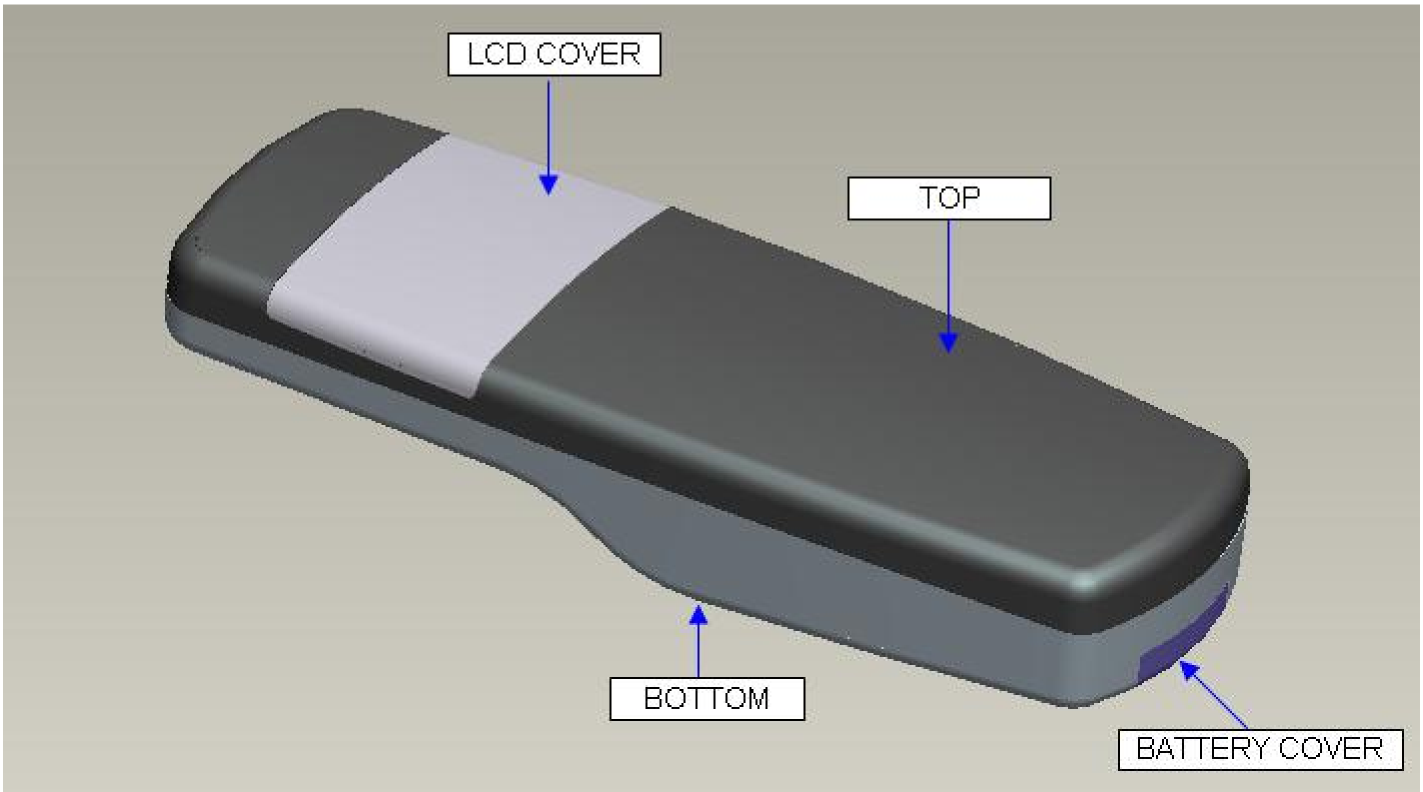
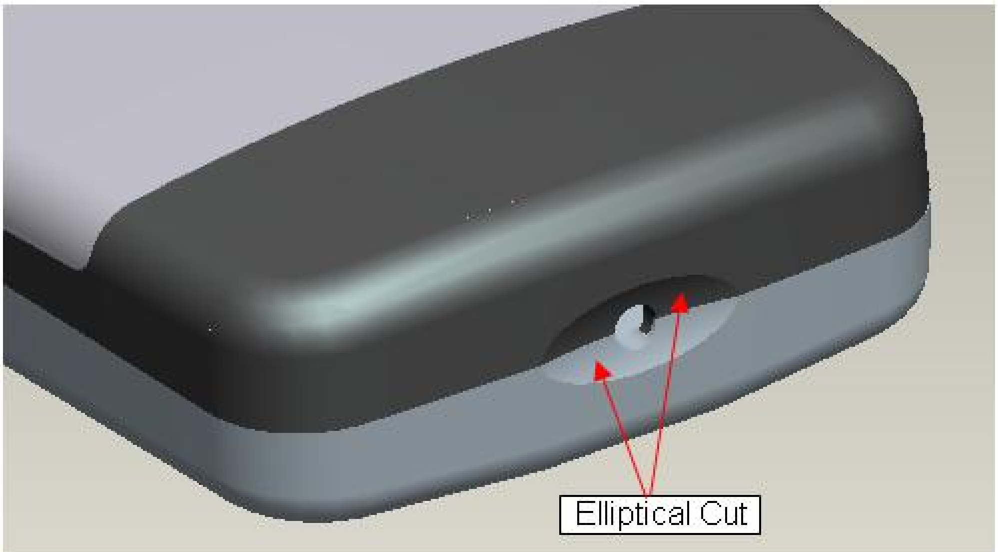
Exercise 2

In this exercise we will create a plastic housing for a remote-control consisting of four parts. The completed product is shown below.



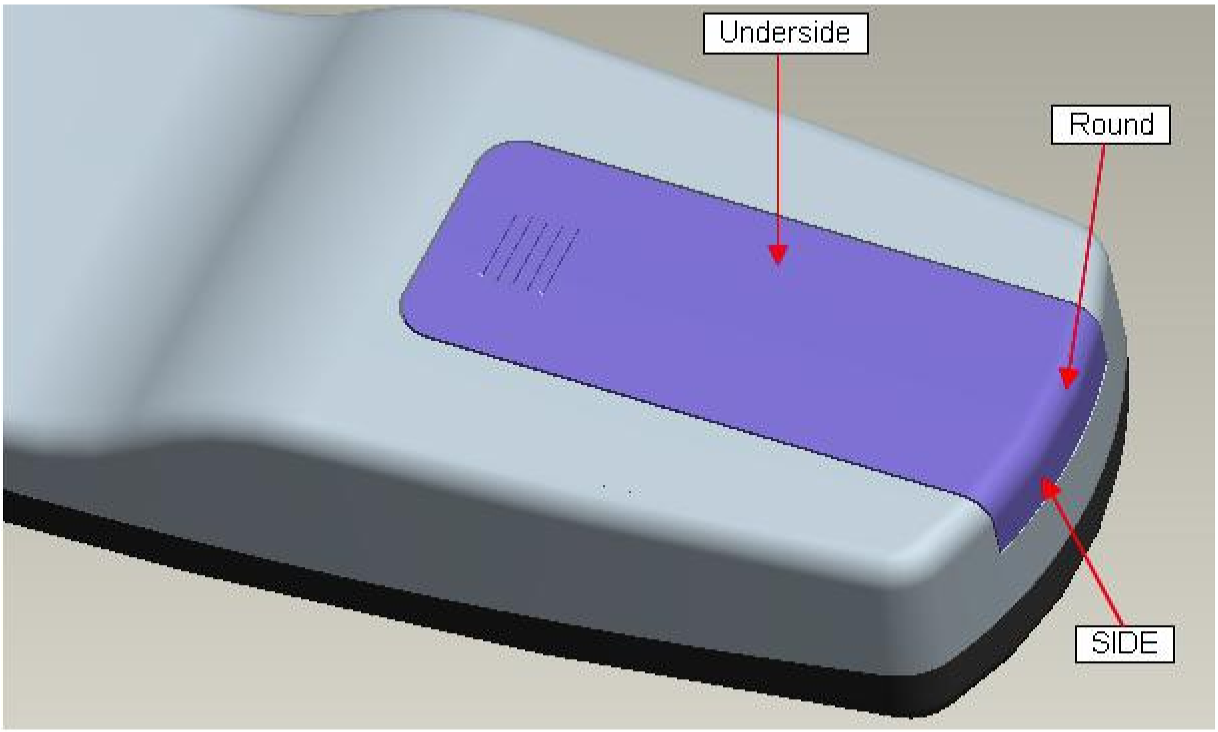
The front side of the remote is shown below



The common attributes between TOP and BOTTOM parts are as follows

1. Profile of both components is same.
2. Both components join along a 3-D surface
3. The hole at front side should be aligned and of same diameter in both parts
4. The elliptical cut at the front side should be of same size and shape in both parts.

The bottom side of the remote is shown below. Nomenclature used for different areas of battery cover is also shown in the figure.



The common attributes between BOTTOM and BATTERY COVER parts are as follows

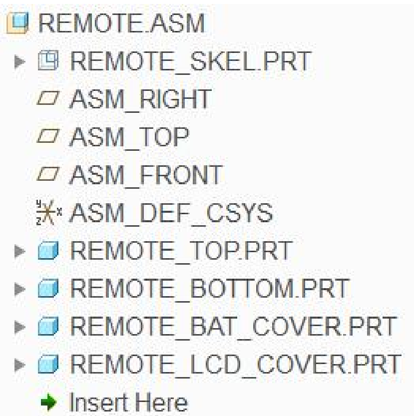
1. Profile of BATTERY COVER and accommodating cut in the BOTTOM part is same.
2. The outer shape of both parts (including underside and side) follow common surfaces
3. The round value is same for both parts

So we will create the geometry in the skeleton model that controls the above common attributes.

Only, the geometry that affects or crosses more than one component (or has to be referenced by more than one component) should be placed in the skeleton.

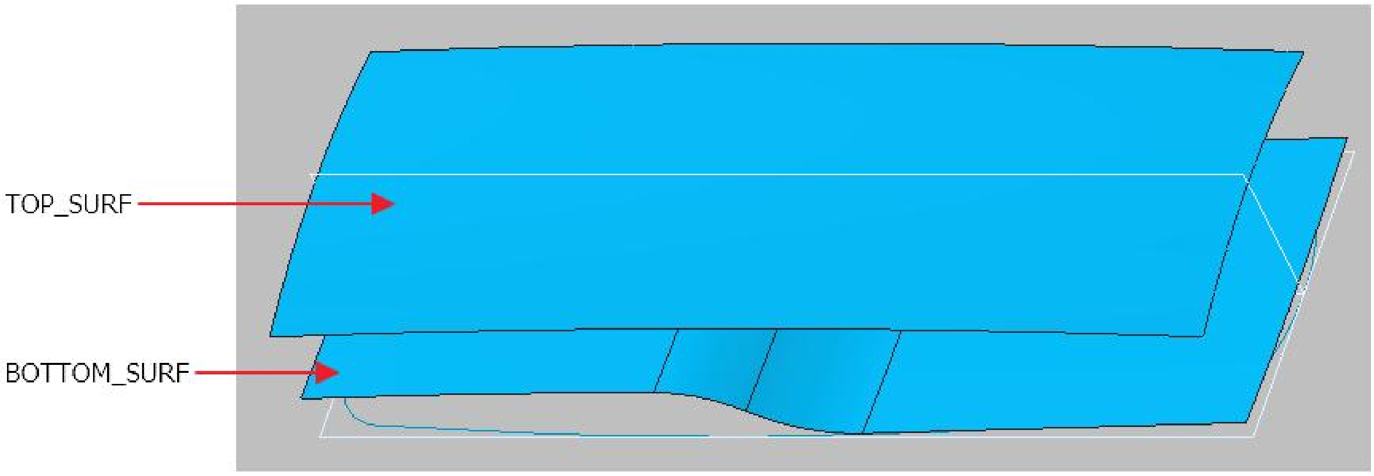
Set the working directory to REMOTE folder and open the assembly REMOTE.ASM

Notice that there are four components assembled with Default constraint.



Building the Skeleton

Open the skeleton model in a separate window and it will appear as shown below. Notice down the name of the quilts in the model tree. These quilts will be referred by these names in the following text.

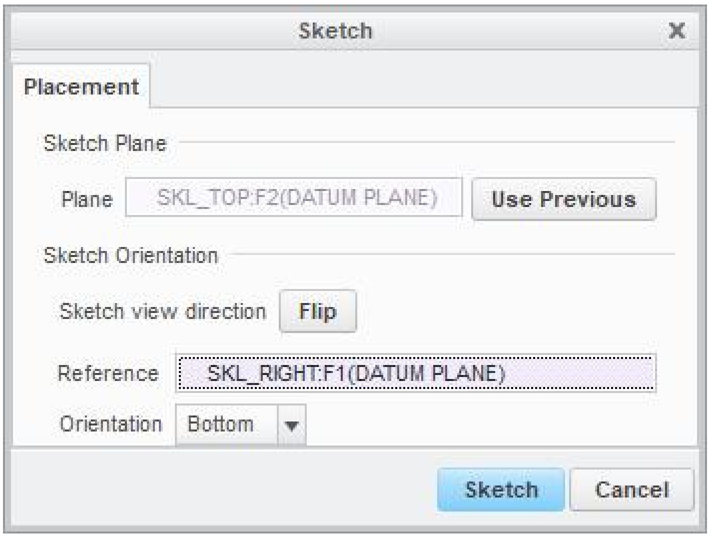


Here you can see top and bottom surfaces and profile datum curve. Other geometry is placed on custom layers and hidden. The white curves represent the horizontal and vertical boundaries of the remote assembly.

We will create the skin of the remote-control completely in the skeleton model because top, bottom and side surfaces and rounds are shared by multiple parts.

Pick the Extrude Tool icon  to invoke Extrude tool.

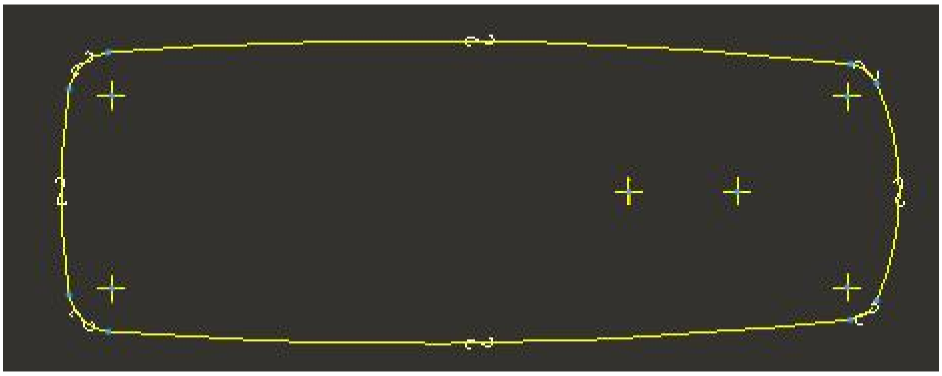
Select the sketching references as shown below



Hide the 06___PRT_ALL_SURFS layer for clear view.

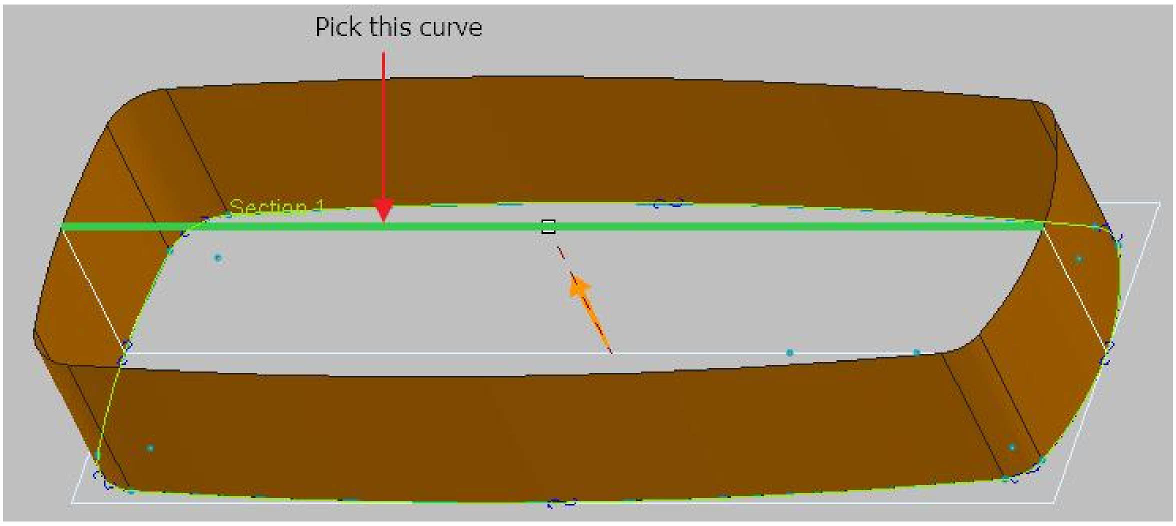
Pick 

icon and select the loop shown in figure below.



After completing the sketch, pick  to create the feature as surface.

Change the depth option to **To Selected** () and pick the following curve.

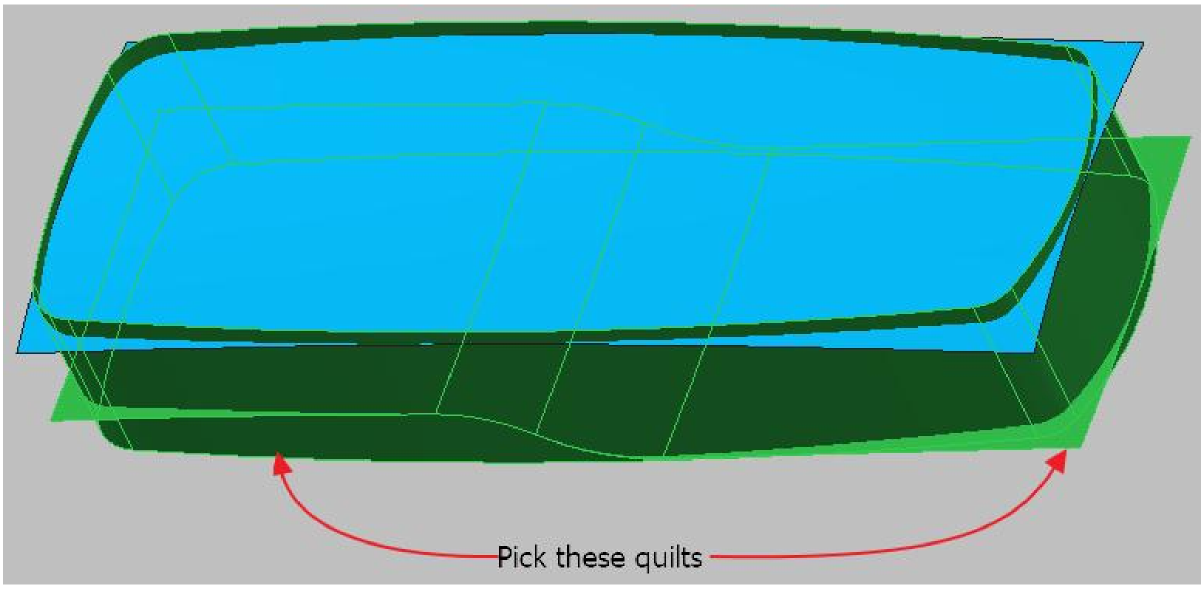


Pick  or middle-click to complete the feature.

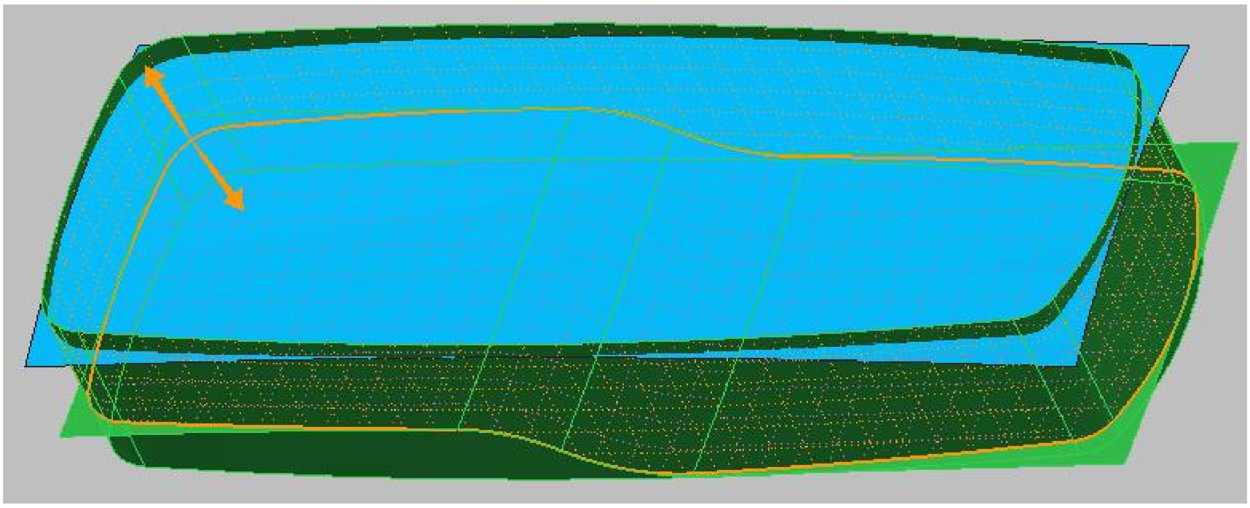
Before proceeding hide the SKL_CURVE_BOUNDAY and SKL_CURVE_PROFILE layers. Also unhide the 06___PRT_ALL_SURFS layer before proceeding.

Now we will merge the surfaces together to get the outer skin of the remote-control body. First change the filter setting to **Quilts** for ease of selection as shown below.

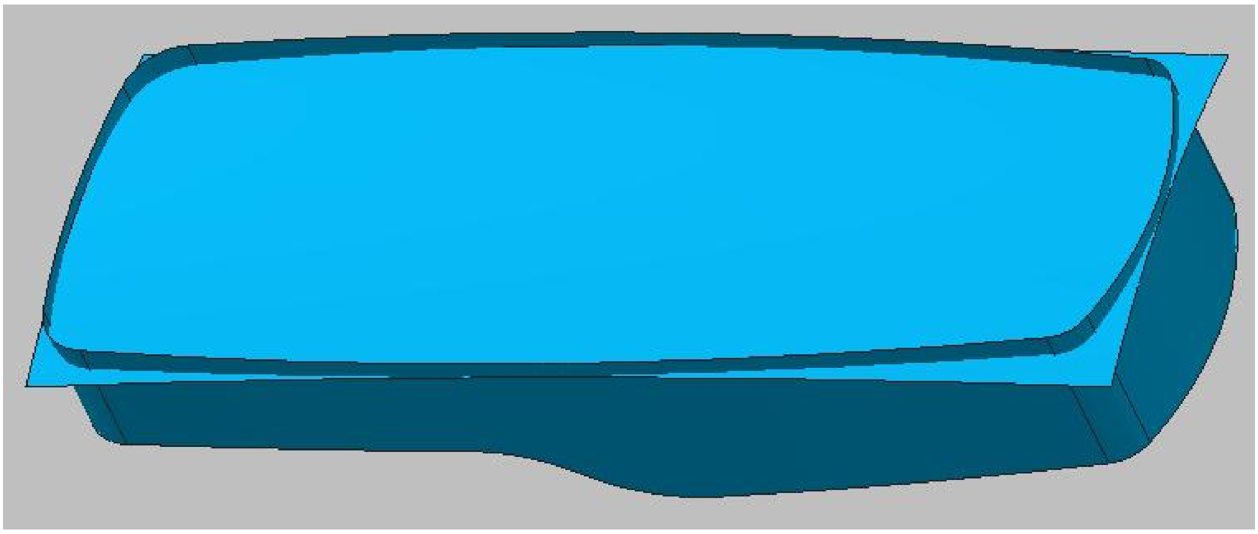
Select the extruded quilt and BOTTOM_SURF quilt while holding down the Ctrl key as shown in the figure below. Set the view to **ISO1** to avoid any ambiguity.



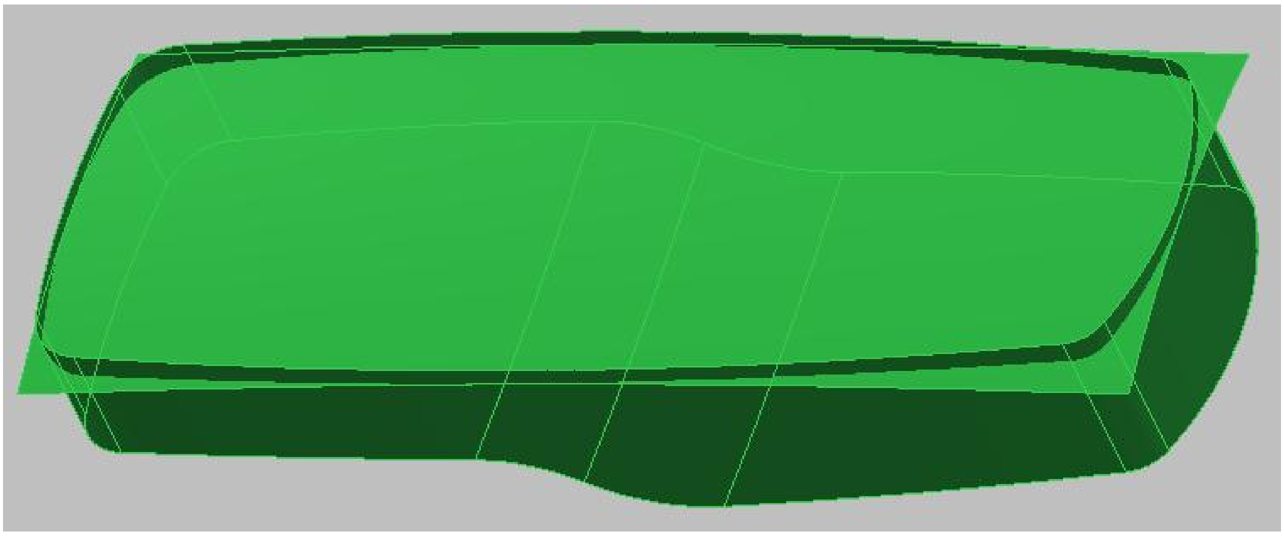
Pick  to invoke the Merge tool. Make sure that preview appears as shown below.



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



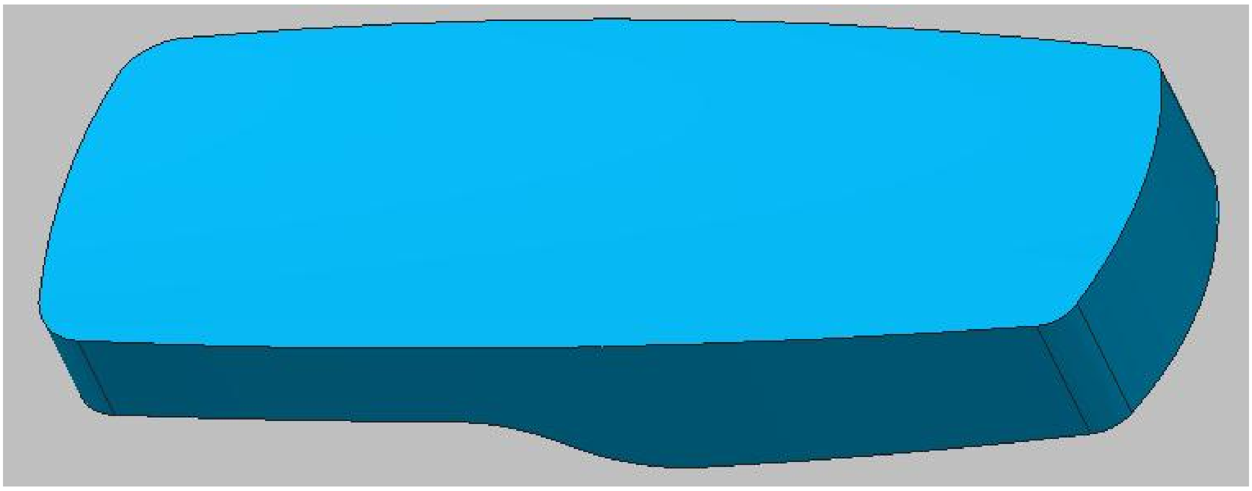
Now select the merged quilt and TOP_SURF while holding down the Ctrl key as shown in the figure below.



Pick 

to invoke the Merge tool.

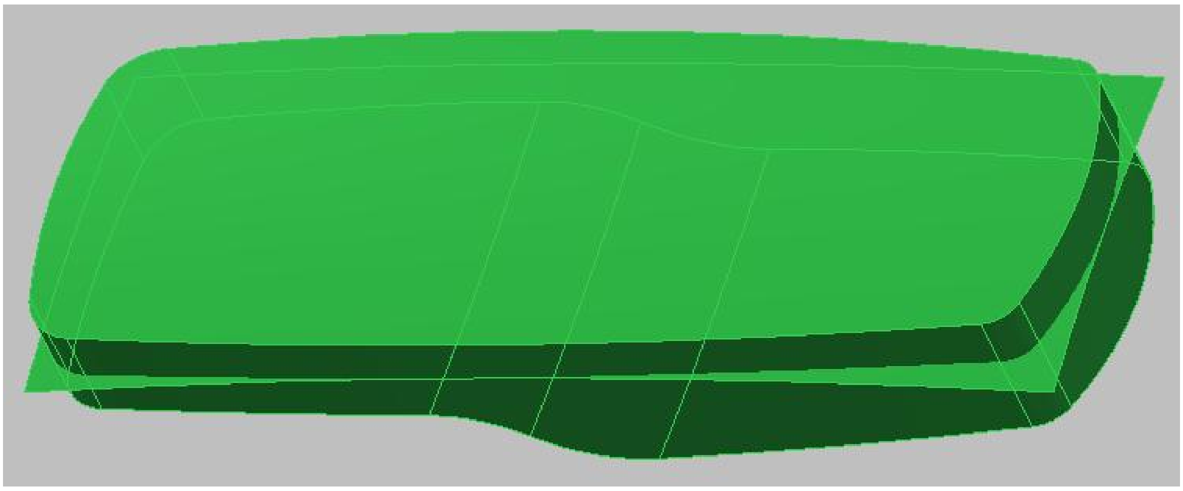
Pick  to complete the feature and part will appear as shown below.



Now we will apply the split draft to the side surfaces so that TOP and BOTTOM parts are manufacturable using injection molding process. The side surfaces will be split about the split surface (the surface at which both components join). To achieve this, we need to create a datum curve to be used as neutral curve.

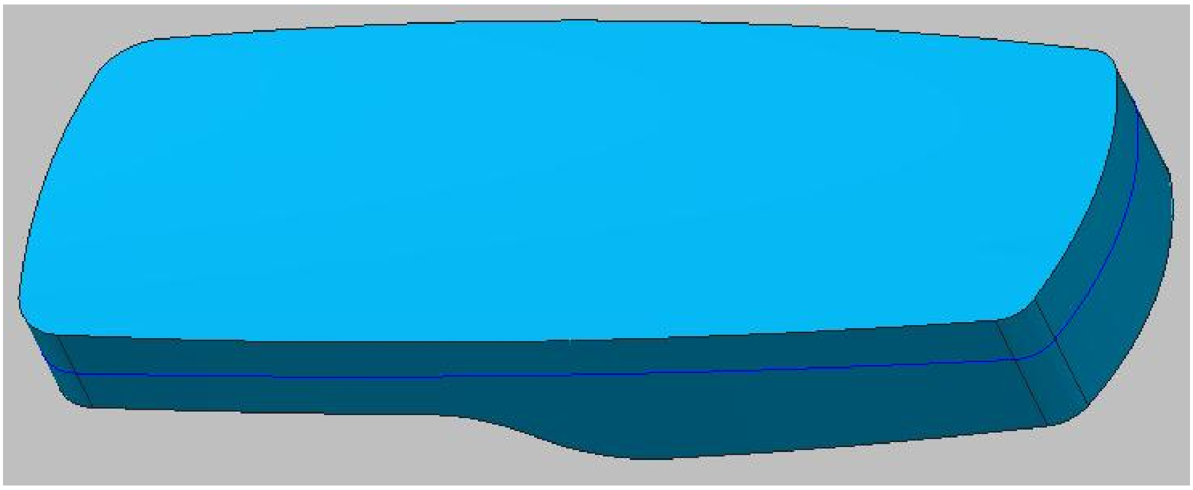
First unhide the **SKL_SURF_SPLIT** layer to view the split surface.

Select the merged quilt and split surface while holding down the Ctrl key as shown in figure below.



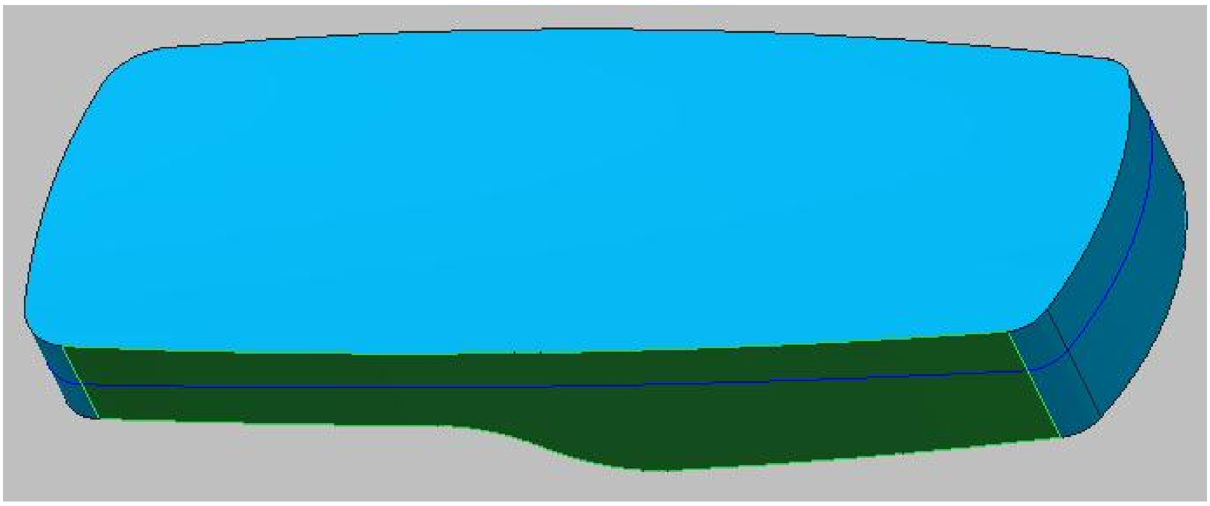
Pick Intersect on the Model tab to invoke the Intersect Tool and intersected datum curve will appear on the screen.

Now hide the **SKL_SURF_SPLIT** layer and component will appear as shown below.



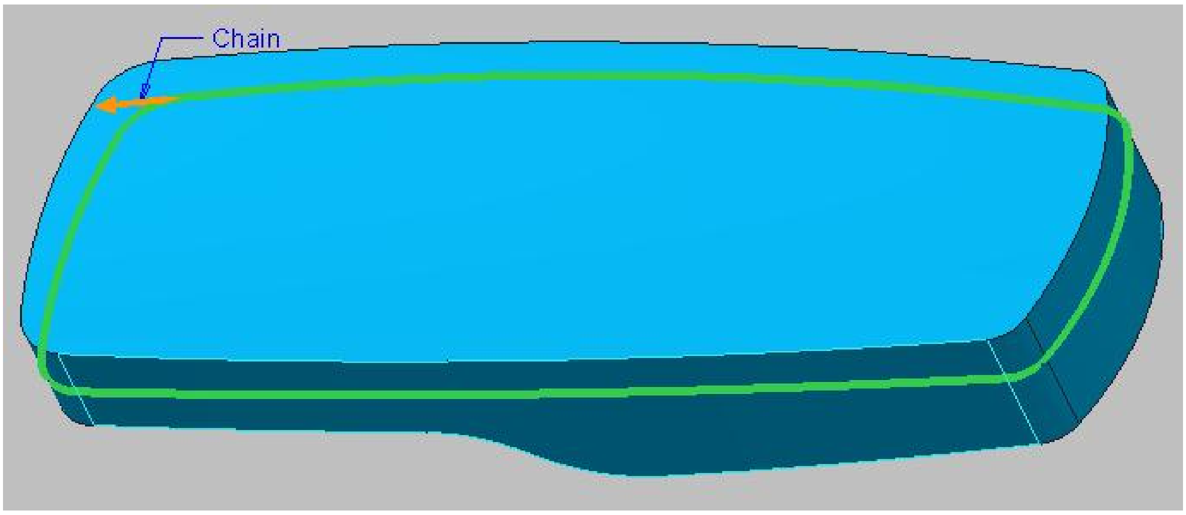
Note: We could also create the intersected curve before merging the quilts. It will result in less parent child dependencies although result will be the same. We have created here this curve just to continue the flow of the exercise.

Select the highlighted surface shown in figure below (make sure that filter is set to Smart).



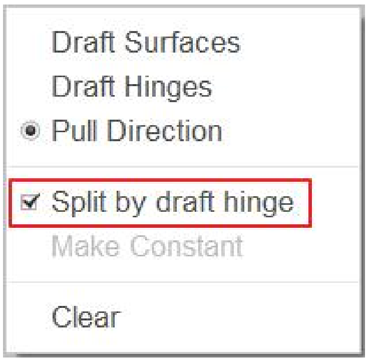
Pick Draft to access the Draft tool.

Notice that Draft hinges collector is active so select the intersected curve as shown below.

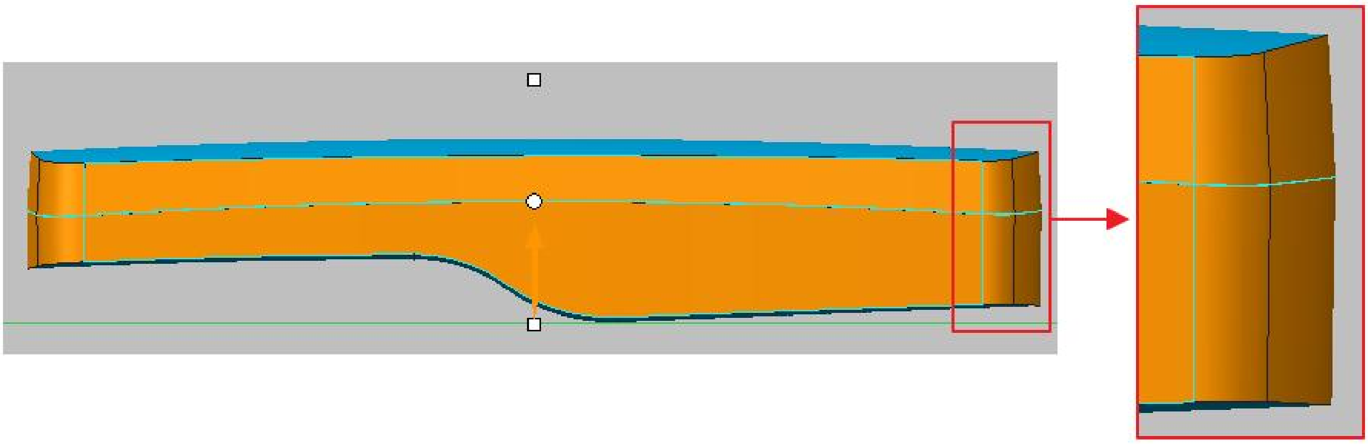


Pick in the Pull Direction collector and select the **SKL_TOP** Datum plane as reference.

