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## **Introduction**

### **CATIA 3DEXPERIENCE Sheet Metal Design**

Upon completion of this course, the student should have a full understanding of the following topics:

- Defining sheet metal parameters
- Creating sheet metal parts
- Performing operations on sheet metal parts
- Modifying sheet metal parts
- Working on sheet metal parts in both the folded and unfolded views
- Recognizing normal parts as sheet metal parts

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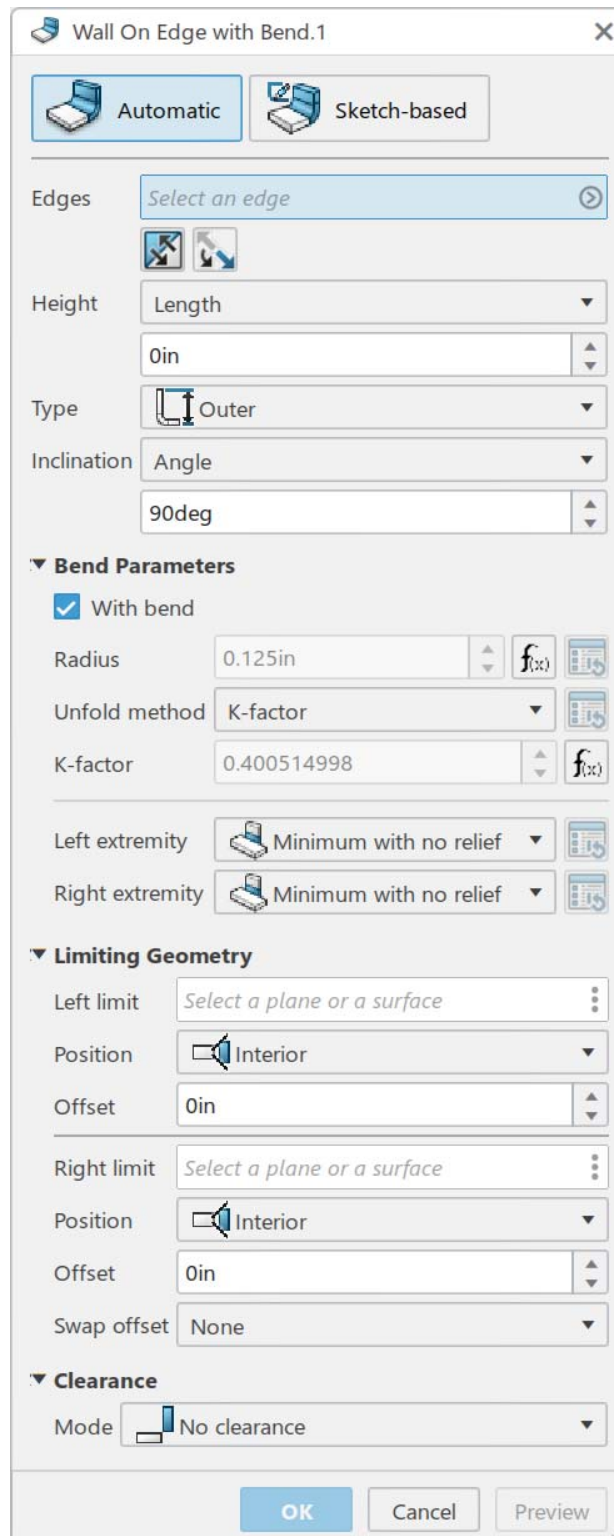
**Sheet Metal**



Most parts can be created by using the Part Design and Wireframe and Surfaces tools. However, there are times when sheet metal specific parts are needed. The sheet metal workbenches allow you to create parts that can be folded and unfolded in order to make them easier to work with. Many times, using sheet metal features can simplify your design. There are a few workbenches in CATIA that deal with sheet metal options. Some of the options appear in more than one sheet metal workbench. They will only be covered once. This course will discuss all of the options found in the Sheet Metal Design and Sheet Metal Hydro-formed workbenches.

## Wall on Edge

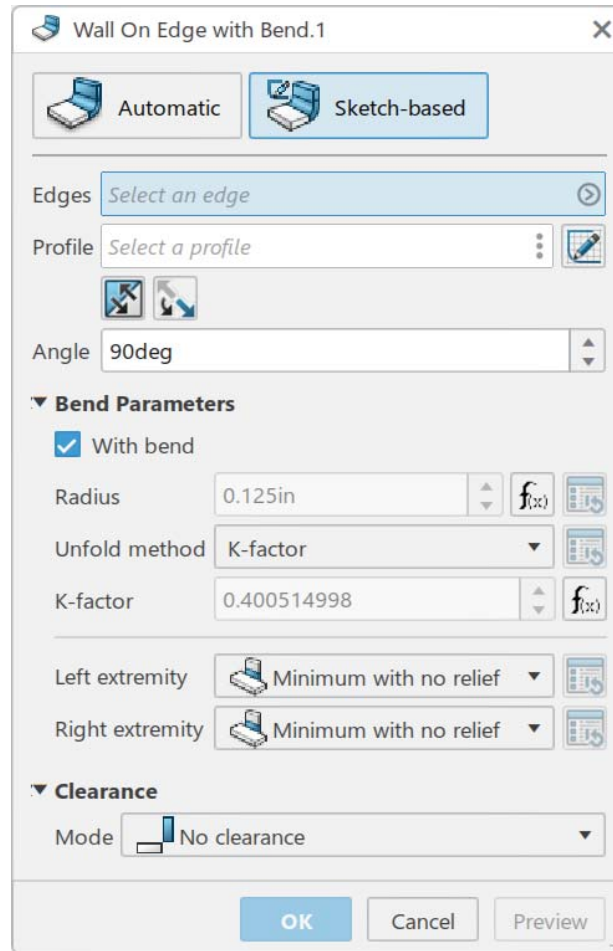
This option creates a wall on an existing edge. With it, you can specify the height and the width of the wall.

Select the **Wall on Edge** icon.  The *Wall On Edge* window appears.



<i>Automatic</i>	Defines the profile for the wall based upon a selected edge
<i>Edges</i>	Specifies the edges the wall will be built along
	Reverses the direction the wall on edge will be built in
	Switches the direction the material thickness will be added in
<i>Height</i>	Defines if the wall will be limited by Length or Up to plane/surface. If <i>Length</i> is selected, the <i>Type</i> will define how it is measured. If <i>Up to Plane/Surface</i> is selected, a limiting element must be defined and the <i>Offset</i> and <i>Position</i> can be used to control the exact definition.
<i>Type</i>	Specifies how the Length value is measured from the supporting wall
<i>Offset</i>	Specifies an additional offset from the plane or surface element
<i>Position</i>	Specifies which side of the wall goes up to the plane or surface
<i>Inclination</i>	Specifies that the angle of the wall will be defined with an <i>Angle</i> parameter or an <i>Orientation plane</i>
<i>With Bend</i>	Specifies whether or not the wall will have a bend radius
<i>Unfold method</i>	Specifies how the material will bend
<i>Left/Right extremity</i>	Specifies the limits for the extremities of the wall
<i>Limiting Geometry</i>	Specifies additional limiting geometry on the sides of the wall on edge if necessary
<i>Clearance</i>	Specifies how far the wall will be offset from the specified edge by the default bend radius; three options are available: <i>No Clearance</i> , <i>Mono-directional</i> , and <i>Bi-directional</i> .

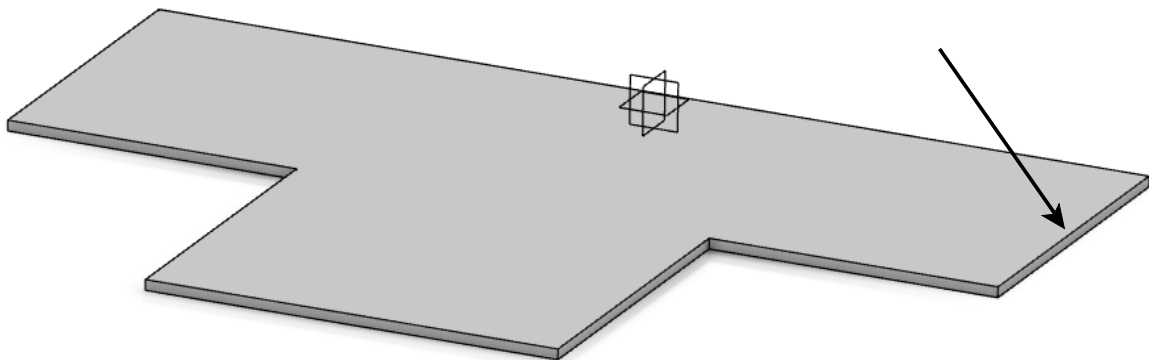
Sketch Based



*Profile* Specifies the sketch to be used for the profile of the wall

*Angle* Specifies the angle or rotation of the wall

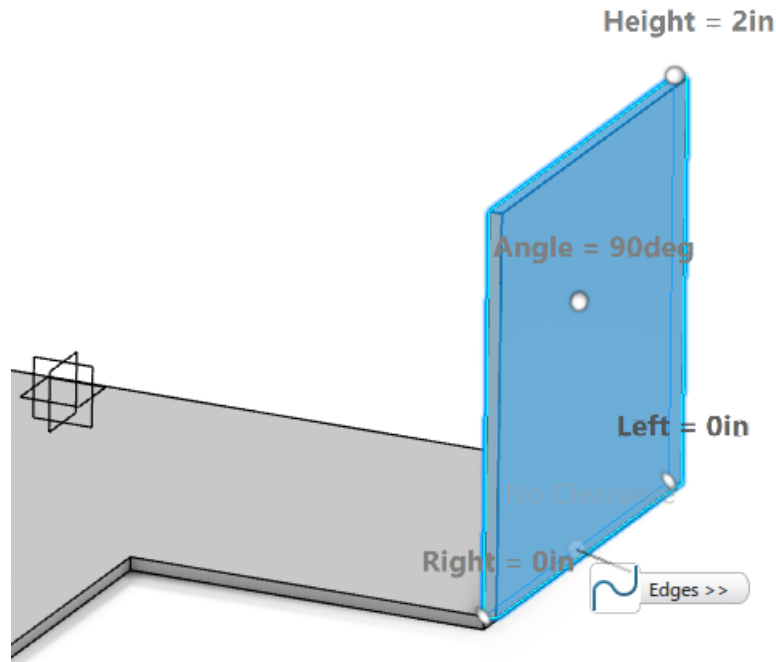
**Specify *Automatic* and select the edge shown below.** Multiple edges can be chosen, if desired.



**Set the *Type* to be *Outer*.** The height will be measured from the bottom face of the wall that the selected edge belongs to.

