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Introduction

CATIA Version 6 Wireframe & Surfaces

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

- Creating wireframe geometry
- Creating surfaces
- Performing operations on surfaces
- Modifying wireframe and surfaces
- Analyzing curves and surfaces
- Utilizing wireframe and surfaces in Part Design

Wireframe & Surfaces

Many parts can be created using just the Part Design tools. However, there are times when surfaces need to be used in order to get the desired shape for your part. Wireframe geometry is also necessary at times to define support geometry for the various Part Design tools as well as the surface tools. Surfaces provide the ability to create complex contours that are often necessary in your design. There are a few workbenches in CATIA V6 that have wireframe and surface options. This class will focus on the Generative Shape Design workbench. The Generative Shape Design workbench has all of the tools that are available on the Wireframe & Surfaces workbench and more. This course will cover all of the options found in the Generative Shape Design workbench.

As covered in previous courses, surfaces can be used within Part Design. This gives the capability of hybrid modeling. To review, you should remember that you can perform four operations with surfaces in Part Design. One option is to add thickness to a surface thereby creating a solid. A second option is to split your part with a surface. A third option is to sew a surface to your part, which will either add or remove material, or both. The last option is to close a surface with planar faces to form a solid. These options can be reviewed via the exercises located in Appendix B. It is also important that you understand how to work with boolean operations in order to fully utilize all of the surface options. These are reviewed in Appendix B as well.

It is important to understand some of the terminology that CATIA uses when working with wireframe and surfaces. You should already be familiar with a PartBody and know that you can have more than one within your part. Wireframe geometry and surfaces are created within geometrical sets. You may also have more than one geometrical set in your part. Geometrical sets are used to organize non-solid geometry. When you create new wireframe or surface geometry, you will need to be sure that the correct geometrical set is the in-work object in order to have an organized tree.

Wireframe

Wireframe geometry is critical to the creation of surfaces and is used as reference elements throughout CATIA.

Points

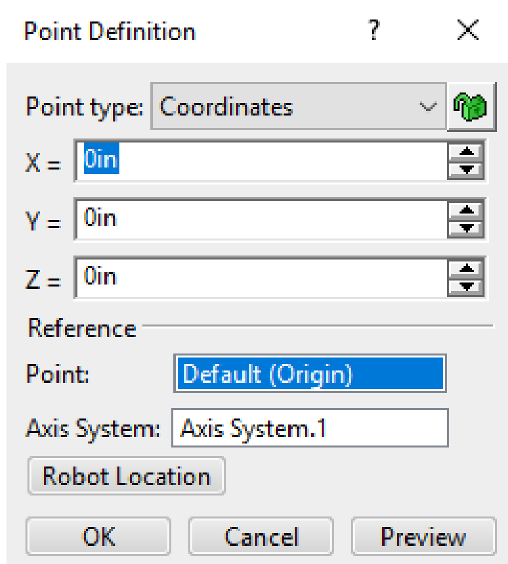
Points are useful to define specific locations and to assist in the creation of other wireframe geometry. You have a variety of options to define points that will be explored in the following exercises.

Coordinate

Open the WFAS - Points document. Remember, you will need to search for <u>WFAS - Points</u> and then open the document. You should see a surface and some wireframe geometry.

Switch to the Generative Shape Design workbench if are not already there. To change workbenches, you can select the My 3D Modeling Apps (West quadrant) section of the compass, then choose *Generative Shape Design*.

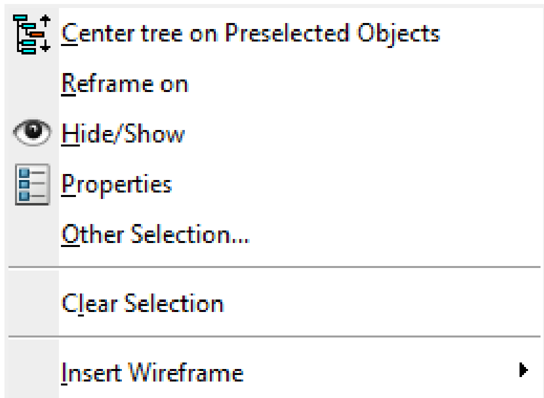
Select the Point icon. The *Point Definition* window appears.



Point type	Specifies what type of point you want to create: *Coordinates*, *On curve*, *On plane*, *On surface*, *Circle/Sphere/Ellipse center*, *Tangent on curve*, or *Between*
X=, Y=, Z=	The coordinate values of the point to be created from the reference point

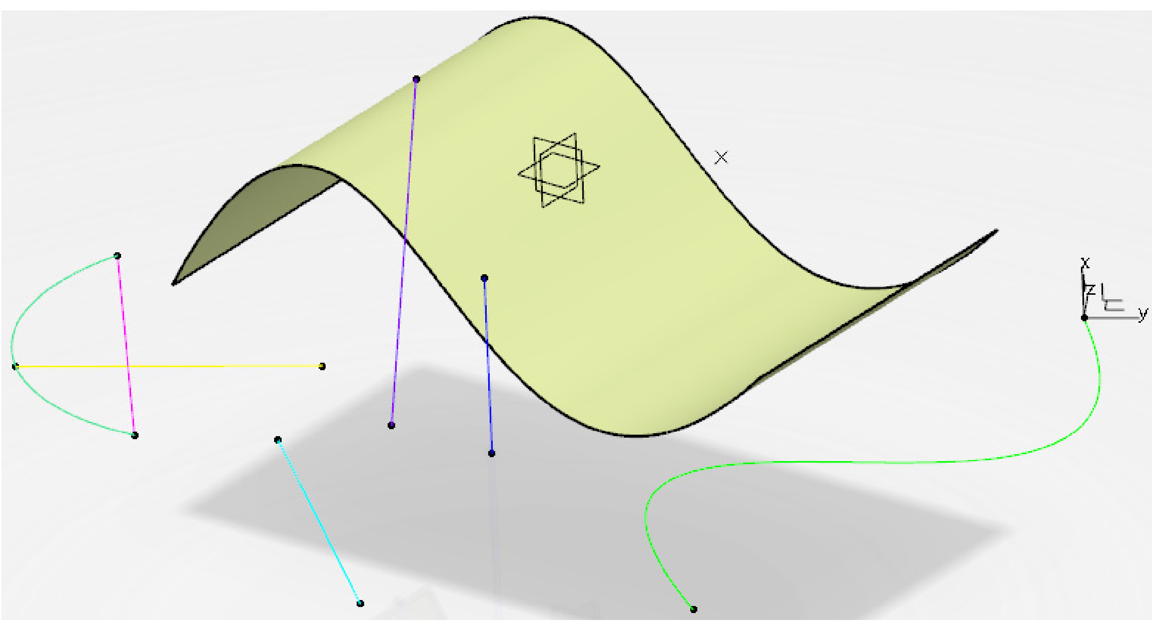
Reference	
Point	The point that the coordinates are based from; the default is the origin
Axis System	Defines the axis system that the point will be based off of
Robot Location	Creates the point at the location of the robot in the display

Press the third mouse button in the *Axis System* field. A contextual menu appears.



Choose *Clear Selection*. By default, the active axis system is used to create a coordinate point. By clearing the *Axis System* field, the absolute axis system will be used to define the point instead.

Enter 2.0, 4.0, and 2.0 for the *X*, *Y*, and *Z* values respectively, then select *OK*. You should have noticed a preview of the point as you were entering the values before selecting *OK*. It should appear similar to the diagram shown below.



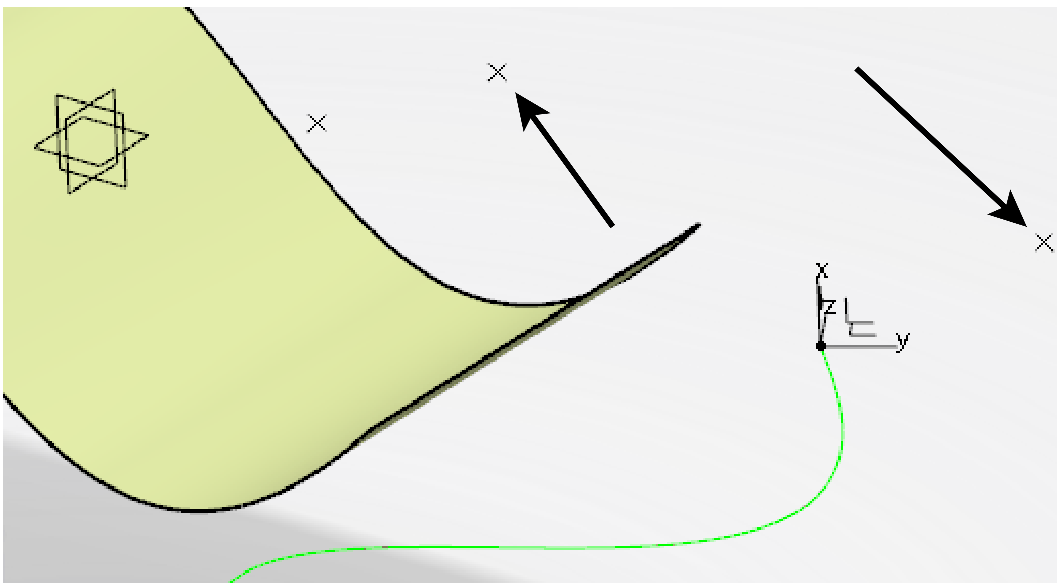
Select the Point icon again. The *Point Definition* window appears. This time, you will use a point as the reference instead of the origin.

Clear the *Axis System* selection box so that the absolute axis system will be used, then select the point you just created to define the *Reference Point*. The point is labeled as *Point* and the name of the point appears in the *Point* field of the window.

Enter 0.0, 2.0, and 1.0 for the *X*, *Y*, and *Z* values respectively, then select *OK*. The new point is created off of the previous point rather than the origin.

Select the Point icon again. The *Point Definition* window appears. This time, you will use the axis system that has been created instead of using the absolute axis system.

Enter 1.0, 2.0, and 0.0 for the *X*, *Y*, and *Z* values respectively, then select *OK*. The point is created off of the origin of *Axis System.1* instead of the absolute axis system. Your model should appear similar to the diagram shown below.

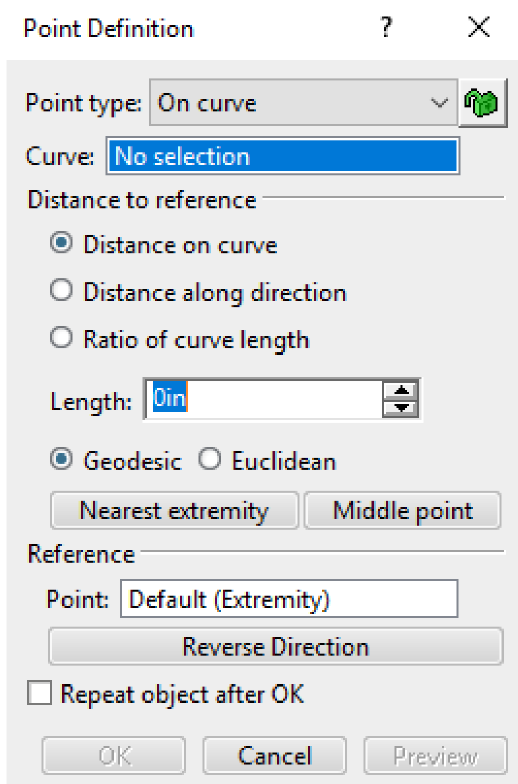


You can also create points on elements such as curves, planes, and surfaces.

On curve

Select the Point icon. The *Point Definition* window appears.

Change the *Point type* to *On curve*. The options change as shown here.



Curve	Specifies the curve on which the point will be created
Distance to reference	Determines the mode to use for the point creation
Distance on curve	The distance along the curve from the reference point
Distance along direction	The distance along the curve in a particular direction
Ratio of curve length	The ratio between the reference point and the extremity
Length/Offset/Ratio	A user-defined value to specify either the *Length* for the *Distance on curve* option, the *Offset* for the *Distance along direction* option, or the *Ratio* for the *Ratio of curve length* option
Geodesic	Forces the length to be measured along the curve

Euclidean	Corresponds to *Distance on curve* only; the length is the absolute value from the reference point
Nearest extremity	Creates the point at the nearest extremity
Middle point	Creates a point in the middle of the curve

Reference

Point	Allows you to specify a reference point if you do not want to use an extremity for the reference
Reverse Direction	Reverses which side of the reference the point is created on, or which extremity is used

Repeat object after OK Allows you to create multiple, equidistant points

Select the curve on the right. The curve is labeled *Curve* in the display and the extremity shows a red arrow.