Right-click in the graphics window and select **Split by draft hinge**.

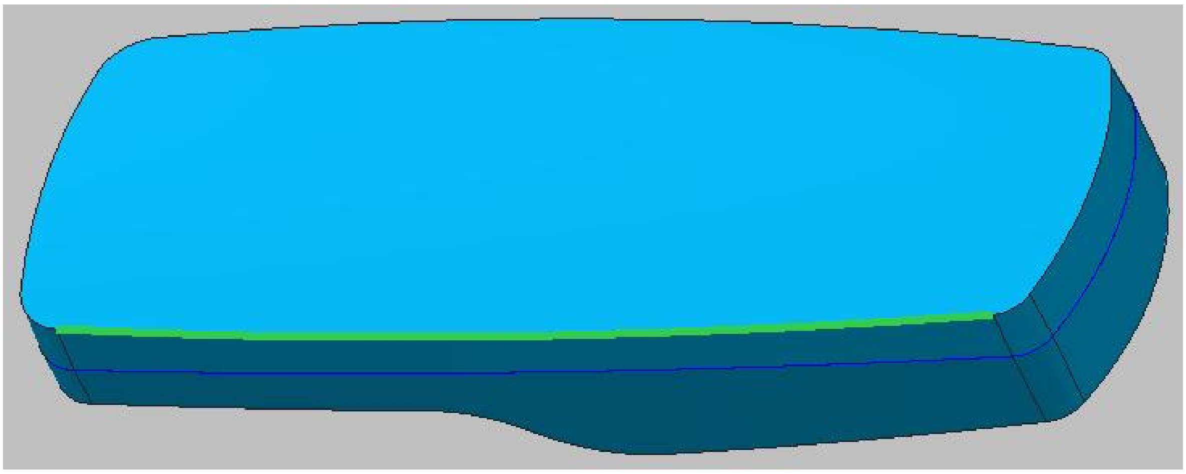


Enter ***3*** as Draft angle for both sides and make sure that preview appear as shown below.



Pick icon to complete the feature.

Now select the following edge for applying round.

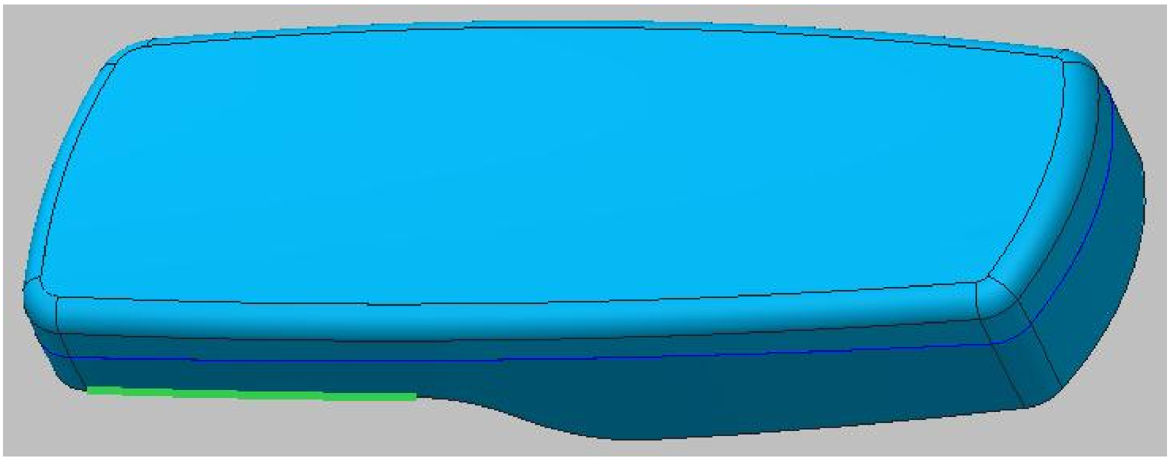


Pick Round to access the Round tool.

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

Pick icon to complete the feature.

Now select the following edge for applying round.



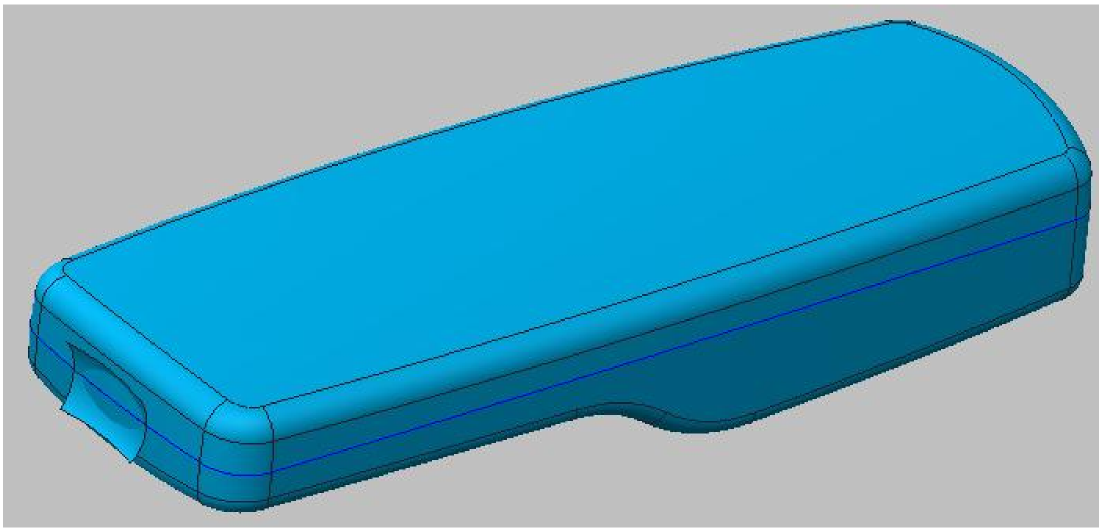
Pick Round to access the Round tool.

Enter ***3*** as the radius value

Pick icon to complete the feature.

We have drafted the side walls and applied rounds in the skeleton because the surfaces created by these features are shared by multiple parts.

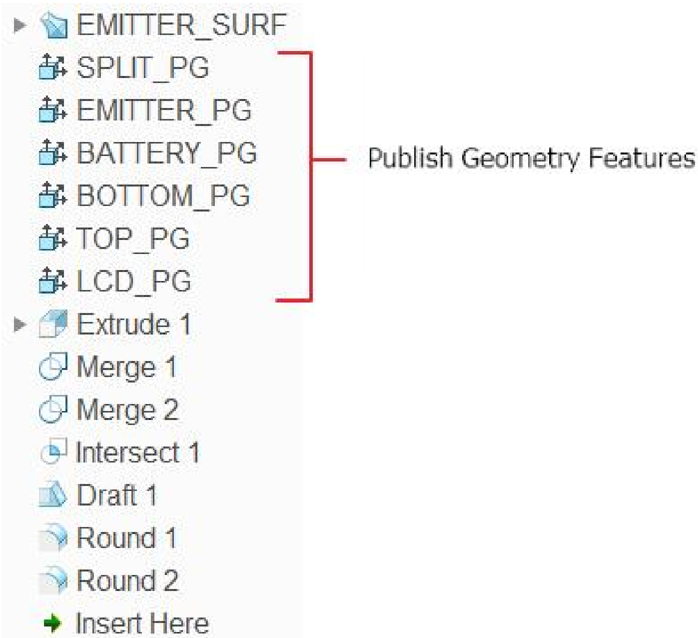
Unhide the SKL_SURF_EMITTER layer and model will appear as shown below.



Although emitter surface should be a constituent of the skin but we will not merge it. Because if we merge it with main quilt, it will create an undercut in individual parts after shelling. So it will be used in the individual parts to create cut features after shell operation.

Communicating the Design Information

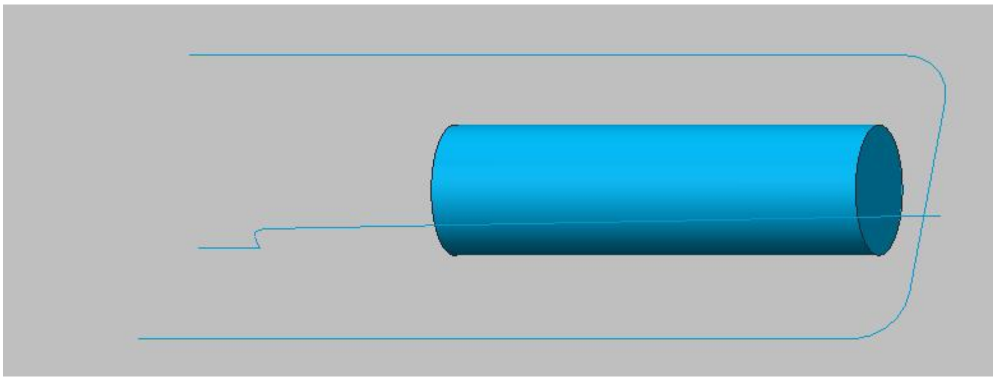
If you look in the model tree you will notice that publish geometry features are already defined as shown below.



The SPLIT_PG publish geometry feature contains the split surface and interlock curve as shown below. (You need to unhide the related layers to see the geometry)

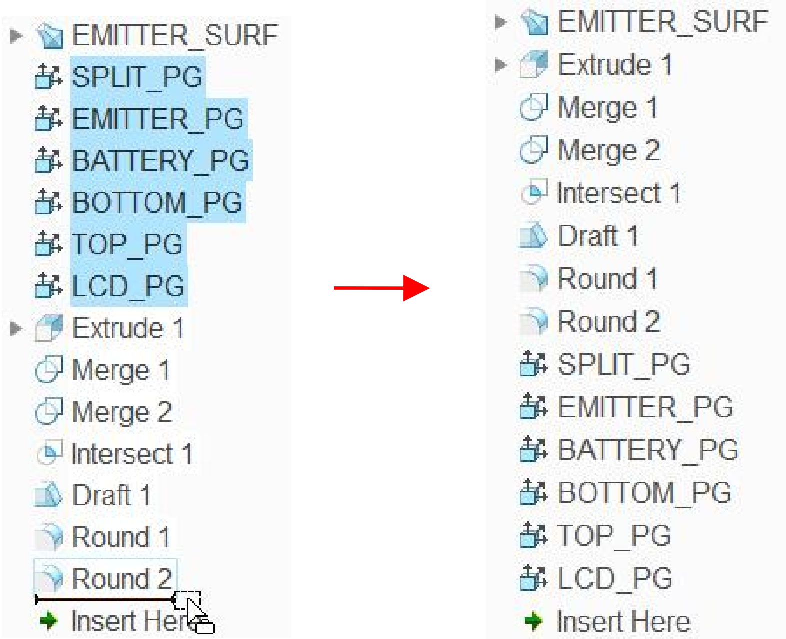
The EMITTER_PG publish geometry feature contains the emitter surface and emitter hole curve as shown below.

The BATTERY_PG feature contains the battery surface and profile curves as shown below.



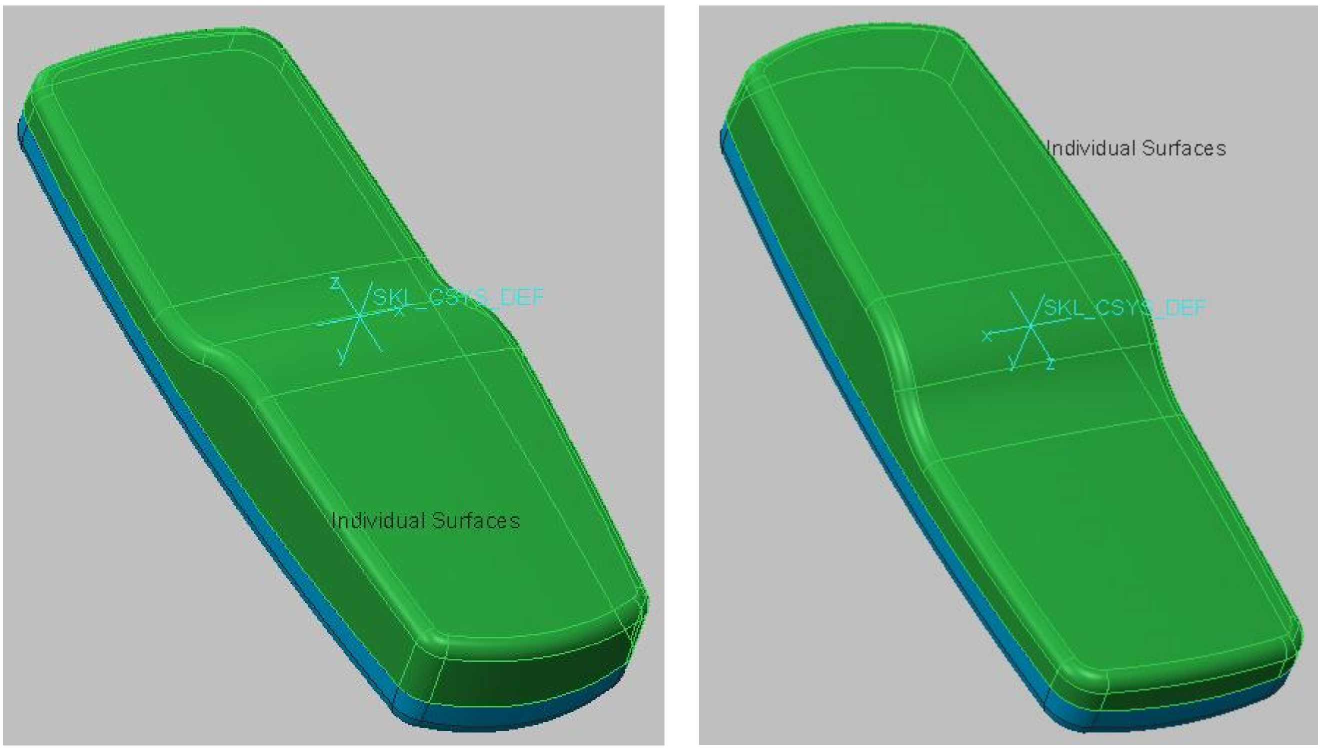
The BOTTOM_PG and TOP_PG publish geometry features are empty (they only contain dummy geometry). So now we will redefine them and copy the desired geometry. But first we need to move all the publish geometry features at the end of the model tree.

So select and drag all the PG features as shown below.



Select the BOTTOM_PG in the model tree and pick  .

Hold down the Ctrl key and select the following surfaces while Surfaces Sets collector is active. These surfaces will be used to create the BOTTOM part.

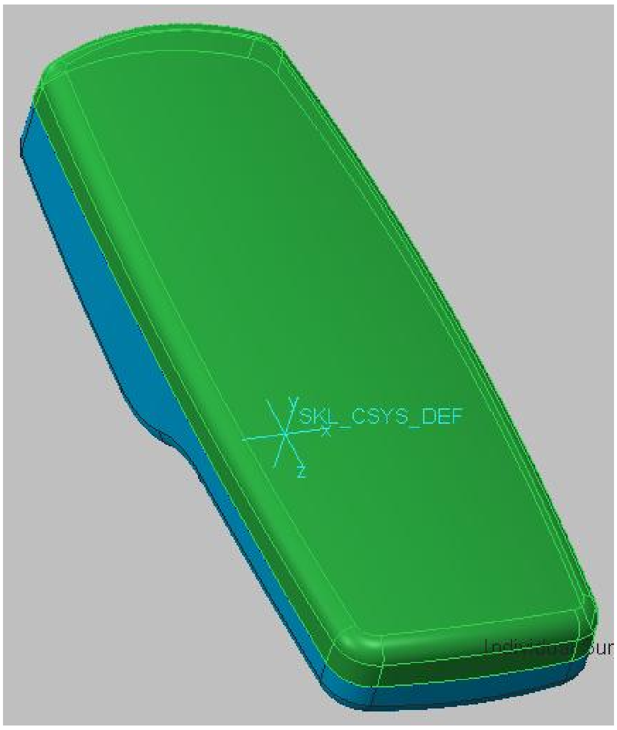


You can remove selections in References collector as they were just meant to create the feature.

Pick  to complete the feature.

Now select the TOP_PG publish geometry feature in the model tree and pick  .

Select the following surfaces while Surfaces Sets collector is active.



You can remove the entries in the Reference collector as they were just meant to create the feature.

Pick icon to complete the feature.

Copy Geometry features have already been defined in individual parts that refer the related Publish Geometry features in the skeleton, so we do not need to create any CGF.

Creating Model Geometry

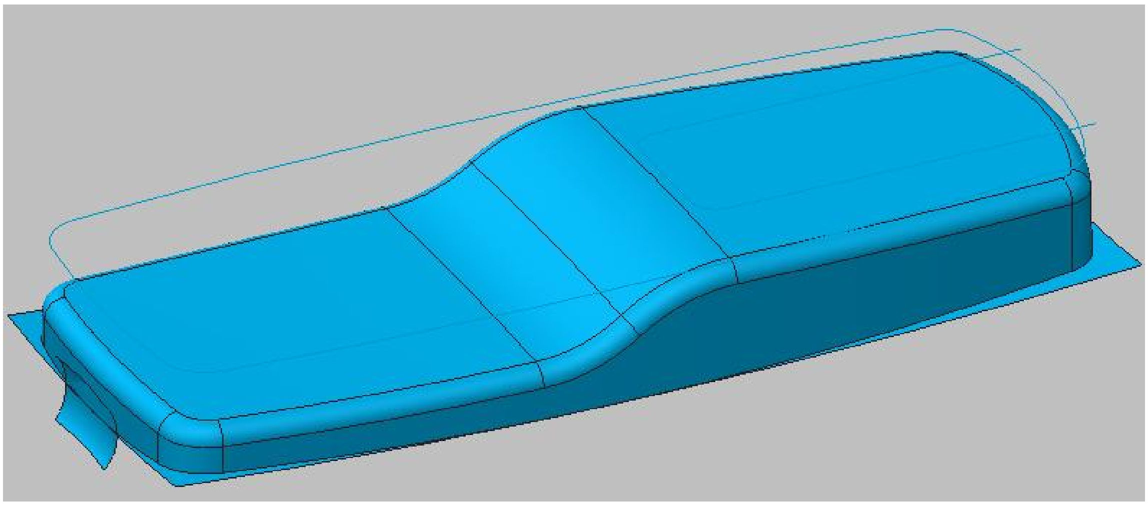
Now we will create the geometry in individual parts. In the exercise we will create geometry for only two parts. You can create the geometry for other parts on your own. You can see all the parts in completed shape in the FINALIZED folder.

First Part

Open the REMOTE_BOTTOM.PRT in a separate window. Notice that it already has four copy geometry features as shown below.

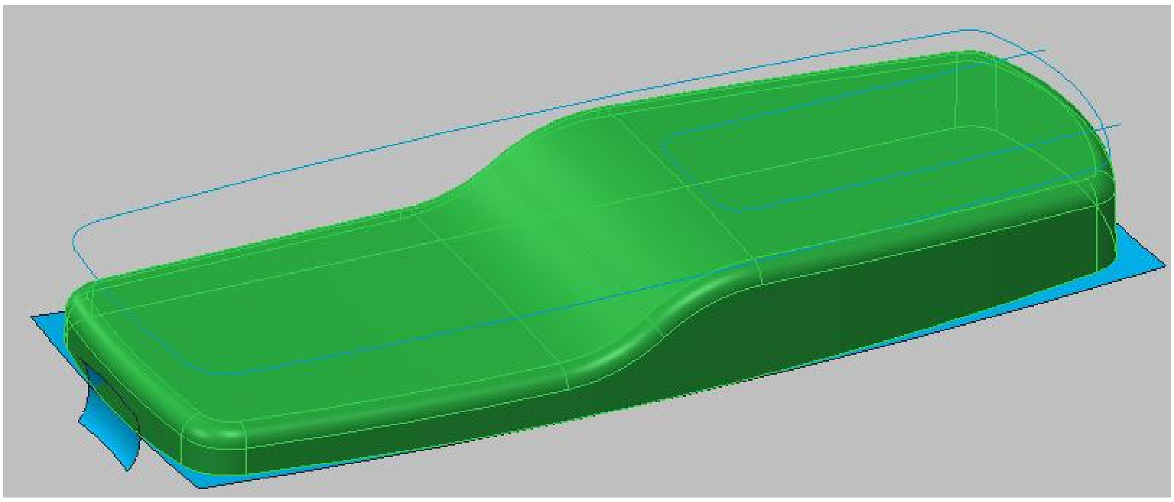
These copy geometry features refer the corresponding publish geometry feature (PGF) in the skeleton.

As we modified BOTTOM PGF in the skeleton so we need regeneration. So regenerate the model by picking icon and model will appear as shown below.



Now we will create surface copy features.

Select the following quilt. Change the filter setting to **Quilts** for ease of selection

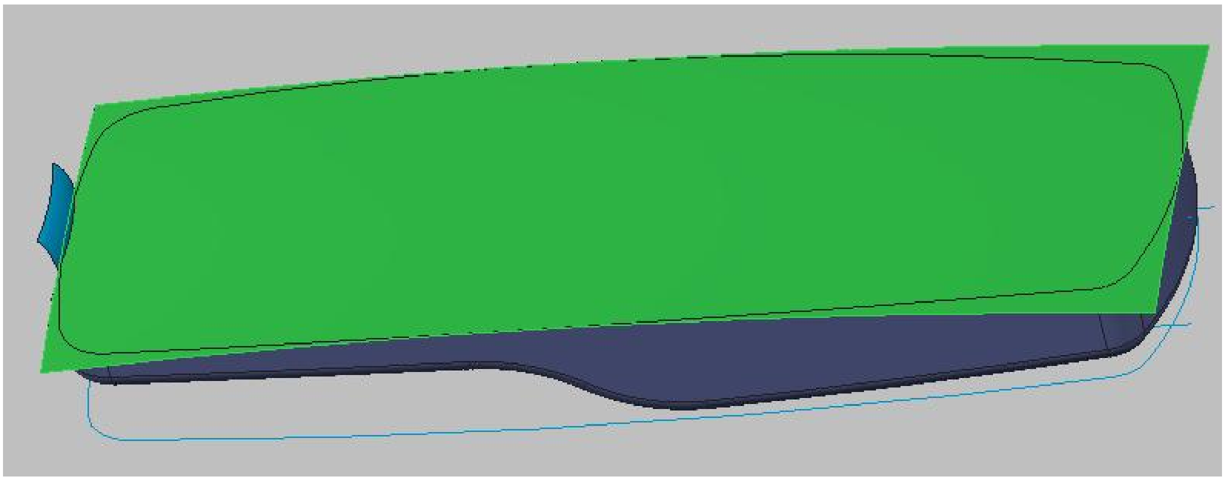


Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

The Copy dashboard will appear so pick to complete the feature.

Now select the following surface.



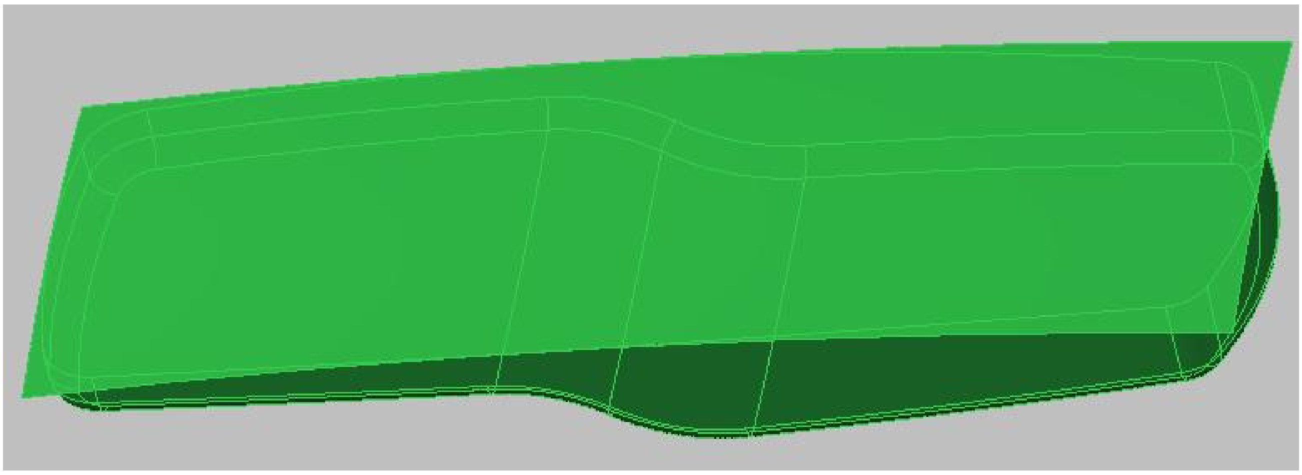
Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

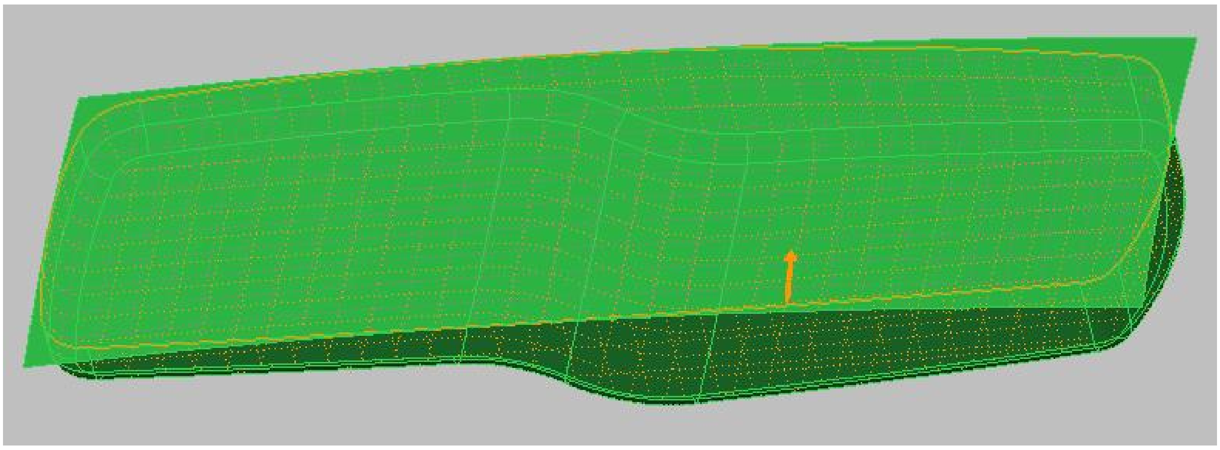
The Copy dashboard will appear so pick to complete the feature.

Now we will merge the copied surfaces together. Before proceeding hide the 07___PRT_COPY_GEOM layer.

Select the copied quilts while holding down the Ctrl key as shown below.

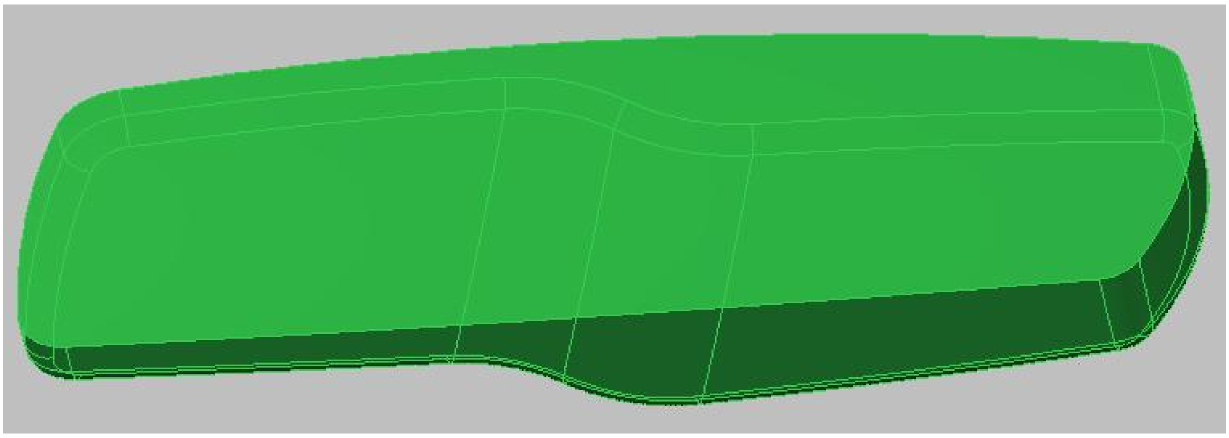


Pick Merge to invoke the Merge tool. Make sure that preview appears as shown below.



Pick  to complete the feature.

Now pick the merged quilt as shown below.



Pick 

to invoke Solidify tool.

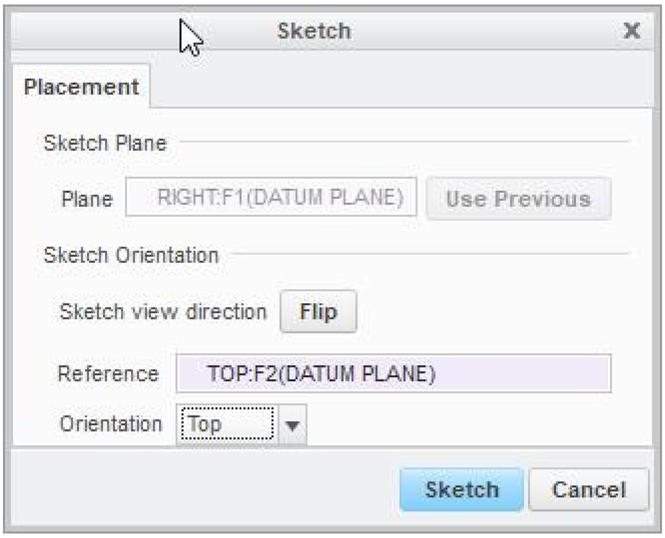
Pick  to complete the feature.

Now we will create two extruded surfaces that will be merged and then used to create a cut for the battery cover.

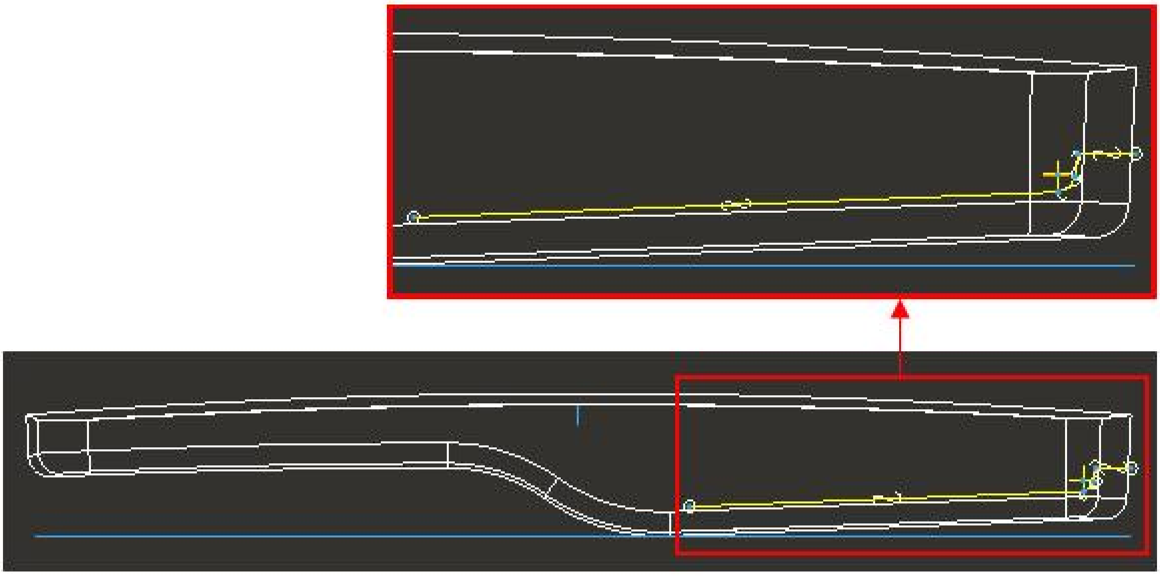
First unhide the 07___ PRT_COPY_GEOM and hide 06___ PRT_ALL_SURFS layer.

Pick  to invoke Extrude tool.

Select the sketching references as shown below



Pick Project icon and select the following loop.

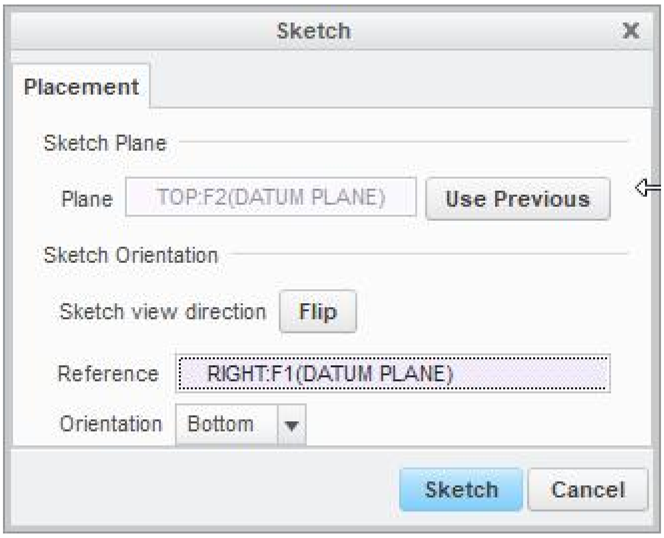


After completing the sketch, change the depth option to Symmetric ( ) and enter **30** as depth value.

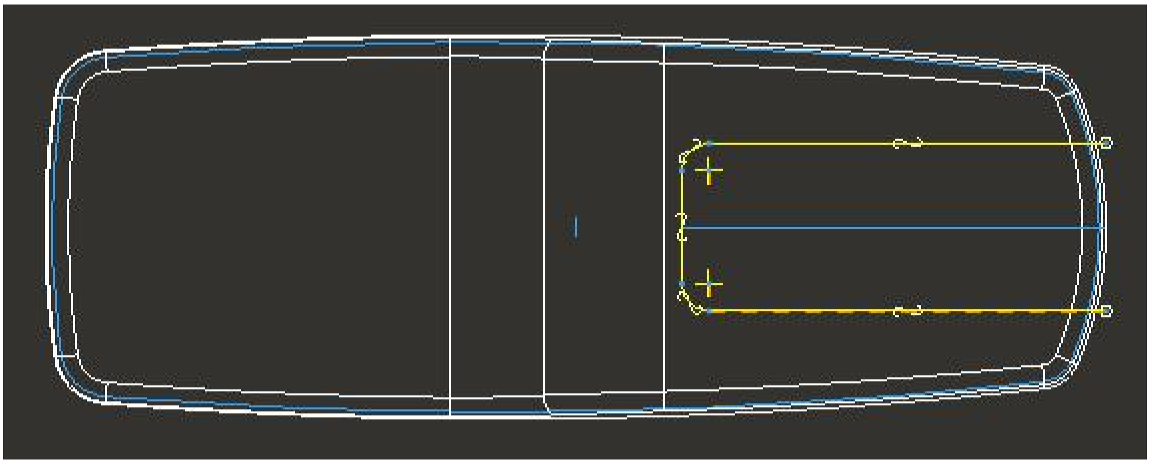
Pick to create the feature as surface.

Pick icon or middle-click to complete the feature.

Again pick to invoke Extrude tool and select the sketching references as shown below



Pick Project icon and select the following loop.



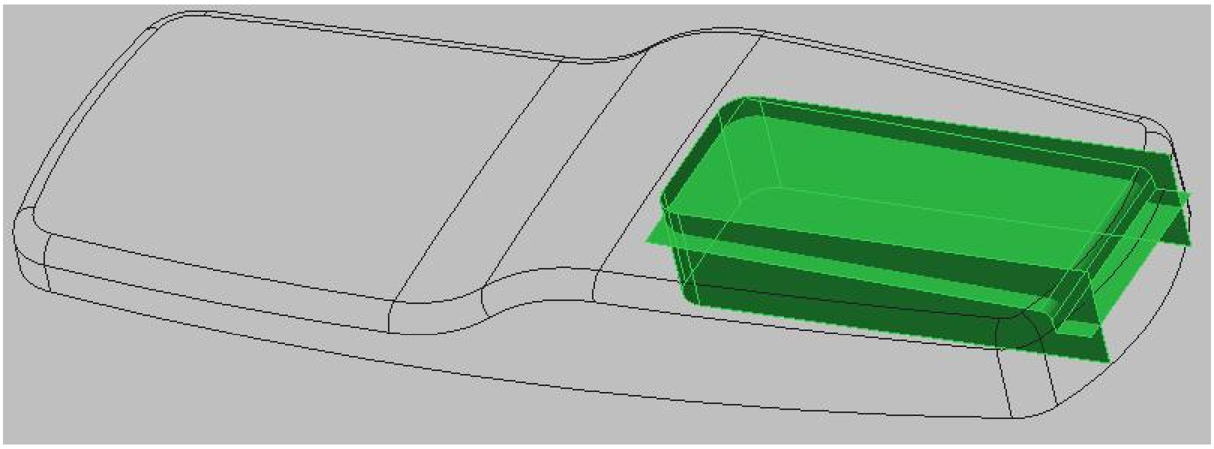
After completing the sketch enter **15** as depth value.

Pick to create the feature as surface.