**Set the *Height* to 2.0.** The height of the wall will be two inches from the bottom of the original wall.

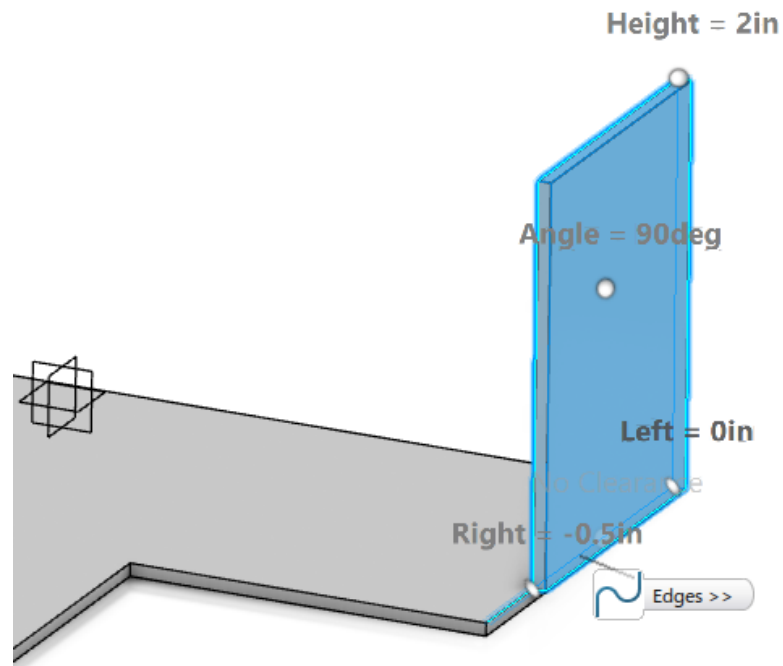
**Turn off the *With Bend* option and select *Preview*.** The wall should appear as shown. Depending on how the original sketch for the wall was created, the *Left* and *Right* side extremities could be switched.



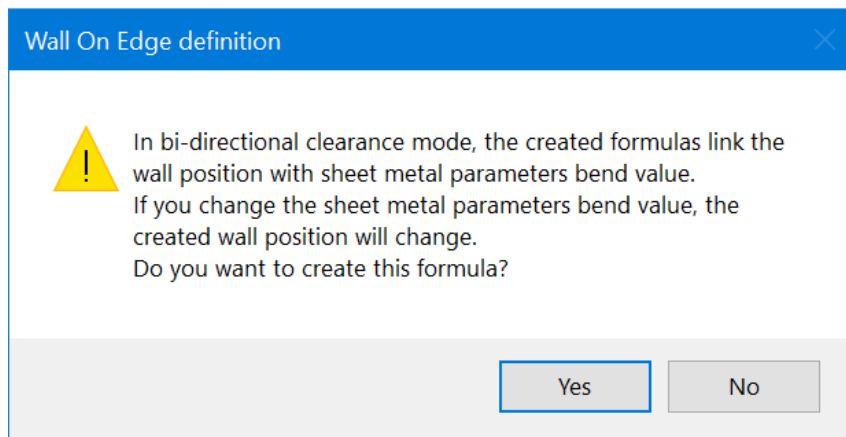
**Key in -0.5 for the *Right Offset* in the *Limiting Geometry* section.** The left limit of the new wall will stop 0.5 inches before the end of the edge. *Note: Your offset direction may be opposite depending on how you created the sketch for the initial wall.*



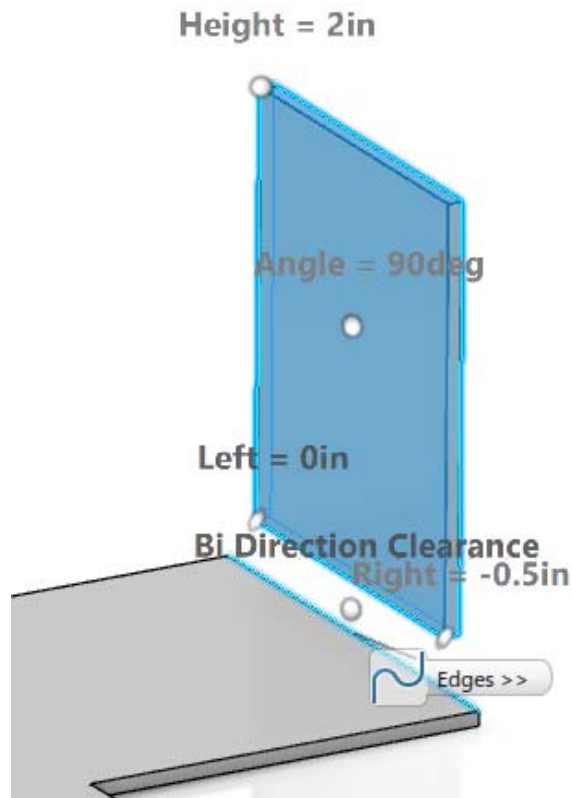
Select **Preview**. The wall should appear as shown.



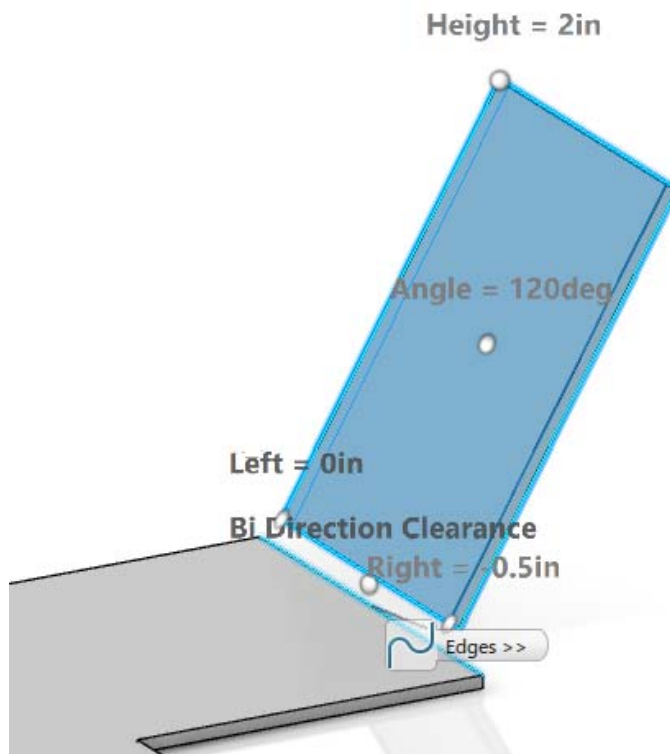
Change the **Clearance Mode** to be **Bi-directional** and click **Preview**. A **Feature Definition Warning** window appears.



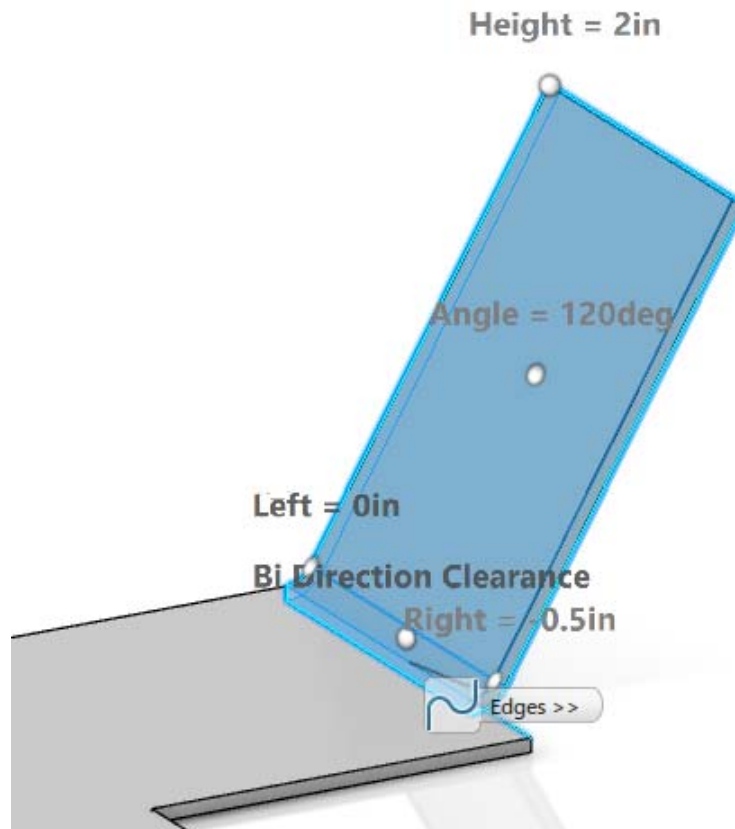
Select **Yes** and click **Preview** again. The wall should appear as shown. It is offset 0.125 inches from the edge corresponding to the *Default Bend Radius* that was specified in *Sheet Metal Parameter.1*.



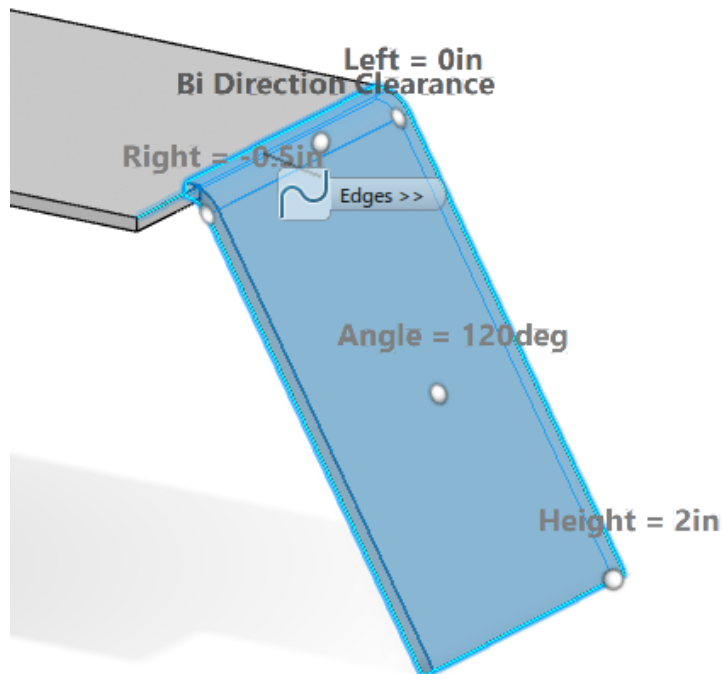
Change the *Inclination* to 120 and select **Preview**. The wall should appear as shown.



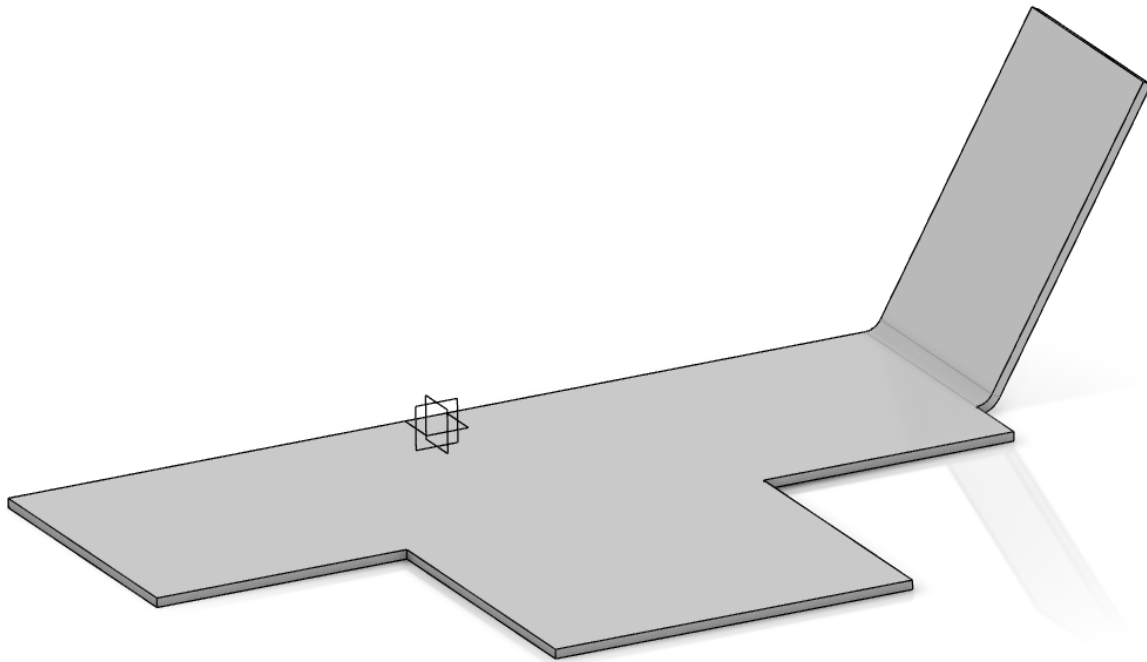
Turn on **With Bend** and click **Preview**. The wall should look like this. The bend radius is defined by the sheet metal parameters.