Select the *Distance on curve* option, then change the *Length* to 5.0 and make sure it is set to *Geodesic*. The point appears at five inches along the curve.

Select the *Euclidean* option instead of *Geodesic*. The point is still on the curve, but it is now five inches from the reference point instead of five inches along the curve.

Select *OK*. The point is created.

Select the Point icon again and ensure the *Point type* is set to *On curve*, then select the curve on the right. 

Set the *Reference Point* to be the origin of the axis system at the end of the curve.

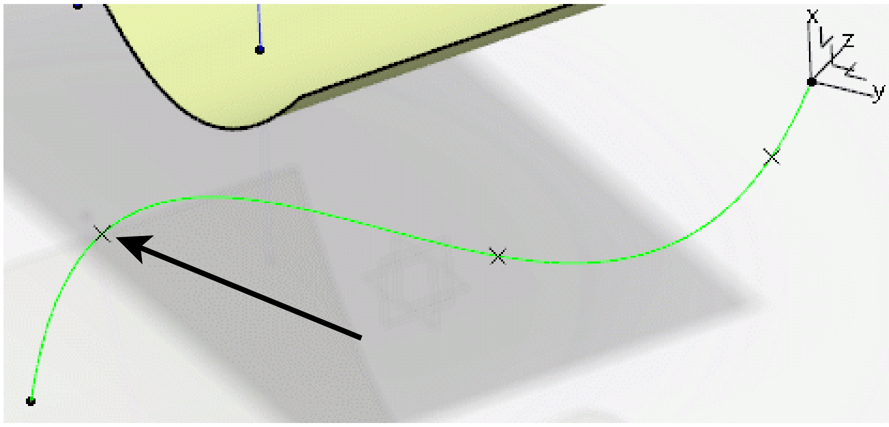
Select the *Distance along direction* option, then select the *yz plane* from the specification tree for the *Direction*. This will be where the offset is measured from.

Change the *Offset* to 1.0 and select *OK*. The point is created and is measured normal from the yz plane along the curve.

Select the Point icon again and ensure the *Point type* is set to *On curve*, then select the curve on the right. 

Select the *Ratio of curve length* and *Geodesic* options, then change the *Ratio* to 0.25 and select *Preview*. A point appears a quarter of the way along the curve. A ratio of 0.5 is the midpoint of the curve. Only the *Distance on curve* option can use a *Euclidean* measurement.

Select *OK.* The point is created and should appear similar to the diagram shown below.



Select the Point icon again and ensure the *Point type* is set to *On curve*, then select the curve on the right. 

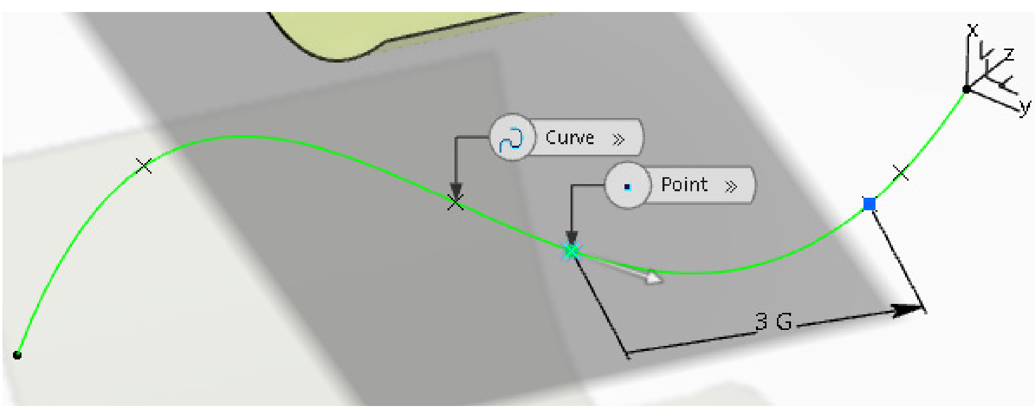
Select the *Distance on curve* option, then select the *Nearest extremity* button. The point appears at the nearest endpoint of the curve.

Select the *Middle point* button and click *OK.* The point appears in the middle of the curve.

Select the Point icon again and ensure the *Point type* is set to *On curve*, then select the curve on the right.  This time, you will use a reference point other than an extremity.

Select *Distance on curve*, then choose *Geodesic* and change the *Length* to 3.0.

Select the *Reference Point* field and choose the point shown below. Notice the direction of the arrow.



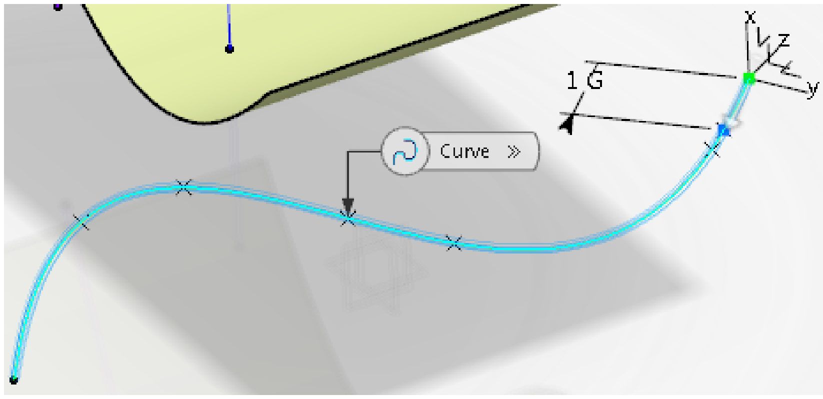
Select the *Reverse Direction* button. The arrow points to the opposite direction. If using an extremity, reversing the direction causes the reference point to switch to the other end of the spline.

Select *OK.* The point is created.

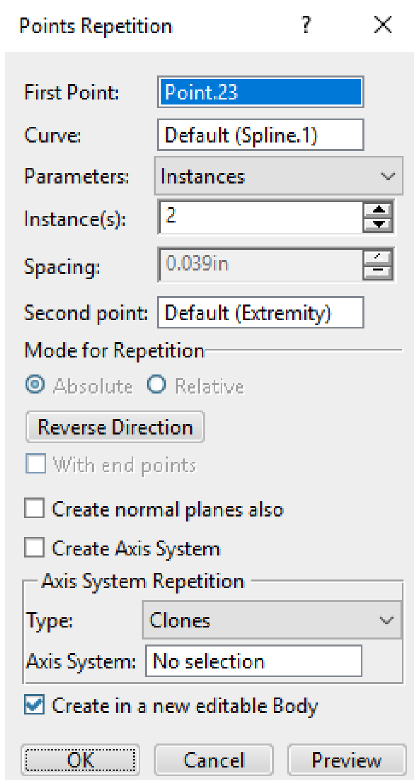
Select the Point icon again and ensure the *Point type* is set to *On curve*, then select the curve on the right.

Choose *Distance on curve* and *Geodesic*, then change the *Length* to 1.0.

Select the *Reverse Direction* button. This moves the reference to the other end of the curve.



Turn on the *Repeat object after OK* checkbox and select *OK*. The *Points & Planes Repetition* window appears.



Select *Cancel*. These options will be covered in detail later in the book. Only the one point is created. This completes the options for creating a point on a curve.

Select in space to release the point.

Surfaces

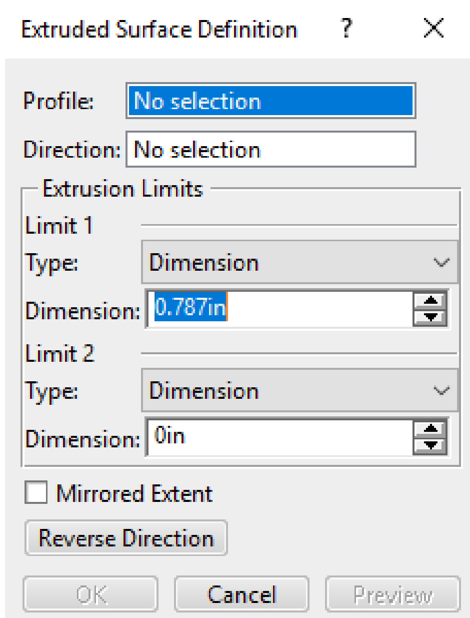
Surfaces are extremely important for defining contours. With the use of wireframe geometry, surfaces can be created to represent any contour needed. Once the surfaces are created, they can then be used in Part Design to make a solid model. There are a variety of options for creating surfaces. Some are straightforward, while others are much more complex.

Extruded

Extruded surfaces are created by extending an element in a linear direction. The resulting object is called an *Extrude* in the specification tree.

Open the WFAS - Basic Surfaces document. You should see some wireframe geometry.

Select the Extrude icon from the *Surface* section.  The *Extruded Surface Definition* window appears.



Profile — Specifies the shape to be extruded

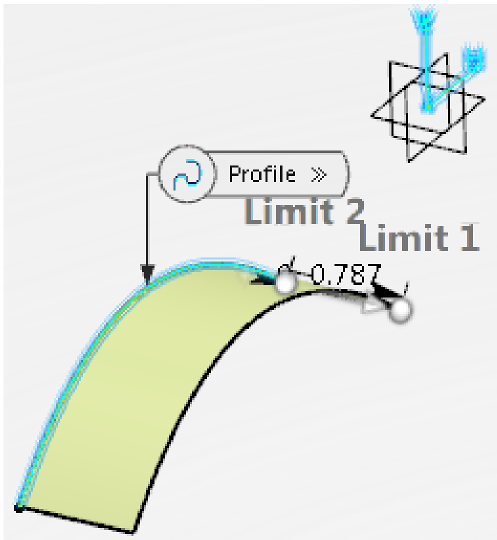
Direction — Defines the direction of the extrusion

Extrusion Limits

- *Limit 1/2* — Defines a distance or a limiting element
  - *Type* — Two options available: *Dimension* and *Up-to element*
    - *Dimension* — Specifies a distance for the extrusion to extend

	Up-to element	Specifies an element that the extrusion will stop at
Mirrored Extent	Forces *Limit 2* to be the same as *Limit 1*; only available when the *Type* is set to *Dimension*	
Reverse Direction	Reverses the direction of the extrusion	

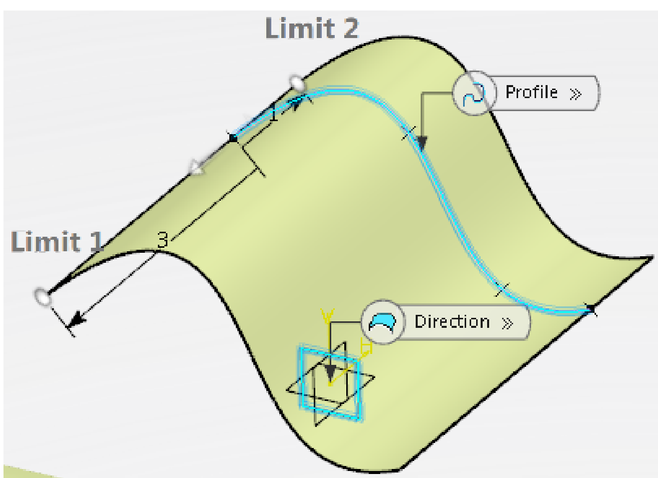
Select the curve shown below. This curve was created in a sketch. Whenever a sketch is selected for an extrusion, the direction will automatically be normal to the sketch plane.



Enter 3.0 for *Limit 1* and 1.0 for *Limit 2*, then select the *Reverse Direction* button and click *OK*. The surface is created.

Select the Extrude icon again.  The *Extruded Surface Definition* window appears.

Select the curve and plane as shown below. When a plane is selected for the direction, the surface will extend normal to the plane.

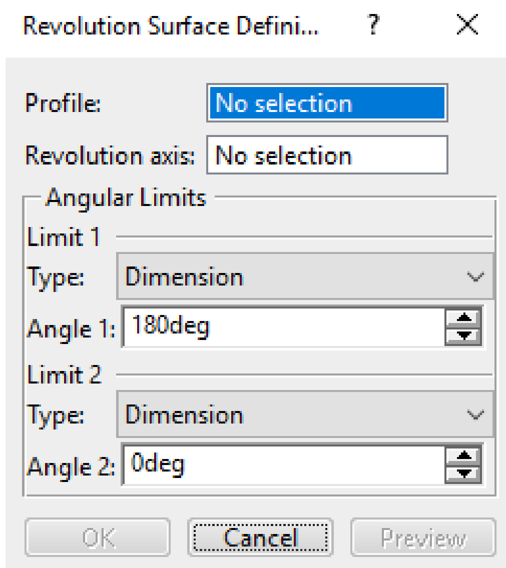


Change both *Dimension* fields to 1.0 and select *OK*. The surface is created.

Revolution

Revolution surfaces are created by rotating an element around an axis. The resulting object is called a *Revolute*.

