing the First Section

The first section is a rectangle of 290x190 units.

1. Draw the sketch of the rectangular section and then add constraints and dimensions to it, as shown in Figure 8-112.

Figure 8-112 First rectangular section with dimensions



After drawing the rectangular section, you need to toggle the section and draw the next section.

Note
While drawing the sections for the blend feature, the start point is very important. The start point should be similar to those shown in the figures. If the start point is not at the desired point then select the point where you need the start point. Hold down the right mouse button to invoke a shortcut menu and choose the Start Point option.

Toggling the Section
Pressing OK, to exit Section completes the first section.
From the section menu, select Section 2 and then Sketch.
For the following sections, from the section menu select insert and then Sketch

The next section is a circle.

1. Draw the sketch of the circular section, refer to Figure 8-113. Modify the diameter of the circle to 145.

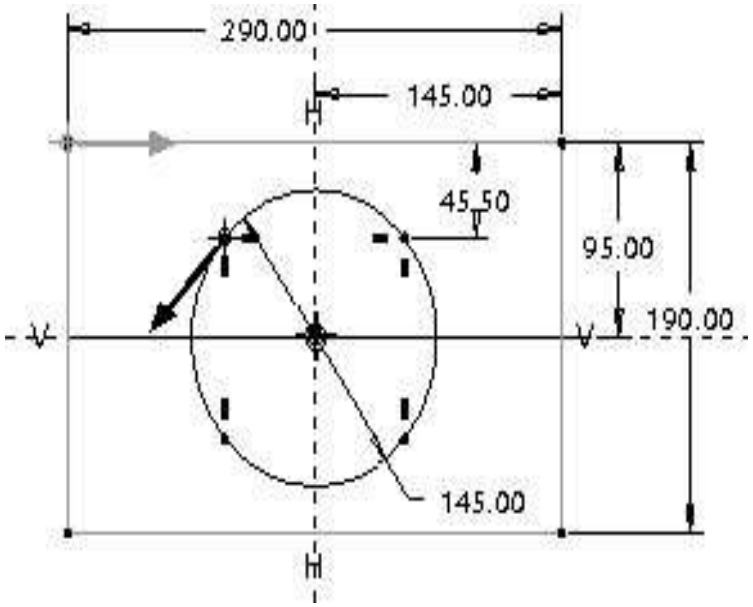


Figure 8-113 Two completed sections

The number of entities per section must be equal in a blend feature. Since, a circle is a single entity, it should be divided at four points.

Dividing the Circular Section

The circular section should be divided at four points because the rectangle and square have four entities. When you divide a circle at four points, the number of entities becomes four.

1. **Divide Segment** button
2. Select the circle at four points, as shown in Figure 8-113.

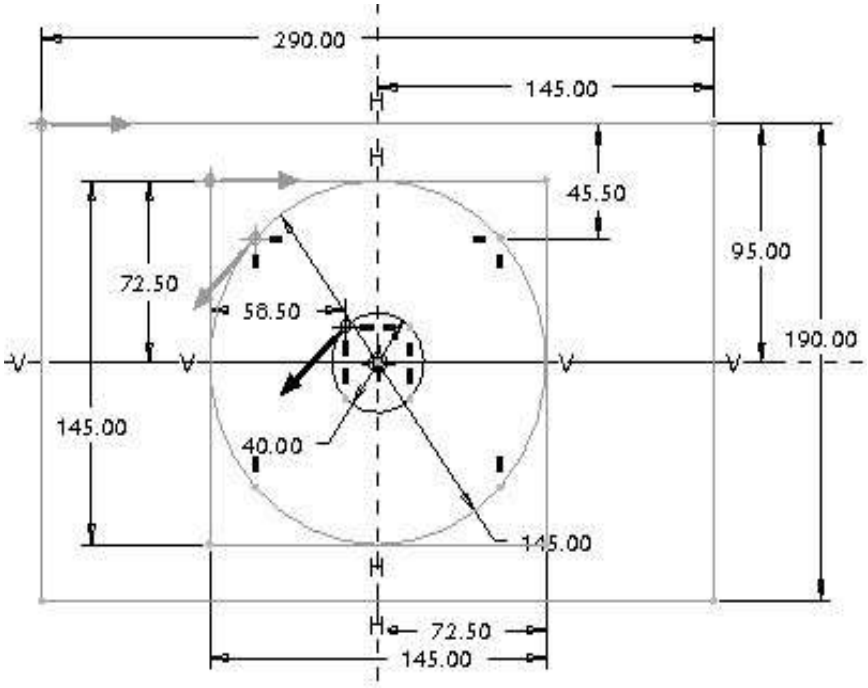
As you select points on the circle to divide it, some weak dimensions appear on the circle. Next, you need to apply constraints on the four points that were selected to divide the circle.

Applying Constraints on the Four Points

1. Choose the **Make line or two vertices vertical** constraint button from the **Constraints** ribbon and select the two division points on the left side of the circle to lie in a vertical line. Similarly, select the two points on the right to apply the constraint.
2. Now, choose the **Make line or two vertices horizontal** constraint button and select the two division points on the upper half and the lower half to lie in a horizontal line.
3. Modify the vertical dimension of the upper left division point, as shown in Figure 8-113. After the circular section is completed, the two sections with dimensions will look similar to the sections shown in Figure 8-113.

5. Now, toggle the section and create the next section. The next section to be drawn is a square. After drawing the square section, draw the circular section. Divide the circular section into four entities similar to section 2 and then constrain and dimension it.
Figure 8-114 shows all sections completed with dimensions.

Figure 8-114 All four completed sections before giving depth



Note
In Figure 8-114, note the direction of the start points indicated by arrows. These arrows are important to avoid a twisted feature. To **change** the direction of the arrows, select the point, right click and select "Start point"

Applying Depth to the Sections

After the sketches of all sections are completed, you need to specify the depth between each section. The dimensions for depth between each section can be referred to from Figure 8-111.

1. From the Sections menu the depth between each section is defined.
2. Enter the value 175 for the second section.
3. Enter 100 for section 3 and 100 for section 4.

Choose OK, to complete the feature.

Saving the Model

1. Choose the Save button from the File toolbar and save the model.

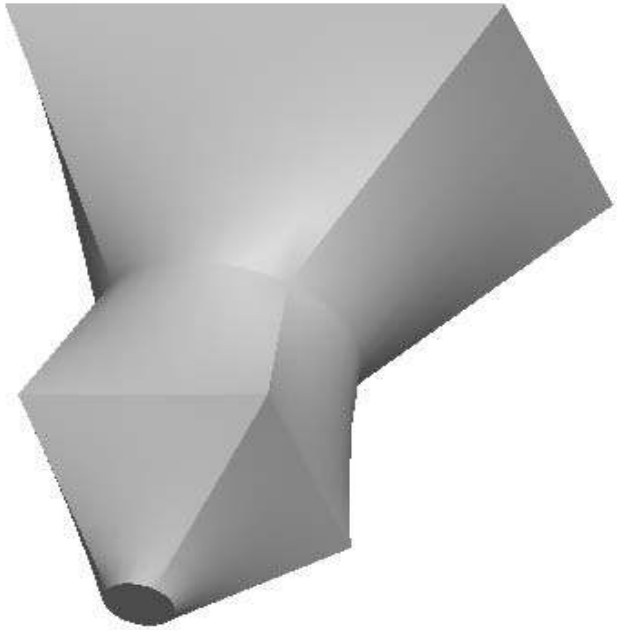
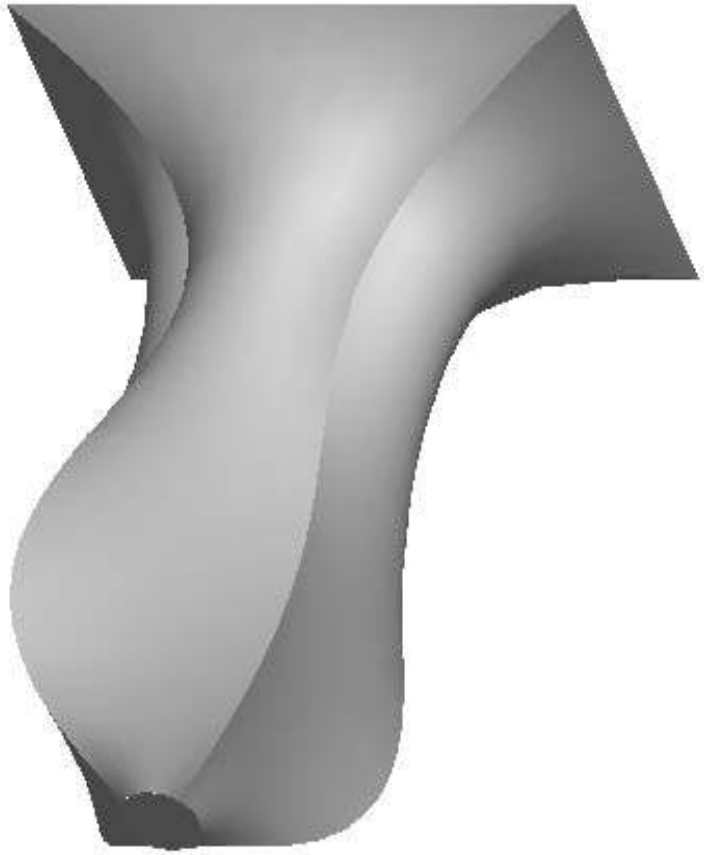