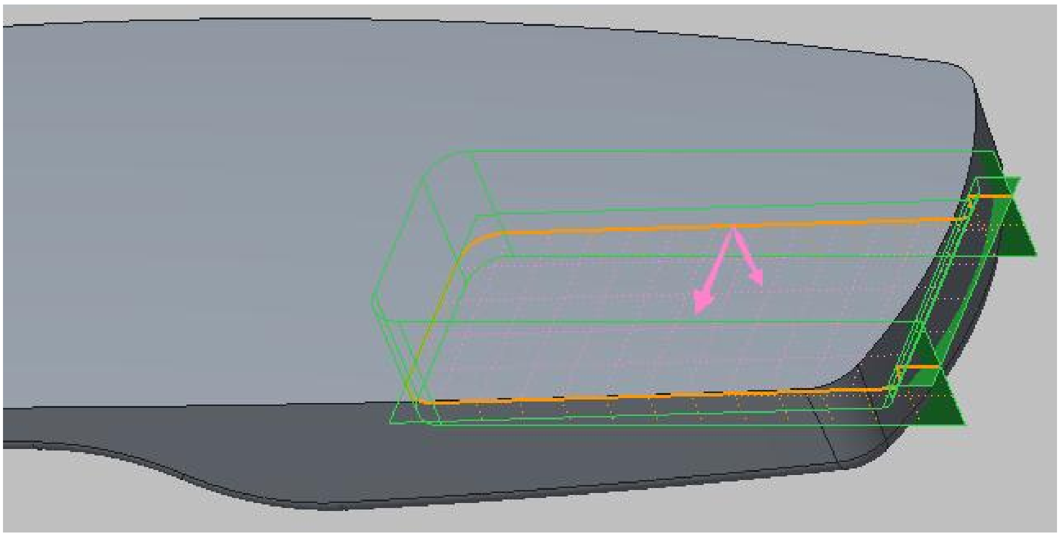
Pick icon or middle-click to complete the feature.

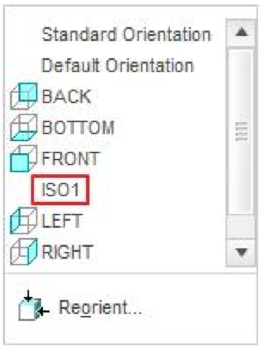
Now we will merge the extruded surfaces together. Before proceeding hide the 07___PRT_COPY_GEOM and unhide 06___ PRT_ALL_SURFS layer.

Select the extruded quilts while holding down the Ctrl key as shown in the figure below.



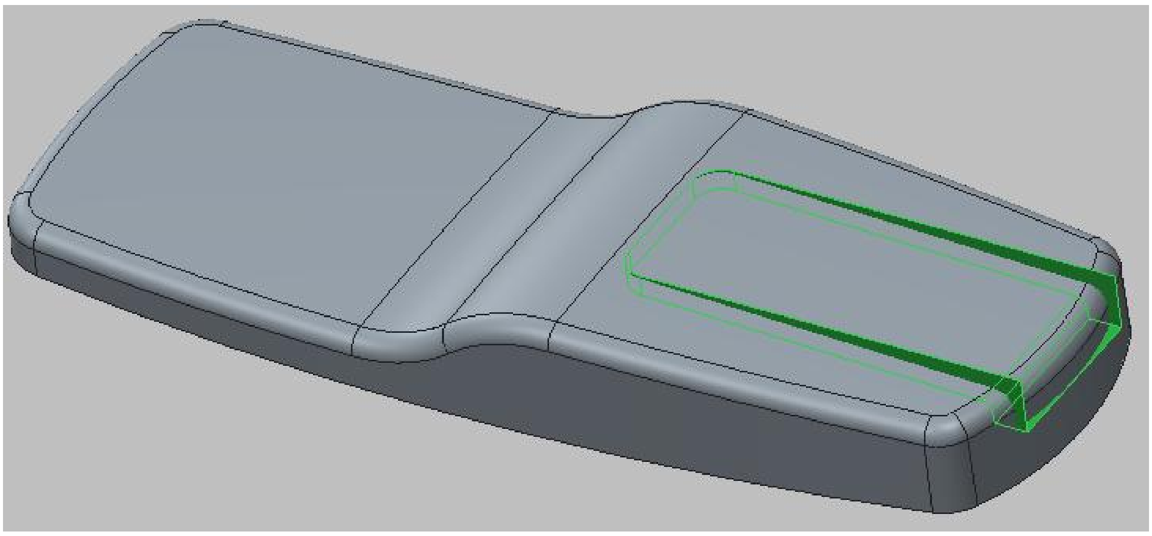
Pick Merge to invoke the Merge tool. Flip the direction of arrow so that preview appears as shown below. Set the view to **ISO1** to avoid any ambiguity.





Pick to complete the feature.

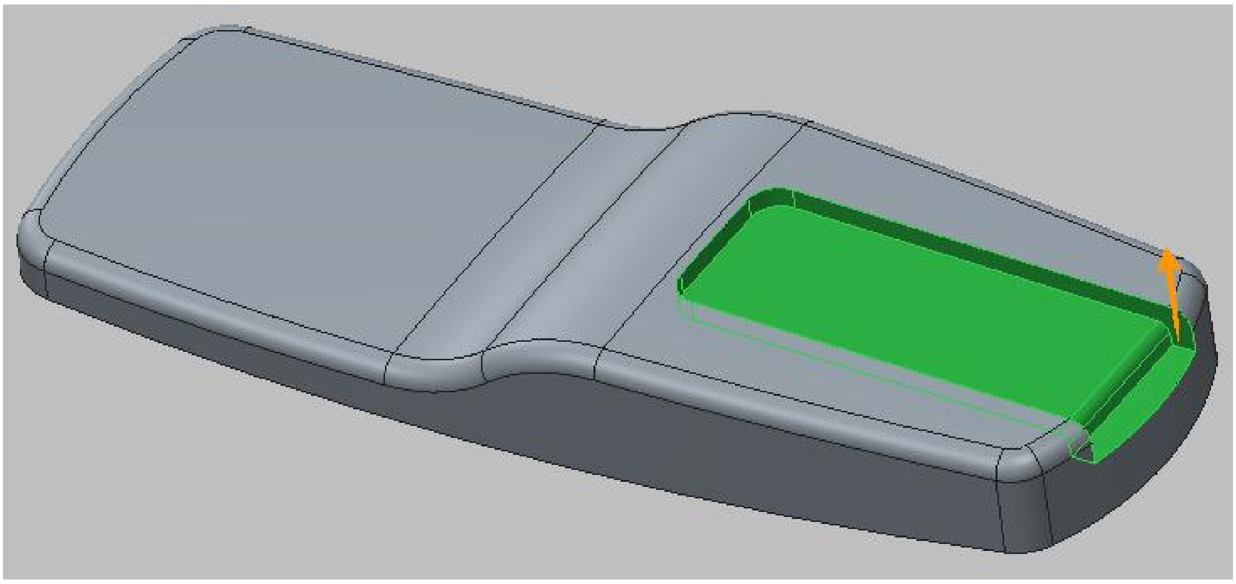
Now pick the merged quilt as shown below.



Pick Solidify to invoke Solidify tool.

In the Solidify dashboard pick icon to create the feature as cut.

Now the preview should appear as shown below.



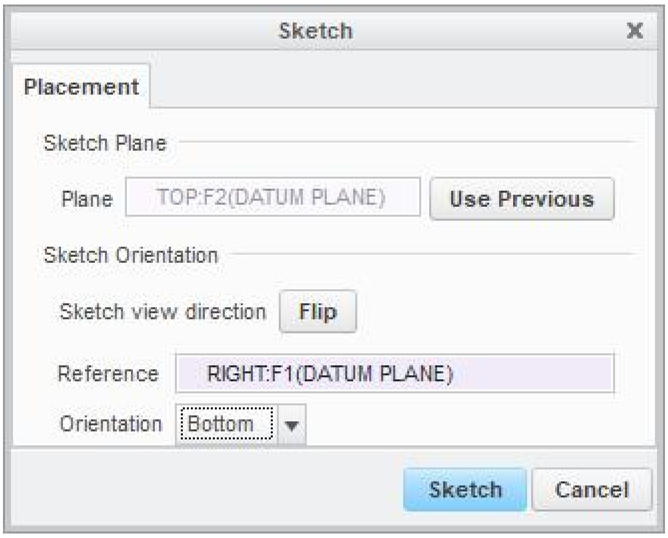
Pick to complete the feature.

Now we will create a cut to create space in the component for placing batteries.

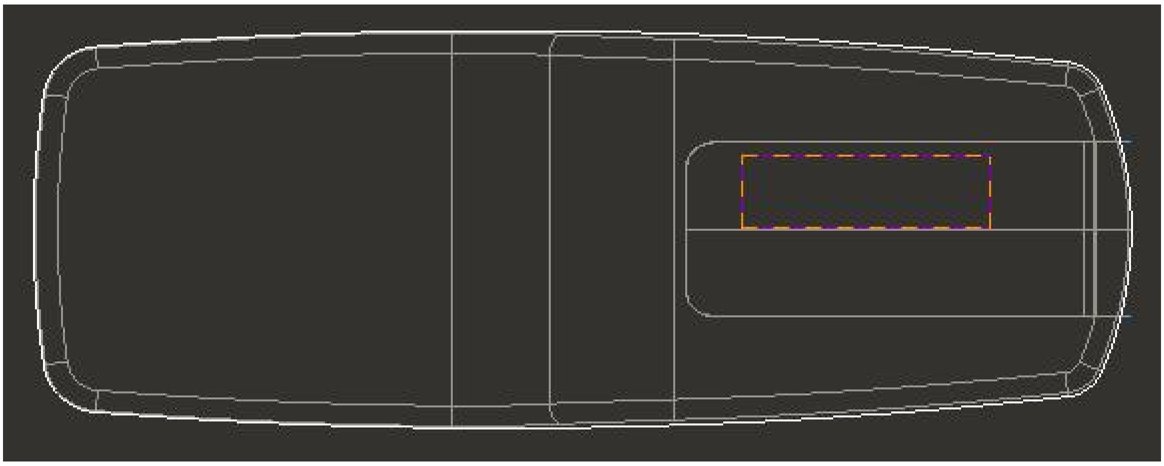
First unhide the 07___ PRT_COPY_GEOM and hide PRT_CG_E_BATT layer (this is a custom layer that contains all copy geometry features except BTTERY_CG).

Pick to invoke Extrude tool.

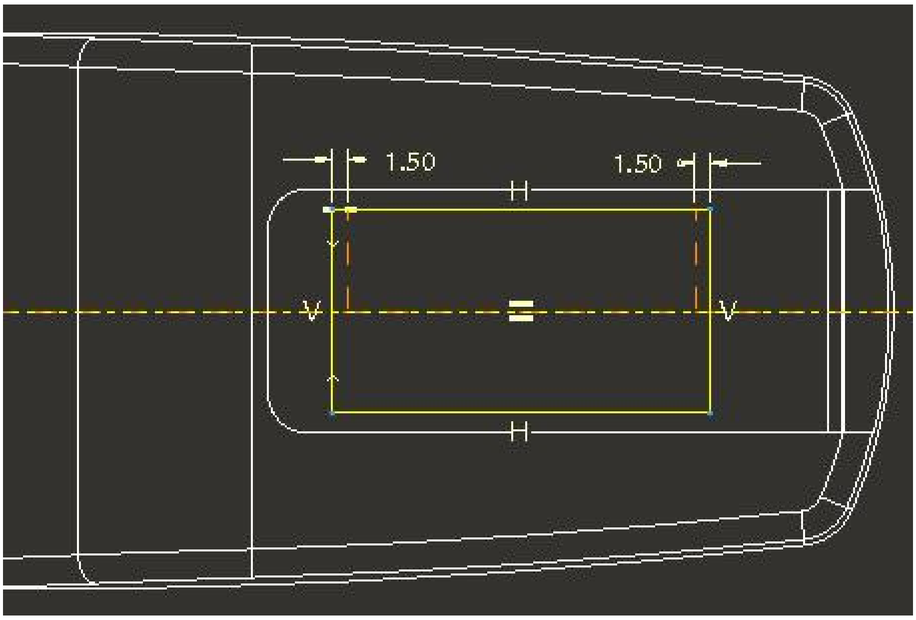
Select the sketching references as shown below



Select the edges of the battery surface as references as shown below.



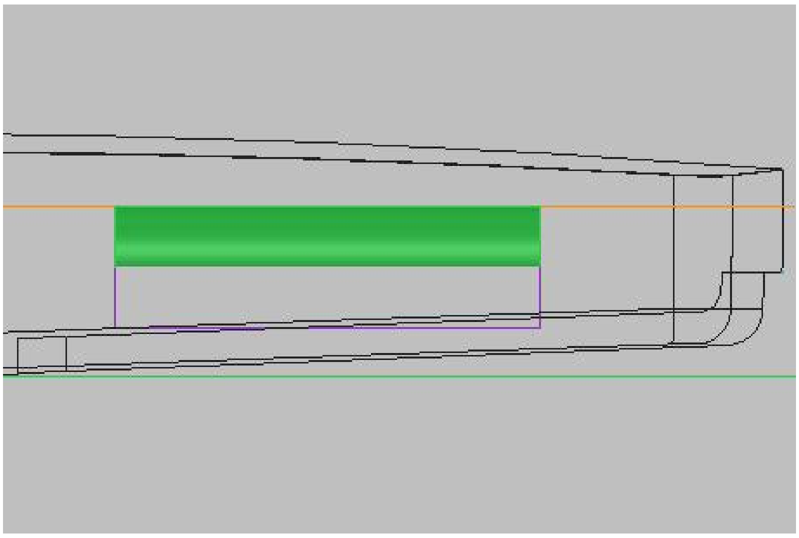
Sketch a centerline and rectangle as shown below

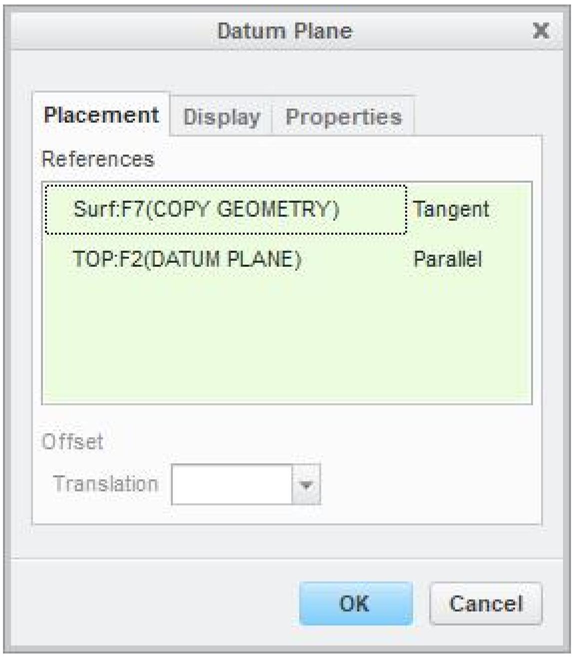


After completing the sketch pick icon to create the feature as cut.

After completing the sketch, change the depth option to **To Selected** ( ). We will create a datum plane on-the-fly to specify the depth.

Pick on the Model tab and select the references as shown below. (View is set to Right in the figure).

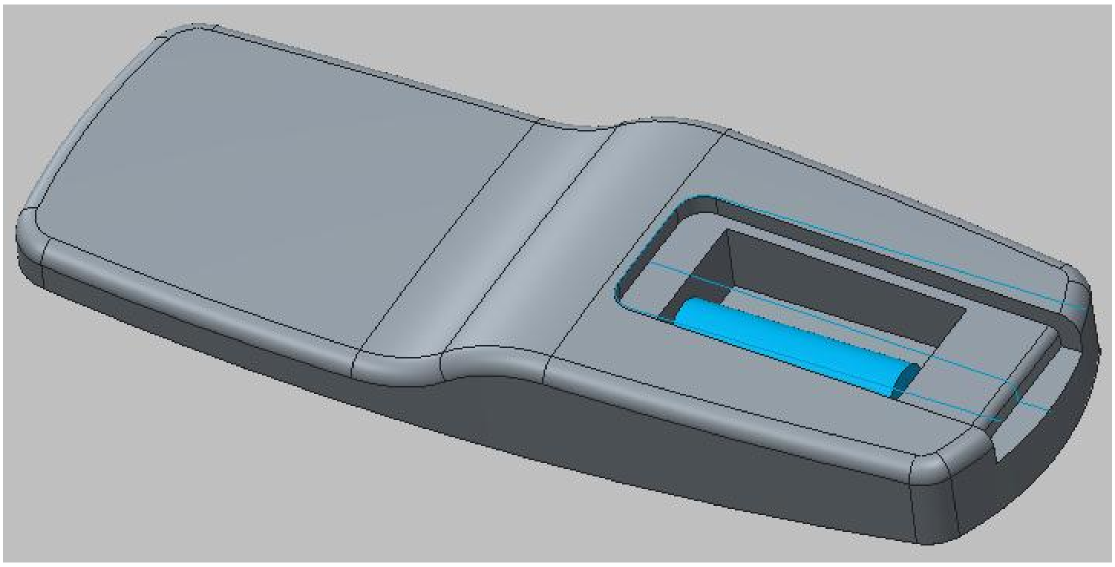




Notice that datum plane is parallel to TOP datum plane and tangent to the upper half of the battery surface.

Pick OK to apply and exit.

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

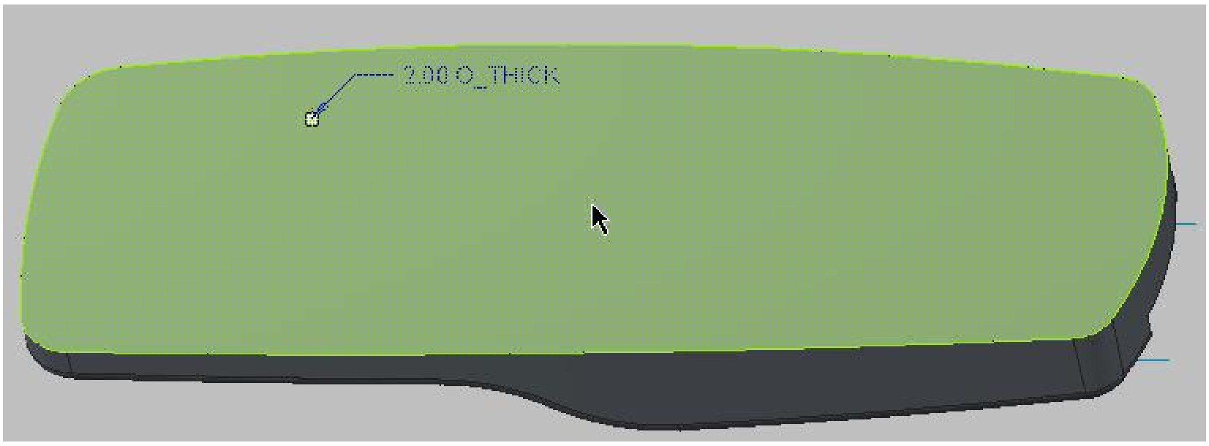


Now you should apply draft to the side walls of the two cut features just created. We will not show step-by-step procedure and is left for you. But if you want, you can skip the draft and follow the remaining exercise. It is not necessary for this exercise but from plastic product design point of view, draft should be applied.

Pick Shell to access the Shell tool.

Enter ***2*** as the thickness value

Select the following highlighted surface to add it to Removed surfaces collector.

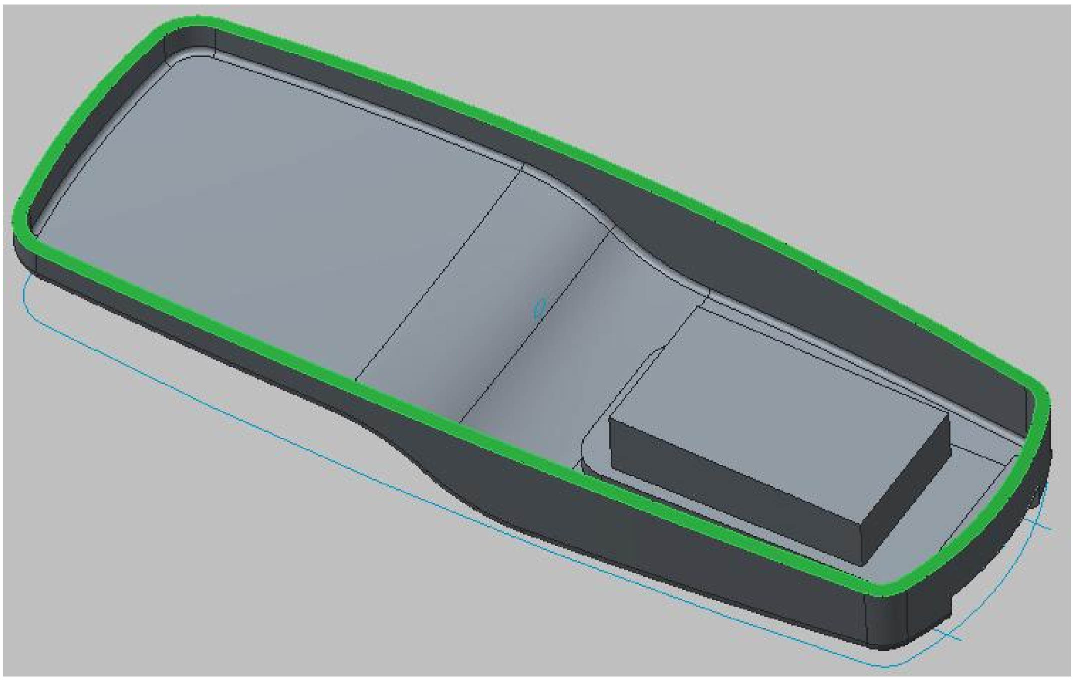


Pick  icon to complete the feature.

Now we will define an Offset feature to create the interlock lip.

First unhide the 07___ PRT_COPY_GEOM, PRT_CG_E_BATT layers and then hide 06___ PRT_ALL_SURFS layer.

Select the following surface of the part.



Pick 

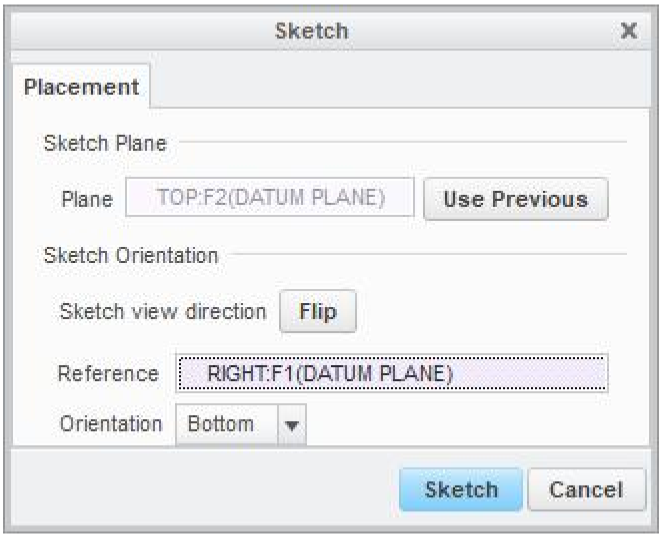
to access the Offset tool.

Change the Offset type to **With Draft** ().

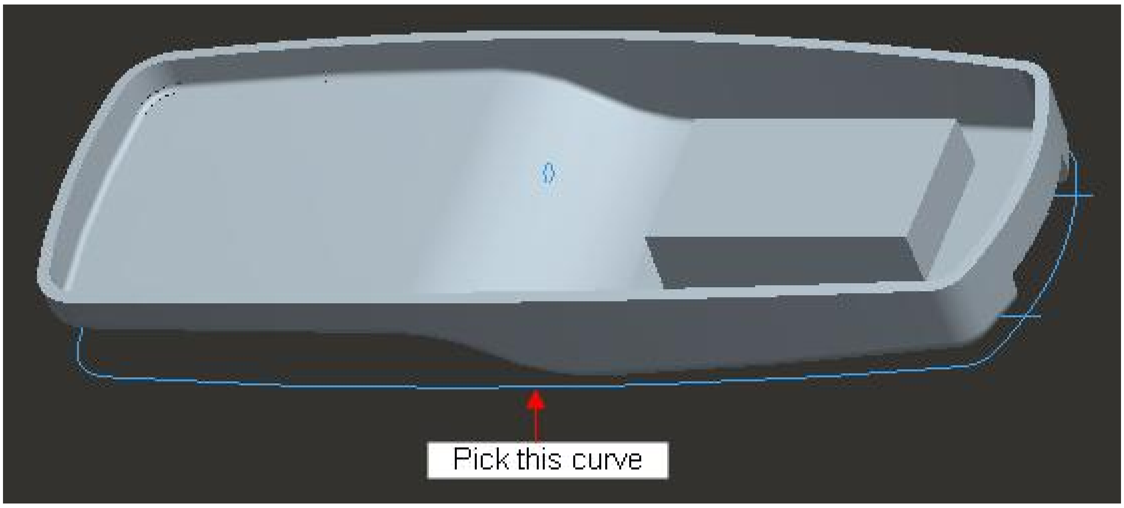
Pick 

tab in the References slide-up panel.

Select the sketching references as shown below.



Pick  icon and select the loop as shown below.



Specify ***0.1*** as the offset value.

The purpose to this offset is to have clearance between lip geometry of both parts.

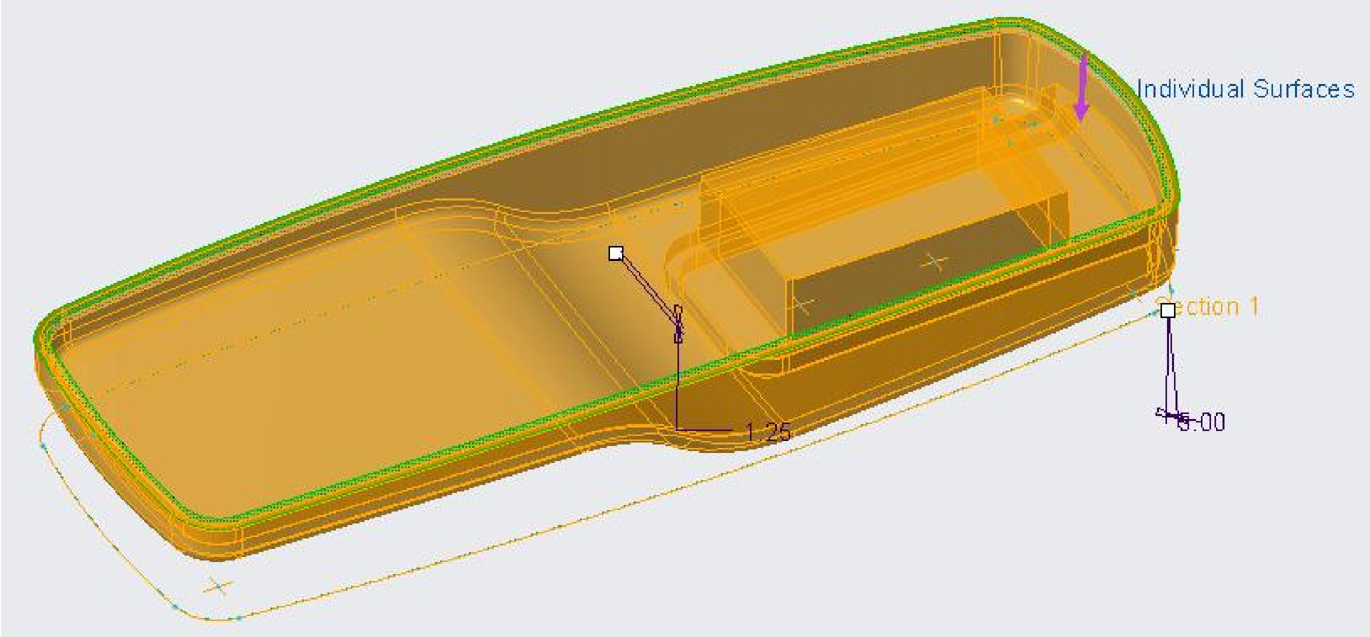
After completing the sketch enter ***1.25*** as the offset value for feature and hit Enter key.

Enter ***5*** as the draft angle and hit Enter key.

Pick  tab and change the “Side surface normal to” option to **Sketch**.

Pick  to flip the direction of feature creation.

The preview will appear as shown below.



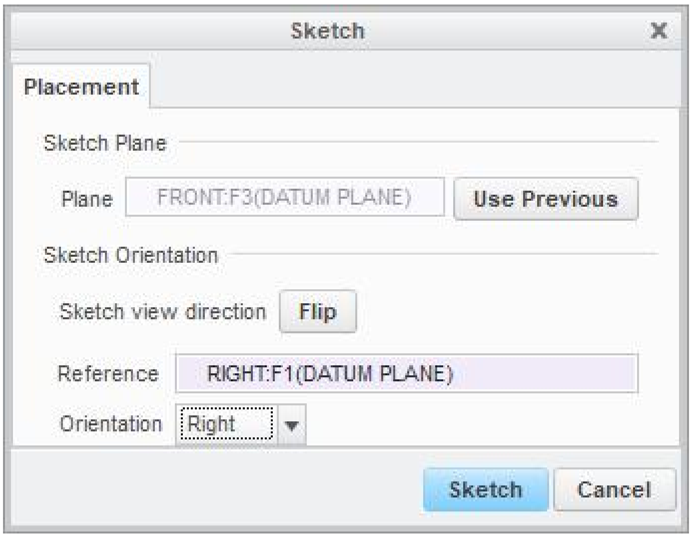
Make sure that Offset Direction arrow points in the direction shown. It will make sure that material is removed from the part.

Pick  icon to complete the feature.

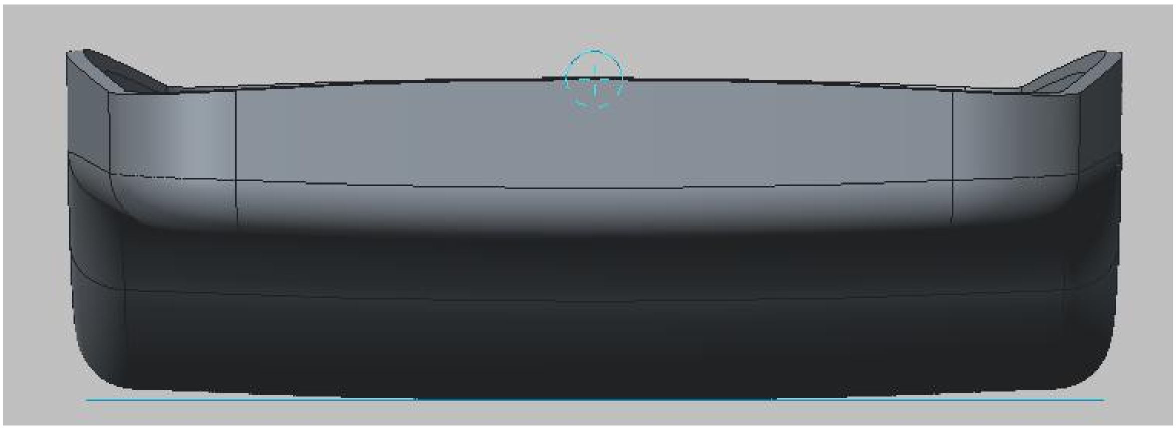
Now we will create the hole for emitter LED.

Pick  to invoke Extrude tool.

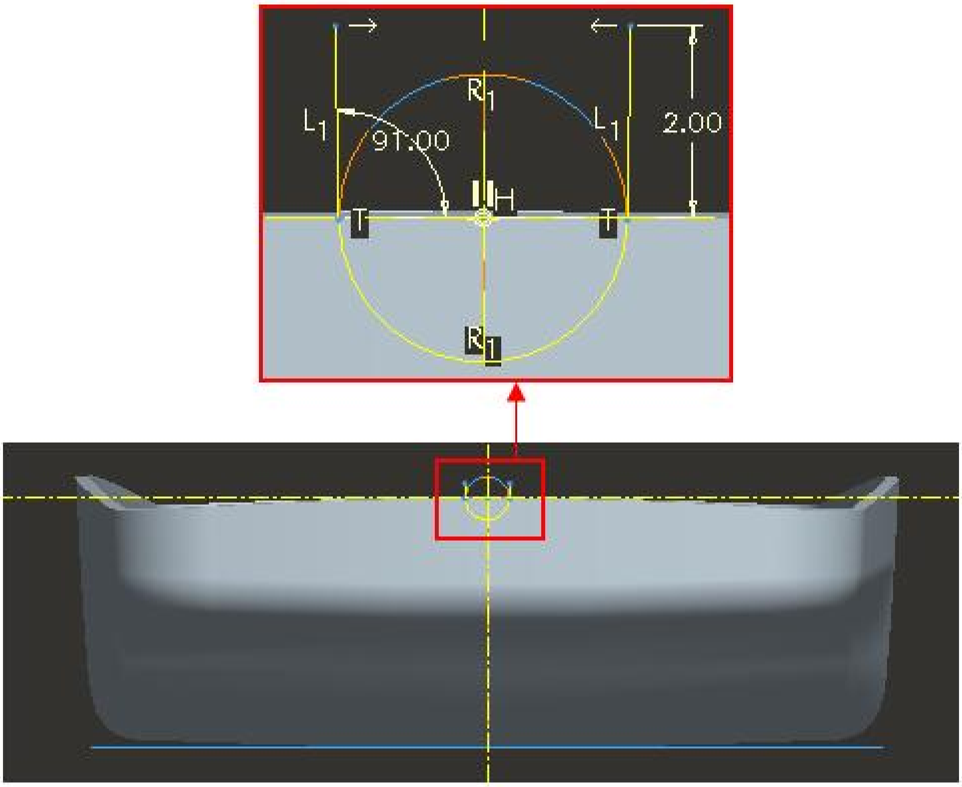
Select the sketching references as shown below



Pick the following curve as reference.



Create two centerlines passing through the center of the selected reference and then sketch as shown below.



After completing the sketch pick icon to create the feature as cut.

Specify the depth option to **Through All.**

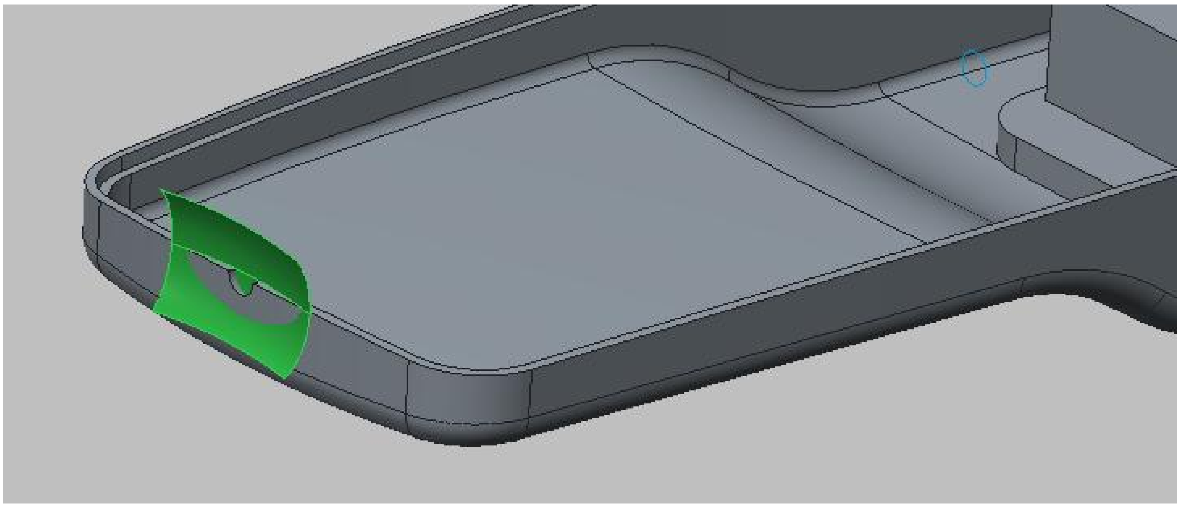
Pick icon or middle-click to complete the feature.

Now we will create the elliptical cut at the front of the part.

First unhide 06___PRT_ALL_SURFS and hide the PRT_CG_E_EMITTER layer.

Now we will create surface copy features.

Select the following highlighted surface.



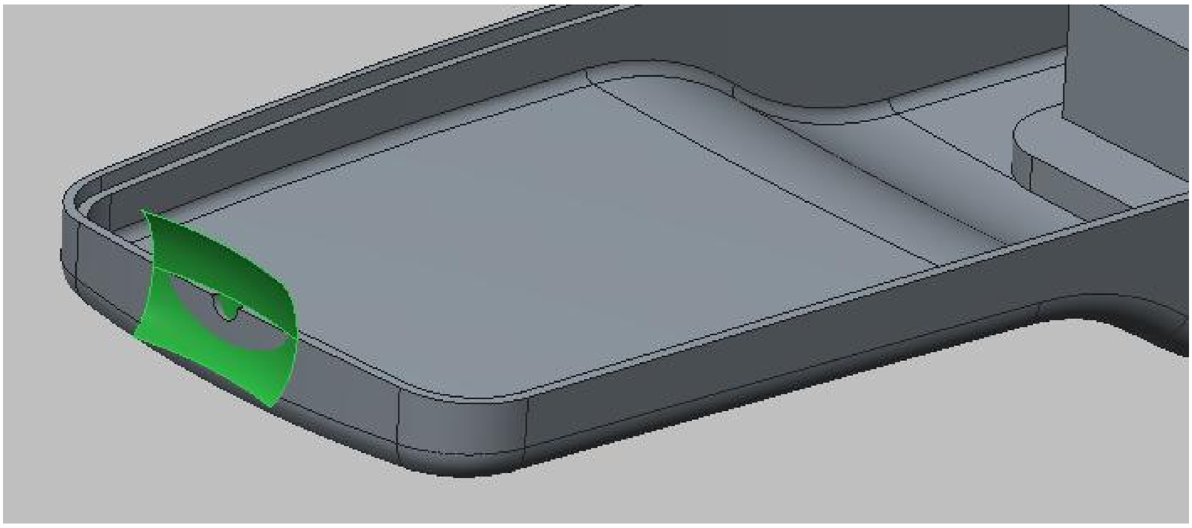
Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

The Copy dashboard will appear so pick to complete the feature.

Hide PRT_CG_E_BATT layer to unclutter the view.

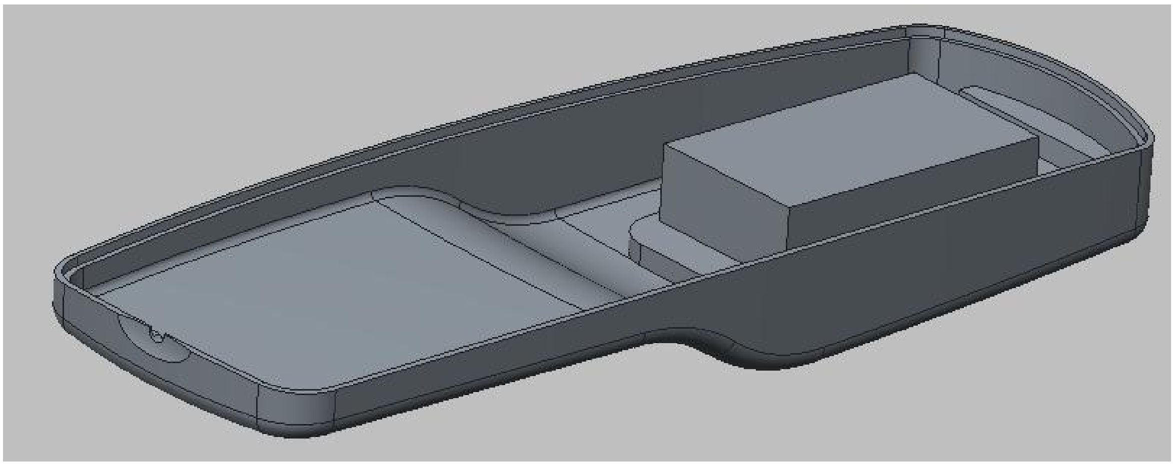
Now select the copied surface as shown below.