Select the **Reverse Position** icon and click **Preview**.  The wall should appear as shown.



Select the **Reverse Position** icon again and click *OK*.  The wall is created.

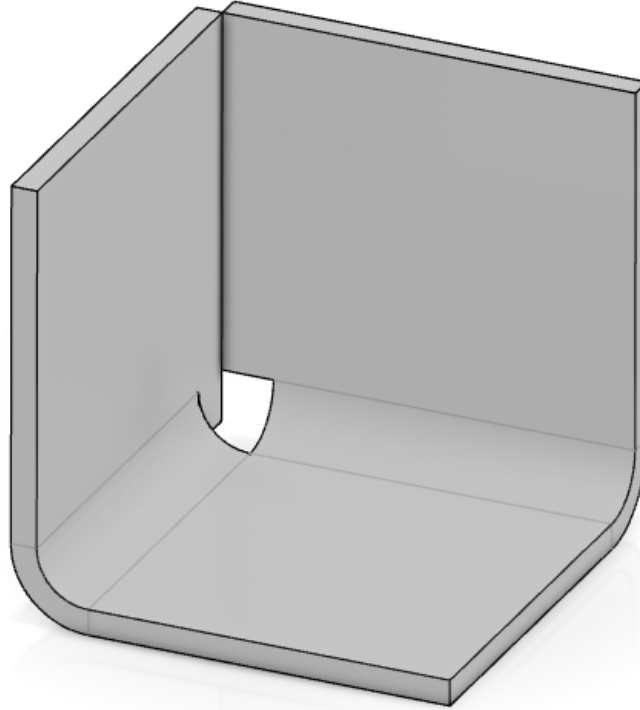


**Save and close the document.**

## Corner Relief

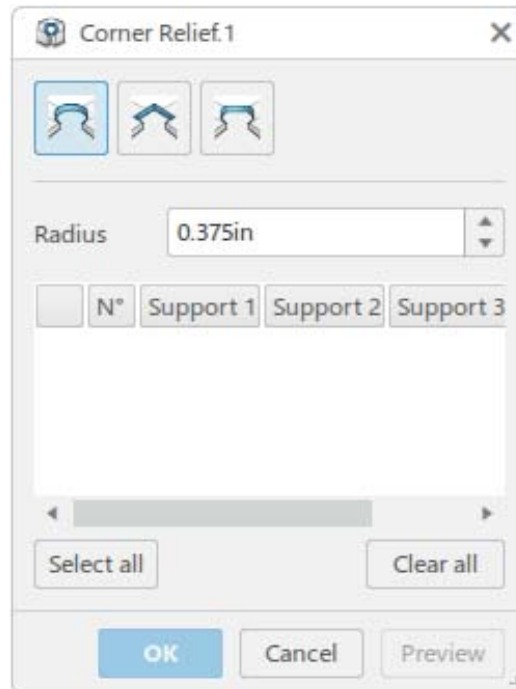
This option creates corner reliefs where two bends come together.




**Open the SMTL - Corner Relief document.** The part consists of three walls with two bends between the bottom wall and side walls.



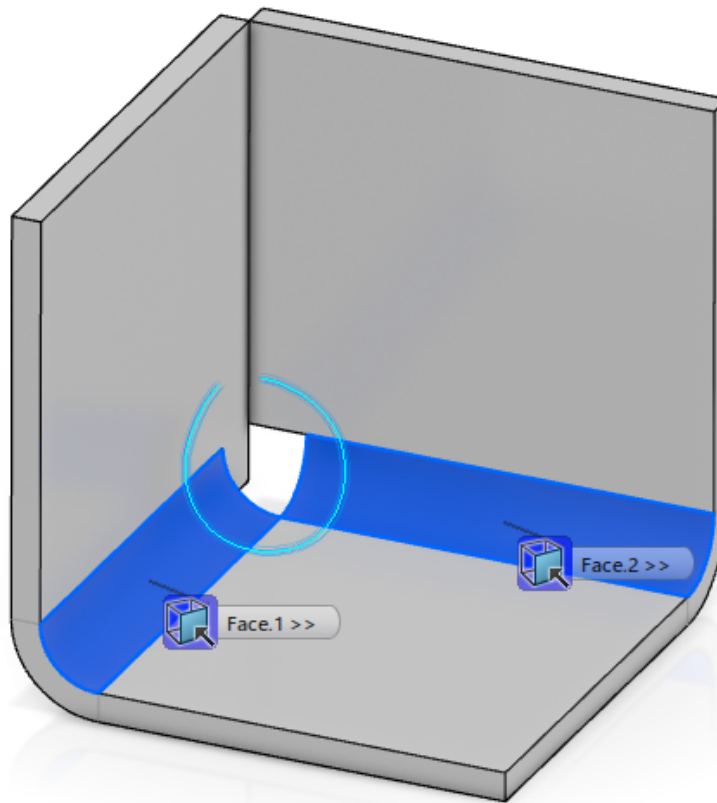
Since no corner relief has been defined, CATIA guessed at how to handle the corner when creating the two bends. The result is undesirable.

Select the **Corner Relief** icon.  The *Corner Relief* window appears.

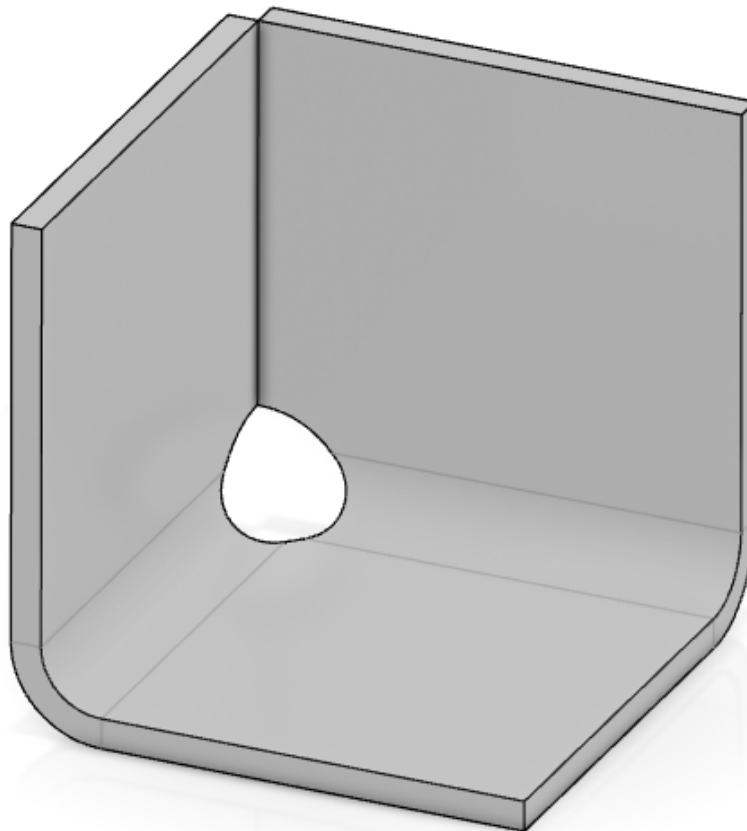



-  **Circular** Specifies a circular relief
  -  **Square** Specifies a square relief
  -  **User Profile** Specifies a user defined profile relief
- Radius* Defines the radius for a *Circular* profile
- Length* Defines the length for a *Square* profile
- Profile* Defines the sketch for a *User* profile
- Support 1, 2, 3* Specifies the supports for the corner relief
- Centers* Defines the center location for a *Circular* profile

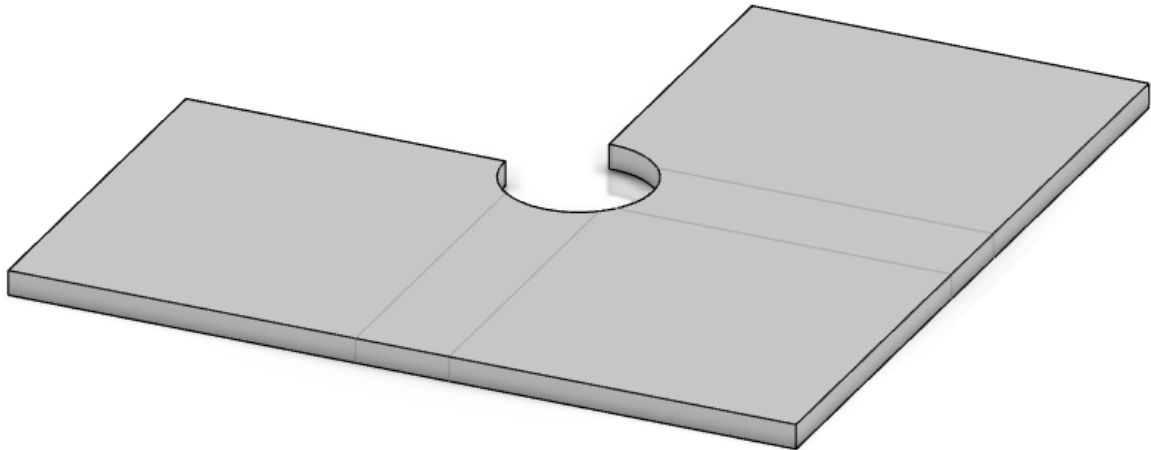
Select the two bend faces shown below to define the supports for the relief.



Set the *Radius* to 0.375 and click *OK*. The corner relief is created.

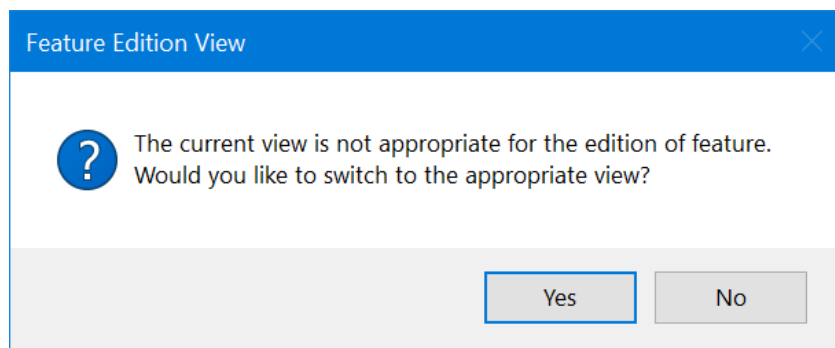


Select the **Fold/Unfold** icon.  The part is unfolded.