Figure 8-115 Trimetric view of the model

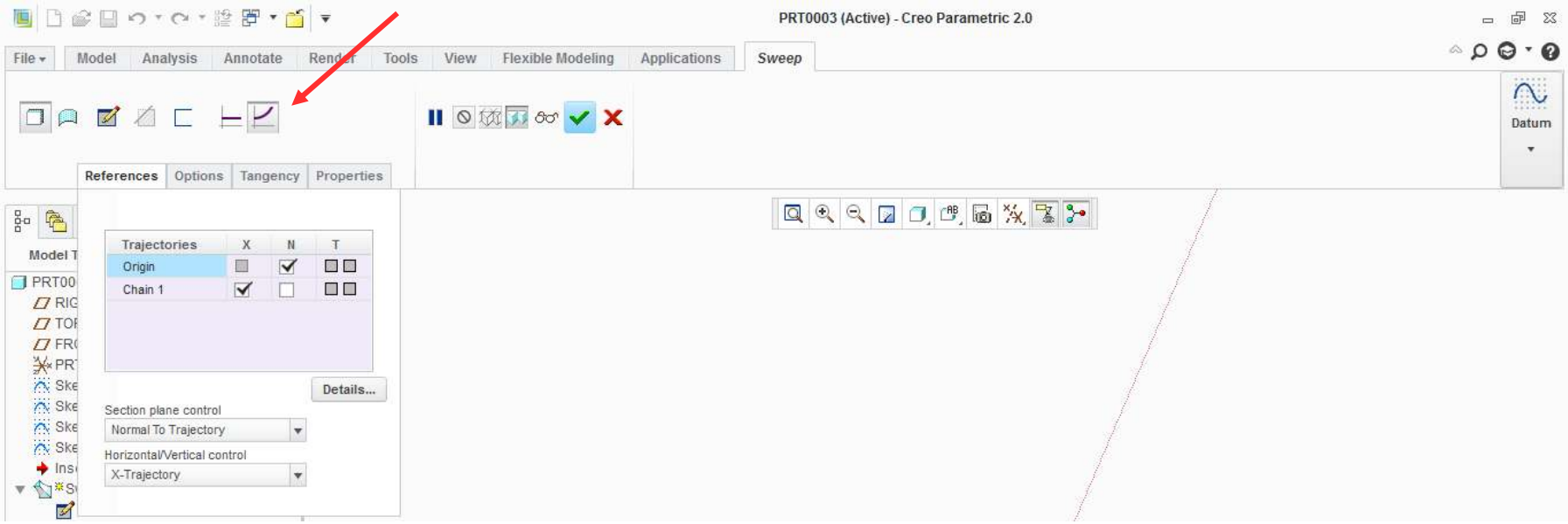
Redefining the Blend Feature After saving the straight blend feature, you will redefine this feature so that it is converted into a smooth blend.

1. Select the model in the drawing area. The edges of the model turn red in color.
2. Press and hold down the right mouse button in the drawing area until a shortcut menu is displayed.
3. Choose the Edit Definition option from the shortcut menu.
4. Select the options menu
5. Choose Smooth
6. Now, choose the OK button; the smooth blend is created, as shown in Figure 8-116.

Figure 8-116 Smooth blend feature



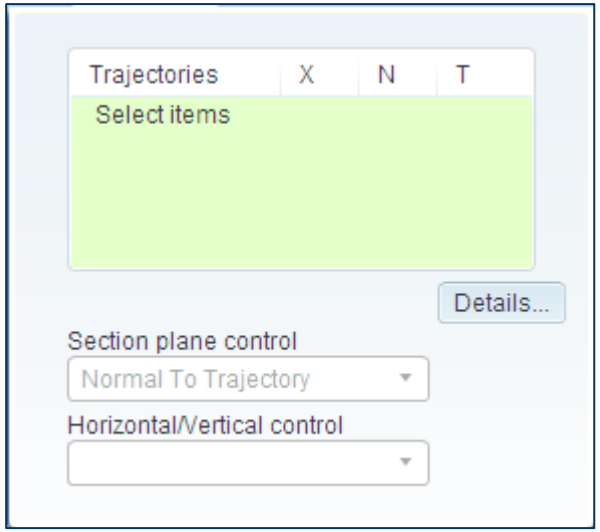
Variable Section Sweep Option



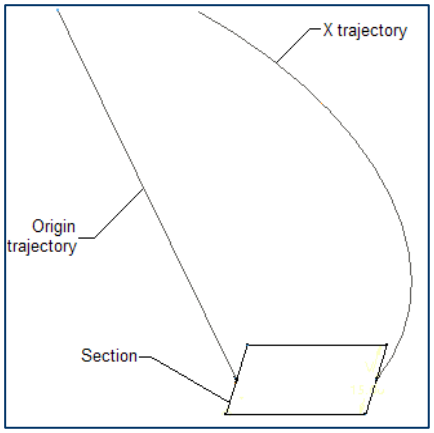
The **Variable Section Sweep** dashboard

From the Sweep ribbon, select the button with the arc for the sweep trajectory

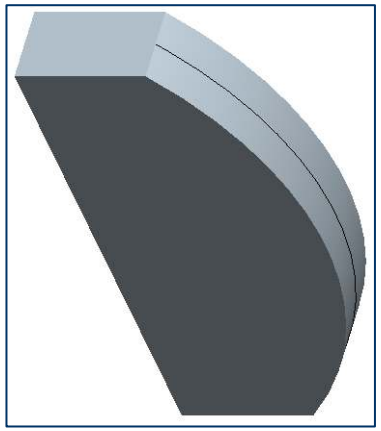
- **References Tab**



The **References** tab slide-down panel



The two trajectories



Variable section sweep feature

Συστήματα CAD – Variable Section Sweep

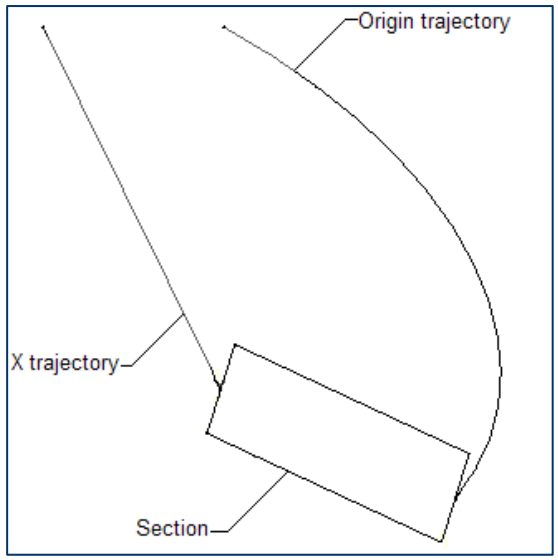


Figure C The two trajectories and section

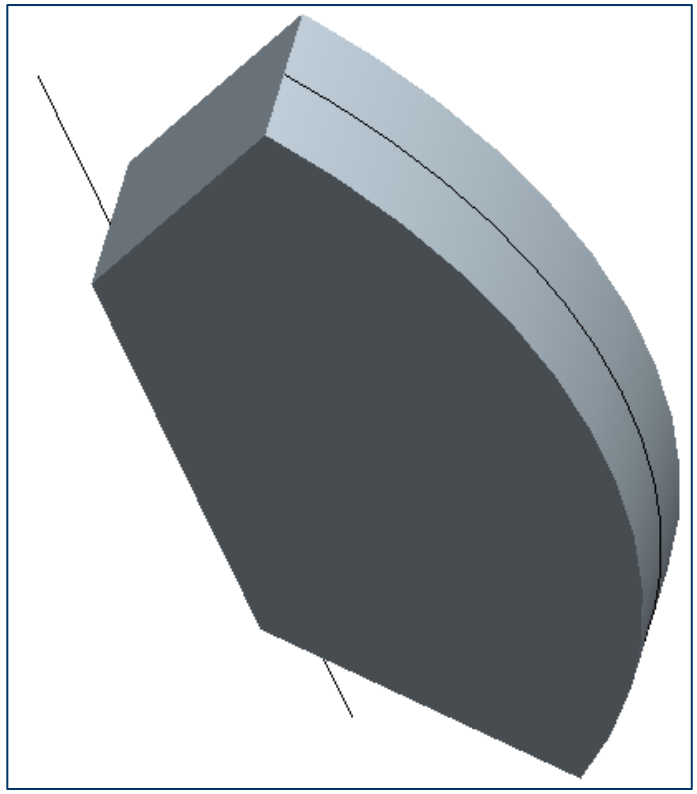
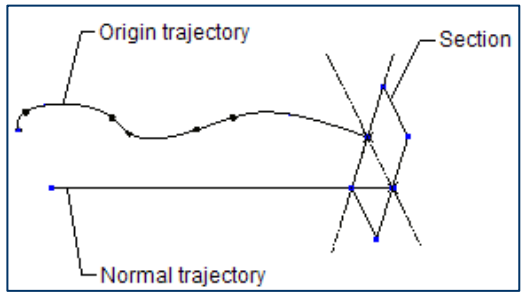
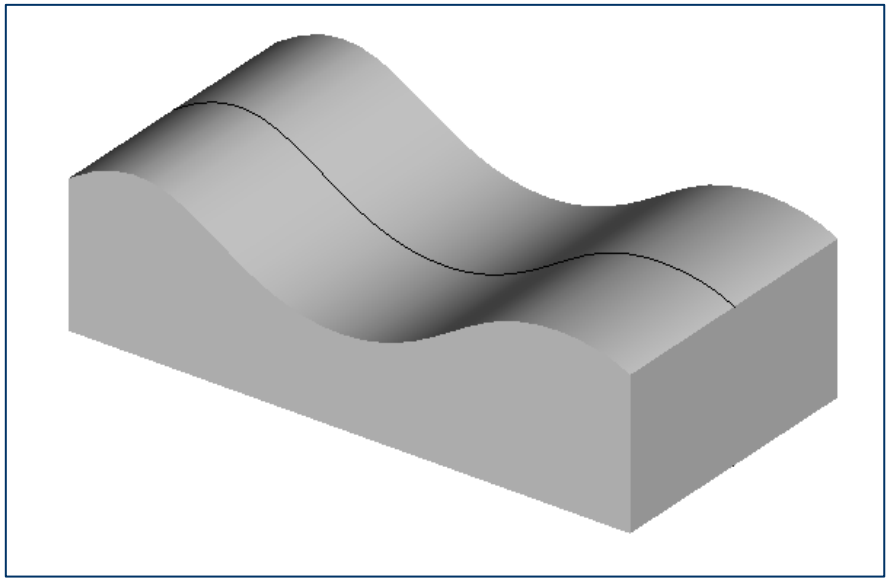


