

Swept Blend

Swept Blend feature combines the power of Sweep and Blend features into one. A swept blend feature is created by blending a series of (at least two) sections along a specified trajectory (Origin Trajectory). So we can say it is “trajectory guided blend”.

The origin trajectory can be a datum curve or edges of a surface or solid.

The sections to be blended are sketched at specified segment vertices or datum points on the Origin Trajectory. We can sketch the sections either within the Swept Blend feature or create as datum curve before defining the swept blend.

Section Plane Control

There are following three options to control the orientation of the section plane along the trajectory.

1. Normal To Trajectory
2. Normal To Trajectory
3. Pivot Direction

Normal To Trajectory

Section Plane stays normal to the selected trajectory. The Z axis is tangent at all points to the trajectory.

Normal To Projection

The Normal to Projection option is used to control the section plane orientation normal to the projection of the origin along a specified direction reference. This is particularly useful when defining geometry that requires the user to maintain a constant section or draft angle relative to the directional reference.

Constant Normal Direction

Section plane normal vector is parallel to the direction vector of the selected reference.

Horizontal / Vertical Control

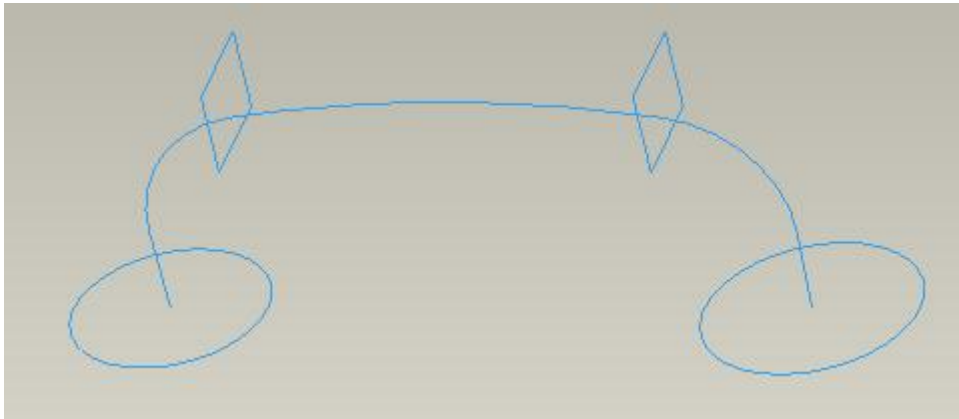
Horizontal / Vertical Control determines how the section plane is oriented in terms of its rotation about the Z axis.

Exercise 1

In this exercise we will learn how to create a swept blend surface.

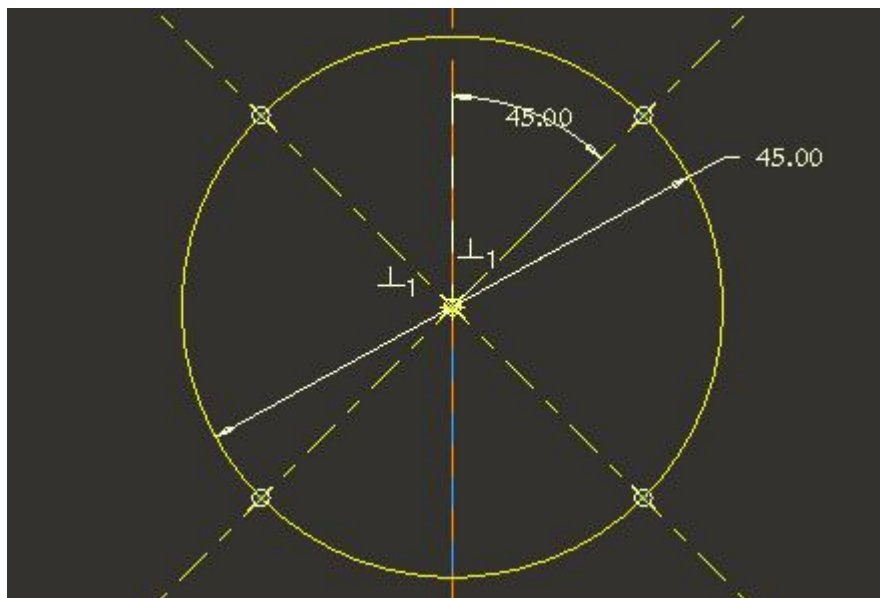
Set the working directory to the BASICS folder and open the model SWEPT_B1.PRT


The model consists of five datum curves. The open curve will be used as Origin Trajectory and other four as sub-sections.

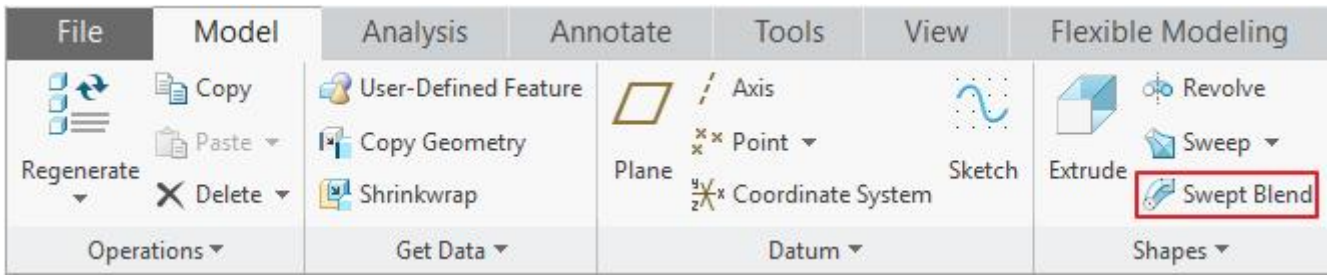


Notice that rectangular sections consist of four entities. So we need to split the circular sections at the end of trajectory into four segments.


If you redefine the end sections, you will notice that they have already been divided into four segments. The sketch for the circle is shown below.

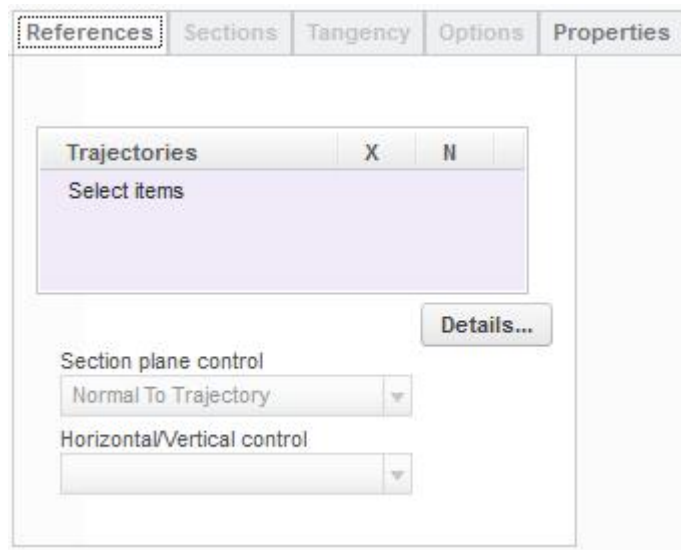


Pick  , located on the Model tab, to invoke Swept blend tool.



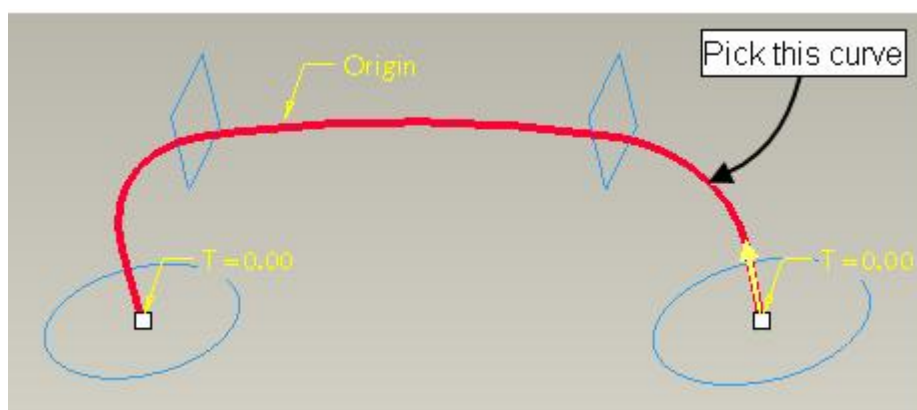
Pick Create as surface icon () if not selected by default.

Pick  tab to open the References slide-up panel.



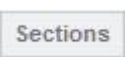
Notice that Trajectories collector is active by default.

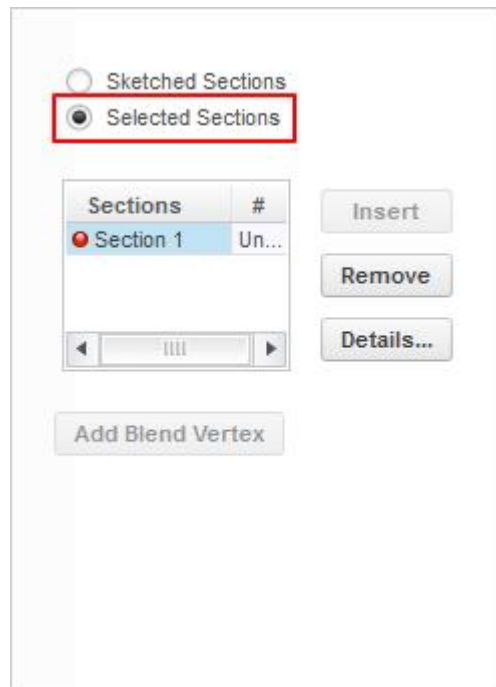
So pick the open datum curve with left mouse pick and trajectory will appear as shown below.



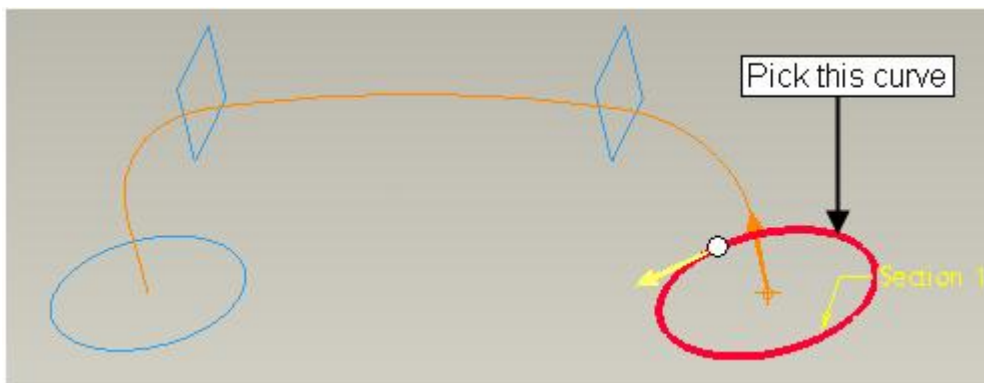
The system has designated the trajectory as Origin trajectory as shown in the figure above.

You can see some other options in the panel but we will not discuss them now to concentrate only on the basic concepts.

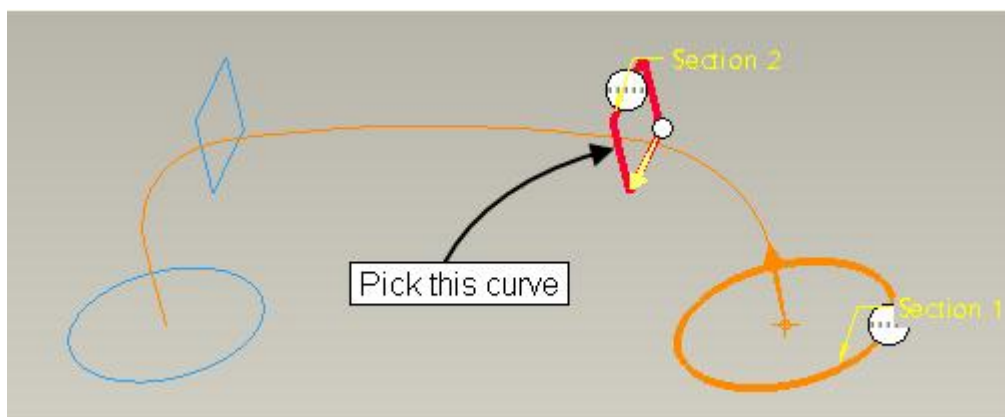
Pick  tab to open Sections slide-up panel and change the section option to **Selected Sections**.



