Exercise 7

In this exercise we will build a surface for a jug. The completed part is shown below.

Set the working directory to the EXAMPLES folder and open the model JUG.PRT

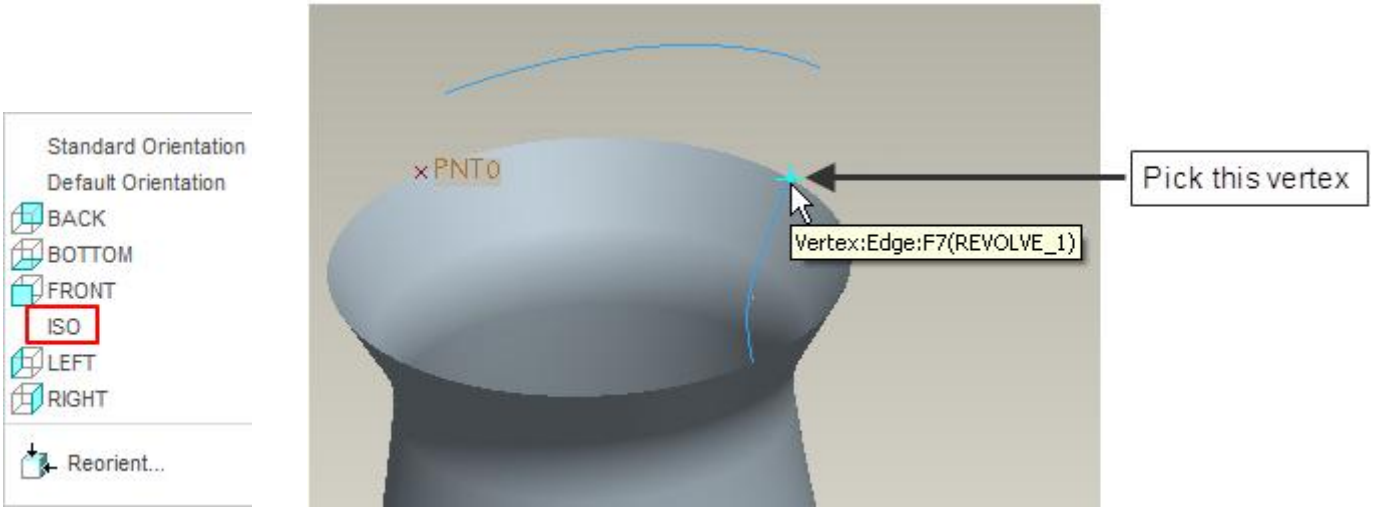
Because this part is symmetrical so we will build the surface only on one half of the part and then mirror it.

Creating the datum curves

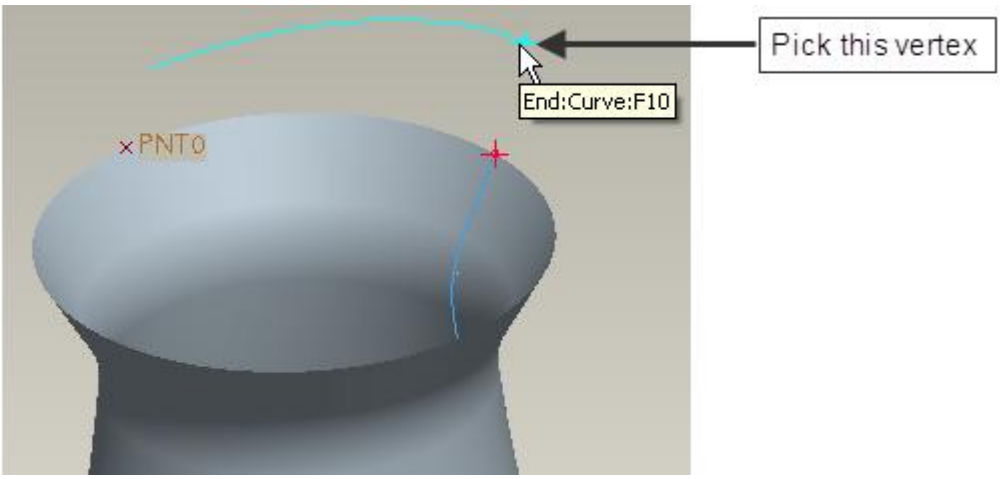
Before creating the desired surface, first we will create datum curves which will be used to construct the surface. We will create the datum curves using Through Points option

Pick Curve through Points on the Model tab to invoke Curve through Points tool.

Pick the following vertex as first point for curve.



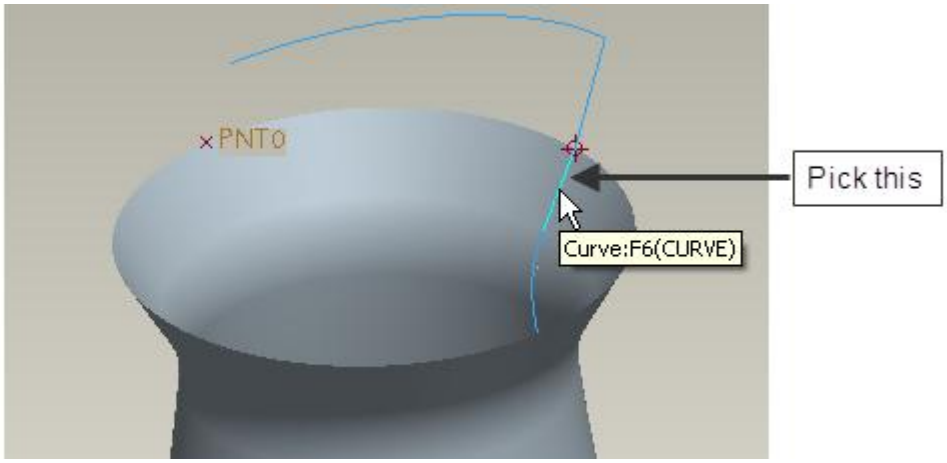
Pick the end of datum curve as last point for new curve as shown in the figure below.



Now we will specify the tangent constraints for the ends of the curve.

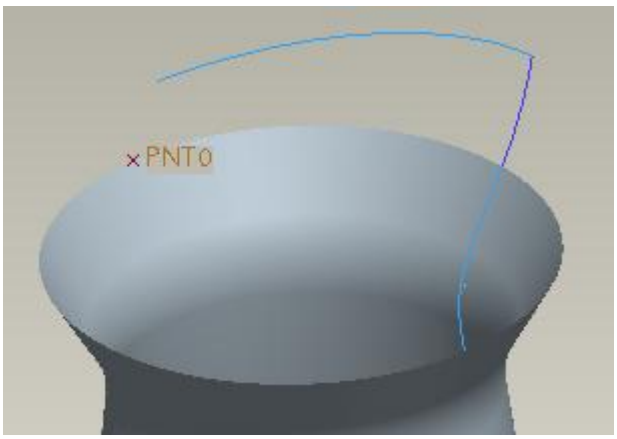
So pick Ends Condition tab and change the End condition for Start Point to Tangent.

Now system wants to know a reference to which the new curve will be tangent, so pick the curve shown in figure below.



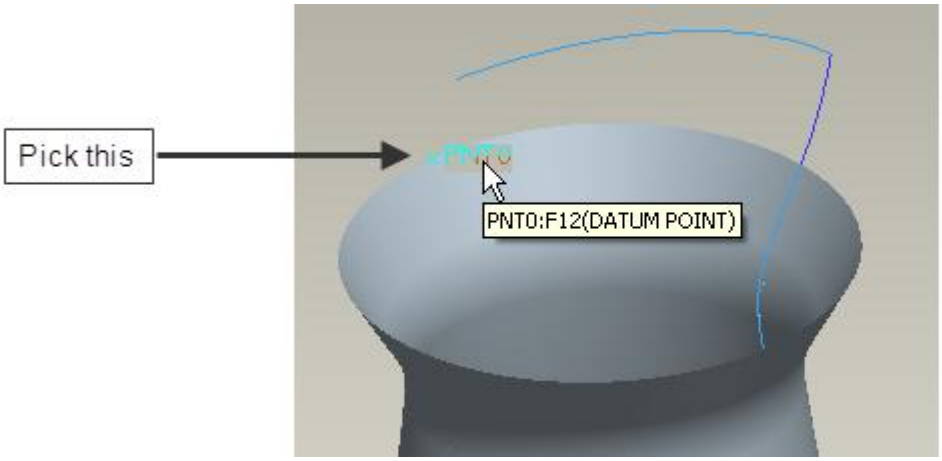
If you look at the end of curve, you will notice that it is twisted. It is due to the wrong direction of normal constraint. So pick Flip to reverse the direction. (Note: if the datum curve is not twisted in your case you do not need to flip.)

Pick to complete the feature. The datum curve will appear as shown below.

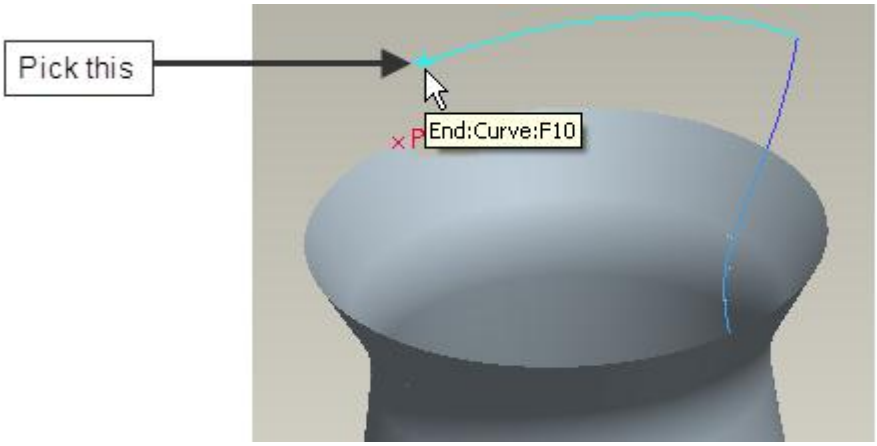


Now we will create second curve so again pick Curve through Points on the Model tab.

Pick the following datum point as first point for curve.

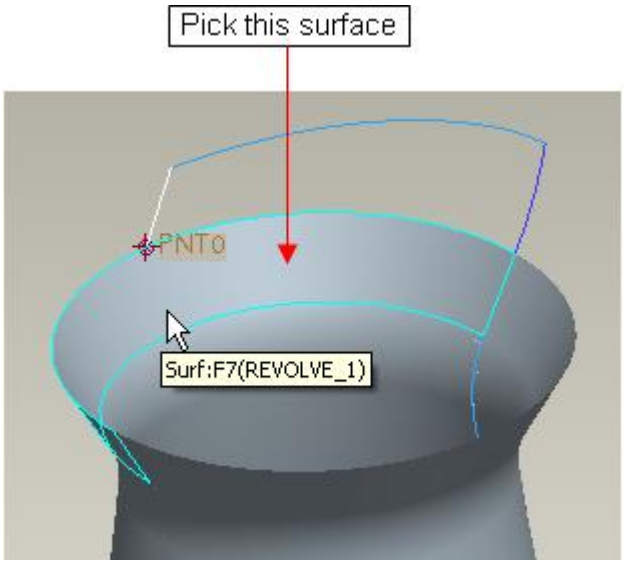


Pick the end of curve as last point for curve as shown in the figure below.

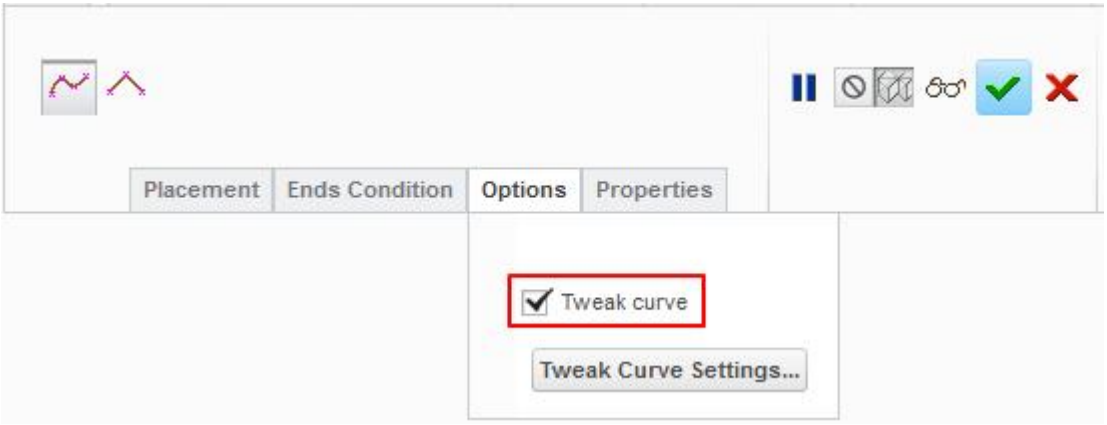


Now we will specify the tangent constraints for the ends of the curve.

So pick Ends Condition tab and change the End condition for Start Point to Tangent and select the following surface as reference.

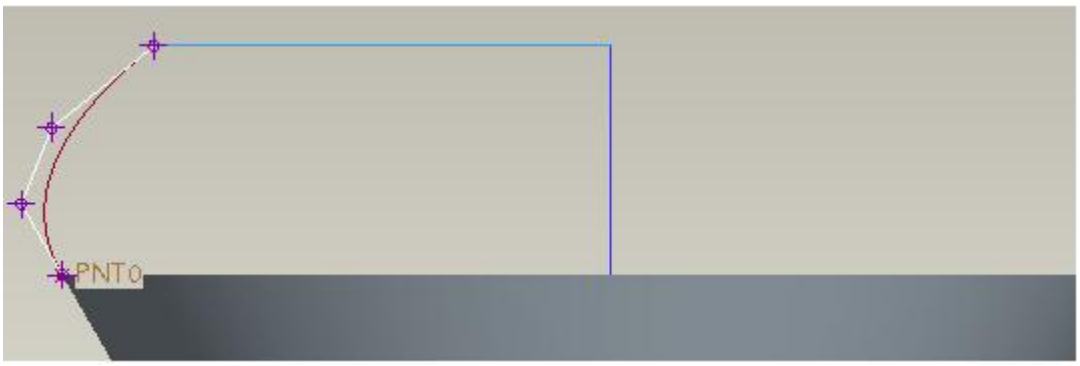


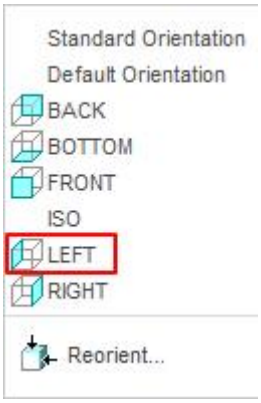
Now we will tweak this curve so check **Tweak curve** option in the Options panel as shown below.



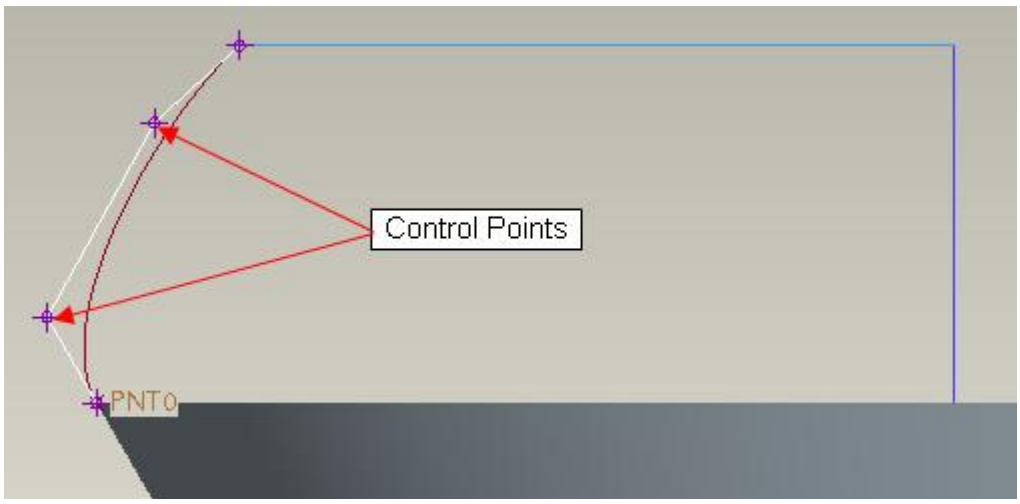
Pick Tweak Curve Settings... and Modify Curve dialog box will appear.

Set the view to LEFT and curve will appear as shown below.



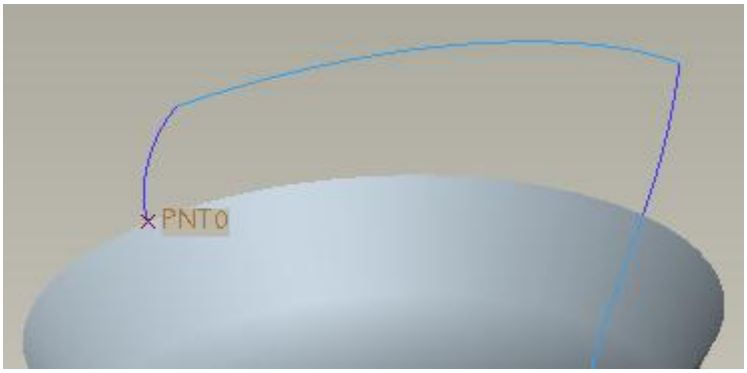


Now move the control points by dragging with mouse. After moving the control points, the curve should appear approximately as shown in the figure below. (Note: shape of the curve is not important for this exercise; purpose is to just introduce the functionality)



Pick OK to apply changes and quit Modify Curve dialog box after tweaking the curve.

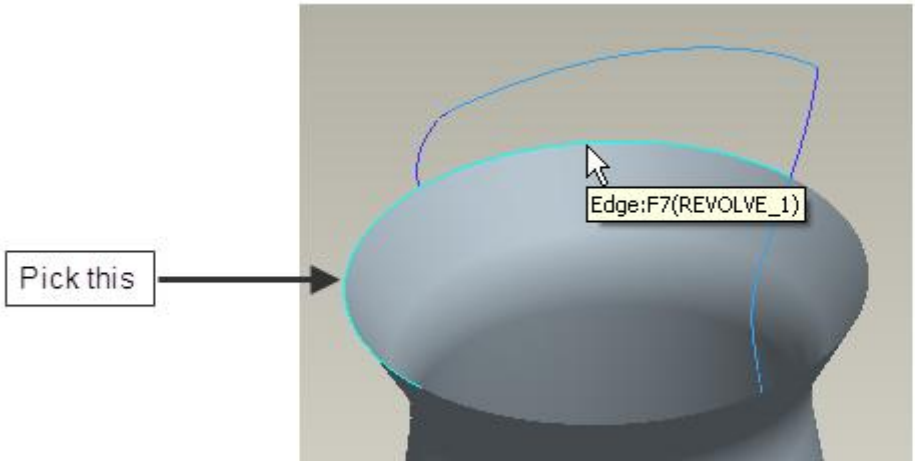
Pick ✔ to complete the curve feature. The datum curve will appear as shown below.



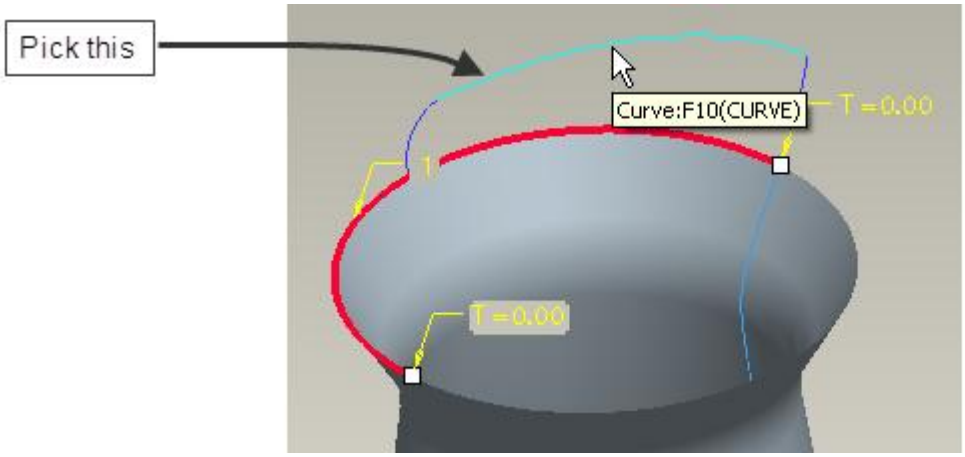
Creating the surface

Now we will create surface with Boundary Blend Tool.

Pick and select the edge shown in the figure below.

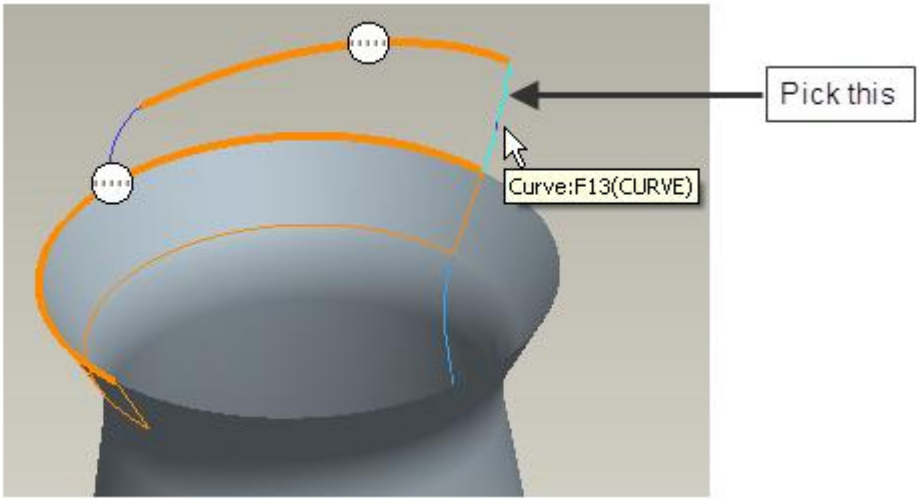


Now hold down the Ctrl key and pick the following curve as second chain.

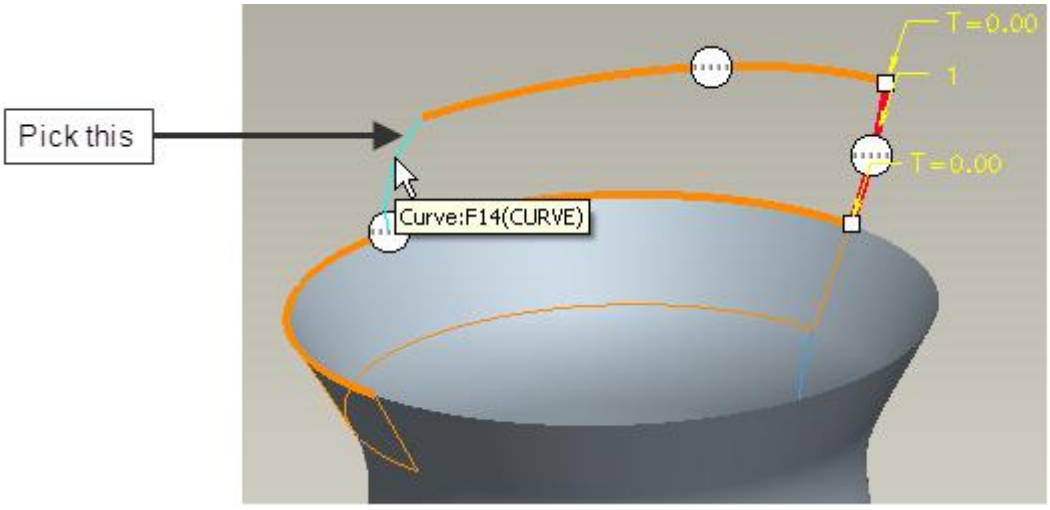


Pick in the **Second Direction Collector** to activate it.

Pick the datum curve, shown in the figure below (preview off) as first chain in second direction.

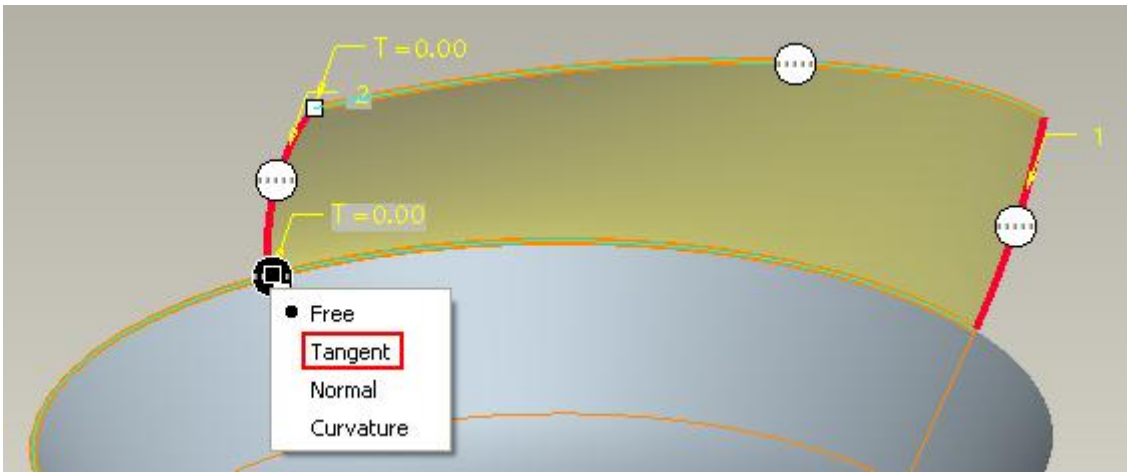


Now hold down the Ctrl key and pick the datum curve, shown in the figure below as second chain in second direction.

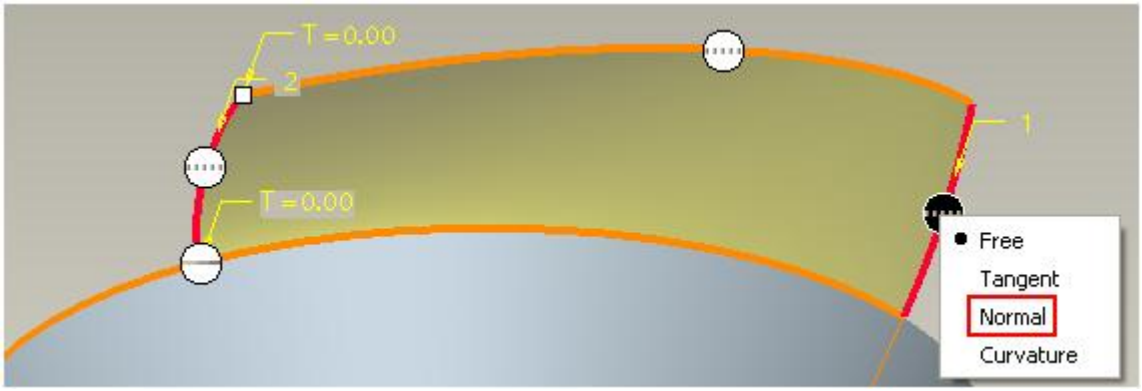


Now we will apply the boundary constraints to surface.

Apply the **Tangent** constraint for the boundary chain adjacent to the existing surface (i.e. first chain in first direction) as shown below.



Apply the **Normal** constraint for first chain in second direction as shown below.