Figure D Resulting variable section sweep feature

Συστήματα CAD – Variable Section Sweep



Trajectories and the section



Resulting swept surface

Συστήματα CAD – Variable Section Sweep

1. Create a part named BOTTLE.PRT.
2. Create on the FRONT datum. Start a line at the intersection with the RIGHT datum and extend it vertically a distance of **140**. This datum curve is the “spine” trajectory.
3. On the same sketching plane sketch the profile in Figure 11.
4. Sketch the same profile to the RIGHT datum plane.
5. Insert Sweep and select Variable Section Sweep.
6. Pick the straight, vertical datum curve at the intersection of the FRONT and RIGHT datum planes.
7. Press Ctrl and pick the other two curves.
8. Press SKETCHER.
9. Sketch a rectangle with the sides aligned with the start of the two sketched profiles.
10. Fillet the top right angle with radius 8.
11. Mirror the feature to complete the shape of the bottle.
12. Complete the feature.
13. Mirror the feature to complete the part.
14. Shell the part, with thickness **2**.

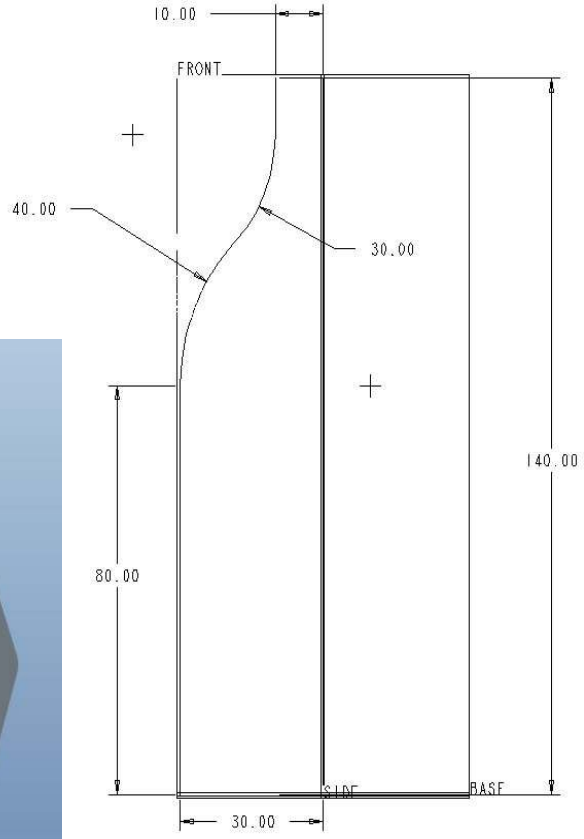
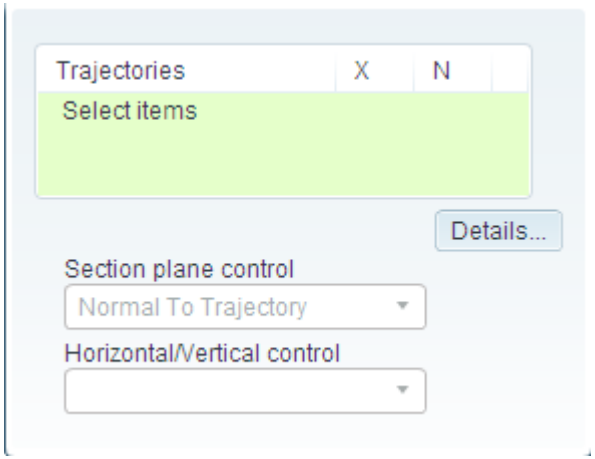


Figure 11

Swept Blend

The **Swept Blend** dashboard

- **References Tab**



The **References** tab slide-down panel

Συστήματα CAD – Swept Blend

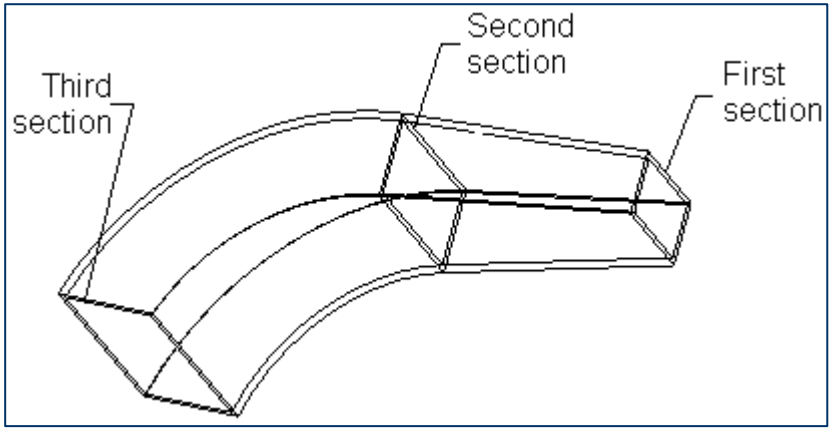


Figure A Three sections in swept blend

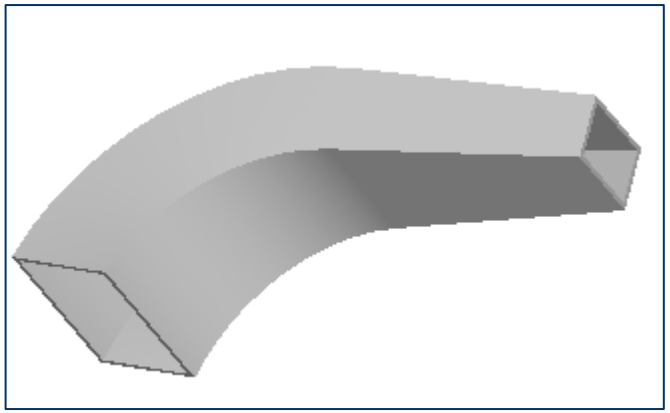


Figure B Shaded view of the feature

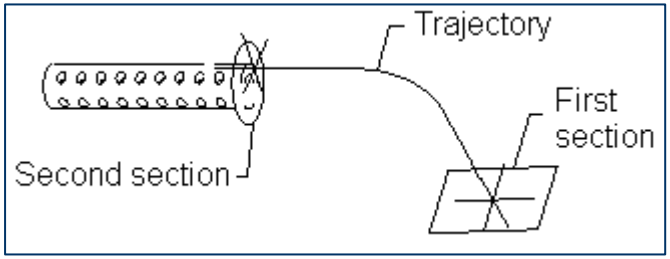


Figure C Two sections and a trajectory

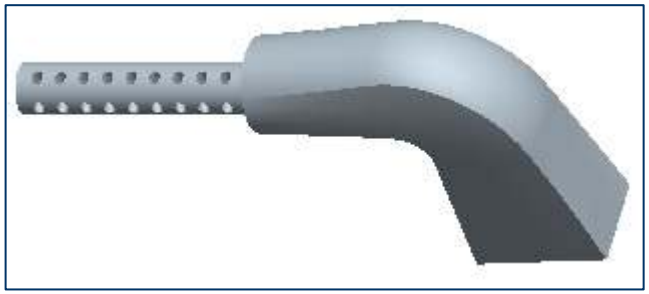
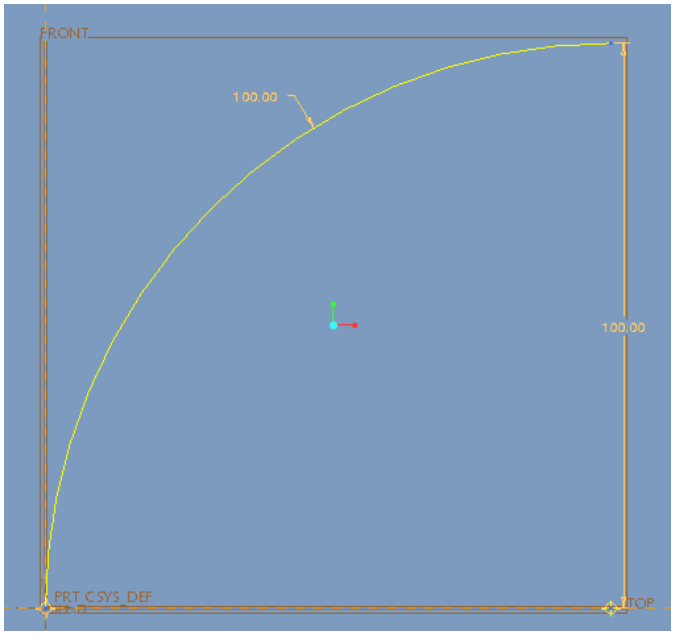
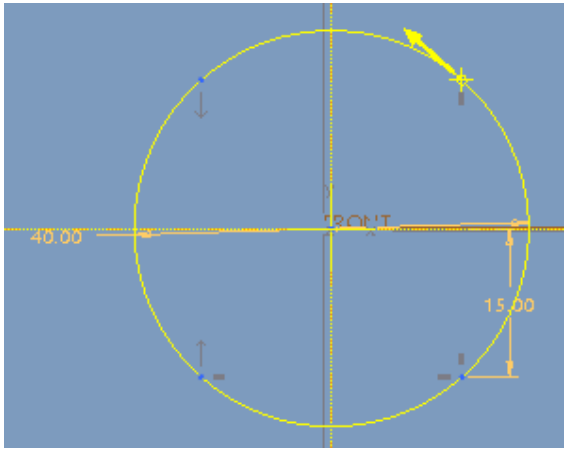