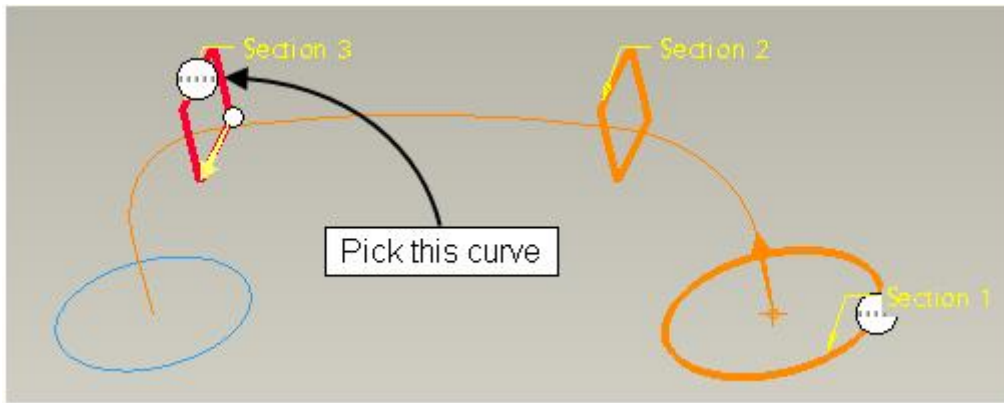
Now select the following section



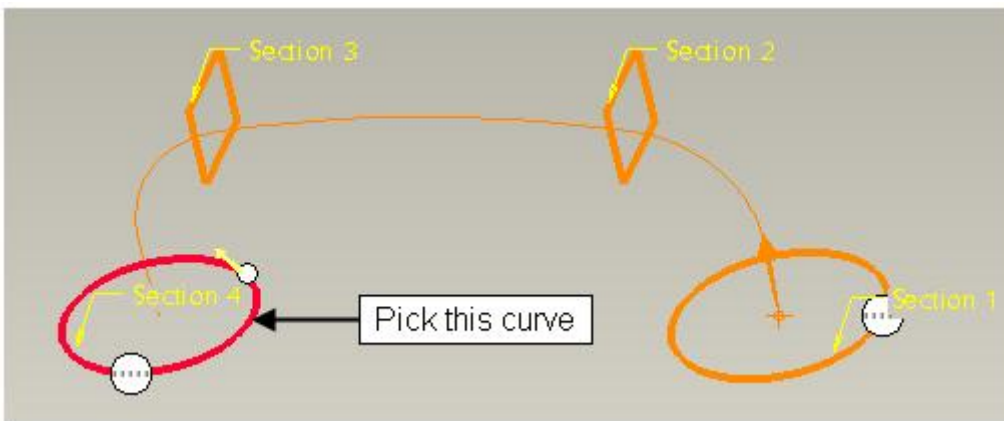
Pick **Insert** tab in the Sections slide-up panel and select the second section.



Again pick **Insert** and select the third section.

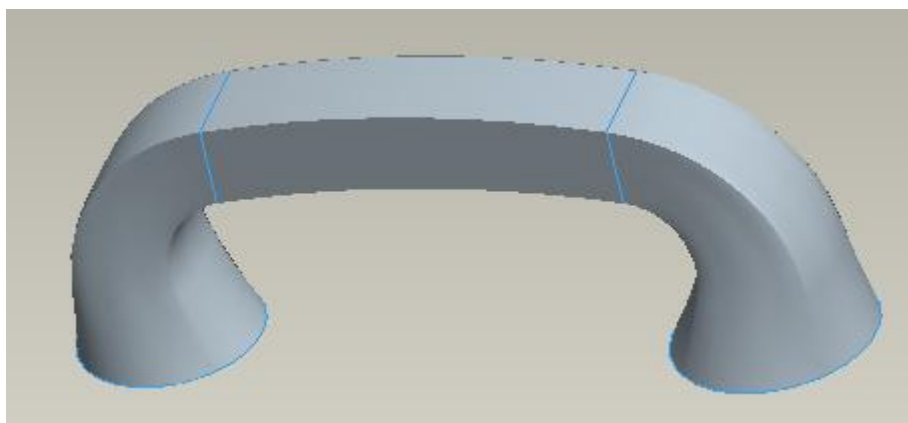


Again pick and select the fourth section.



Pick icon or middle-click to complete the feature.

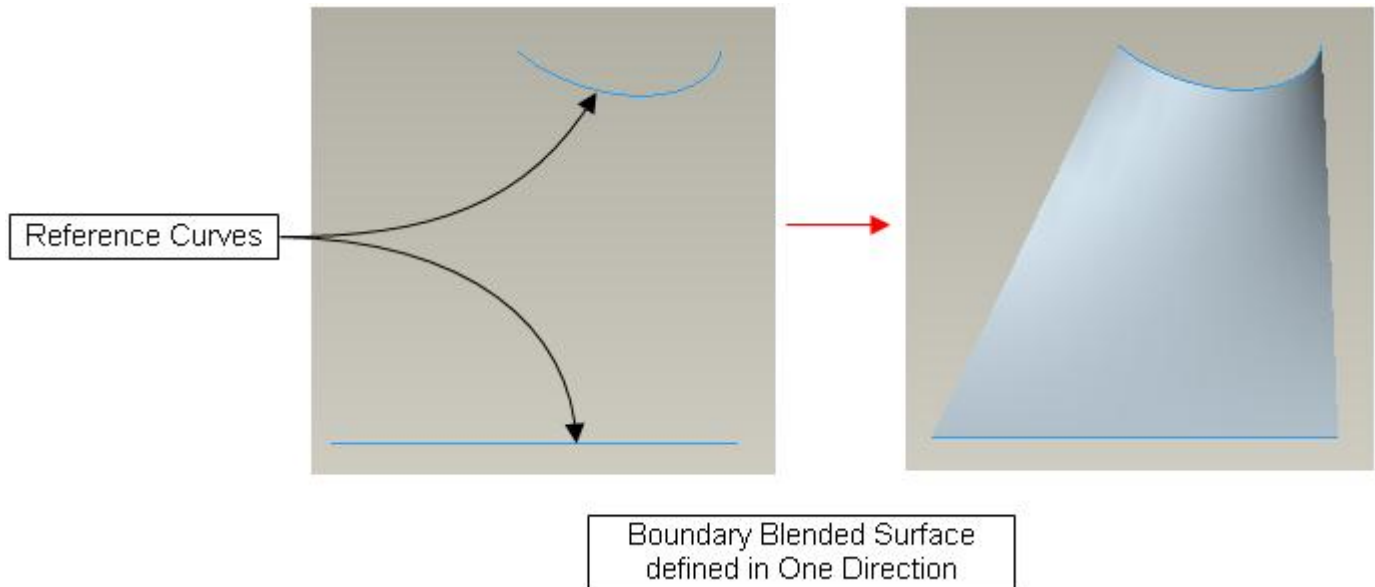
The part will appear as shown below.



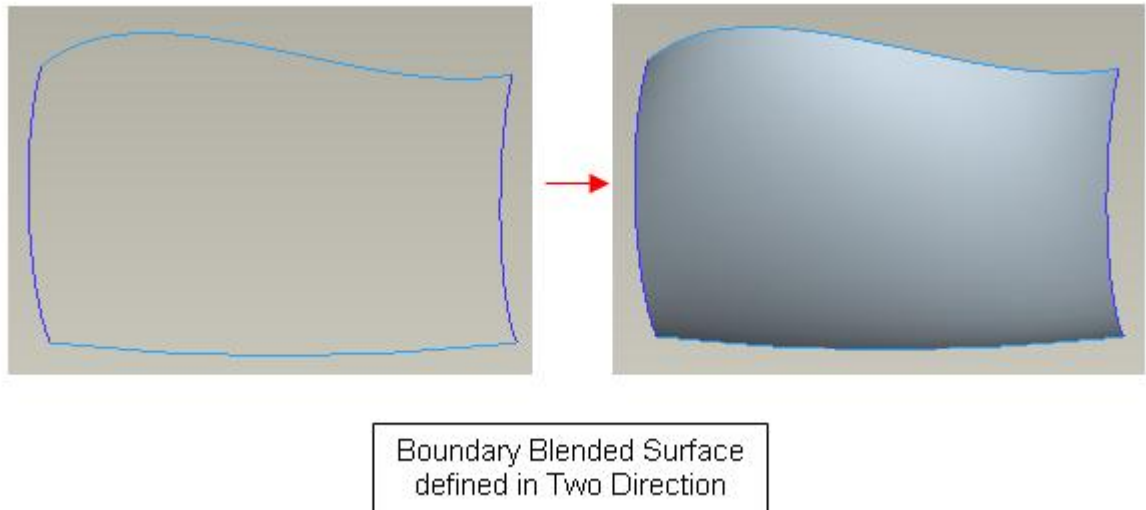
Select **File > Save** to save the work done so far.

Boundary Blend

With the Boundary Blend tool, you can create a surface that is blended between reference curves or edges. Blended surfaces can be defined either in one or two direction. In the figure below, a surface is shown blended between two datum curves in one direction.



In the figure below, a surface is shown blended in two directions.



For blended surfaces defined in two directions, the outer boundaries must form a closed loop. This means that the outer boundaries must intersect. If the boundaries do not terminate at the intersection points, Creo Parametric automatically trims them and uses the relevant portion.

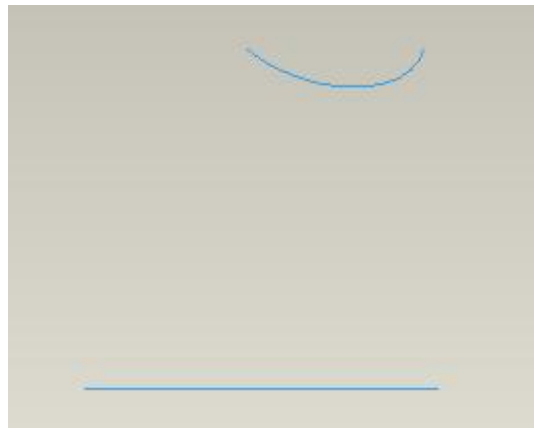
In each direction, reference entities must be selected in consecutive order. However, reference entities can be reordered.


Exercise 1

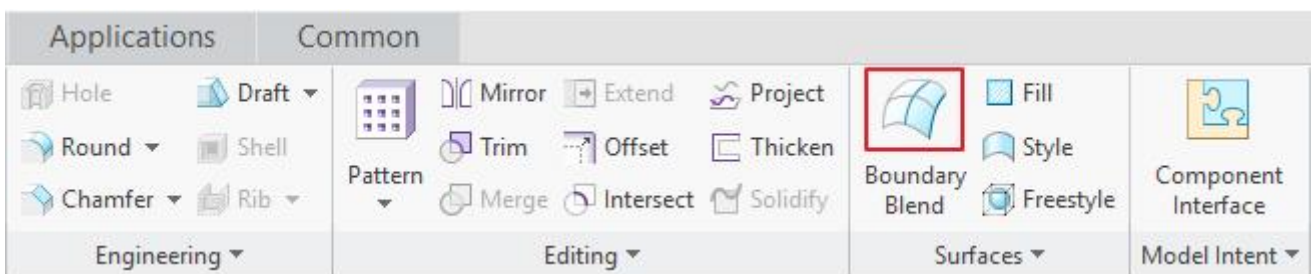
In this exercise we will learn how to create a blended surface by referencing curves in one direction.