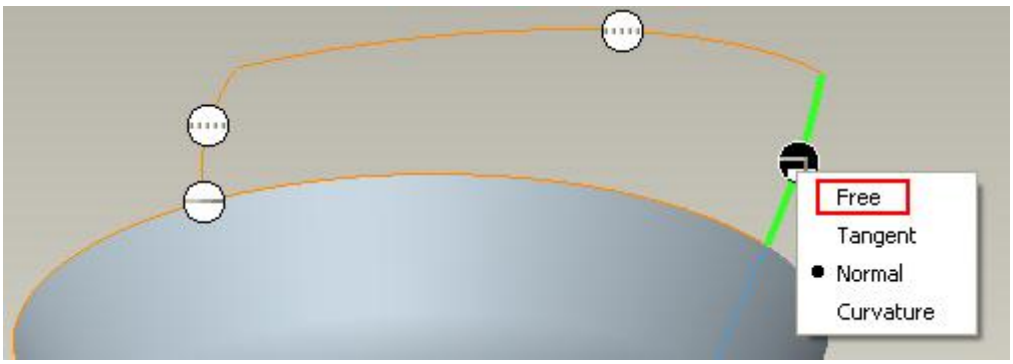


Notice that system is unable to apply the Normal constraint for this chain and displays the following message

Select a surface that lies on the highlited boundary component.

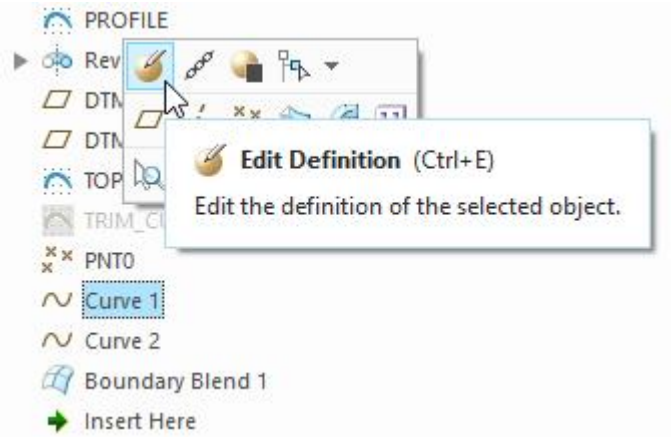
It is because system does not know to which reference (a surface or datum plane) our created surface should be normal to. There are two ways to fix this issue; one is to select the desired reference during the surface definition and the second is to redefine the underlying datum curve and constrain it to the desired surface or plane. In this exercise we will use the second approach.

So again right-click over the constraint balloon and apply the **Free** constraint as shown below.



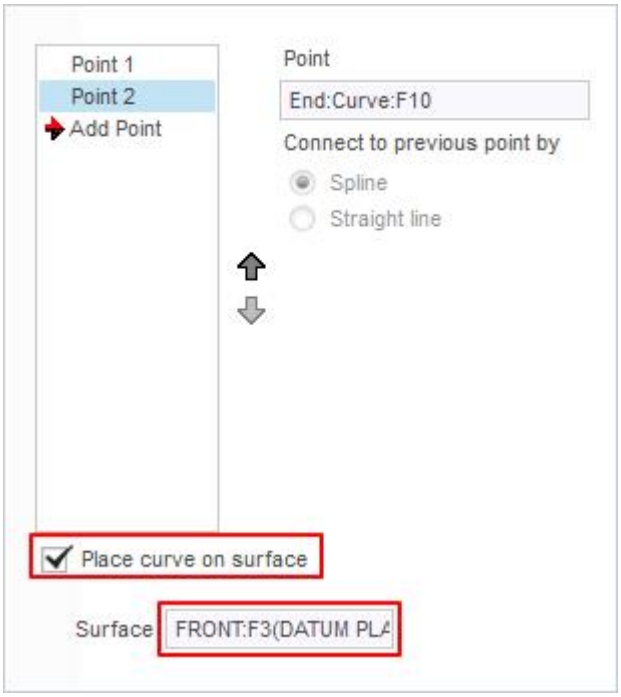
Pick  to apply the changes and exit the dashboard.

Select the first “Thru Points” datum curve feature (Curve 1) and pick  as shown in figure below.



Pick Placement to access the Placement panel and check the 'Place curve on surface' option.

As the Surface collector becomes active so select the **FRONT** datum plane as reference.

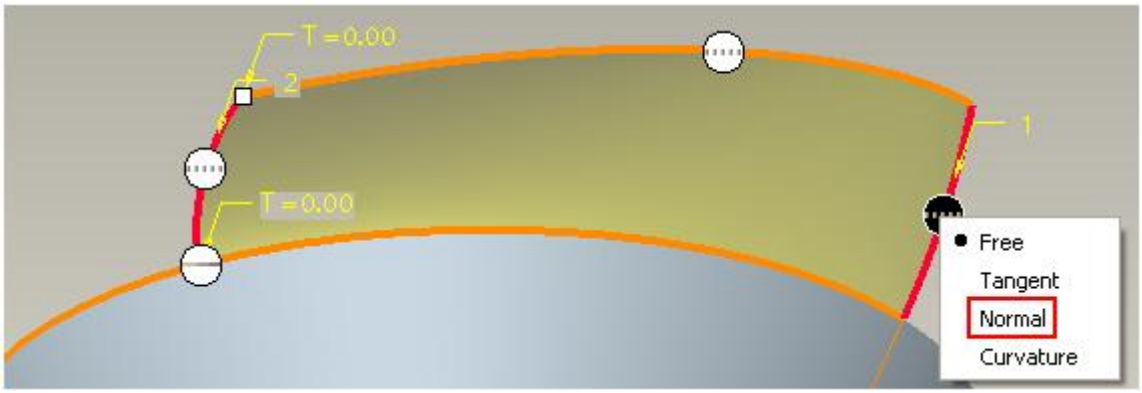


'Place curve on surface' option is used to create a spline curve that passes through the selected points and lies on a specific surface patch.

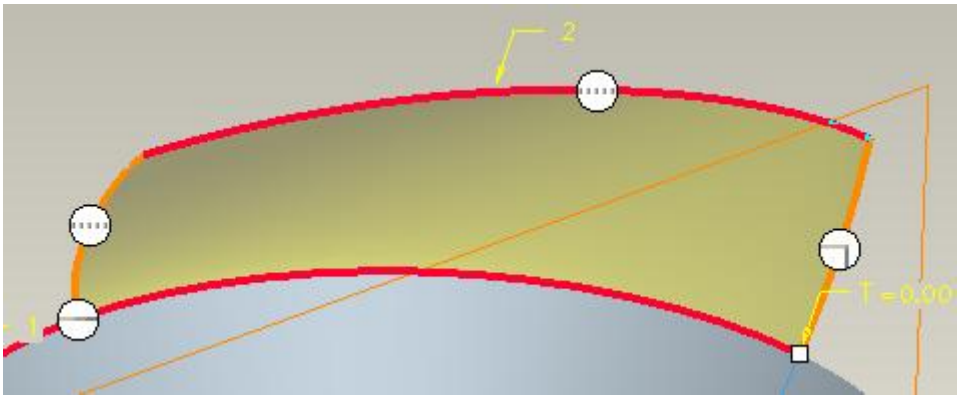
Pick  to apply the changes and exit the dialog box.

Select the Boundary Blend feature and pick  .

Apply the **Normal** constraint for first chain in second direction as shown below.



Now the preview will appear as shown below.



This surface will be mirrored about FRONT datum plane. Therefore to maintain tangency between original and mirrored surface, the surface has been constrained Normal to the FRONT datum plane.

To make the mirrored and parent surface tangent to each other we need to make the parent surface normal to mirror plane.

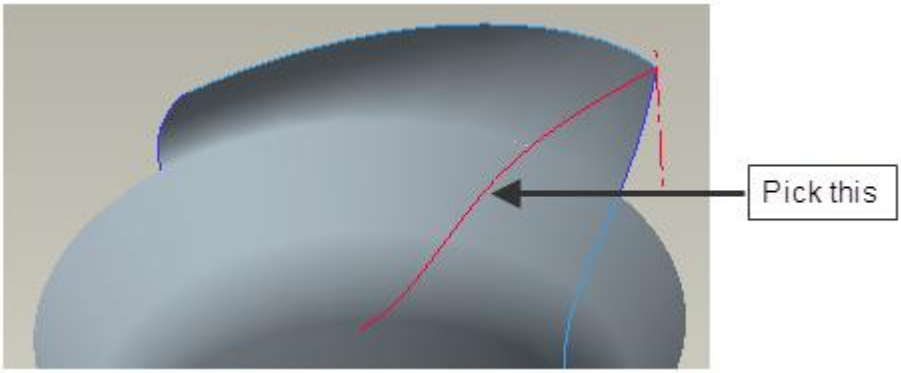
Pick icon or middle-click to complete the feature.

Trimming the surface

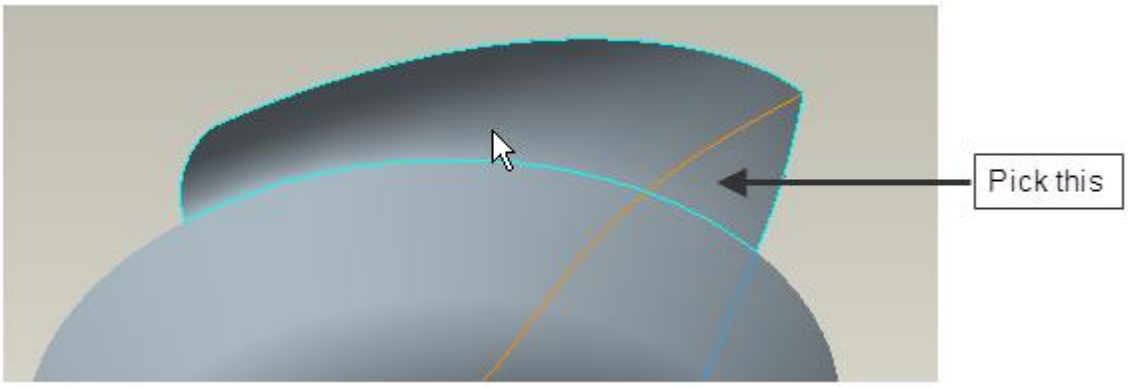
Now we will trim the unwanted part of the created surface. For this purpose we have already created a sketched datum curve but it is hidden.

So right-click the “TRIM_CURVE” feature in the model tree and pick **Show**.

Select the datum curve on screen as shown below.

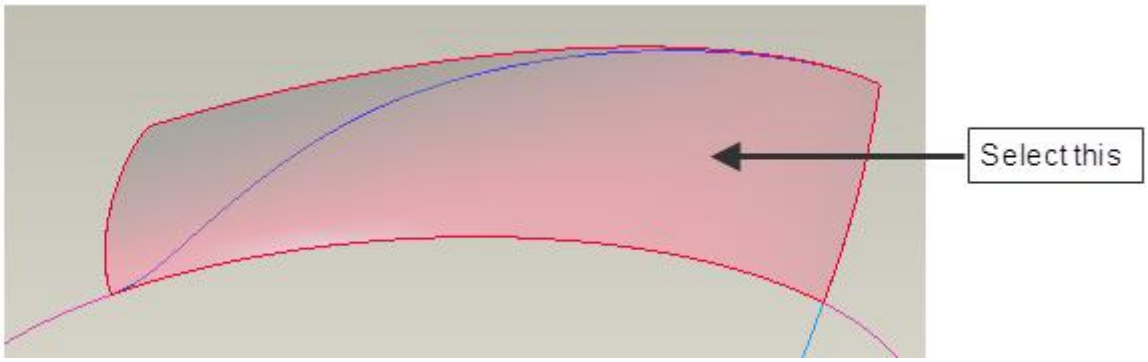


Pick Project on the Model tab to invoke the Project tool and select the boundary blended surface to project selected curve onto.



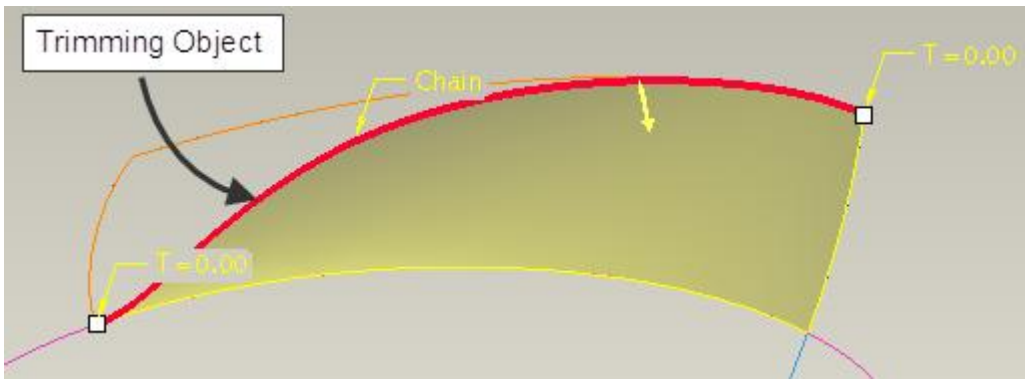
Pick to complete the feature.

Now we will trim the quilt with the curve. So first select the quilt.



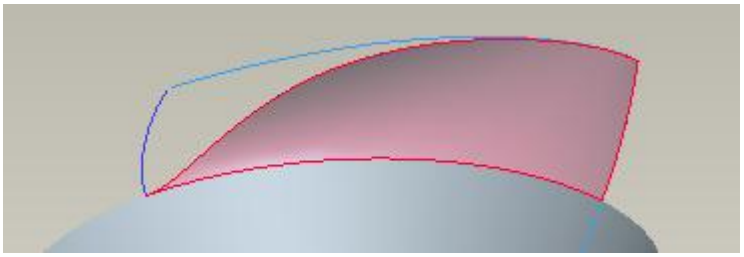
Pick Trim on the Model tab to invoke Trim tool.

Select the projected datum curve as trimming object and preview will appear as shown below.



Pick  or middle-click to complete the feature.

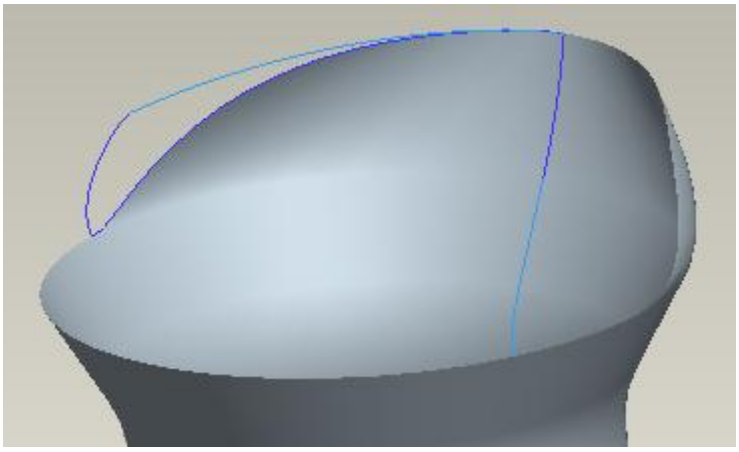
Now we will mirror the quilt so pick the trimmed quilt as shown below.



Pick 

on the Model tab to invoke Mirror tool and select the **FRONT** datum as Mirror plane.

Pick  to complete the feature. Part will appear as shown below.



Now you can merge surfaces together to form a quilt.

Select **File > Save** to save the work done so far.

Exercise 8

In this exercise we will create surface for the scoop of a car.

In the following figure (Source: Wikipedia) a car is shown with a scoop on its bonnet.

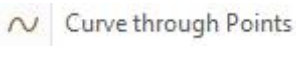
We will not create all details of the bonnet instead we will only create scoop surface by using a simple surface as base.

Set the working directory to the EXAMPLES folder and open the model SCOOP.PRT

Because this part is symmetrical so we will create half side of the part and then mirror it.

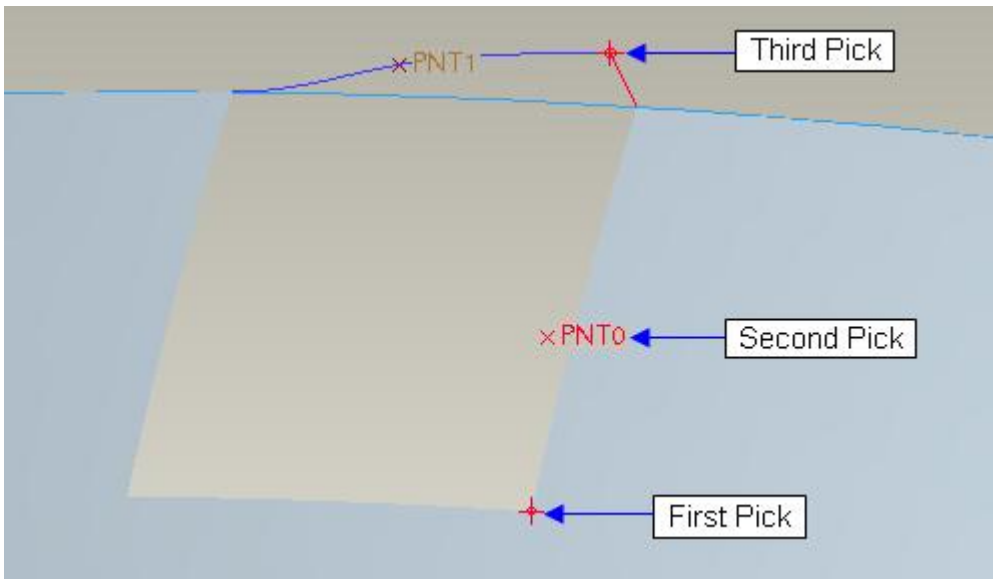
Creating the datum curve

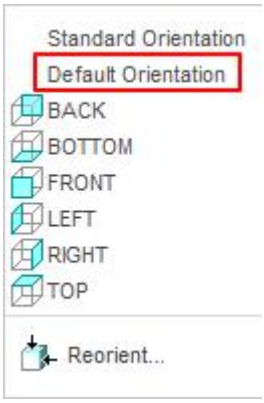
Before creating the desired surface, first we will create a datum curve which will be used to construct the surface. We will create the datum curve using Through Points option

Pick 

on the Model tab to invoke Curve through Points tool.

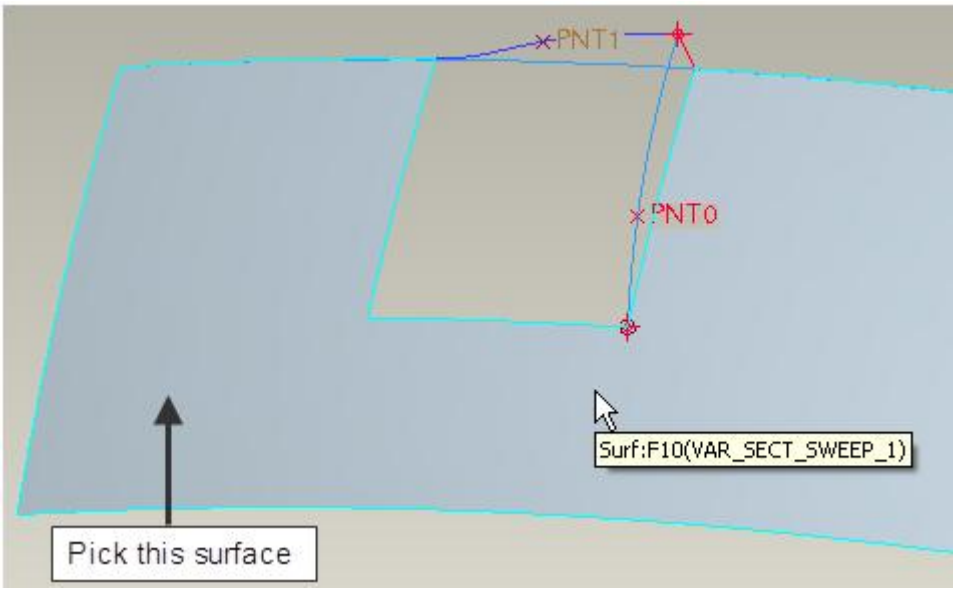
Pick the following vertices/point in the order shown. (Set the view to Default Orientation to avoid any confusion)





Pick Ends Condition tab and change the End condition for Start Point to Tangent.

Now system wants to know a reference to which the new curve will be tangent, so select the following surface as reference.

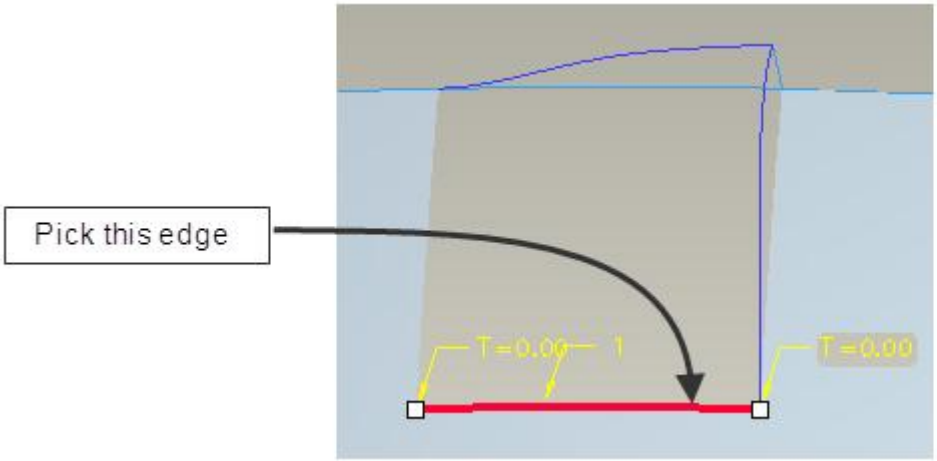


Pick  to complete the feature.

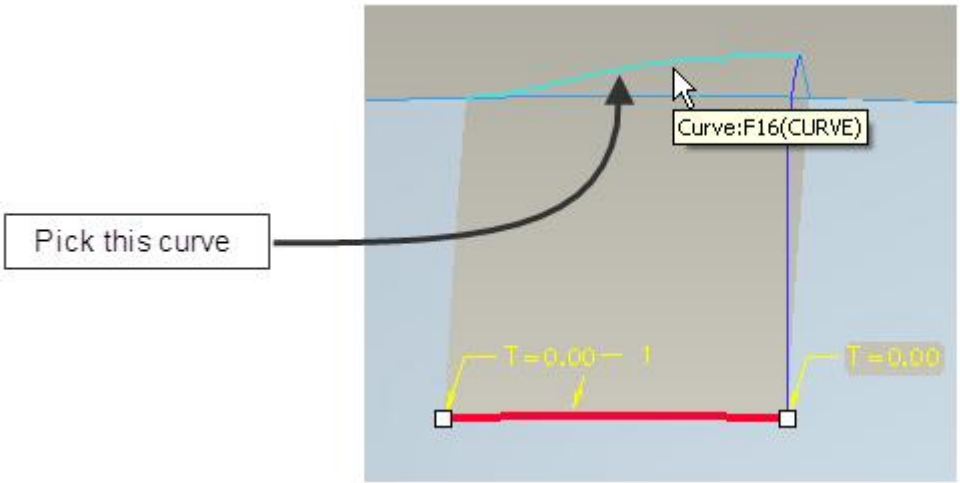
Creating the surface

Now we will create surface with Boundary Blend Tool.

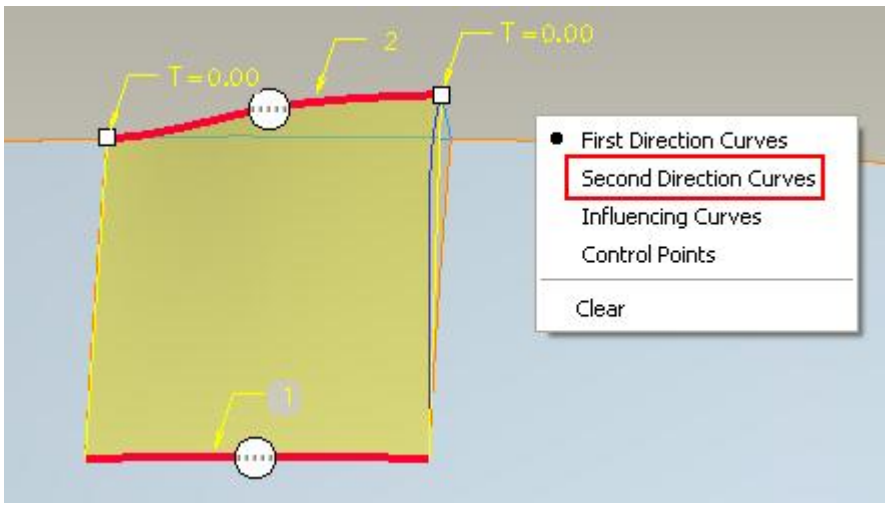
Pick  and select the edge shown in the figure below.



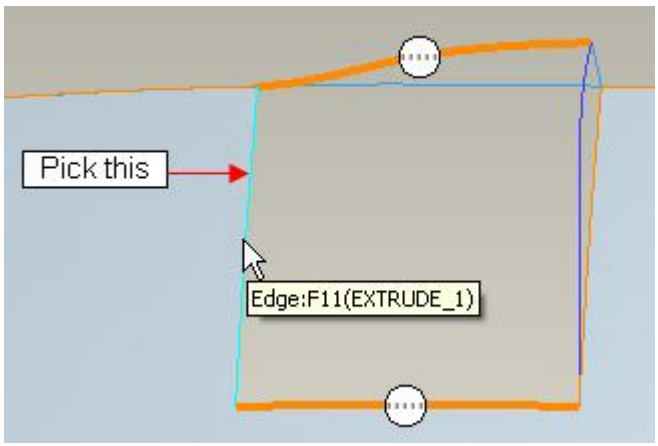
Now hold down the Ctrl key and pick the following curve as second chain.



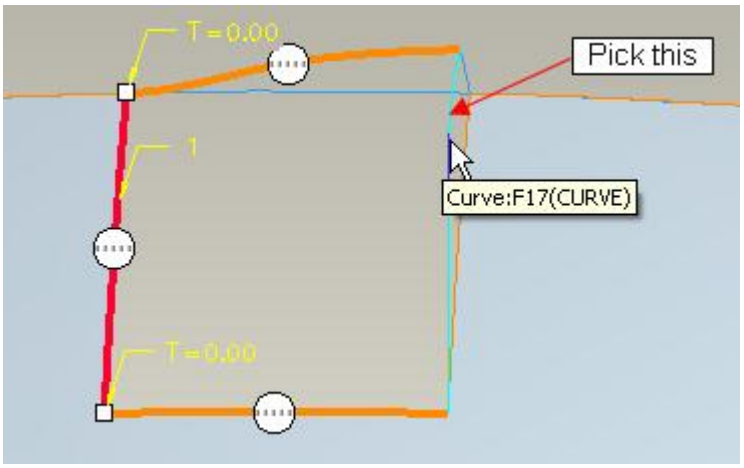
Right-click in the graphics window and select **Second Direction Curves** in the short-cut menu as shown below.



Pick the edge, shown in the figure below (dynamic preview off) as first chain in second direction.