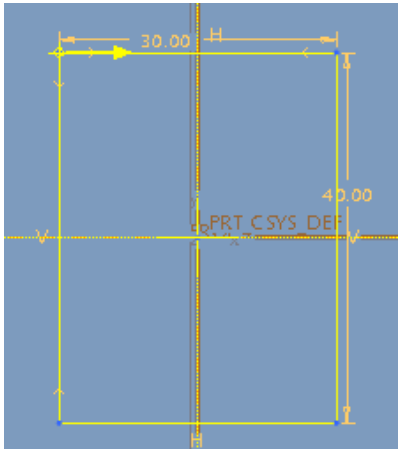
Figure D Resulting Feature

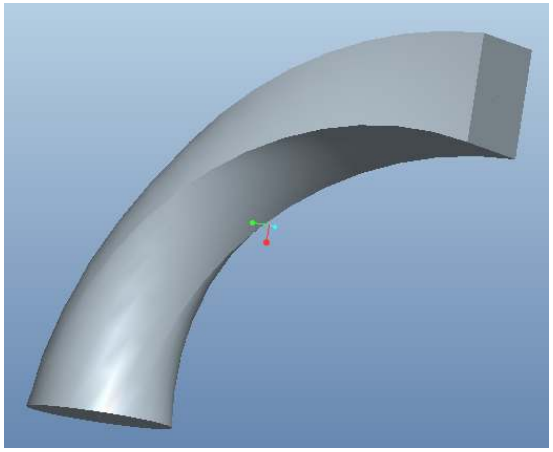
Συστήματα CAD – Swept Blend

1. On a new prt, select FRONT plane and SKETCHER, to sketch the arc shown in the figure.
2. Insert Swept Blend feature.
3. Select the arc as the curve of the blend.
4. Go to SECTIONS, select Section 1, location Start and press sketch to sketch a circle as shown.
5. Divide to **4** sections, as shown and close Sketcher.
6. Press insert to add Section 2, location End and press sketch to sketch a rectangle as shown. Close Sketcher.
7. Finish the feature.
8. It looks twisted, because the points the circle was divided do not perfectly align with the rectangle vertices.





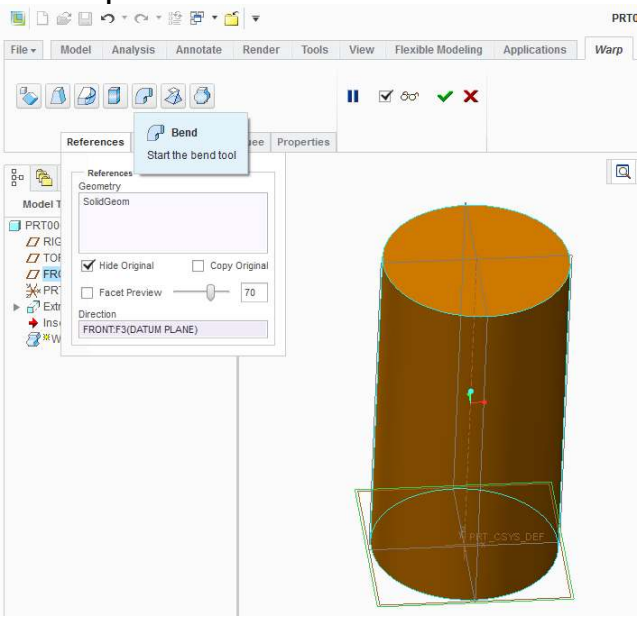


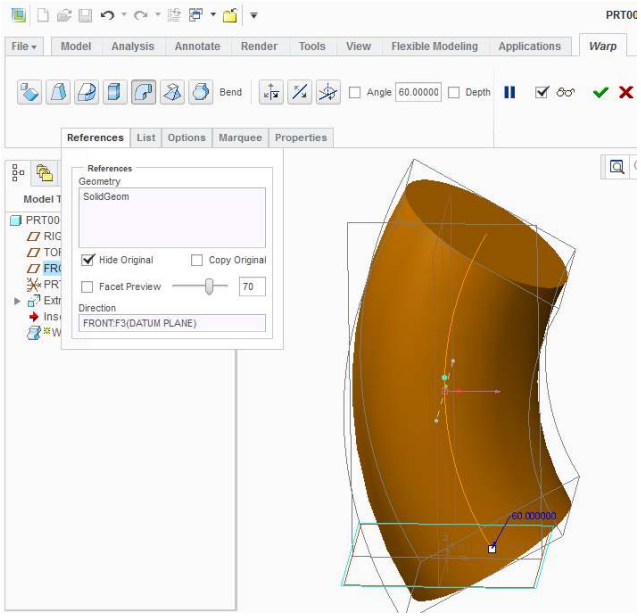


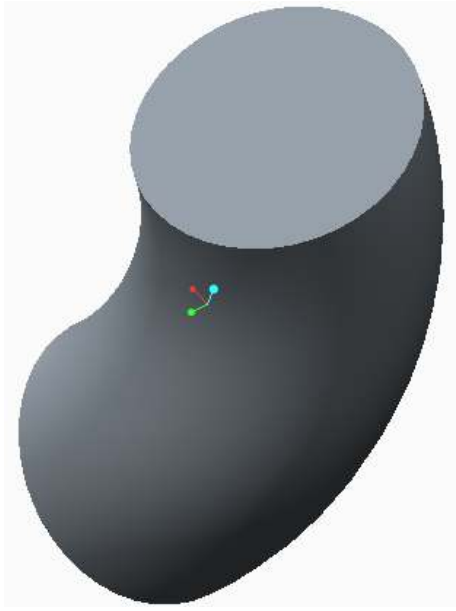
Συστήματα CAD – Warp

Επεξεργασία γεωμετρίας, αλλάζοντας το σχήμα της προς διάφορες διευθύνσεις (Bend, Twist, Sculpt, etc.)

1. Create a new part
2. With Extrude draw a **80** diameter, **150** height cylinder at the intersection of the datum planes in any plane.
3. Select Editing > warp
4. Select the cylinder and at the direction field, select the plane at the bottom of the cylinder.
5. Select Bend and select **60** at the dimension.
6. The cylinder bends to create a **60** degree angle
7. Now select twist and add **60** to the dimension, to further twist the cylinder by **60** degrees in the longitudinal direction.
8. Complete the feature and save the model.







Ασκήσεις

Στη συνέχεια ακολουθούν ασκήσεις σχετικές με το λογισμικό CAD.
Τις ασκήσεις αυτές πρέπει να τις υλοποιήσετε στο αντίστοιχο λογισμικό του PTC Pro Engineer Creo 2.0 και να τις αποστείλετε σε ηλεκτρονική μορφή μέχρι το επόμενο μάθημα (μέχρι την ημέρα του επόμενου μαθήματος, πριν από αυτό). Μπορείτε να χρησιμοποιήσετε τις οδηγίες που παρέχονται, ή να χρησιμοποιήσετε δικό σας τρόπο σκέψης και υλοποίησης των ασκήσεων.

Για το λογισμικό CAD ζητείται να παραδοθούν τα αντίστοιχα αρχεία σχεδίων. Από τις 4 ασκήσεις σχεδίων που ακολουθούν να υλοποιήσετε αυτήν που απαιτείται για την άσκηση του CAM και 1 από τις άλλες 3, όποια επιθυμείτε.