Show **Sketch.2** from the specification tree. You will replace the circular corner relief with a square profile.

Double-select **Corner Relief.1** in the specification tree. A *View Edition View* window appears.



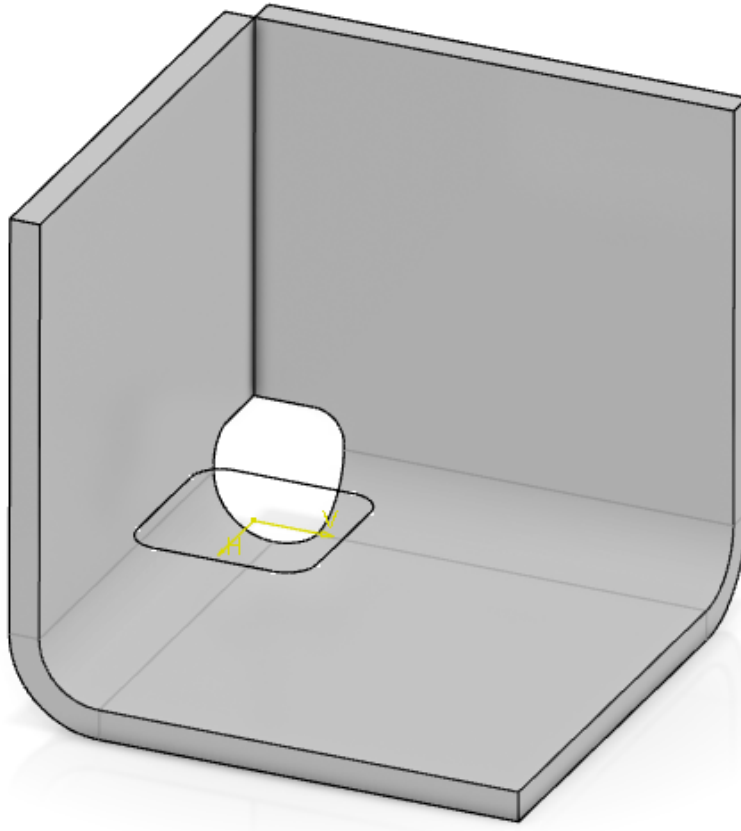
Select **Yes**. The *Corner Relief* window appears.

Select the **User Profile** icon. 

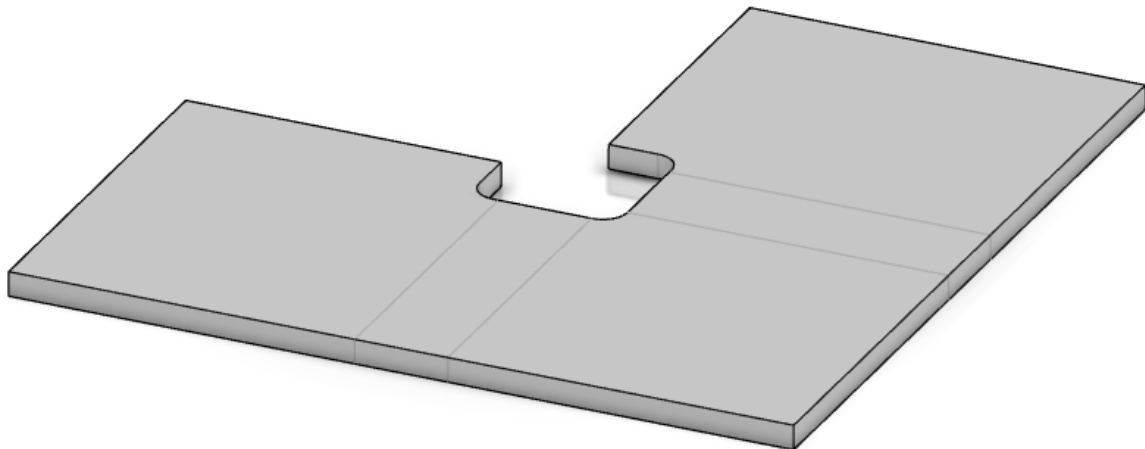
Click in the *Profile* field and choose **Sketch.2** from the specification tree.



Select **OK**. The corner relief is created. It is using the profile defined in *Sketch.2* rather than a circle.



Hide *Sketch.2*, then select the **Fold/Unfold** icon.  The part is unfolded again.



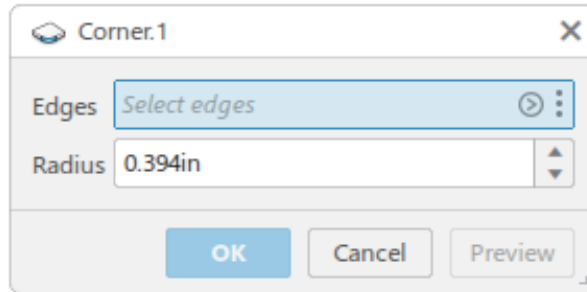
Select the **Fold/Unfold** icon again.  The part is folded.

You will use this model for the next exercise.

**Corner**

The Corner tool is much like the filleting function in Part Design. It rounds off edges on sheet metal parts.

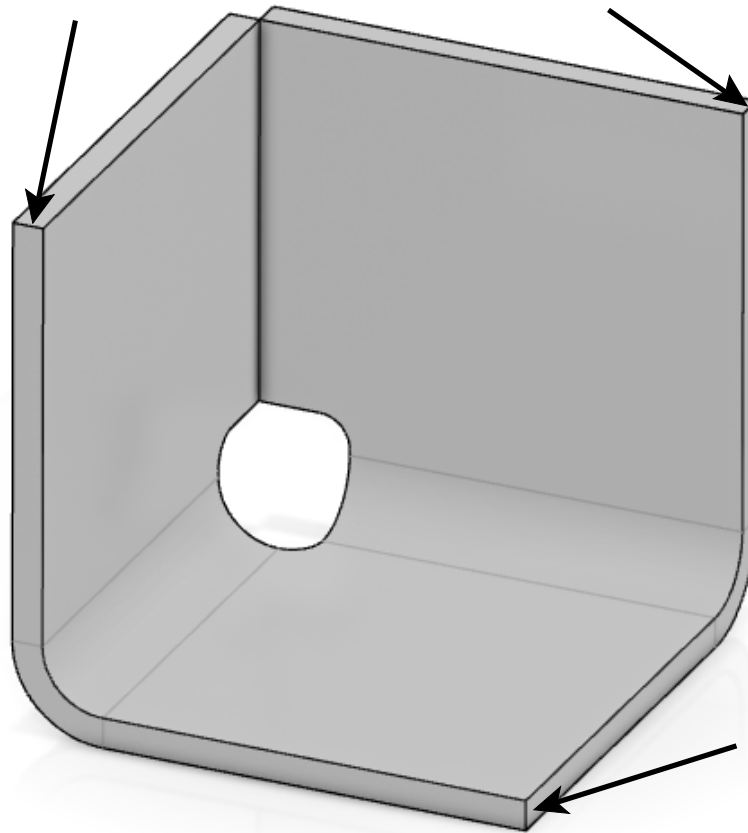
Select the **Corner icon**.  The *Corner* window appears.



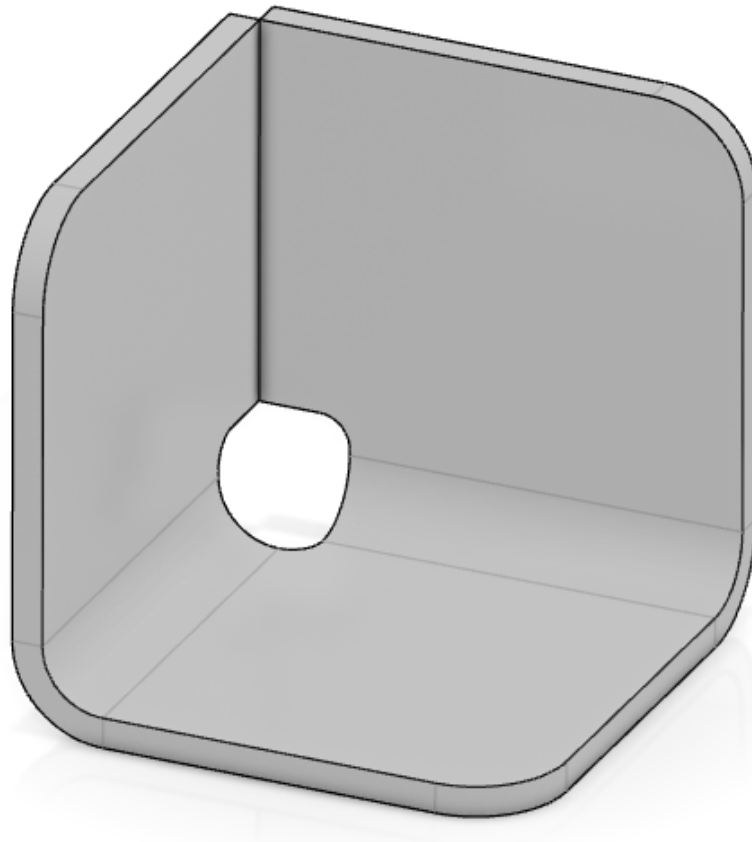
*Edges* Specifies the edges that are to be rounded

*Radius* Specifies the size of the corner

Select the edges shown below.



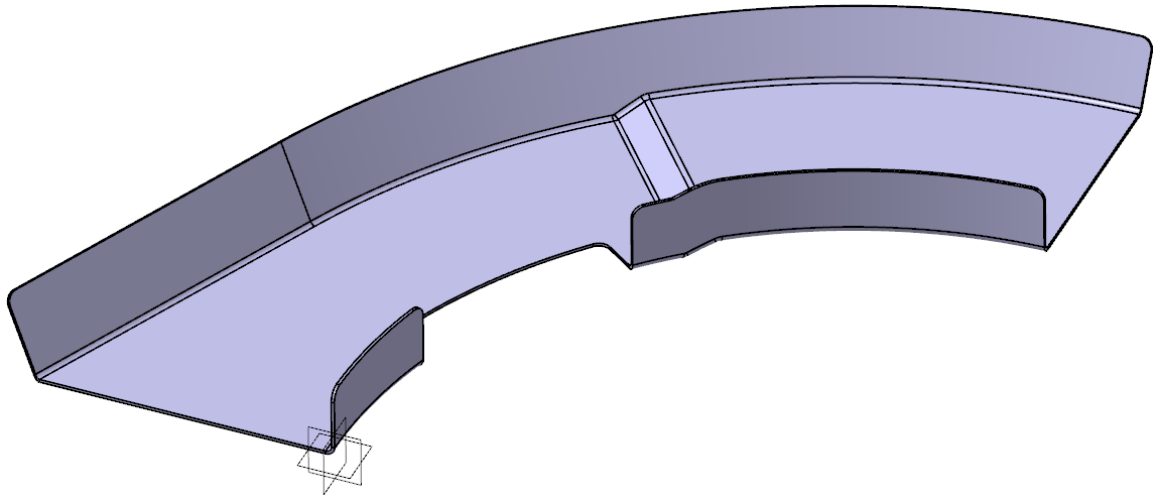
Set the *Radius* to 0.375 and click *OK*. The model appears as shown.