Set the working directory to the BASICS folder and open the model BB1.PRT

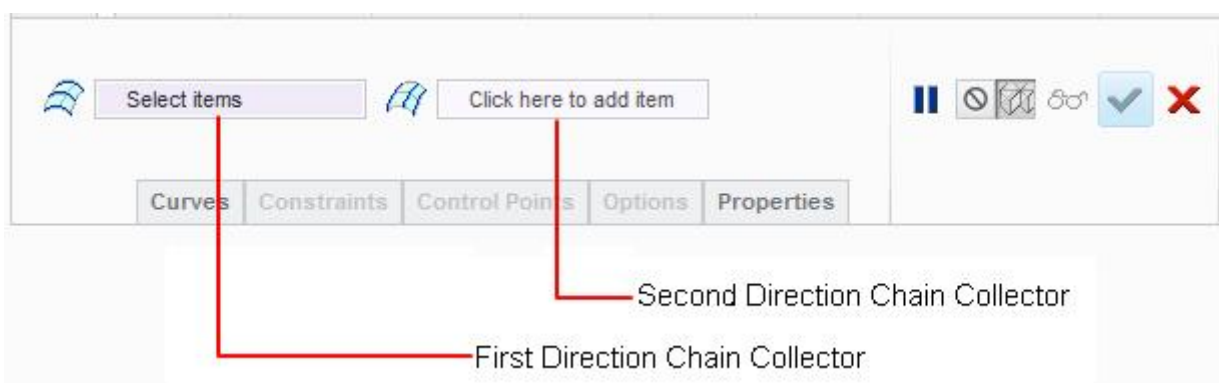
The model consists of two sketched datum curve that will be used create the blended surface



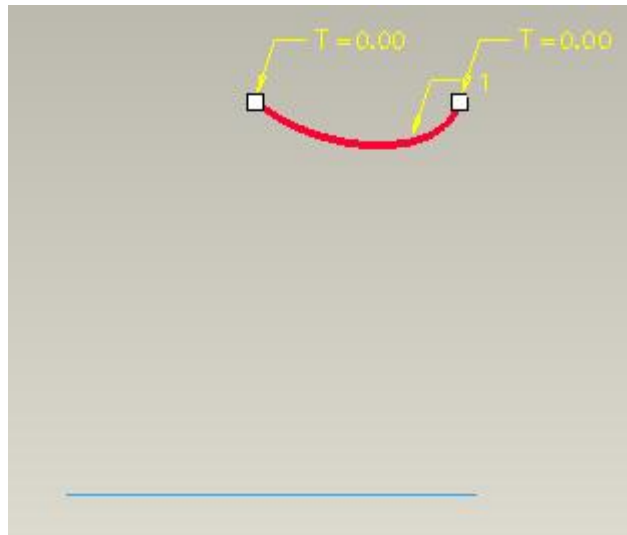
Pick  to invoke boundary blend tool.



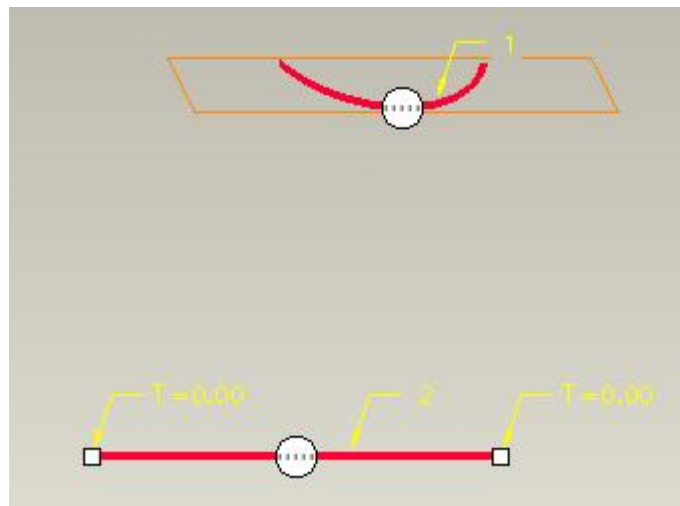
The dashboard will appear as shown below.



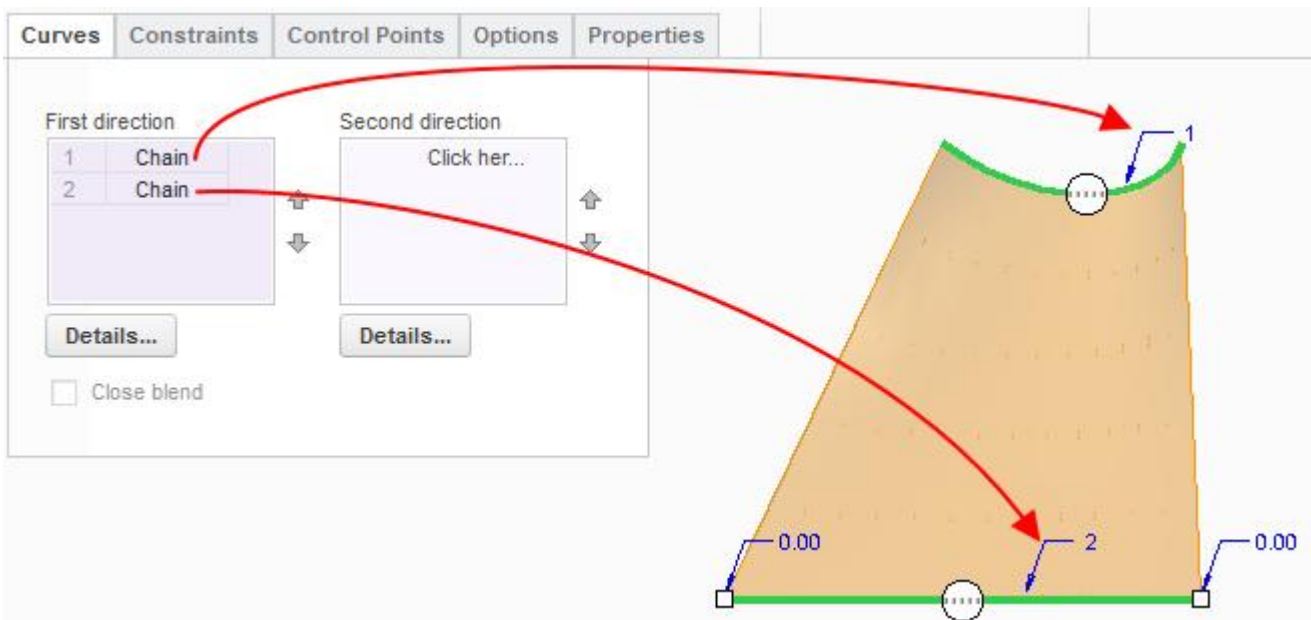
Notice that First Direction Chain collector is active by default. So pick the datum curve with left mouse pick and screen will appear as shown below.




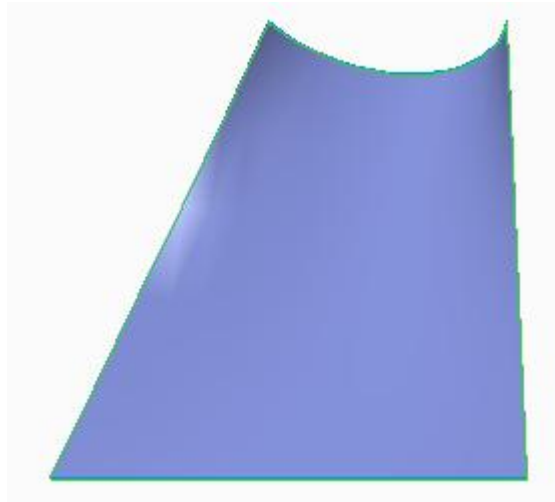
Now hold down the Ctrl key and pick the second curve as shown below (with dynamic preview off).



If you look in the Curves slide-up panel you will notice that there are two chains listed in the First Direction collector. These chains are numbered and corresponding numbers are also displayed on screen as shown below.



Pick  to complete the feature and component will display as shown below.



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


Exercise 2

In this exercise we will learn how to create a blended surface by referencing curves in two directions.

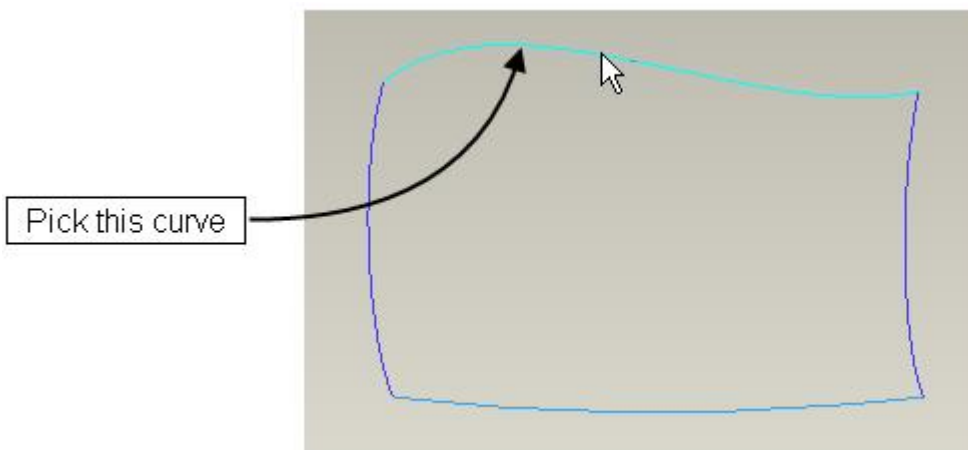
Set the working directory to the BASICS folder and open the model BB2.PRT

The model consists of four datum curve that will be used to create the blended surface

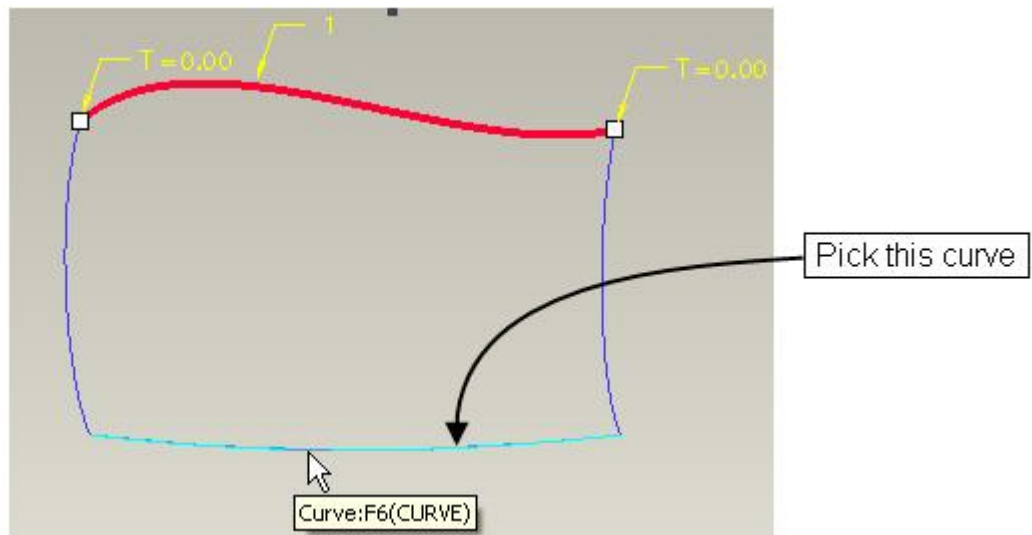


Pick  to invoke boundary blend tool.

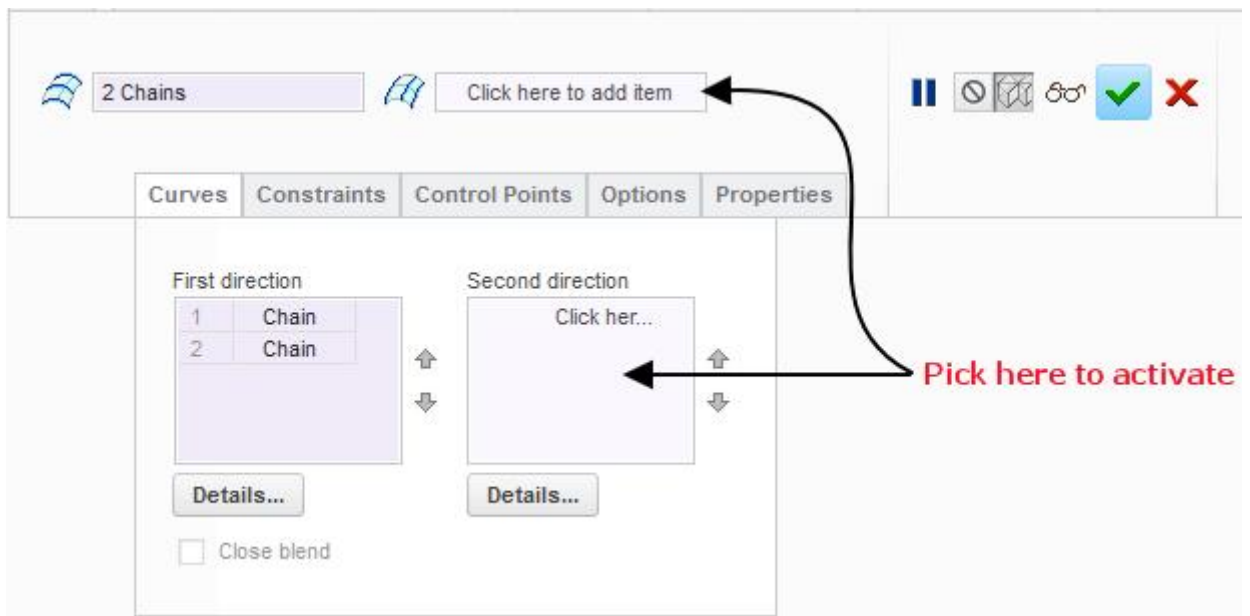
Notice that First Direction Chain collector is active by default. So pick the following datum curve with left mouse button.