Μαζί με τα αρχεία αυτά πρέπει να παραδοθεί και αναφορά, στην οποία να περιγράφεται:

- Η διαδικασία υλοποίησης των ασκήσεων
- Η μεθοδολογία που ακολουθήσατε και η ροή των εντολών στο λογισμικό
- Πιθανά προβλήματα και δυσκολίες που συναντήσατε
- Πιθανές σκέψεις που μπορεί να σας αναπτυχθούν, σε σχέση με τη διαδικασία χειρισμού του λογισμικού, οι οποίες πιστεύετε ότι θα έκαναν απλούστερη την υλοποίηση των ασκήσεων (με αντίστοιχη τεκμηρίωση)

Συστήματα CAD – Blend - Άσκηση

In this exercise, you will create the model of a carburetor cover shown in Figure 8-129. Figure 8-130 shows the sectioned top view, sectioned front view, sectioned right-side view, and the sectioned bottom view with dimensions. (Expected time: 45 min)

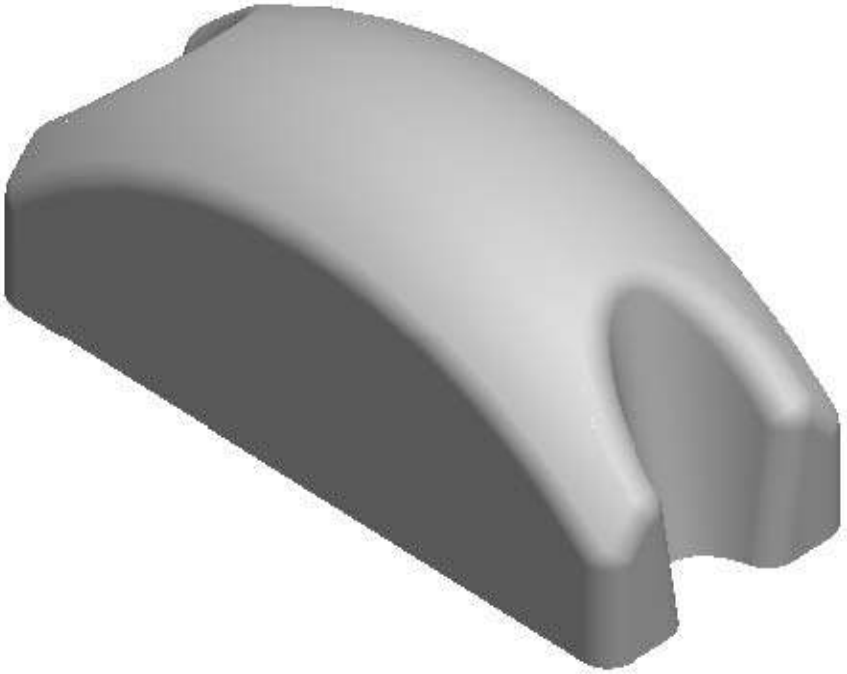


Figure 8-129 Isometric view of the carburetor cover

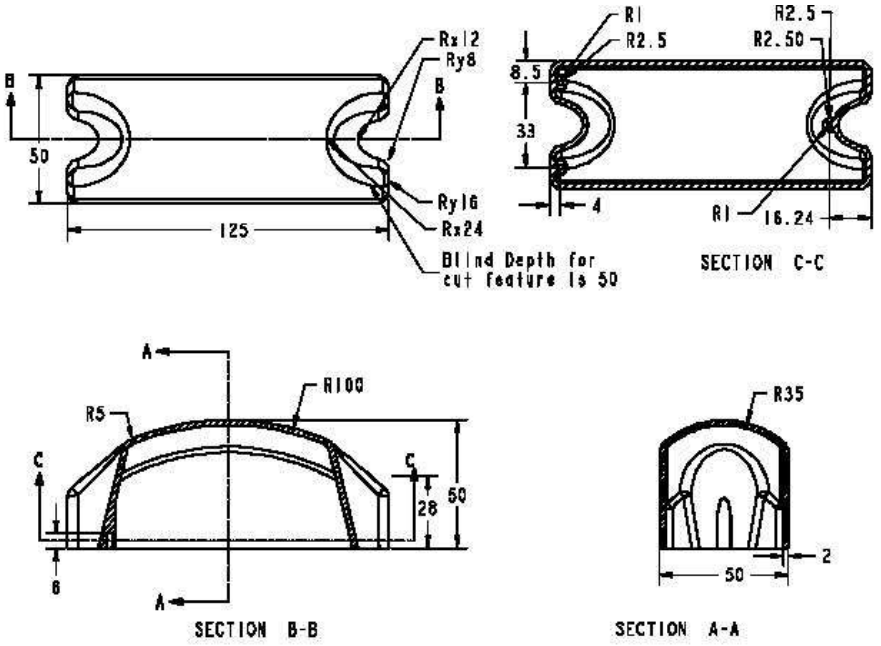


Figure 8-130 Top view, sectioned front view, sectioned right-side view, and sectioned bottom view of the carburetor cover

Συστήματα CAD – Blend - Άσκηση

Hint

1. Create the sketch of the base feature that includes a rectangle of 125x50 and then extrude it.
2. Choose Sweep from the menu bar. Select the Remove material button
3. Select the FRONT datum plane as the sketching plane for sketching a trajectory.
4. Create a trajectory. The start point and the endpoint of the arc should be at a distance of 28 from the bottom of the base feature and the radius of the arc should be 100.
5. Exit the sketcher environment using the Done button.
6. Choose the Merge Ends option from the menu.
7. You will again enter the sketching environment to create the section for the sweep feature. The arc created should be tangent to the reference lines, and the endpoints of the arc should be aligned with the edges of the base feature and should have a radius of 35.
8. Choose OK from Sweep dialog box.
9. Choose Blend from the menu bar. Select the Remove material button
10. Select the bottom face of the base feature as the sketching plane.
11. Choose Okay from the DIRECTION submenu.

Συστήματα CAD – Blend - Άσκηση

Hint (continue)

12. Choose the TOP option from the SKET VIEW submenu and select the RIGHT datum plane.
13. Create an ellipse of Rx12 and Ry8 using the Ellipse button.
14. Toggle Section.
15. Create another ellipse of Rx24 and Ry16.
16. Exit the sketcher environment using the Done button.
17. Choose Okay from the DIRECTION menu.
18. Enter the depth of cut.
19. Mirror the cut feature about the RIGHT datum plane.
20. Create a round feature on all edges except the edges enclosing the bottom planar surface of the base feature.
21. Invoke the Shell option from the menu bar. Remove the bottom face of the base feature.
22. Using the bottom face of the base feature as the sketching plane create the three protrusion feature that are the supporting structures for the screws. Extrude these features using the Extrude up to next surface button.
23. Using the Hole dashboard, create the hole in the protrusion feature create earlier.

Συστήματα CAD – Blend - Άσκηση

In this exercise, you will create the surface (or solid if you prefer) model shown in Figure 15-105. The orthographic views and the detailed view of the surface model are shown in Figure 15-106. Ignore the elliptic pocket. (Expected time: 55 min)

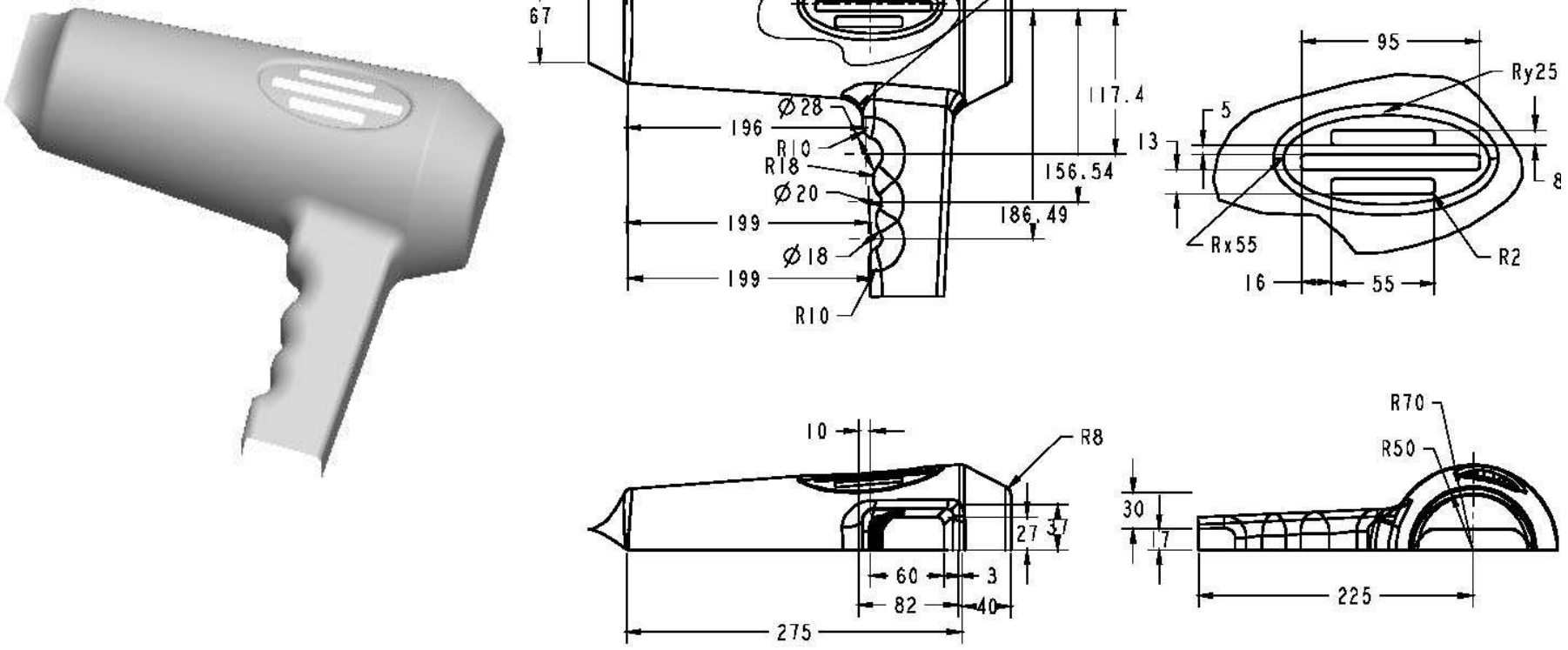


Figure 15-106 Top, front, right-side, and detailed views of the surface model

➢ Tutorial 1

In this tutorial, you will create the model shown in **Figure A**. This figure also shows the sectioned top, front, and right-side and isometric views of the model.