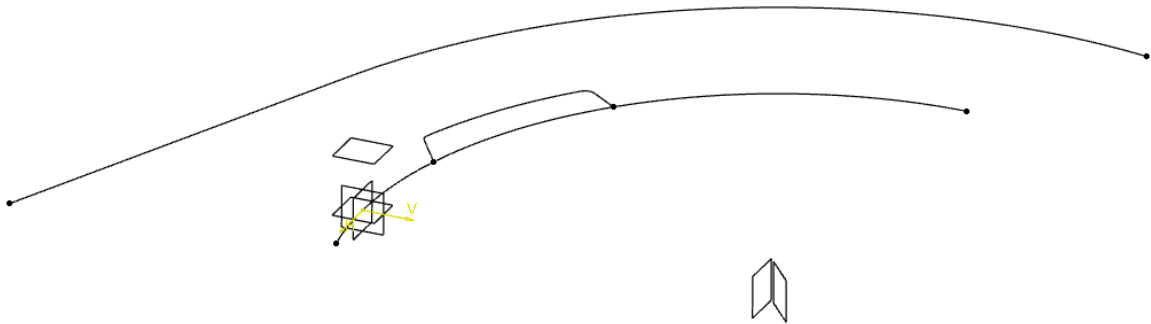
Delete *Corner.1* from the specification tree. You will use the same model for chamfers.

## Hydro-formed Review Exercise

This exercise will review the three, major icons in the Sheet Metal Hydro-formed workbench. You will use them along with some Generative Shape Design tools to build the part below.



Open the **SMTL - Hydro-formed Review** document. It should appear as shown. You will use the existing wireframe geometry to complete the exercise.



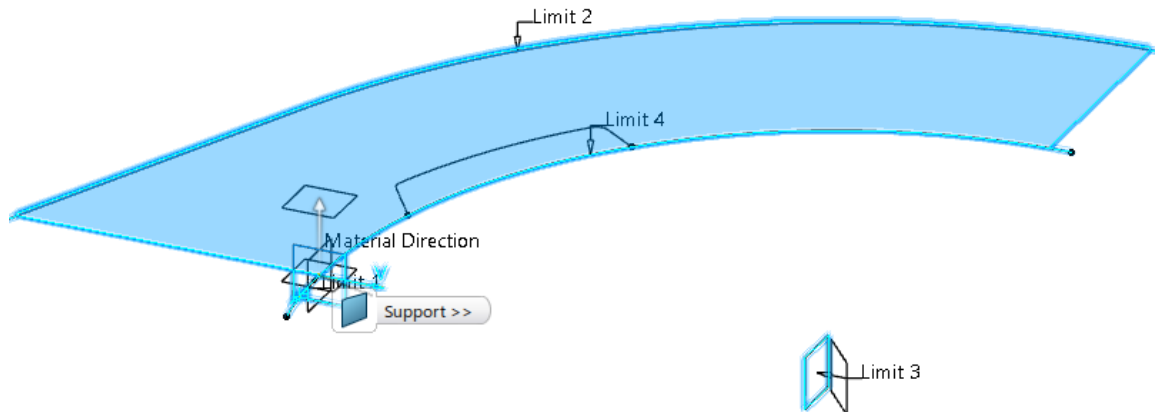
Select the **Sheet Metal Parameters** icon.  The *Sheet Metal Parameters* window appears.

Set the *Thickness* to 0.0625 and the *Bend radius* to 0.125, then select **OK**.

Select the **Web** icon.  The *Web* window appears.

Select the *xy plane* from the specification tree to define the *Support* for the web. You will select multiple wireframe elements to define the *Boundary*.

Select the *yz plane*, *Sketch.1*, *Plane.1*, and *Circle.1* to define the *Boundary* for the web. It should appear like this.

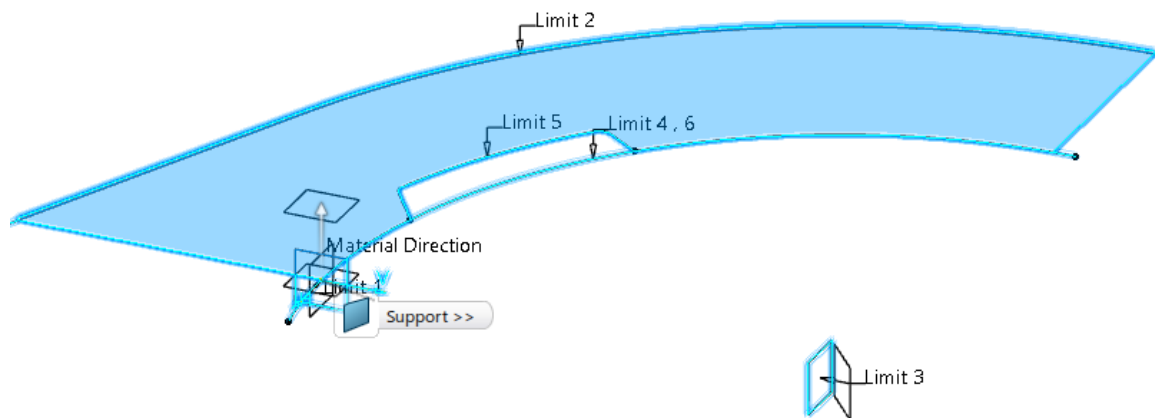


You will incorporate a cutout into this profile within the definition of the web.

Add *Sketch.2* to the list of *Boundary* elements. The profile is no longer closed, so the preview disappears.

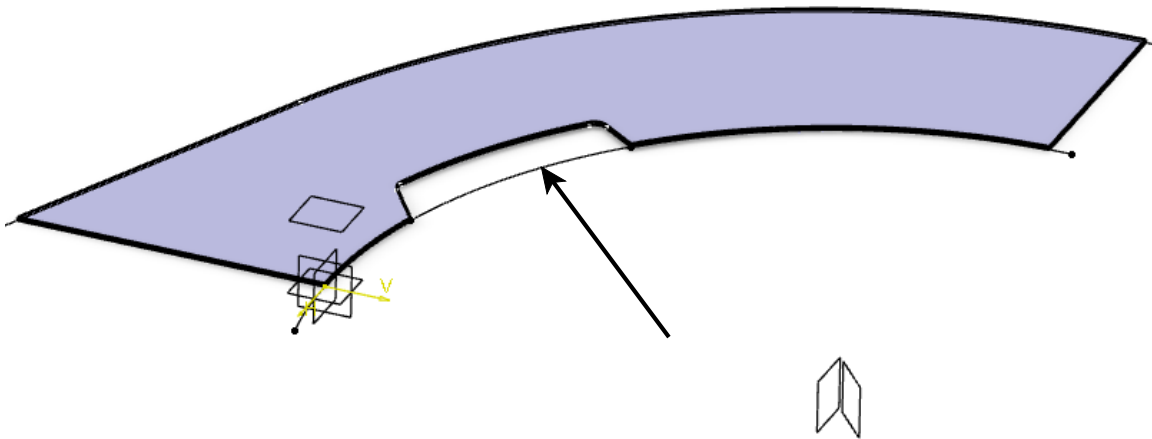
Select *Sketch.2* from the *Boundary* area in the *Web Definition* window. You will add another selection after the sketch.

Right select on *Sketch.2* in the window and select *Add After, 1 Limit* and pick *Circle.1* again. It will be used twice to define the *Boundary*. The preview should appear again.




You can use as many wireframe elements as necessary to define the *Boundary* for the web, as long as they form a closed profile at the end.

Ensure the *Material Direction* arrow is pointing upward and select *OK*. The web is defined.



Now you are ready to build the surfacic flanges that will make up the sides of the part.

**Switch to the Generative Shape Design workbench.** You will need to build the support surfaces for the surfacic flanges.

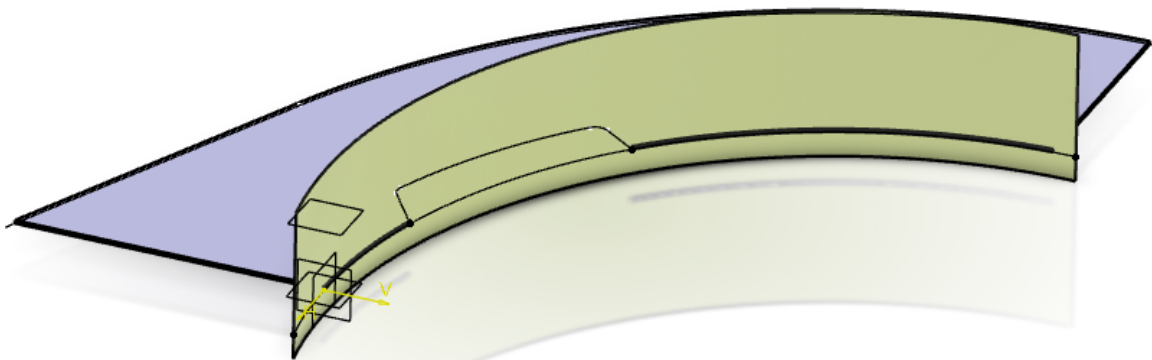
**Select the Extrude icon.**  It is located in the *Surface* section. The *Extrude* window appears.

**Select *Circle.1* as shown above to define the *Profile*, then select the *xy plane* to define the *Direction*.**

Ensure that you do not use the edges of the web to create your support surfaces. This will cause a problem if you need to add a joggle to the web. Building your support surfaces off the existing solid features is usually not a good idea for modeling in the Aerospace Sheet Metal Design workbench.


**Set the *Length* for the *First Limit* to 2.5 and the *Length* for *Second Limit* to 0.5.** This will allow the surface to extend far enough to support the surfacic flange.

**Select *OK*.** The surface should appear as shown.

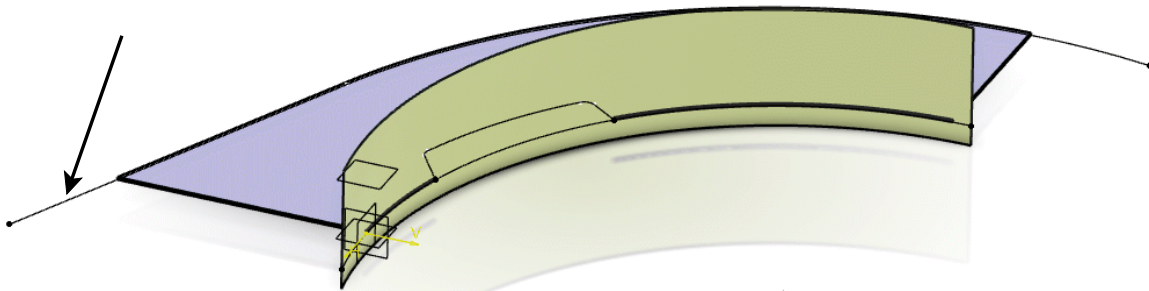


You will define the second surface in a slightly different manner.