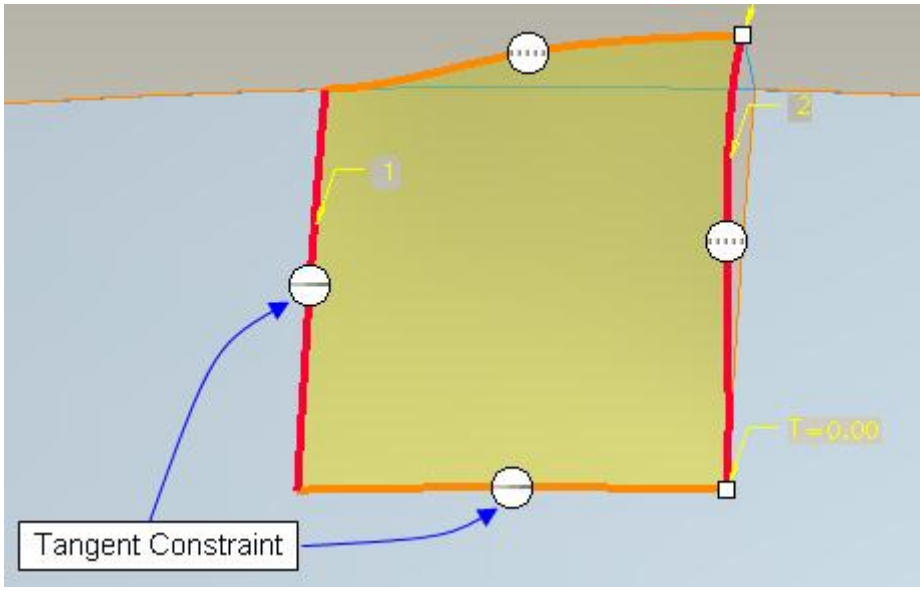


Now hold down the Ctrl key and pick the datum curve, shown in the figure below as second chain in second direction.

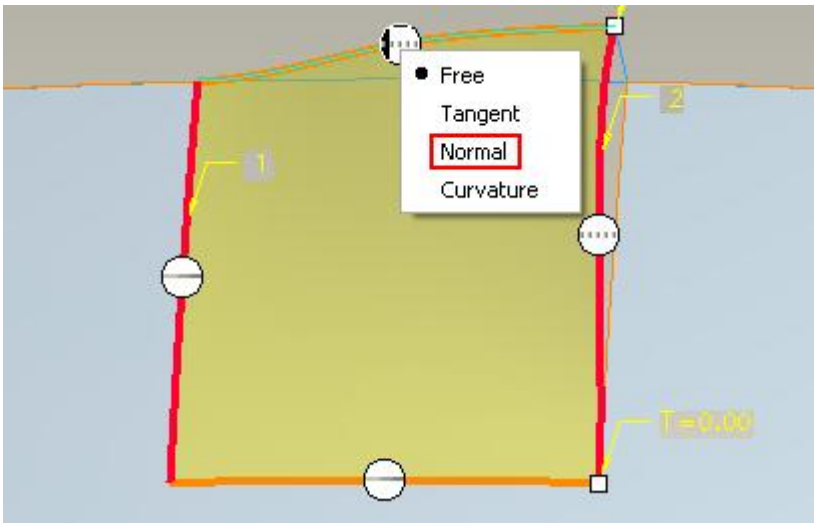


Now we will apply the boundary constraints to surface.

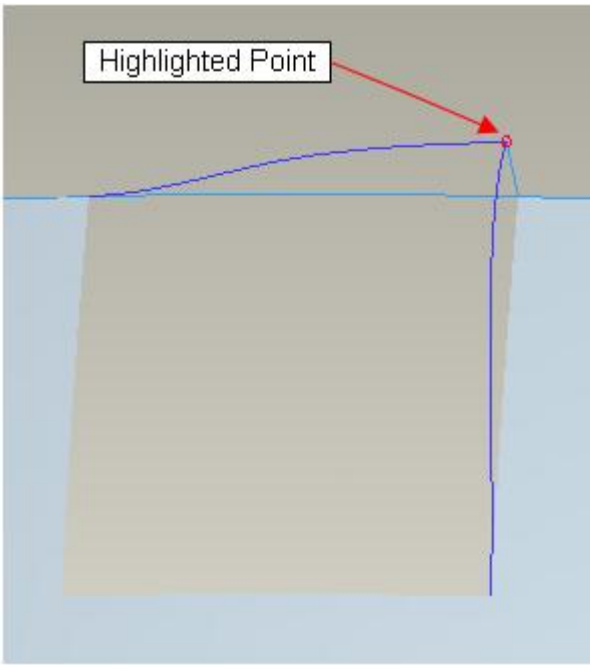
Apply the **Tangent** constraint for the chains shown in the figure below.

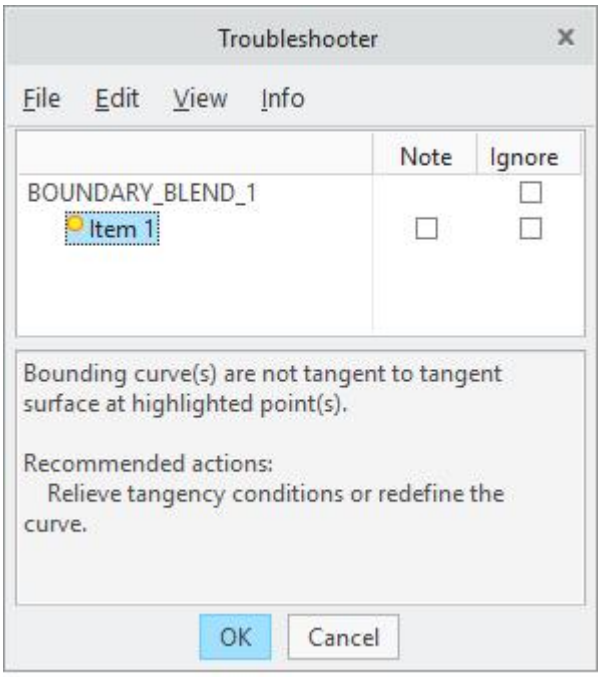


Apply the **Normal** constraint (to make surface normal to FRONT datum plane) for second chain in second direction as shown below.



Pick  to verify the surface and Troubleshooter dialog box will appear. Pick “Item 1” in the dialog box and system will highlight the troubled point as shown below.





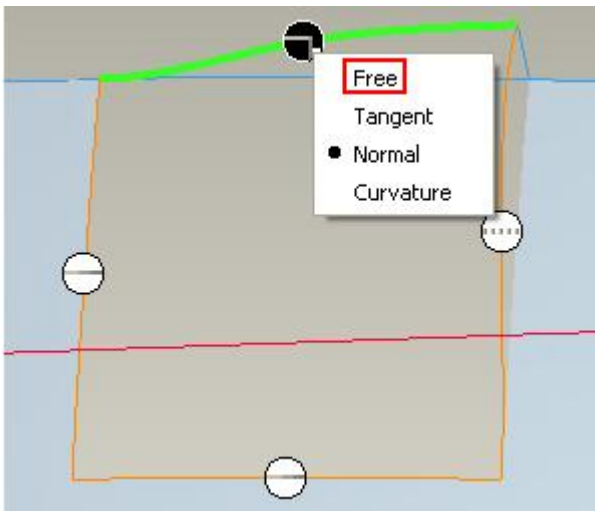
It is because the datum curve that we just created is not normal to the FRONT datum plane.

For a surface to be normal or tangent to a reference, underlying datum curves must be normal or tangent respectively to desired reference.

So first we will make the datum curve normal to the FRONT datum plane and then apply the Normal constraint to surface.

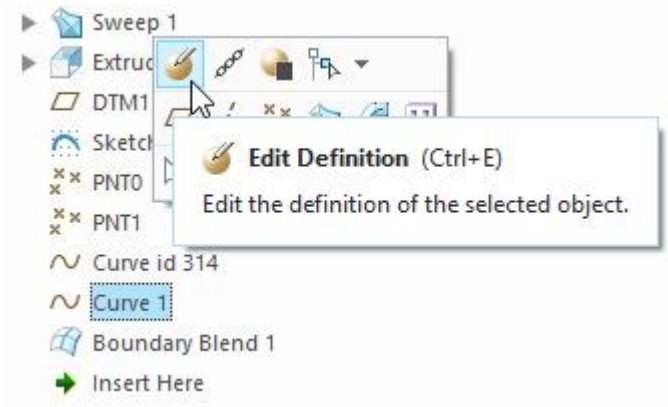
Now close Troubleshooter dialog box and pick  to Resume the boundary blended surface.

Again right-click over the constraint balloon and apply the **Free** constraint as shown below.



Pick  to apply the changes and exit the dashboard.

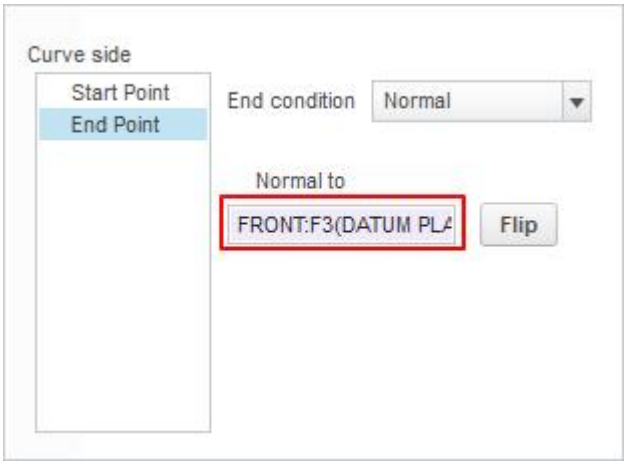
Select the datum curve feature and pick  .



Pick 

tab and change the End condition for 'End Point' to Normal.

Now system wants to know a reference to which the new curve will be normal, so select the **FRONT** datum plane as reference



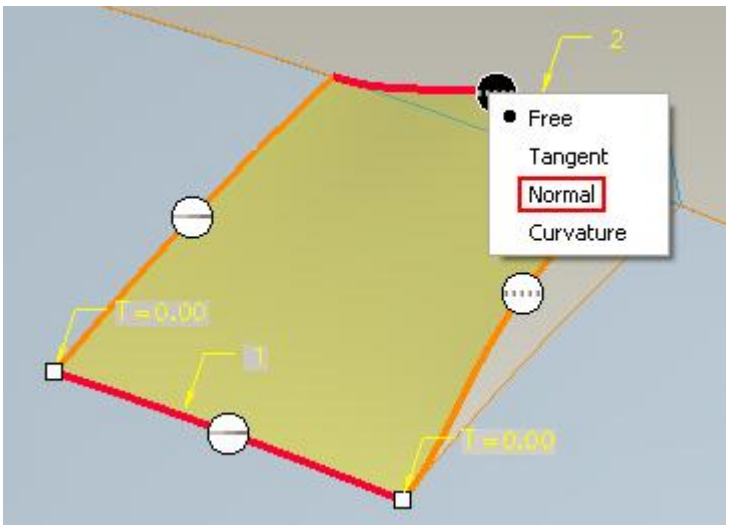
If you look at the end of curve, you will notice that it is twisted. It is due to the wrong direction of normal constraint. So pick 

to reverse the direction.

Pick  to complete the feature.

Select the Boundary Blend feature and pick  .

Apply the **Normal** constraint for second chain in second direction as shown below.



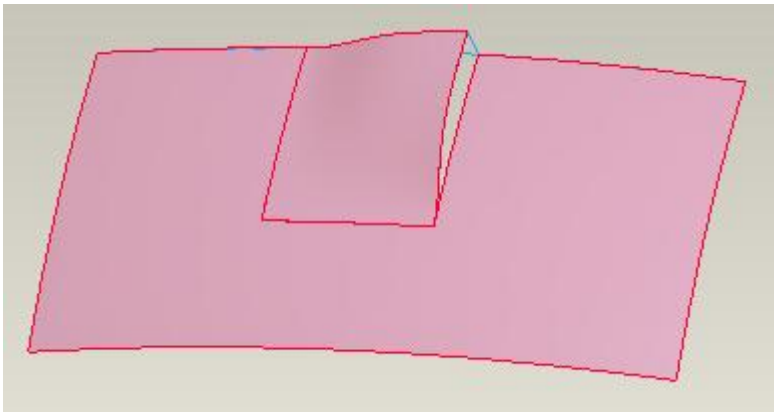
This surface will be mirrored about FRONT datum plane. Therefore to maintain tangency between original and mirrored surface, the surface has been constrained Normal to the FRONT datum plane.

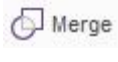
To make the mirrored and parent surface tangent to each other we need to make the parent surface normal to mirror plane.

Pick  icon or middle-click to complete the feature.

Merging the Quilts

Now we will merge the surfaces together. So select both surfaces while holding down Ctrl key as shown below.



Pick 

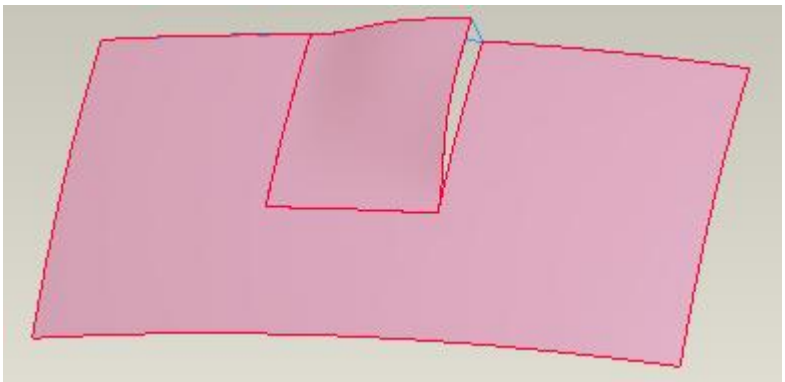
on the Model tab to invoke Merge tool.

Pick Options tab and change the merge type to **Join**.

Join option is used when two surfaces have a common edge. Join option is faster than Intersect option.

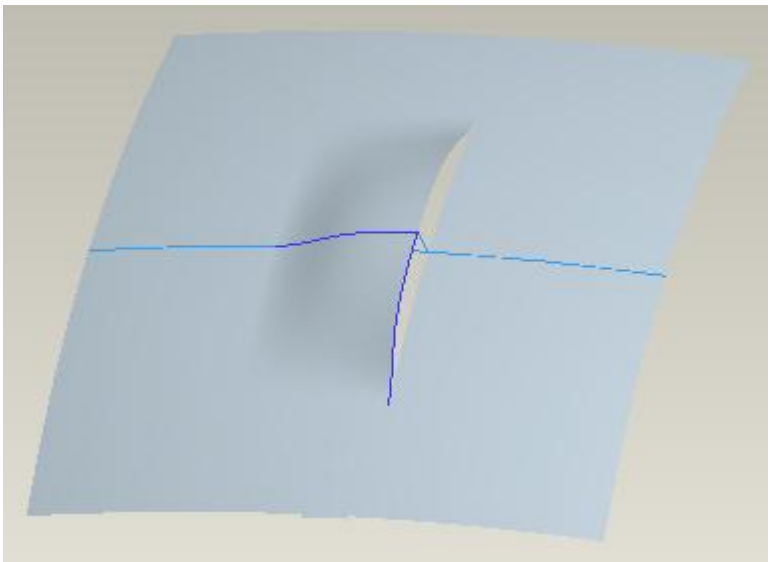
Pick or middle-click to complete the merge feature.

Now we will mirror the quilt so pick the merged quilt as shown below.



Pick Mirror on the Model tab to invoke Mirror tool and select the **FRONT** datum as Mirror plane.

Pick to complete the feature. Part will appear as shown below.



Now you can merge both quilts together.

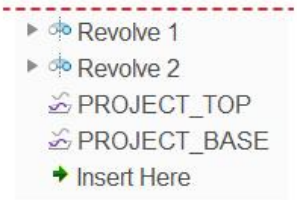
Select **File > Save** to save the work done so far.

Exercise 9

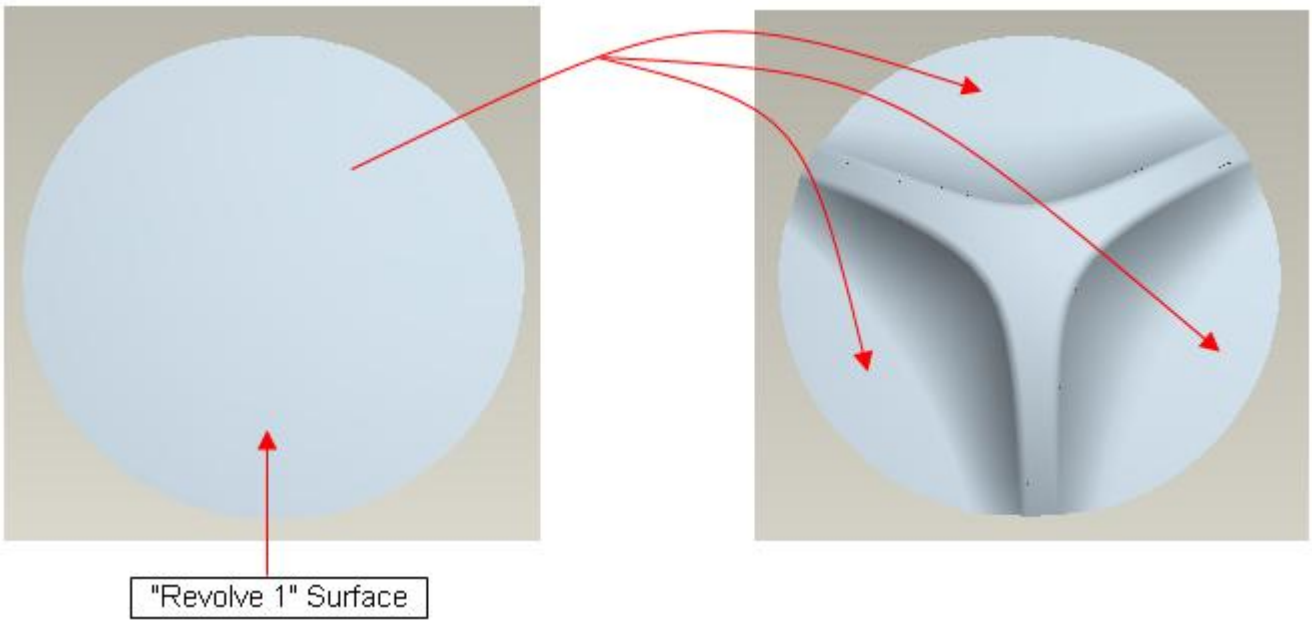
In this exercise we will create a model for washing machine pulsator. The completed part is shown below.

Set the working directory to the EXAMPLES folder and open the model PULSATOR.PRT

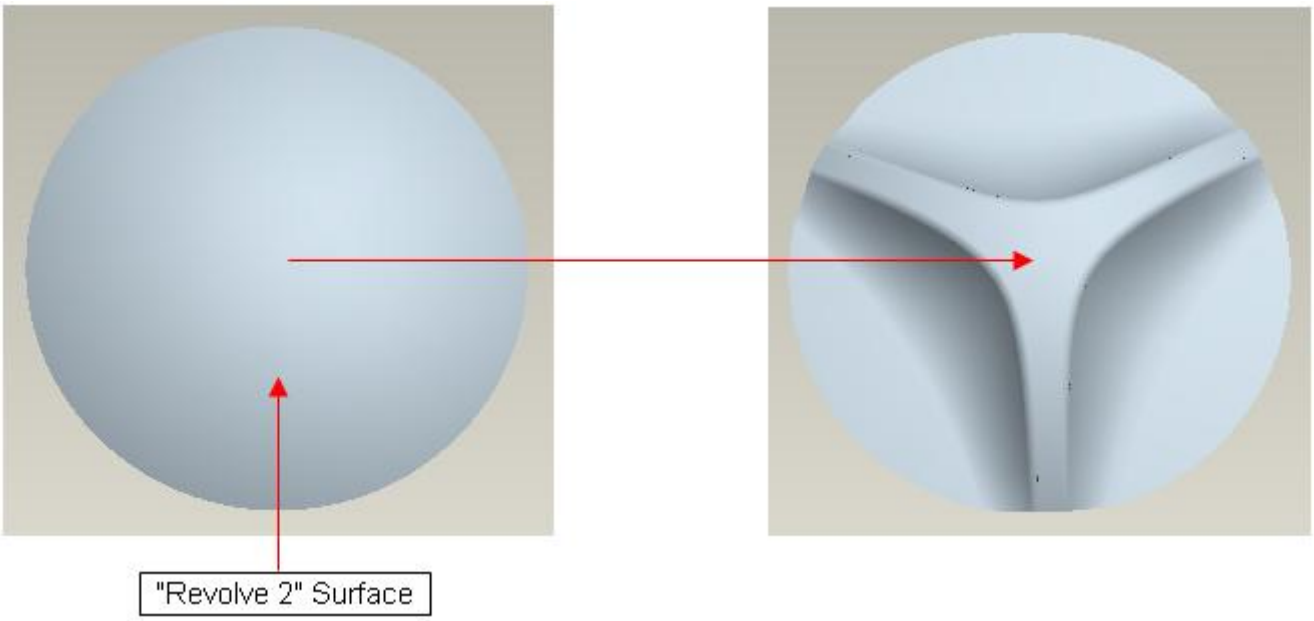
If you look in the model tree, you will notice that in addition to datum curves it also contains two revolve features and two projected curves.



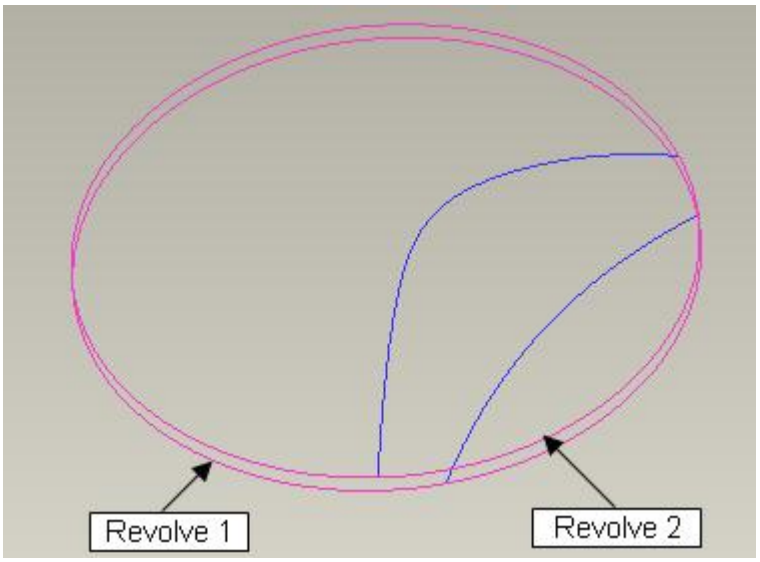
The “Revolve 1” feature creates surface that makes up the base surface of the pulsator as elaborated below.



The “Revolve 2” feature creates surface that makes up the top surface of the pulsator as elaborated below.



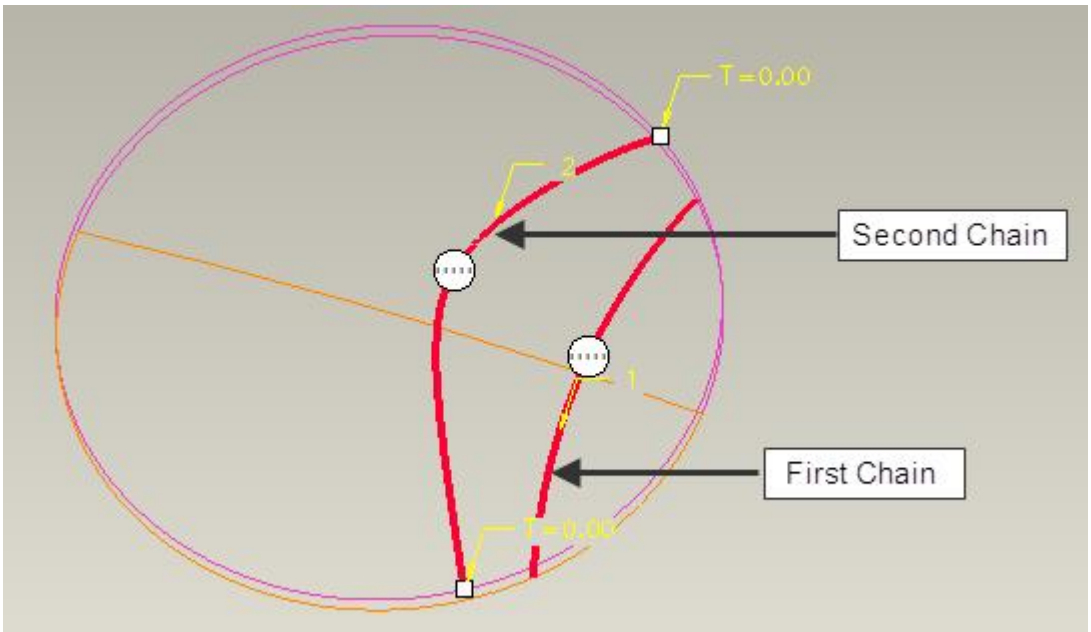
We will create a boundary blend surface that will reference the projected datum curves. You can see the projected datum curves in the figure below.



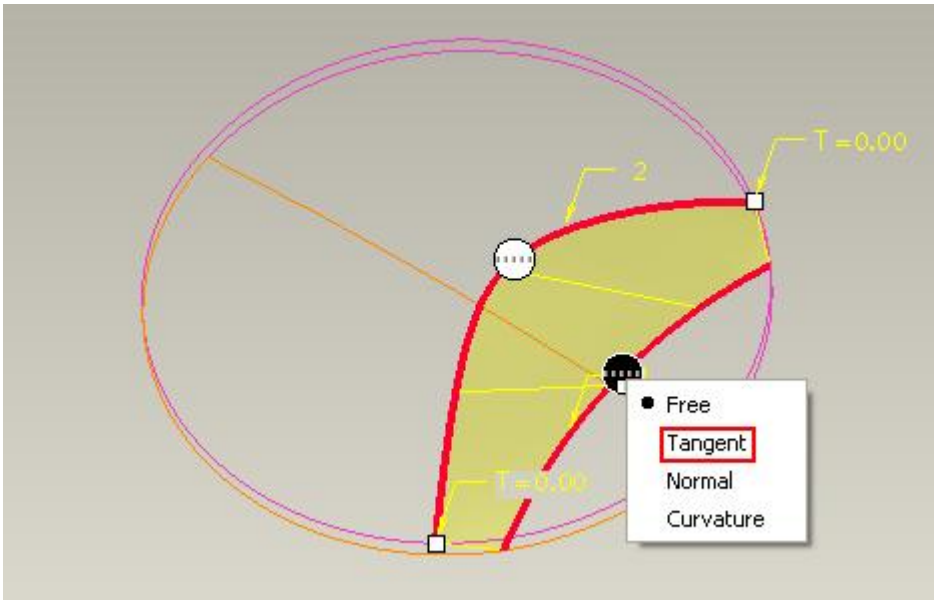
Creating the surface

We will create a boundary blended surface by using the curves shown in the above figure.

Pick and select the curves, while holding down the Ctrl key, as shown in figure below.



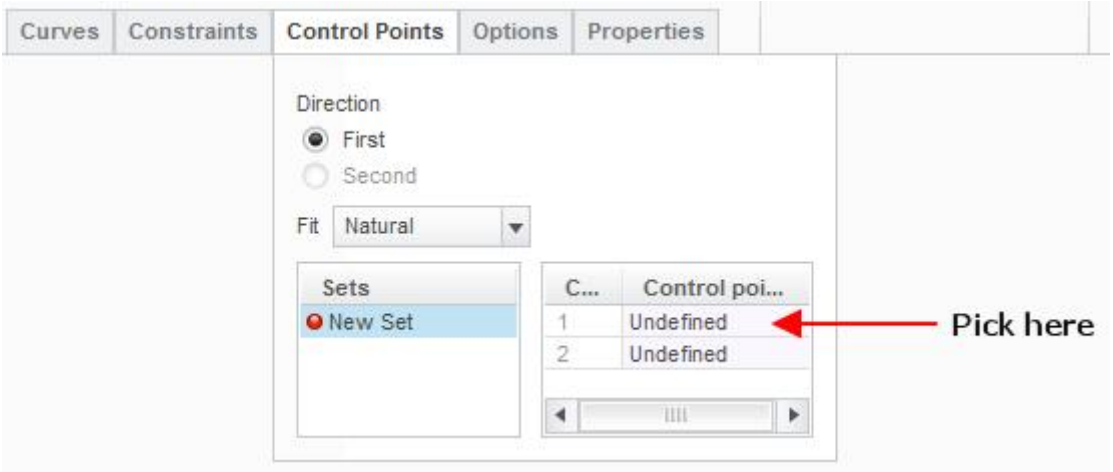
Apply the **Tangent** constraint for the bottom curve (that lies on “Revolve 1”) as shown below.