Pick Solidify to invoke Solidify tool.

In the Solidify dashboard pick icon to create the feature as cut.

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



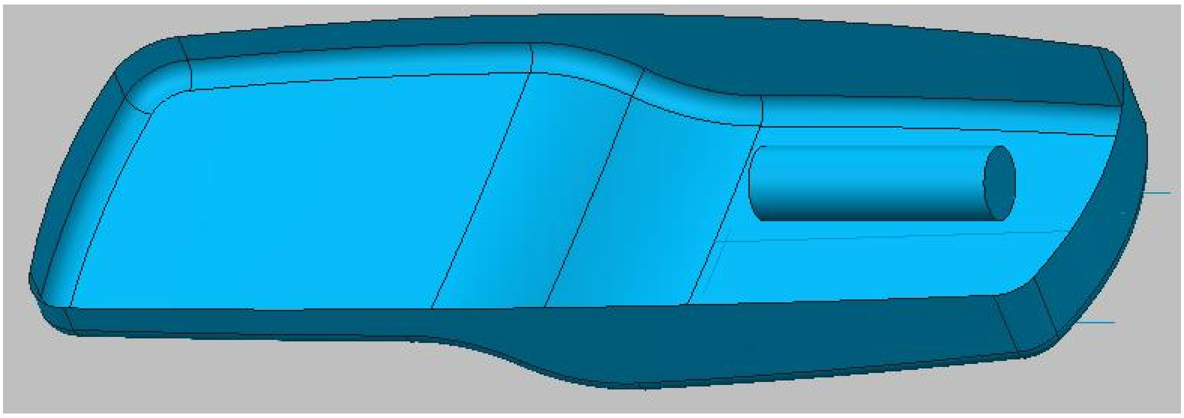
Second Part

Now we will create the geometry for battery cover. So open the REMOTE_BAT_COVER.PRT in the new window.

Notice that it has already two copy geometry features as shown below.

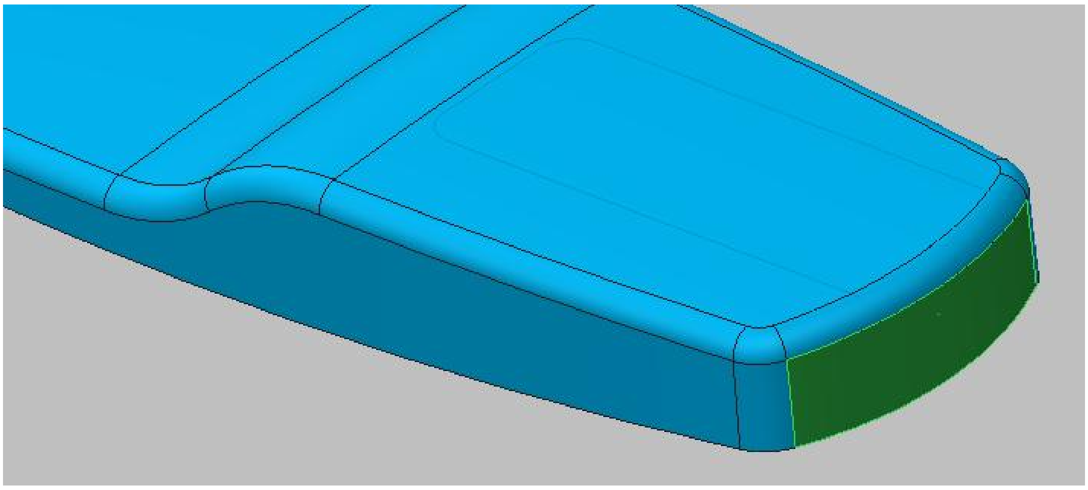
These copy geometry features refer the corresponding publish geometry features in the skeleton.

As we modified BOTTOM_PG feature in the skeleton so we need regeneration. So regenerate the model by picking icon. Now the model will appear as shown below.



Now we will create a surface copy feature.

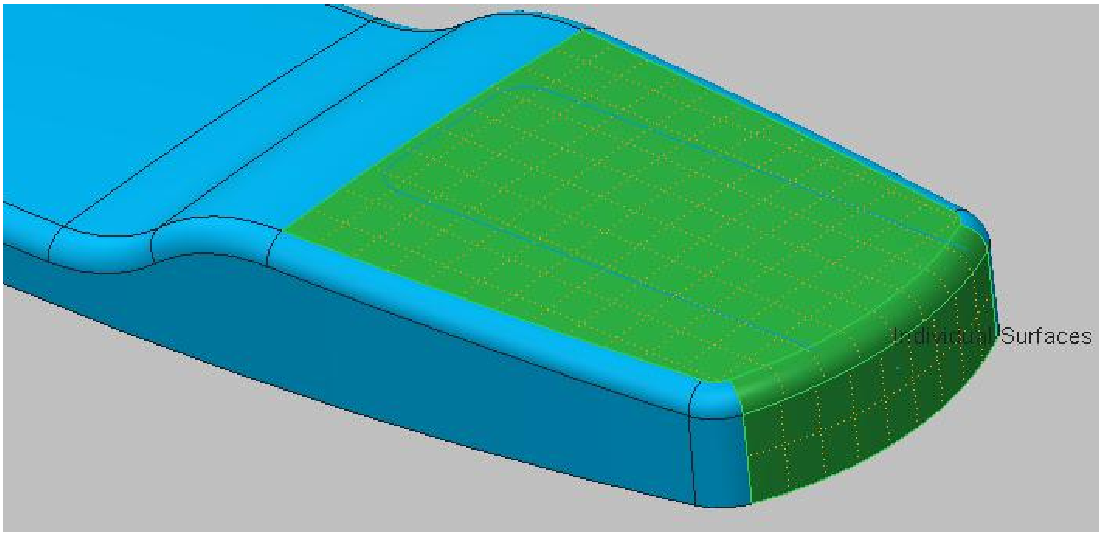
Select the surface highlighted in the figure below. (Make sure that filter is set to Smart)



Pick copy icon () on the Model tab or press Ctrl+C on the keyboard.

Select paste icon () on the Model tab or press Ctrl+V on the keyboard.

Add two more surface patches to the selection shown in figure below while holding down Ctrl key.



Pick  to complete the feature.

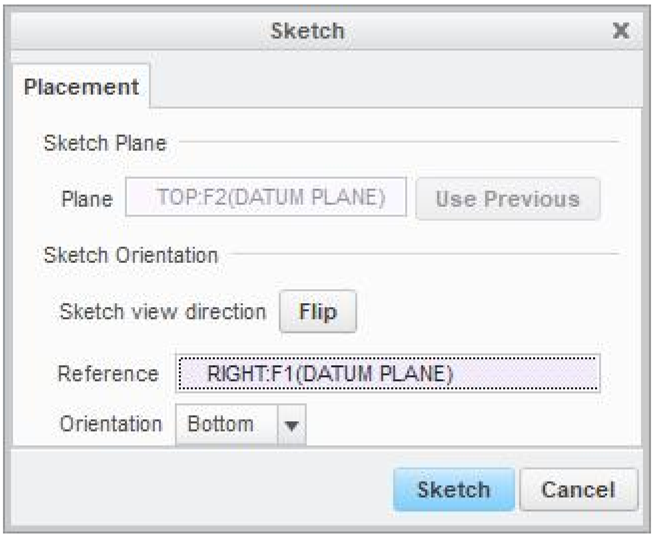
Now we will create two extruded surfaces and then merge them with the copied surface.

First hide the PRT_CG_BOTTOM layer.

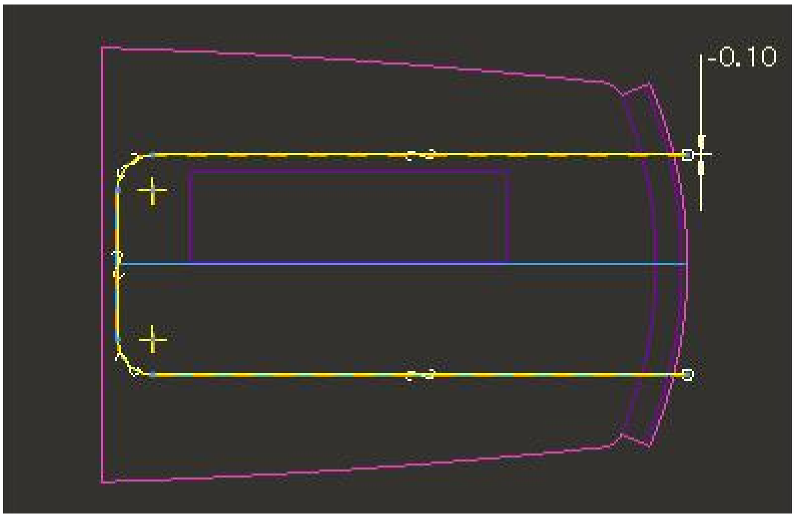
Pick  to invoke Extrude tool.

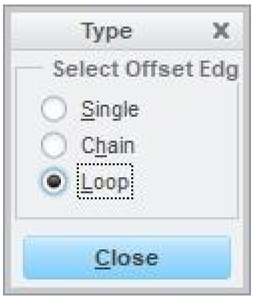
Pick  to create the feature as surface.

Select the sketching references as shown below.

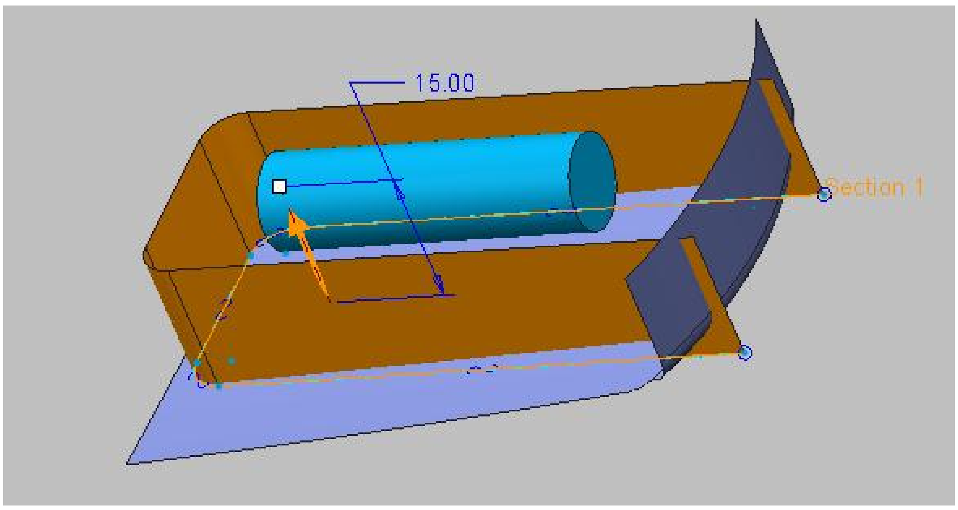


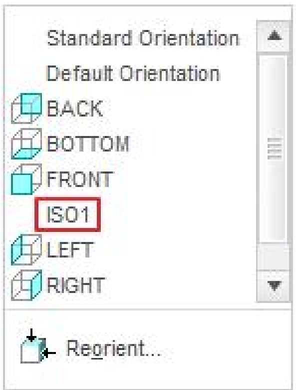
Pick Offset icon and create a loop that is offset by ***-0.1*** (i.e. towards inside) from the profile curve as shown below





After completing the sketch enter **15** as depth value.



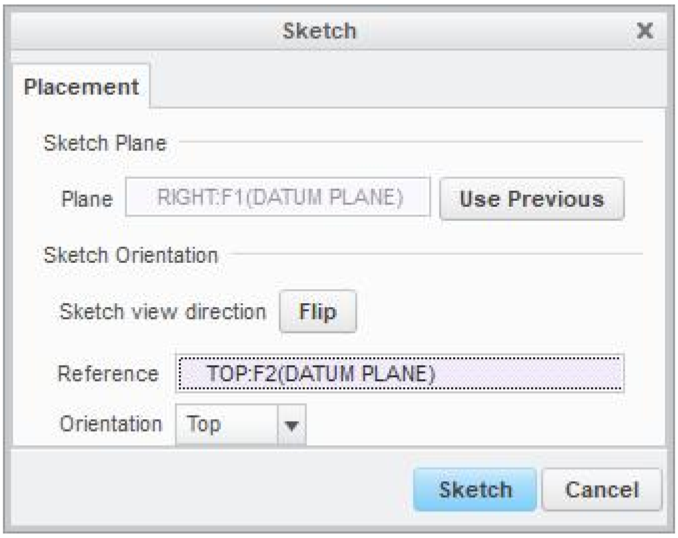


Pick icon or middle-click to complete the feature.

Again pick to invoke Extrude tool

Pick to create the feature as surface.

Select the sketching references as shown below.



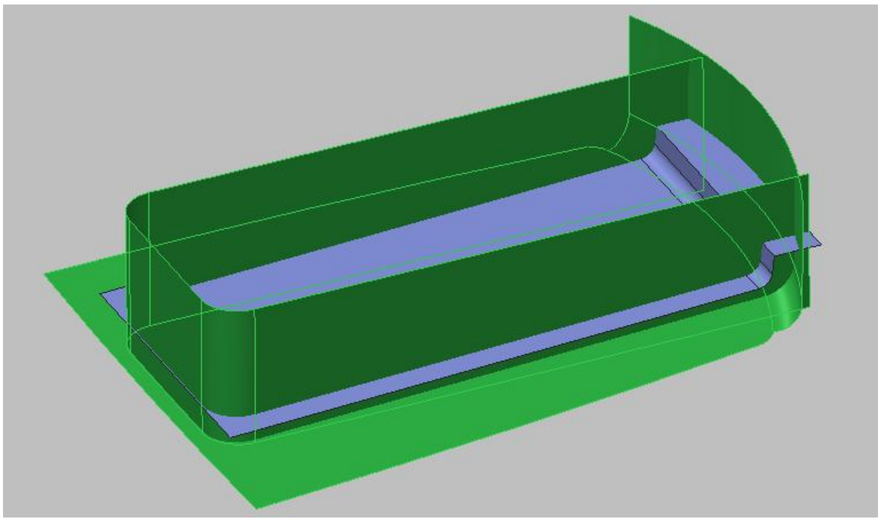
Pick Project icon and select the following loop.

After completing the sketch, change the depth option to Symmetric ( ) and enter **30** as depth value.

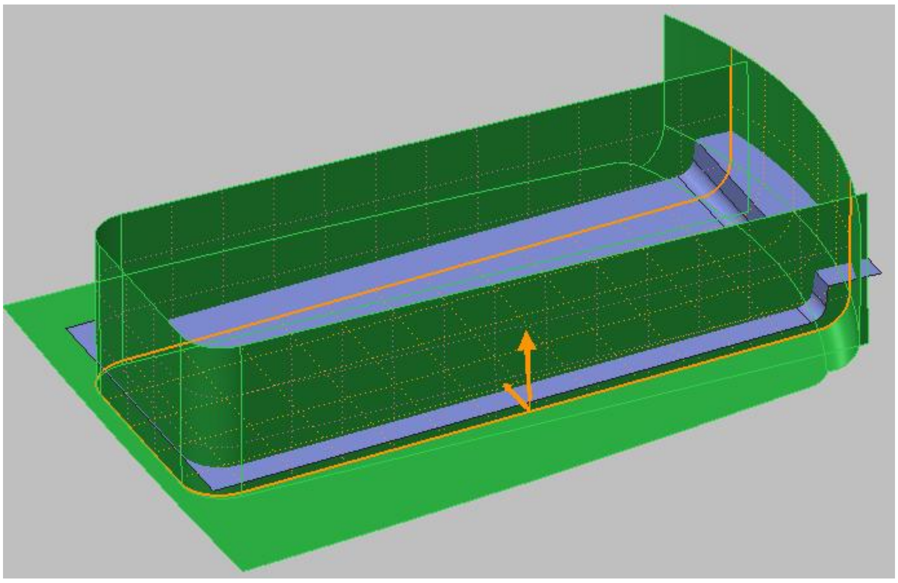
Pick icon or middle-click to complete the feature.

Now we will merge the copied and extruded surfaces together. Before proceeding hide the 07___PRT_COPY_GEOM layer.

Select the copied quilt and extruded surface while holding down the Ctrl key as shown in figure below.

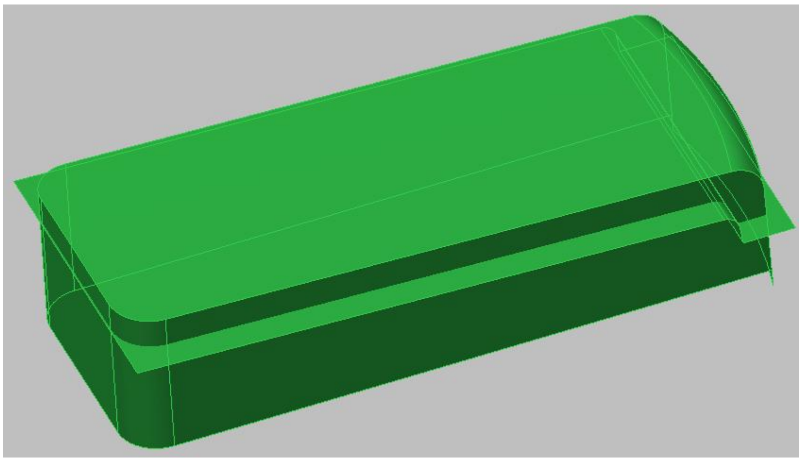


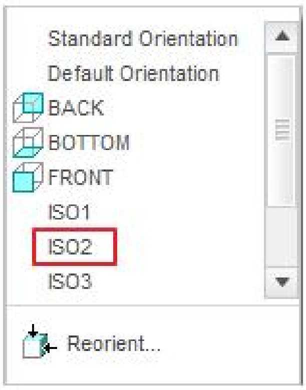
Pick  to invoke the Merge tool. Make sure that preview appears as shown below.



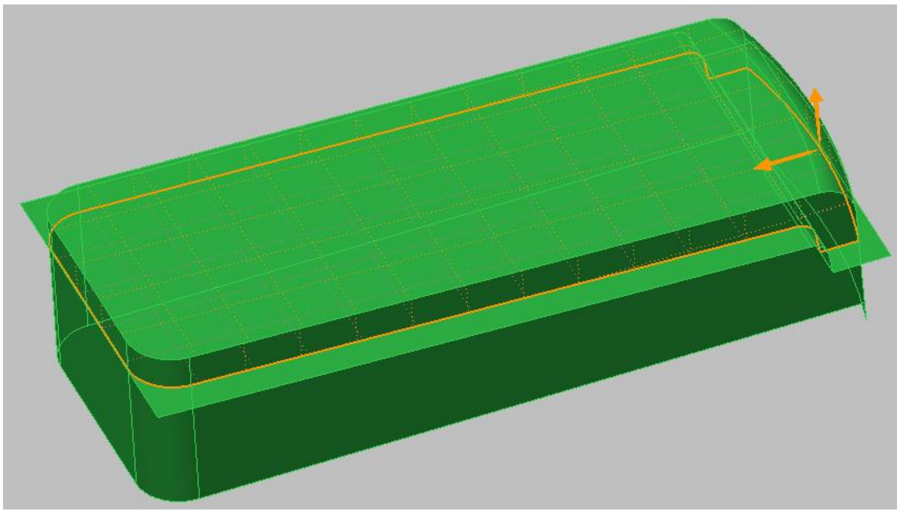
Pick  to complete the feature.

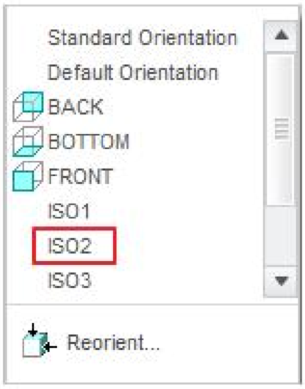
Now select the merged quilt and extruded surface while holding down the Ctrl key as shown below.





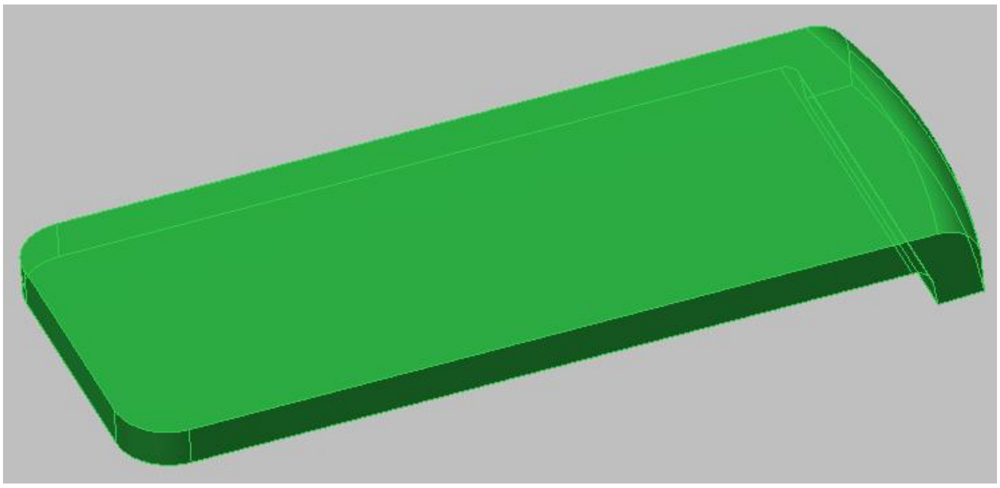
Pick  to invoke the Merge tool. Flip the arrow to make sure that preview appears as shown below.





Pick  to complete the feature.

Now pick the merged quilt as shown below.

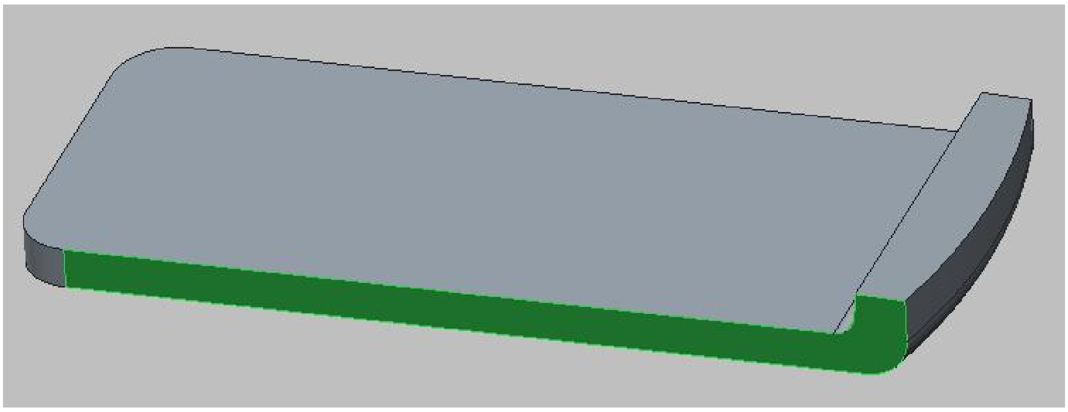


Pick  to invoke Solidify tool.

Pick  to complete the feature.

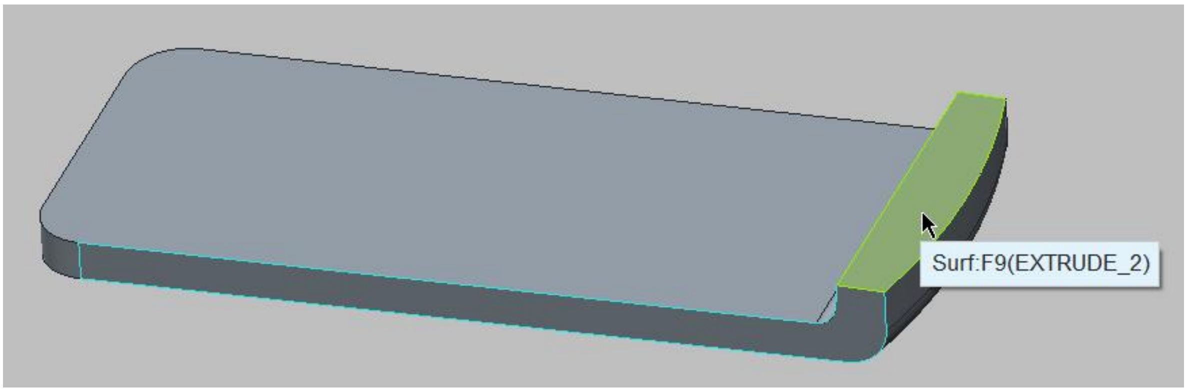
Now we will apply draft and shell the part.

Select the following surface.



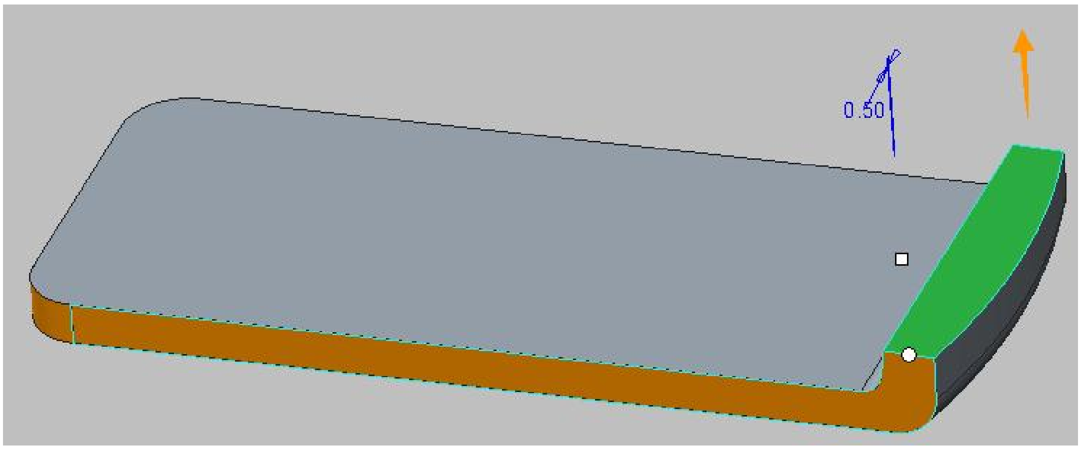
Pick Draft to access the Draft tool.

Notice that Draft hinges collector is active so pick the surface highlighted in the figure below, as the draft hinge.



Enter ***0.5*** in the Draft angle field on the dashboard.

The preview of the drafted surface will appear as shown below.

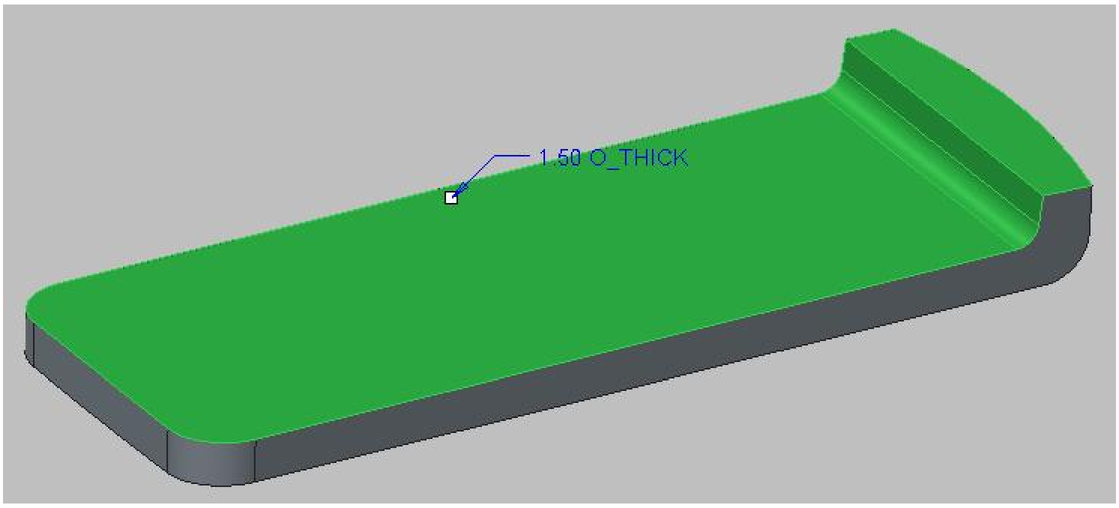


Pick icon to complete the feature.

Pick Shell to access the Shell tool.

Enter ***1.5*** as the thickness value

Select the following highlighted surfaces to add them to Removed surfaces collector. (Preview is off in the figure.)



Pick  icon to complete the feature.

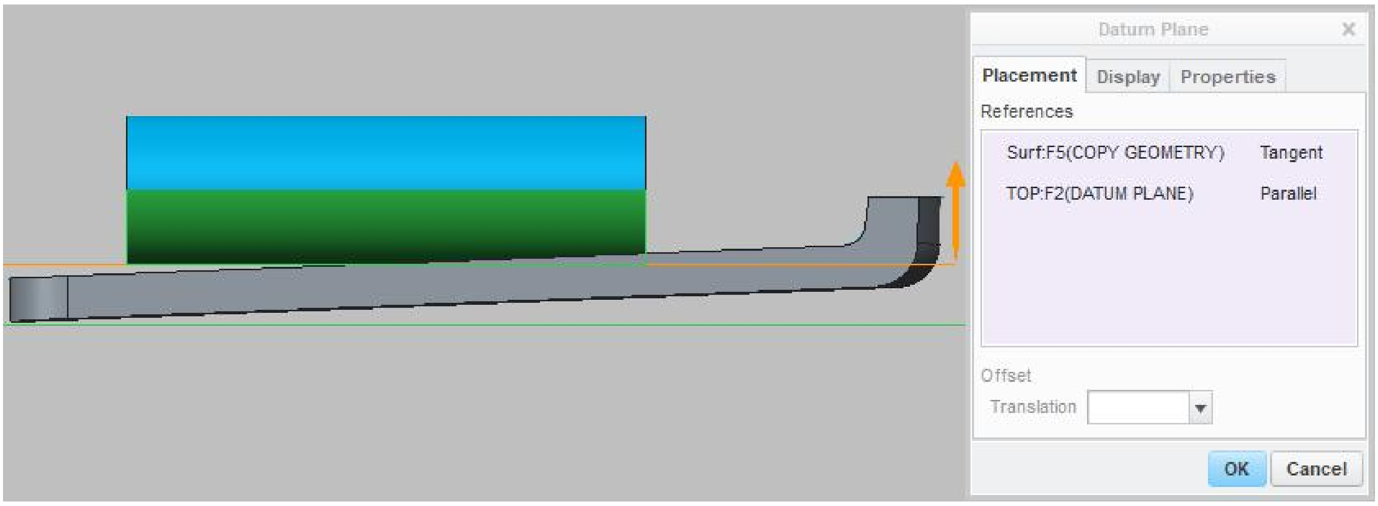
Now we will create a wall that will hold down the batteries in their position.

First unhide the 07___PRT_COPY_GEOM layer.

Pick  to invoke Extrude tool.

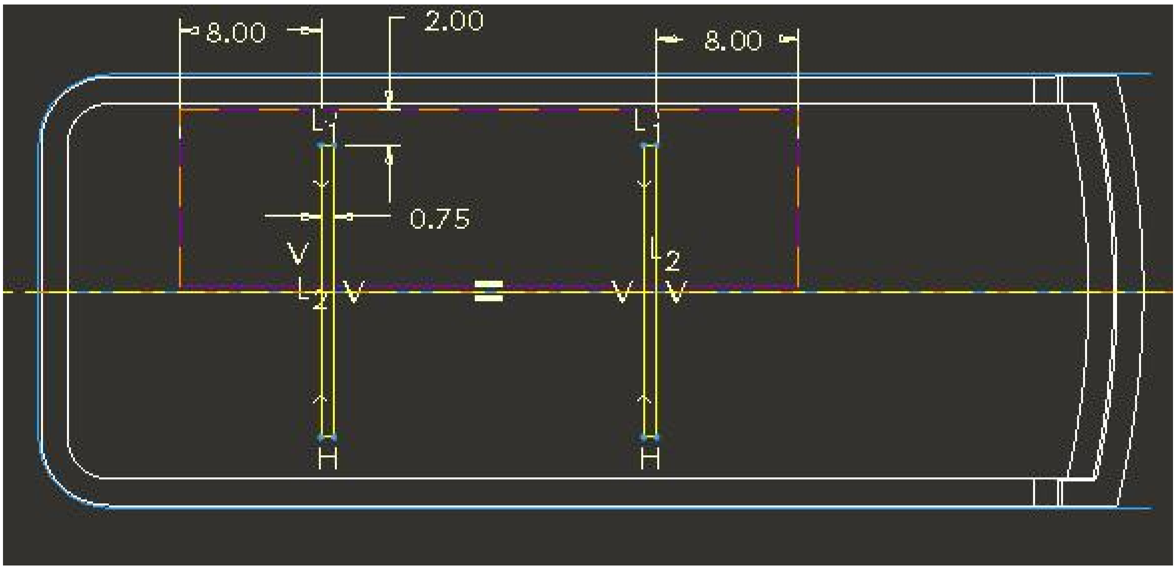
We will create a datum plane on the fly to be used as sketching plane.

Pick  on the Model tab and select the references as shown below. (View is set to Right in the figure).



Notice that datum plane is parallel to the TOP datum plane and tangent to the lower half of the battery surface.

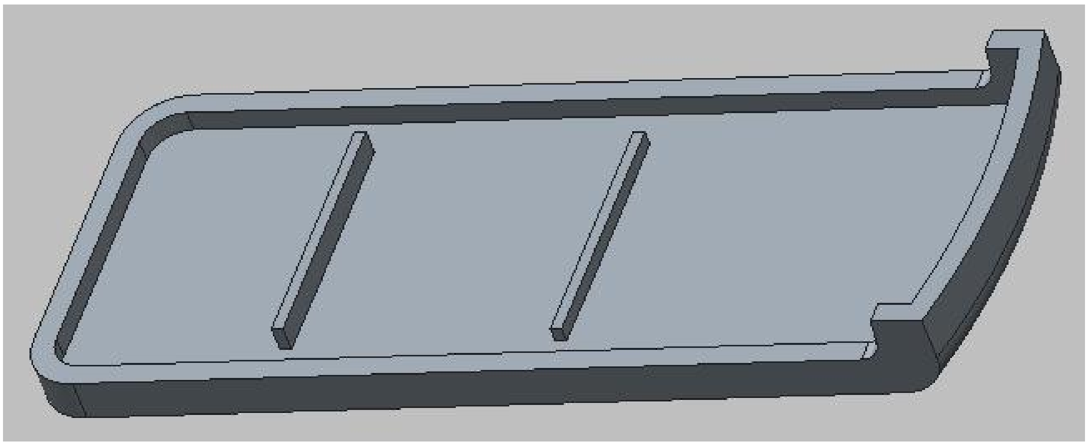
After completing the datum plane, select the surfaces of the battery as reference and sketch rectangles as shown below



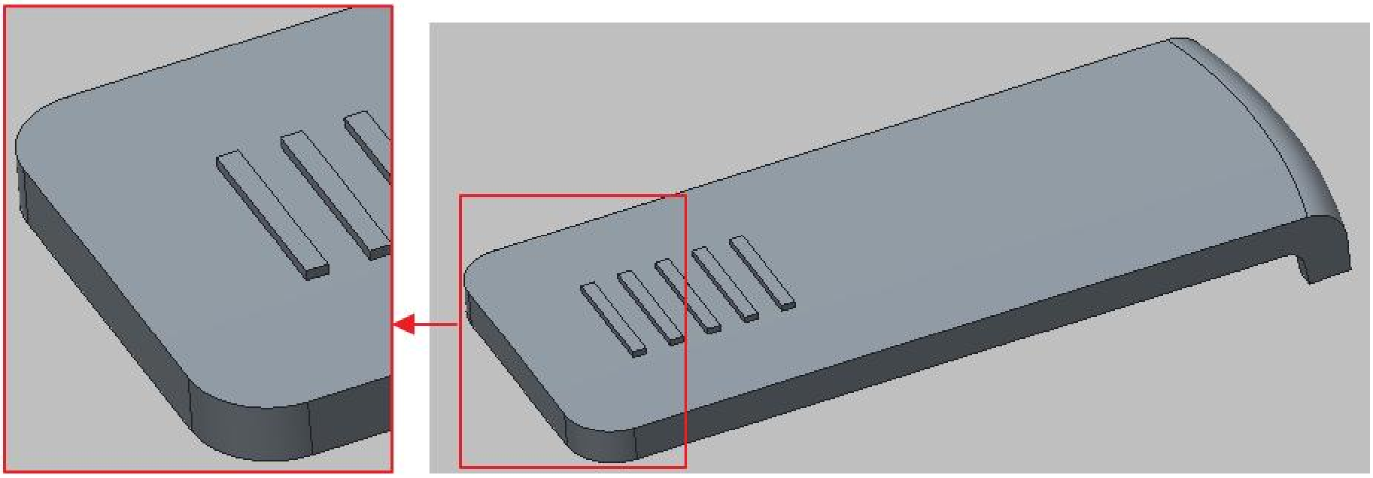
After completing the sketch, pick  to flip the direction of feature creation.

Change the depth option to **To Next** ().

Pick  icon to complete the feature and part will appear as shown below (after hiding all layers).