(Expected time: 45 min)

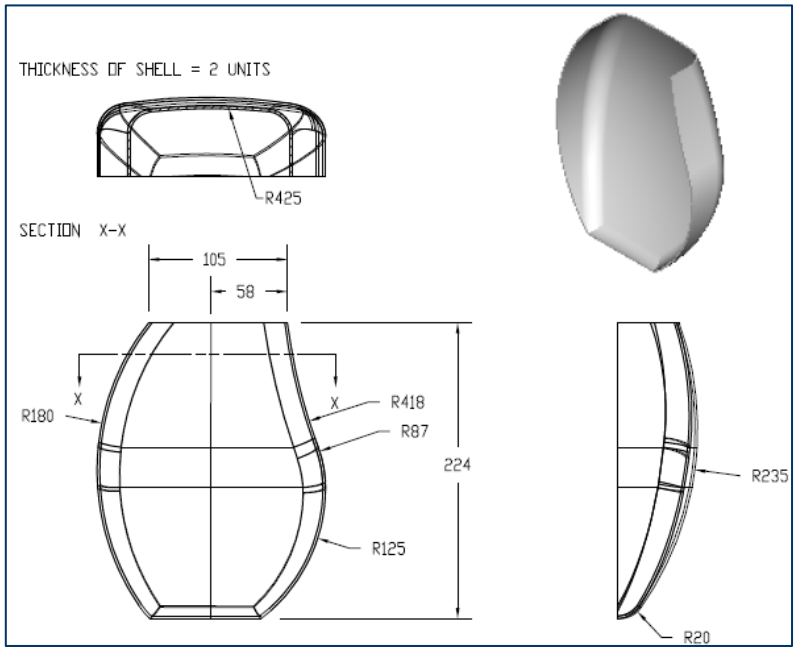


Figure B Sectioned-top, front, right- side, and isometric views of the model

The following steps are required to complete this tutorial:

1. The base feature is a variable section sweep feature. First, the origin trajectory will be sketched **(Figure B)**, and then the X trajectory will be sketched **(Figure C)**. Two additional trajectories will be sketched that will sweep the section along their paths **(Figure D)** and **(Figure E)**. Then the section that will vary with the shape of the trajectories will be sketched **(Figure F)**. These four trajectories will be used to create the closed section **(Figure G)** for variable section sweep feature. **(Figure H)**.

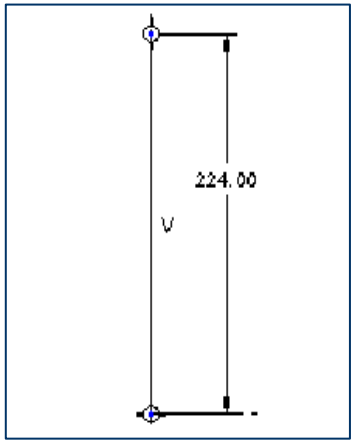


Figure B Sketch with dimension of the origin trajectory

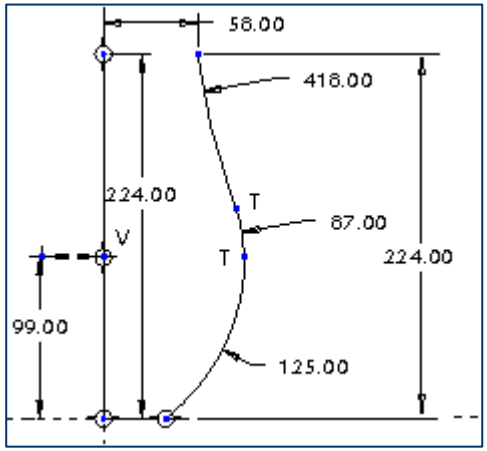


Figure C Sketch with dimensions of the X trajectory

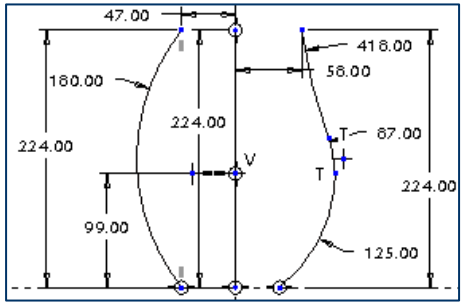


Figure D Sketch with dimension of the additional trajectory

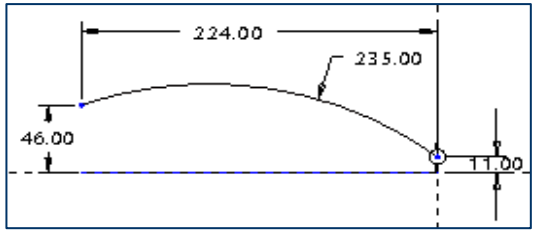


Figure E Sketch with dimension of additional trajectory

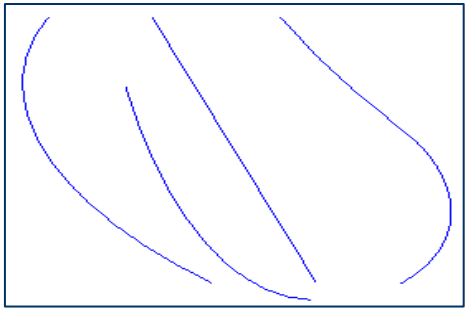


Figure F Default view of the four trajectories

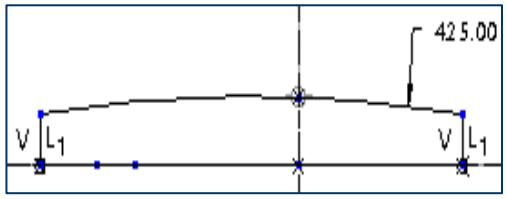


Figure G Sketch of the section with dimensions

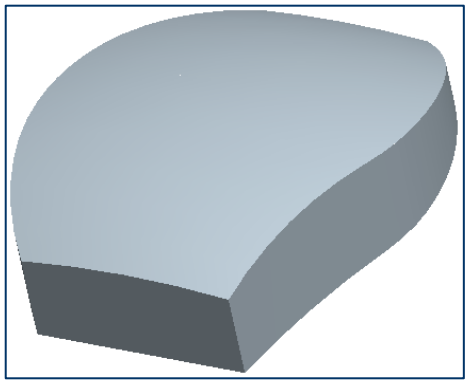


Figure H Variable section sweep feature

2. The second feature is a round **(Figure I** and **Figure J)**

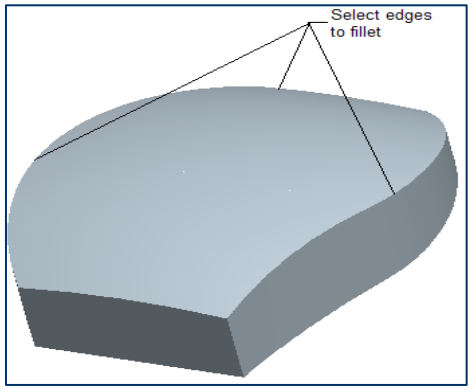


Figure I Edges to be selected to round

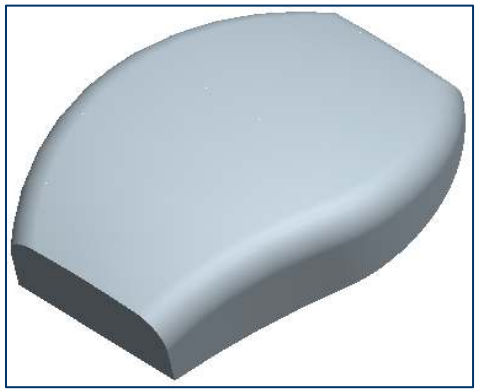


Figure J Model after creating round

3. The third feature is a shell of thickness 2 **(Figure K** and **Figure L)**

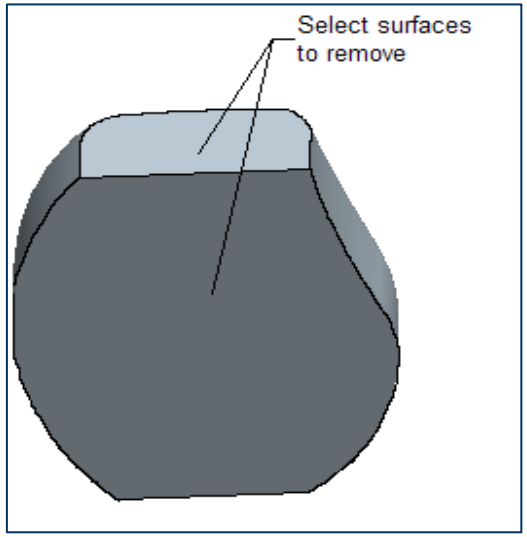


Figure K Surfaces to be selected to remove

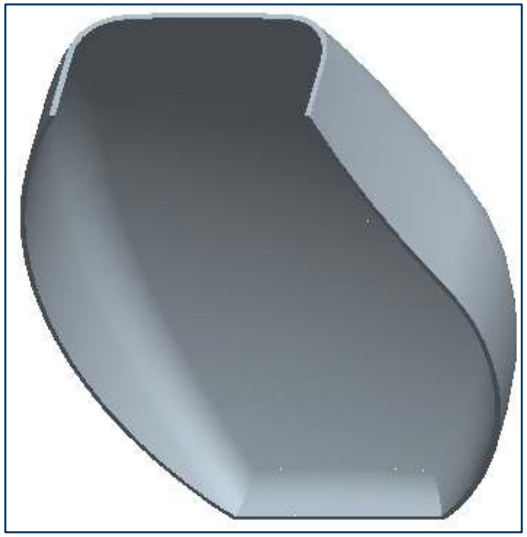


Figure L Final model after shelling

4. Save the model and close the window.

➢ Tutorial 2

In this tutorial, you will create the solid model of a Upper Housing of a motor blower assembly shown in **Figure A**. **Figure B** shows the left-side view, top view, front view, and the sectioned left-side view of the model.