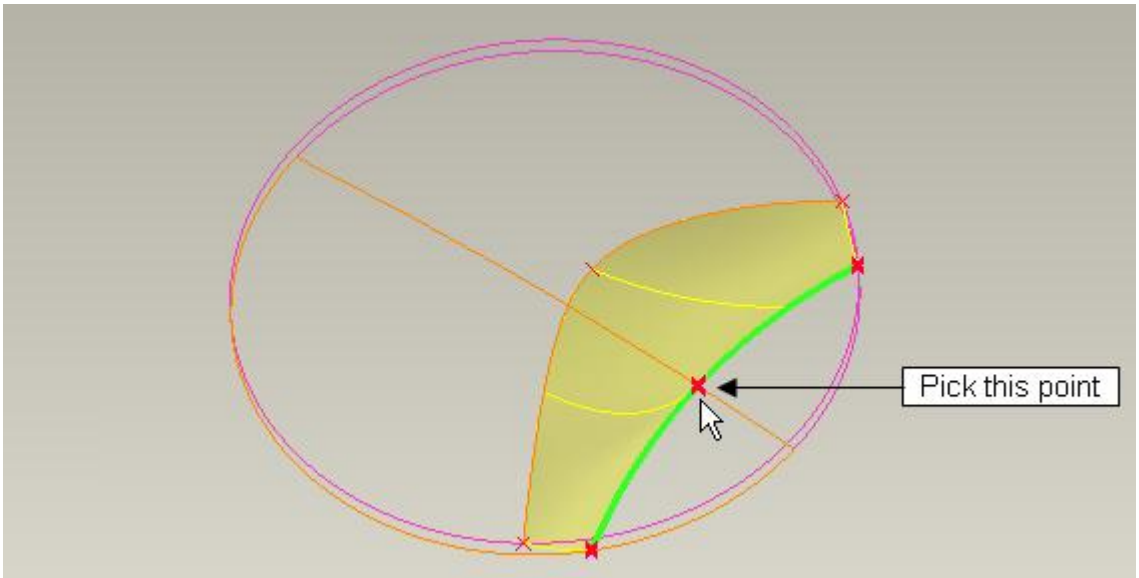
You should notice that surface shape is not as smooth as we expected. It is because the system cannot map the points on one curve to the corresponding points on the second curve. So we need to fix the surface by ourselves.

Pick Control Points icon to access the Control Points panel.

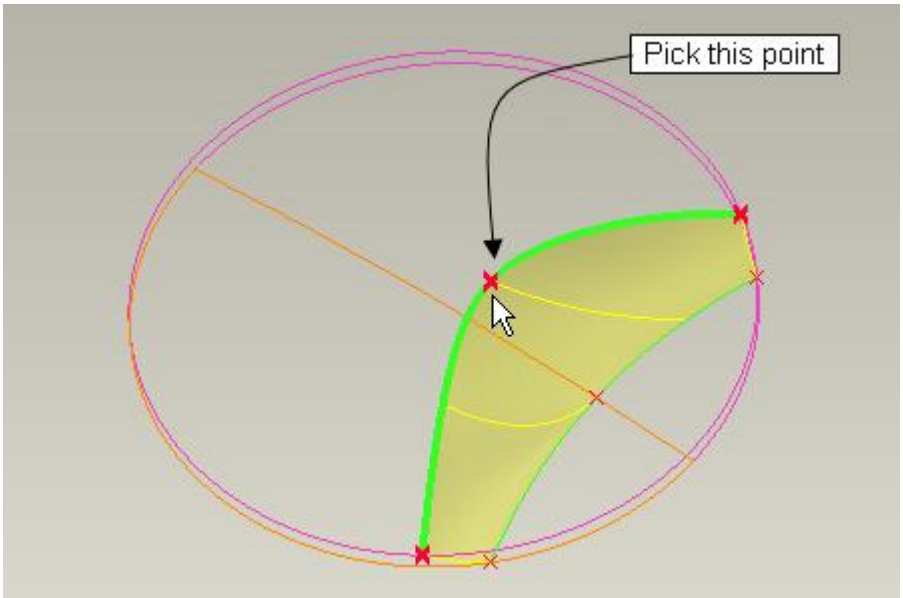
Pick on the first row in the collector for points as shown below.



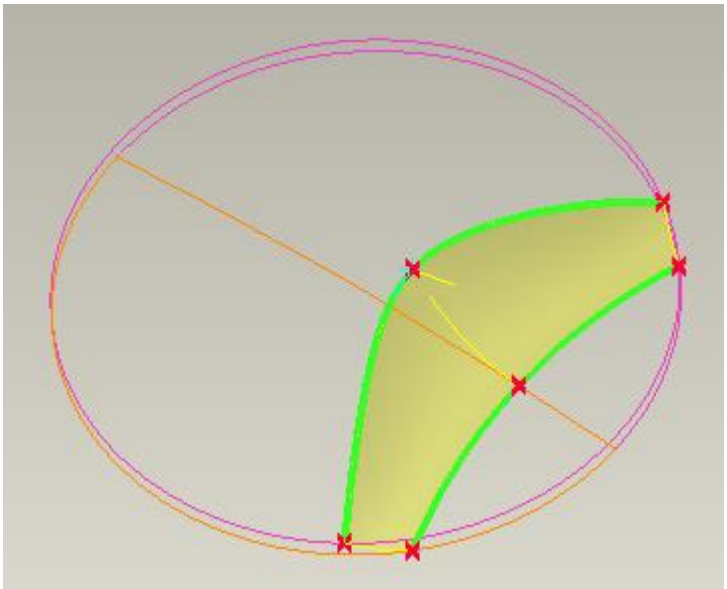
Now pick the mid point as shown below.



System will highlight the control points on second curve. Again pick the mid point as shown below.



The preview will appear as shown below.

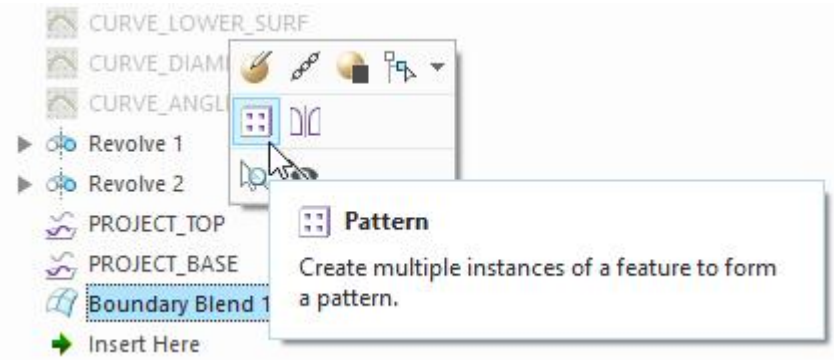


Pick  to complete the feature.

Pattern the surface

Now we will pattern the boundary blended surface.

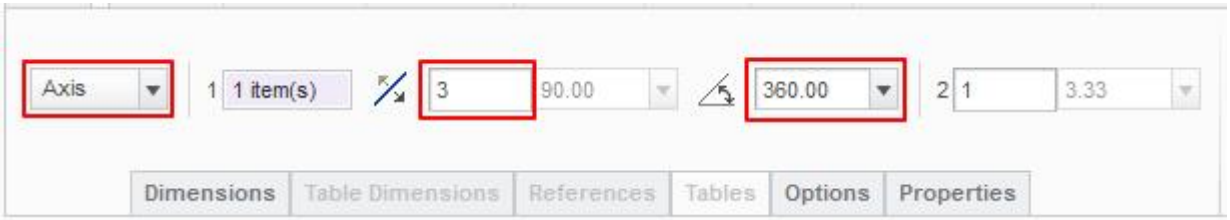
Select the **Boundary Blend 1** in the model tree and pick **Pattern**.



Change the pattern type to **Axis** and select axis A_2

Enter ***3*** as the number of pattern members.

Pick  and enter ***360*** as the angular extent then hit Enter key.



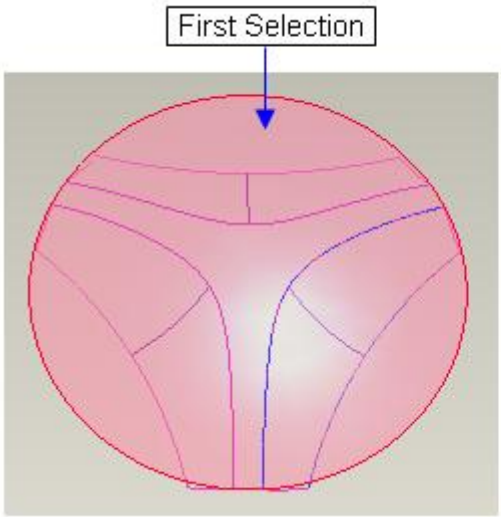
Pick  to complete the pattern.

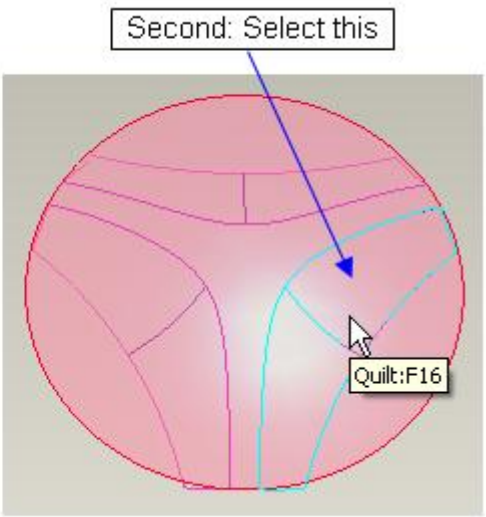
Merging the Quilts

Now we will merge the quilt together. For ease of selection first we need to hide the surfaces that are not needed immediately.

Select the “Revolve 1” feature and pick **Hide**.

So first select the revolved surface and then, while holding down the Ctrl key, first instance of pattern of boundary blended surface as shown below.

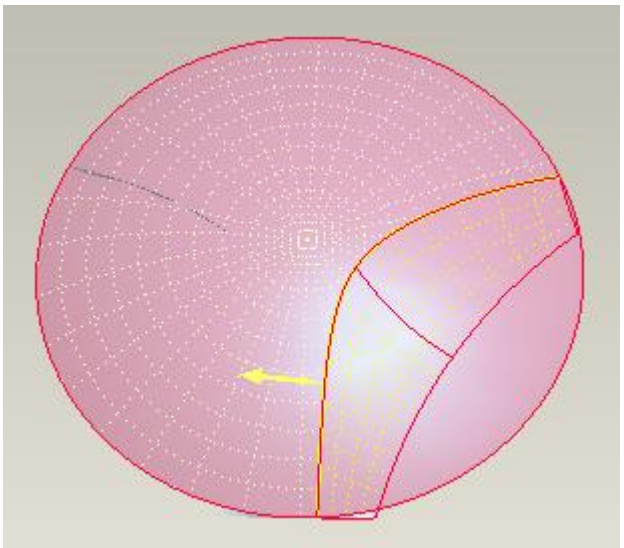




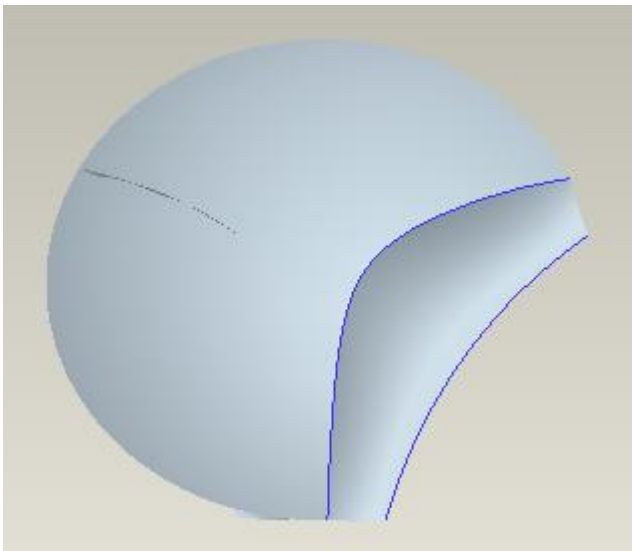
Pick 

on the Model tab to invoke Merge tool.

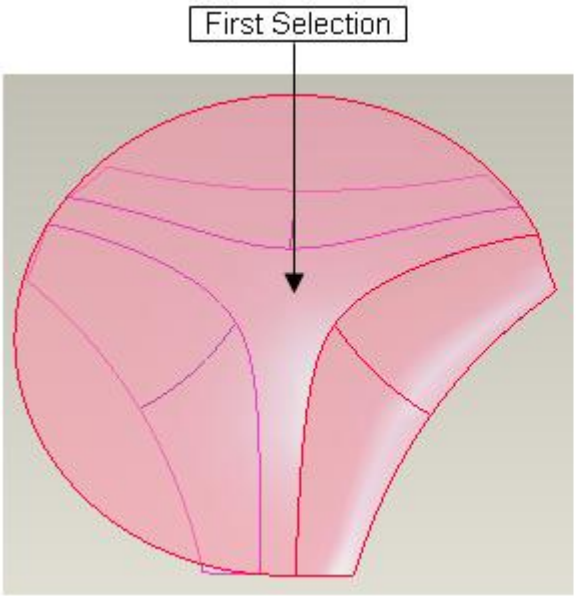
Pick  so that preview change to image shown below.

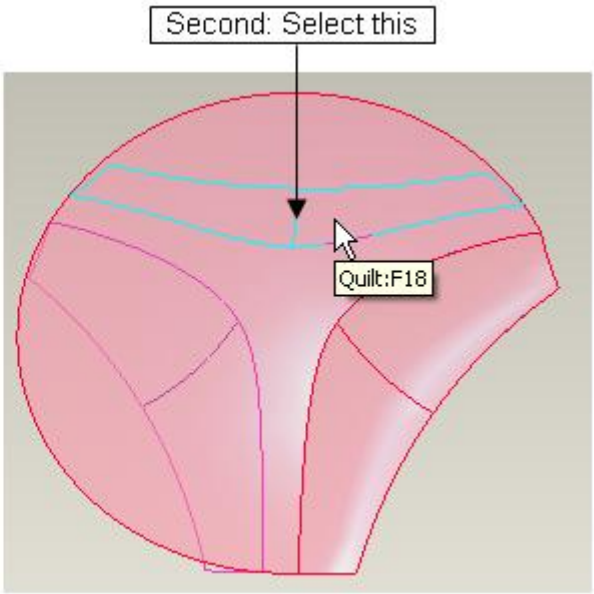


Pick  or middle-click to complete the merge feature. The quilt will appear as shown below.



Select the new quilt (created after merge operation) and then, while holding down the Ctrl key, boundary blended surface as shown below.

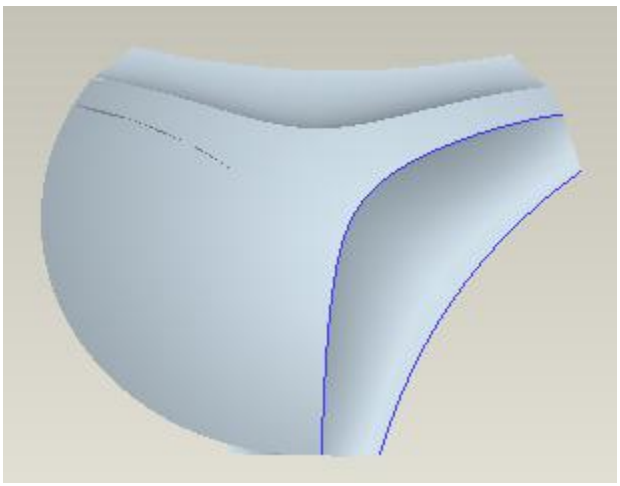




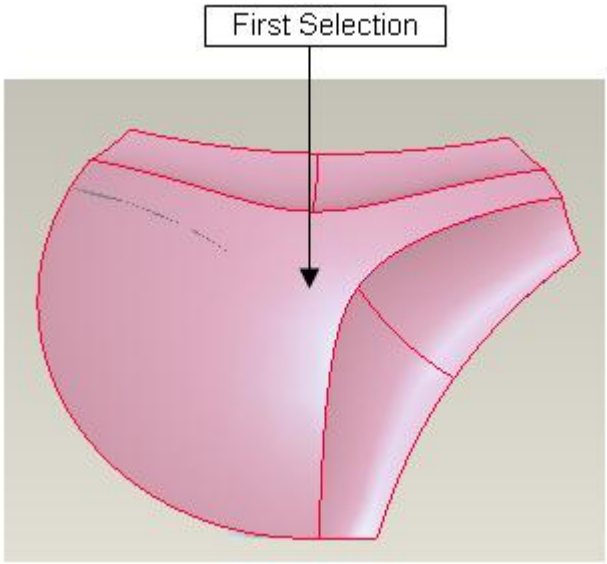
Pick  on the Model tab to invoke Merge tool.

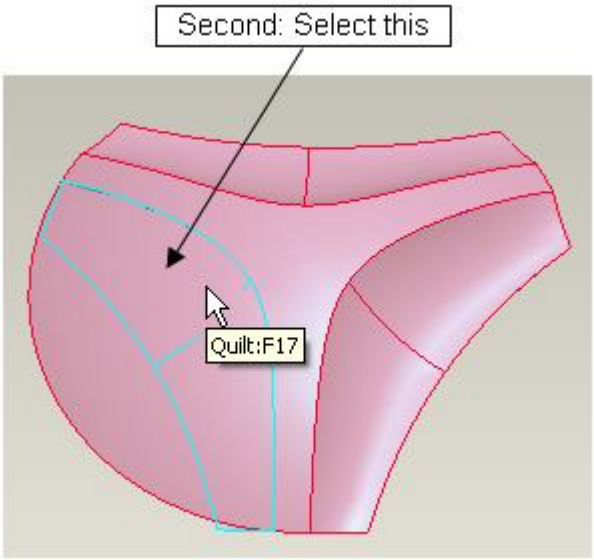
Pick  to flip the quilt side to keep.

Pick  or middle-click to complete the merge feature. The quilt will appear as shown below.



Select the new quilt (created after merge operation) and then boundary blended surface as shown below.



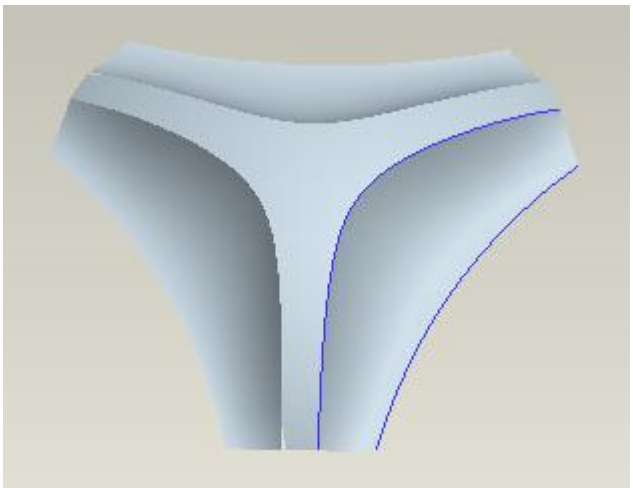


Pick 

on the Model tab to invoke Merge tool.

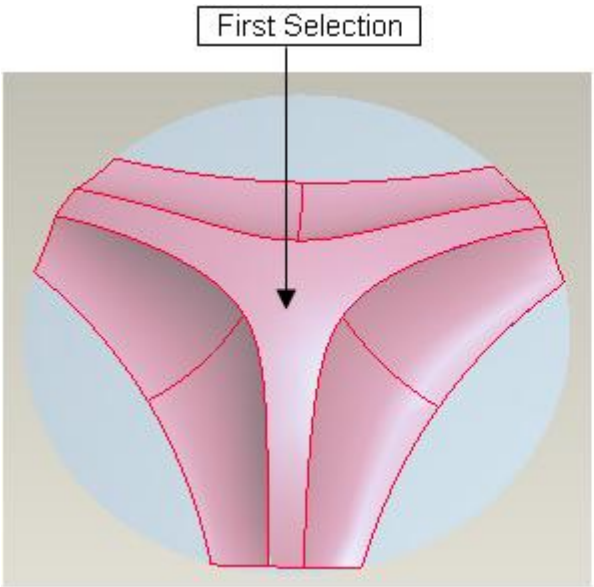
Pick  to flip the quilt side to keep.

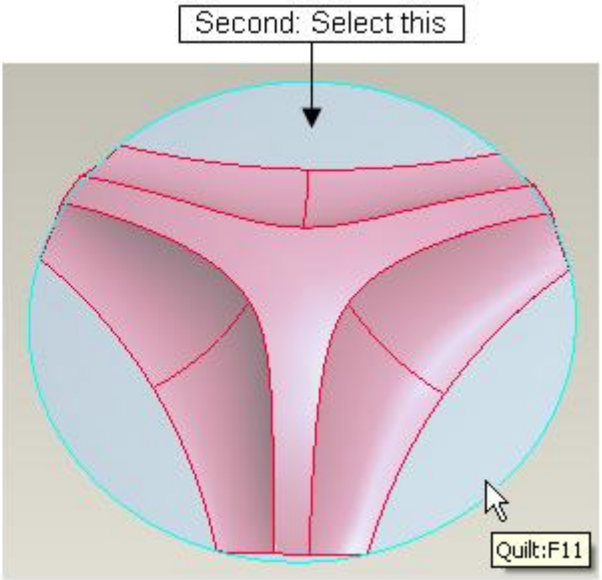
Pick  or middle-click to complete the merge feature. The quilt will appear as shown below.



Now we need to merge the quilt created by “Revolve 1” feature. So Select the “Revolve 1” in the model tree and pick **Show**.

Select the new quilt (created after merge operation) and then, while holding down the Ctrl key, revolved surface as shown below.



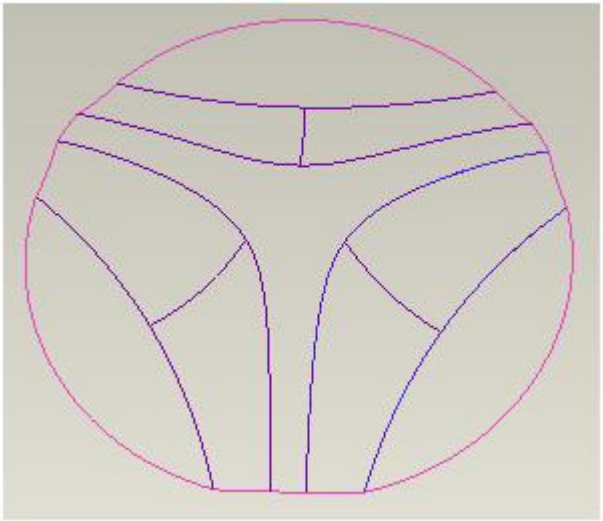


Pick 

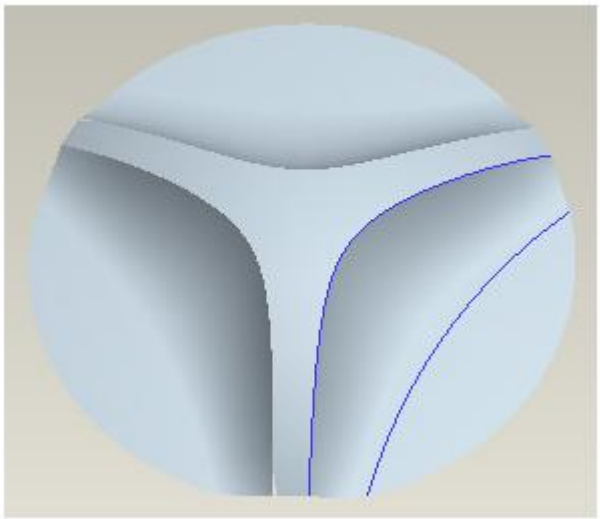
on the Model tab to invoke Merge tool.

Pick  or middle-click to complete the merge feature. The quilt will appear as shown below.

Wireframe View

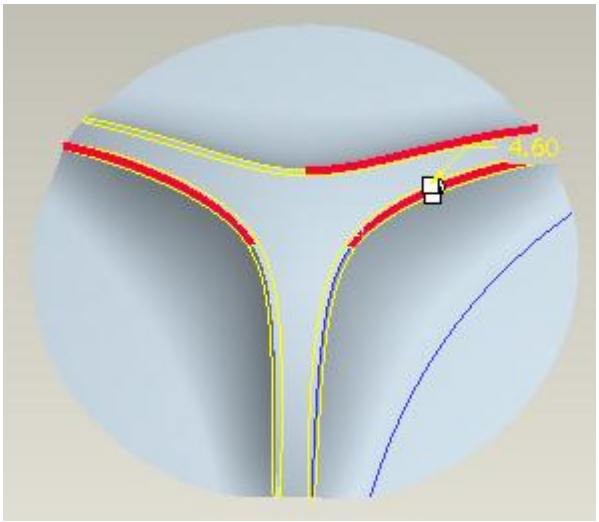


Shaded View



Pick  to access the Round tool.

Select the following edge chains while holding down Ctrl key.



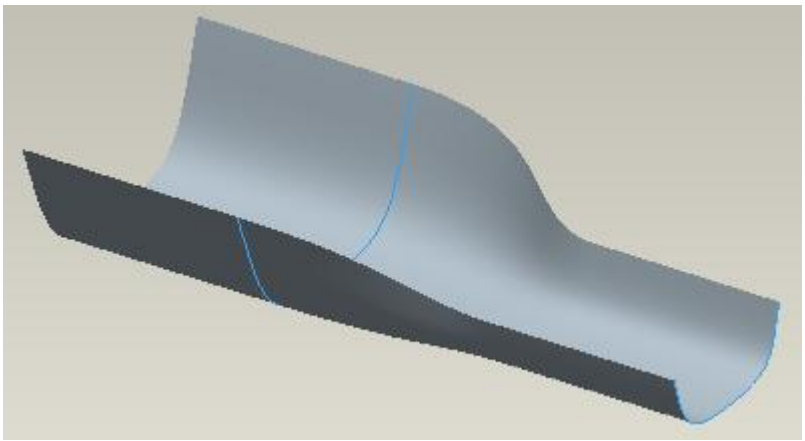
Enter ***5*** as the radius value

Pick  icon to complete the feature. The part will appear as shown below (after hiding the curves).

Select **File > Save** to save the work done so far.