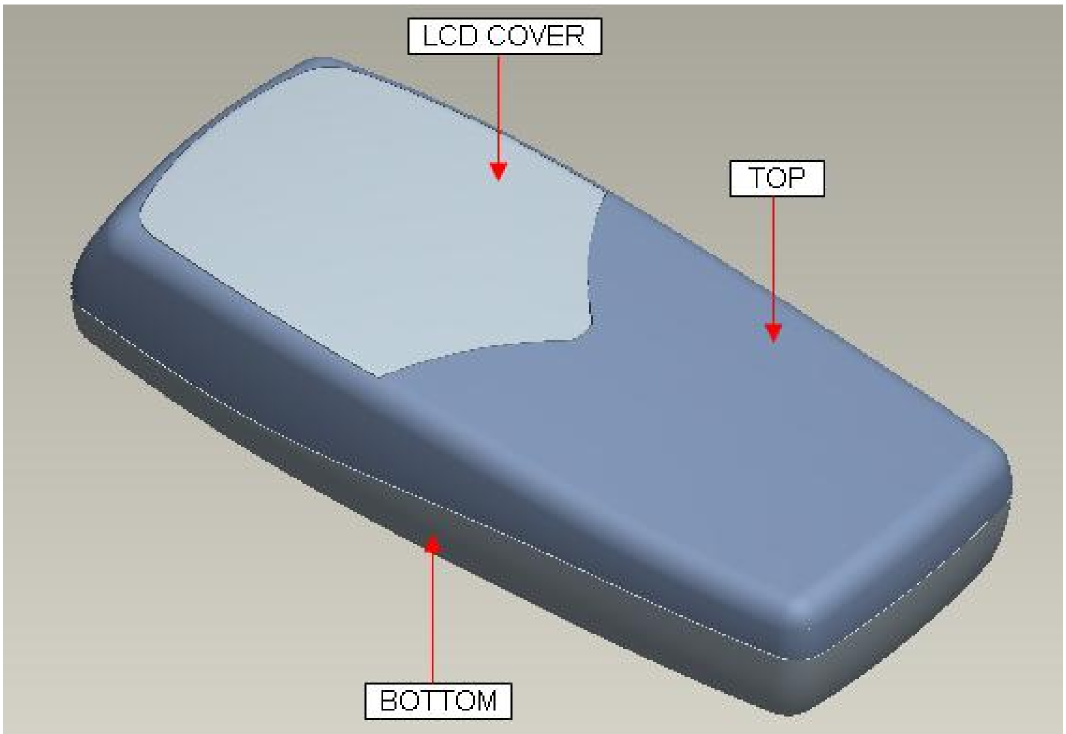
Now you can create the raised pattern as shown below. You do not need the reference geometry for this feature in the skeleton as it has no relation to any other part.

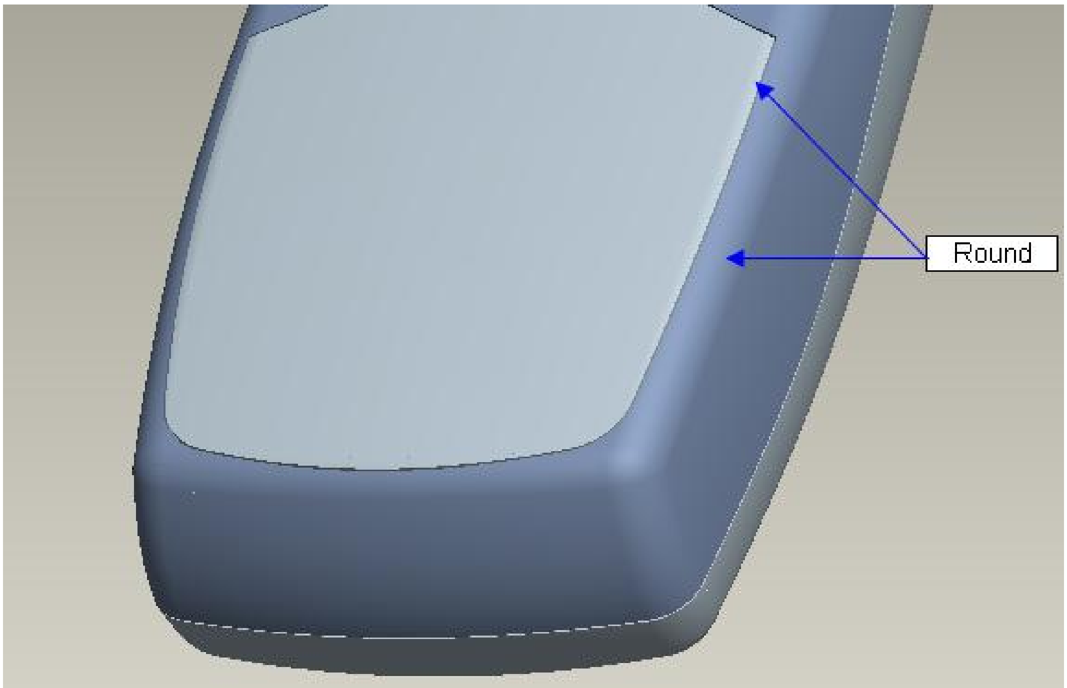


It completes the basic shape of this part. We will not create the locking details for this part. You can do so, on your own and see how to achieve fit between parts. In the FINALIZED folder you can see the other parts in completed form.

Exercise 3

In this exercise we will create a plastic housing for a mobile phone consisting of three parts. The completed product is shown below.





The common attributes between TOP and BOTTOM parts are as follows

1. Profile of both components is same.
2. Both components join along a 3-D surface

The common attributes between TOP and LCD_COVER parts are as follows

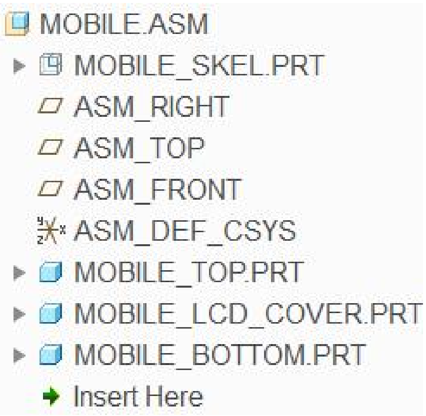
1. Profile of LCD COVER and accommodating cut in the TOP part is same.
2. The outer shape of both parts follow common surface
3. The round surfaces are shared by both (see following figure).

So we will create the geometry in the skeleton model that controls the above common attributes.

Only, the geometry that affects or crosses more than one component (or has to be referenced by more than one component) should be placed in the skeleton.

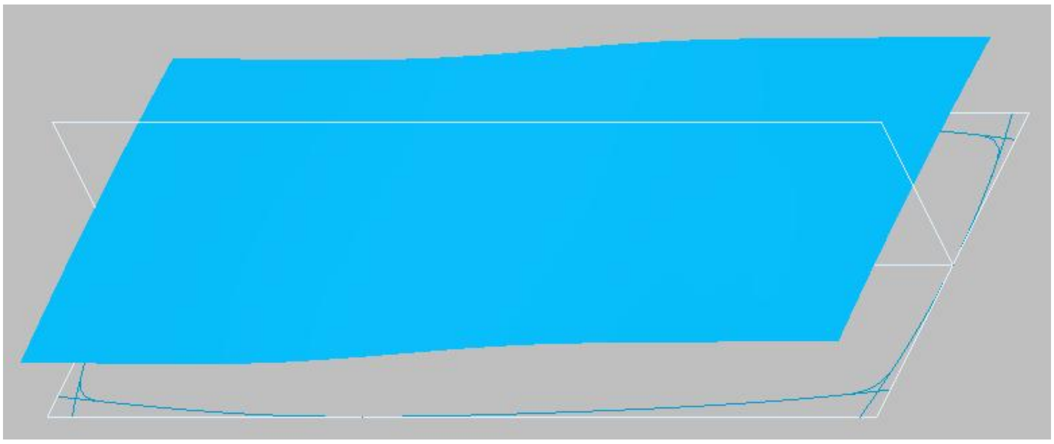
Set the working directory to MOBILE folder and open the assembly MOBILE.ASM

Notice that there are three components assembled with Default constraint.



Building the Skeleton

Open the skeleton model in a separate window and it will appear as shown below.



Here you can see split surface and profile datum curve. Profile datum curve is built in two stages (features). First, a datum curve is sketched without radius applied at the corners and then a second datum curve that references the first curve but with radius added at corners. (Look at the MAIN_PROFILE_WO_RAD and MAIN_PROFILE datum curve features for clear understanding)

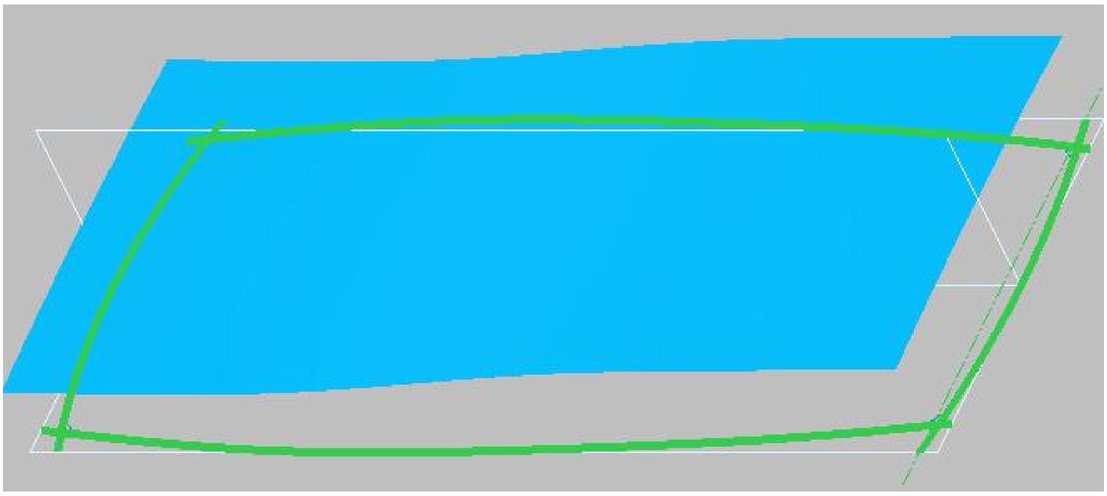
Other geometry has been placed on custom layers and hidden. The white curves represent the horizontal and vertical boundaries of the mobile assembly.

We will create the skin of the upper half of the mobile completely in the skeleton model because top surface and rounds are shared by multiple parts.

This project requires creating separate variable section sweep features to build side surfaces. The trajectory for these surfaces will be the profile curve (MAIN_PROFILE_WO_RAD) projected on the split surface. So first we will project the profile datum curve (without radius) on the split surface.

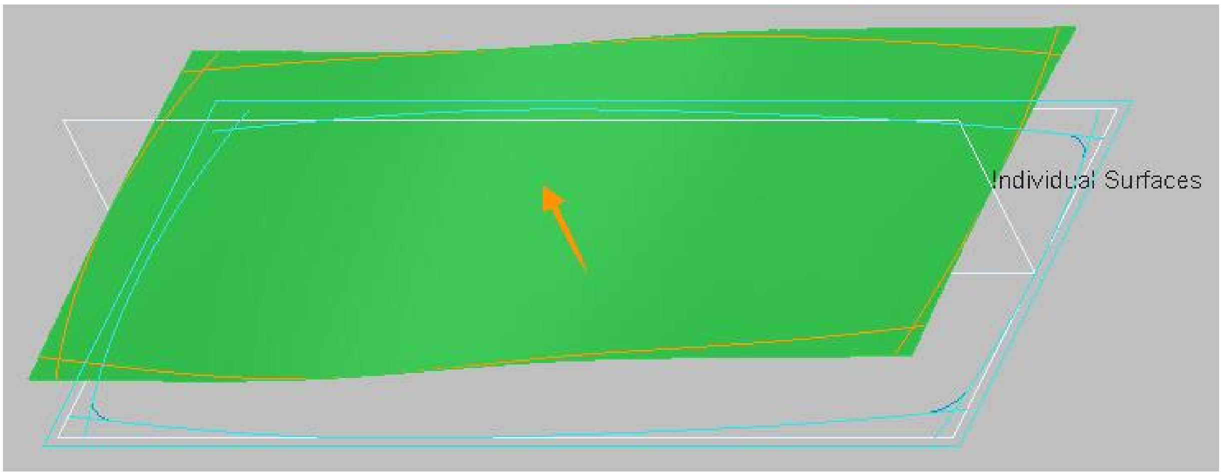
When the split surface is non-planar then it becomes necessary to begin with split surface.

Pick the following highlighted curve. Change the filter setting to **Feature** for ease of selection.



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.



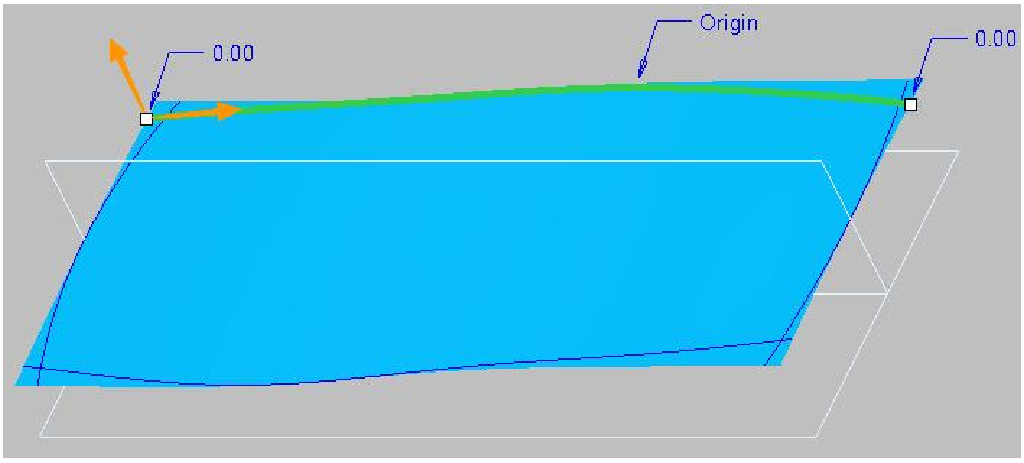
Pick to complete the feature.

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

Before proceeding hide the SKL_CURVE_PROFILE layer.

Pick Sweep icon to invoke the sweep tool.

Pick the following curve as reference. Set the view to **ISO3** to avoid any confusion.

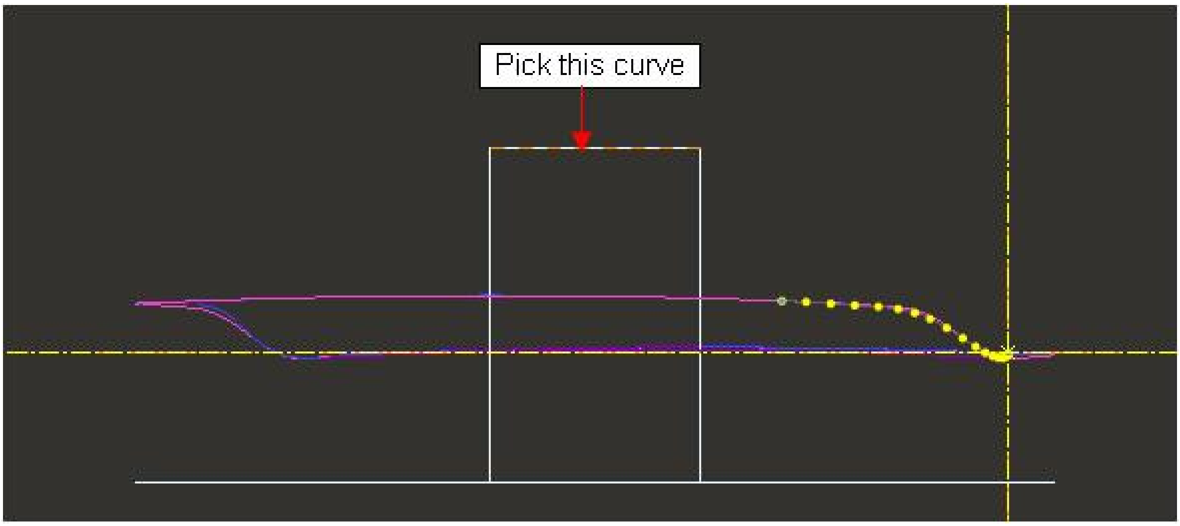


Pick References tab and change the “Section plane control” option to **Normal To Projection**.

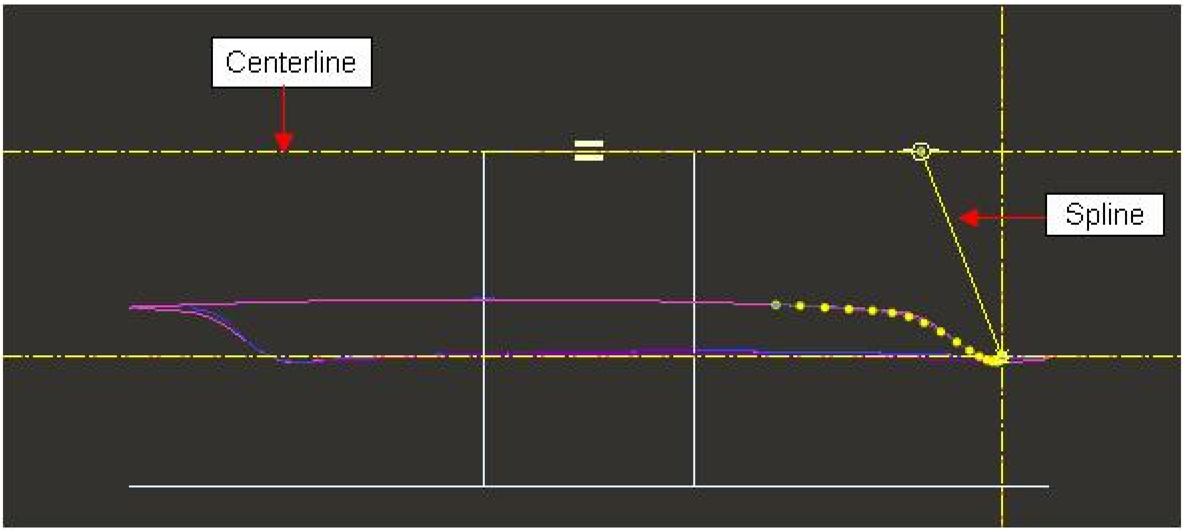
Select **SKL_TOP** datum plane as direction reference.

Pick  to create the feature as a surface.

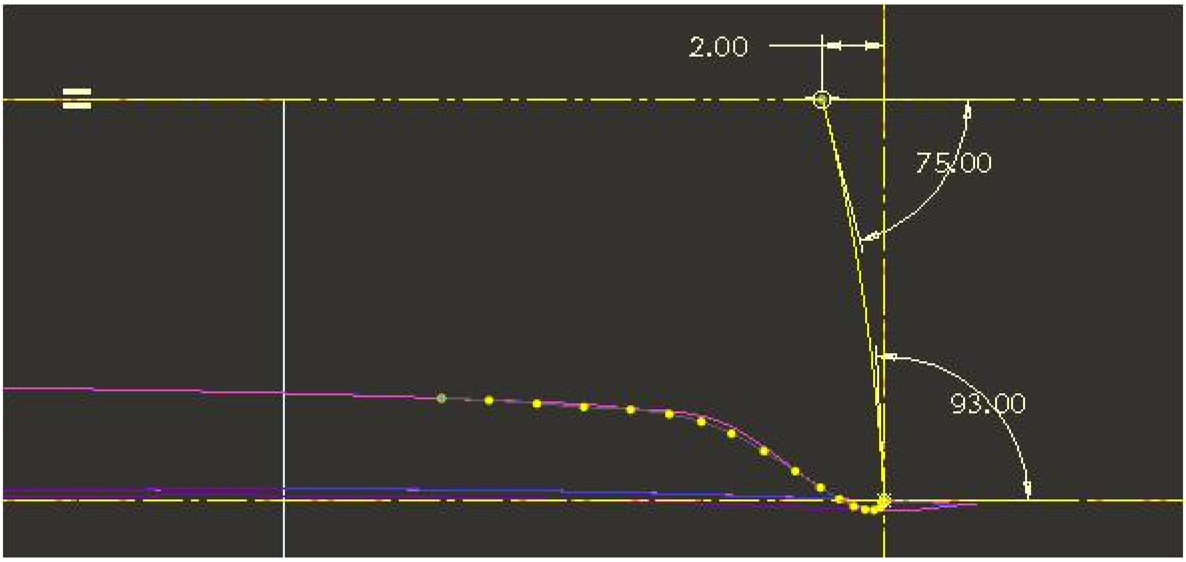
Pick  and select the following curve as reference.



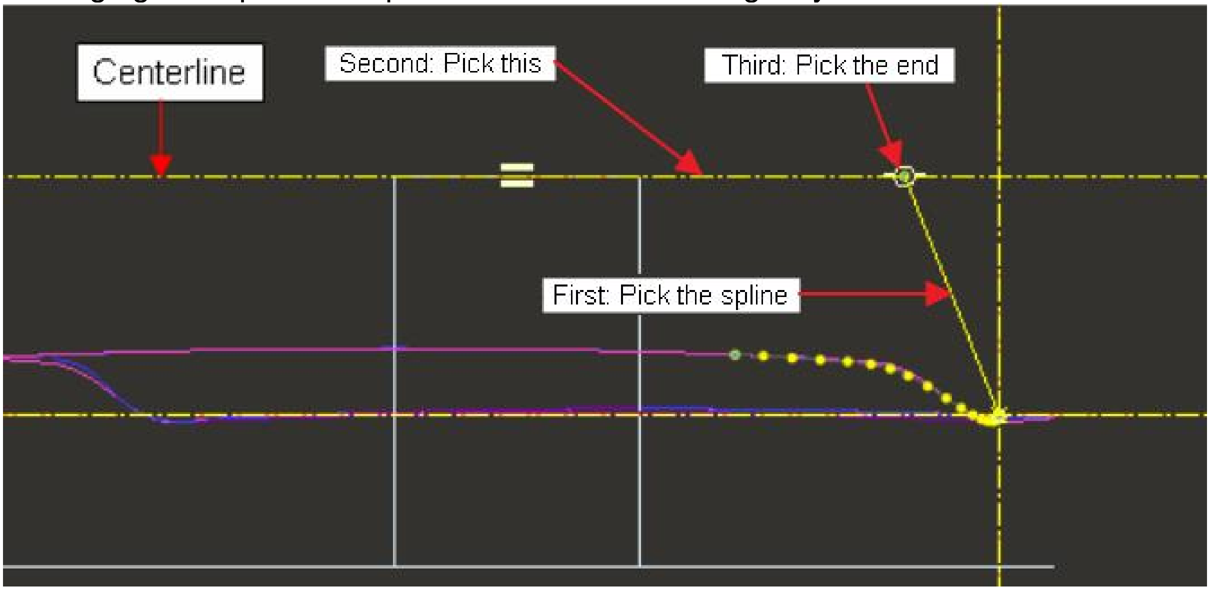
Sketch a centerline and two point spline (by picking Spline) as shown below.



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 following figure explains the procedure to create a tangency dimension.



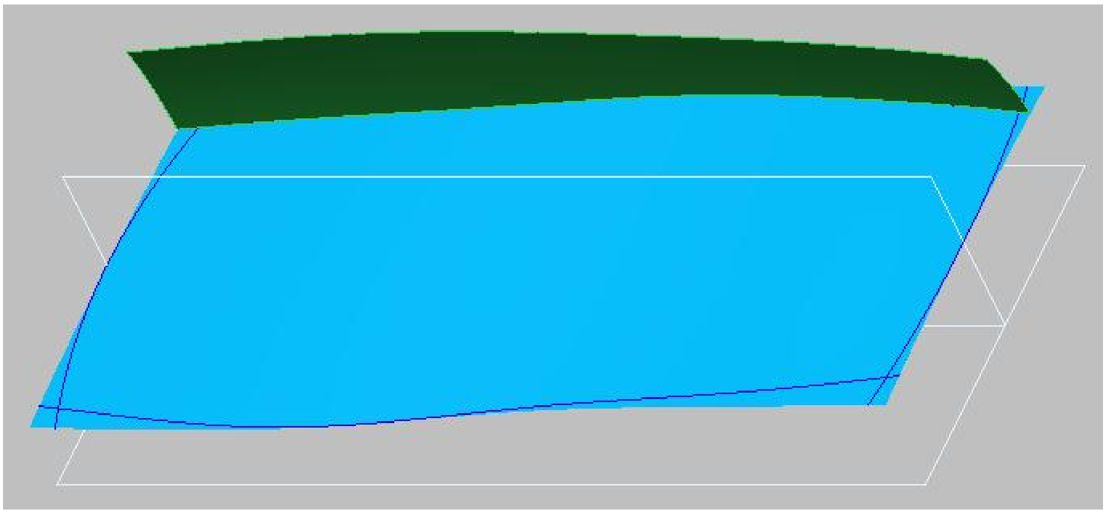
This dimensioning scheme builds the draft in the surface definition.

Pick  after completing the sketch.

Pick  to create the feature as variable section sweep.

Pick  to complete the sweep feature.

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



Pick the Mirror Tool icon 

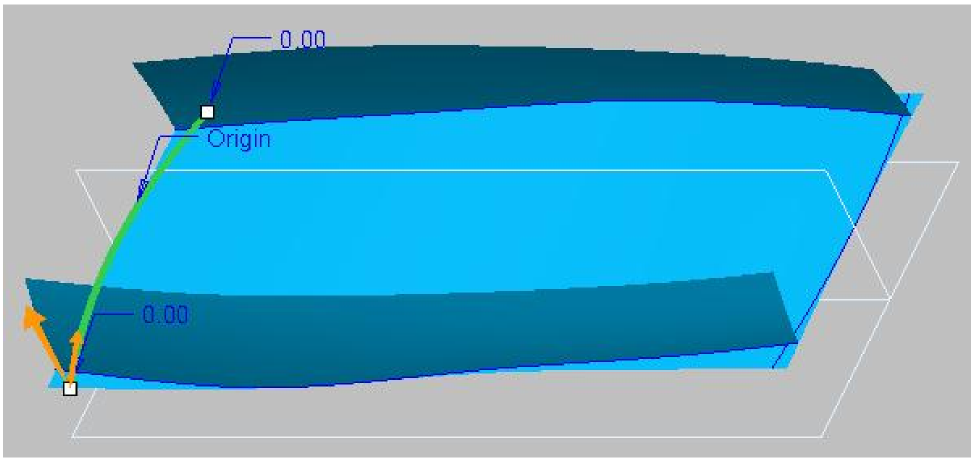
Select the **SKL_RIGHT** datum as Mirror Plane.

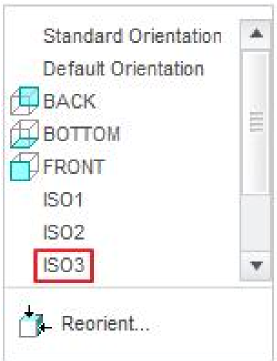
Pick  to complete the feature

Now we will create the front side surface.

Pick 

and select the following curve as trajectory. Set the view to **ISO3** to avoid any ambiguity.



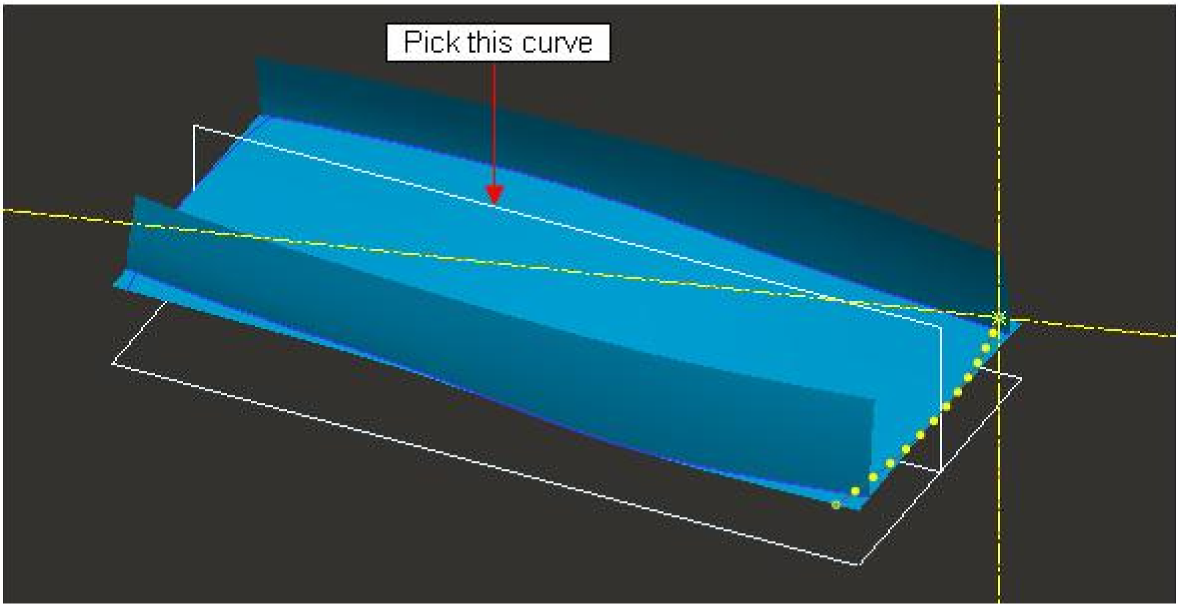


Change the “Section plane control” option to **Normal To Projection**.

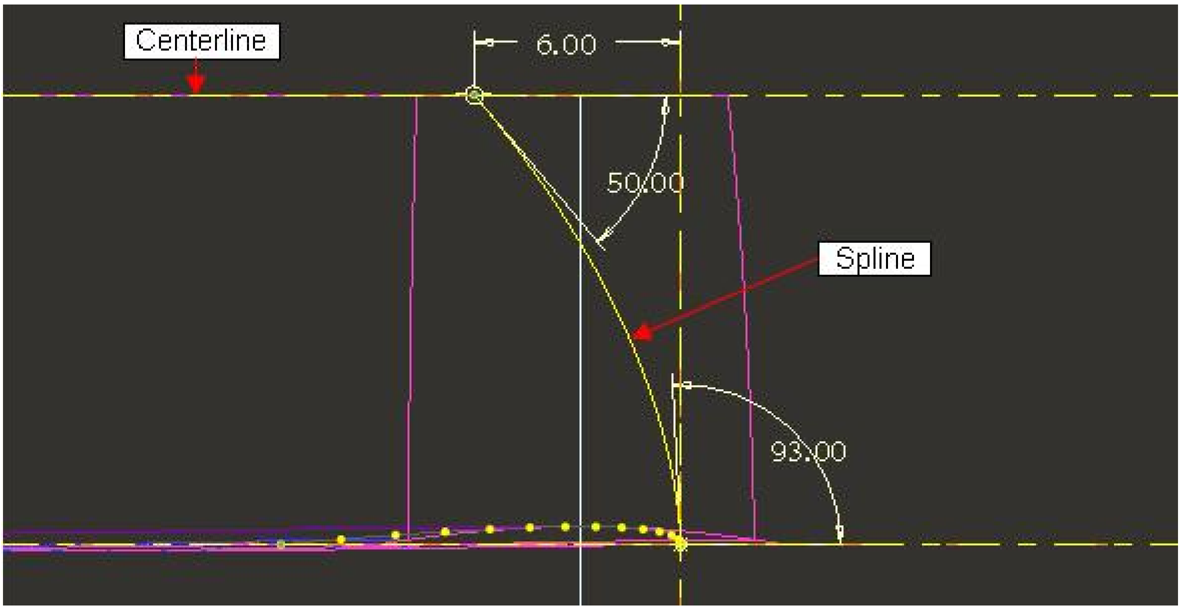
Select **SKL_TOP** datum plane as direction reference.

Pick  to create the feature as a surface.

Pick  and select the following curve as reference.



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



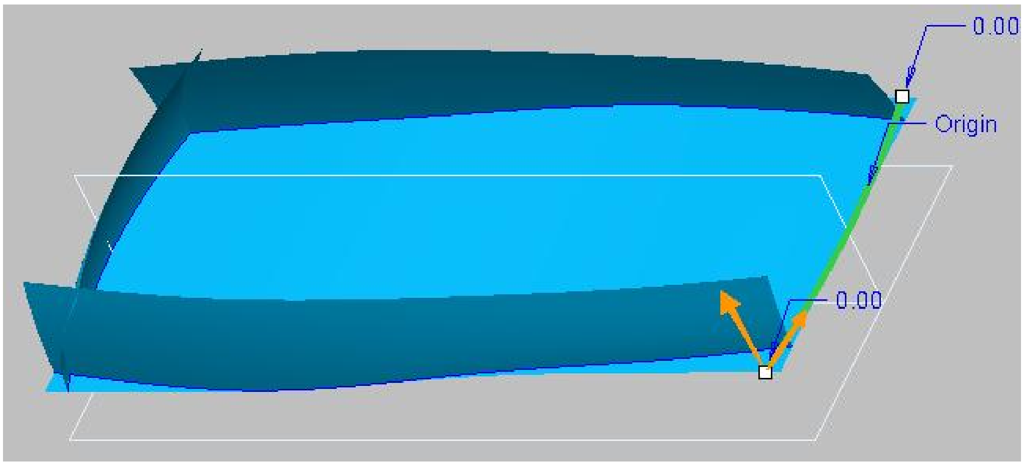
Pick after completing the sketch.

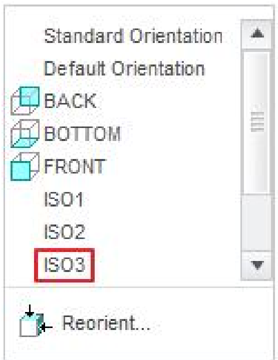
Pick to create the feature as variable section sweep.

Pick to complete the Variable Section Sweep feature.

Now we will create the back surface.

Pick Sweep and select the following curve as trajectory.



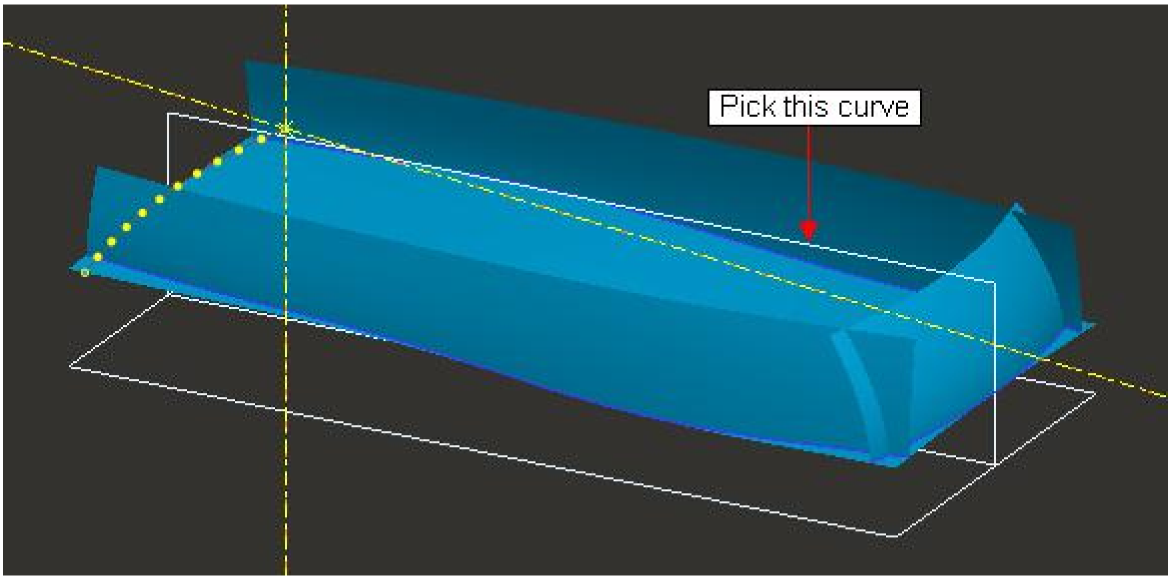


Change the “Section plane control” option to **Normal To Projection**.

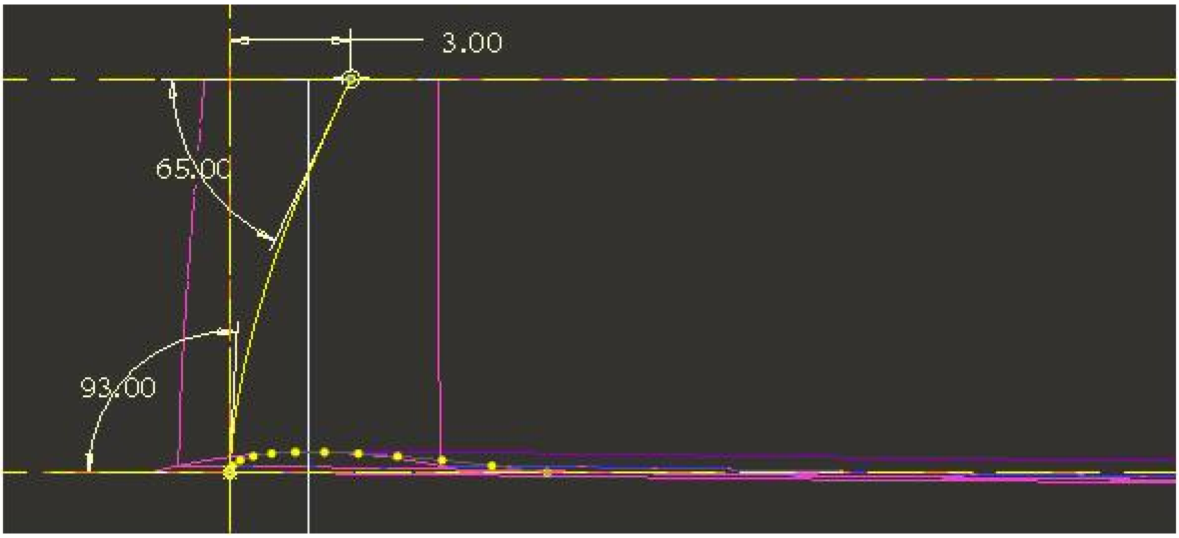
Select **SKL_TOP** datum plane as direction reference.

Pick to create the feature as a surface.

Pick and select the following curve as reference.



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



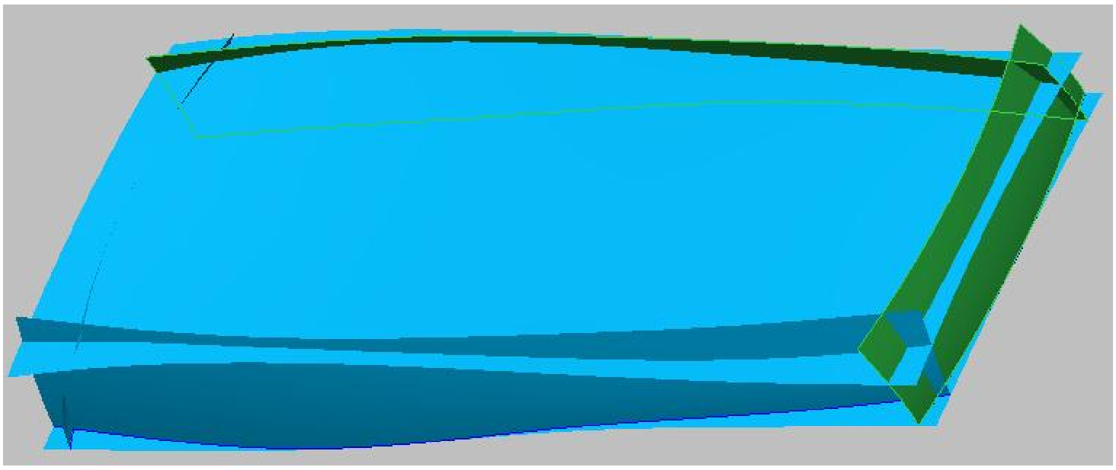
Pick  after completing the sketch.