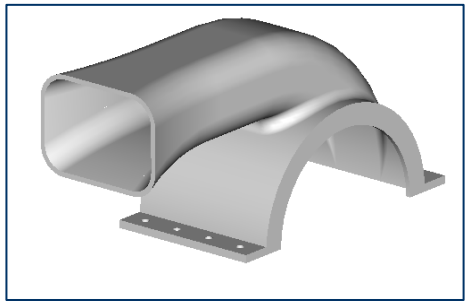


Figure A Solid model of the Upper Housing

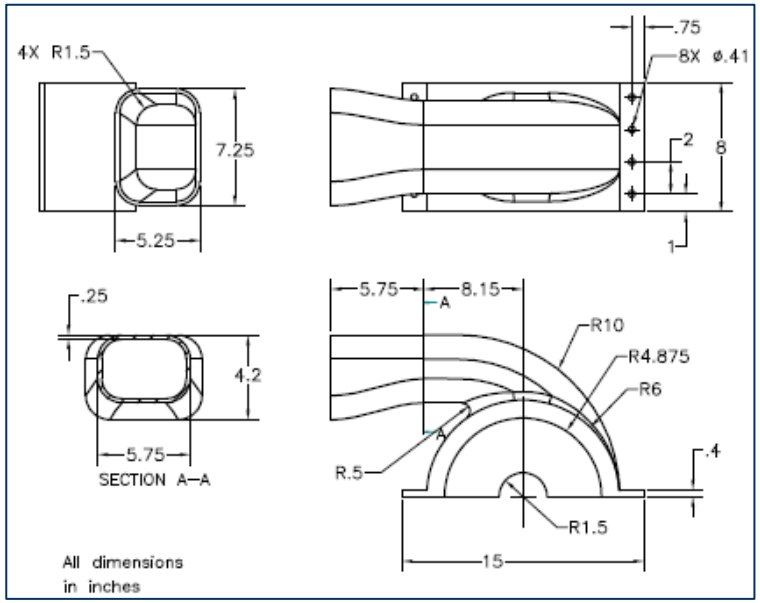


Figure B Left-side view of the top view, top view, front view, and the sectioned left-side view of the model

Συστήματα CAD – Swept Blend - Άσκηση

The following steps are required to complete this tutorial:

1. Examine the model and determine the number of features in it. The model is composed of ten features.

2. The base feature is an extruded feature. Select the sketching plane for the base feature, draw the sketch using the sketcher tools, and apply dimensions **(Figure C)**. Then extrude the feature to the given depth. **(Figure D)**

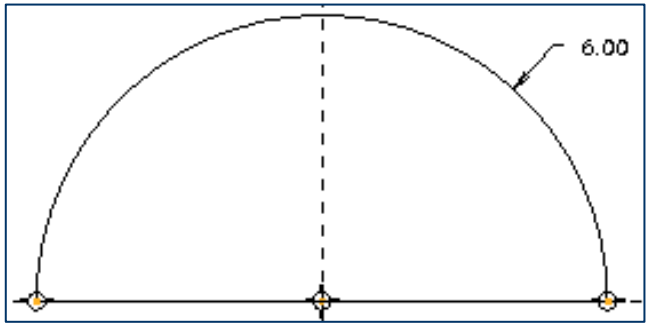


Figure C Sketch for the base feature

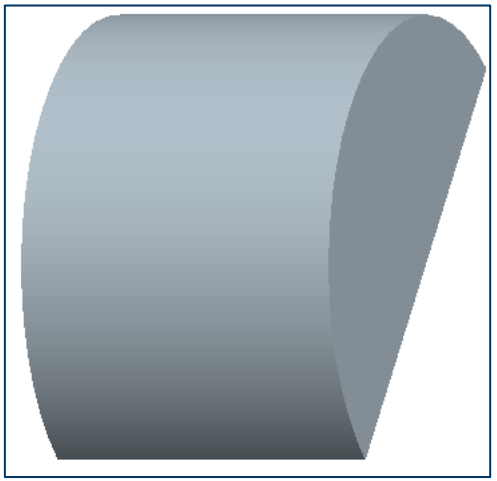


Figure D Default trimetric view of the base feature

Συστήματα CAD – Swept Blend - Άσκηση

3. The second feature is a swept blend feature in which the section is normal to the origin trajectory. First the sketch for the origin trajectory will be drawn and dimensioned (**Figure E**) and then three sections **(Figure F, G, and H)** will be defined along the origin trajectory to create the swept blend feature. **(Figure I)**