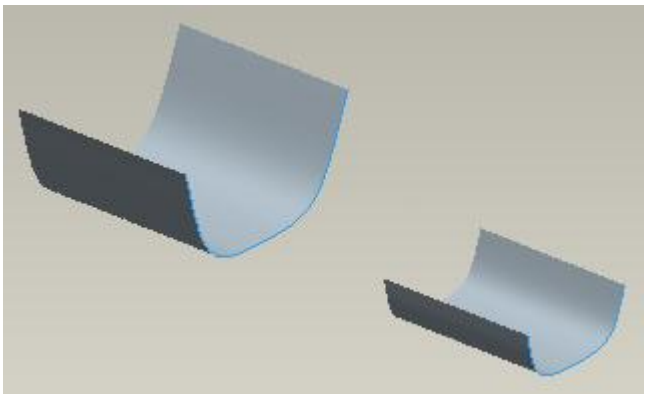
Exercise 10

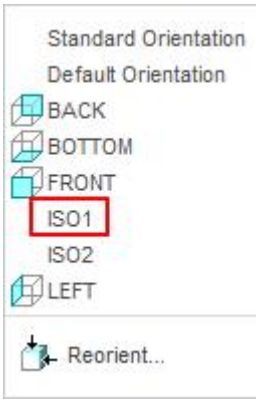
In this exercise we will create a blended surface that connects two parts of a channel. The completed part is shown below.



Set the working directory to the EXAMPLES folder and open the model CHANNEL.PRT

Set the view to **ISO1** and part will appear as shown below.

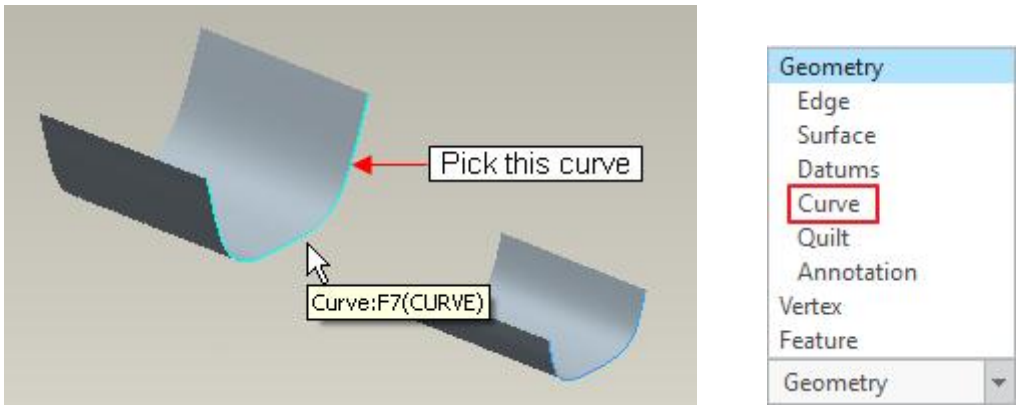




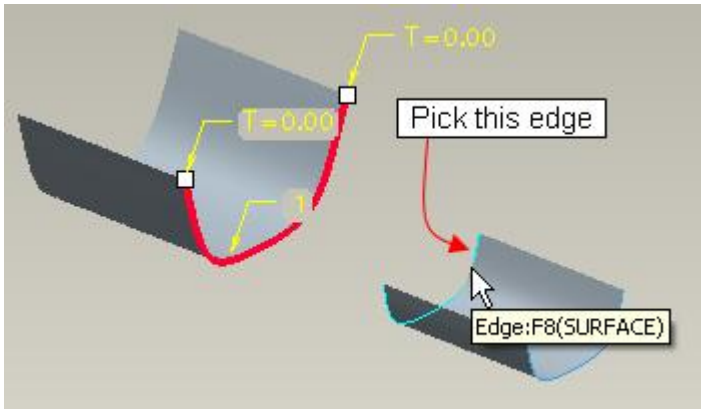
Creating the surface

Now we will create boundary blended surface that connects the shown parts of the channel. You must set the view to ISO1 to avoid any confusion.

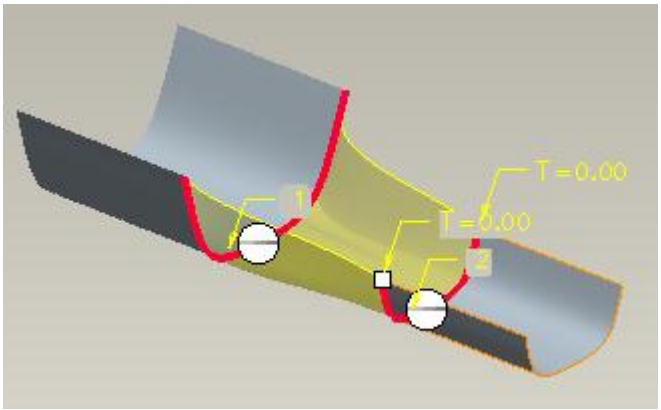
Pick and select the sketched datum curve as shown in figure below. (Make sure that you pick the datum curve and not the edge of the surface. You can use Curve filter to make sure that you select the curve and not the edge.)



Now hold down the Ctrl key and pick the edge shown in figure below as second chain.

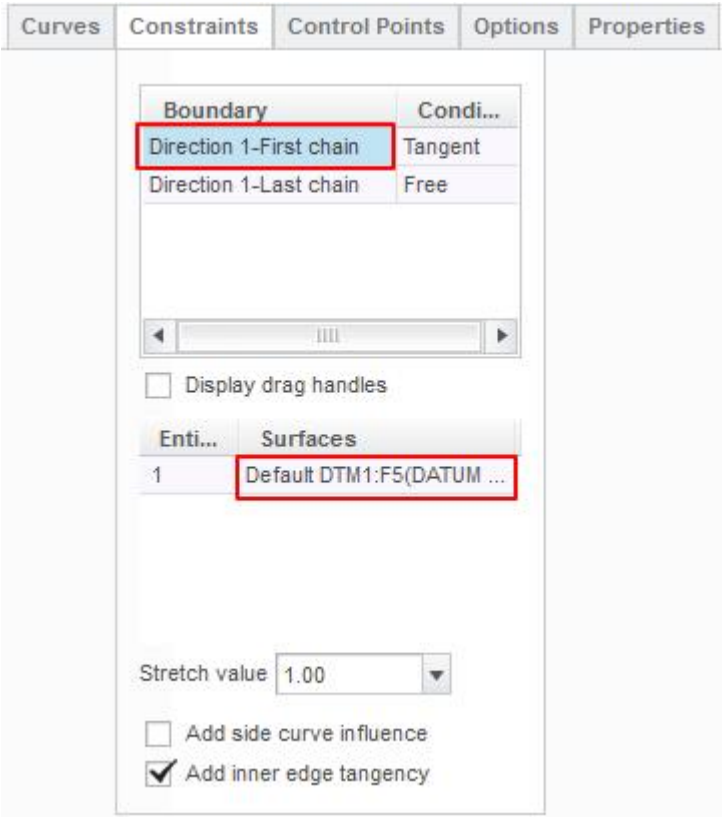


Apply the **Tangent** constraint for both chains. After applying the constraint surface will appear distorted as shown below.



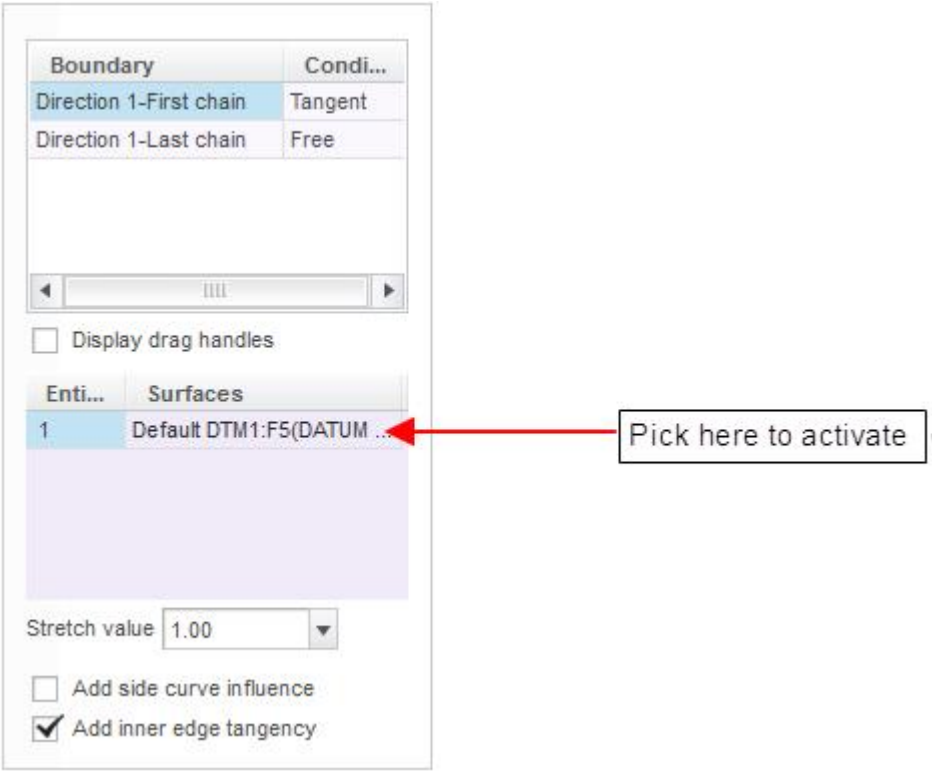
Now we will investigate the cause of this distorted surface.

Pick Constraints icon to access the Constraints panel and pick “Direction1-First chain”. Notice that system has selected the Datum plane “DTM1” as reference for the tangency constraint as shown below.

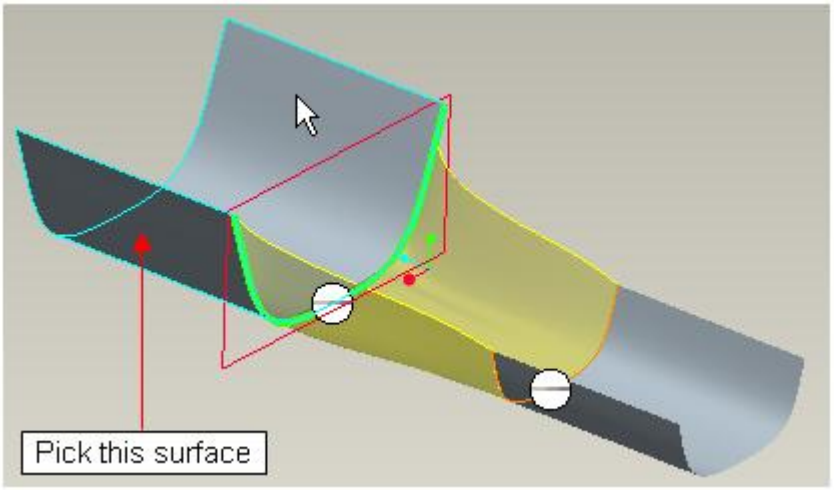


This is because selected datum curve (i.e. first chain) lies on “DTM1” and system has made the new surface tangent to this plane. We need to select the proper reference for this tangency constraint.

So pick in the Reference collector to make it active as shown below.



Now pick the surface shown in figure below as reference

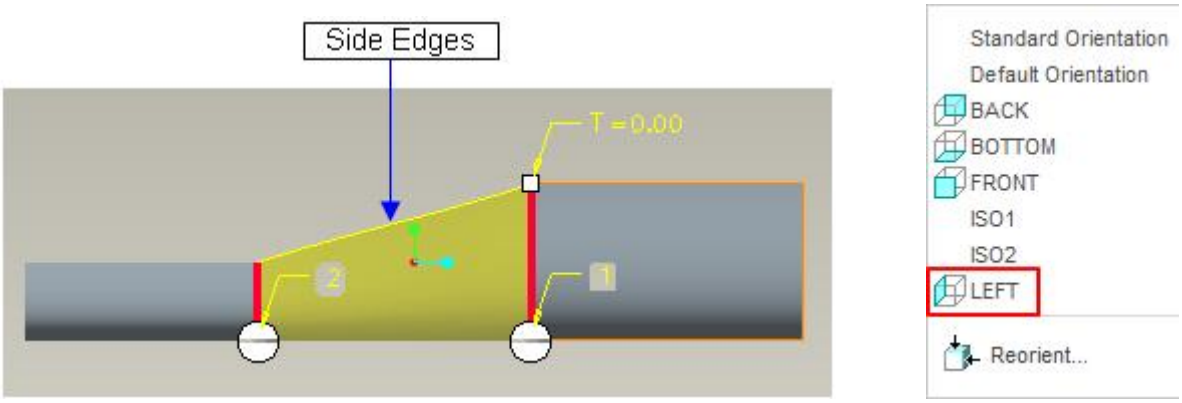


Now pick "**Direction1-Last chain**" in the panel and notice that system has selected the surface, whose edge was selected, as reference for the tangency constraint as shown below.

So it is always advantageous to select the edge of a surface instead of underlying curve. When you select the edges of a surface as boundary chain, then system automatically selects that surface as reference for boundary conditions.

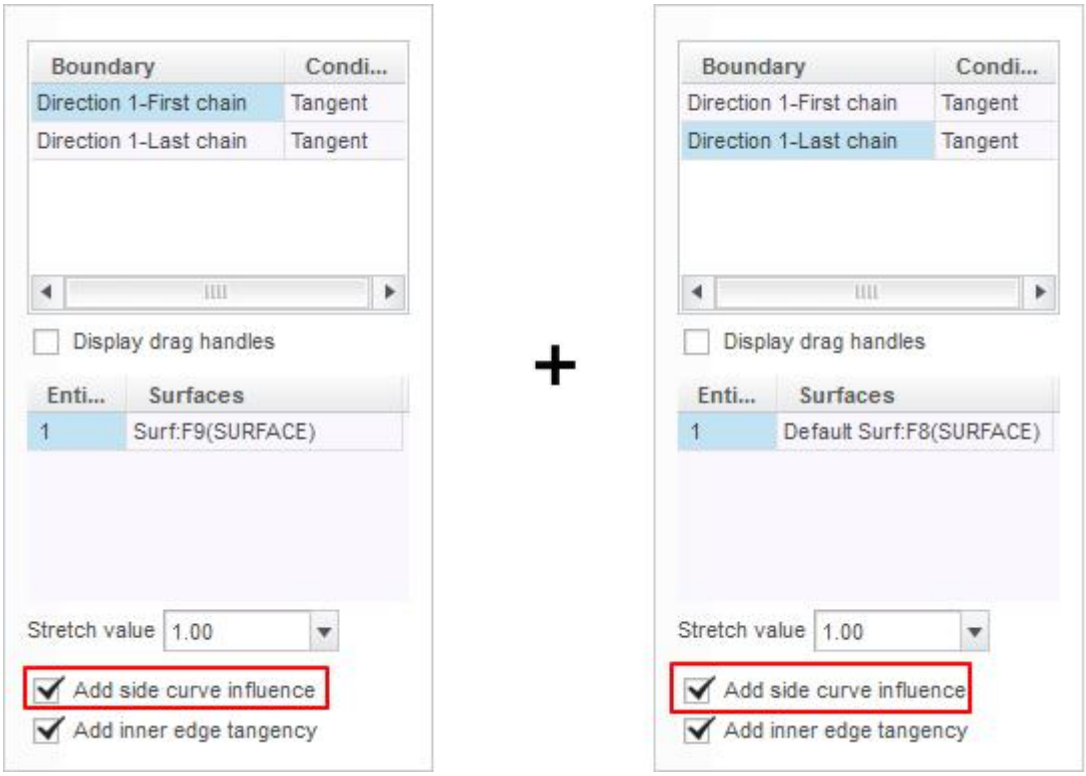
We purposely chose the datum curve instead of edge of surface, just to show you the advantages.

Set the view to **LEFT** and part will appear as shown below.



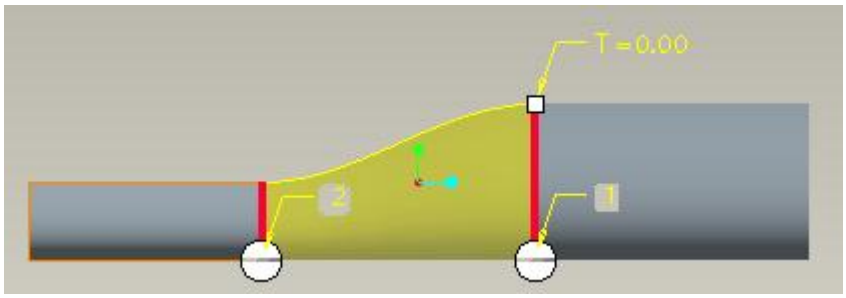
You should notice that although the blended surface is now tangent to the neighboring surfaces but the side edges of the blended surface are not tangent to the side edges of the neighboring surface. Now we will fix this.

Pick Constraints and pick “Direction1-First chain” and check the **Add side curve influence** option. Then pick “Direction1-Last chain” and again check the **Add side curve influence** option as shown in the figure below.



Add side curve influence option makes the side edges of he blended surface tangent to the side edges of the references.

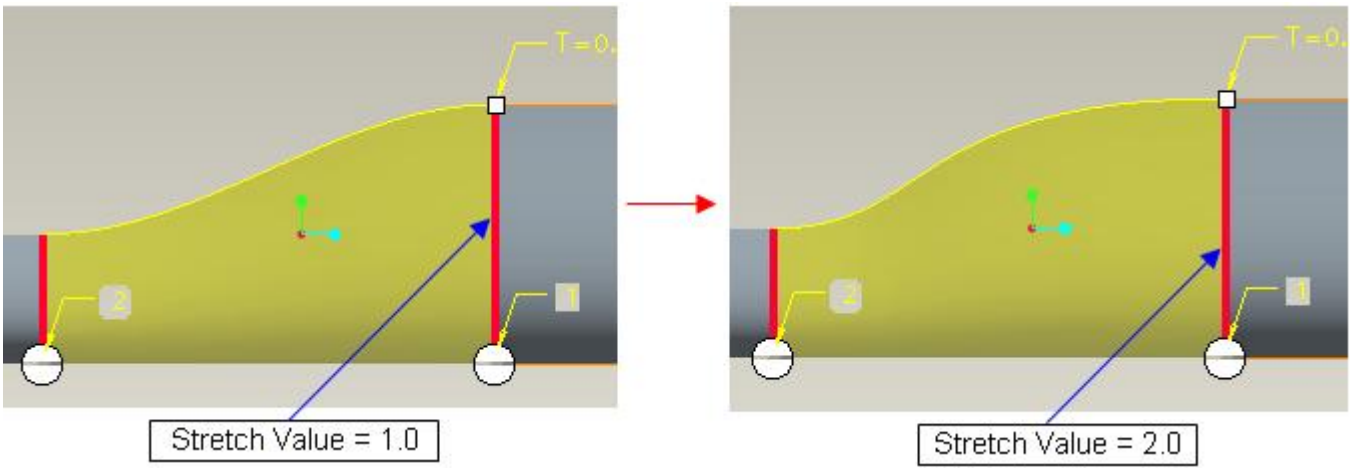
Now preview of the surface will appear as shown below.



Now we will modify the surface so that shape of the large surface is more dominant in the boundary blended surface.

Pick Constraints and pick “Direction1-First chain”. Then change the Stretch value to ***2*** as shown below.

Shape of surface will change as shown below.

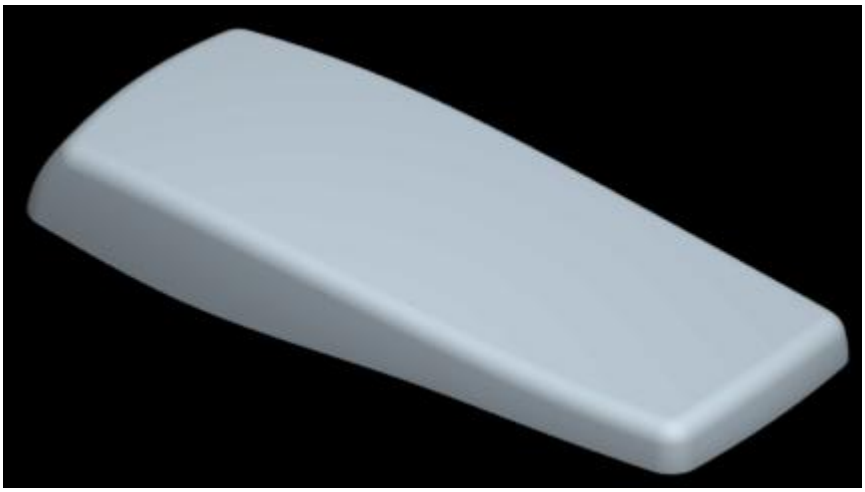


Pick to apply the changes and exit the dashboard.

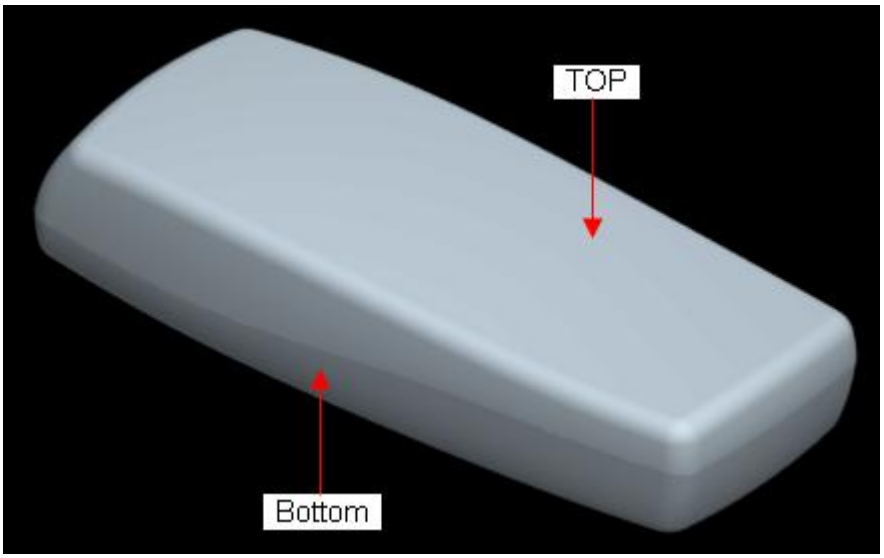
Select **File > Save** to save the work done so far.

Exercise 11

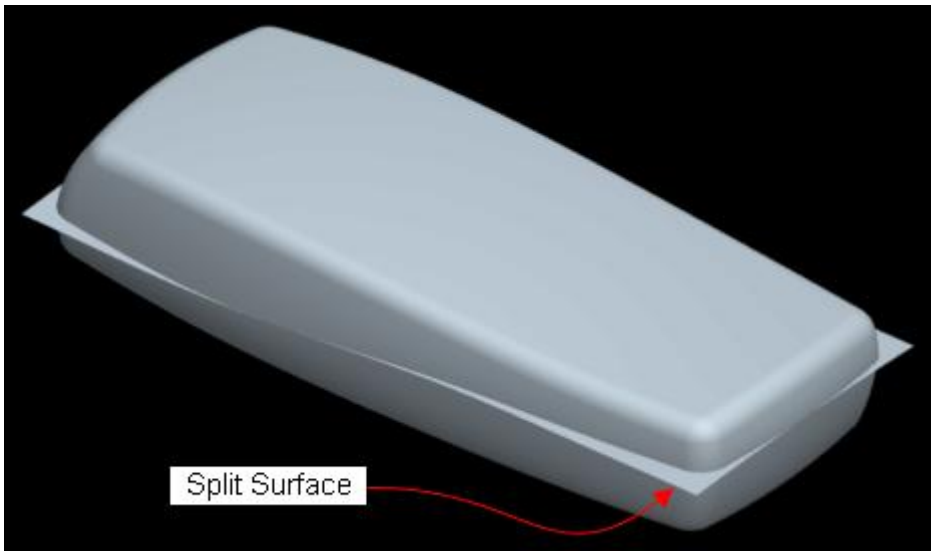
In this exercise we will create the side surfaces of a plastic component with sweep tool and learn to build the draft angle in the surface definition. The completed part is shown below.



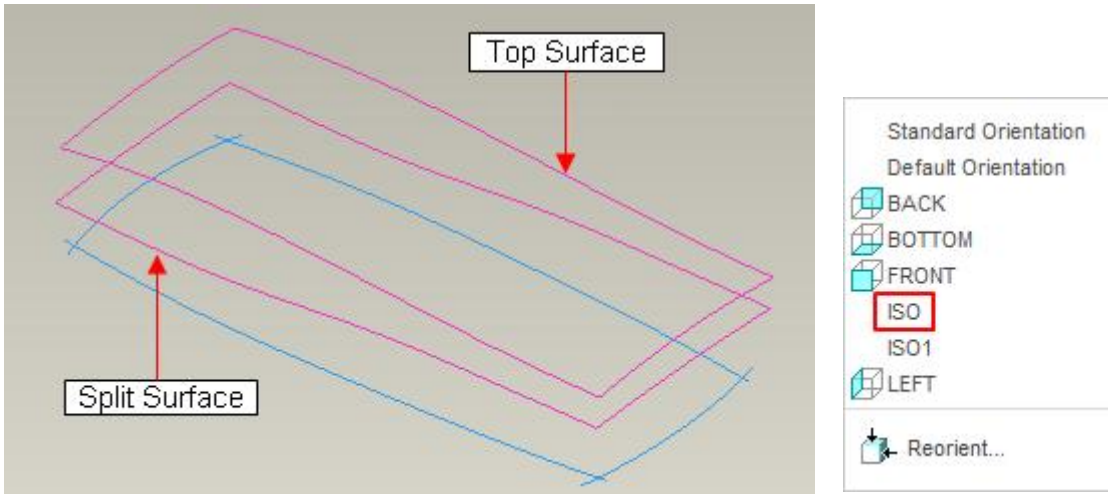
The part shown in the above figure forms the top part of the plastic housing of a mobile phone. The product assembly is shown below.



You should notice that both parts i.e. TOP and BOTTOM join along a 3-D surface (i.e. it is non-planar). The surface along which both components join is called Split Surface.



Set the working directory to the EXAMPLES folder and open the model MOBILE_TOP.PRT
Set the view to **ISO** and part will appear as shown below.

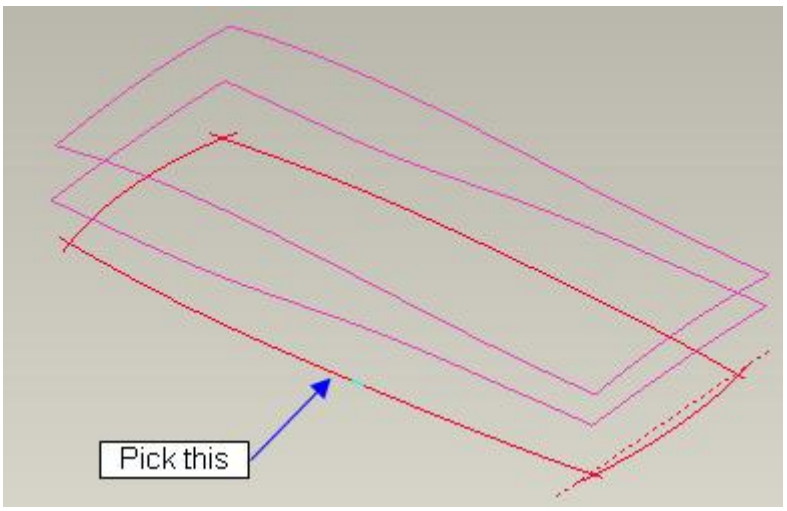


Creating the surface

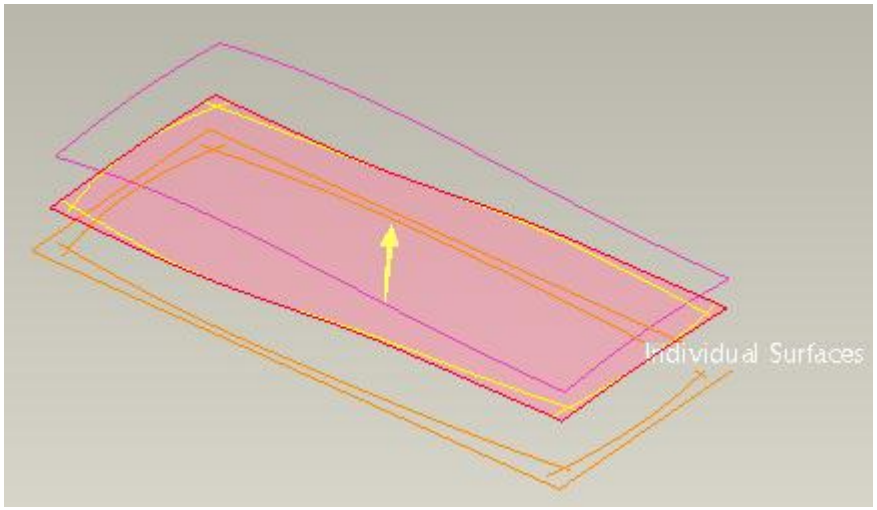
Before defining the surfaces, first we will project the datum curves on the split surface and these projected curves will be used as trajectories for the surfaces.