Pick  to create the feature as a variable section sweep.

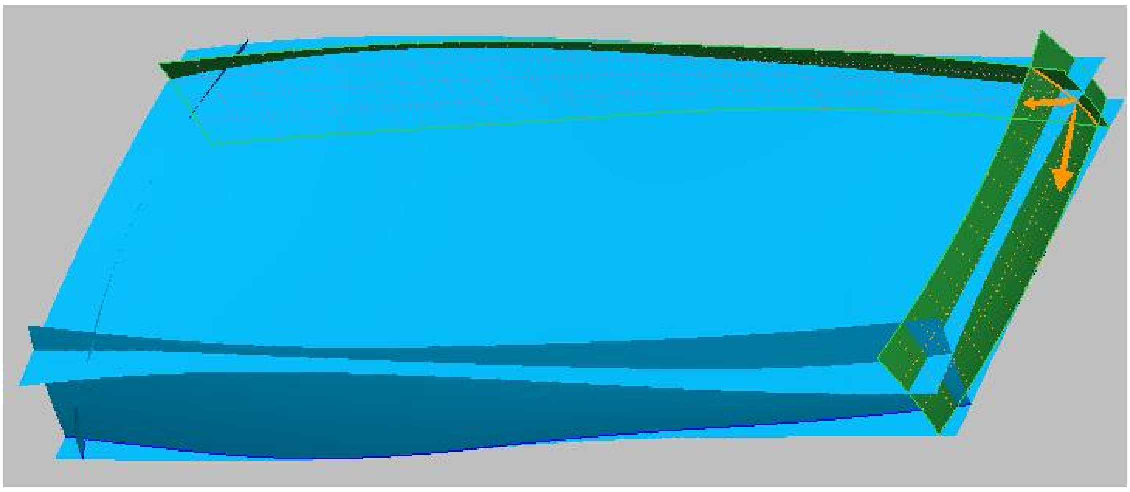
Pick  to complete the sweep feature.

Now we will merge the surfaces together. But before that unhide the SKL_SURF_TOP layer and hide SKL_CURVE_BOUNDARY layer. Set the view to **ISO3** to avoid any ambiguity.

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

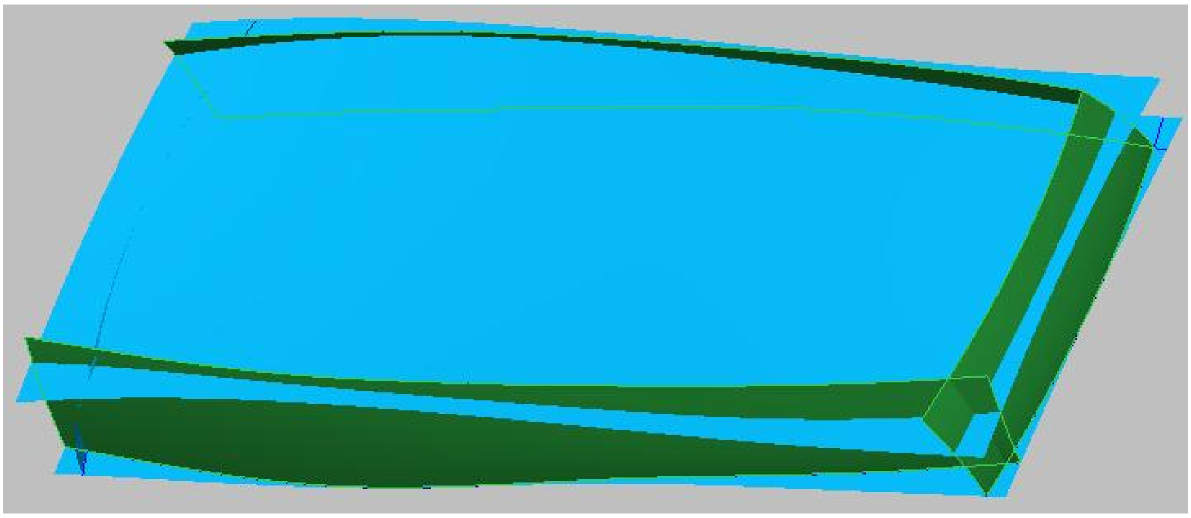


Pick Merge to invoke the Merge tool. Make sure that preview appears as shown below.

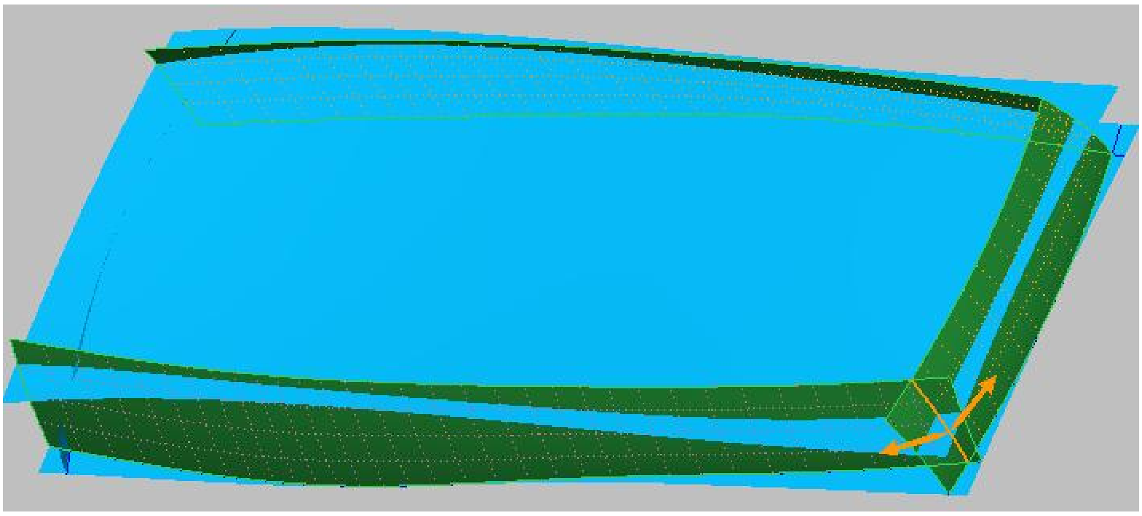


Pick to complete the feature.

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

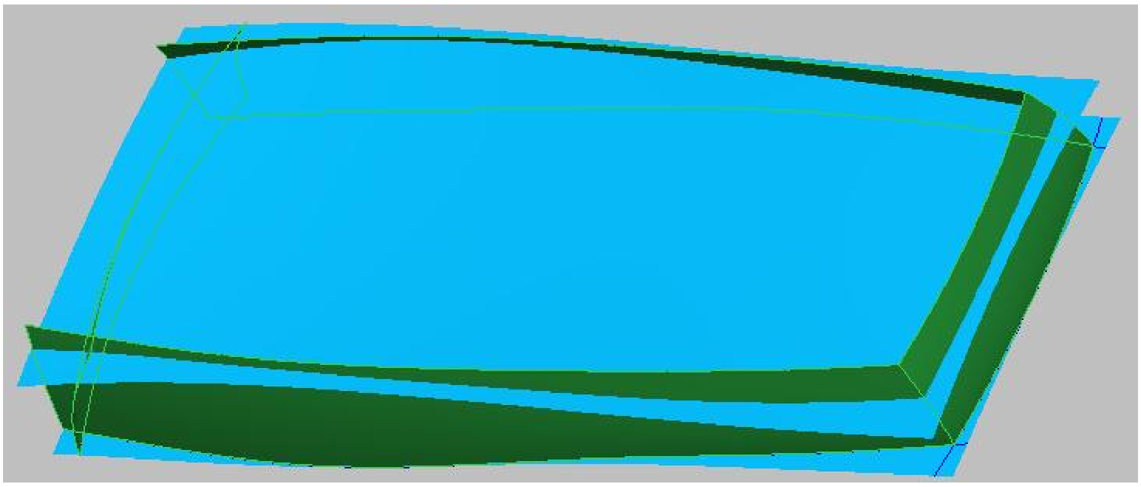


Pick Merge to invoke the Merge tool and make sure preview appears as shown below.



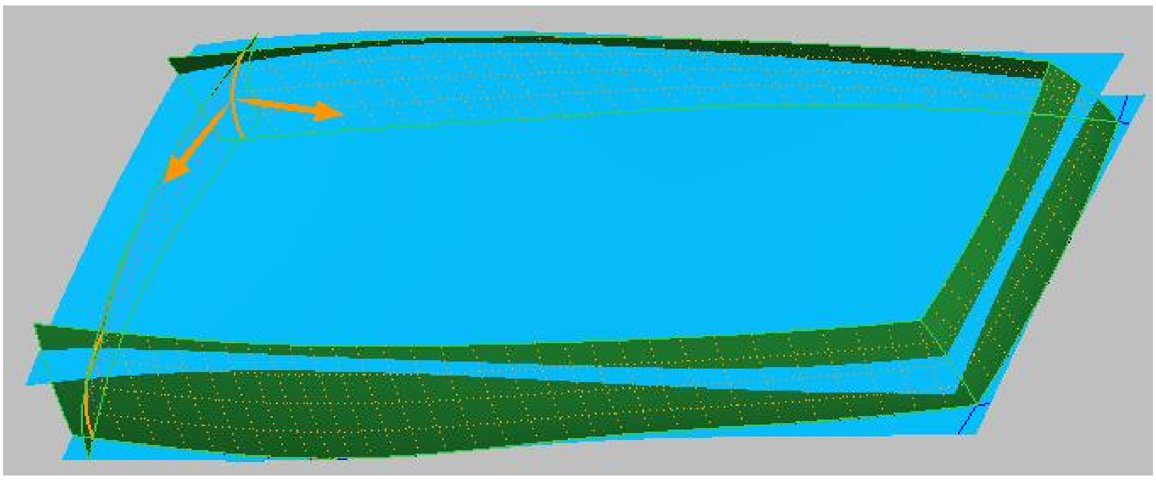
Pick  to complete the feature.

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



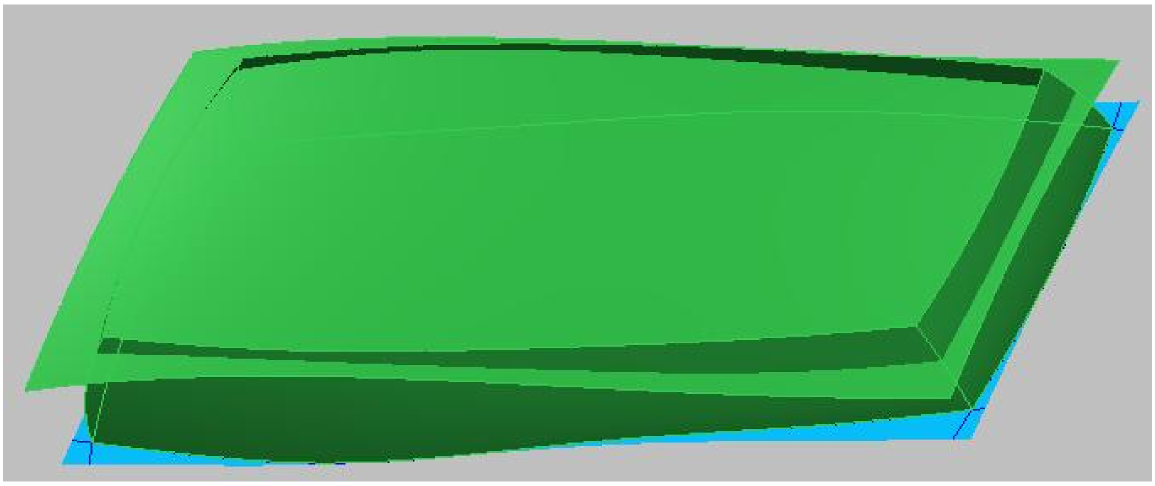
Pick 

to invoke the Merge tool and make sure preview appears as shown below.



Pick  to complete the feature.

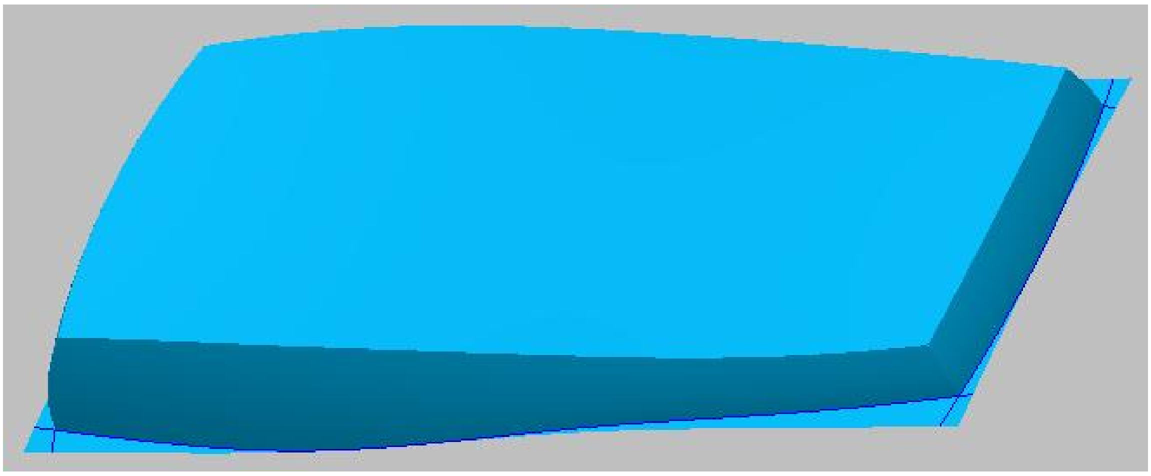
Now select the merged quilt and top surface, while holding down the Ctrl key, as shown below.



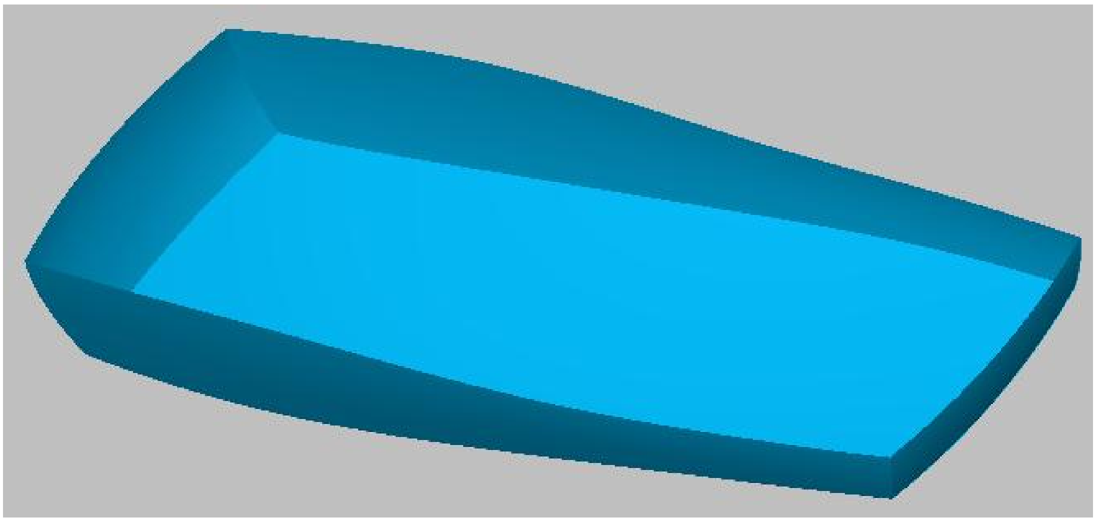
Pick 

to invoke the merge tool.

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

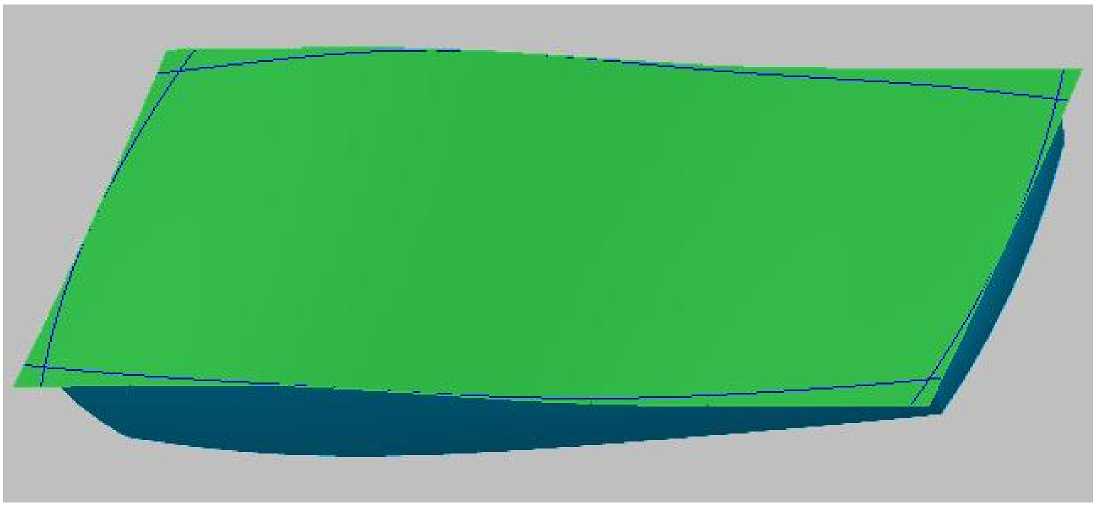


If you look at the merged quilt (after hiding other geometry), you will notice that at the lower side it is open. If we apply rounds to the side edges, the results are unpredictable at the open sides of the edge. The surfaces created by such round features may create problems later. So we will first merge the split surface with this quilt and then apply the round.



Now we will create a surface copy of the split surface that will be merged with the quilt. Although we can use the original split surface, but if we use it, the merge operation will consume this surface. So it is better to copy the surface and then use the copied surface for merge operation. So that we have the original split surface available for other operations.

So select the split surface.



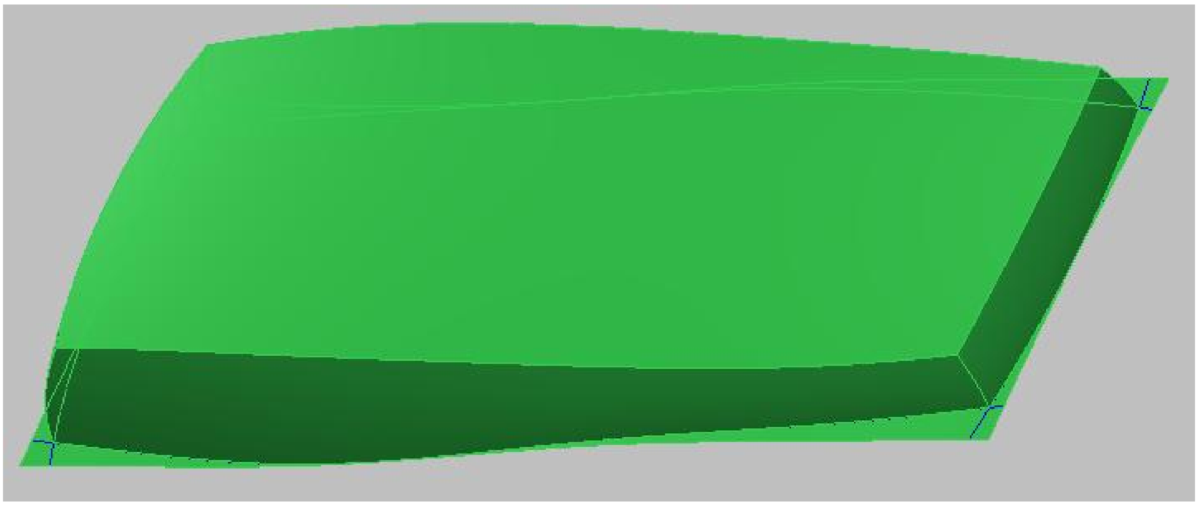
Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

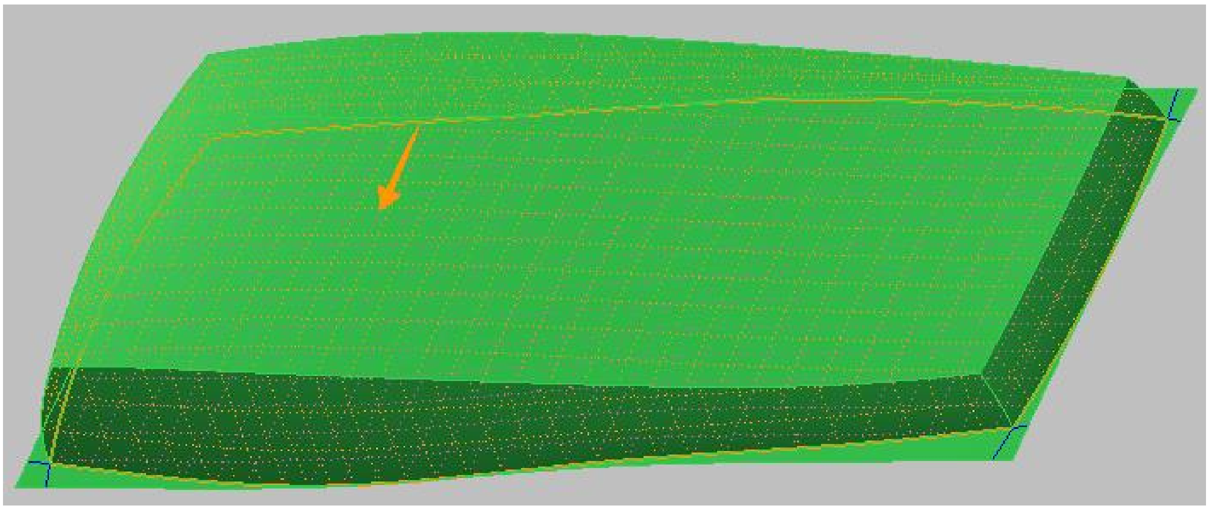
The Copy dashboard will appear so pick to complete the feature

Before proceeding hide the SKL_SURF_SPLIT layer.

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



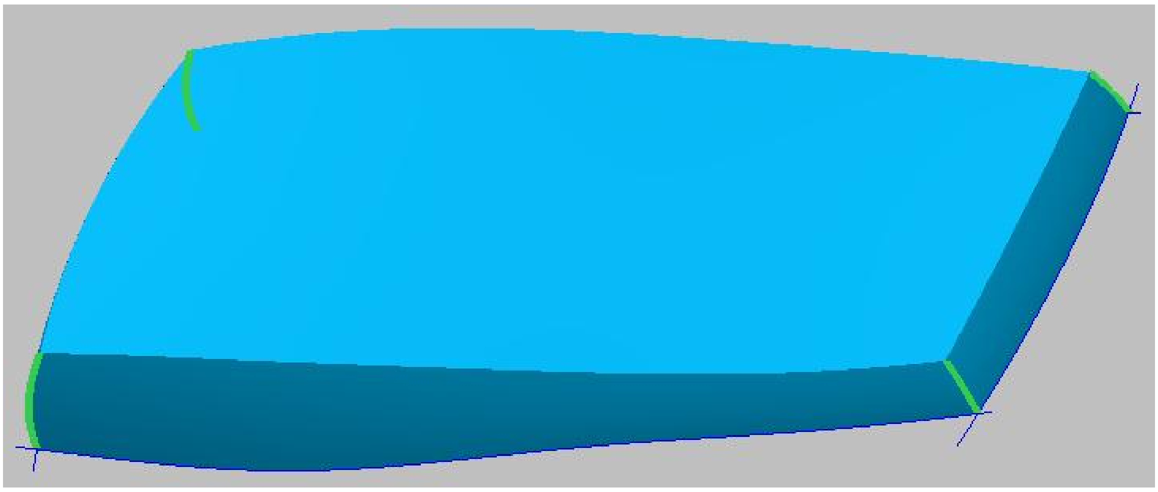
Pick Merge to invoke the merge tool and make sure preview appears as shown below.



Pick to complete the feature.

Now we will apply round at the edges.

Now select the following edges while holding down Ctrl key.



Pick  to access the Round tool.

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

Pick  to complete the feature.

Now we will create a relation to make sure that after any change our skeleton model confirms to the design intent. This step is optional; if you are not familiar with relations then you can skip it and continue with the next step of applying round to other edges.

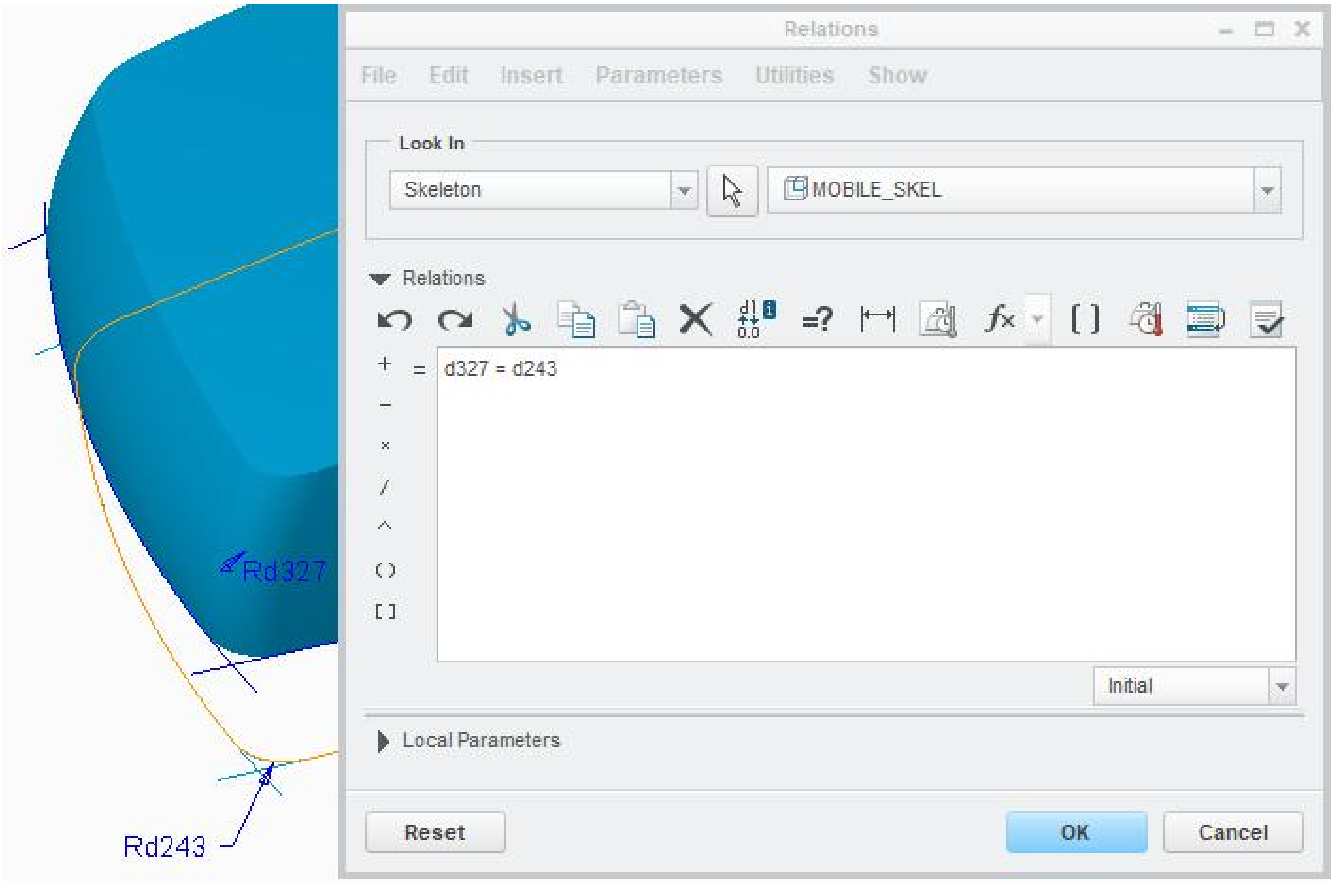
The radius value applied at the edges should be equal to the value of radius applied in the main profile curve. So we will write a relation to achieve this intent.

So first unhide the SKL_CURVE_PROFILE layer.

Pick  on the Tools tab and Relations dialog box will appear.

Pick on the round and the main profile curve on screen and system will show symbols assigned to the dimensions.

Add the relation as shown in the figure below.

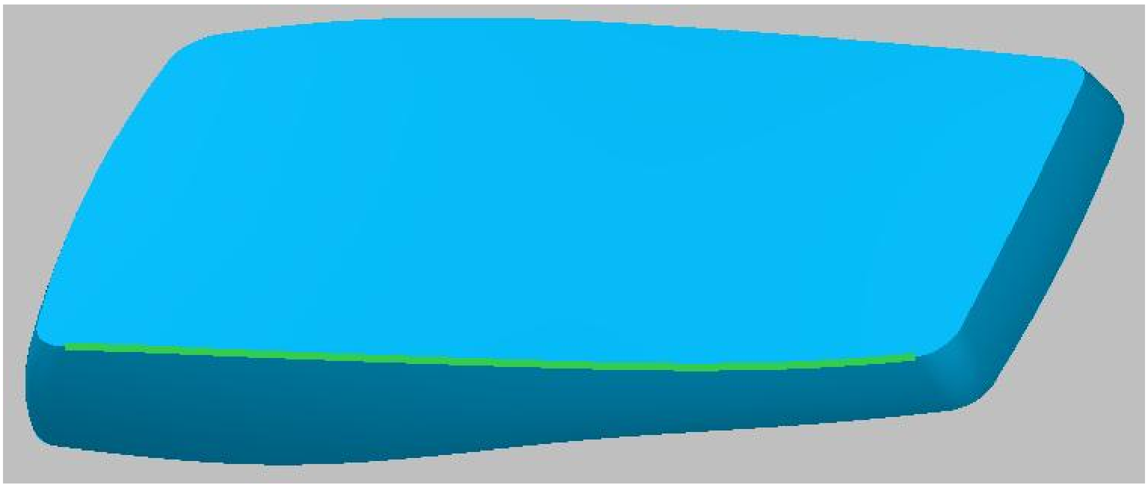


The dimension symbols may be different for your case.

This relation makes sure that interlock details and outer shape of the part will update accurately as the geometry of the skeleton changes. (INTERLOCK_LIP datum curve is dependent on the profile curve).

Hide the SKL_CURVE_PROFILE layer.

Now select the following edge.

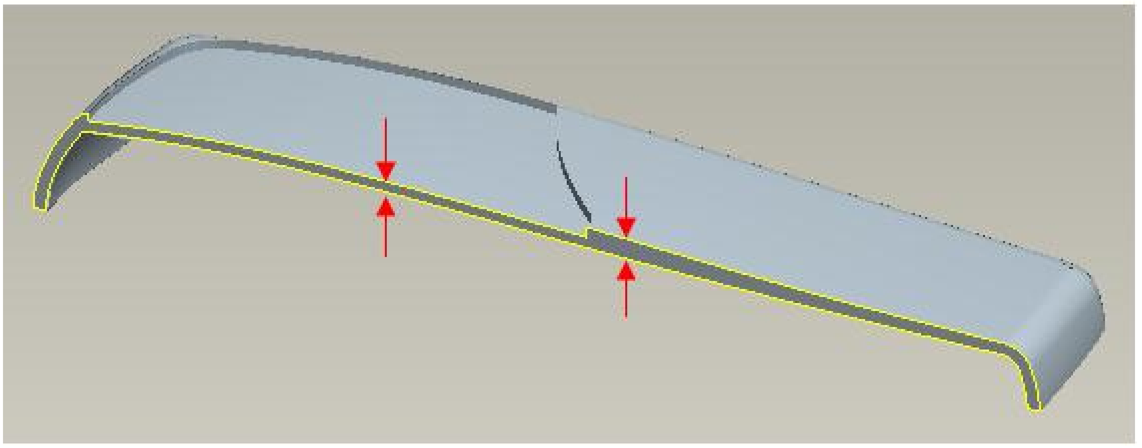


Pick Round to access the Round tool.

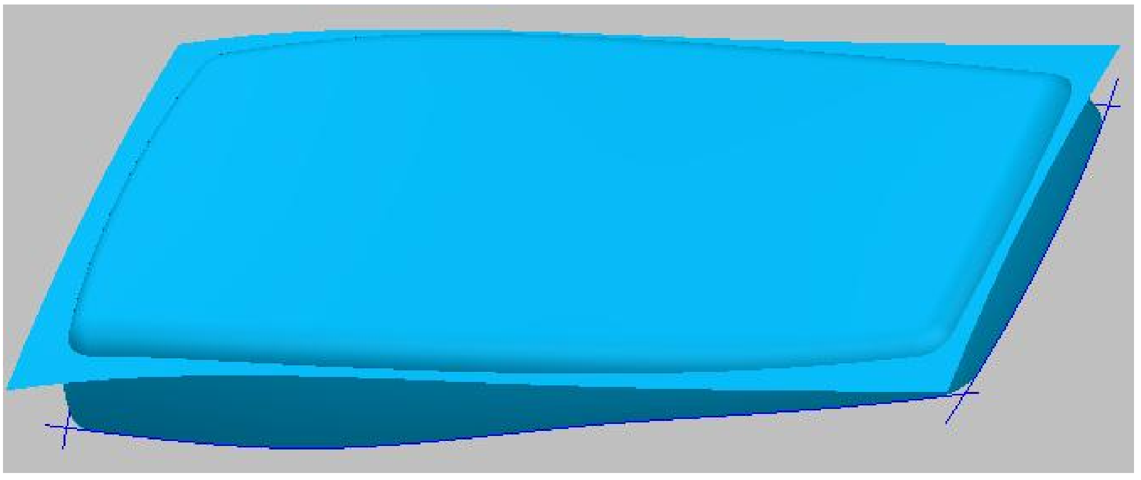
Enter ***3.25*** as the radius value

Pick ✓ to complete the feature.

The following figure is the cross-sectional view of the completed MOBILE_TOP part. You can see that inner surfaces are not uniform offset from the outer surfaces. So we will use surfacing techniques, instead of shell tool, to hollow out this part.

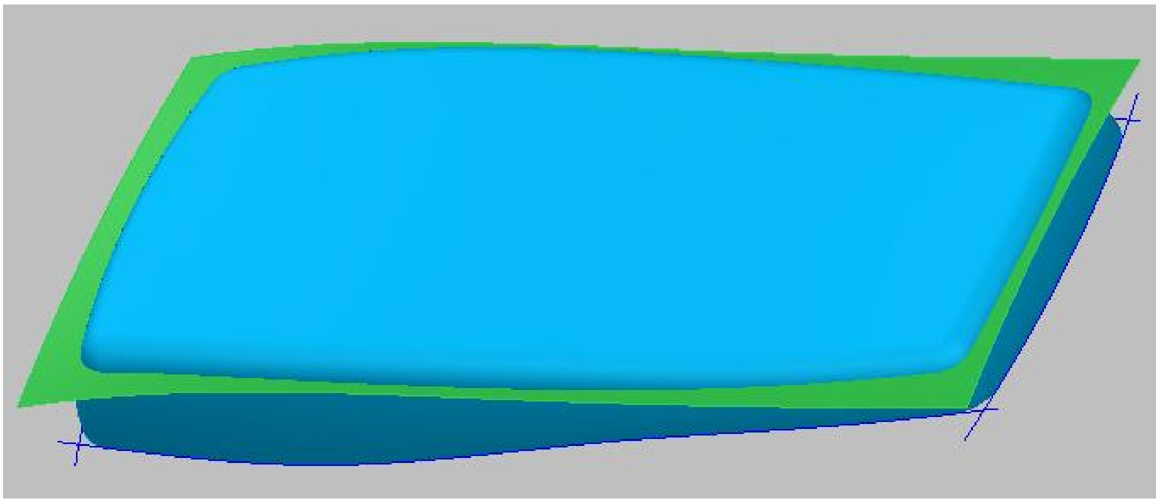


Therefore we will need a separate surface to create inside geometry of the part. The surface to build the inner top surface is already created and lies on SKL_SURF_TOP_INNER layer that is hidden. So unhide this layer and the model will appear as shown below. We will call this surface "Top Inner Surface" in this exercise.



Now we will create a surface that will be the interface between the TOP and LCD_COVER parts.

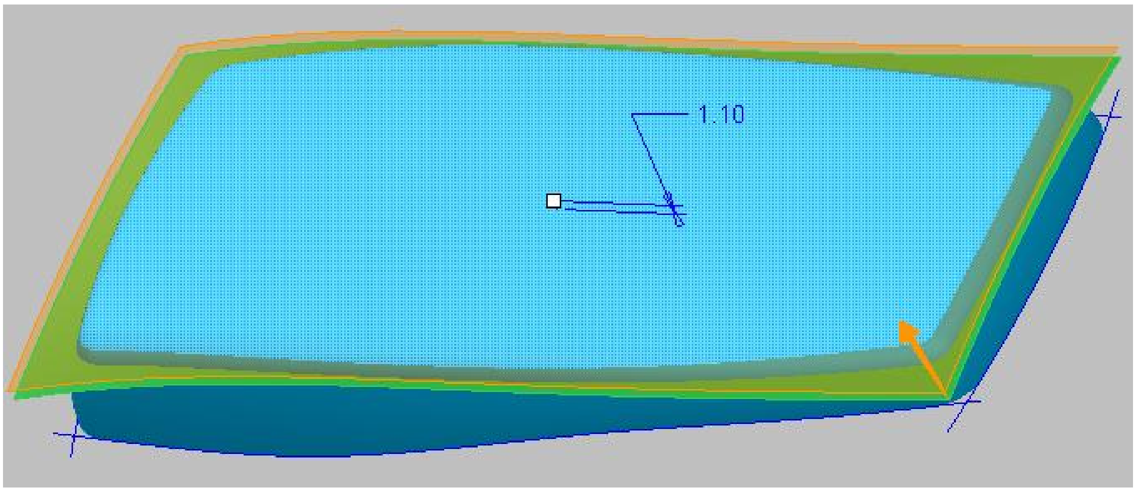
So change the filter setting to **Quilt** and select the following surface.



Pick  to access the Offset tool.

Specify ***1.1*** as the offset value.

The preview should appear as shown below.