Select the Revolve icon.  It is located within the sub-toolbar of the Extrude icon. The *Revolution Surface Definition* window appears.



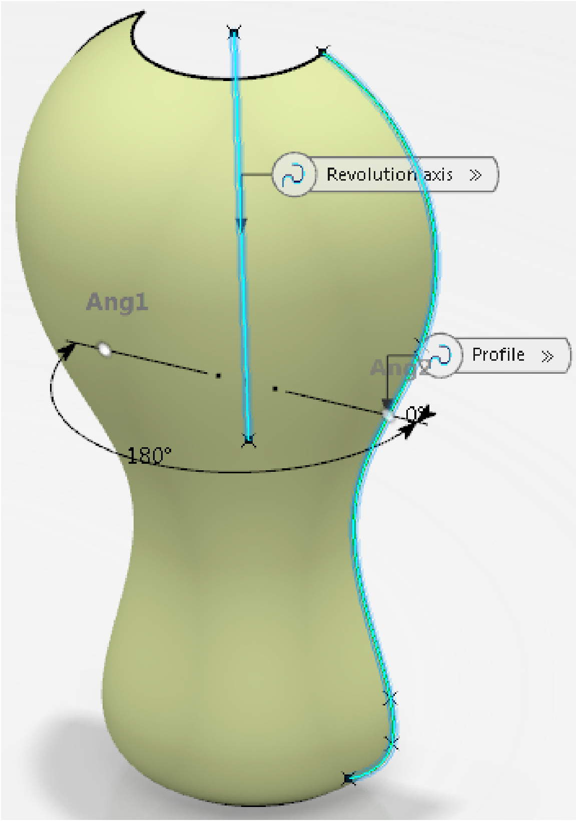
Profile — Specifies the shape that will be revolved

Revolution axis — Defines the axis around which the profile will revolve; if the profile is a sketch that has an axis defined within it, CATIA will use that axis for the revolution

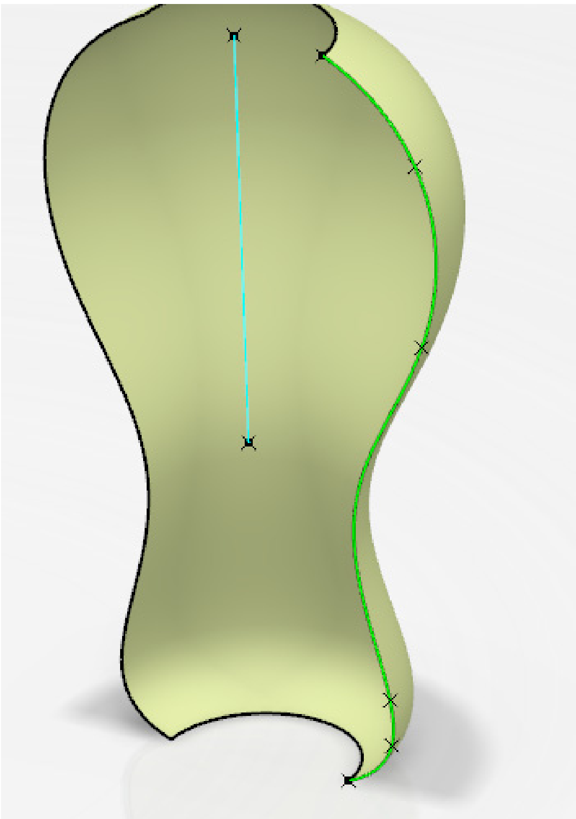
Angular Limits

- *Angle 1* — Defines the starting angle for the revolution
- *Angle 2* — Defines the ending angle for the revolution

Select the profile and line as shown below.



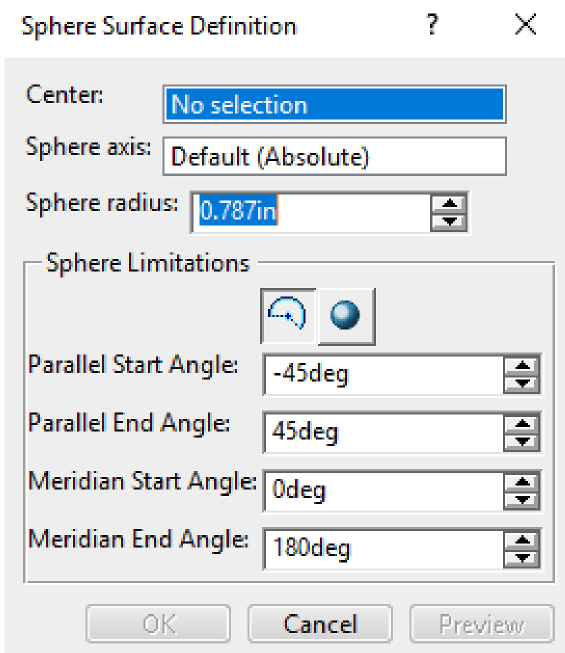
Change *Angle 1* to 0.0 and *Angle 2* to 180, then select *OK*. The surface is created.



Sphere

Spherical surfaces are created by defining a center point and a radius. The resulting object is called a *Sphere* in the specification tree.

Select the Sphere icon.  It is located within the sub-toolbar of the Extrude icon. The *Sphere Surface Definition* window appears.



Center — Specifies the center point of the sphere

Sphere axis — Determines the orientation of the *Parallel* and *Meridian* curves

Sphere radius — Defines the radius of the sphere

Sphere Limitations

 Angles — Creates a partial sphere

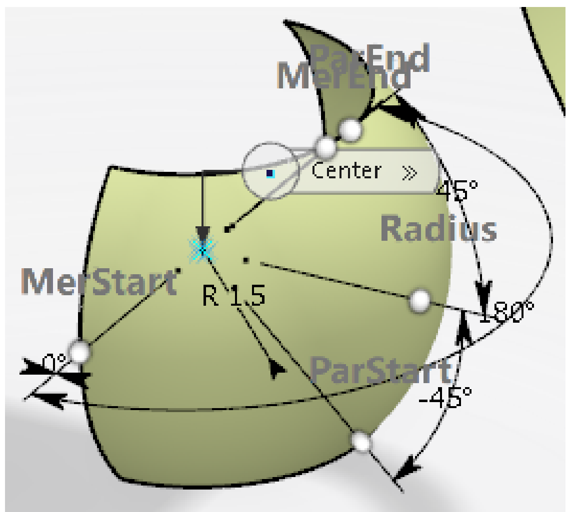
 Whole Sphere — Creates a full sphere

Parallel Start Angle — Defines the starting angle in the parallel direction; only available when the ...Angles icon is selected

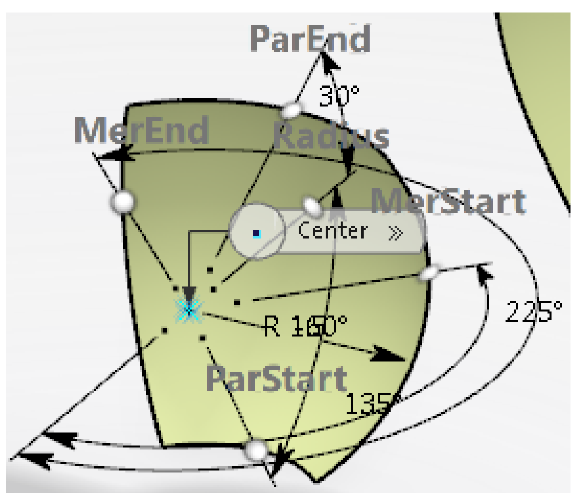
Parallel End Angle — Defines the ending angle in the parallel direction; only available when the ...Angles icon is selected

Meridian Start Angle	Defines the starting angle in the meridian direction; only available when the ...Angles icon is selected
Meridian End Angle	Defines the ending angle in the meridian direction; only available when the ...Angles icon is selected

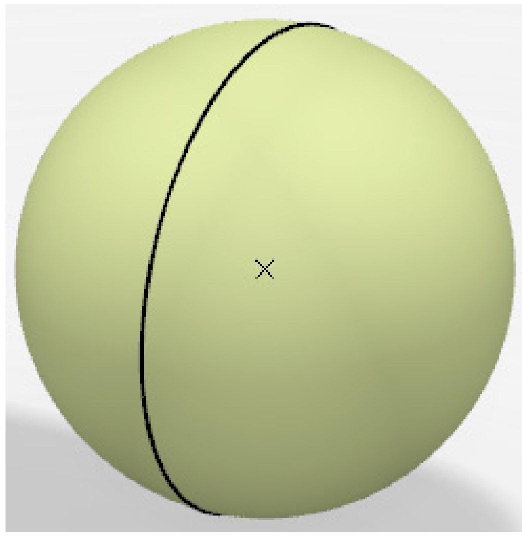
Select *Point.6* either graphically or from the tree and enter 1.5 for the *Sphere radius*, then select *Preview*. Since you do not have an axis to select, you will use the default. The *Parallel* limits have a range of -90 degrees to 90 degrees, while the *Meridian* limits have a range of -360 degrees to 360 degrees. Basically, the *Parallel* limits are up and down, while the *Meridian* limits are side to side. This will depend on the axis selected, however.



Change the *Parallel Start Angle* to -60, the *Parallel End Angle* to 30, the *Meridian Start Angle* to 135 and the *Meridian End Angle* to 225, then select *Preview*.



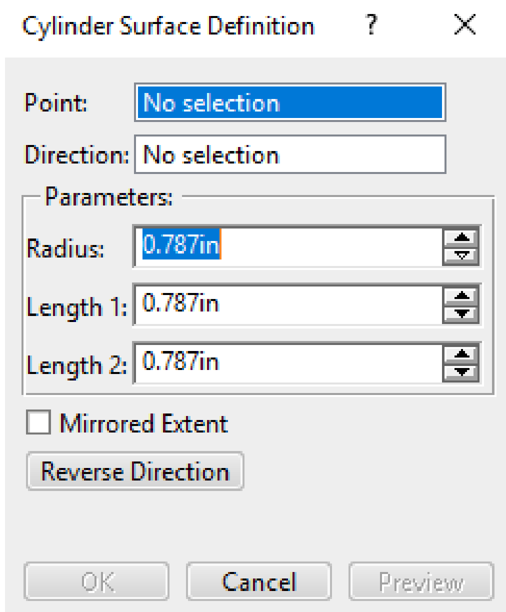
Select the Whole Sphere icon from the window and click *OK.*  All the limit options became unavailable.



Cylinder

Cylindrical surfaces are created by defining a point, a direction, a radius, and a length. The resulting object is called a *Cylinder* in the tree.

Select the Cylinder icon.  It is located within the sub-toolbar of the Extrude icon. The *Cylinder Surface Definition* window appears.



Point Specifies the center point of the cylinder

Direction Specifies the direction the cylinder will extrude

Parameters

Radius Defines the radius of the cylinder

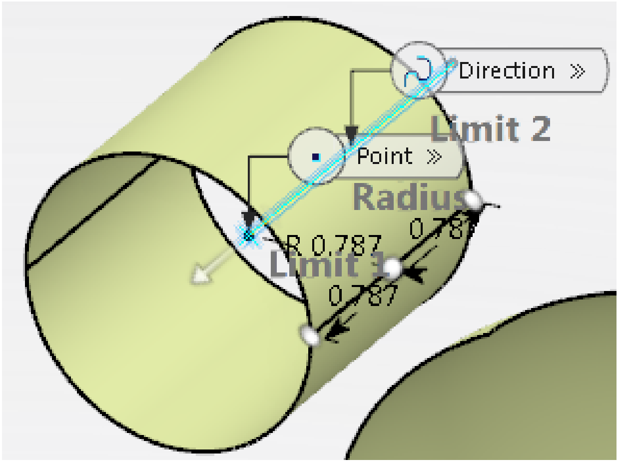
Length 1 Defines the length of the cylinder in the first direction

Length 2 Defines the length of the cylinder in the second direction

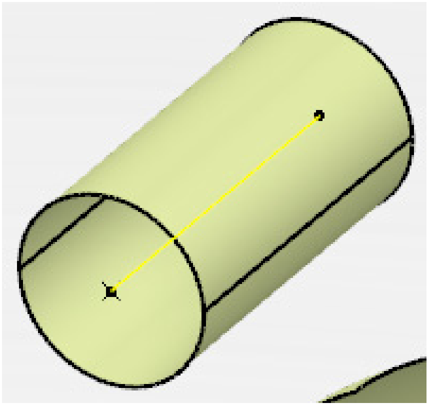
Mirrored Extent Changes *Length 2* to be the same as *Length 1*

Reverse Direction Reverses the direction of the cylinder

Select the point and the line as shown below.