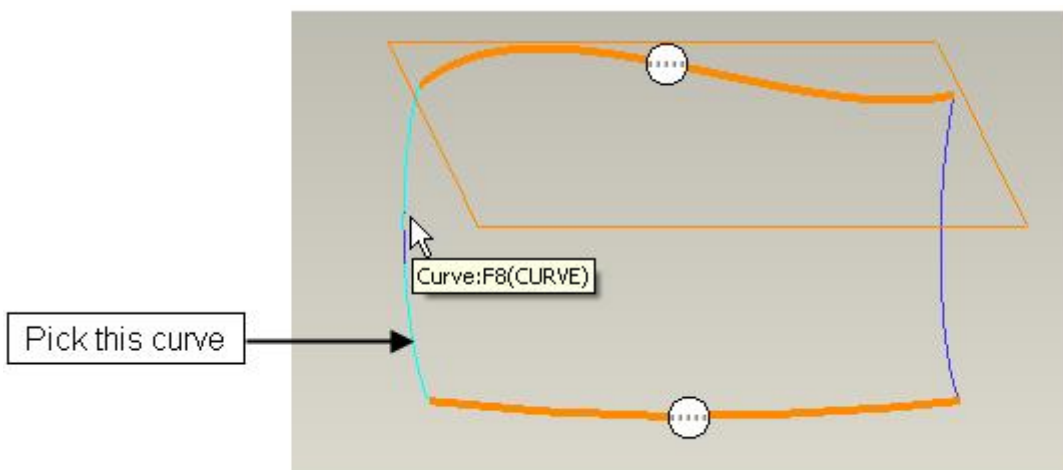
Now hold down the Ctrl key and pick the second curve as shown below.



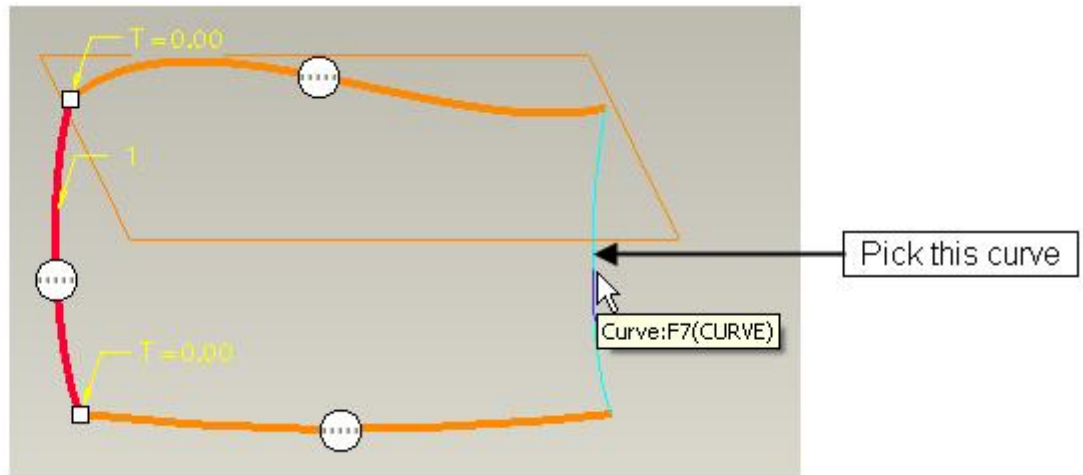
Pick **Curves** and look in the Curves slide-up panel. You will notice that there are two chains listed in the First Direction collector. To add curves for second direction, pick in the “Second direction” collector, either in the Curves panel or dashboard, to activate it as shown below.




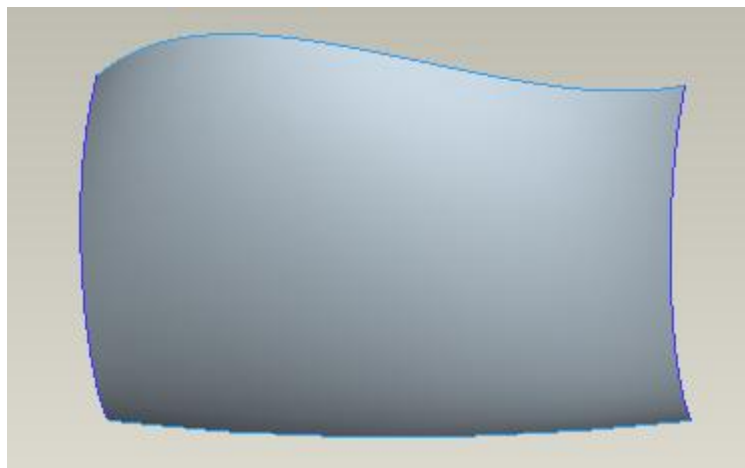
Pick the following datum curve with left mouse button as shown below (with dynamic preview off).



Now hold down the Ctrl key and pick the second curve as shown below (with dynamic preview off).



Pick  to complete the feature and surface will display as shown below.



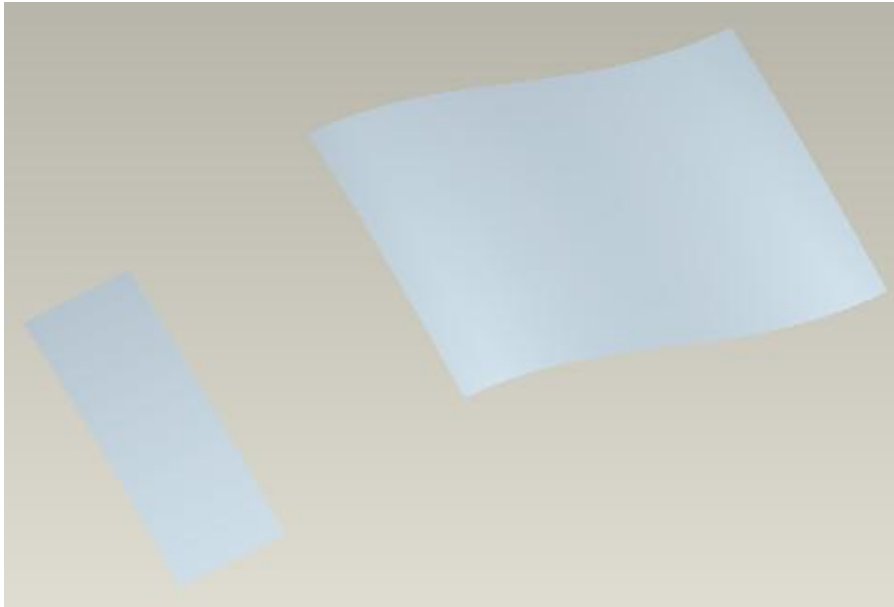
Select **File > Save** to save the work done so far.

Exercise 3


In this exercise we will learn how to create a blended surface that connects two existing surface. We will also see how to make the surface tangent to adjacent surface.

Set the working directory to the BASICS folder and open the model BB3.PRT

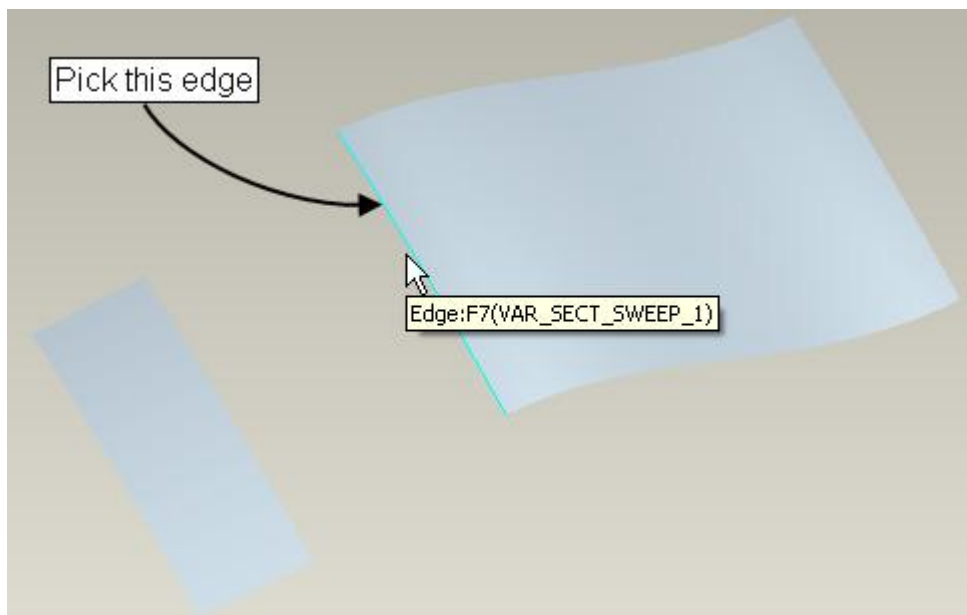
The model consists of two surfaces as shown below.



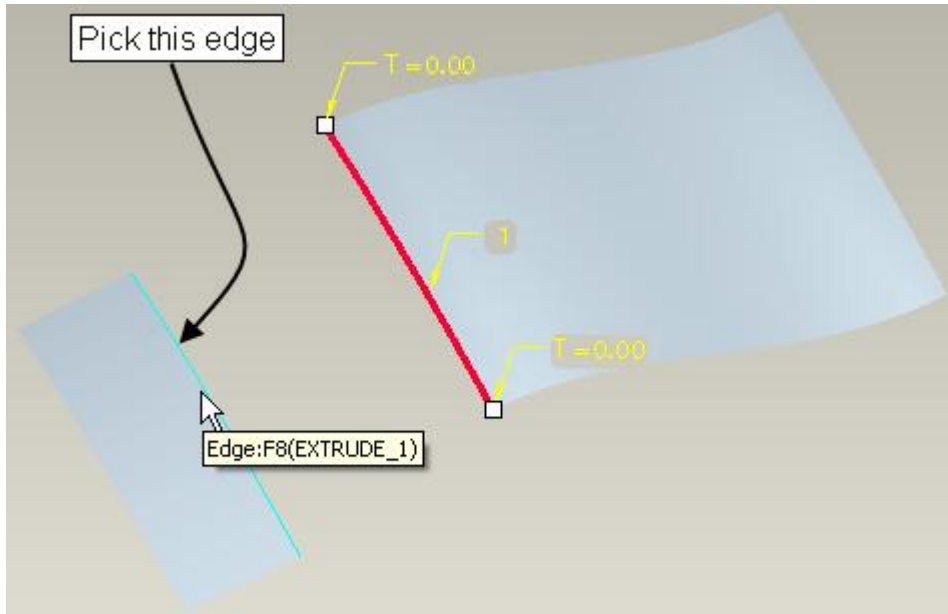
We will create a surface that connects the two surfaces shown in the above figure.

Pick  to invoke boundary blend tool.

Notice that First Direction Chain collector is active by default. So pick the following edge with left mouse button.



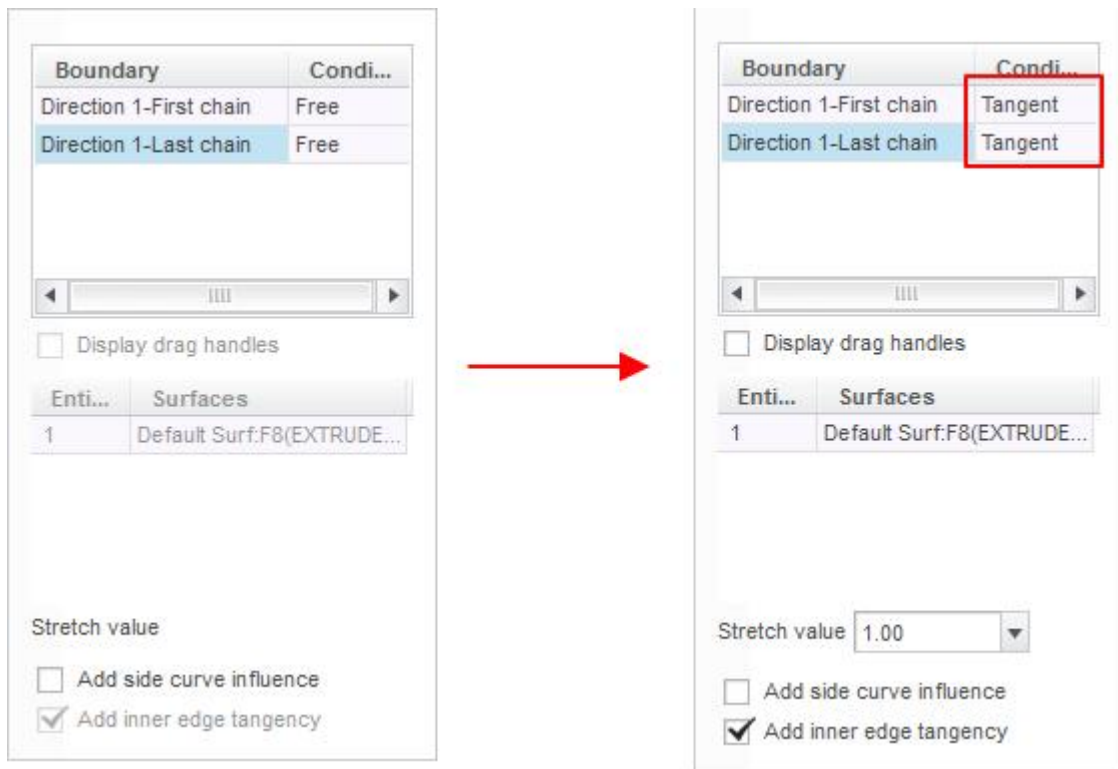
Now hold down the Ctrl key and pick the second edge as shown below.