When the split surface is non-planar then it becomes necessary to begin with split surface.

So pick the following highlighted curve feature using Feature filter.

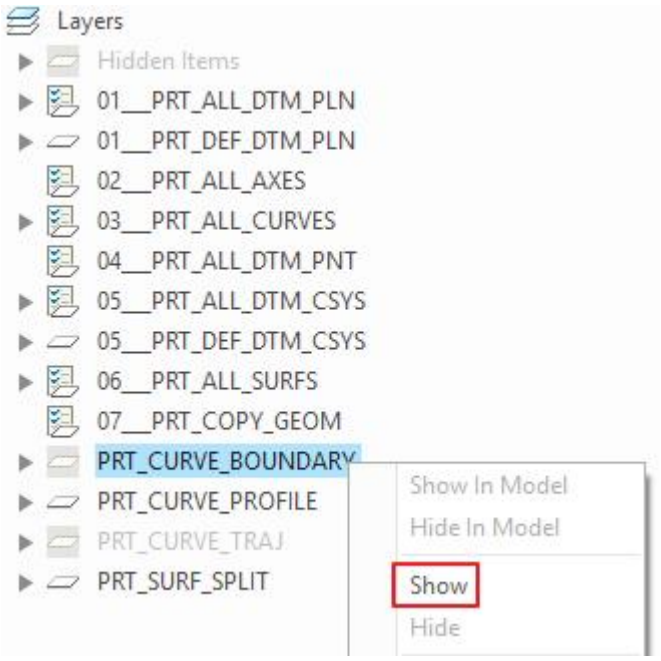


Pick Project on the Model tab to invoke the Project Tool and select the split surface to project selected curve onto. The preview will appear as shown below.



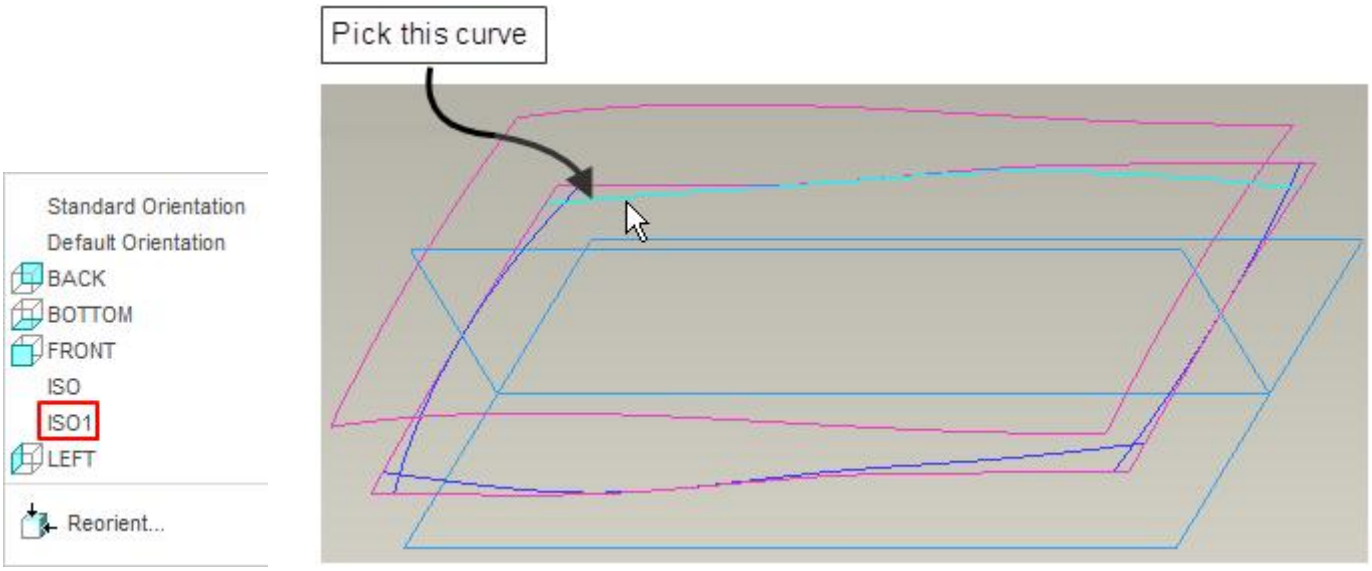
Pick to complete the feature.

Before creating the surface, unhide the PRT_CURVE_BOUNDARY layer. This layer contains the curves which represent the horizontal and vertical boundaries of the mobile assembly. Layer tree can be shown by picking on View tab.

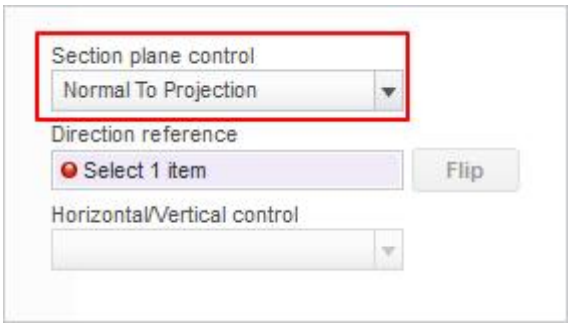


Now we will create the side surfaces with VSS feature. We will build the draft angle in the surface definition.

Pick Sweep icon to invoke the sweep tool and select the following curve as reference (Origin Trajectory). Set the view to **ISO1** to avoid any confusion.



Pick References tab and change the “Section plane control” option to **Normal To Projection**.

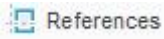


Select **TOP** datum plane as direction reference.

When Normal To Projection option is used, system projects the Origin Trajectory along the selected references (TOP Datum Plane in this case) and the section plane stays normal to this trajectory as the section is swept along the trajectory.

Pick  to create the feature as a surface.

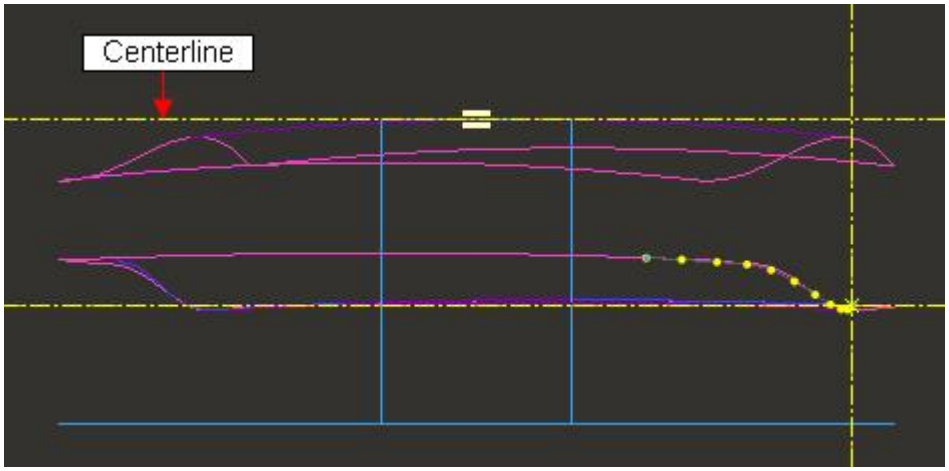
Pick  to sketch the section.

Pick 

and select the following curve as reference.

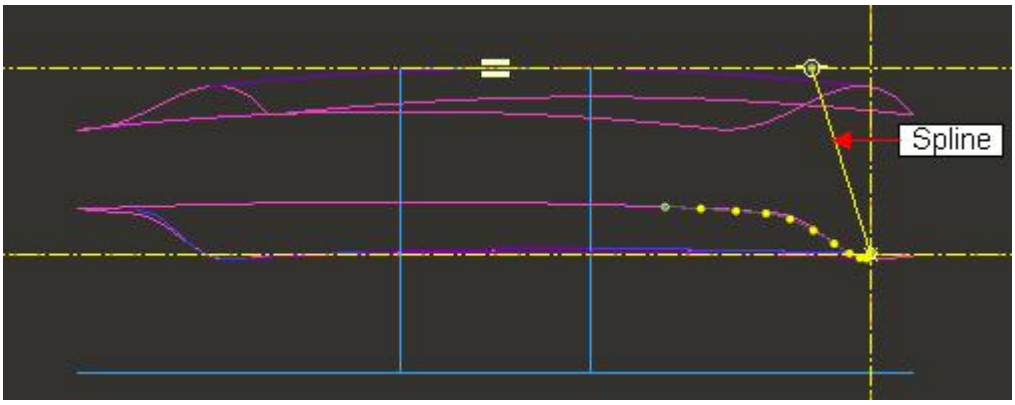
Pick 

and Sketch a centerline that passes through the selected reference as shown below.

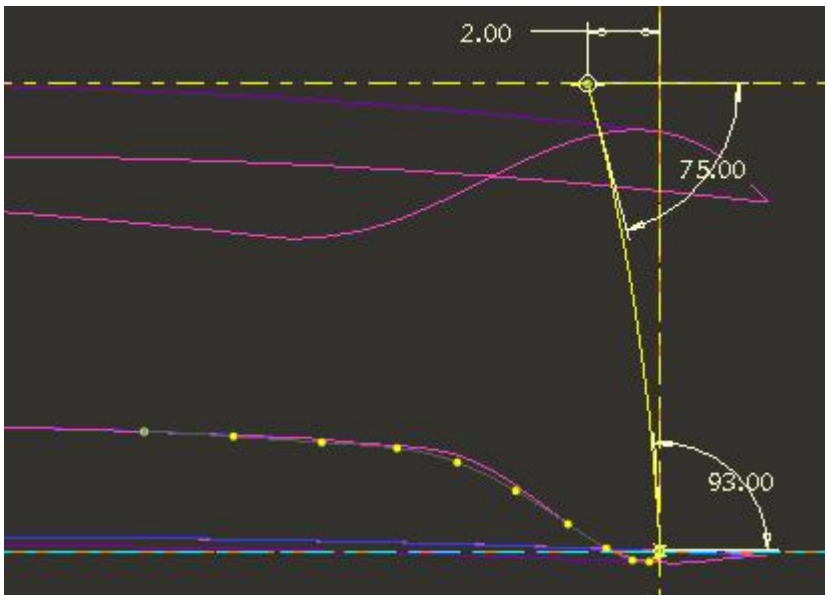


Sketch a two point's spline by picking 

icon. One end of the spline should lie at cross-hair as shown below.



Dimension the spline as shown below. Note: The tangency dimensions (75 and 93 in the figure) are created by first picking spline, then the centerline and then the endpoint where the tangency is defined. After this you should pick middle mouse button to place the dimension.



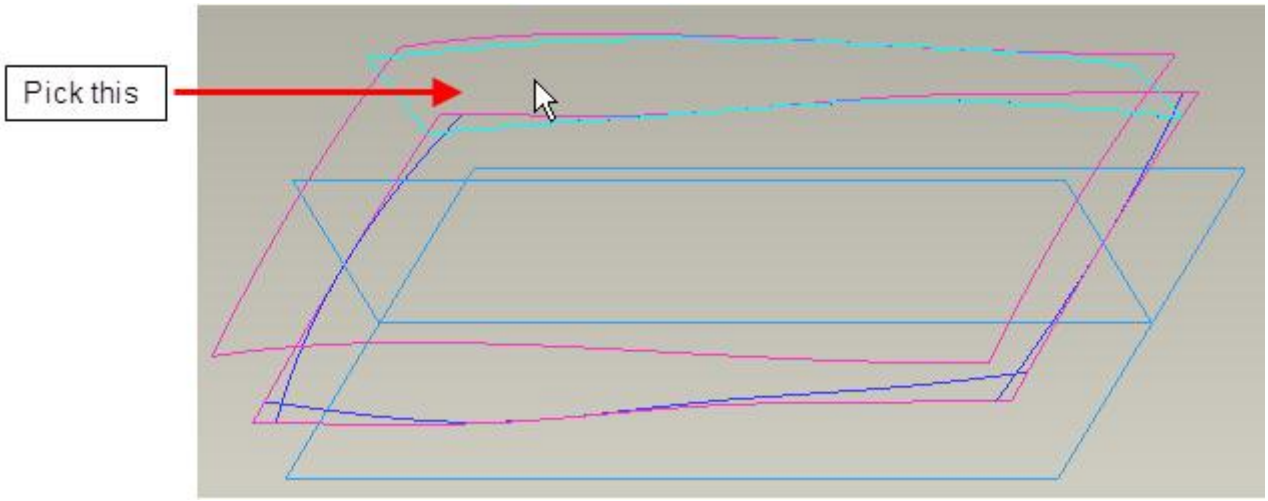
The tangency dimension of “93” builds the draft in the surface definition.

Pick after completing the sketch.

Pick to complete the sweep feature.

“Normal To Projection” option makes sure that the surface does not overlap or intersects itself and hence there is no manufacturing problem for the final components using this surface as reference.

Now we will mirror this feature as the part is symmetric. So select the newly created surface.



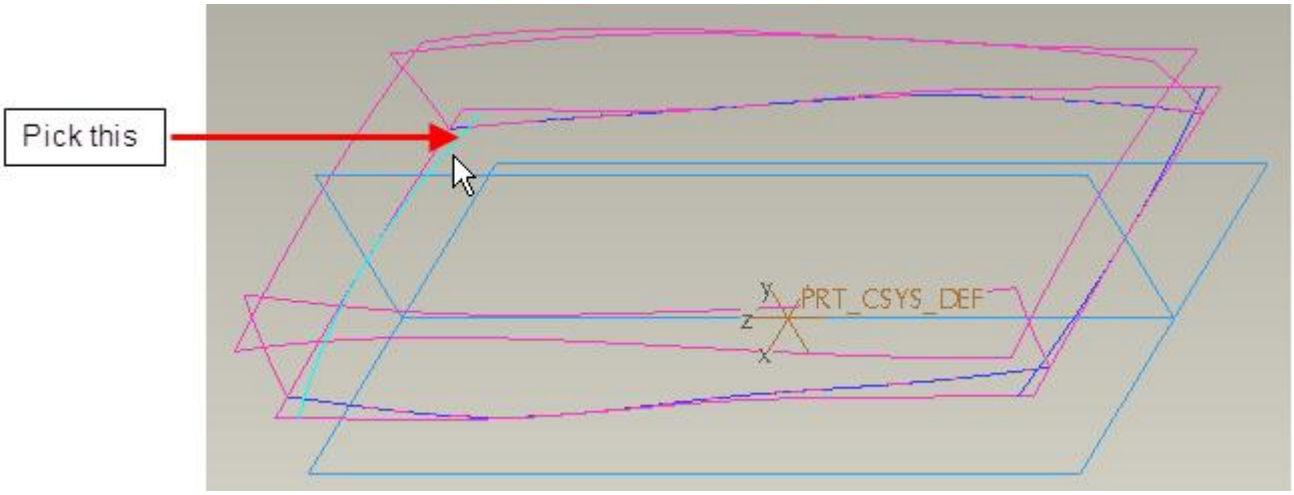
Pick Mirror on the Model tab to invoke Mirror tool.

Select the **RIGHT** datum as Mirror Plane.

Pick to complete the feature.

Now we will create the front side surface.

Pick Sweep and select the following curve as trajectory. (Set the view to **ISO1** to avoid any confusion.)



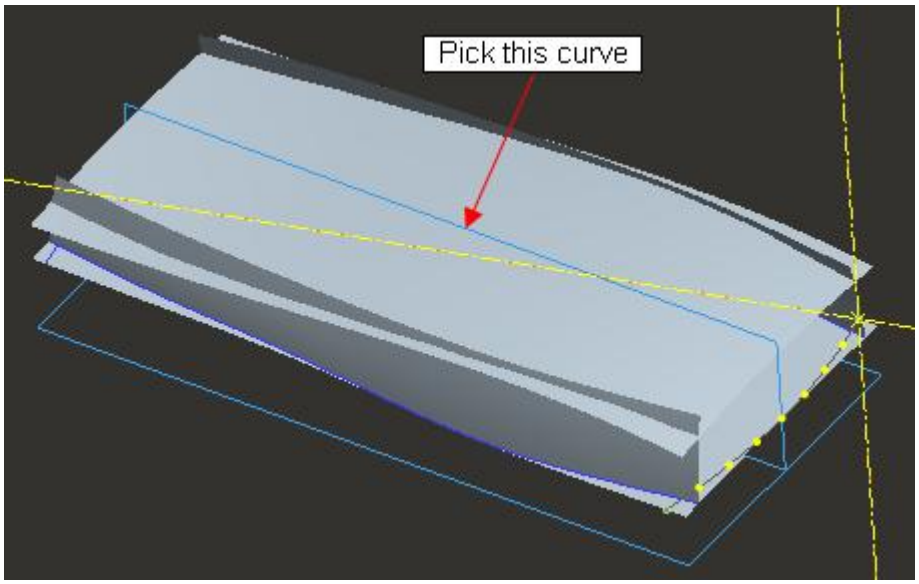
Change the “Section plane control” option to **Normal To Projection**.

Select **TOP** datum plane as direction reference.

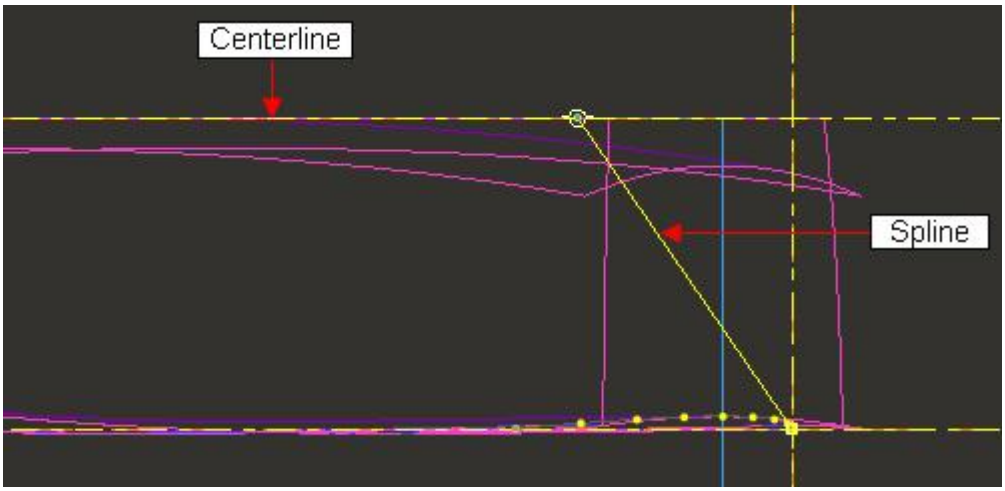
Pick to create the feature as a surface.

Pick to sketch the section.

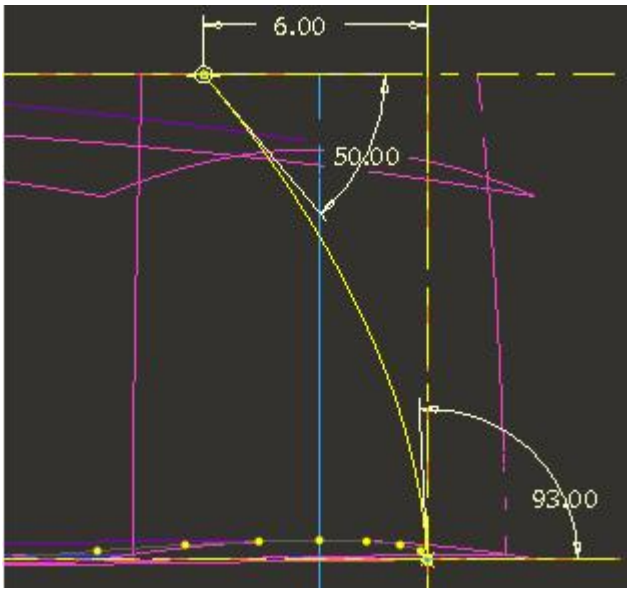
Pick References and select the following curve as reference.



Sketch a centerline and two point's spline as shown below.



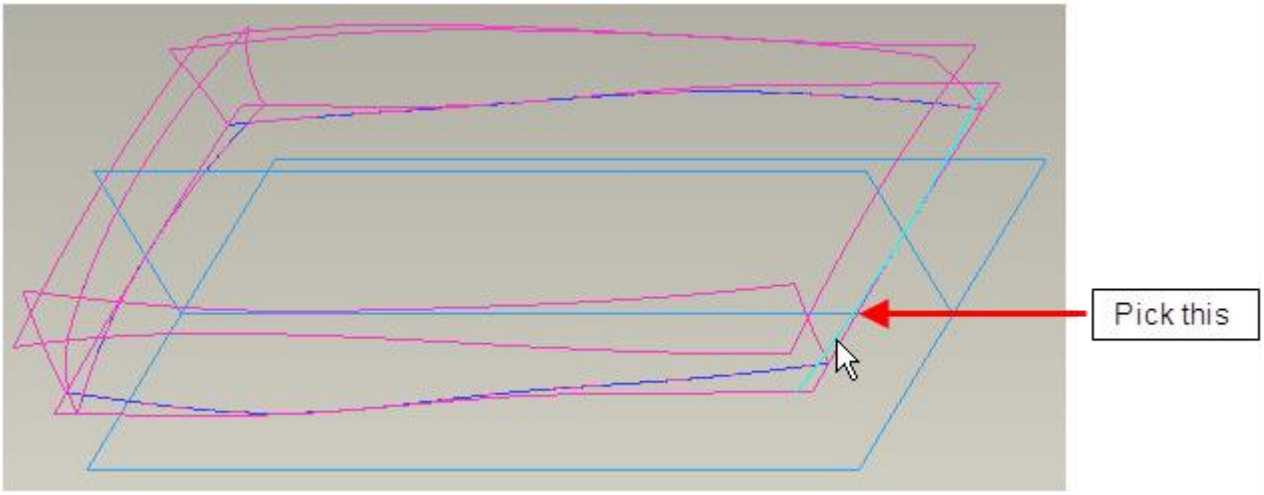
Dimension the spline as shown below.



Pick ✔ after completing the sketch.

Pick  to complete the sweep feature.

Now we will create the back surface. So pick  and select the following curve as trajectory.

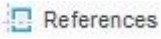


Change the “Section plane control” option to **Normal To Projection**.

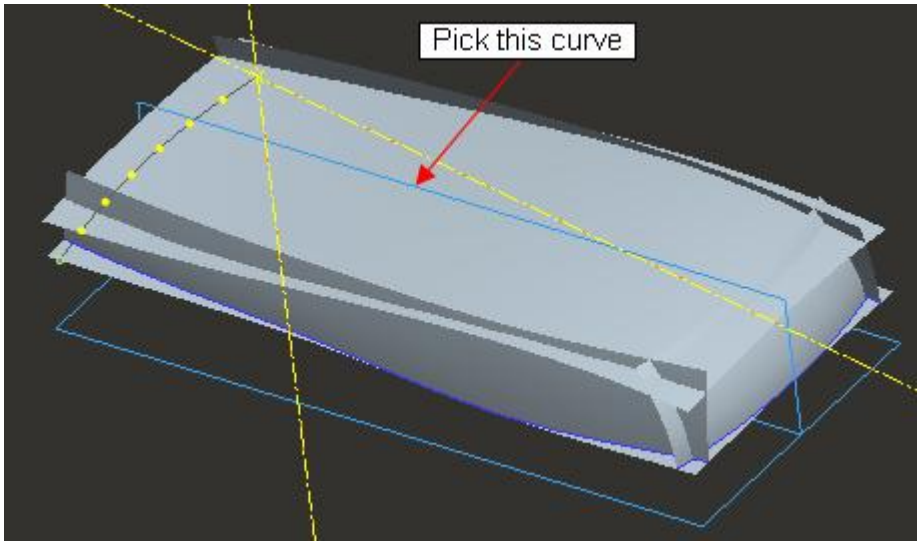
Select **TOP** datum plane as direction reference.

Pick  to create the feature as a surface.

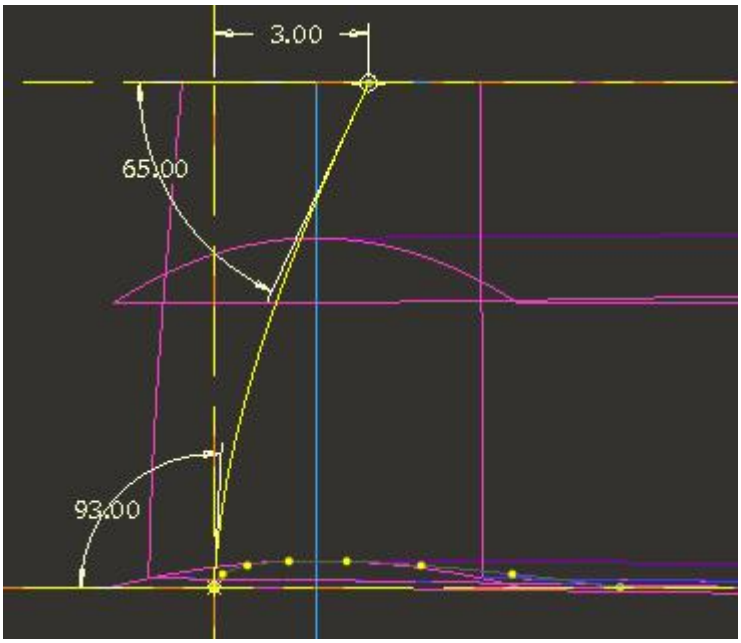
Pick  to sketch the section.

Pick 

and select the following curve as reference.



Sketch a centerline and two point’s spline as shown below.



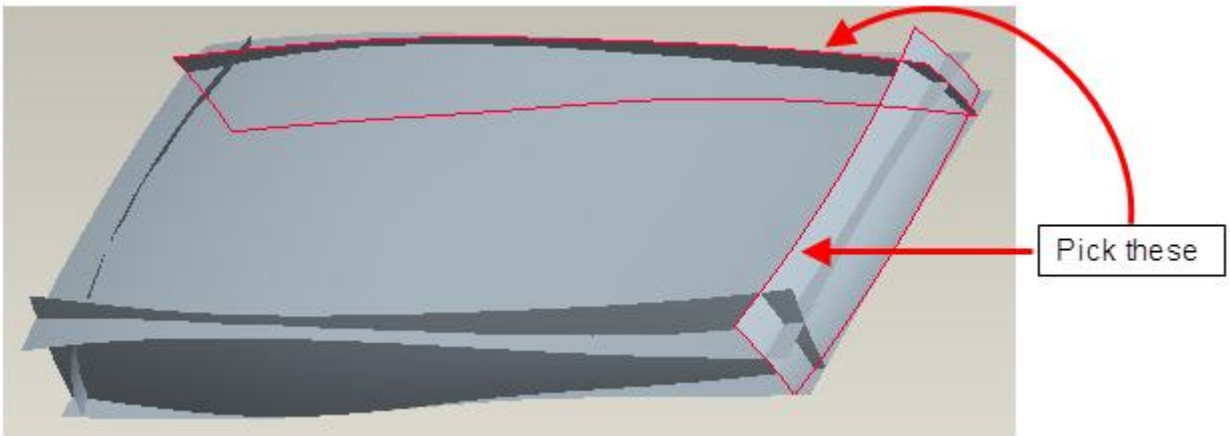
Pick ✔ after completing the sketch.