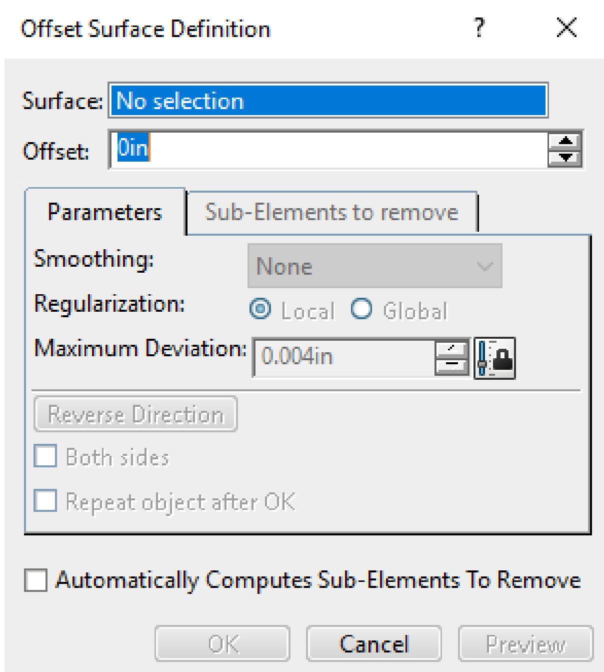
Change the *Radius* to 0.5, *Length 1* to 2.0, and *Length 2* to 0.0, then select the *Reverse Direction* button and click *OK*. A cylindrical surface is created.



Offset

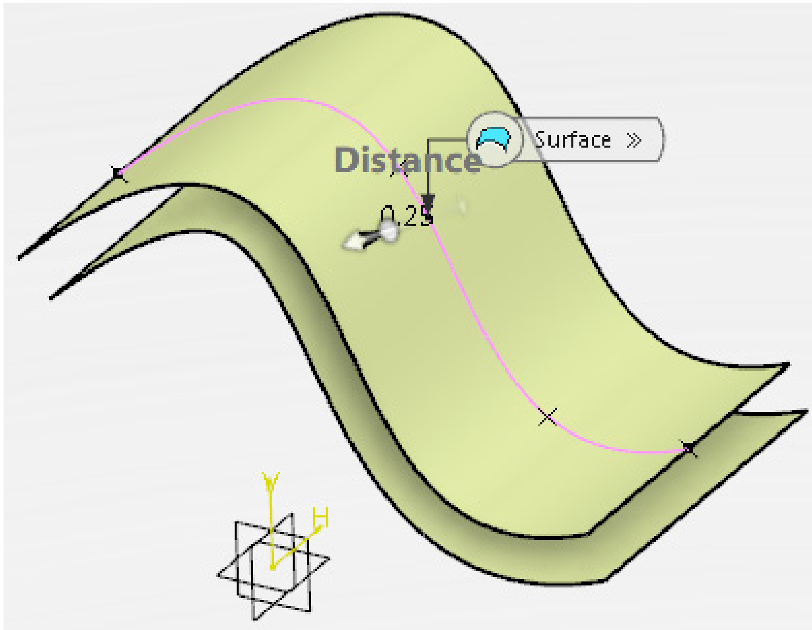
Offset surfaces are created by offsetting an existing surface by a specified distance. Offsets are always extended normal to the original element. The resulting object is called an *Offset*.

Select the Offset icon from the *Transform* section.  The *Offset Surface Definition* window appears.



Surface	Specifies the surface to be offset
Offset	Defines the distance of the offset
Parameters	
Smoothing	Creates deviation in the surface in order to assist in creating the offset
Maximum Deviation	Defines the maximum amount the new surface can vary from the original
Reverse Direction	Reverses the direction of the offset
Both sides	Offsets the surface in both directions
Repeat object after OK	Repeat the offset numerous times
Sub-Elements To Remove	Excludes problematic surfaces from the offset; they are added to a list under the *Sub-Elements To Remove* tab. Sub-elements can be added to the list or removed from it in order to determine which element is causing the offset to fail.

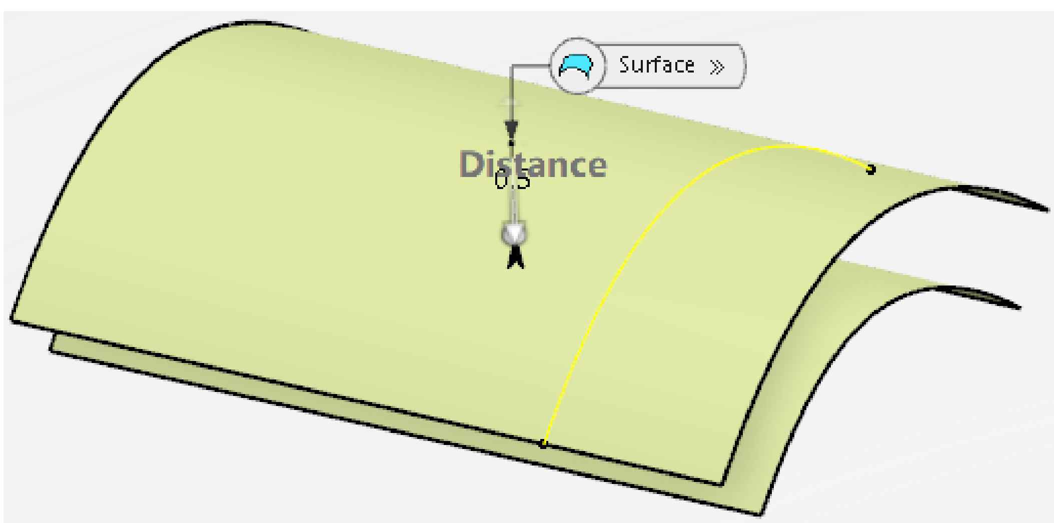
Select the surface as shown below, then enter 0.25 for the *Offset* and select *Preview*. An offset surface appears.



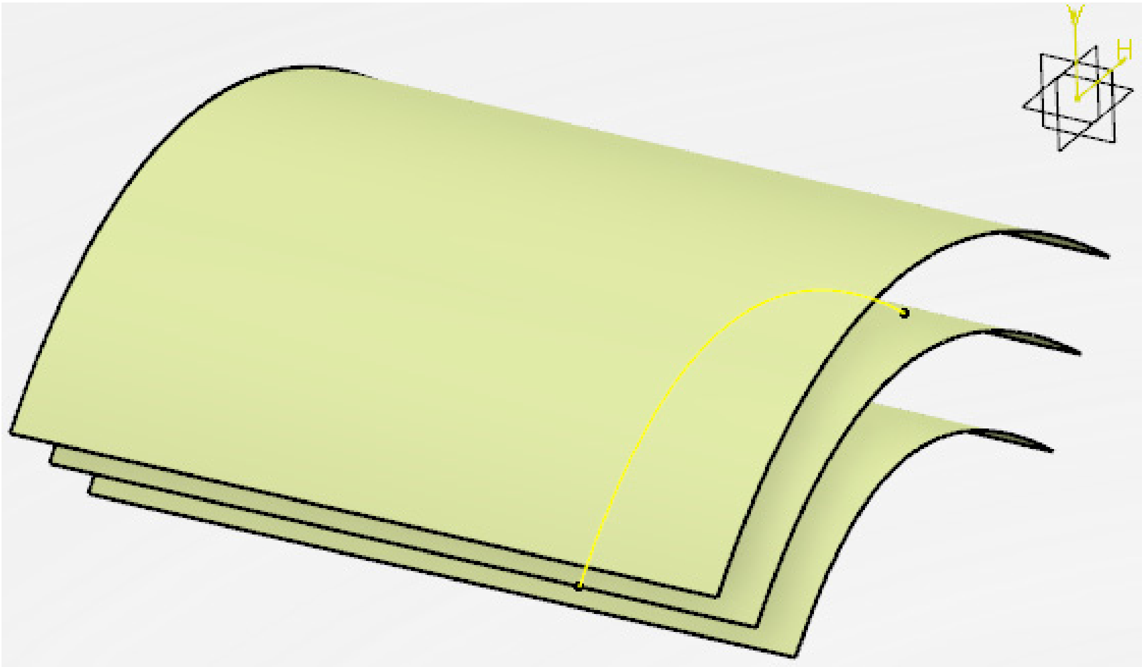
Select the *Reverse Direction* button and click *OK*. The offset surface appears above the original surface instead of below it.

Select the Offset icon again.  The *Offset Surface Definition* window appears.

Select the surface shown below and enter 0.5 for the *Offset*, then select *Preview*. An offset surface appears below the original.



Turn on the *Both sides* option and select *OK*. Offset surfaces appear above and below the original surface.



Note: Since the offset surface has a Repeat object after OK option in its definition window, you can use the Object Repetition *icon to duplicate it, if desired.* 

Save and close your document.

Review

For this review exercise, you will create a computer mouse. The intention of the exercise is to demonstrate the process of building a solid model by utilizing wireframe and surface geometry.

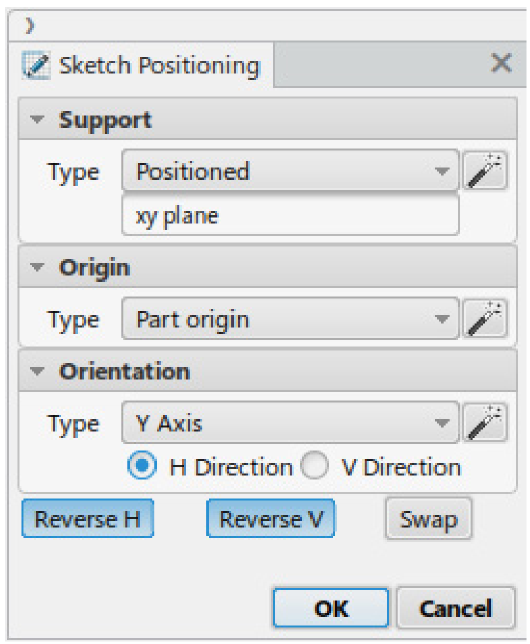
Note: Set your view mode to Shading With Edges Without Smooth Edges in order to obtain the same results shown in the following images.

Mouse Body

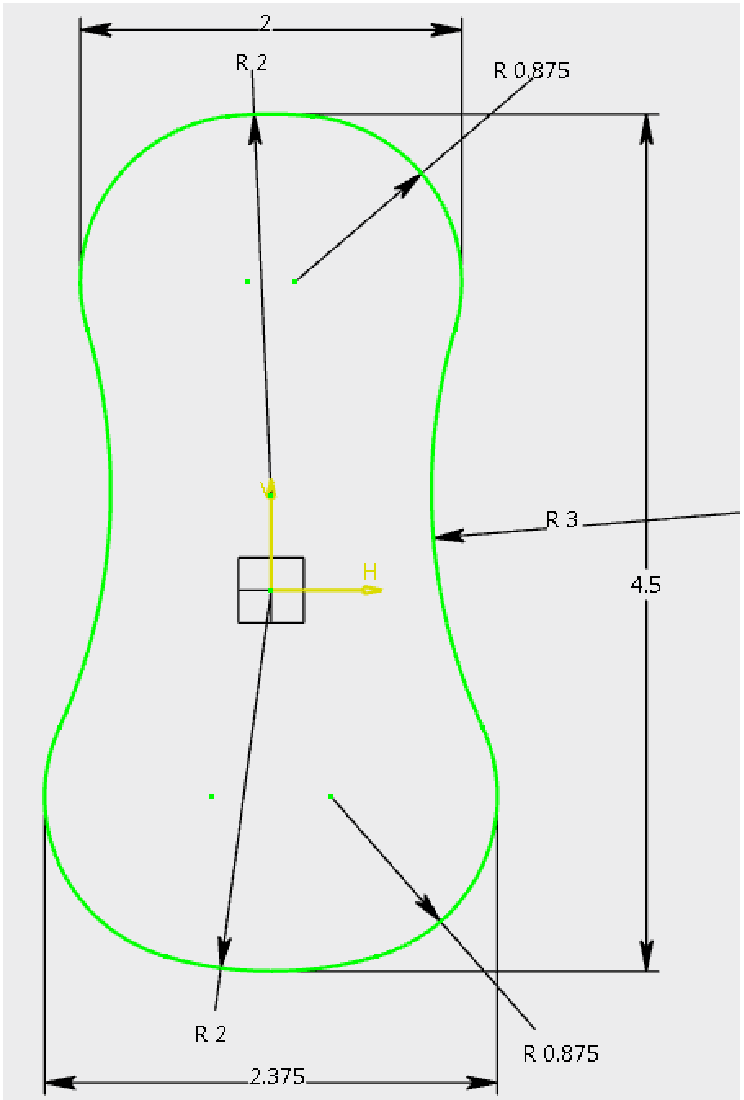
You will first create the mouse body, followed by the buttons and wheel.