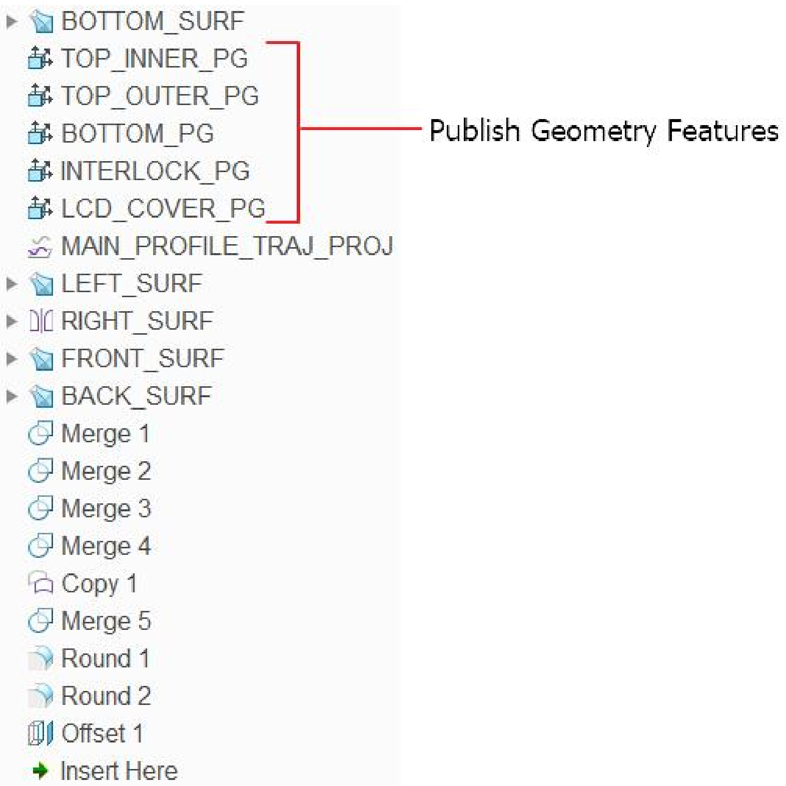


Pick  icon to complete the feature.

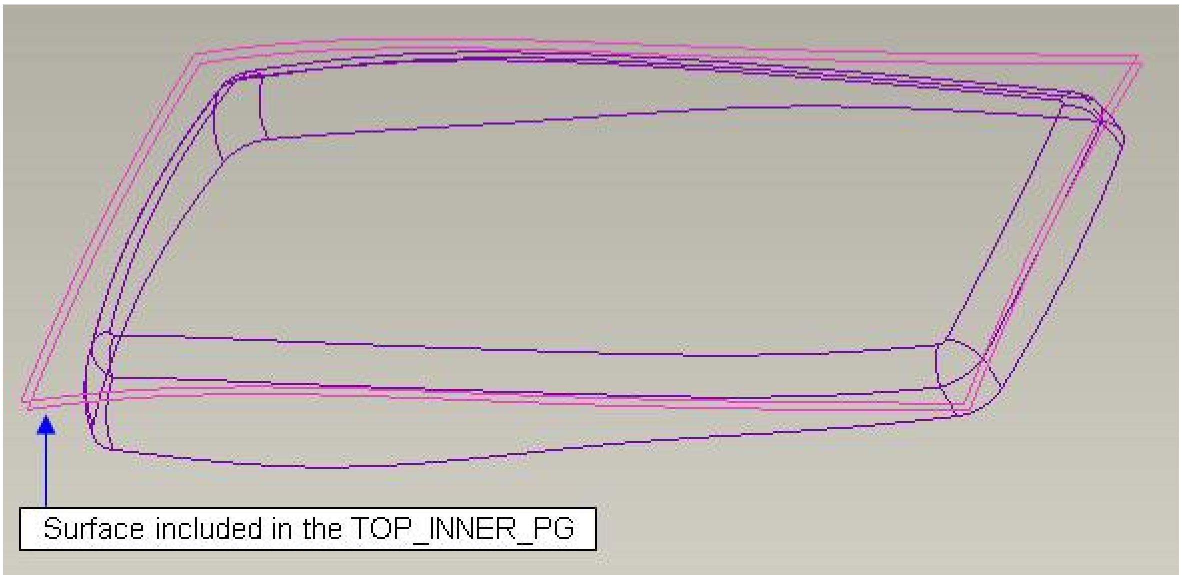
It completes the definition of required features. Now you can rename the features in the model tree. It is always a good practice to rename the features for easy recognition.

Communicating the Design Information

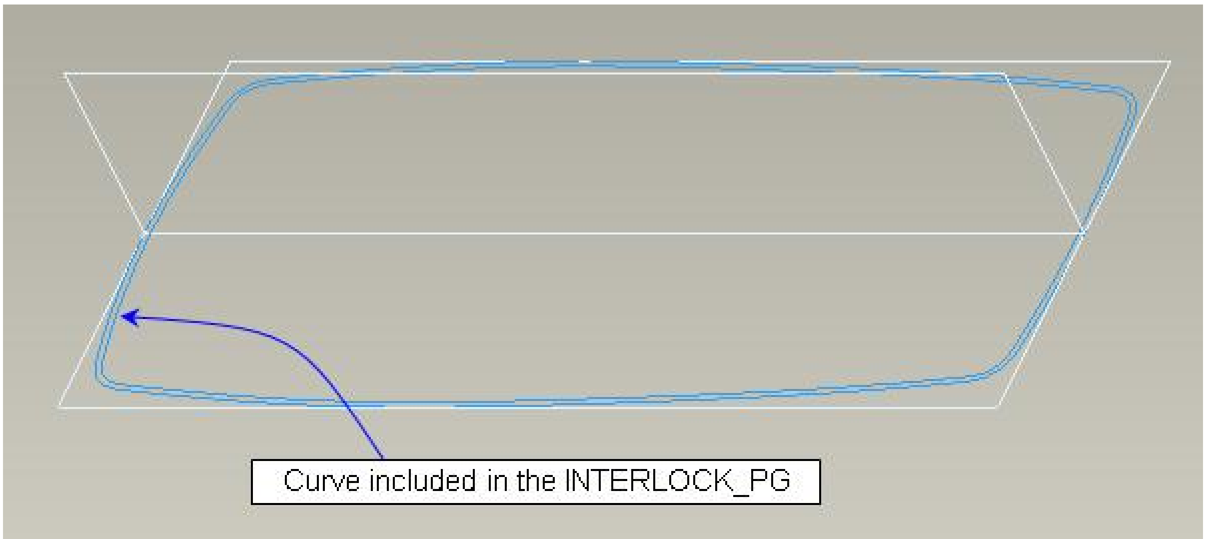
If you look in the model tree you will notice that publish geometry features are already defined as shown below.



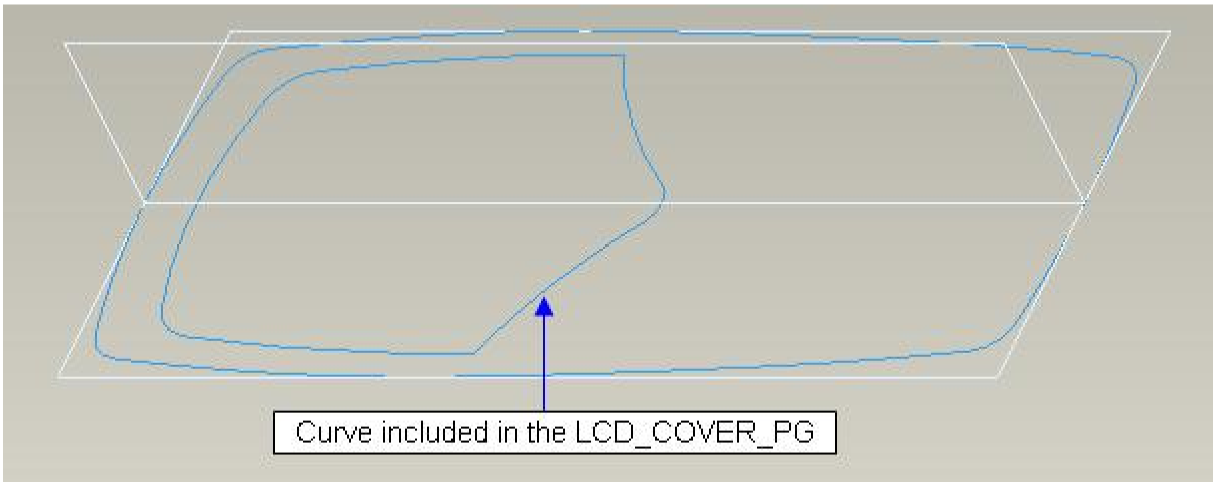
The TOP_INNER_PG feature contains the Top Inner Surface as shown below. (You need to unhide the related layers to see the geometry)



The INTERLOCK_PG feature contains the interlock curve as shown below. (You need to unhide the related layers to see the geometry)



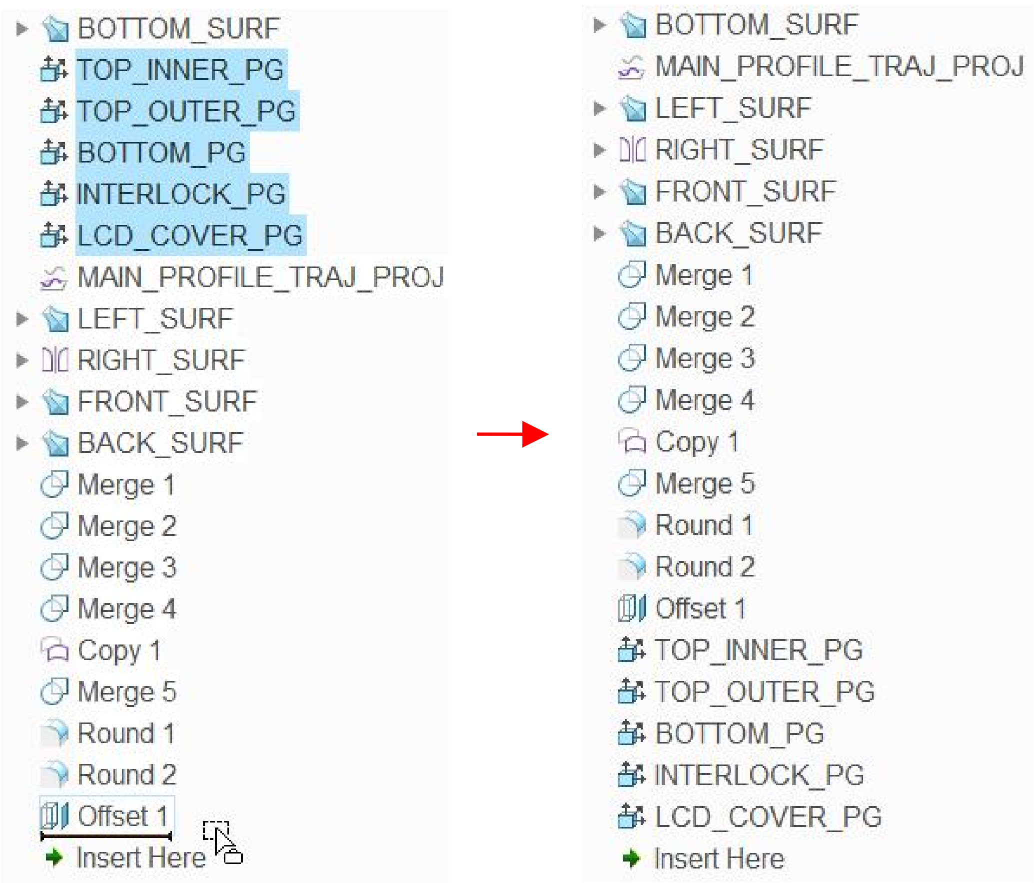
The LCD_COVER_PG feature contains the LCD cover profile curve as shown below. (You need to unhide the related layers to see the geometry)



The BOTTOM_PG and TOP_OUTER_PG publish geometry features are empty (they only contain dummy geometry).

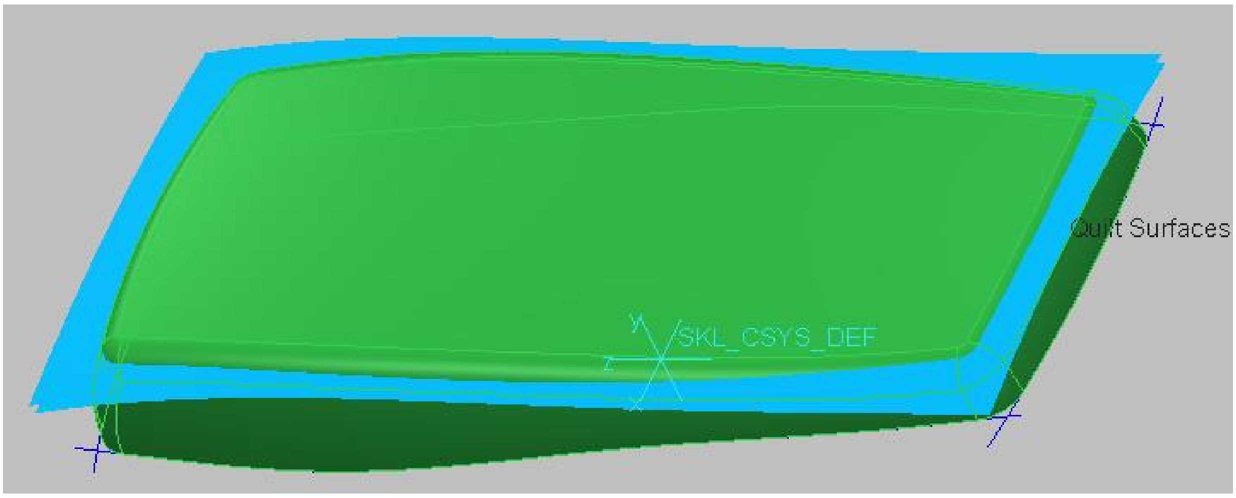
Now we will redefine copy geometry features and add the desired geometry. But first we need to move all the publish geometry features at the end of the model tree.

So select and drag all the PG features as shown below.



Select the TOP_OUTER_PG in the model tree and pick .

Select the following quilt. You should use the Quilt filter if you find the difficulty to select the quilt.

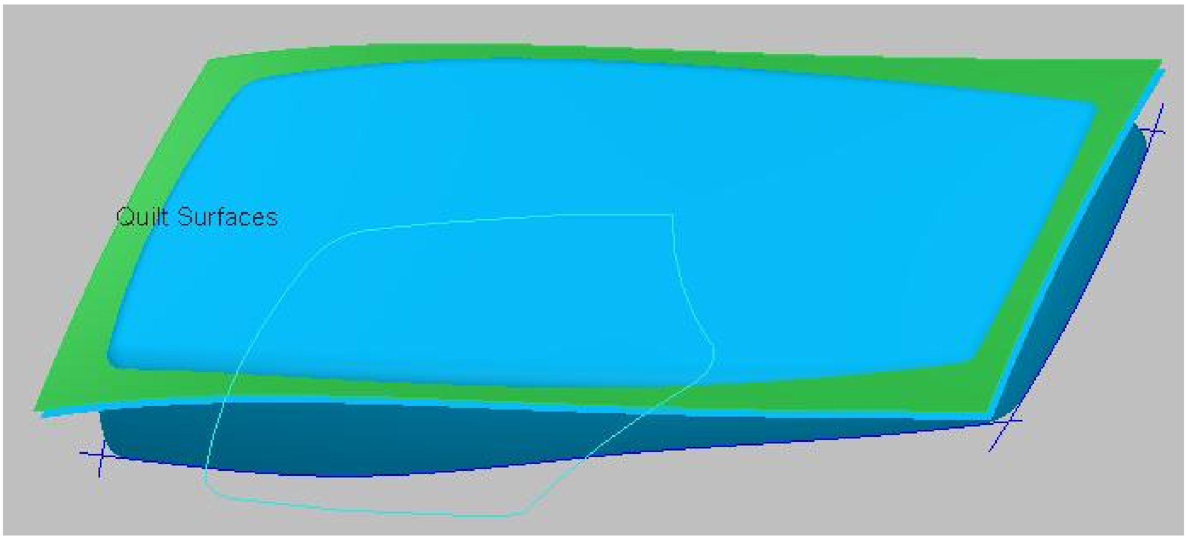


You can remove selections in Reference collector as they were just meant to create the feature.

Pick icon to complete the feature.

Select the LCD_COVER_PG in the model tree and pick .

Select the following surface. (Be careful to select the offset surface)



Pick icon to complete the feature.

Copy Geometry features have already been defined in individual parts so we do not need to create any CGF.

Creating Model Geometry

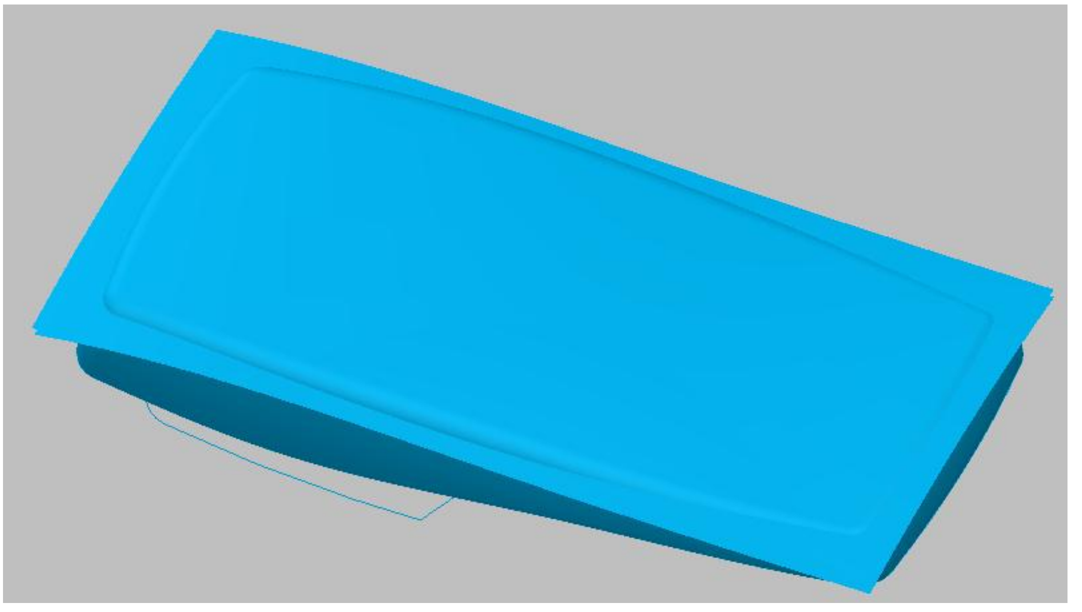
Now the information is captured from the skeleton model into the individual components. We will use the Copy Geometry features as a reference to build geometry in the parts. In the exercise we will create geometry for only two parts. You can create the geometry for other parts on your own. You can see all the parts in completed shape in the FINALIZED folder.

First Part

Open the MOBILE_TOP.PRT in a separate window. Notice that it has already three copy geometry features as shown below.

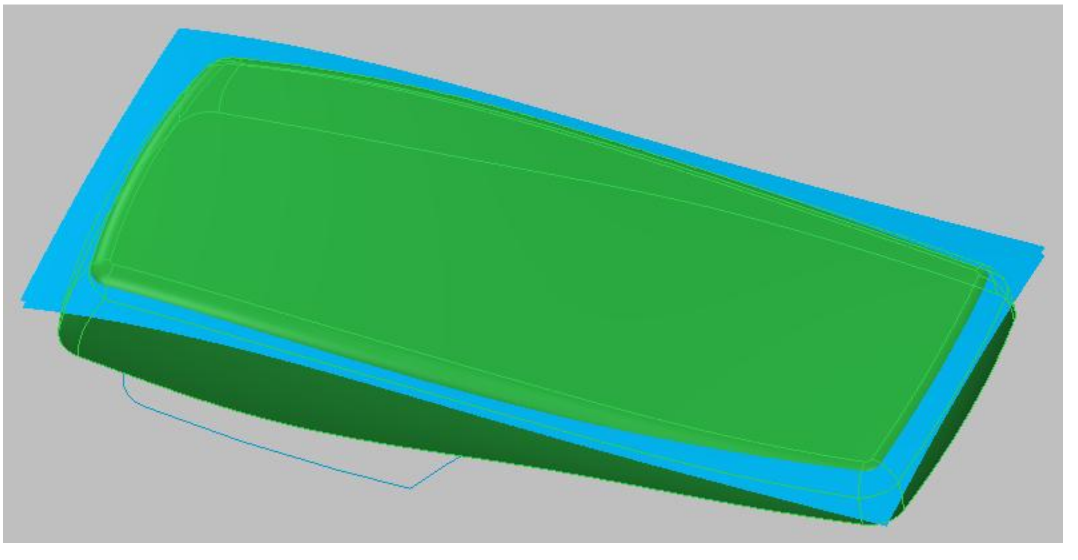
These copy geometry features refer the corresponding publish geometry features in the skeleton.

As we modified publish geometry features in the skeleton so we need regeneration. So regenerate the model by picking icon and model will appear as shown below.



Now we will create surface copy features.

Select the following quilt.



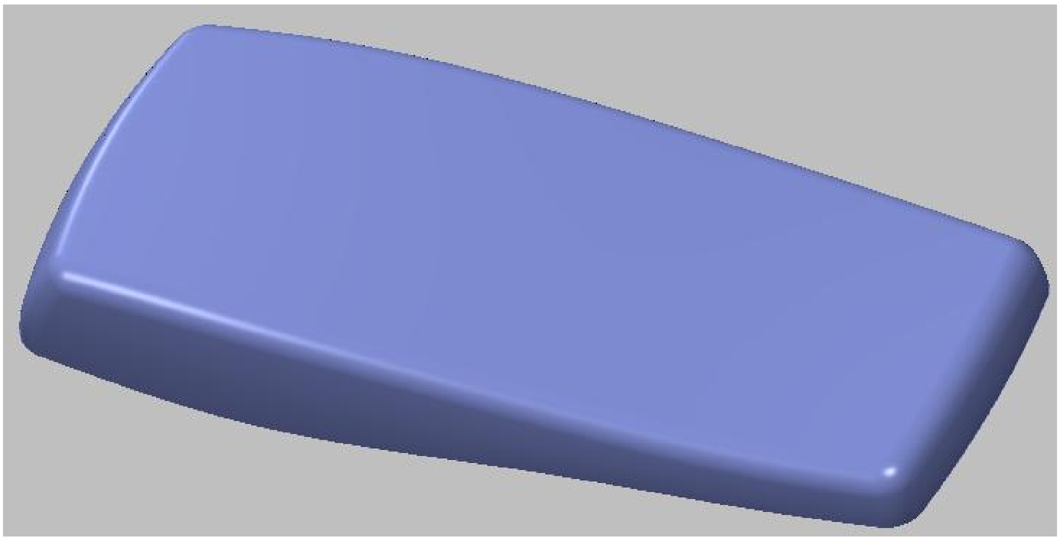
Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

The Copy dashboard will appear so pick to complete the feature.

Hide the 07___PRT_COPY_GEOM layer.

Now pick the copied quilt as shown below.



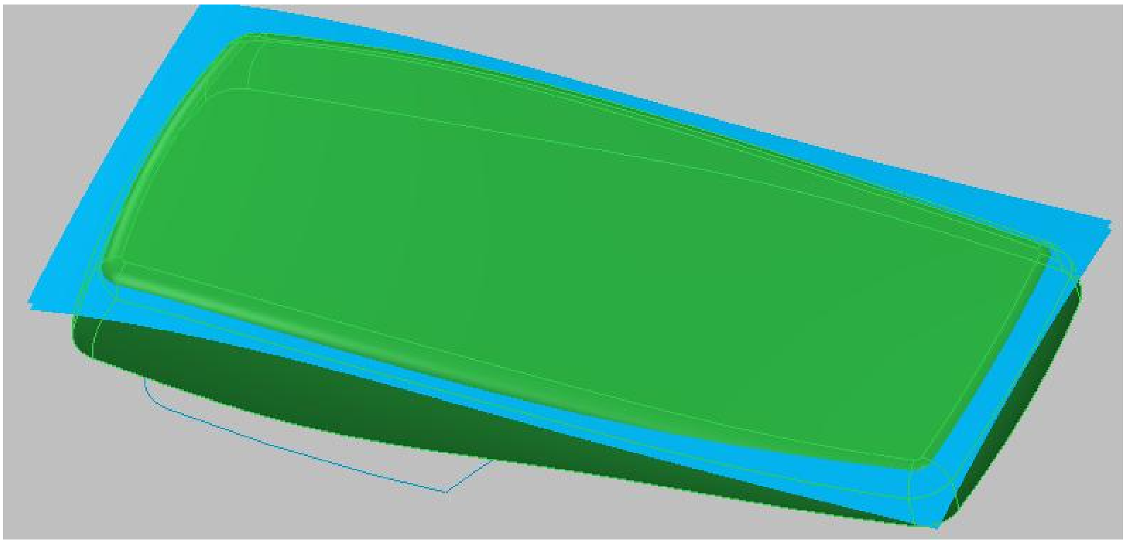
Pick Solidify to invoke Solidify tool.

Pick to complete the feature.

Now we will create the quilt that will be used to hollow the part.

Unhide the 07___PRT_COPY_GEOM layer.

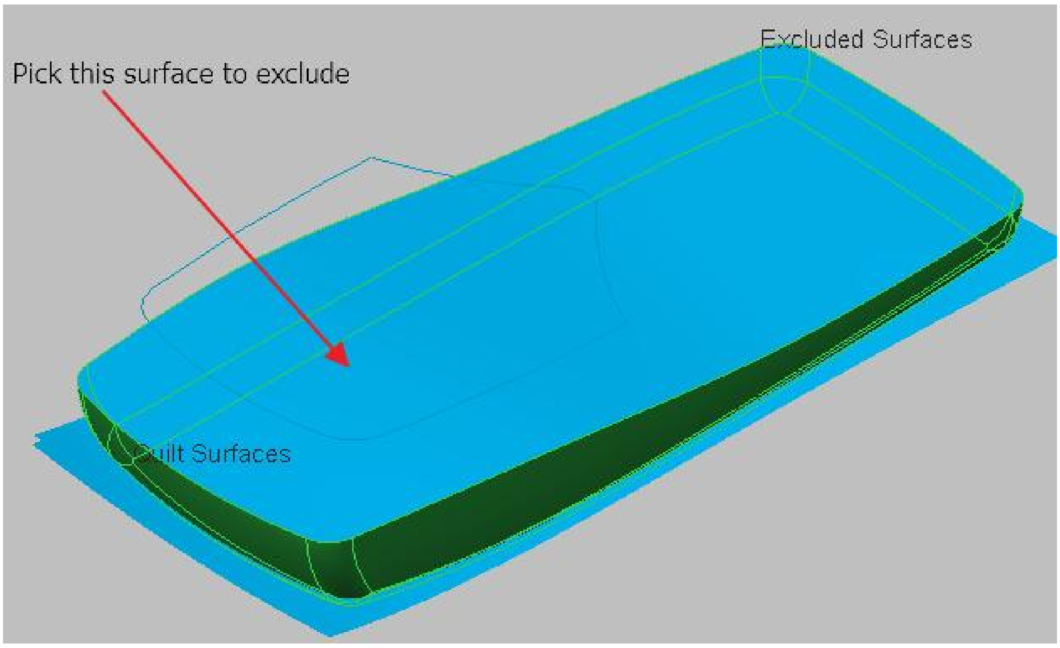
Select the following quilt belonging to Copy Geometry feature.



Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

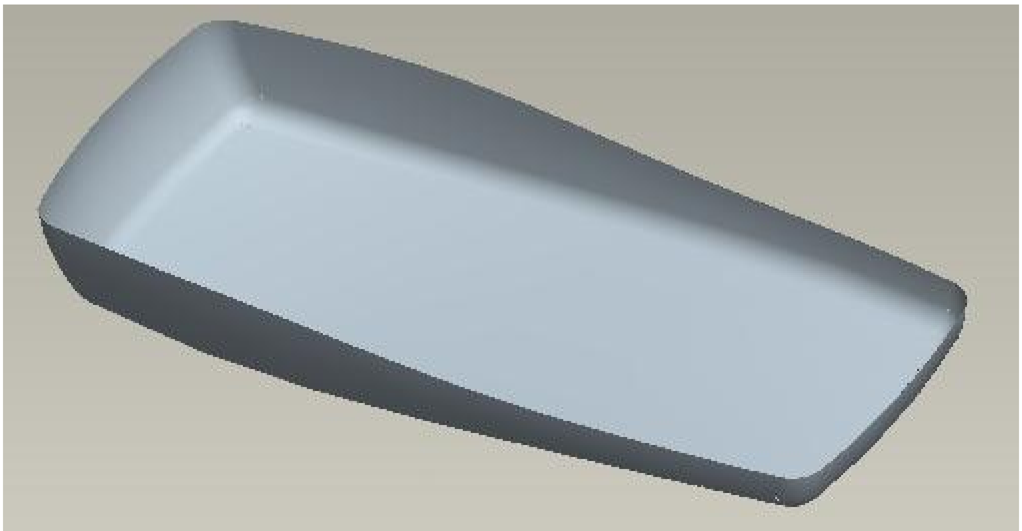
Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

Press Ctrl key and pick the following surface to exclude it form the selection. (Model is displayed with the preview mode off)



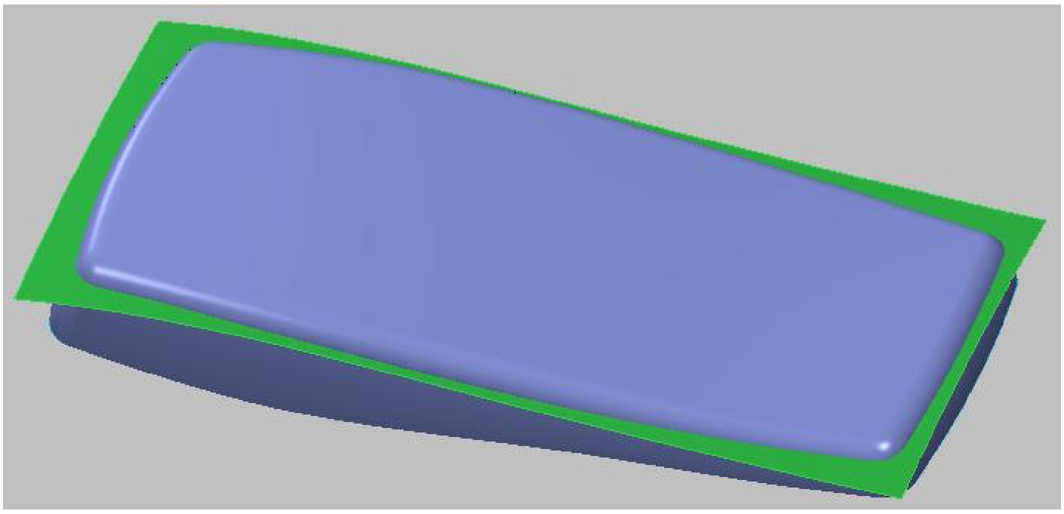
You should make sure that you pick the correct surface from the copy geometry quilt. If you pick any other surface, system will add it to the current selection. Notice the “Excluded Surfaces” text in the above figure.

Pick to complete the feature. If shown separately, the result of this copy surface feature will appear as shown below.



Select LCD_COVER_CG feature in the model tree and pick **Hide** ( ).

Now select the following surface (“Top Inner Surface”).



Pick copy icon () on the Model tab or press Ctrl+C on the keyboard.

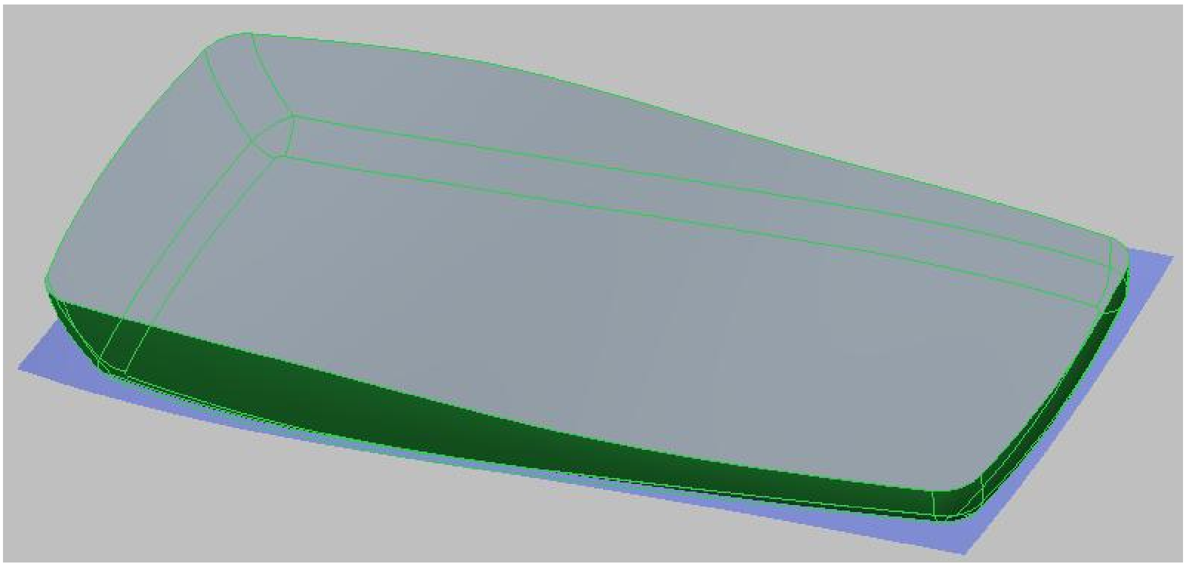
Select paste icon () on the Model tab or press Ctrl+V on the keyboard.

Pick  to complete the feature.

Right-click LCD_COVER_CG feature in the model tree and pick **Unhide**.

Now we will offset the outside quilt that we copied before. First hide the 07___PRT_COPY_GEOM layer.

So select the copied surface as shown below. (this is the surface for which we performed Exclude operation).

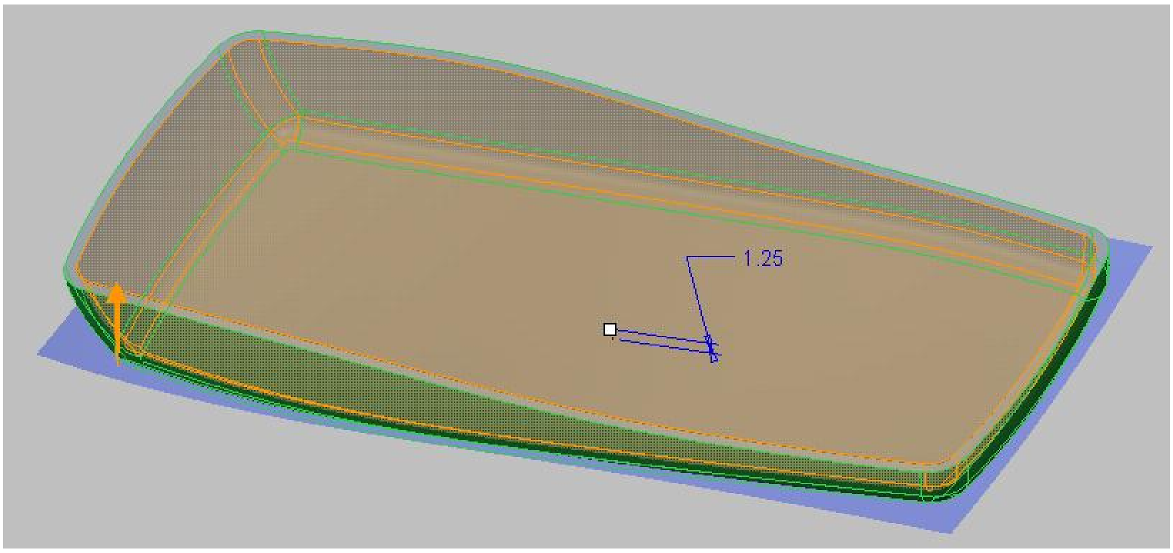


Pick 

to access the Offset tool.

Specify ***1.25*** as the offset value.

Pick  to reverse the direction of feature creation. The preview should appear as shown below.

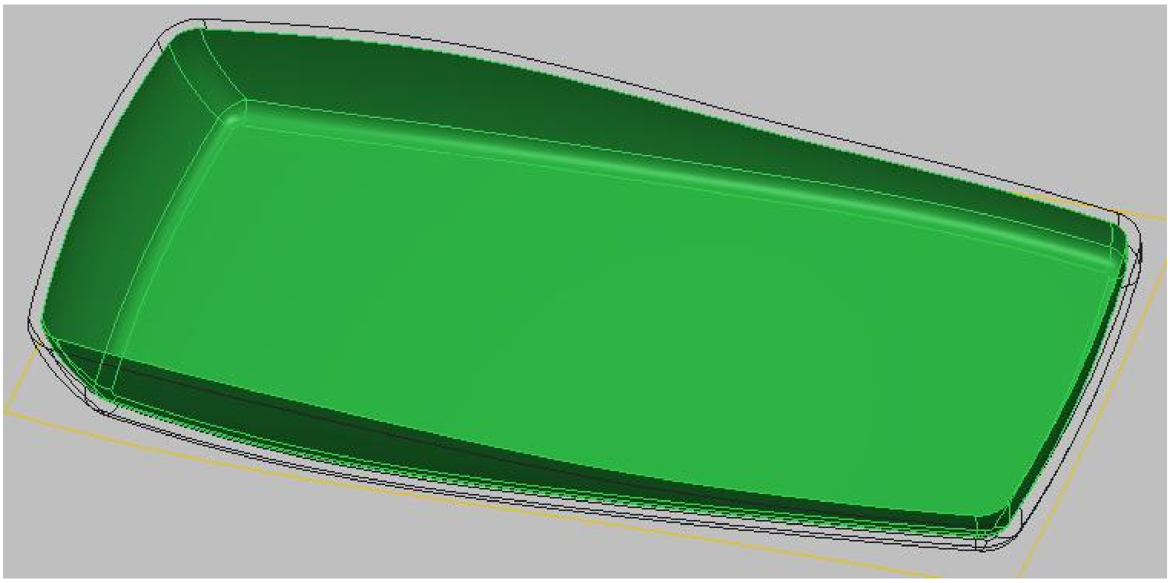


Pick icon to complete the feature.