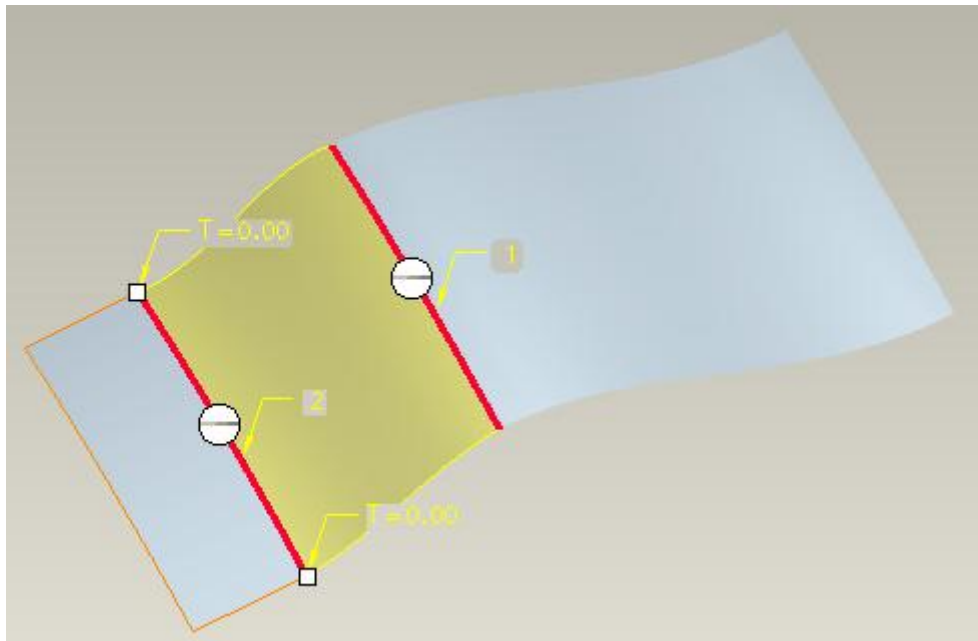
Now we will add constraints to the boundaries of this surface. So pick **Constraints** tab.




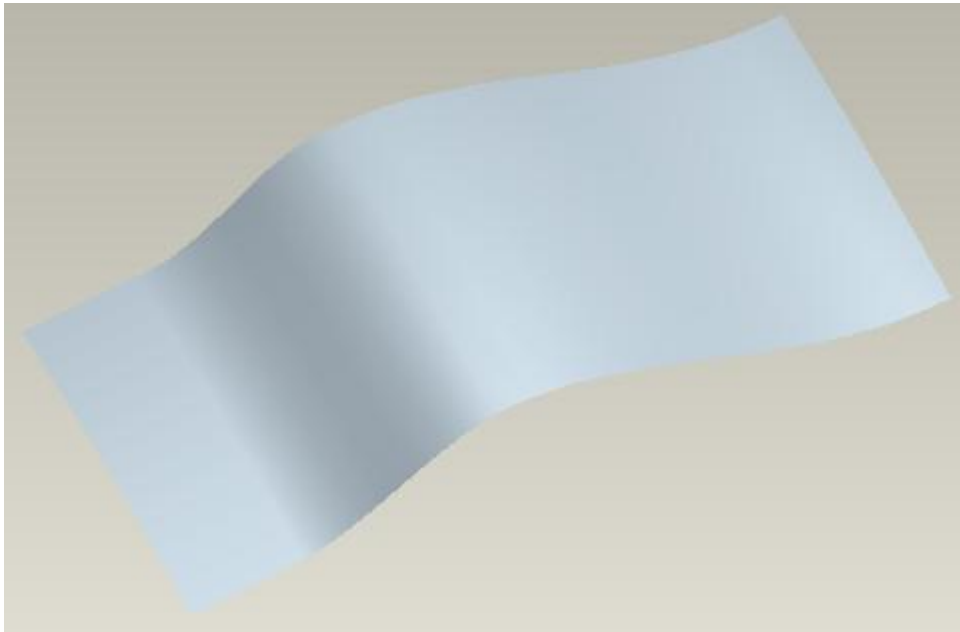
Change the constraint type from Free to **Tangent** for both chains as shown below.



Now the preview will appear as shown below.



Pick  to complete the feature and surface will display as shown below.




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

Offset

Offset tool is used to build a variety of geometry. It can be used to offset either a surface or a curve with a constant or variable distance.

It can create new geometry or manipulate existing surfaces or solids to create the desired shape.

Offset tool is used both in surface and solid modeling. But here we will confine our discussion only to surface modeling.

Offset Tool  is used to create offset feature.

You can create the following types of Offset features by using the Offset tool:

Standard—Offsets a single quilt, surface, or solid face.

Expand—Creates a continuous volume between the selected surface and offset surface. You can also constrain the offset area of an open quilt or solid surface with a sketch.

With Draft—Offsets the region of the quilt or surface that is included inside a sketch, and drafts the side surfaces. You can also create straight or tangent side surface profiles with this option.

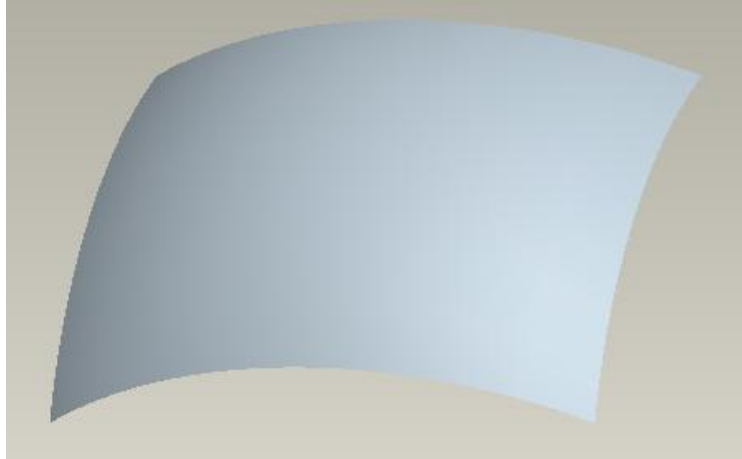
Replace—Replaces a solid face with a quilt or datum plane. The Replace function differs from other features as it can add material in some places and remove it in others.

Exercise 1

In this exercise we will learn to create an offset surface by using the Standard offset type.

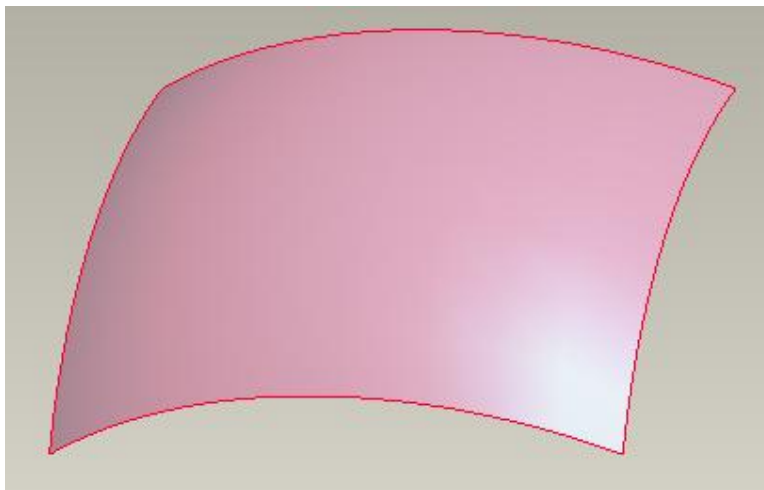
Set working directory to the BASICS folder and open the model OFF1.PRT

The model will appear as shown below.



By default Offset Tool is inactive and becomes active when we make any valid selection (Object-Action behavior of Creo).

So first pick the surface as shown below.



Pick  to invoke Offset tool.



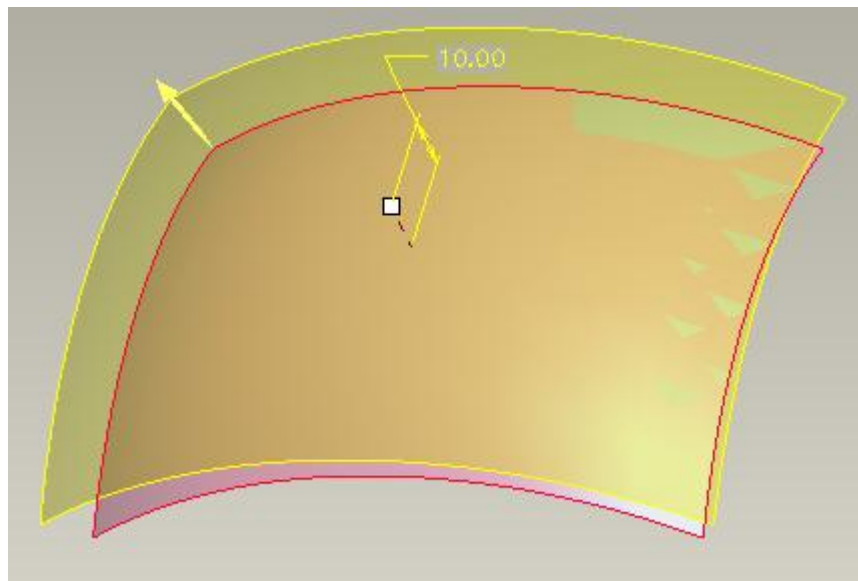
Notice that Standard () option is selected by default in the Offset Types list.




Enter **10** as the offset value on the dashboard.



The preview will appear as shown below.



You can pick the  icon to reverse the direction of feature creation.

Pick  icon to complete the feature.

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

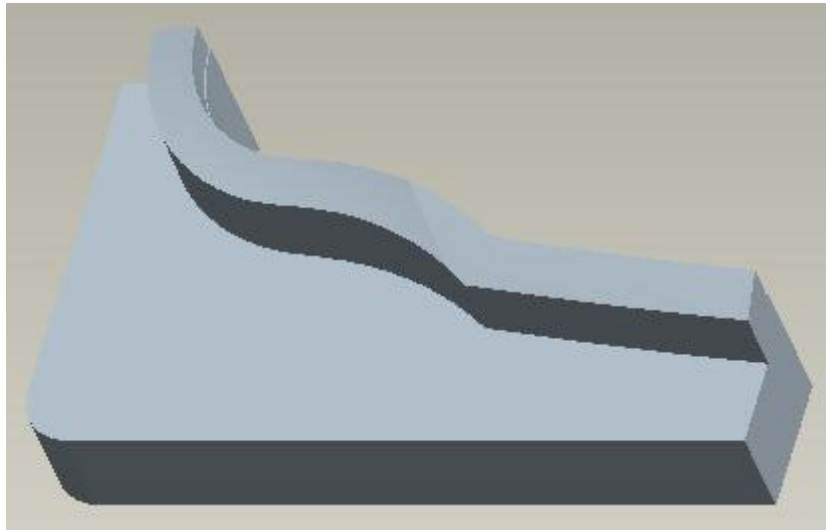
Copy

Copy surface tool creates a new quilt by copying existing quilts or surfaces. Creo Parametric creates the surface feature directly on top of the selected surfaces.

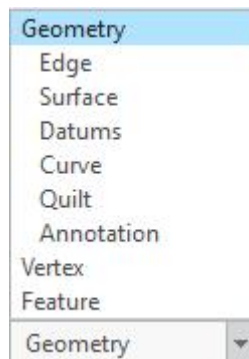
Exercise 1

In this exercise we will learn to create a copy surface feature.