Create a new 3D part.

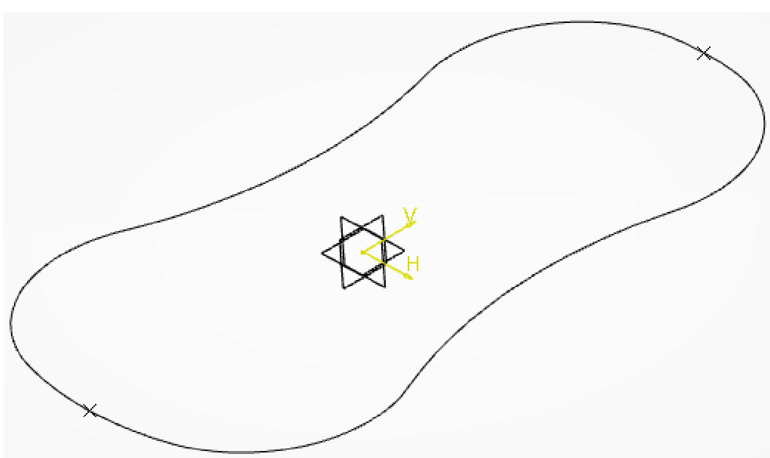
Insert a geometrical set named Mouse Body, then select the Positioned Sketch icon and set the options as shown below.



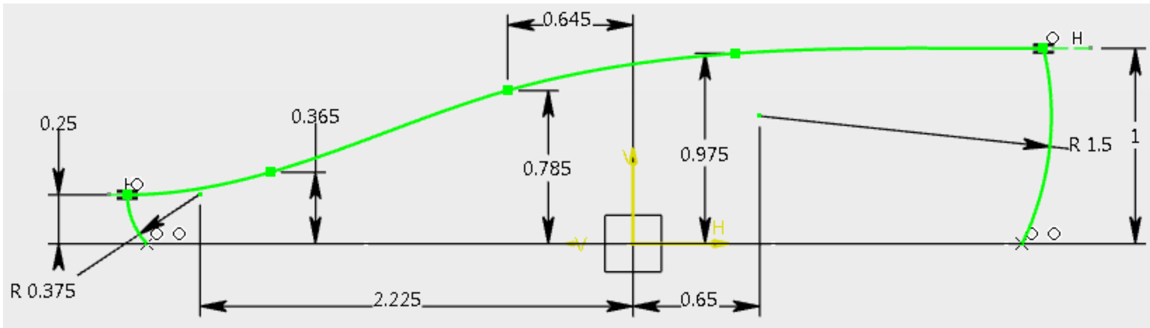
Create the following sketch. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.



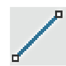
Create an extremum point at each end of the sketch. 

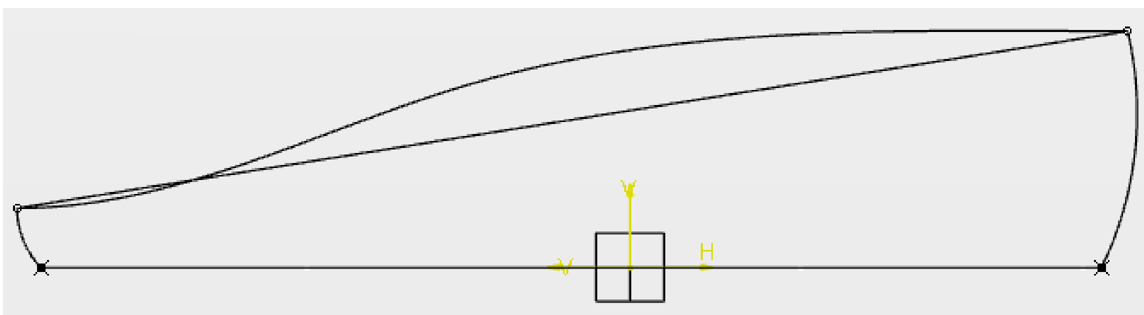


Create the following sketch on the zx plane.  The bottom of the arcs are coincident to the extremum points. Be sure the taller end of this sketch is towards the wider end of the first sketch.

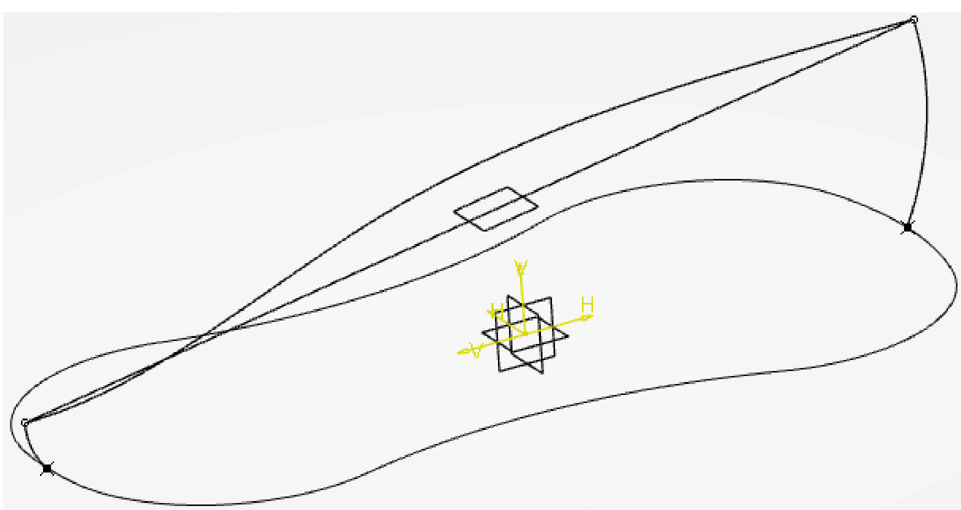


Extract each curve from the sketch. 

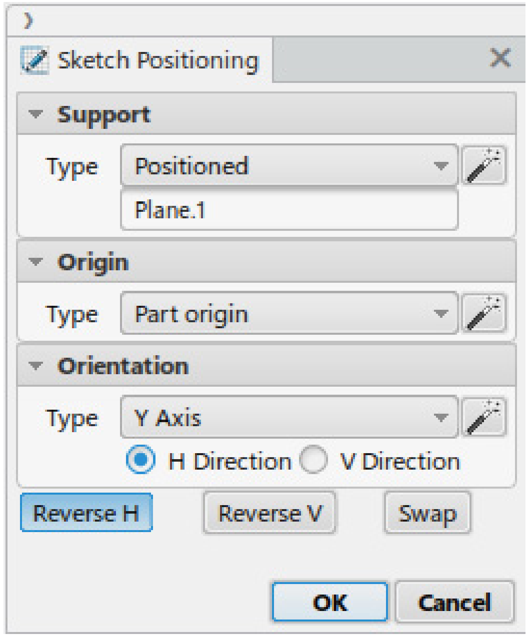
Create a *Point-Point* line between the top points of the arcs. 



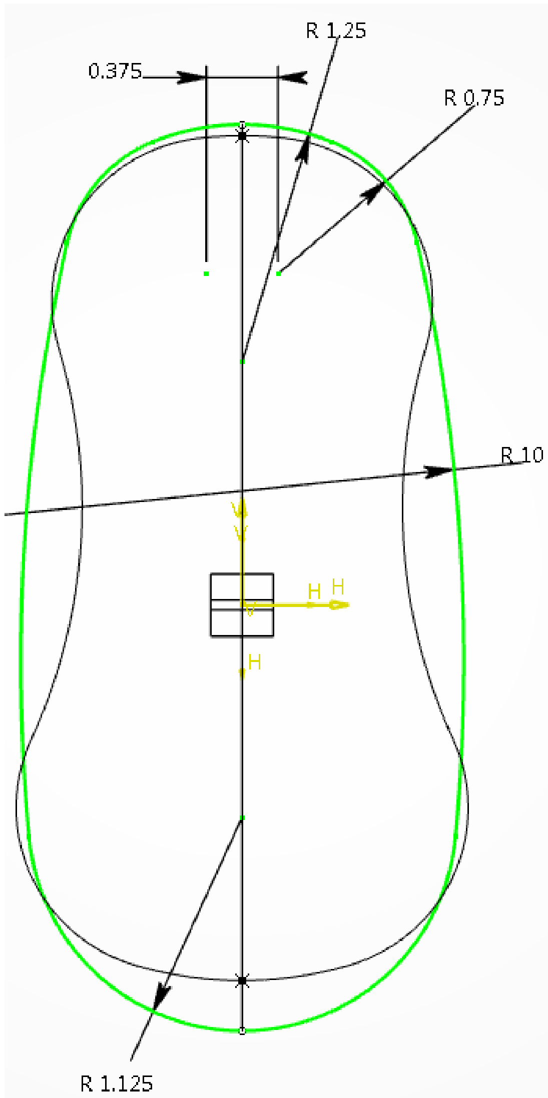
Create a plane through the line.  Use the line for the *Rotation axis* and the zx plane for the *Reference*. The plane should be normal to the reference.



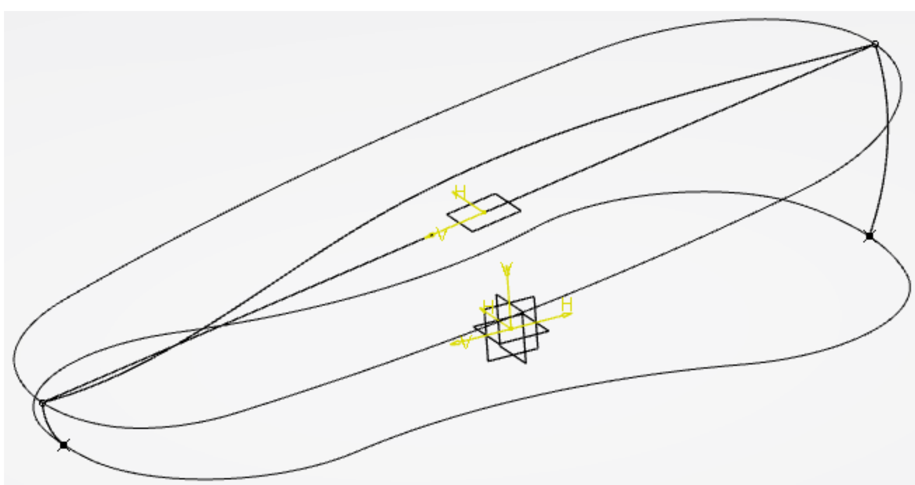
Select the Positioned Sketch icon and set the options as shown below. 



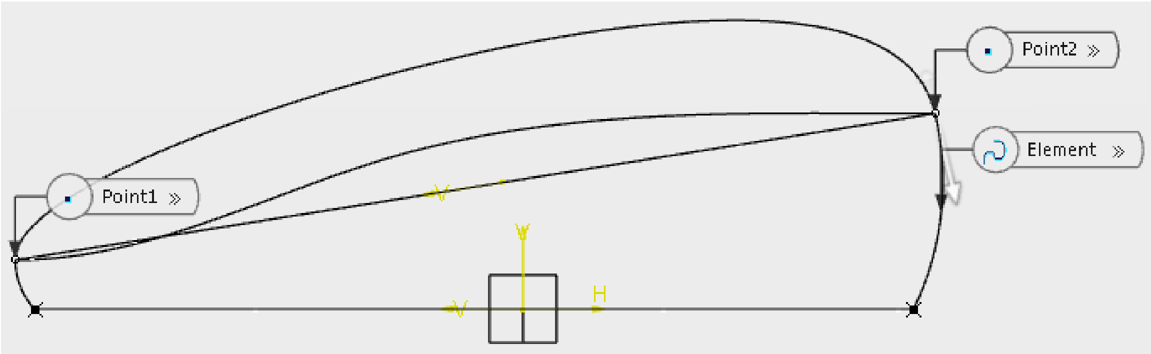
Create the following sketch. The top and bottom arcs in this sketch are coincident to the upper end points of the extracted arcs. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.



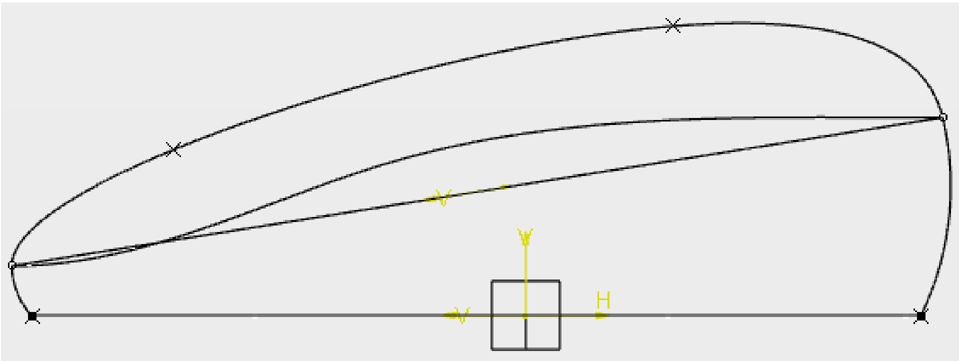
Your model should look like this.