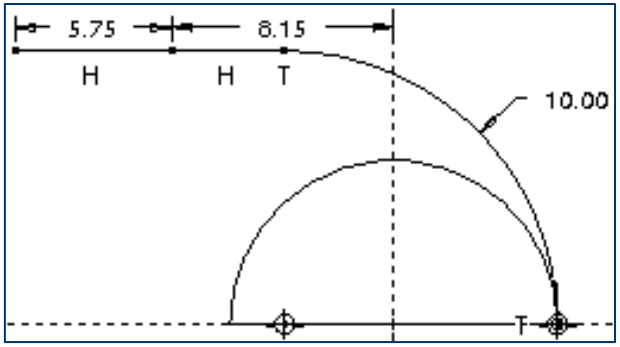


Figure E Sketch of the origin trajectory

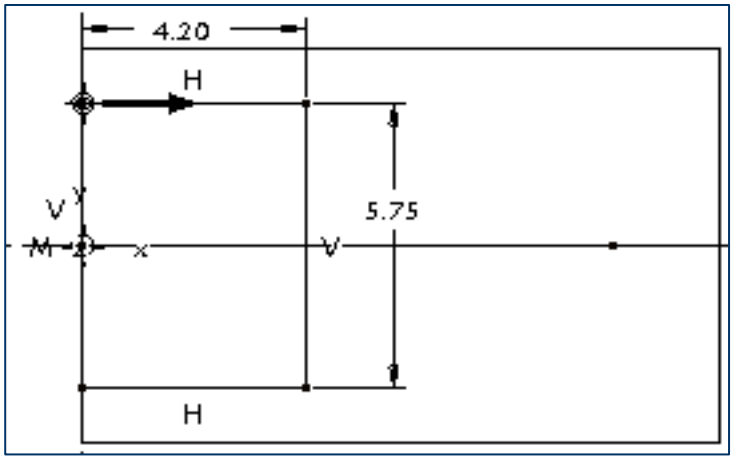


Figure F Sketch for the first section with dimensions

Συστήματα CAD – Swept Blend - Άσκηση

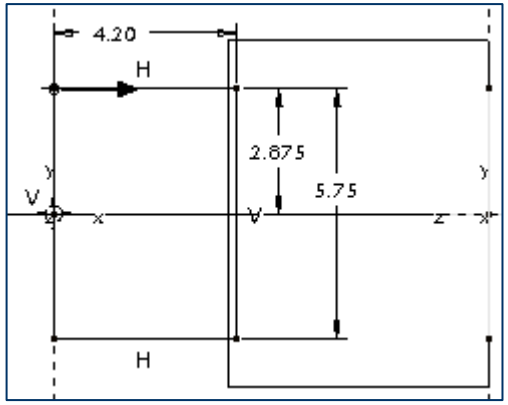


Figure G Sketch for the second section with dimensions

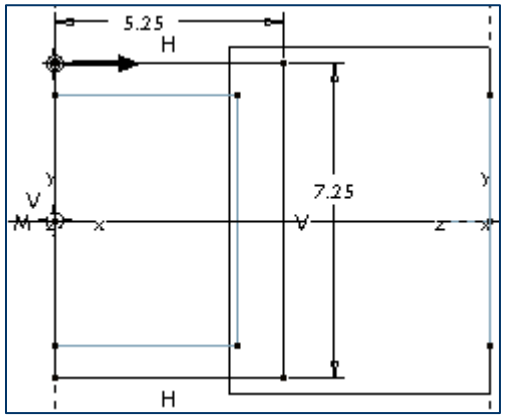


Figure H Sketch for the third section with dimensions

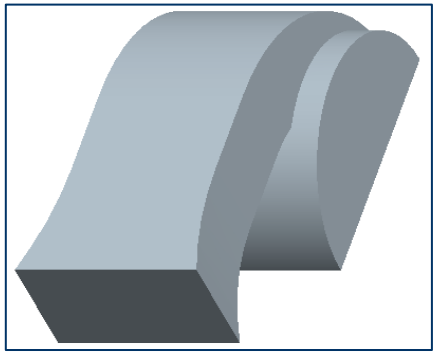


Figure I Default view of the swept blend

4. The third feature is a round of radius 1.5. **(Figure J** and **Figure K)**

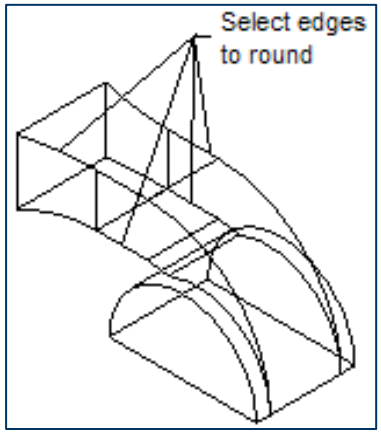


Figure J Edges to round

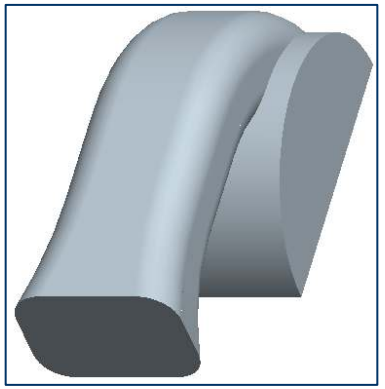


Figure K Default view of the model after creating a round feature of radius 1.5

5. The fourth feature is a round of radius 0.5. **(Figure L** and **Figure M)**

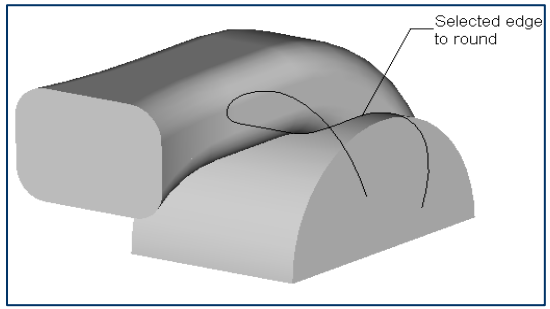


Figure L Edge selected to round

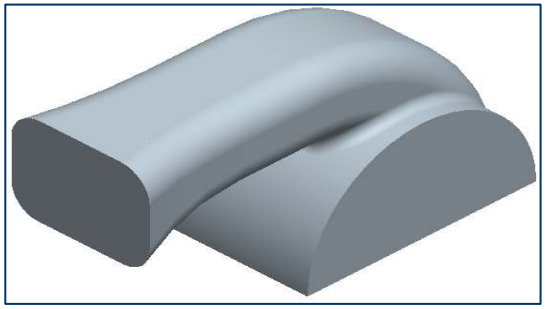


Figure M Model after creating a round feature of radius 0.5

6. The fifth feature is a shell of thickness 0.25 that will be created on the front planar surface of the swept blend feature and on the bottom planar surface of the base feature. (**Figure N**)

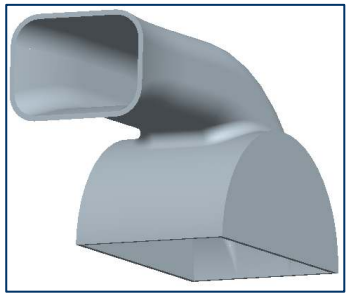


Figure N The model after shelling

7. The sixth feature is an extruded cut. Select the sketching plane for the cut feature, draw the sketch using sketching tools, and apply the dimensions **(Figure O)**. Extrude the sketch to the given distance. **(Figure P)**

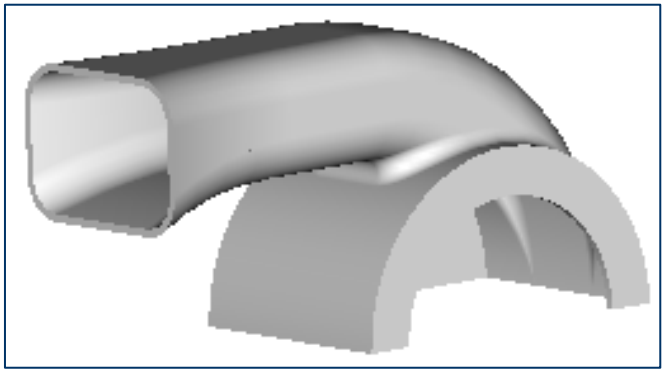


Figure O Sketch with dimension for the cut feature

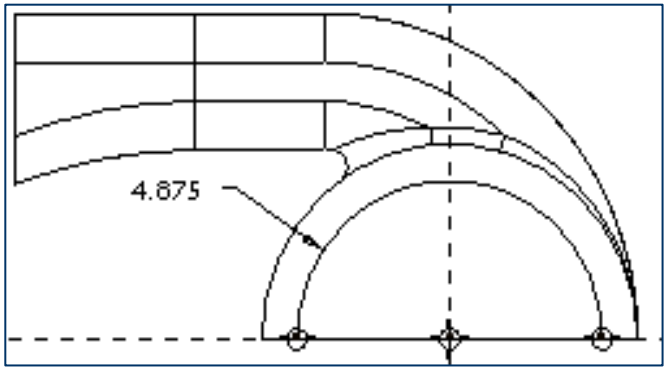


Figure P Model with the cut feature

8. The seventh feature is also an extruded cut. Select the sketching plane for the cut feature, draw the sketch using sketching tools, and apply the dimensions **(Figure Q)**. Extrude the sketch to the given depth. **(Figure R)**

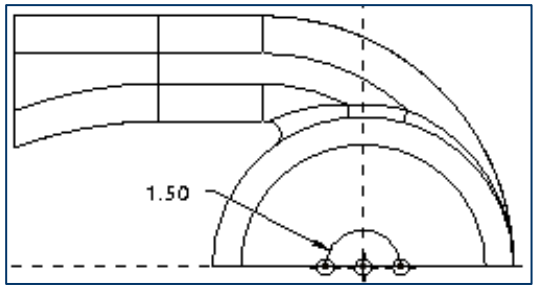


Figure Q Sketch with dimension for the cut feature

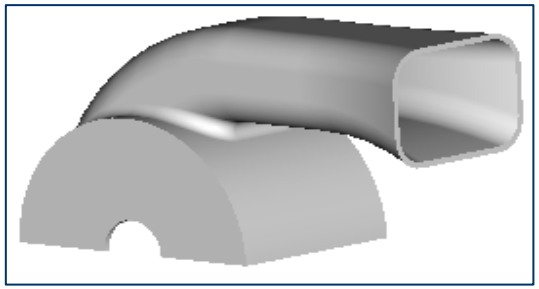


Figure R Model with the cut feature

9. The eighth feature is an extruded feature. Select the sketching plane for the extruded feature, draw the sketch, and apply constraints and dimensions (**Figure S**). Extrude the sketch to the given depth. **(Figure T)**

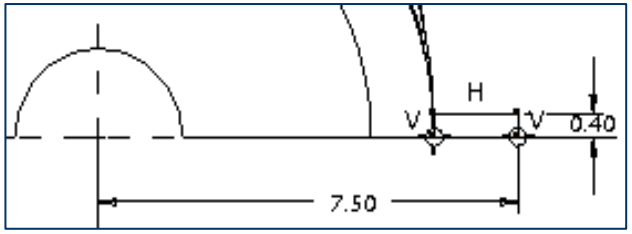


Figure S Sketch for the extruded feature

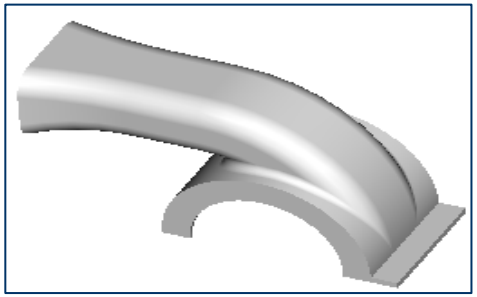


Figure T Model after creating the extrude feature

10. The ninth feature is a copy of the eighth feature. **(Figure U)**

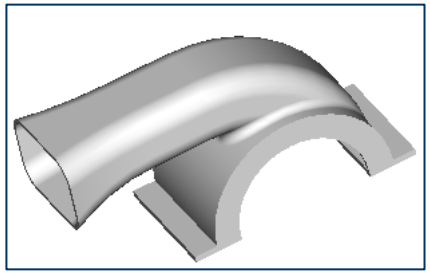


Figure U Model after mirroring the extrude feature

Συστήματα CAD – Swept Blend - Άσκηση

11. The tenth feature is a hole **(Figure V)**, and this hole will be patterned. After you pattern the hole **(Figure W)**, the hole and the pattern feature will become a single feature.

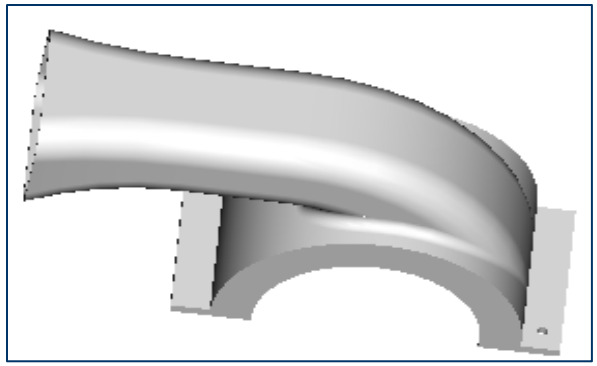


Figure V Model after creating the hole

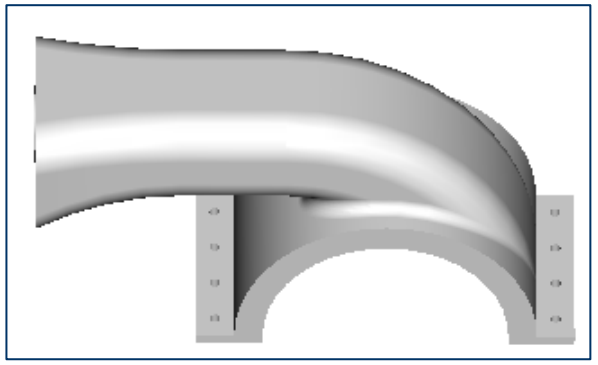


Figure W Hole patterned on the extruded features

12. Save the model and close the window.