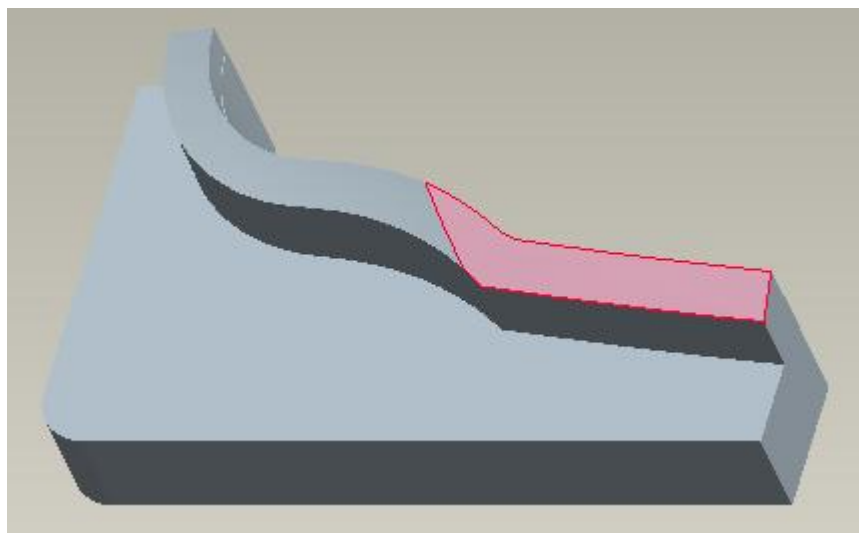
Set working directory to the BASICS folder and open the model COPY1.PRT. The model will appear as shown below.



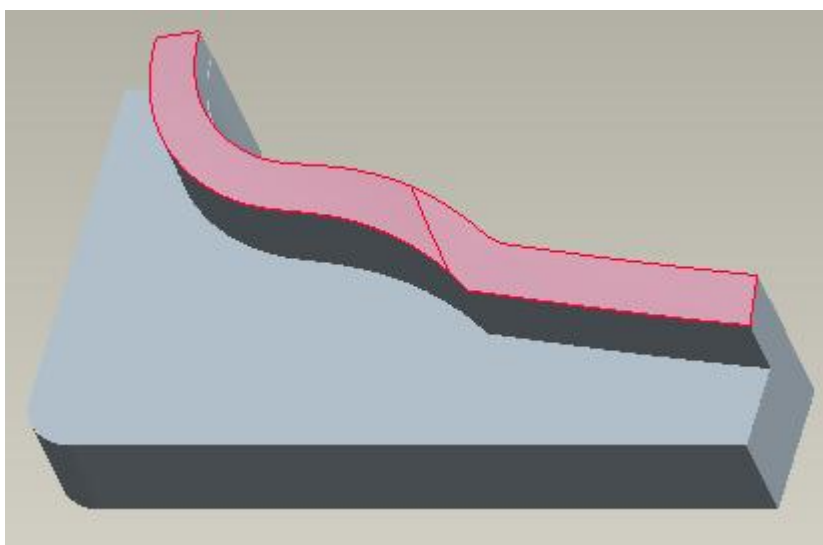
Notice that default the default selection filter is **Geometry** as shown below. (You can find this filter at the lower right corner or graphics window)



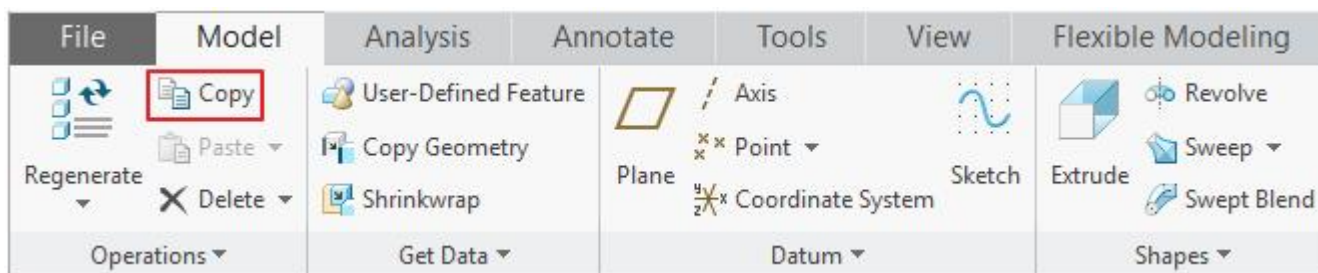
Select the following surface by picking on it.



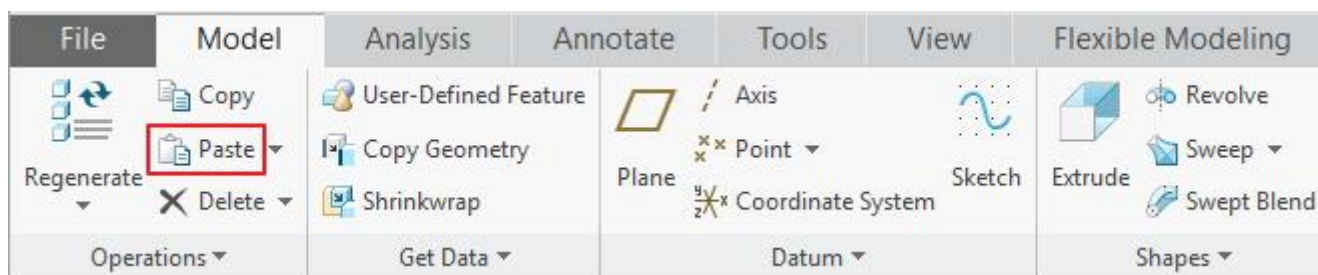
Now hold down the Ctrl key and pick the adjacent surface as shown below.



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




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



Now the dashboard will appear as shown below.



Pick  icon to complete the feature.

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

You can see the copied surface clearly if you view the model in Wireframe mode.



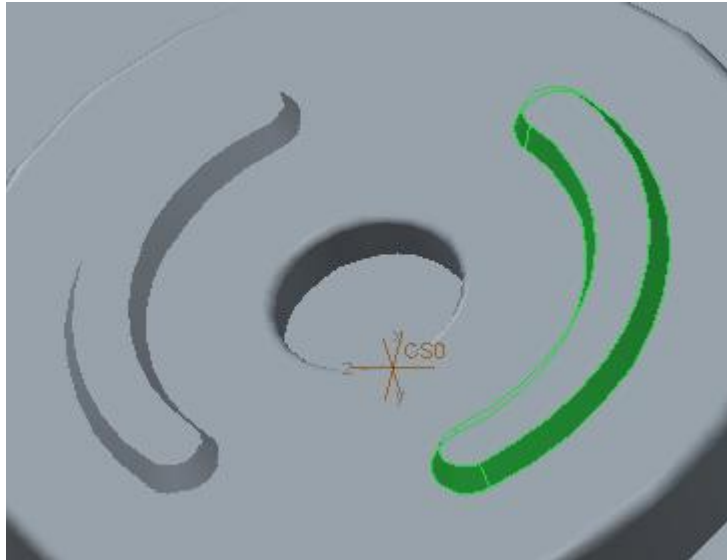
A Copy Surface feature is represented by  icon in the model tree.

Exercise 2

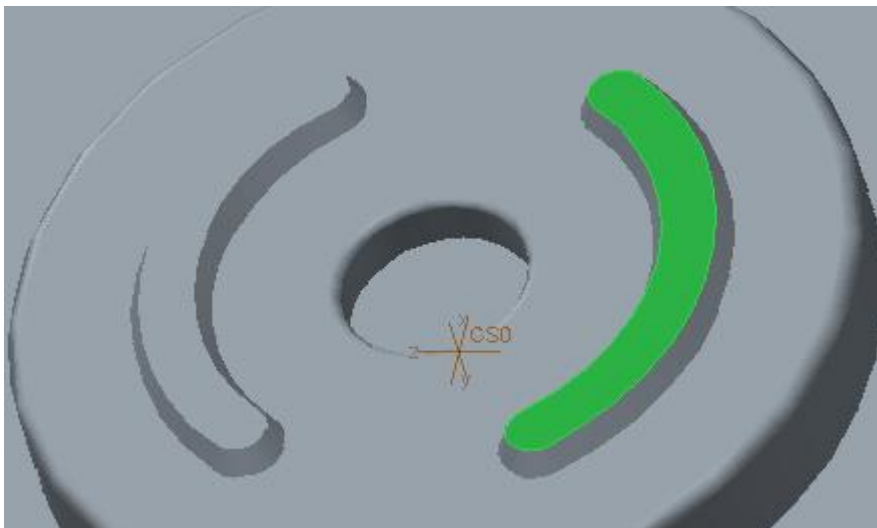
In this exercise we will learn to select surfaces using the Loop selection technique.

Set working directory to the BASICS folder and open the model COPY2.PRT

Here our target is to copy the surfaces highlighted in the figure below.



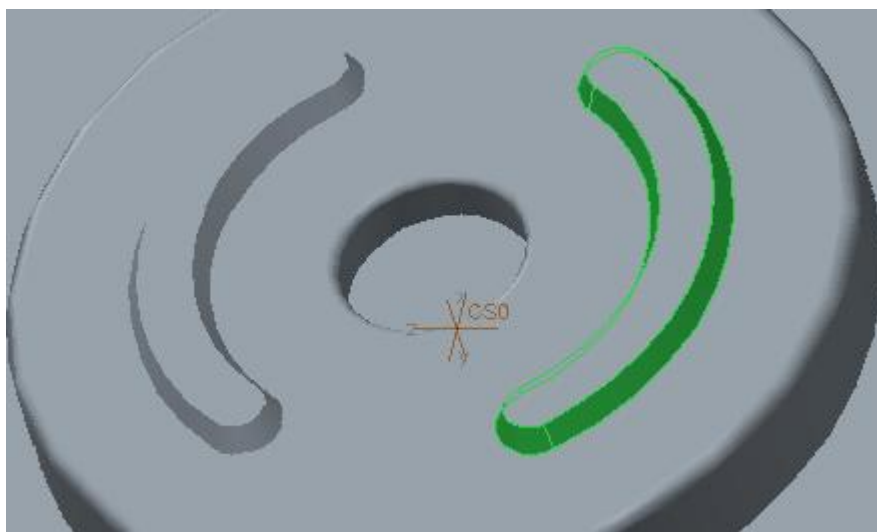
Select the following surface by picking on it.



Now hold down Shift key and place mouse pointer over any edge forming the boundary of the selected surface. The system will highlight the Loop Surfaces as shown below.



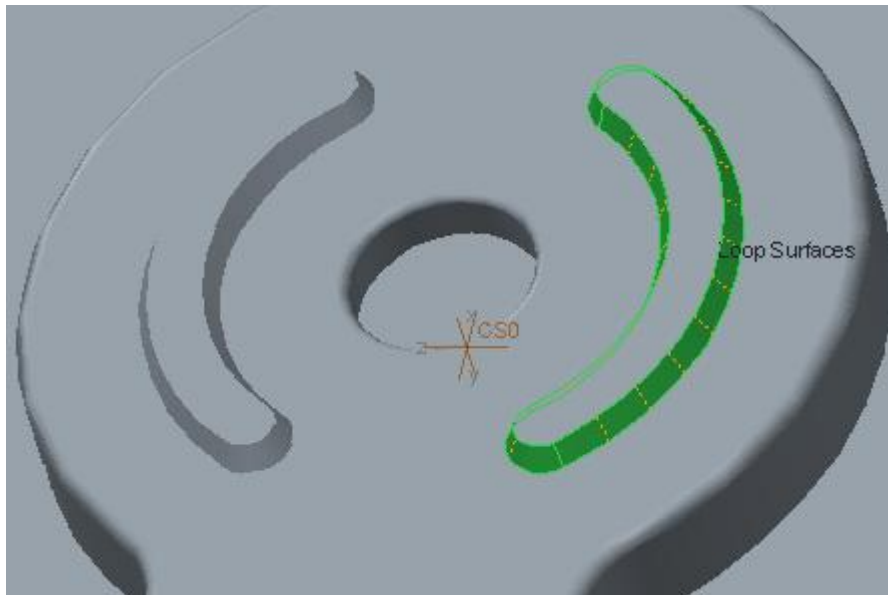
Pick on the edge with left mouse to select the Loop Surfaces while Shift key is held down. The selected surfaces will appear as shown below.




Now release the Shift key and pick copy icon () on the Model tab or press Ctrl+C on the keyboard.

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

If you look in the window you will notice that system shows a tooltip informing the fact that these surfaces have been selected using the Loop Selection technique.