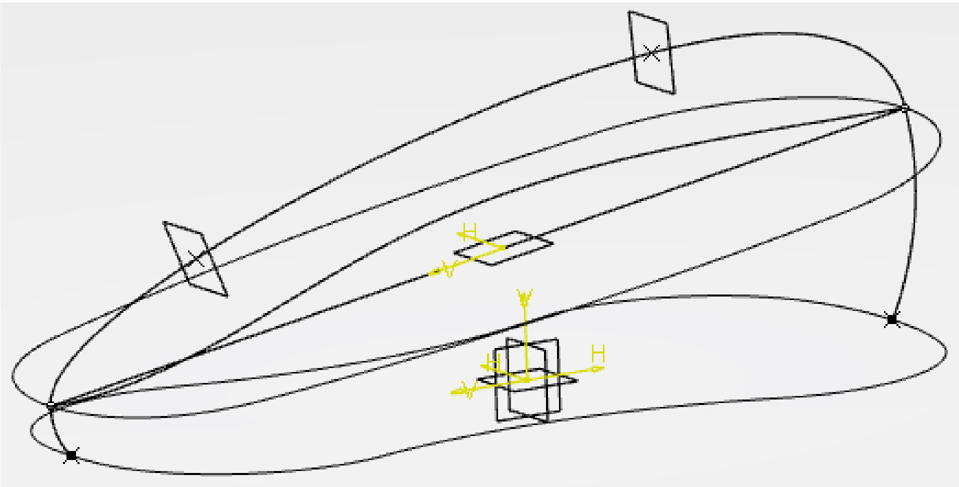
Create a spline between the two points at the top of each extracted arc. The spline will be tangent continuous to both arcs with a tension of 0.375 at the first point, and 0.75 at the second point.



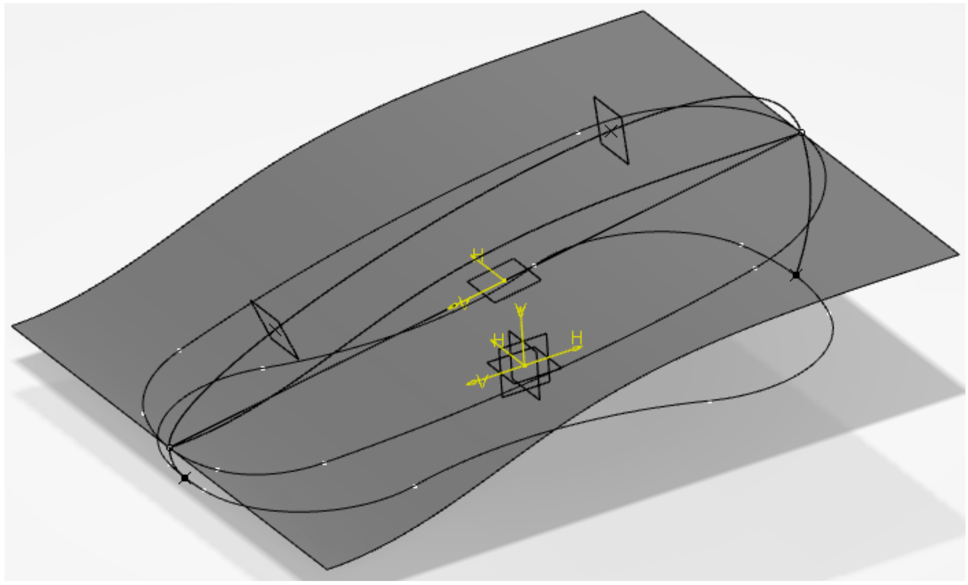
Create two geodesic points on the new spline. The left point will have a ratio value of 0.2 from the left end of the spline, and the right point will have a ratio value of 0.3 from the right end of the spline.



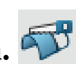
Create a plane normal to the upper spline at both points.

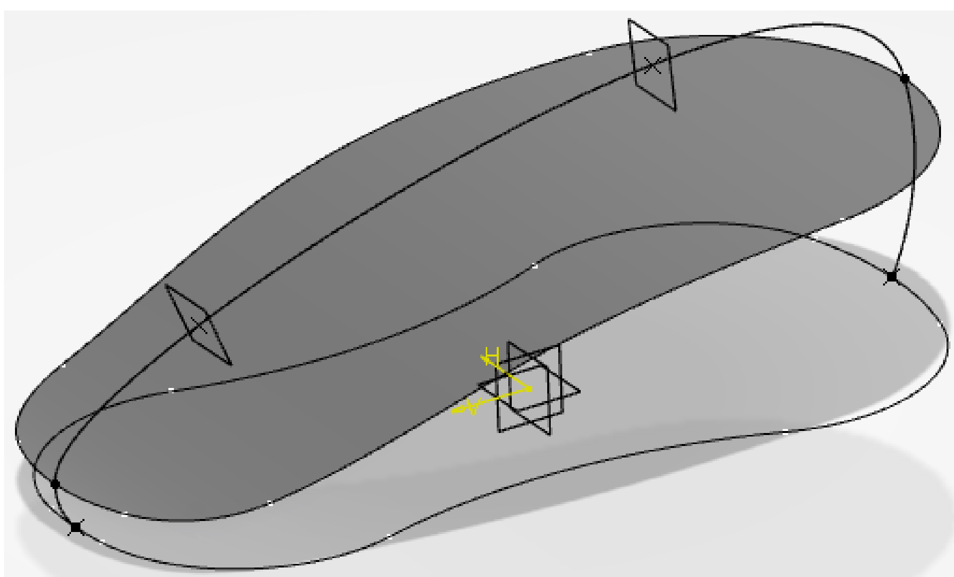


Extrude the extracted spline two inches in both directions normal to the zx plane, then change the name of the extrude to <u>CHANNEL SURFACE</u> in the specification tree. 

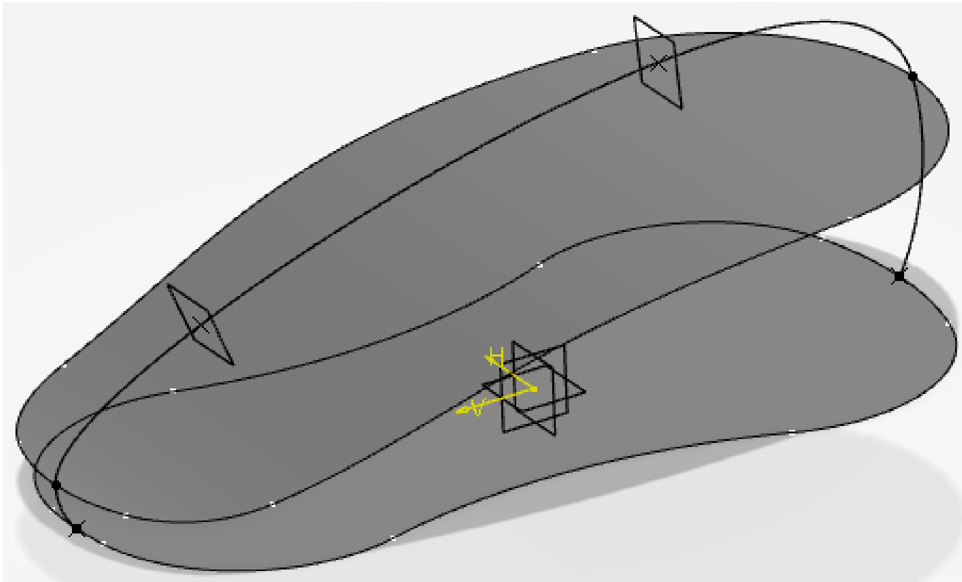


Project the last sketch you created to *CHANNEL SURFACE* along the normal direction of the first plane created. 

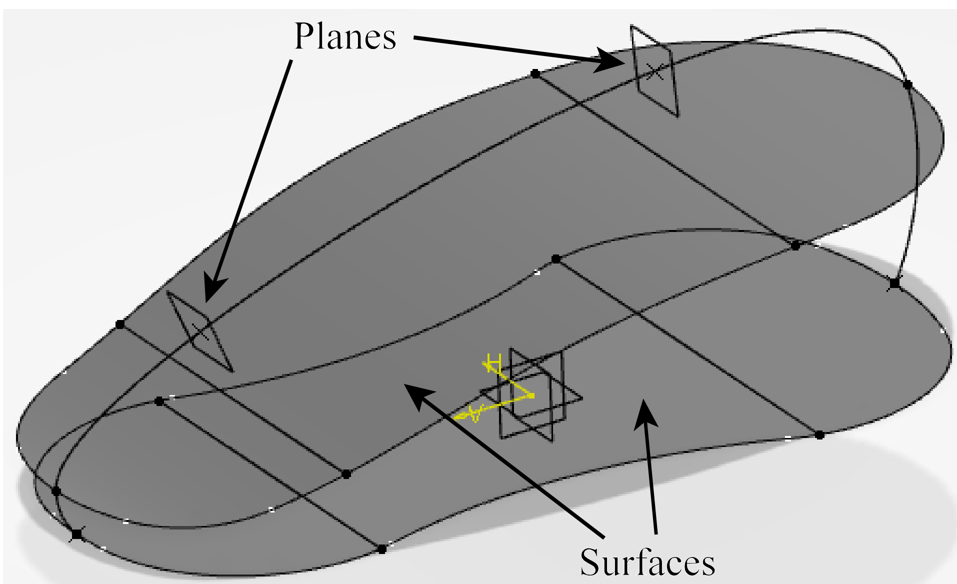
Split *CHANNEL SURFACE* with the projected curve, keeping the inner portion. 
The split surface is shown below. Much of the geometry has been hidden. Feel free to hide your geometry as necessary to reduce clutter.

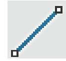


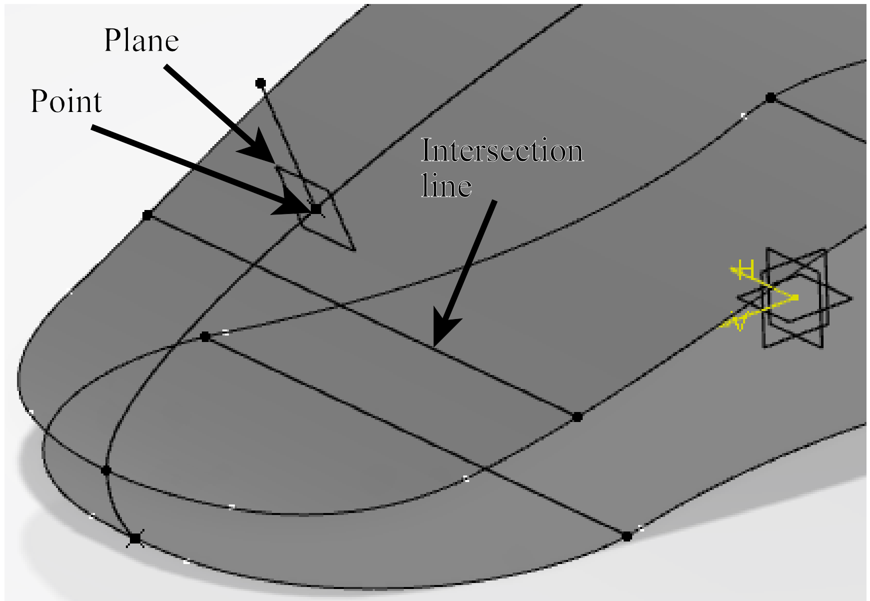
Fill your first sketch with a surface.  This is the bottom profile of the mouse.

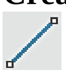


Create an intersection line between both planes and surfaces shown below.  You should have four, separate intersection lines.