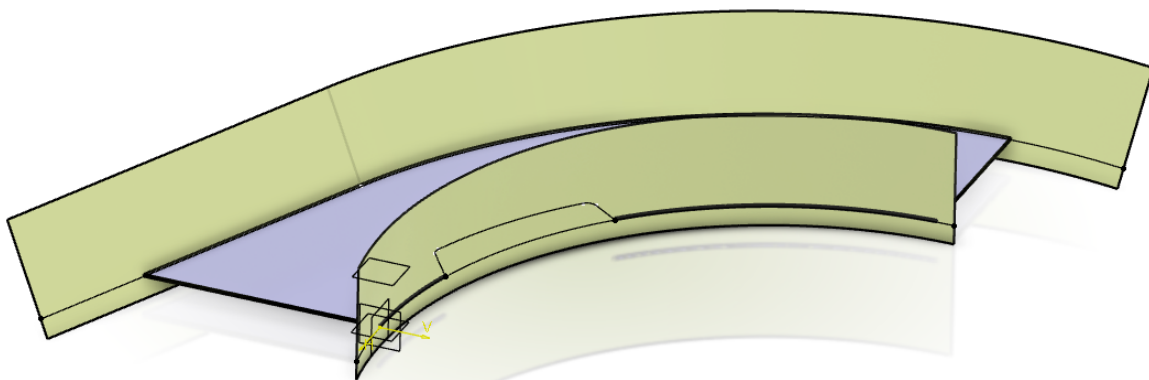
Select the **Sweep** icon.  The *Swept Surface Definition* window appears.

Set the *Profile type* to **Line** and the *Subtype* to **With draft direction**.  This surface will be at a 30 degree angle to the web.

Select *Sketch.1* for *Guide curve 1* and the *xy* plane for the *Draft direction*.




Set the *Angle* to **-30**, *Length 1* to **0.5**, and *Length 2* to **2.5**, then select **OK**. The surface should appear as shown.

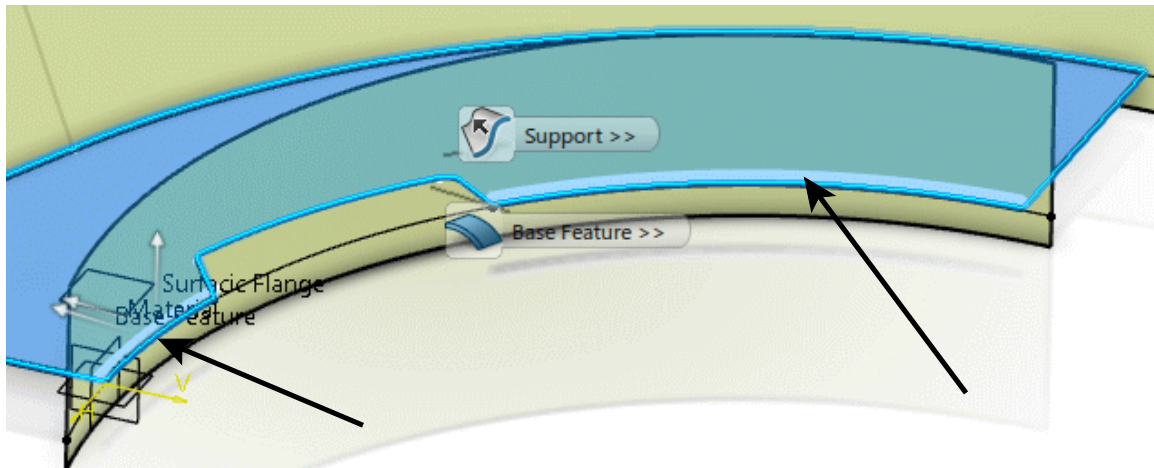


These two surfaces will act as the supports for the surfacic flanges.

Make the *PartBody* the in work object.

Switch to the Sheet Metal Hydro-formed workbench and select the **Surfacic Flange** icon.  The *Surfacic Flange* window appears.

Select the web for the *Base Feature* and the extruded surface for the *Support Geometry*. You will notice that there are actually two separate common edges between the web and the extruded surface because of the cutout that was added to the web definition.



Select **Cancel**. Because there are separate, common edges, it will be best to create the surfacic flanges individually. The surface will need to be split so that there is a separate surface support for each surfacic flange.

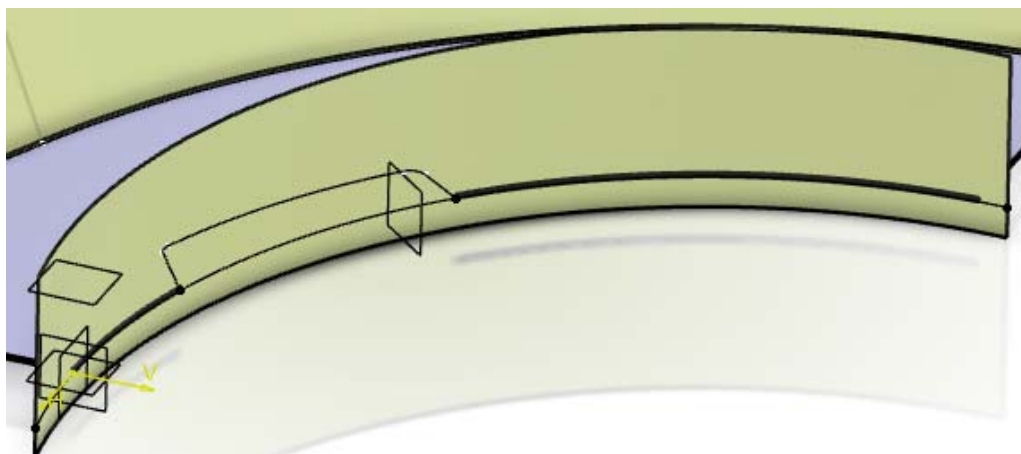
Switch back to the **Generative Shape Design** workbench and select the **Plane** icon.



The *Plane* window appears.

Change the *Plane type* to **Normal to Curve** and select *Circle.1* from the tree. The plane should default to the center of the curve, which actually falls within the cutout between where the surfacic flanges will be created.

Select **OK**. The plane is created.



Select the **Split** icon.  The *Split* window appears.

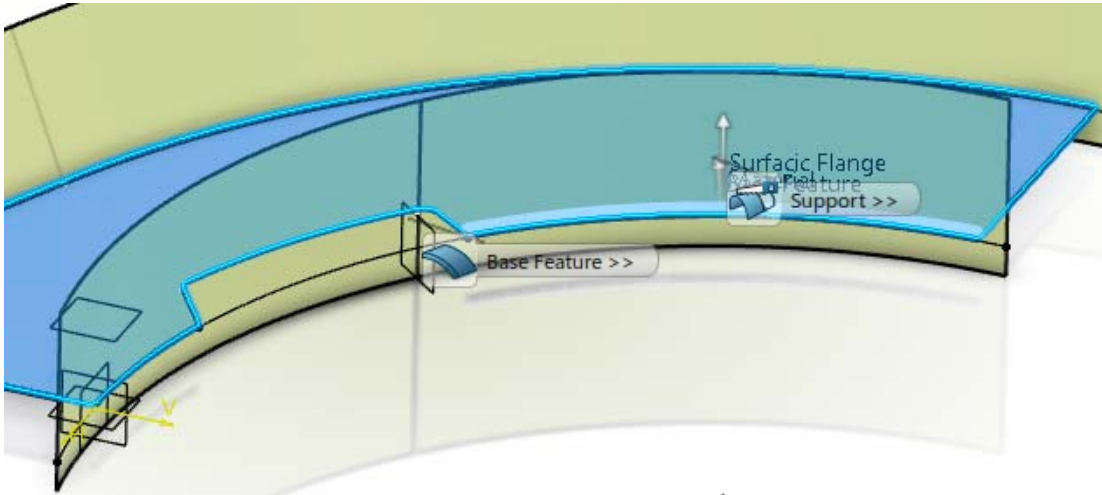
Select *Extrude.1* for the *Element to cut* and the new plane for the *Cutting elements*, then turn on **Keep both sides** and click **OK**. The surface is split and both sides are kept.



Switch back to the Sheet Metal Hydro-formed workbench and make the *PartBody* the in work object.

Select the **Surfacic Flange** icon.  The *Surfacic Flange* window appears.

Select the web for the *Base Feature* and the split surface on the right for the *Support Geometry*. Now, only the edge that intersects the new surface is highlighted.

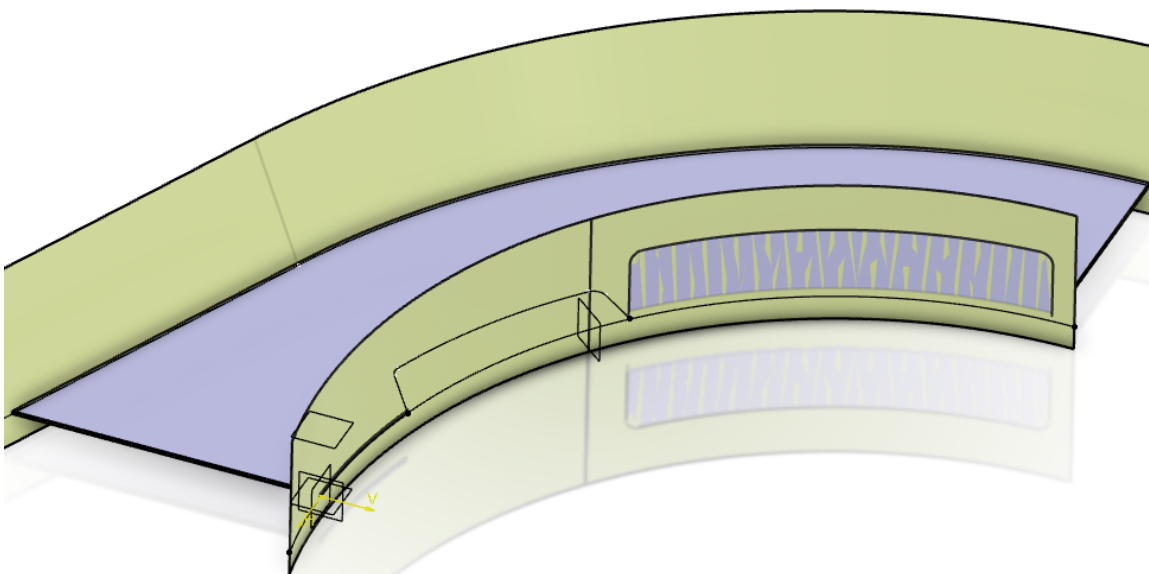



Ensure the direction arrows are pointing inward and upward. You want the flange to be created on the inside of the extruded surface and extend upward from the web.

Expand the *Edge of Part* section and set the *Type* to *Length From OML* and change the *Length* to 1.5.