Pick  icon to complete the feature.

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


Exercise 3

In this exercise we will learn to fill a hole in a copied surface feature.

Set working directory to the BASICS folder and open the model COPY3.PRT

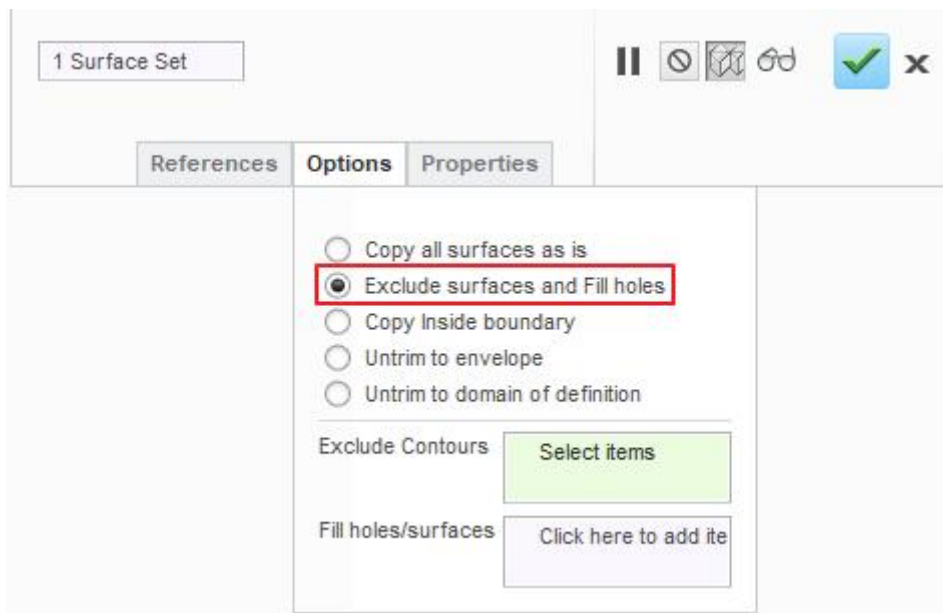
Select the following surface by picking on it.



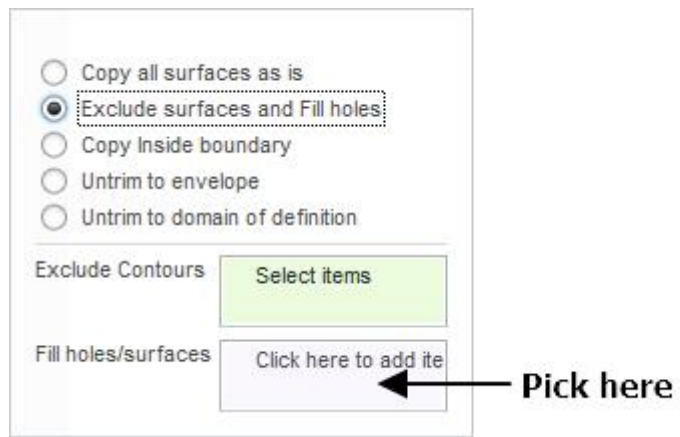
Pick copy icon () on the Model tab or press Ctrl+C on the keyboard.

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

Now the dashboard will appear. Pick **Options** tab and select the **Exclude surfaces and Fill holes** option as shown below.



Now pick in the Fill hole/surfaces collector to make it active.

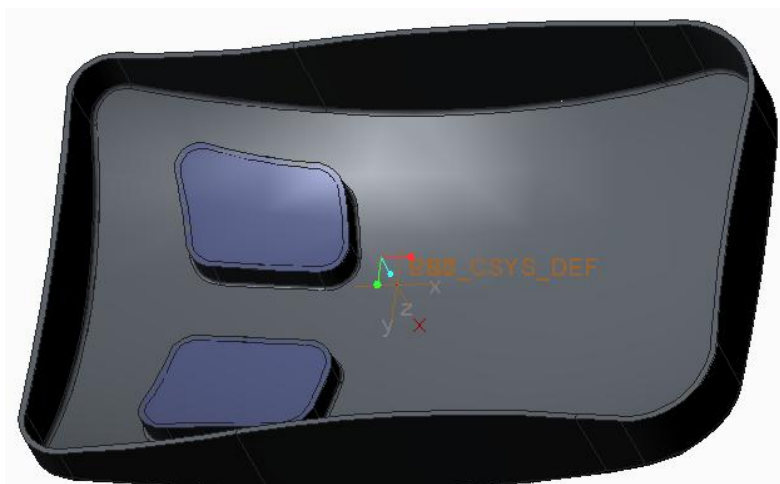


Select the surface highlighted in the figure below.



System fills all the inner loops of the surface added in the Fill hole/surfaces collector.

Pick icon to complete the feature. The new surface will appear as shown below.



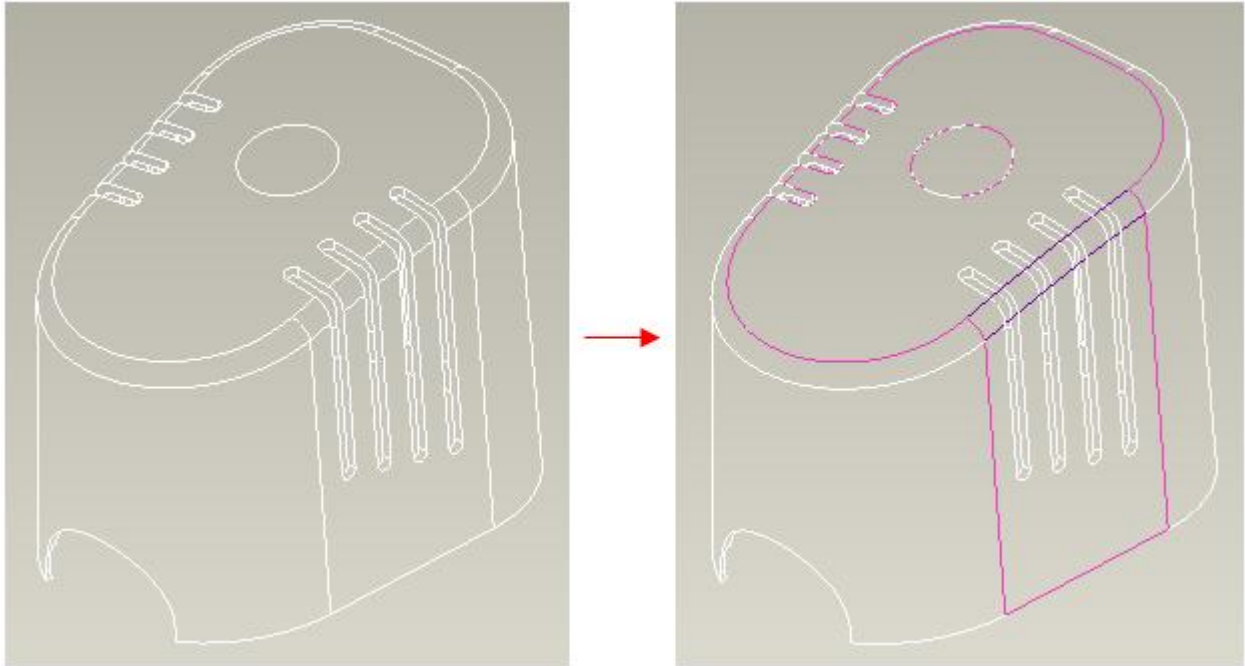
Select **File > Save** to save the work done so far.

Exercise 4

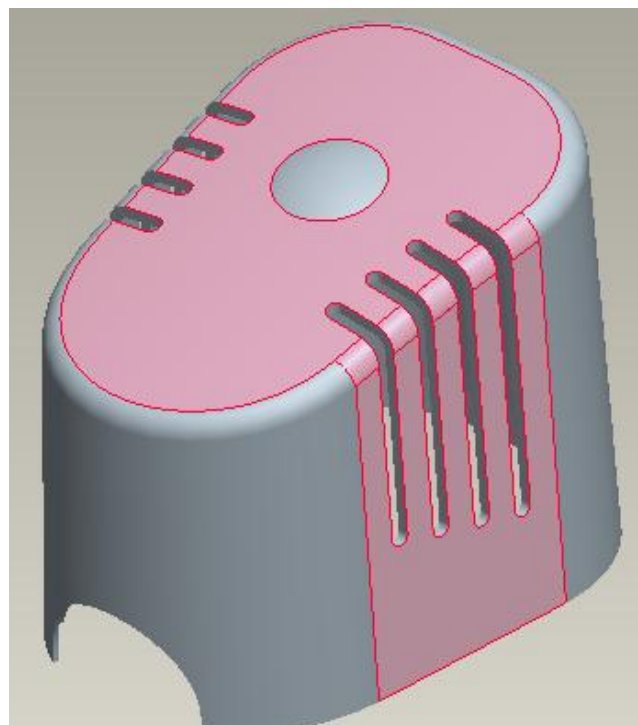
In this exercise we will learn to fill a hole in a copied surface feature.


Set working directory to the BASICS folder and open the model COPY4.PRT


Here our target is to copy the part surfaces and fill the openings in the copied surface as elaborated in the figure below.



Select the three surface patches by picking on them one-by-one while holding down the Ctrl key, as shown below.

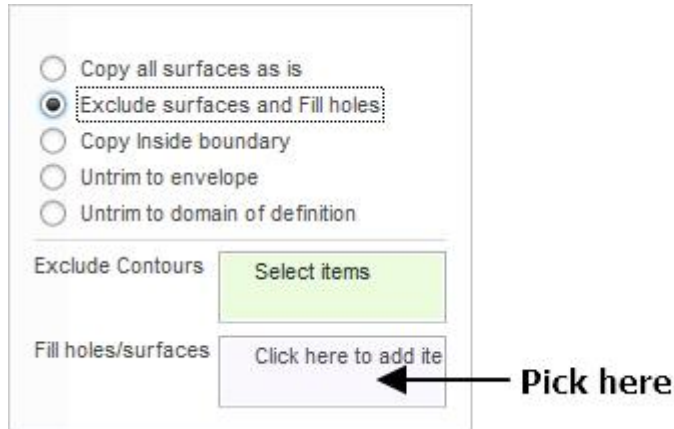


Pick copy icon () on the Model tab or press Ctrl+C on the keyboard.

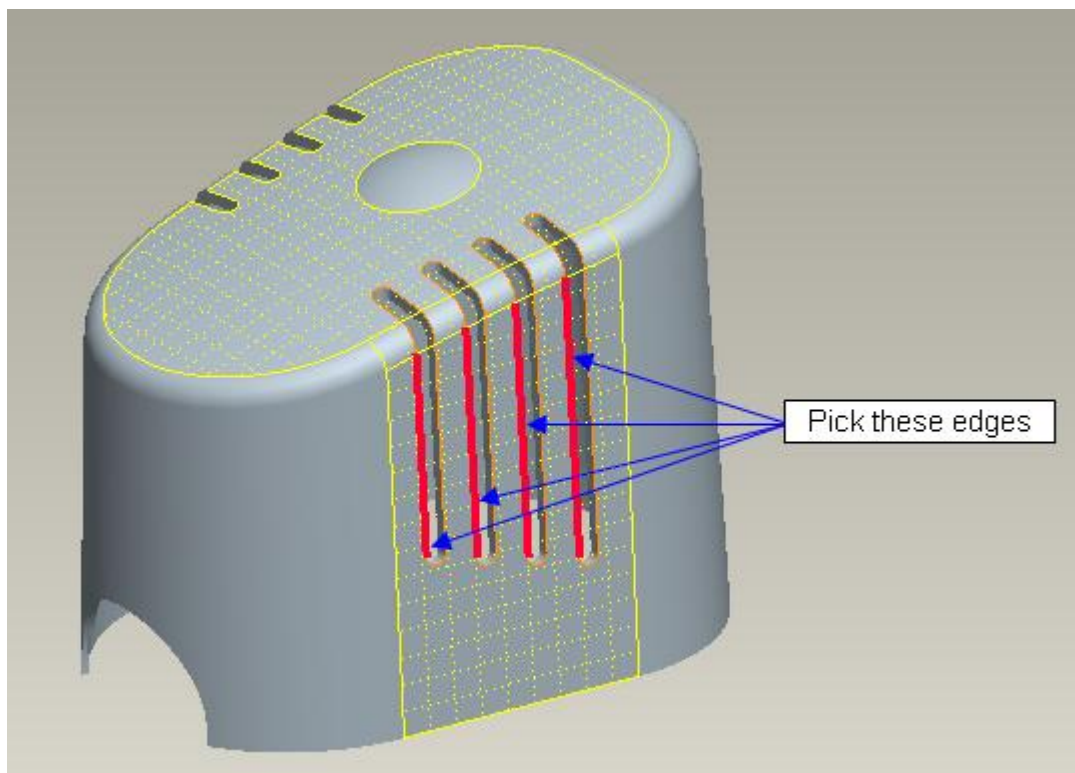
Pick paste icon () on the Model tab or press Ctrl+V on the keyboard.