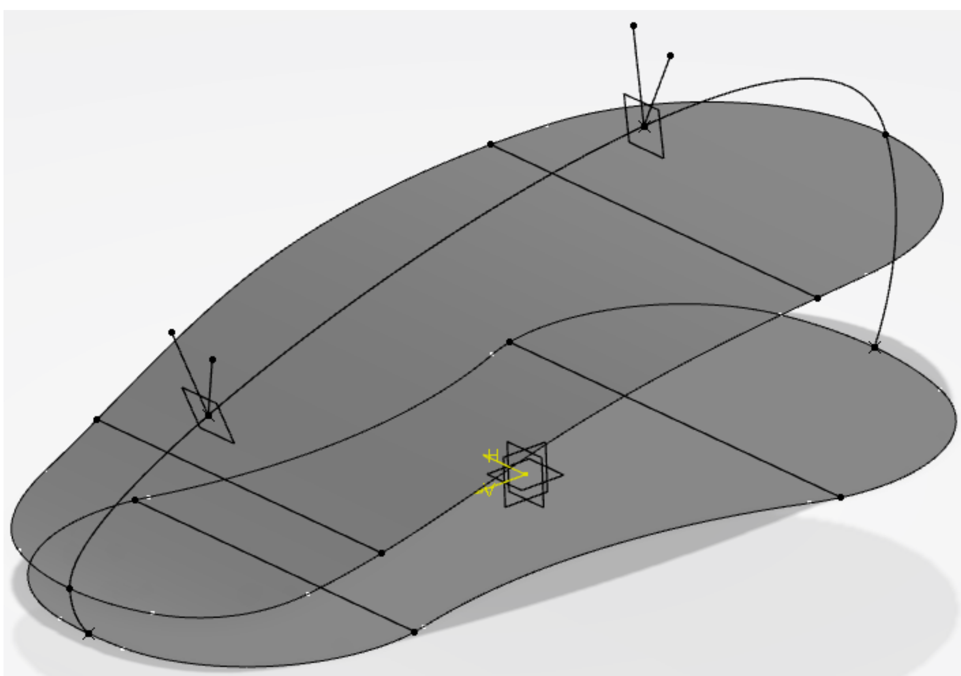


Create a 0.5 inch line normal to the intersection line indicated below that starts at the point and uses the plane as its support. 

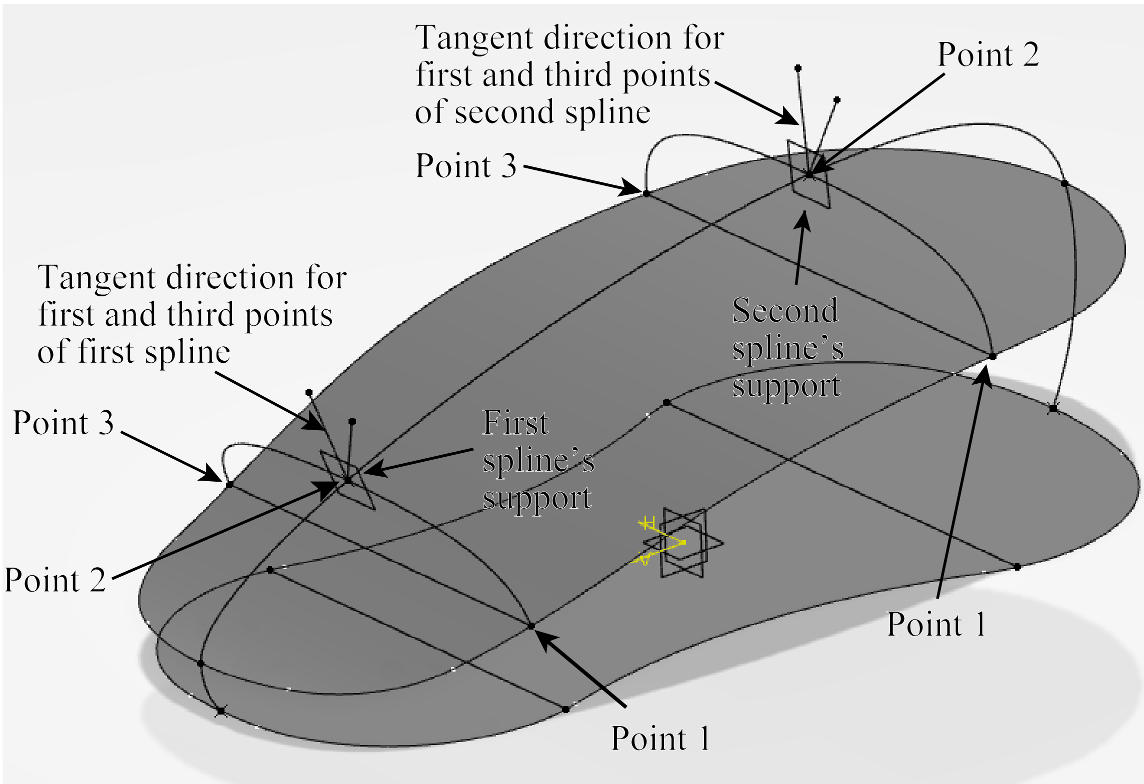


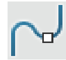
Create another line using the same geometry, but this time specify an *Angle* of -60.
 Ensure the line extends upward.

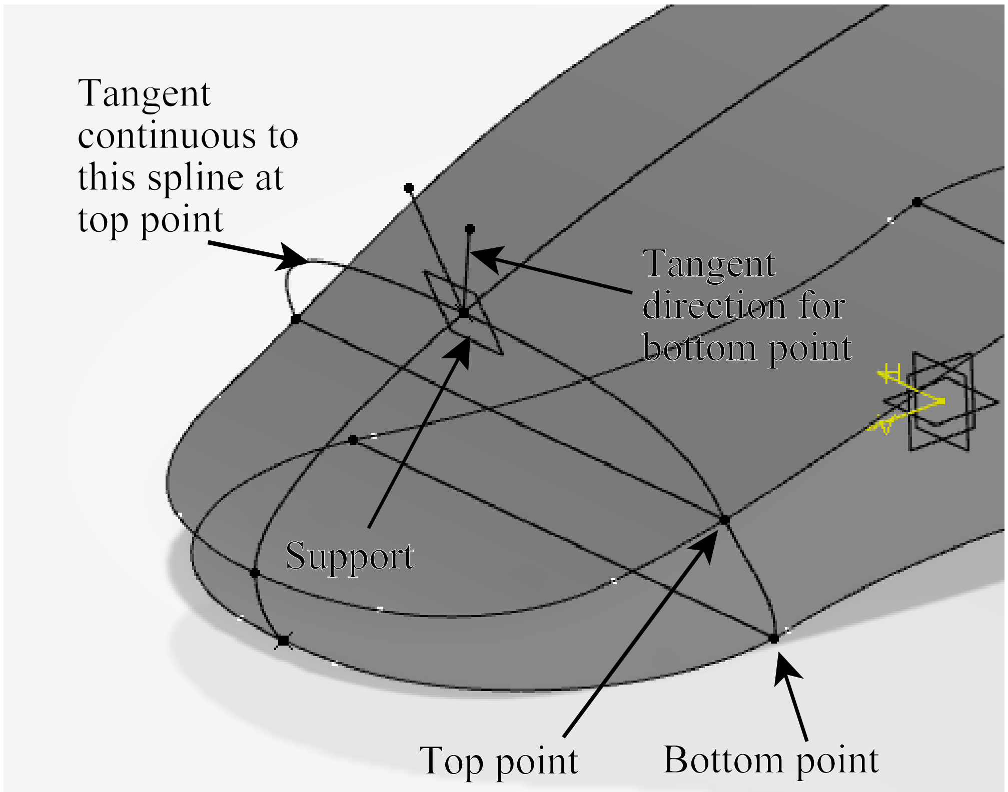
Repeat this process at the other point so that your model looks the same as below.



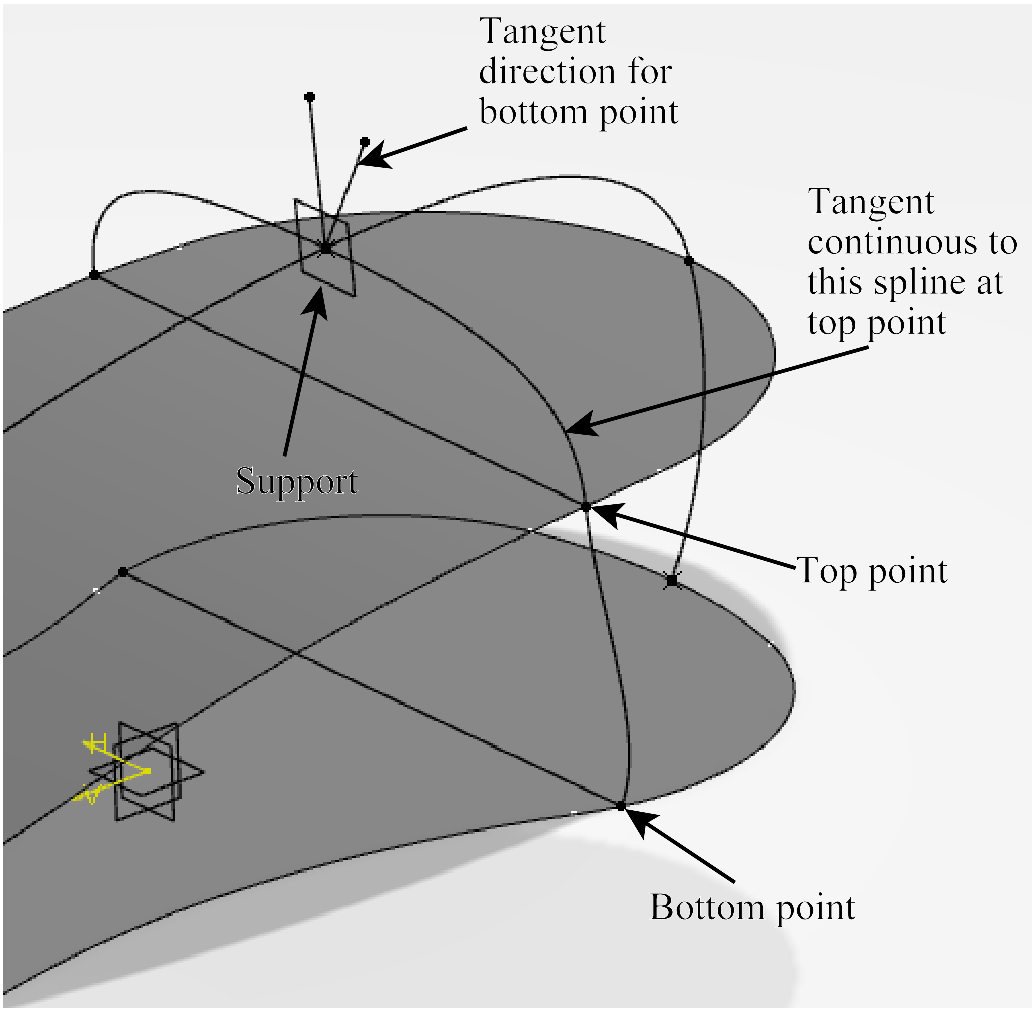
Create the two splines shown below.  The tension at all three points for both splines should be 1.0. Use the normal lines you just created for the tangent direction of the first and third points, and the zx plane for the tangent direction of the second points. Ensure each spline lies on the appropriate support plane indicated below.



Next, create the spline shown below.  This spline is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.5 at the bottom point. Ensure the spline lies on the support plane indicated below.

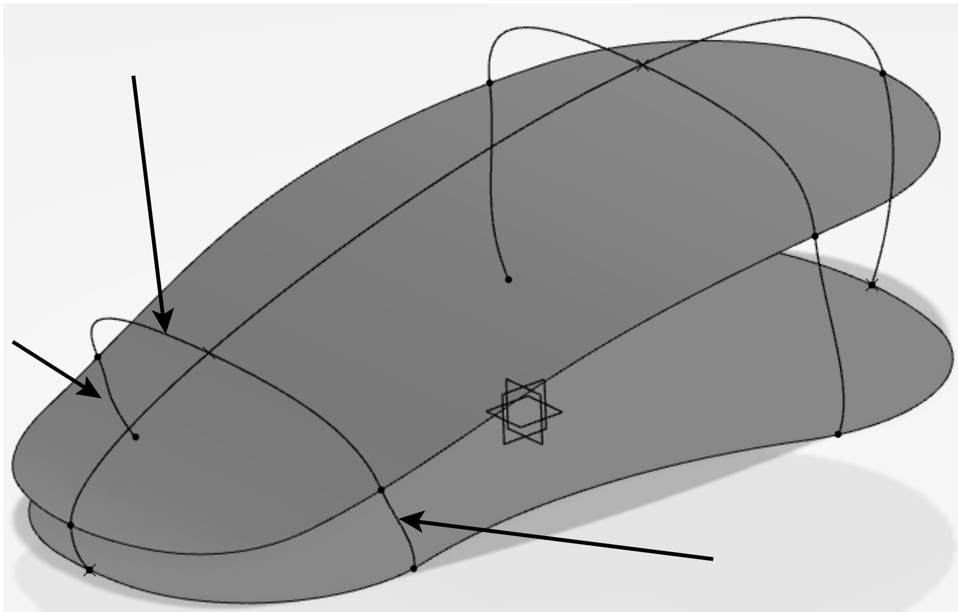


Create the spline shown below using the same method as the previous spline. It is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.25 at the bottom point. Ensure the spline lies on the support plane indicated below.