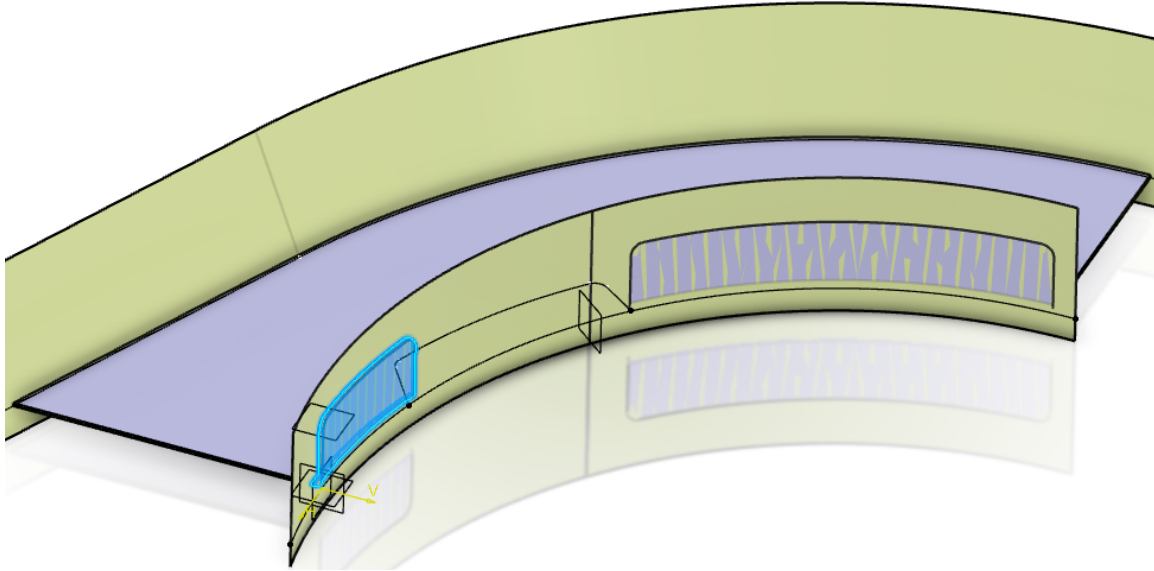
Expand the *Sides and Corners* section and set the *Side 1* and *Side 2* drop-down menus to *Standard*, turn on the *Corner* options, and set both radii to 0.25.

Select *OK*. The surfacic flange should appear as shown.



Select the **Surfacic Flange** icon again, then select the web for the *Base Feature* and the split surface on the left for the *Support Geometry*. 

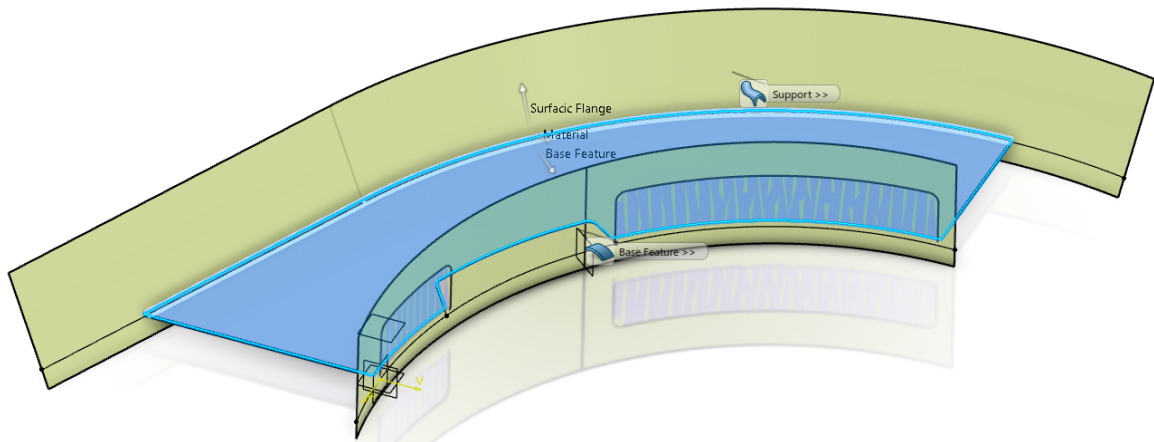
Specify the same parameters as the previous flange and select **OK**. The model should appear as shown.



Now you will create the surfacic flange between the web and the linear swept surface.

Select the **Surfacic Flange** icon.  The *Surfacic Flange* window appears.

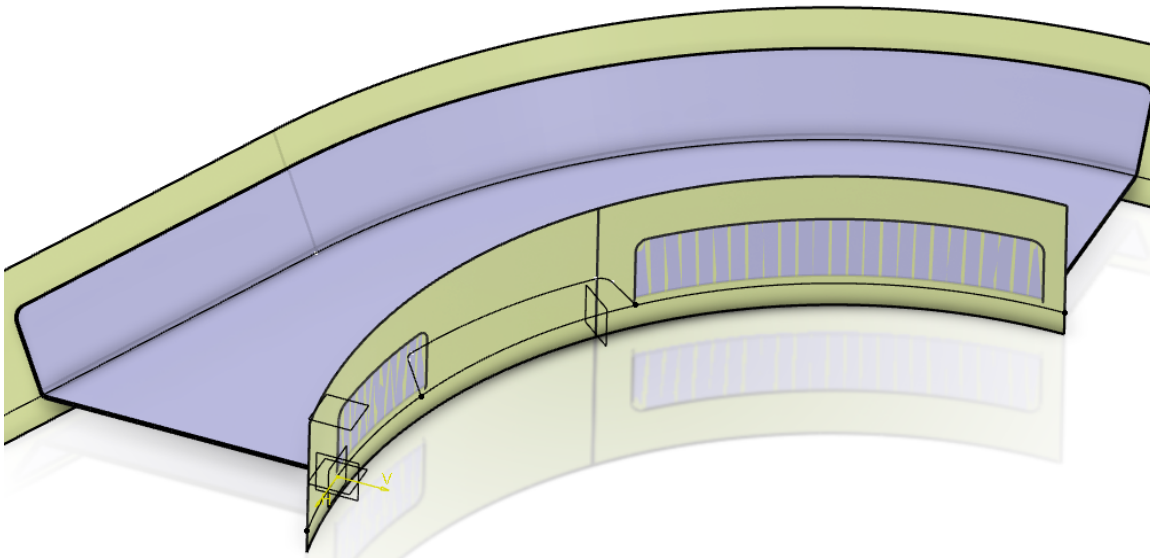
Select the web for the *Base Feature* and the swept surface for the *Support Geometry*. The preview should appear as shown.



**Ensure the direction arrows are pointing inward and upward.** You want the flange to be created on the inside of the swept surface and extend upward from the web.

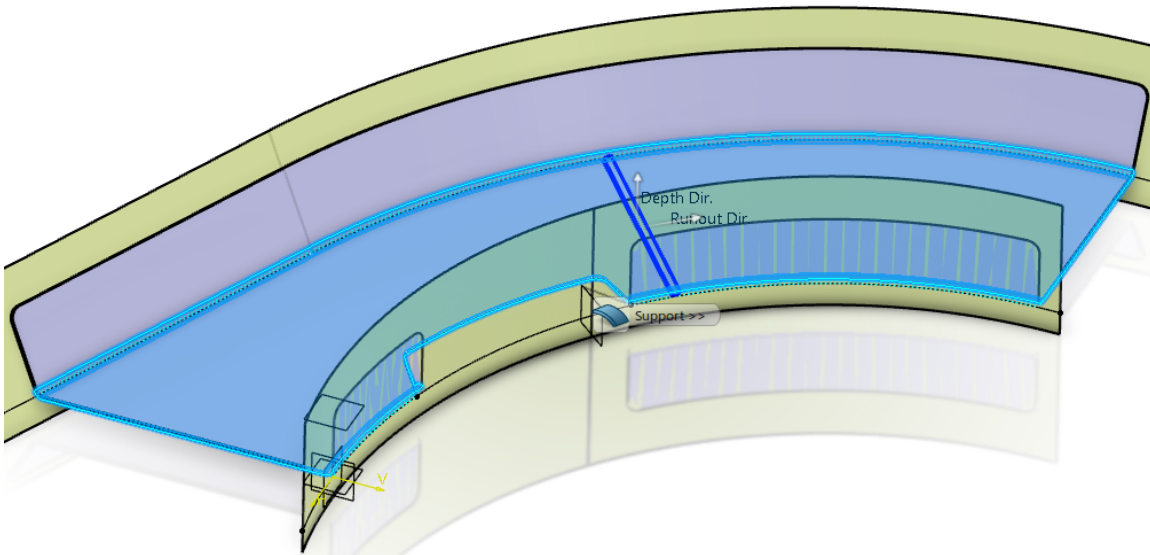
**Expand the *Edge of Part* section, change the *Type* to *Element FD* and select *Plane.2* for the *Boundary Element*.** The surfacic flange will extend up to the plane. The sides and corners should remain the same as the previous surfacic flanges.

Select **OK**. The flange should appear as shown. The joggle will be added next.



Select the **Joggle** icon.  The *Joggle* window appears.

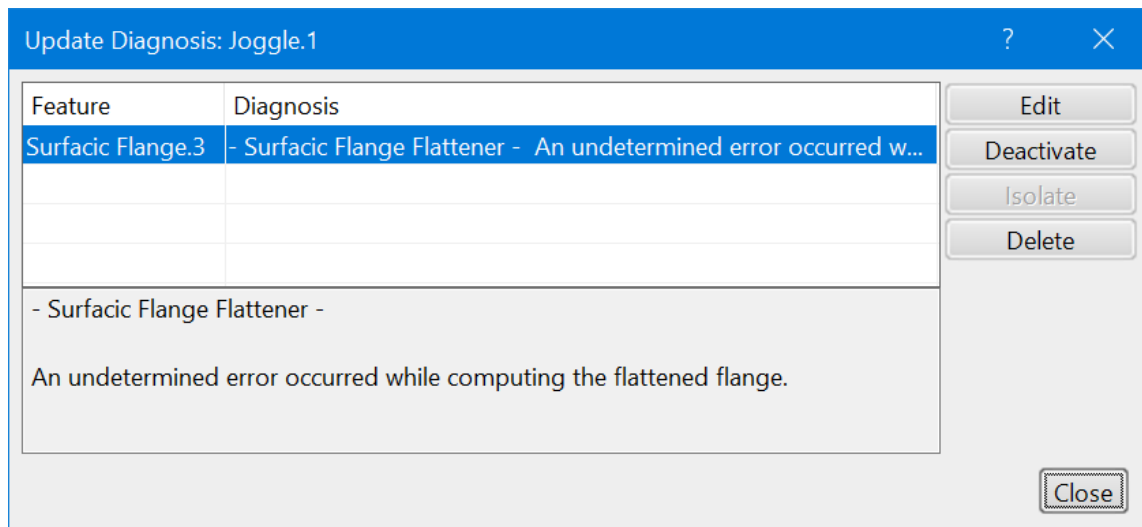
Select the web for the *Support* and *Plane.3* for the *Joggle Plane*. The preview should appear as shown.



Change the *Plane Position* to *Start*, then set the *Depth* to 0.25 and both the *Start Radius* and *End Radius* to 0.125.

Ensure the direction arrows match the image above.

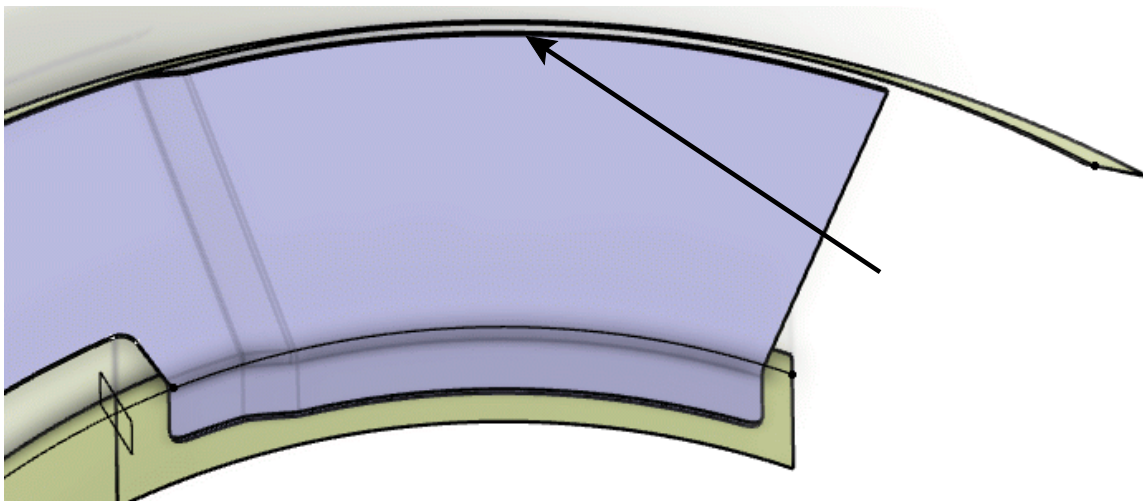
Select **OK**. An *Update Diagnosis* window appears.