Now the dashboard will appear. Pick **Options** tab and select the **Exclude surfaces and Fill holes** option.

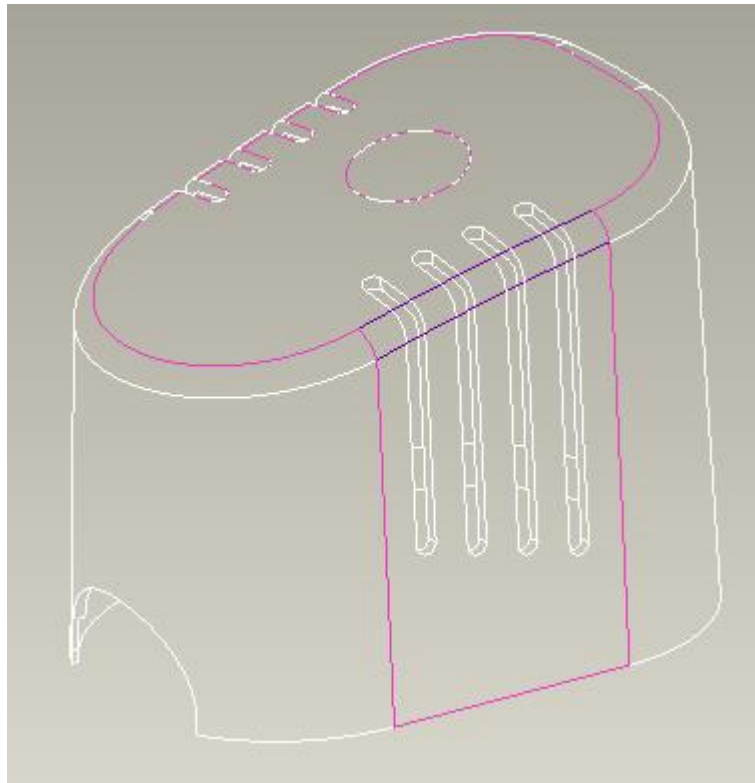
Now pick in the Fill hole/surface collector to make it active.



Now hold down the Ctrl key and select the following edges.



Pick  icon to complete the feature. The new surface will appear as shown below.



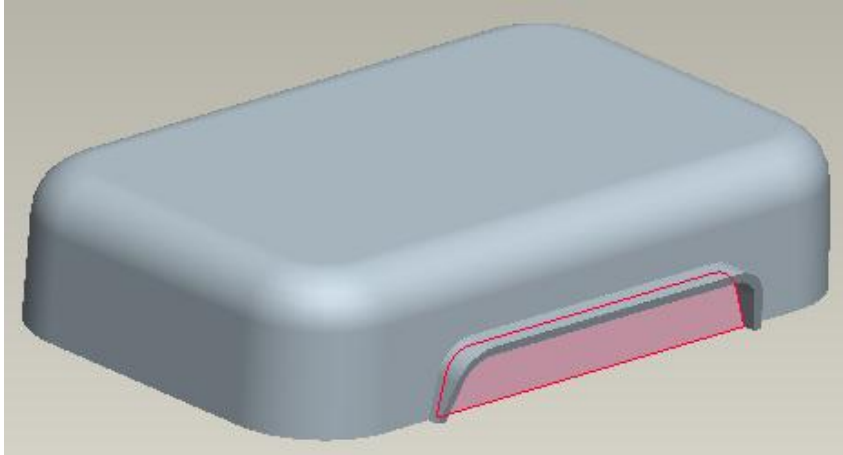
Select **File > Save** to save the work done so far.

Exercise 5

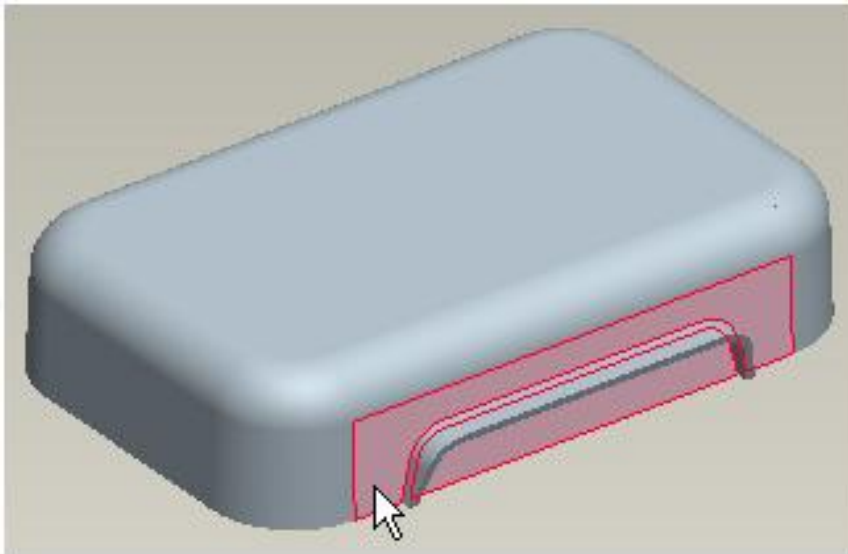
In this exercise we will learn to exclude a surface patch from a copied surface feature.


Set working directory to the BASICS folder and open the model COPY5.PRT

Here our target is to copy the surface patch highlighted in the figure below.



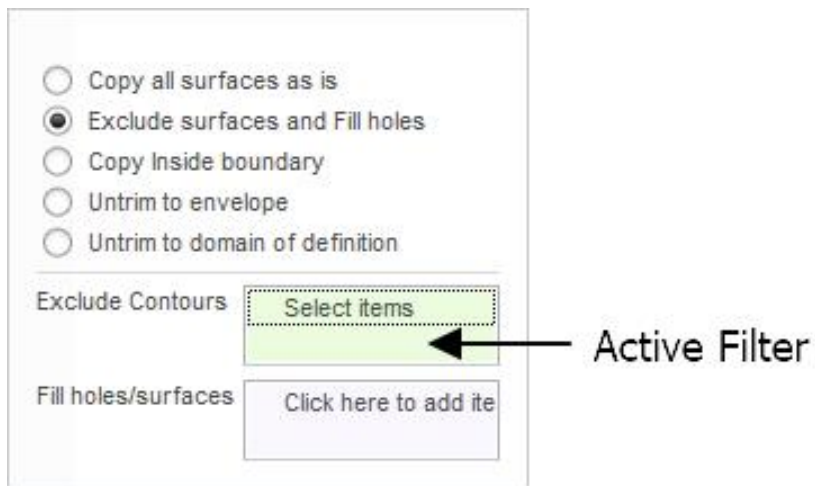
Pick the side surface of the part as shown below.



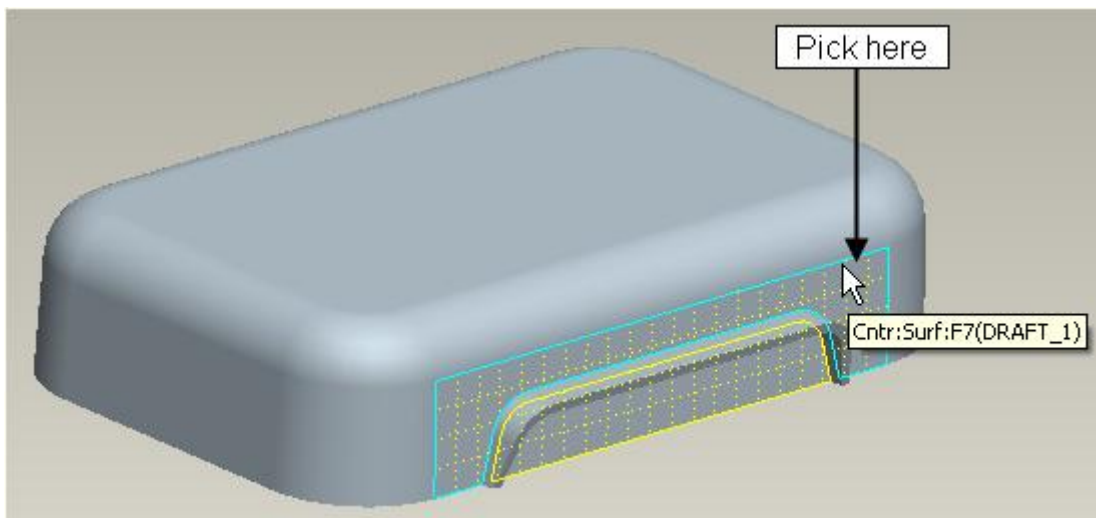
Pick copy icon () on the Model tab or press Ctrl+C on the keyboard.

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

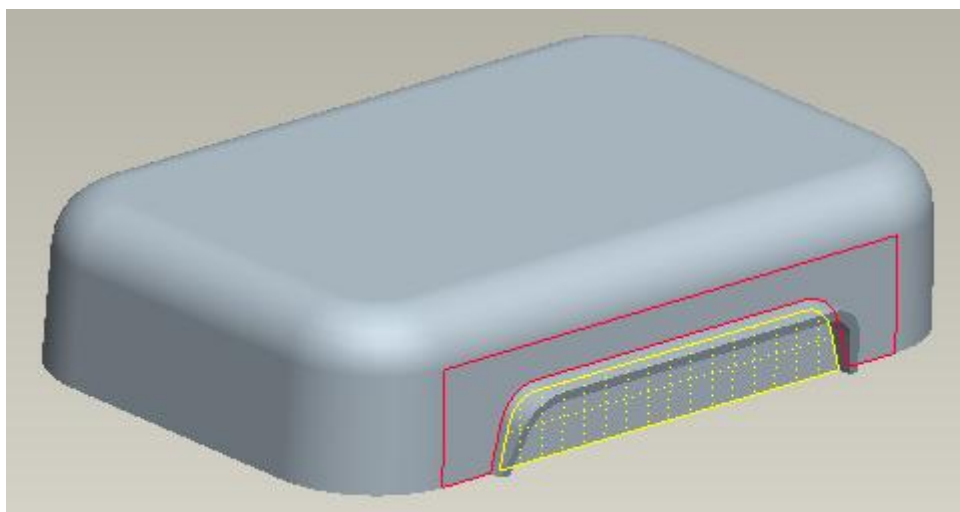
Now the dashboard will appear. Pick  tab and select the **Exclude surfaces and Fill holes** option. Notice that **Exclude Contours** collector is selected by Default as shown below.




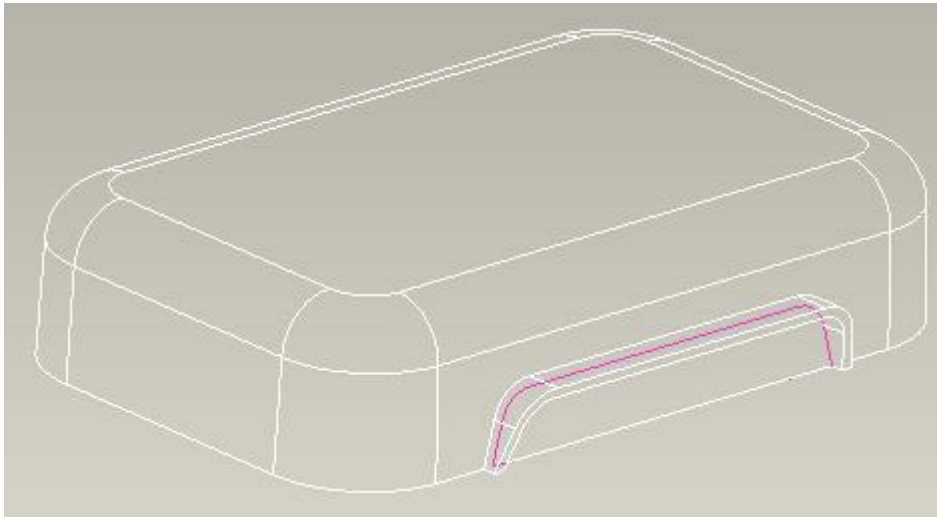
Pick on the boundary curve of the outside surface loop as shown in the figure below.



After picking the highlighted boundary curve the part will appear as shown below.



Pick  icon to complete the feature. The new surface will appear as shown below.



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