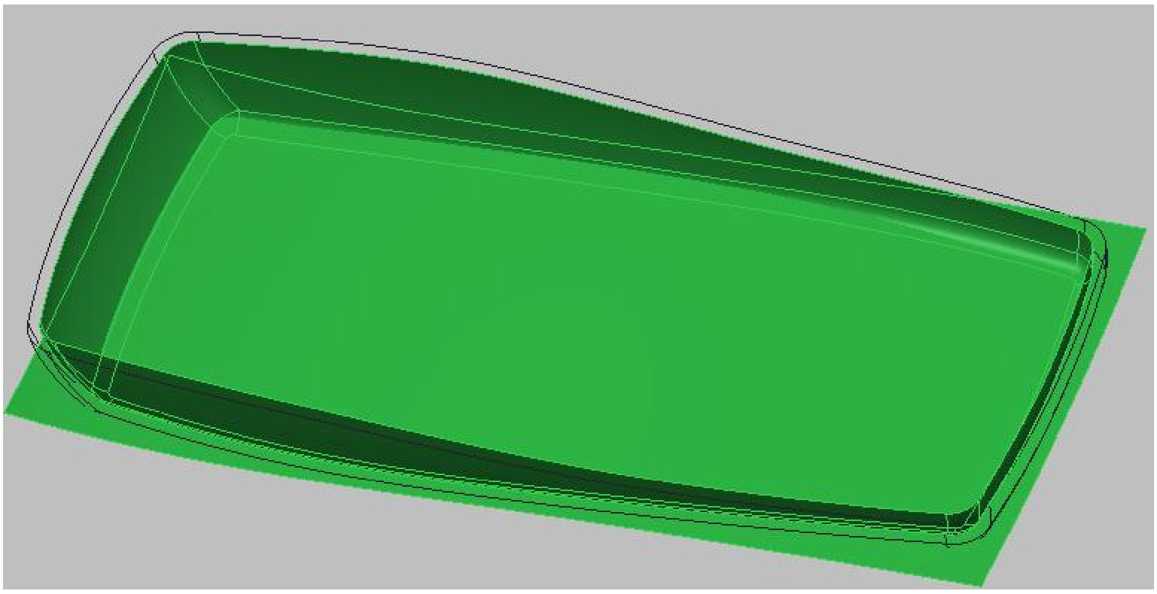
Now we will merge the offset quilt and “Top Inner Surface” together.

As we do not need the copied surface so right-click “Copy2” feature in the model tree and pick **Hide**.

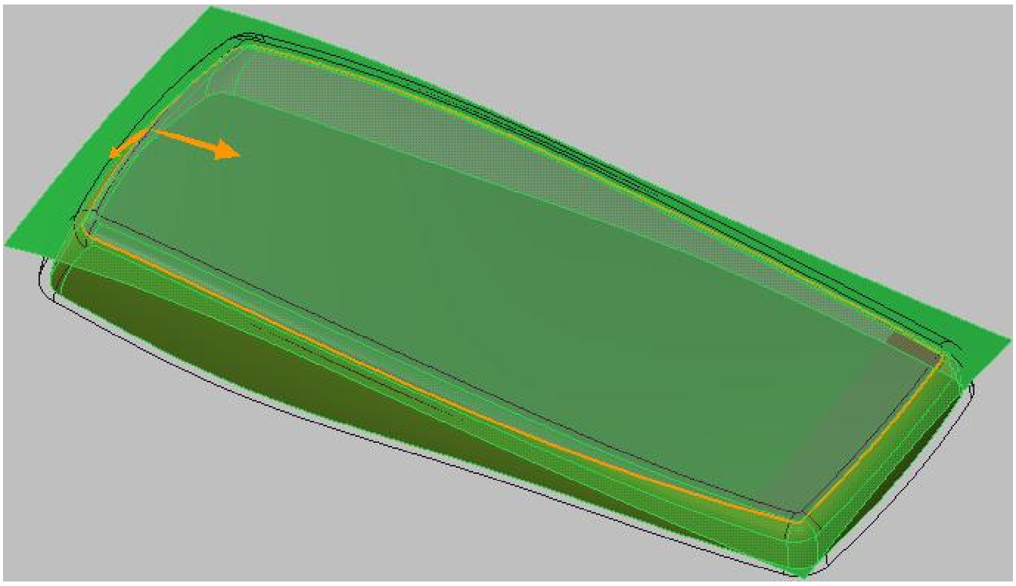
Select the offset quilt as shown below. (Note: Model is shown in wireframe mode.)

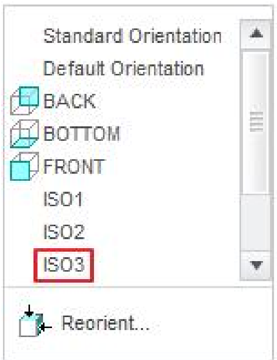


Hold down the Ctrl key and select the second quilt as shown below.



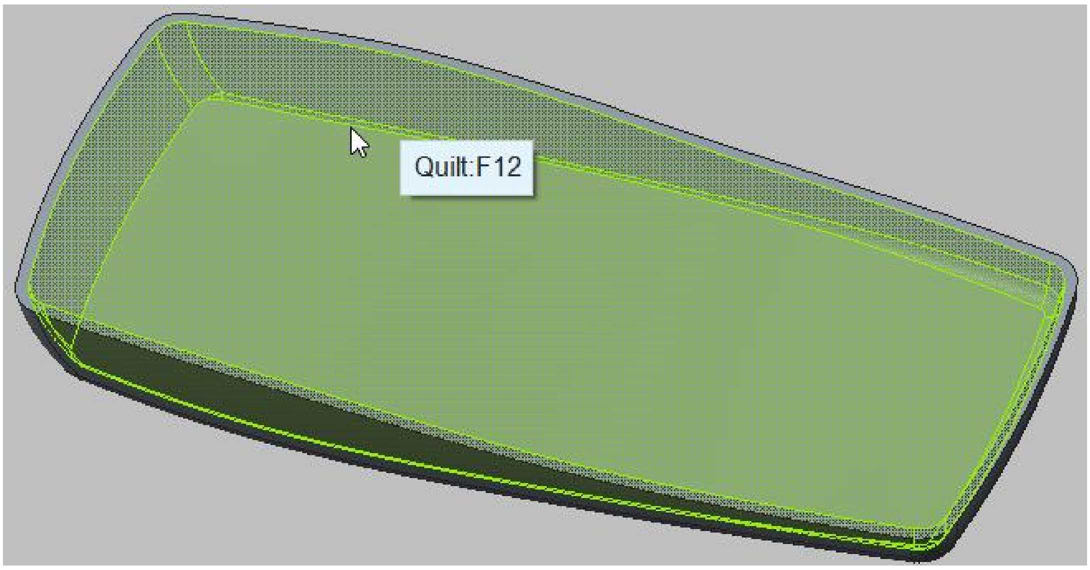
Pick Merge to invoke the Merge tool. Make sure that preview appears as shown below.





Pick to complete the feature.

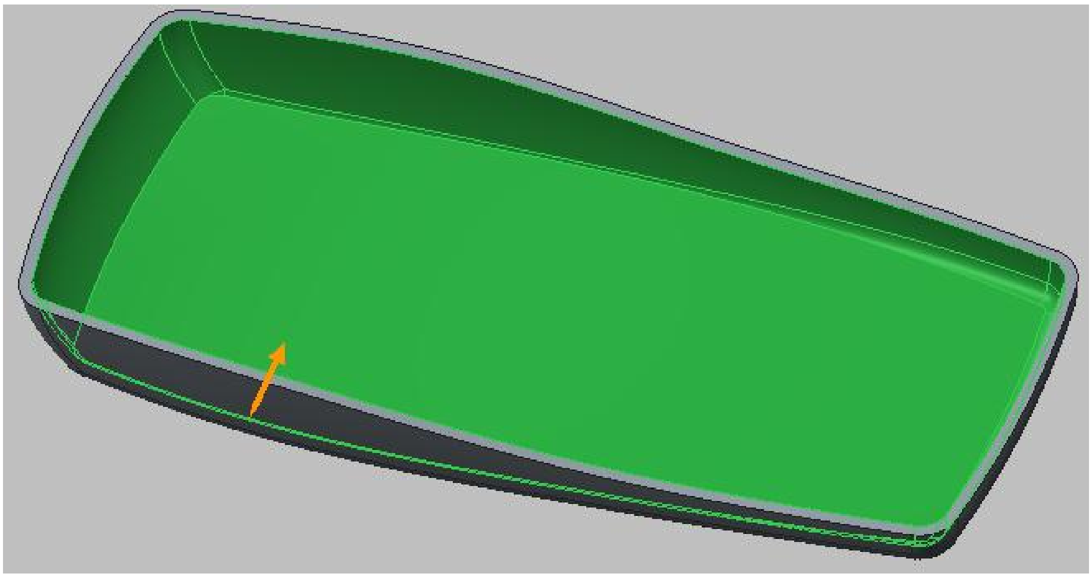
Now pick the merged quilt as shown below.



Pick Solidify to invoke Solidify tool.

Pick icon to create the feature as cut.

Pick to reverse the direction of feature creation.



Make sure that Direction arrow points towards inside. It will make sure that material is removed from the part.

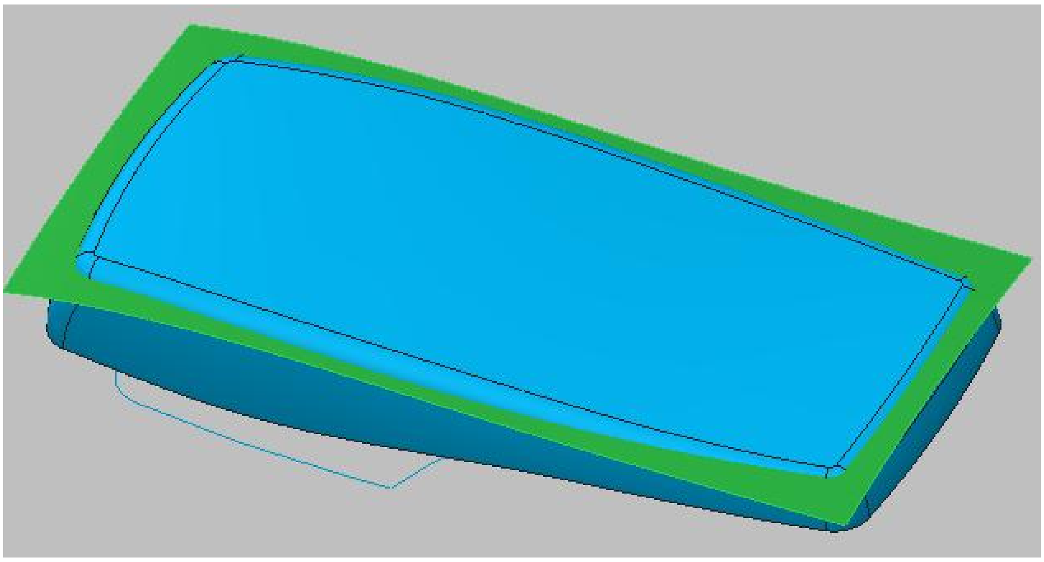
Pick to complete the feature.

Now we will create the cut for accommodating the LCD COVER.

First unhide the 07___PRT_COPY_GEOM layer.

Right-click TOP_INNER_CG feature in the model tree and pick **Hide**.

Now select the following surface.



Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

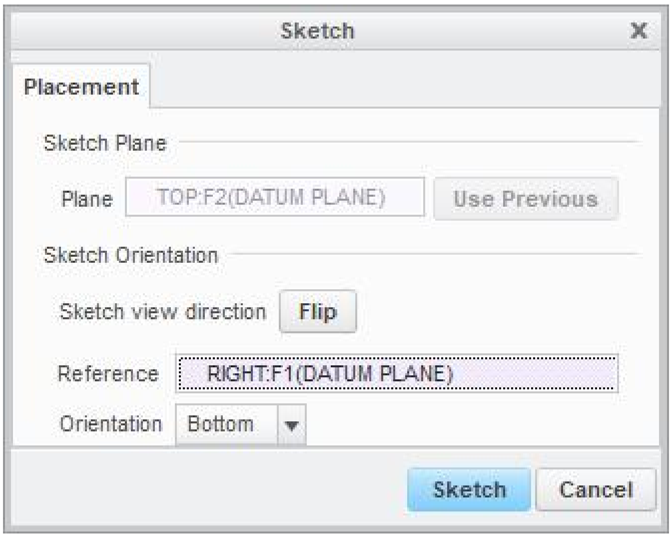
Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

Pick to complete the feature.

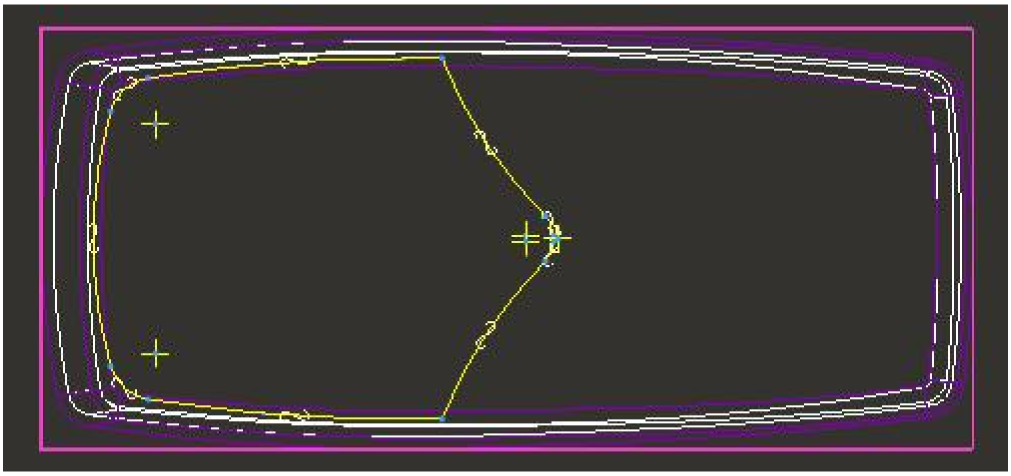
Now we will extrude a surface that will be merged with this copied surface.

Pick to invoke Extrude tool.

Select the sketching references as shown below



Pick Project icon and select the following loop.



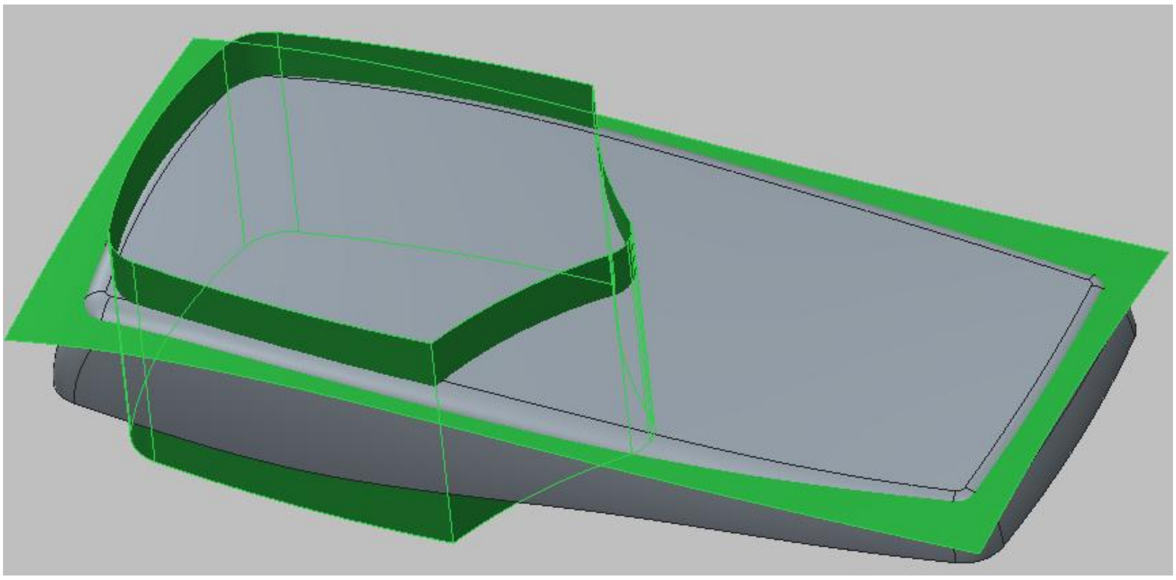
After completing the sketch, enter ***25*** as depth value.

Pick to create the feature as surface.

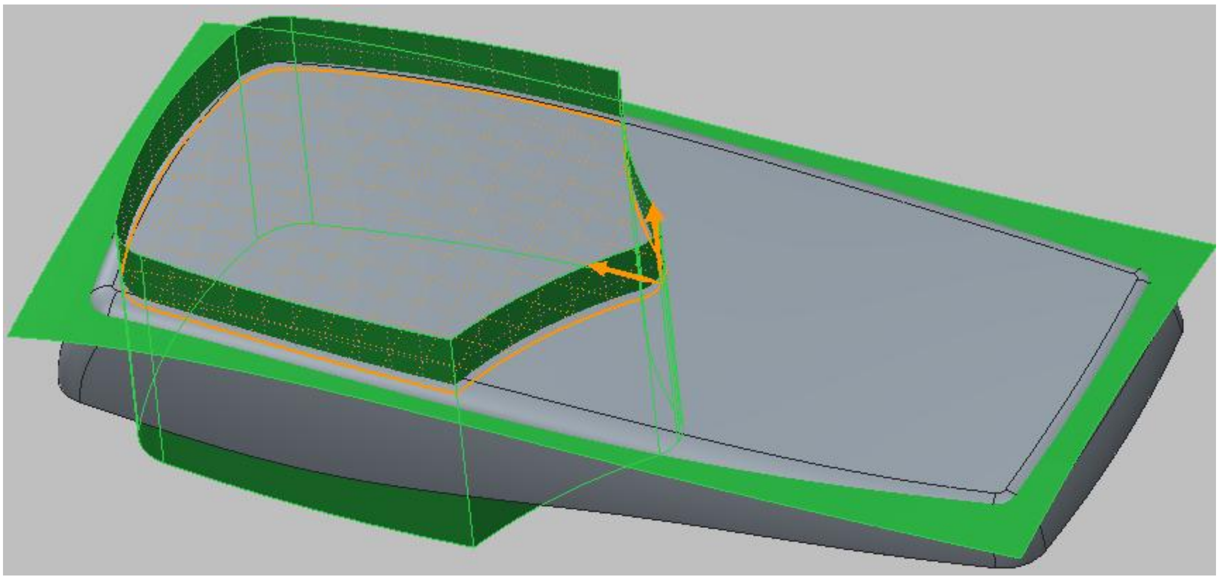
Pick icon or middle-click to complete the feature.

Hide the 07___PRT_COPY_GEOM layer.

Select the copied and extruded surface features (highlighted) while holding down the Ctrl key.



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



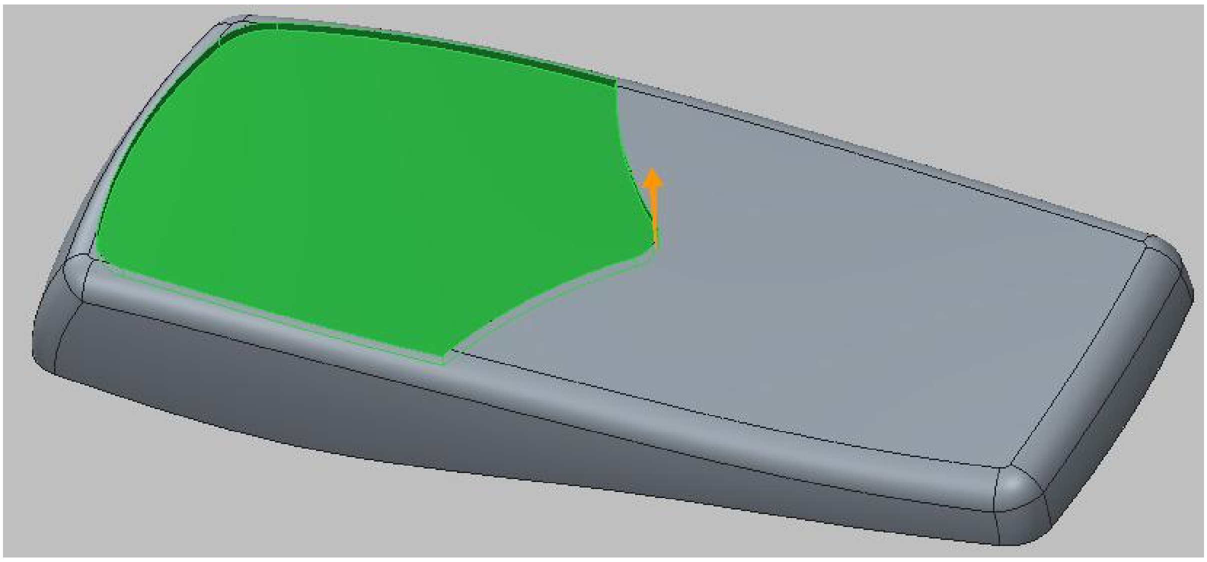
Pick  to complete the feature.

Now pick this merged quilt.

Pick 

to invoke Solidify tool.

Pick  icon to create the feature as cut and preview will appear as shown below.



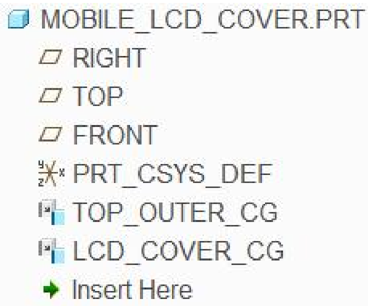
Pick  to complete the feature.

Now you can create the lip feature on your own by copying the INTERLOCK_PG publish geometry feature from the skeleton in this part.

Second Part

Now we will create the geometry for LCD COVER. So open the MOBILE_LCD_COVER.PRT in the new window.

Notice that it has already two copy geometry features as shown below.

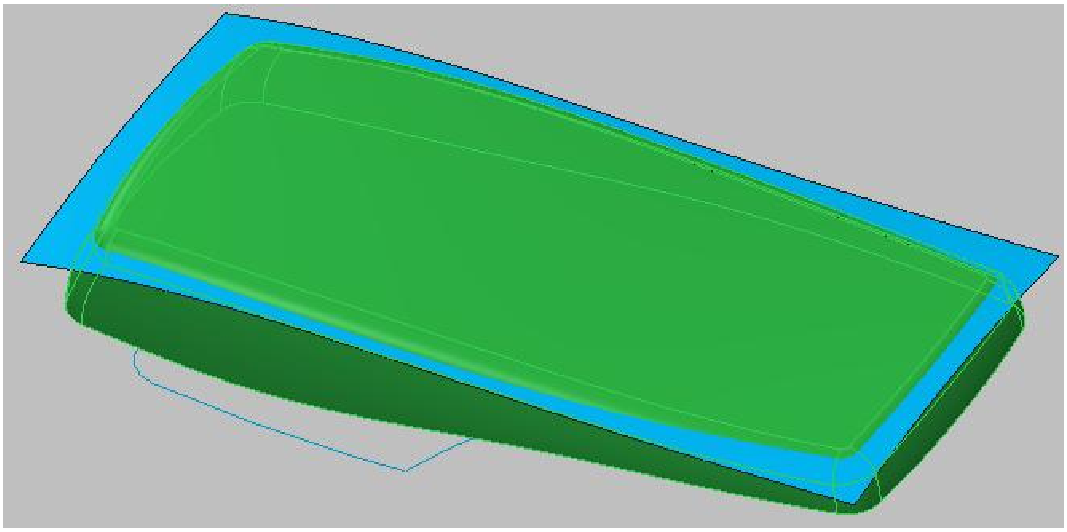


These copy geometry features refer the corresponding publish geometry features in the skeleton.

As we modified publish geometry features in the skeleton so we need regeneration. So regenerate the model by picking icon.

Now we will create surface copy features.

Select the following quilt.

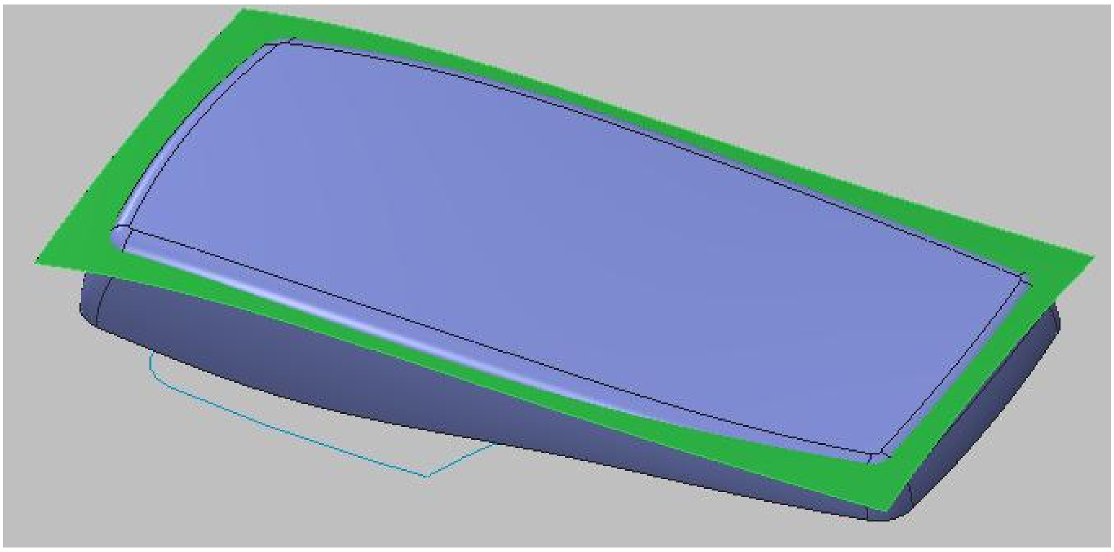


Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

The Copy dashboard will appear so pick to complete the feature.

Now select the following surface.



Pick copy icon ( ) on the Model tab or press Ctrl+C on the keyboard.

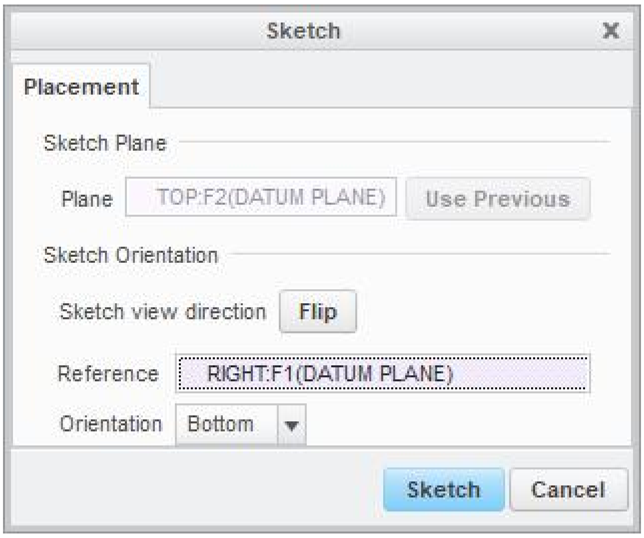
Select paste icon ( ) on the Model tab or press Ctrl+V on the keyboard.

Pick to complete the feature.

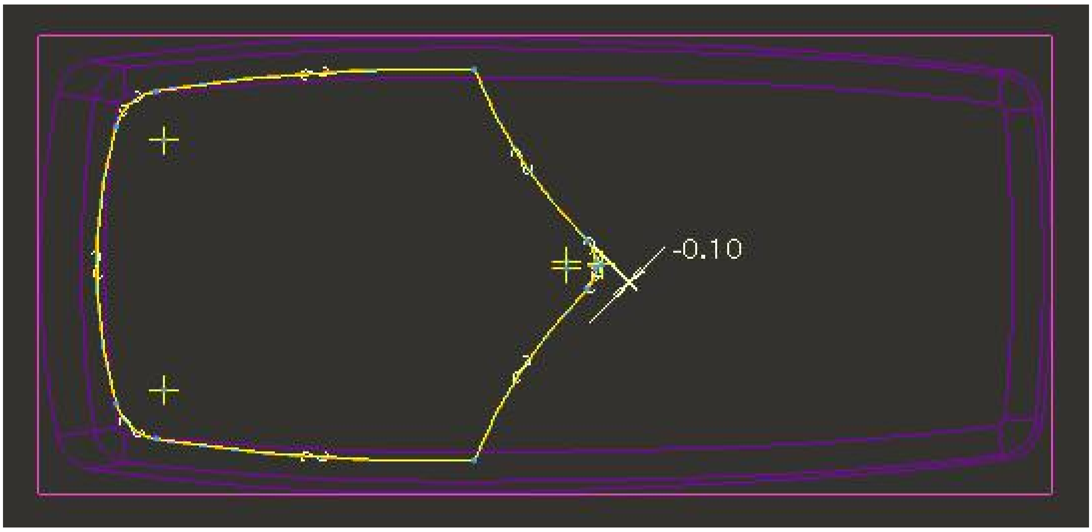
Now we will extrude a surface that will be merged with this copied surface.

Pick to invoke Extrude tool.

Select the sketching references as shown below



Pick Offset icon and create a loop that is offset by ***-0.1*** from the profile curve as shown below.



After completing the sketch, enter ***25*** as depth value.

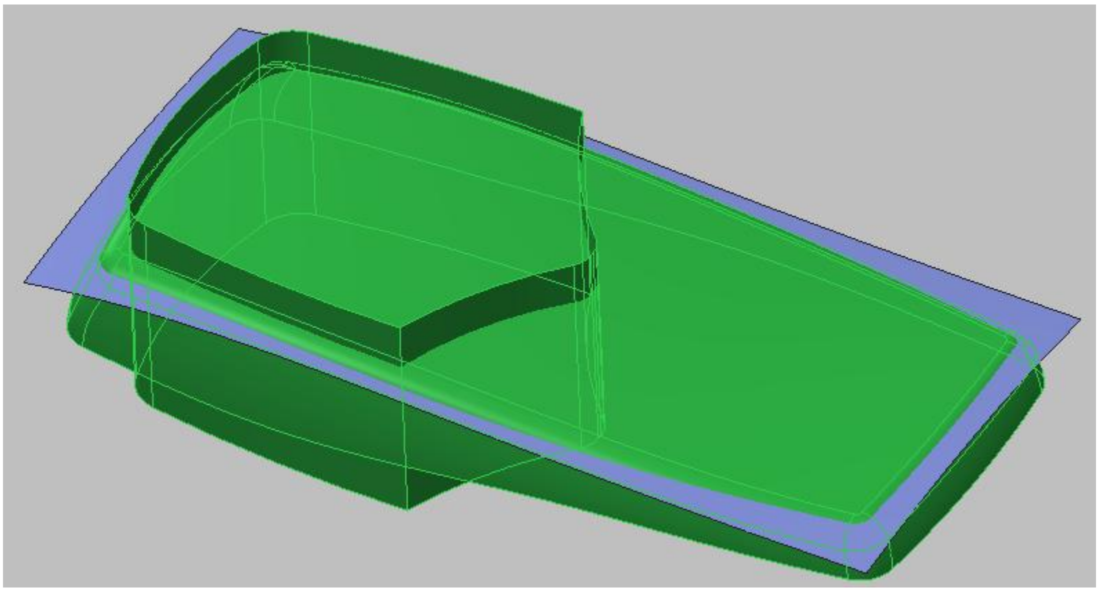
Pick to create the feature as surface.

Pick icon or middle-click to complete the feature.

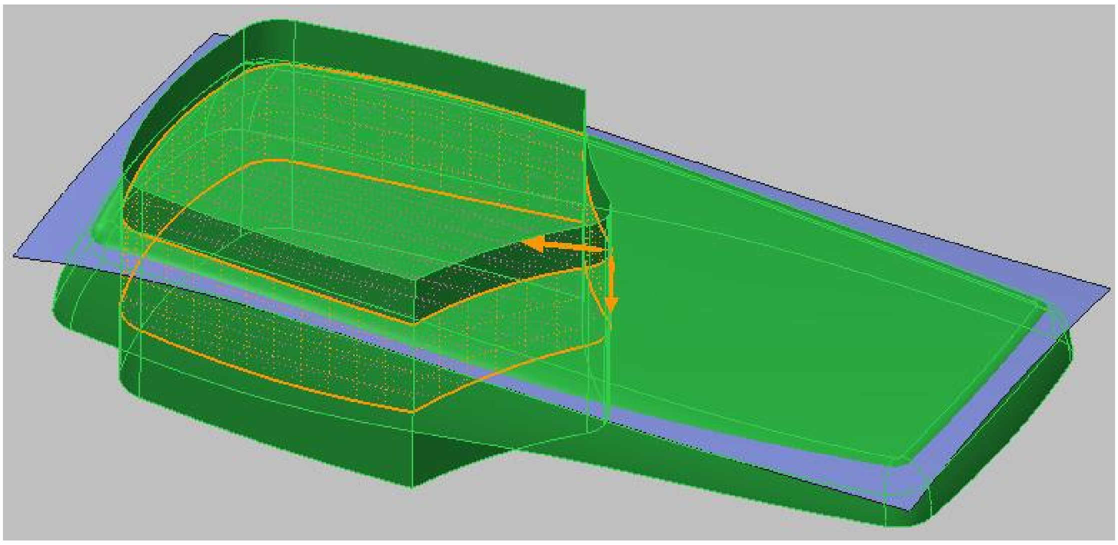
Now we will merge these surfaces together and then create a solid protrusion out of merged quilt.

Hide the 07___PRT_COPY_GEOM layer.

Select the copied and extruded surface features (highlighted) while holding down the Ctrl key.

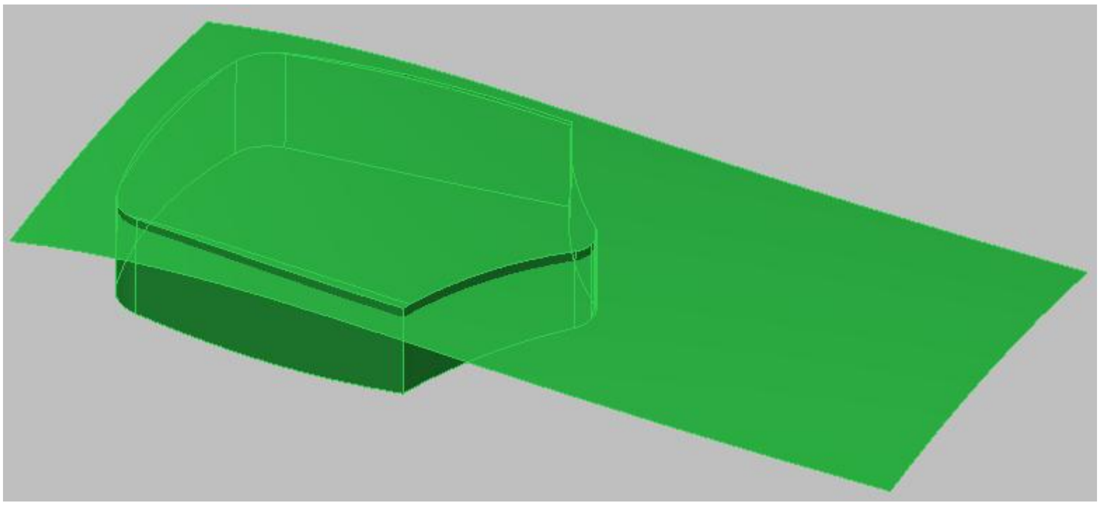


Pick to invoke the Merge tool. Make sure that preview appears as shown below.



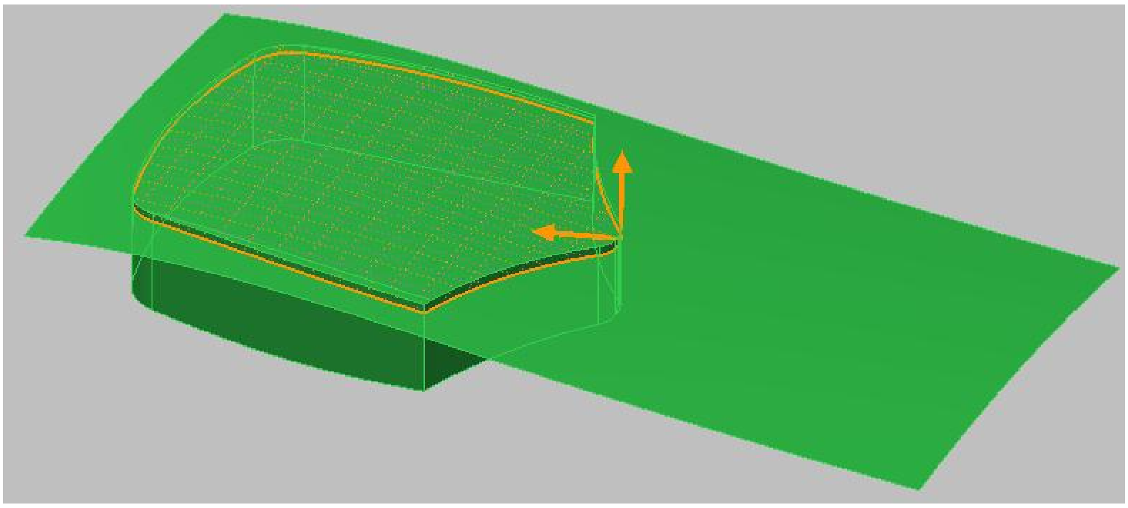
Pick  to complete the feature.

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



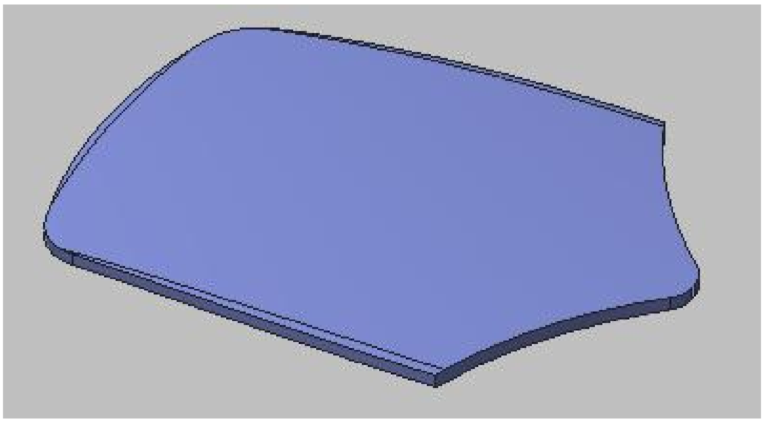
Pick 

to invoke the Merge tool and make sure preview appears as shown below.



Pick ✓ to complete the feature.

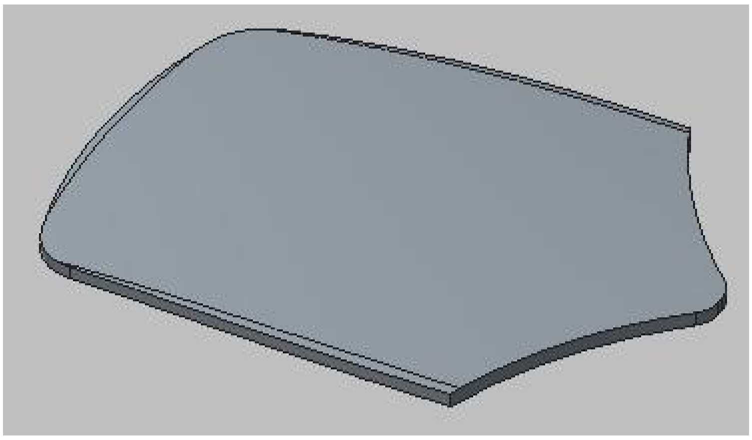
Now pick the merged quilt as shown below.



Pick Solidify to invoke Solidify tool.

Pick ✓ to complete the feature.

The part will appear as shown below.



It completes the basic shape of the part.