Pick ✔ to complete the sweep feature.

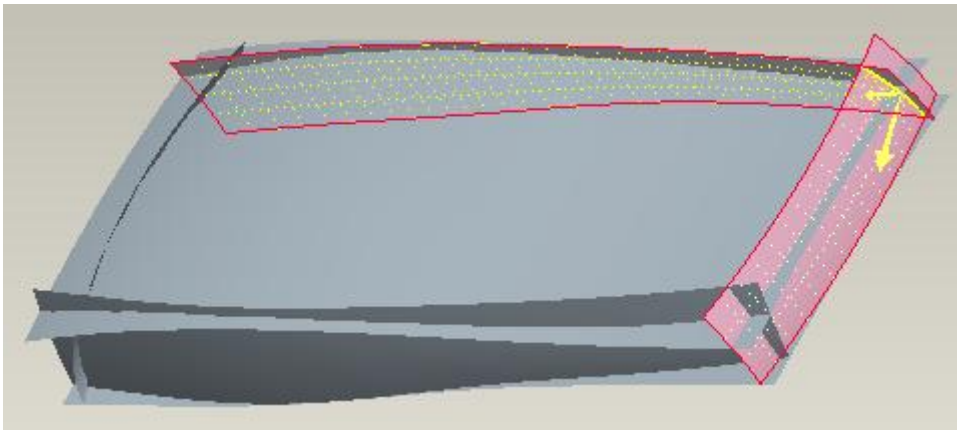
Merging the Quilts

Now we will merge the surfaces together. But before that hide PRT_ALL_CURVES layer.

Select the following features (highlighted) while holding down the Ctrl key.

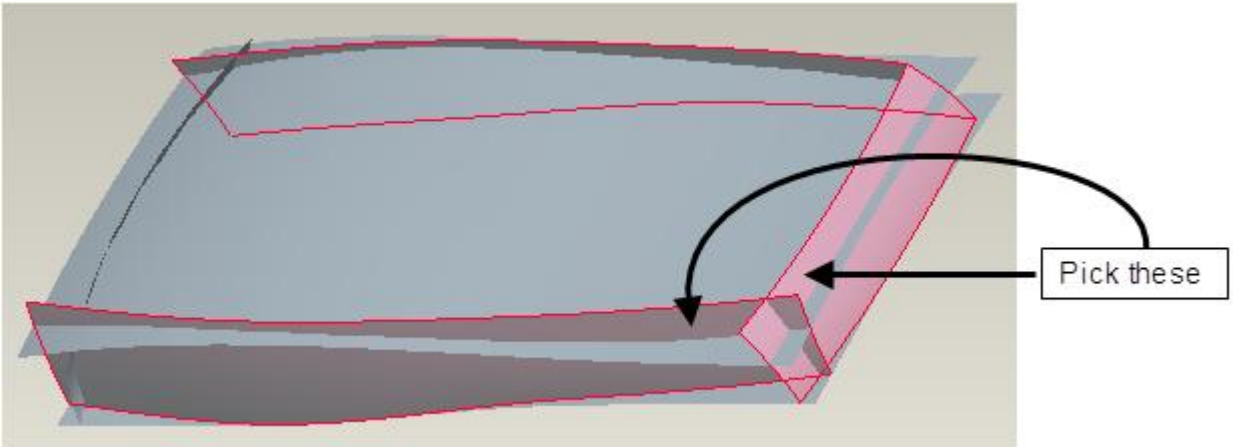


Pick Merge to invoke the Merge tool. Make sure that preview appears as shown below.



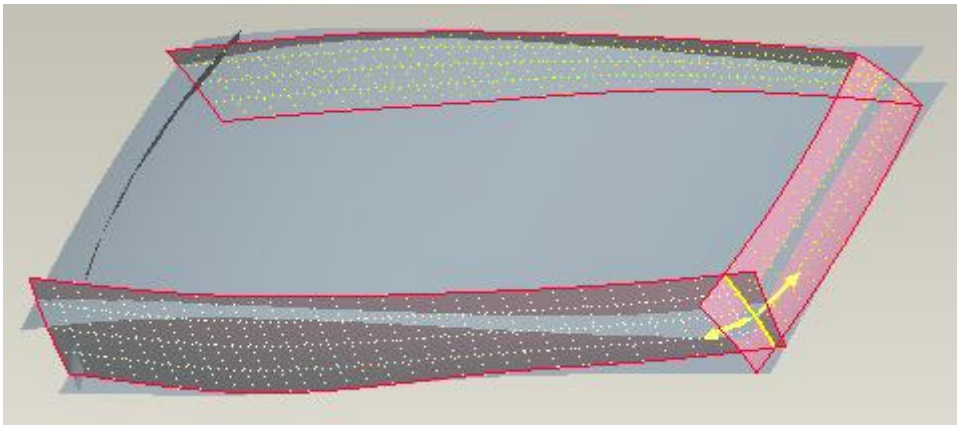
Pick  to complete the feature.

Now select the merged quilt and mirrored surface feature, while holding down the Ctrl key, as shown below.



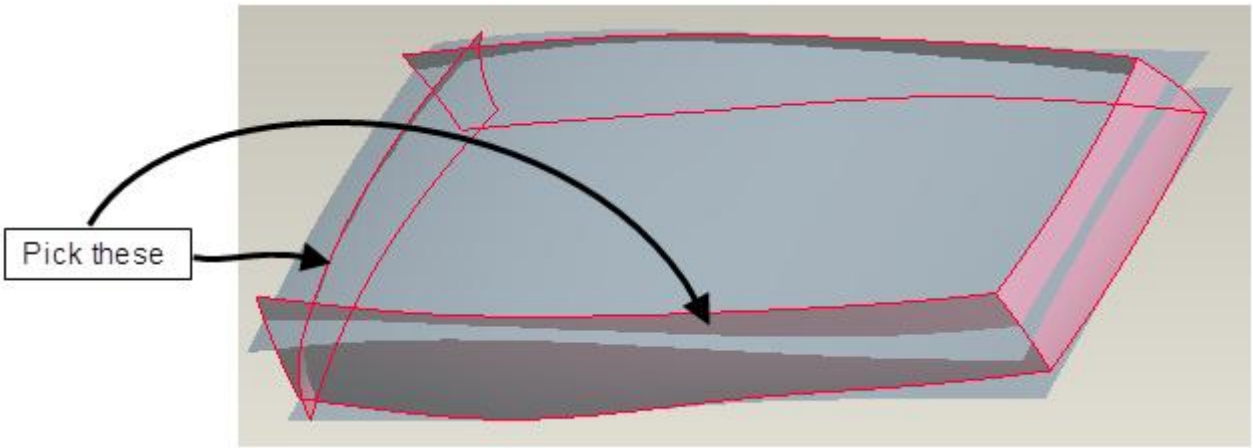
Pick 

and make sure preview appears as shown below.

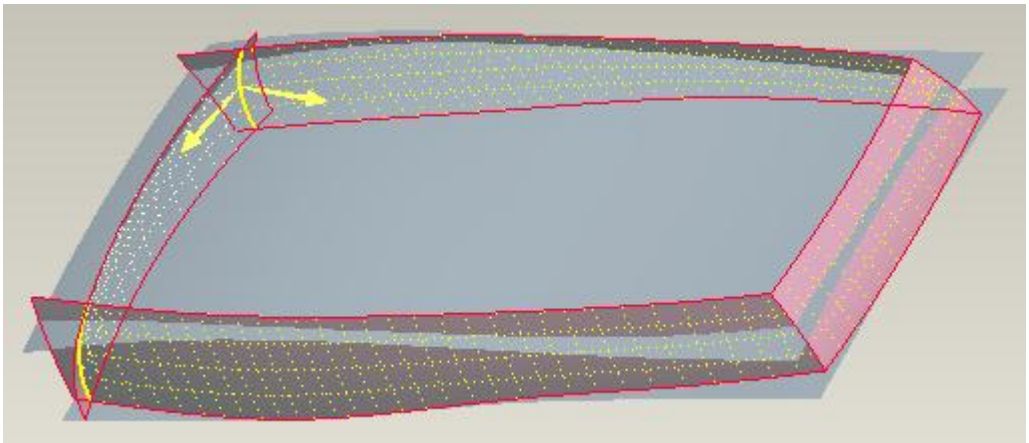


Pick  to complete the feature.

Now select the merged quilt and front surface feature, while holding down the Ctrl key, as shown below.

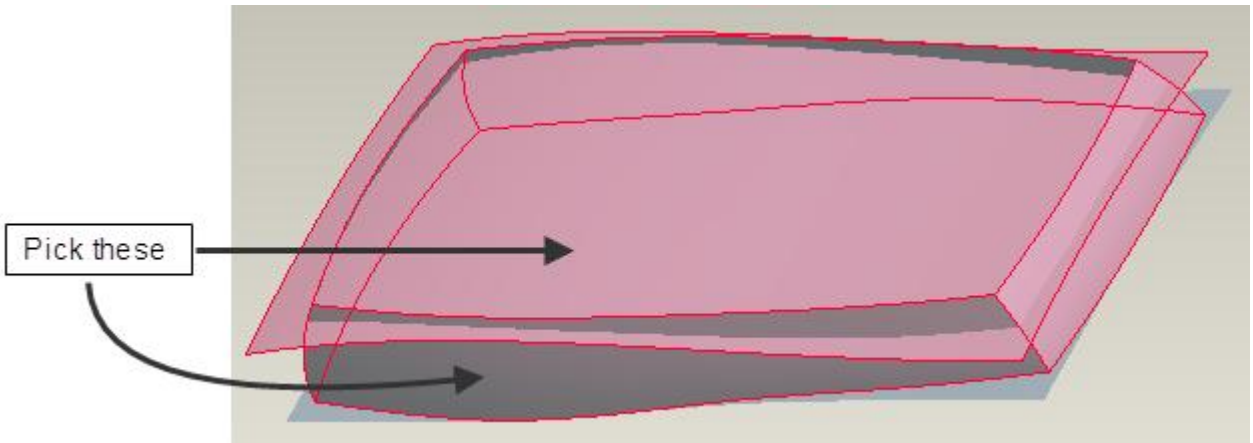


Pick Merge and make sure preview appears as shown below.



Pick to complete the feature.

Now select the merged quilt and top surface feature, while holding down the Ctrl key, as shown below.



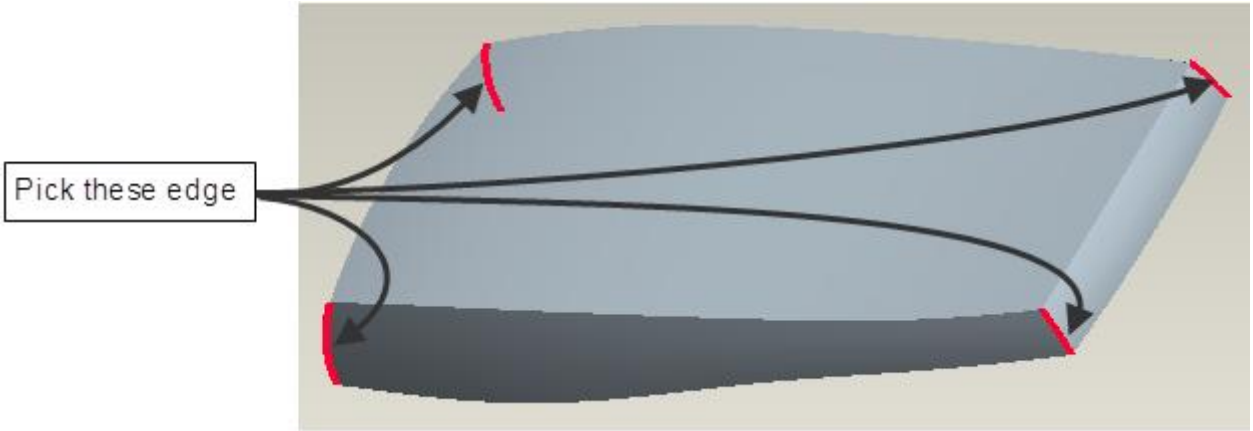
Pick Merge to invoke the merge tool.

Pick to complete the feature. The model will appear as shown below.

Hide the PRT_SURF_SPLIT layer to unclutter screen.

Now we will apply round at the edges.

So select the edges highlighted in the figure below while holding down Ctrl key.



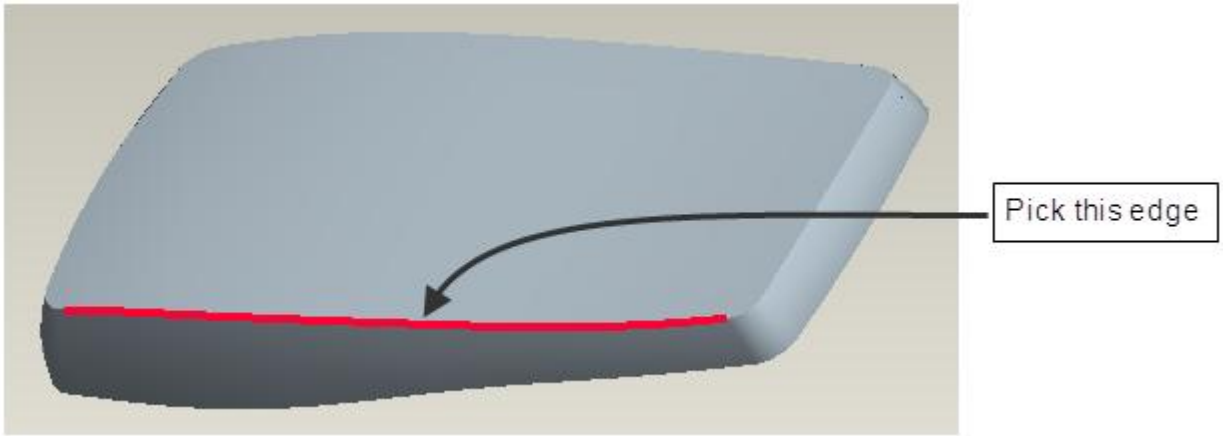
Pick 

to access the Round tool.

Enter ***4*** as the radius value

Pick  to complete the feature.

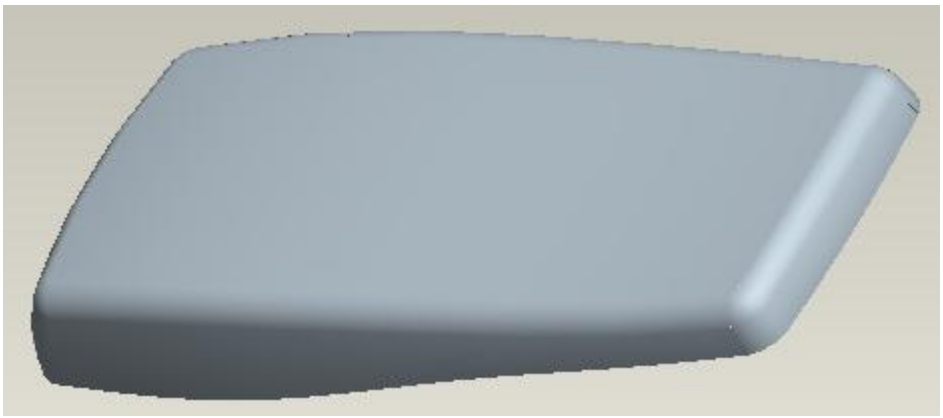
Now select the following edge.



Pick  to access the Round tool.

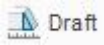
Enter***3.25*** as the radius value

Pick  to complete the feature and part will appear as shown below.



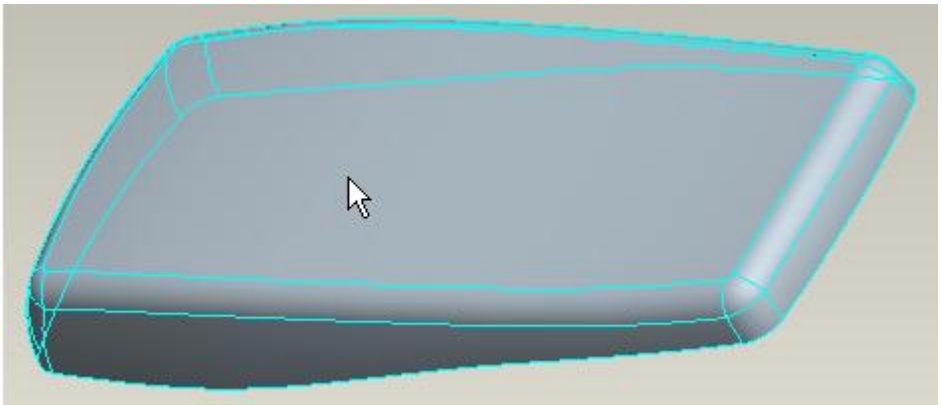
Analyzing the Quilt

Now we will analyze the quilt to determine if it has sufficient draft by using Draft Check tool.

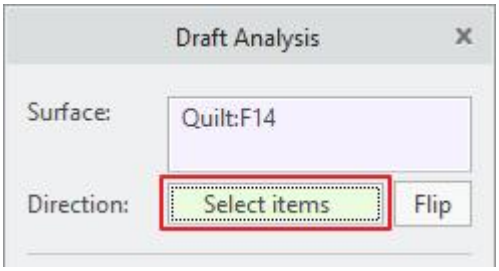
Pick 

on the Analysis tab to access the Draft Check tool.

Now system will ask you to select a surface or quilt to be analyzed. So pick the following quilt.

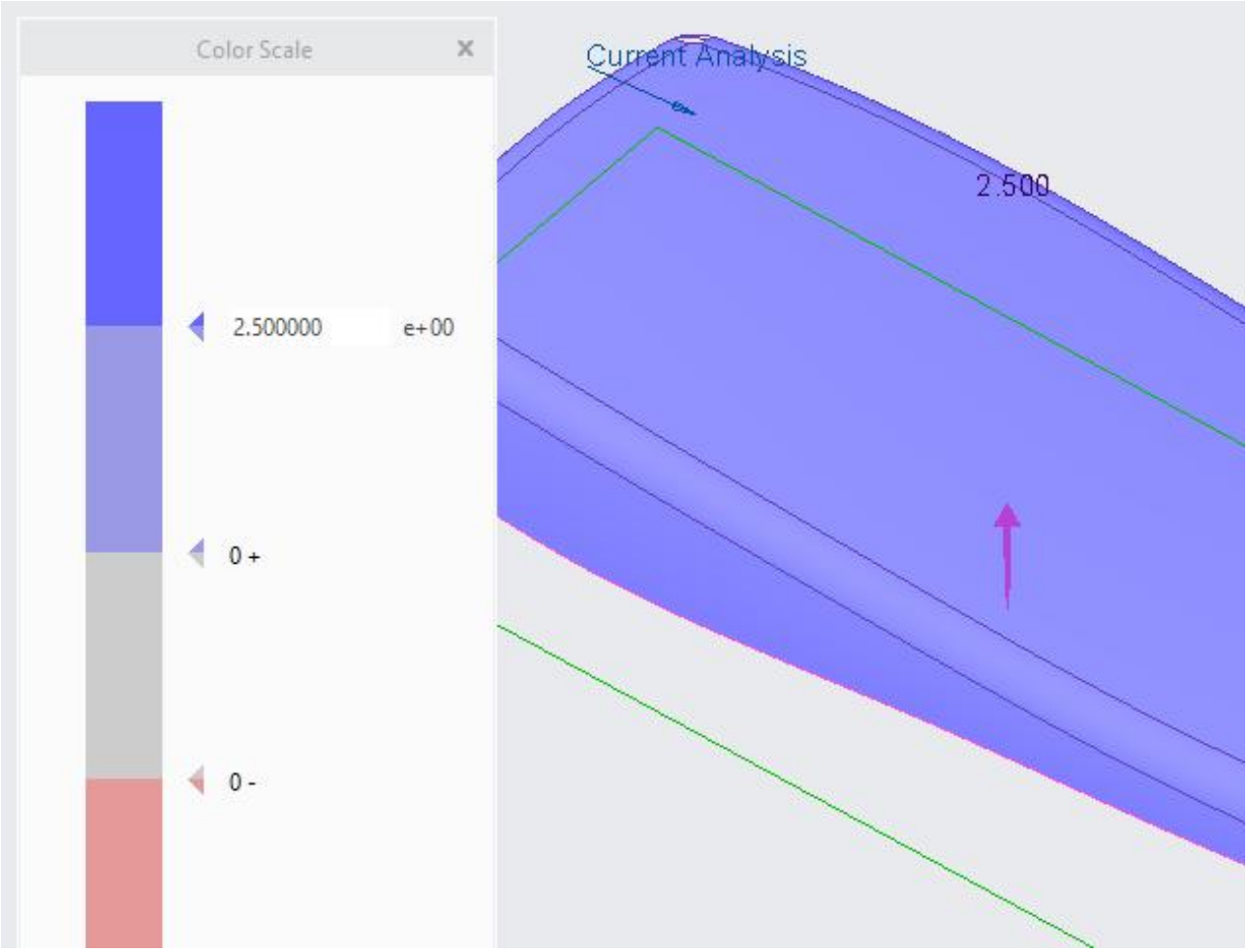


Now pick the Direction reference collector as shown below.



Pick **TOP** datum plane as reference.

Enter ***2.5*** for the draft angle to plot for. Notice that the whole surface has the required draft.

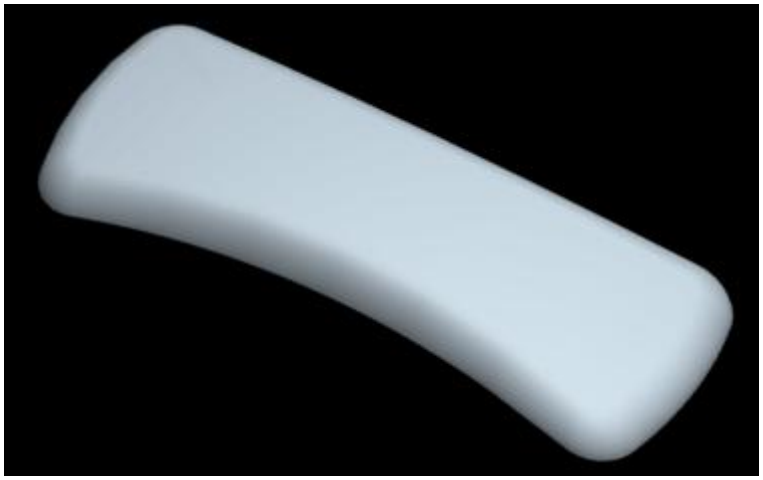


Pick OK to exit the dialog box.

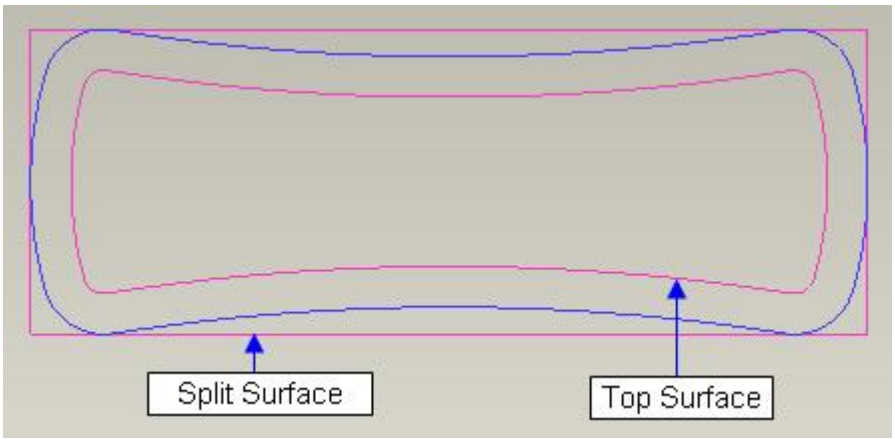
Select **File > Save** to save the work done so far.

Exercise 12

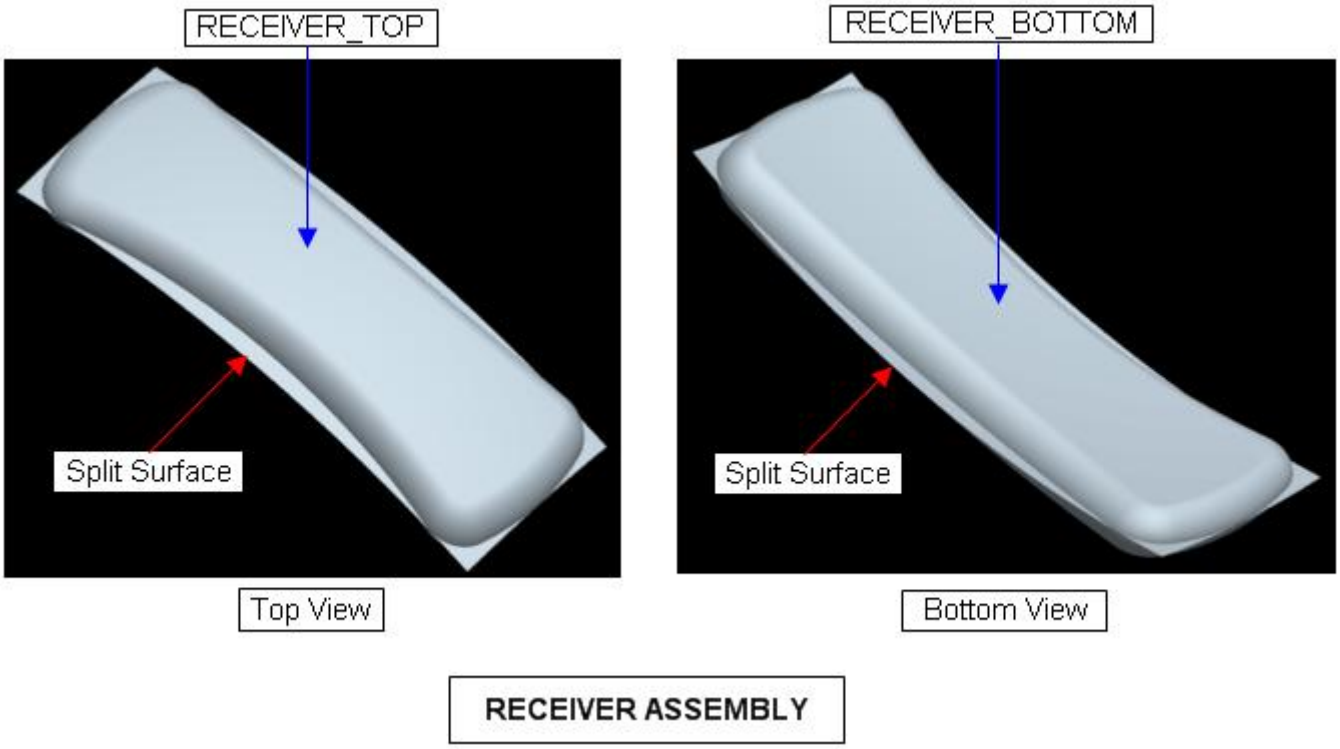
In this exercise we will create the side surface of a plastic component with sweep tool. We will learn to build the draft angle and control tangency with neighboring surface while creating a sweep surface. The completed part is shown below.



Set the working directory to the EXAMPLES folder and open the model RECEIVER_TOP.PRT The part will appear as shown below.