The joggle caused *Surfacic Flange.3* to fail. This is the flange that was built from the angled swept surface.

Select **Close** in the window.

Rotating the model reveals a gap between the web and the angled swept surface. This is the reason the flange failed.



In order for the surfacic flange to be created, the web must always intersect with the support surface. You will need to modify the web in order to make it intersect with the swept surface, even after the joggle is added.

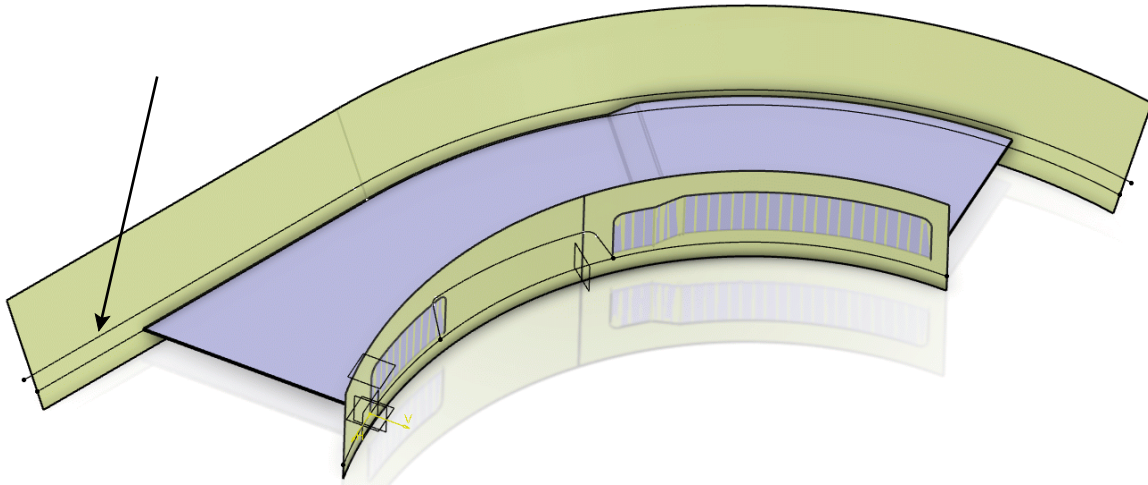
**Switch to the Generative Shape Design workbench.**

You will offset the original curve that was used to build the web, then swap it out in the web definition. This will extend the web so that it intersects with the swept surface after the joggle.

Select the **Parallel Curve** icon.  The *Parallel* window appears.

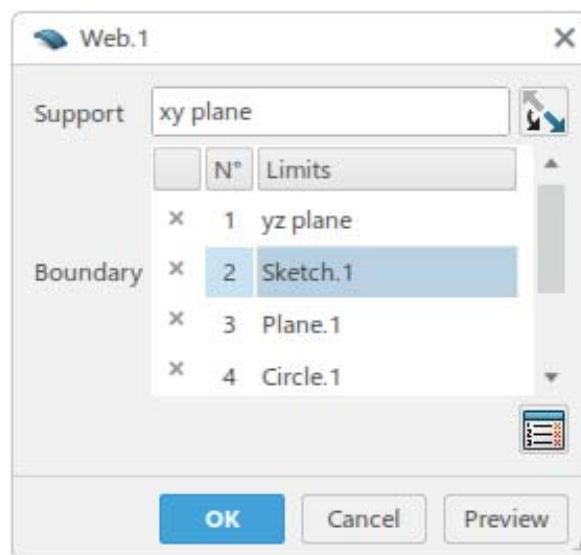
Select *Sketch.1* to define the *Curve* for the operation.

Ensure the direction is pointing away from the web, then set the *Distance* to 0.5 and click **OK**. The curve should appear as shown.



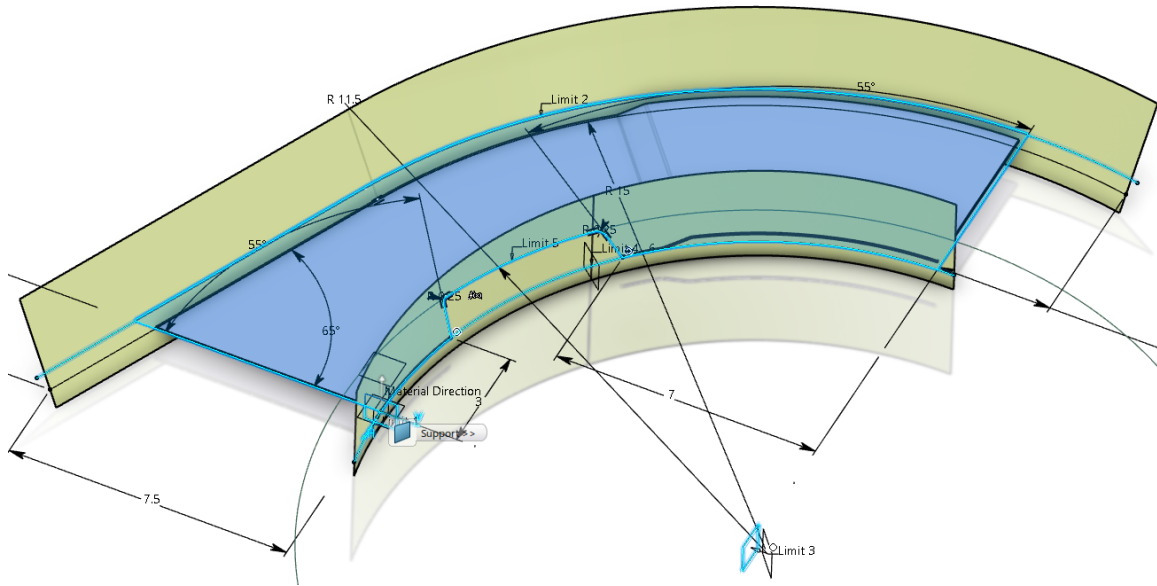
Double-select on *Web.1* in the specification tree. The *Web* window appears.

*Sketch.1* was the second element selected in the original *Boundary* definition of the web. You will replace it with the new parallel curve to extend the web outside of the swept surface.

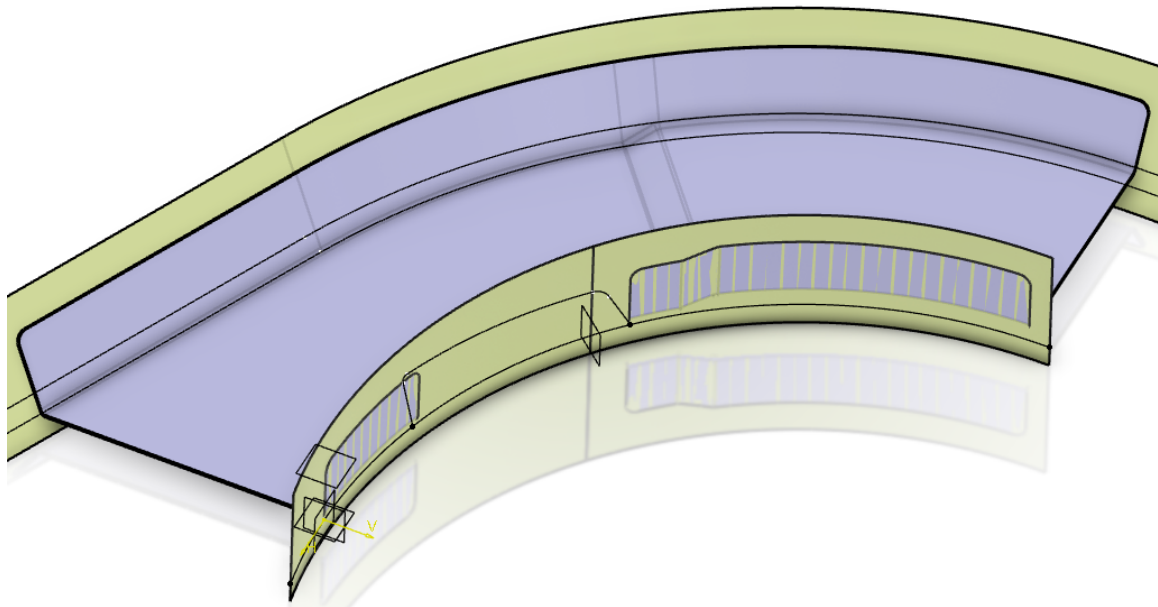


Select *Sketch.1* from the *Boundary* area in the *Web* window.

Right select on *Sketch.1* and select *Replace, 1 Limit* and select the new parallel curve. The preview updates.

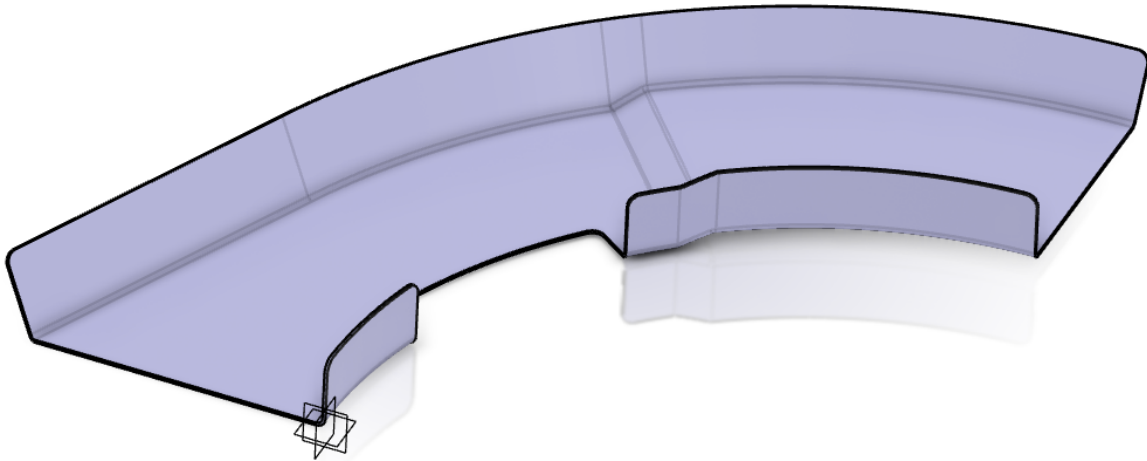


Select **OK**. The model updates, and should appear as shown.



The web now intersects with the swept surface, even after the joggle, so the surfacic flange is able to be created.

**Hide Geometrical Set.1.** The part should appear as shown.



**Save and close the document.**