RECEIVER_TOP forms the top part of RECEIVER assembly. RECEIVER assembly consists of RECEIVER_TOP and RECEIVER_BOTTOM components. Split surface separates both of these parts as elaborated in the figure below.



Split Surface is the surface where two components of a product join.

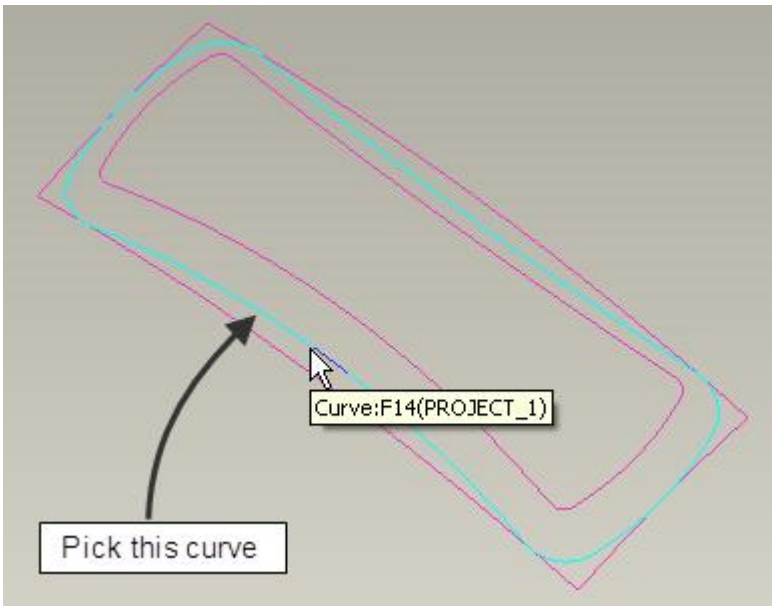
In this exercise we will create side surface of the RECEIVER_TOP part.

Creating the surface

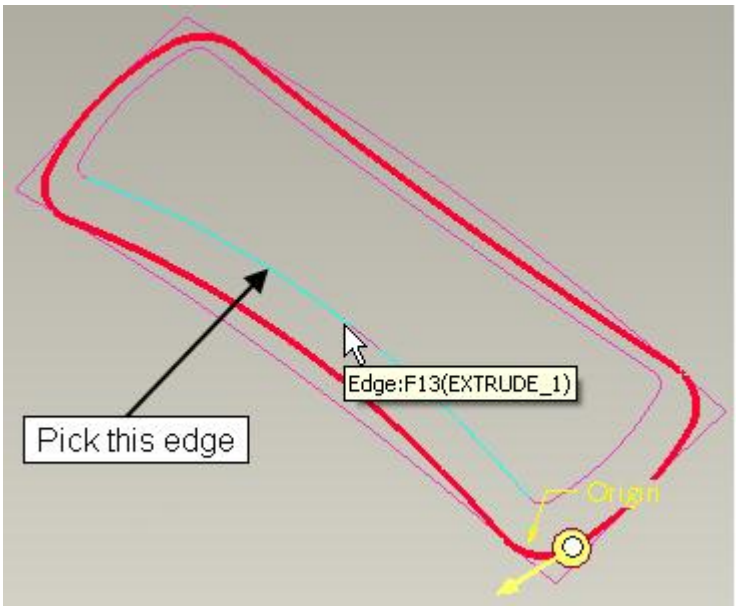
Now we will create the side surfaces with VSS feature. We will build the draft angle in the surface definition.

Pick Sweep icon to invoke the sweep tool.

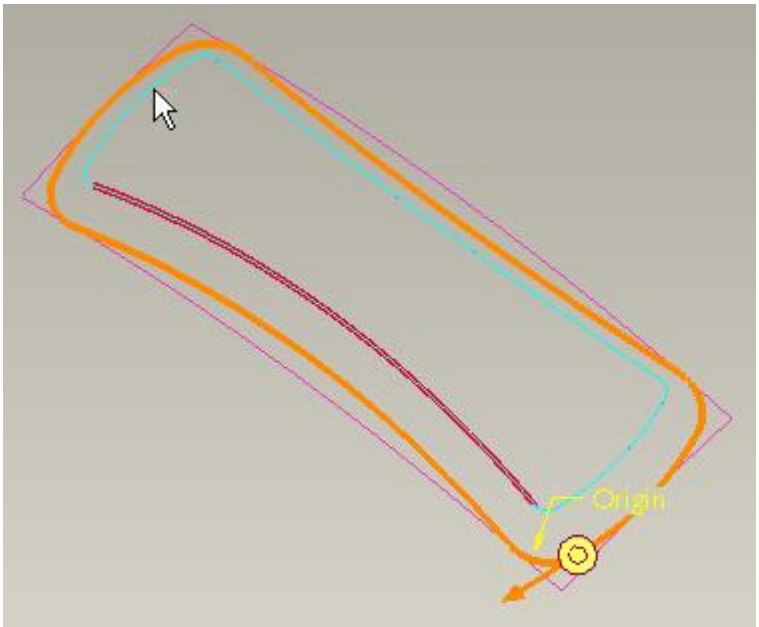
Pick the projected curve as reference.



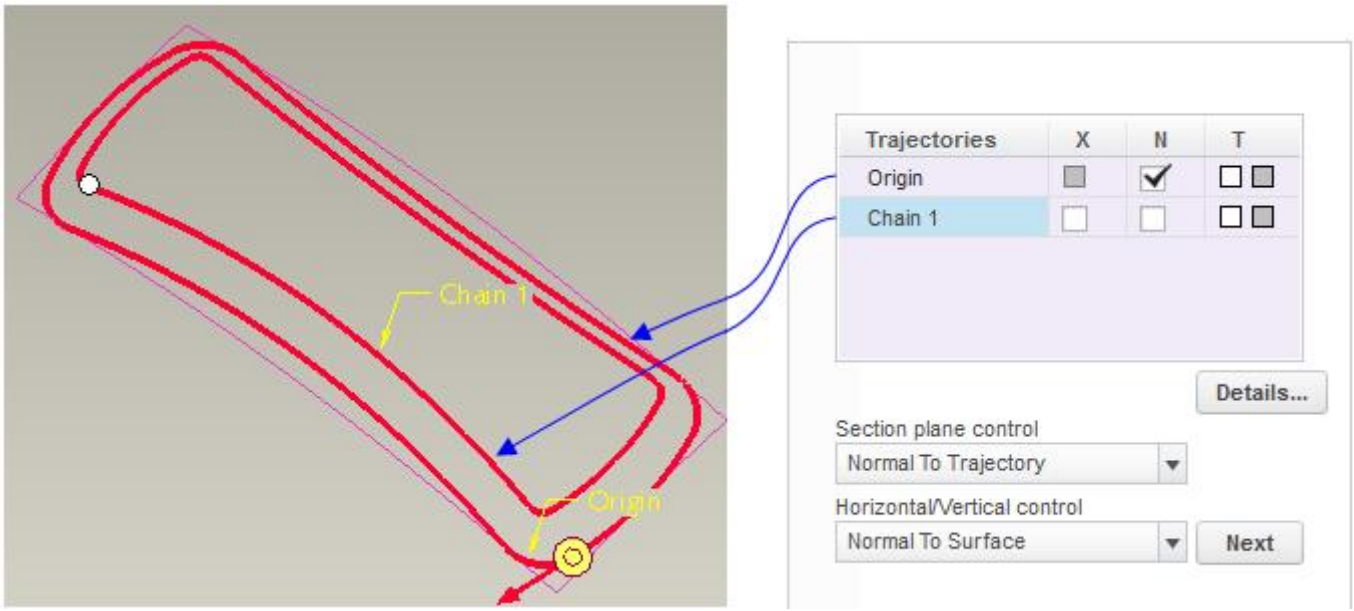
Hold down the Ctrl key and pick following edge. (Note: After picking on edge release the Ctrl key)



Hold down the Shift key and place the mouse pointer over any edge forming the outer loop of the surface and Right-Click to query.



Pick with left mouse button when system highlights the Tangent chain. Selected trajectories will appear as shown below.



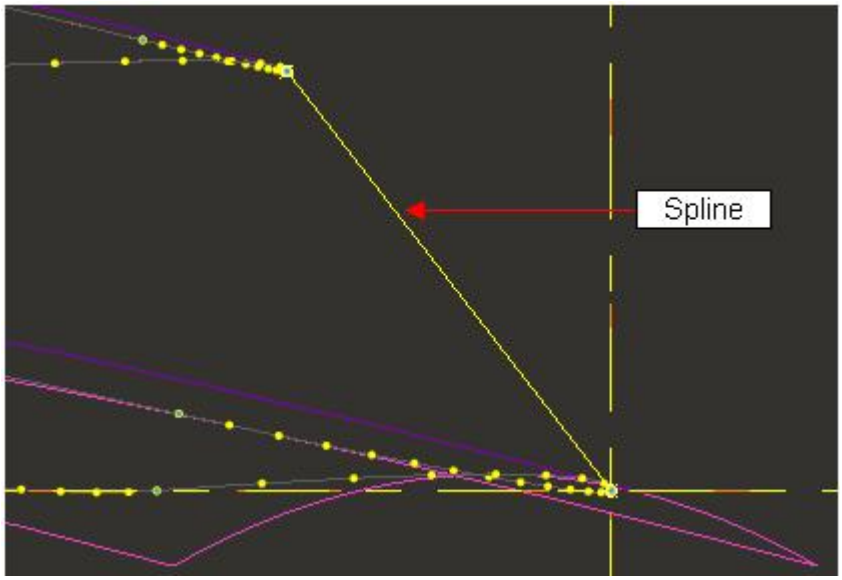
Pick References tab and change the “Section plane control” option to **Normal To Projection**.

Select **TOP** datum plane as direction reference.

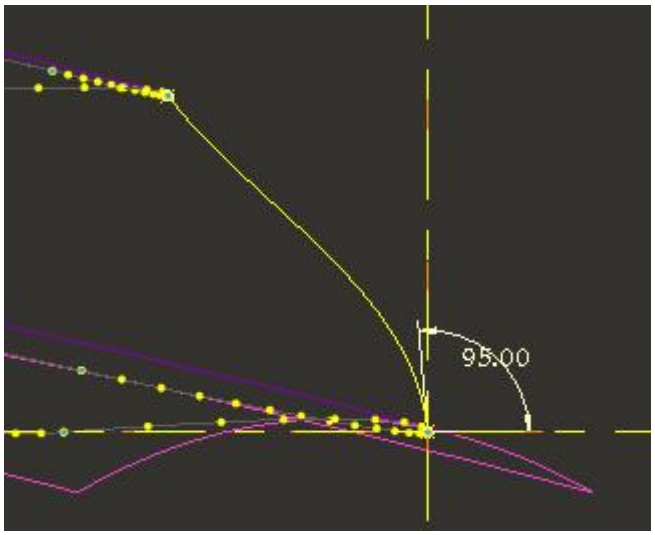
Pick to create the feature as a surface.

Pick to create the sweep section.

Sketch a two point’s spline by picking Spline icon. One end of the spline should lie at cross-hair and other at the reference point for second trajectory as shown below.



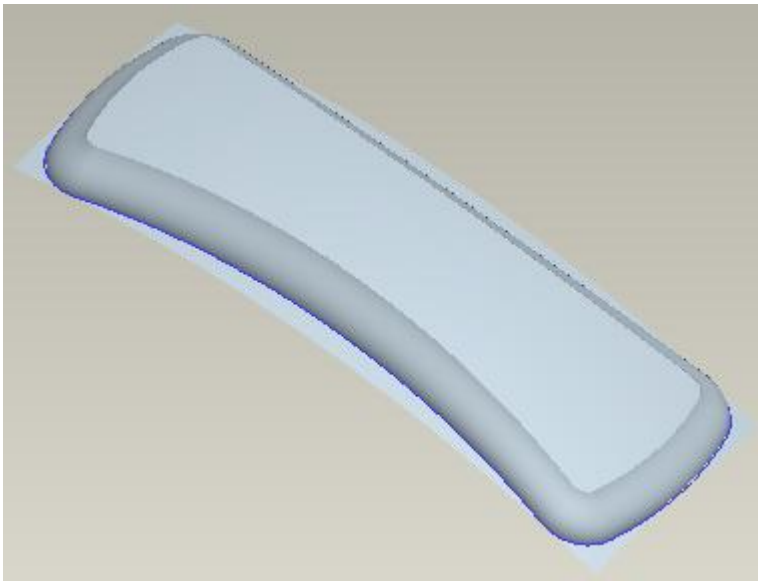
Dimension the spline as shown below. Note: The tangency dimension is created by first picking spline, then the centerline and then the endpoint where the tangency is defined. After this you should pick middle mouse button to place the dimension.



This dimensioning scheme builds the draft in the surface definition.

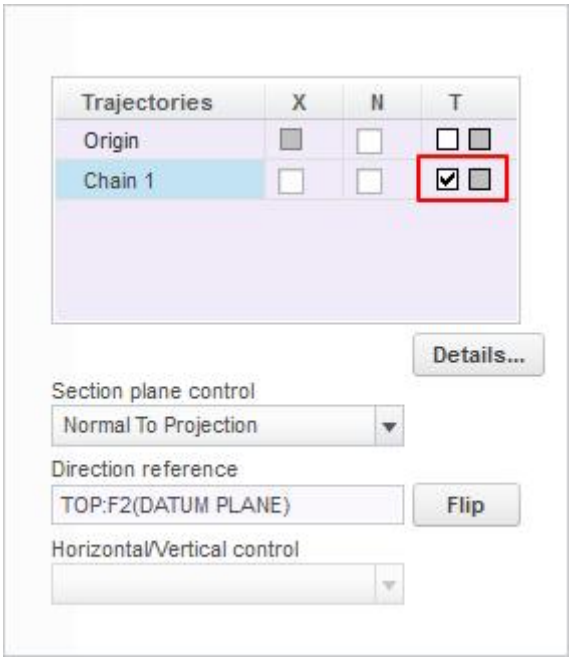
Pick ✔ after completing the sketch.

Pick 👓 to see the preview and it will appear as shown below.



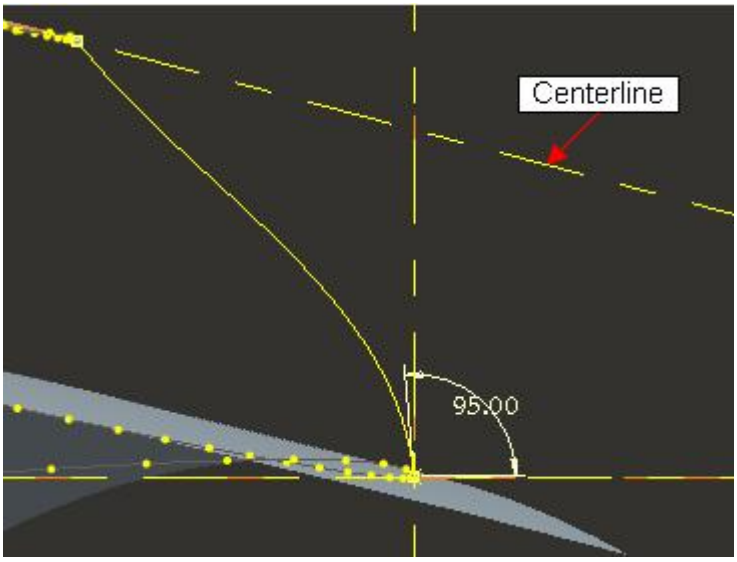
Notice that created surface is not tangent to the top surface. So we will make changes so that the created surface is tangent to the top surface.

Pick References tab and check Tangent constraint box in front of the “Chain 1” as shown in the figure below.



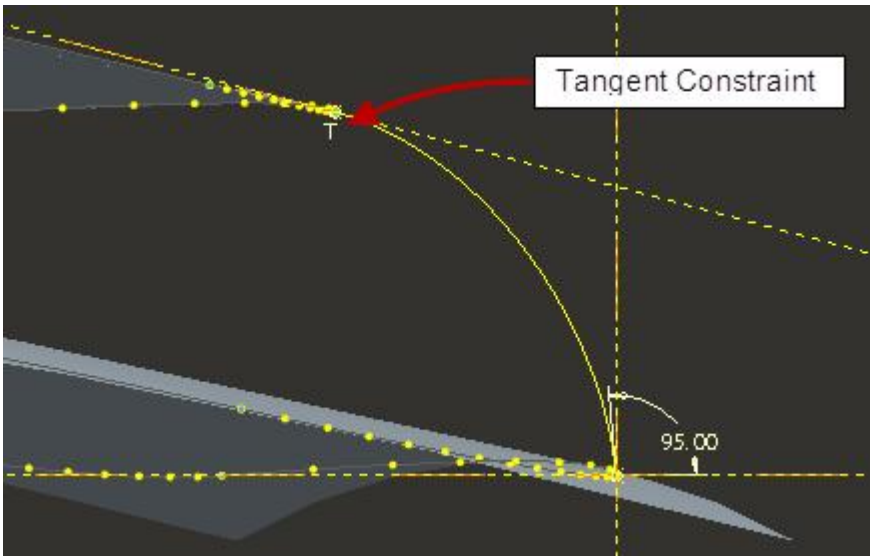
When you assign a trajectory to be a tangent trajectory, Creo Parametric adds a centerline to the sweep section's sketch. This centerline is tangent to the neighboring surfaces at the intersection point of the trajectory and the sketch plane and this tangency is maintained as section is swept.

Pick  and notice the existence of an additional centerline as shown below.

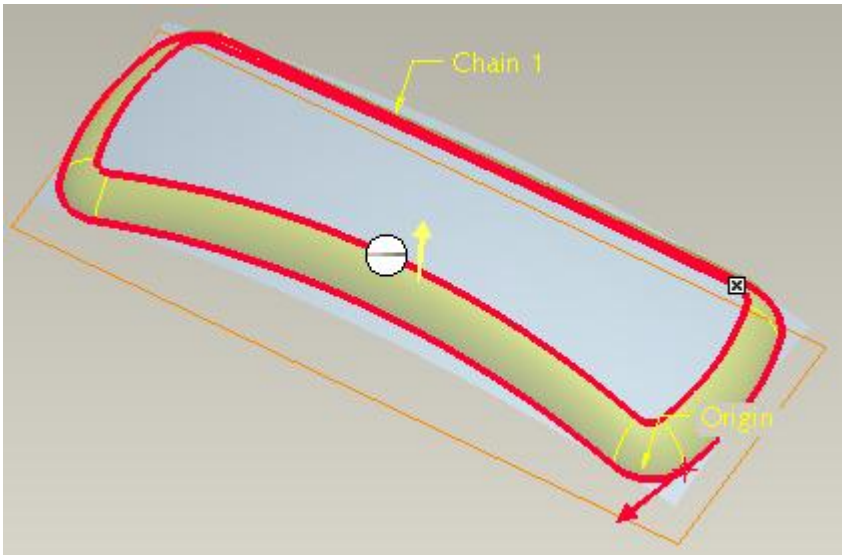


Now pick 

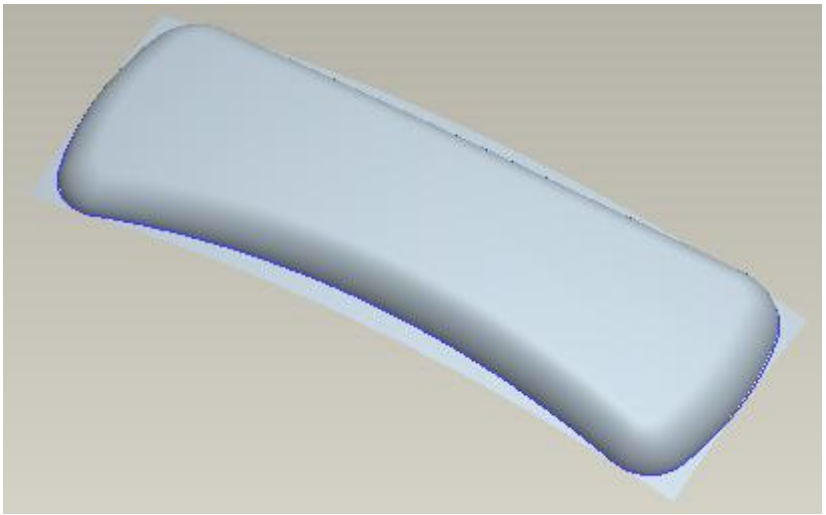
and add Tangent constraint between the centerline and spline (to apply tangent constraint you need to pick the centerline and spline)



Pick ✔ after completing the sketch. Notice that system shows tangent constraint on screen as shown below.



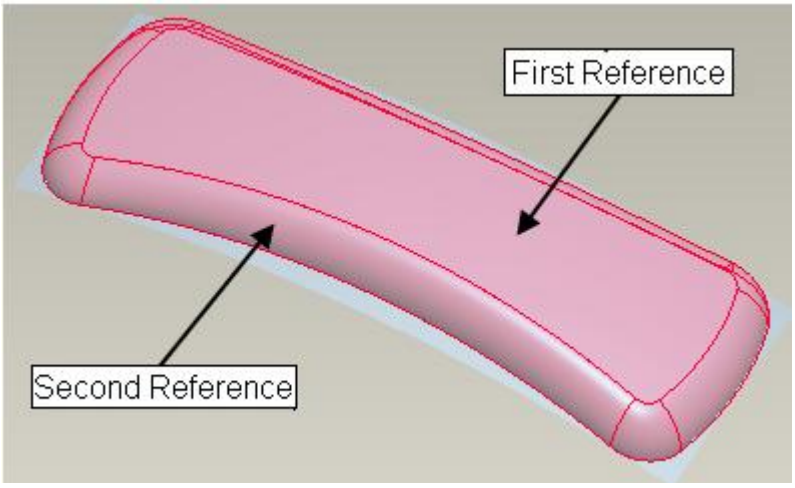
Pick ✔ to complete the feature and part will appear as shown below.



Merging the Quilts

Now we will merge the surfaces together.

First select the top surface and then newly created surface as shown below.



Pick Merge on the Model tab to invoke Merge tool.

Pick Options tab and change the merge type to **Join**.

Join option is used when two surfaces have a common edge. Join option is faster than Intersect option.

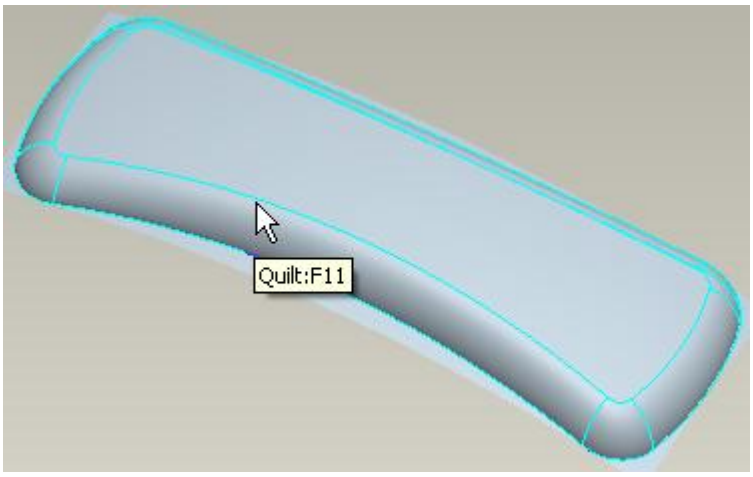
Pick or middle-click to complete the merge feature.

Analyzing the Quilt

Now we will analyze the quilt to determine if it has sufficient draft by using Draft Check tool.

Pick Draft on the Analysis tab to access the Draft Check tool.

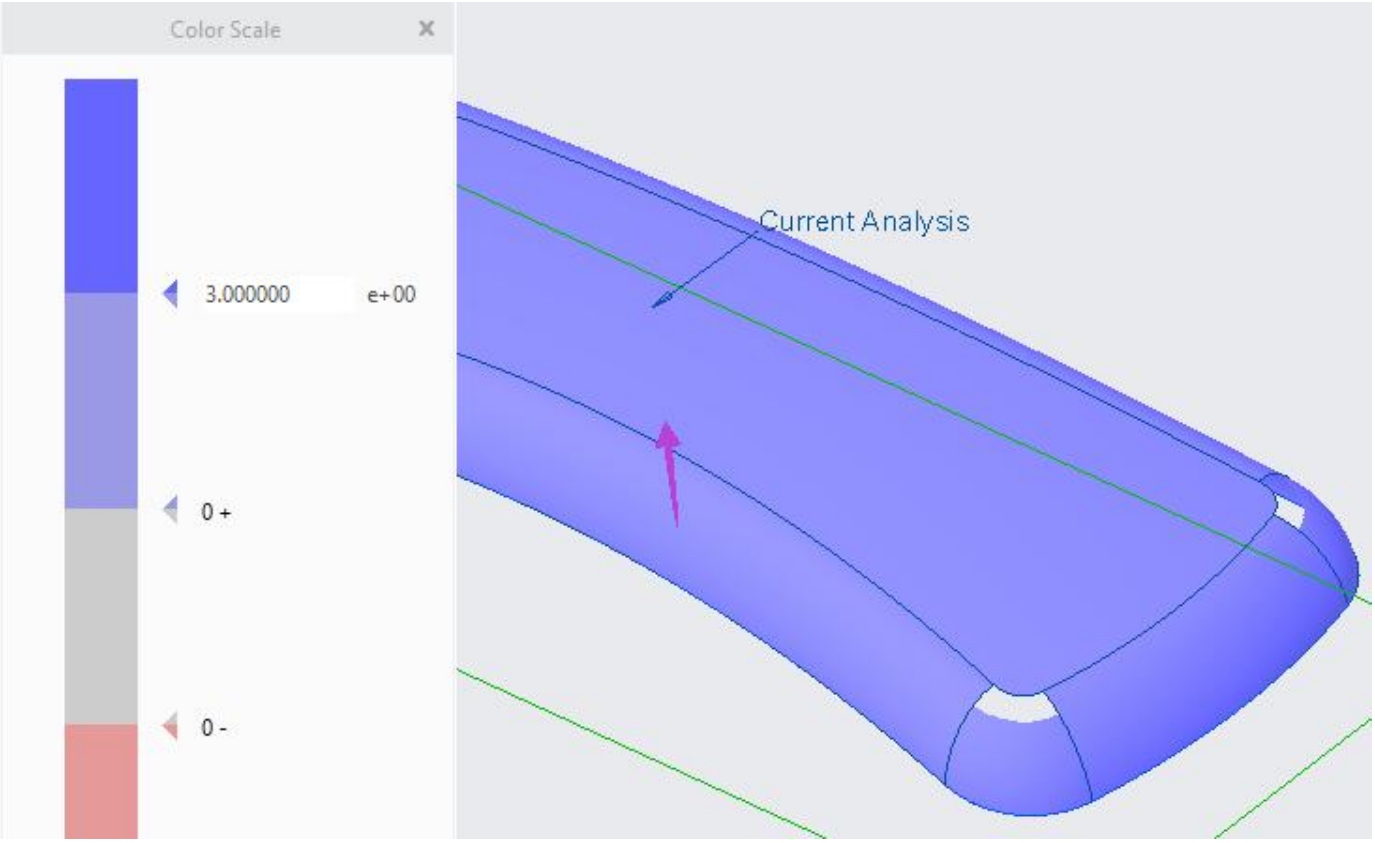
Now system will ask you to select a surface or quilt to be analyzed. So pick the following quilt.



Now pick the Direction reference collector and pick **TOP** datum plane as reference.

Enter ***3*** for the draft angle to plot for.

Notice that the whole surface has the required draft.



Pick OK to exit the dialog box.

Select **File > Save** to save the work done so far.