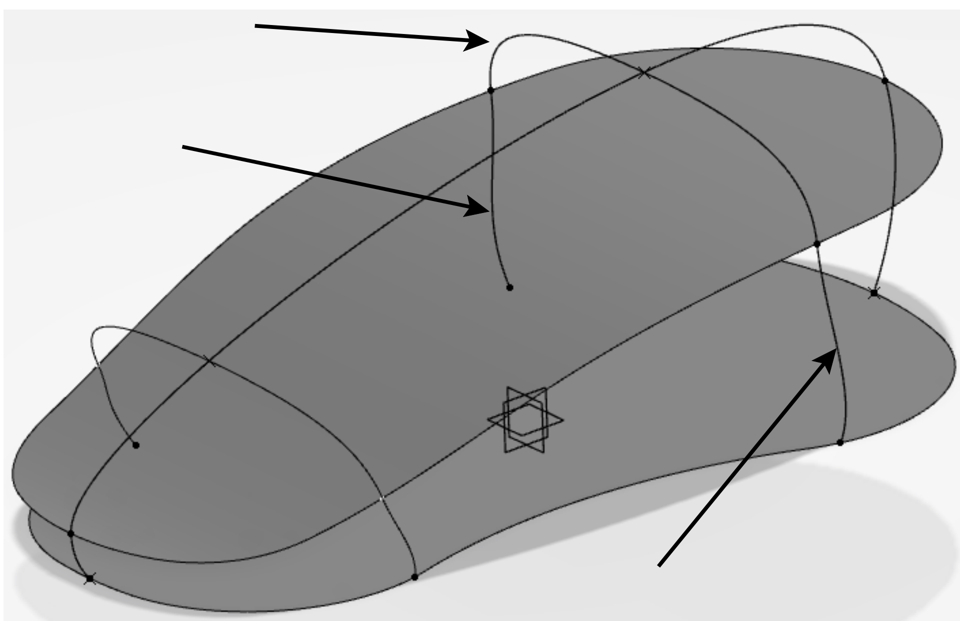


Mirror each of the last two splines created across the zx plane.

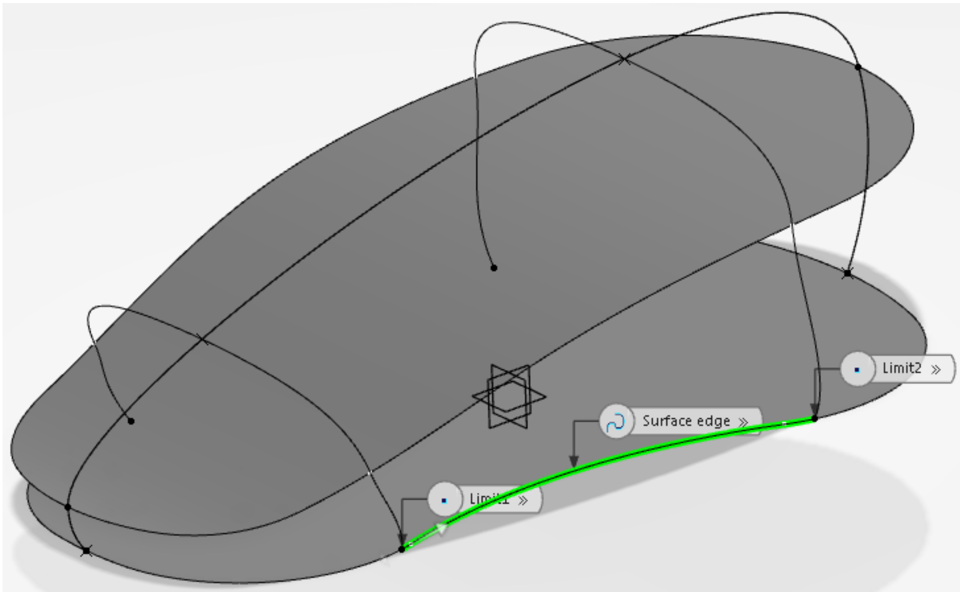
Join the three curves indicated below. Ensure they are tangent continuous.



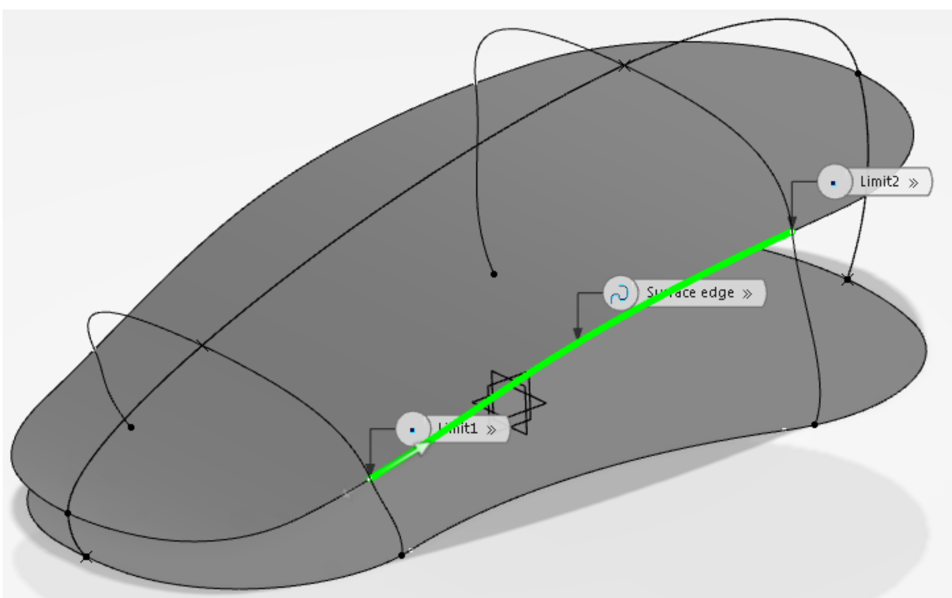
Create another join for the three curves shown below. Ensure they are tangent continuous.



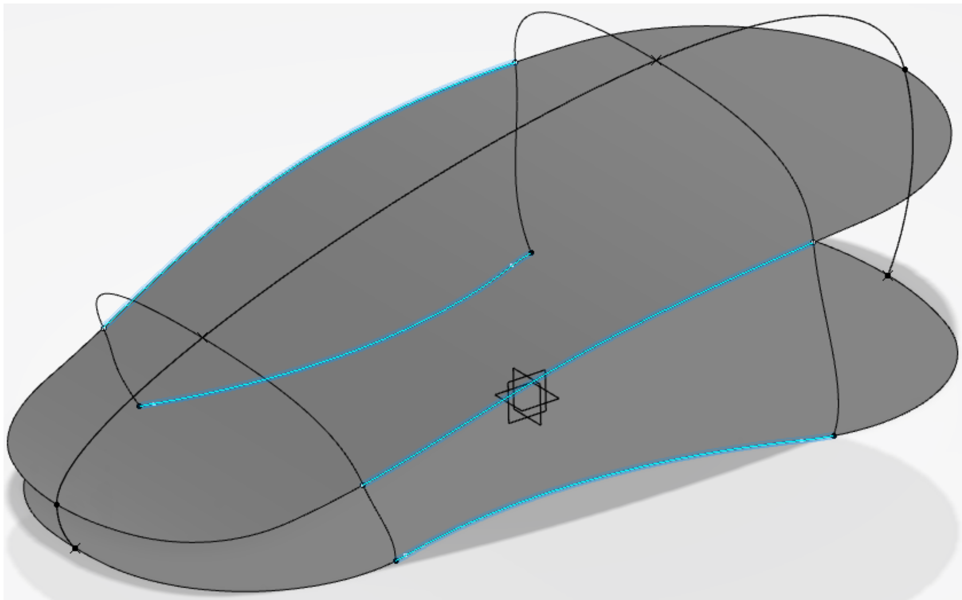
Create a boundary curve on the following edge. 



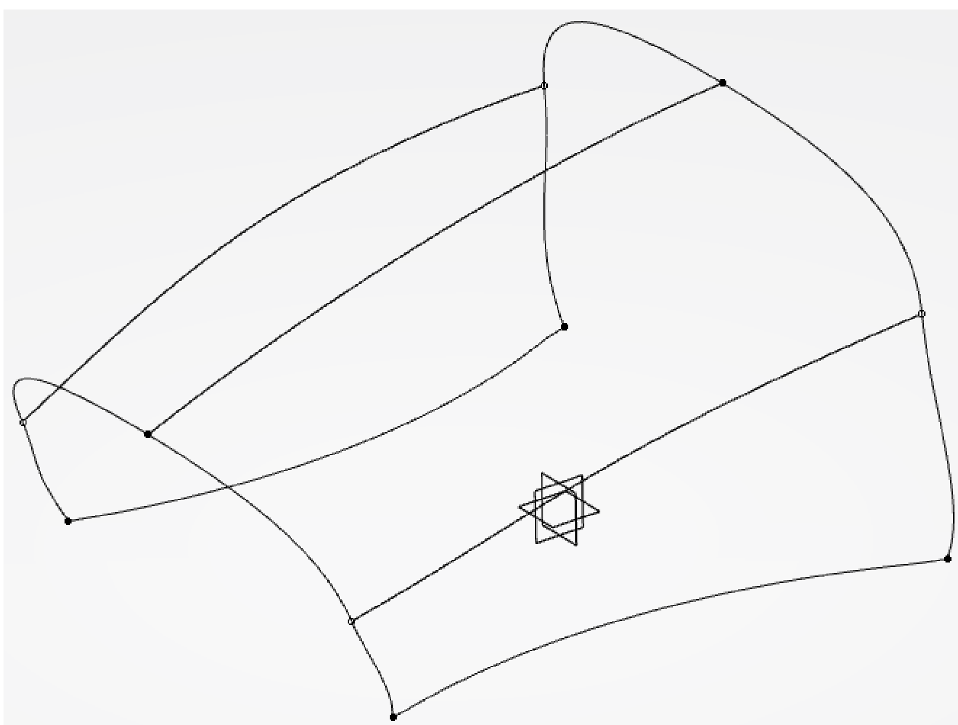
Create another boundary curve on the edge shown below. 



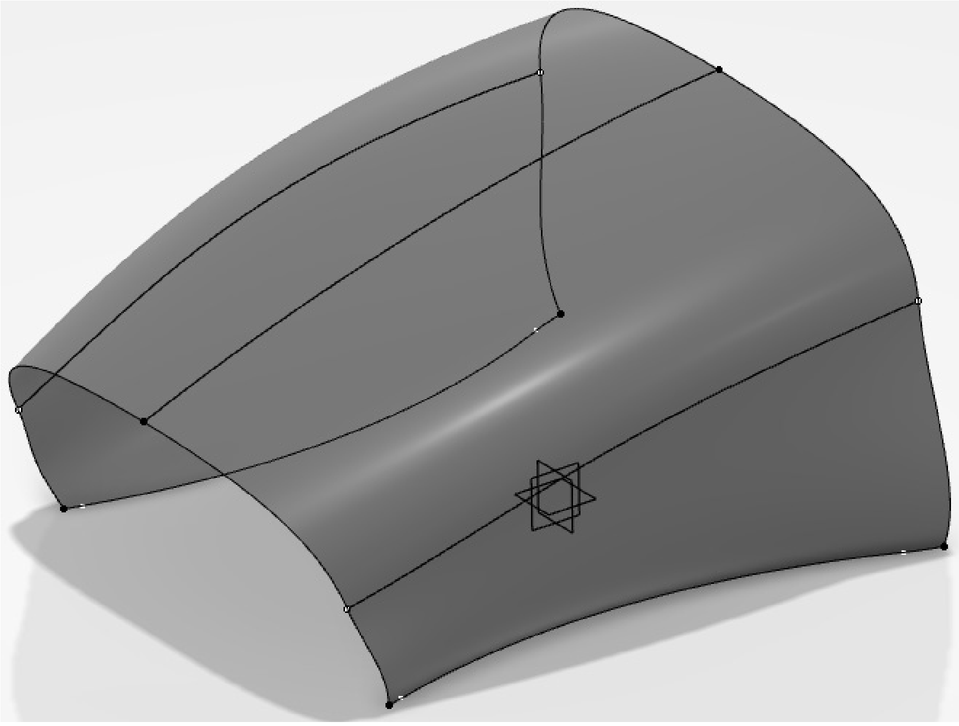
Create the same two boundaries on the opposite side of the surfaces.  Only the boundary curves, the joined curves, the split surface, and the filled surface are shown below.



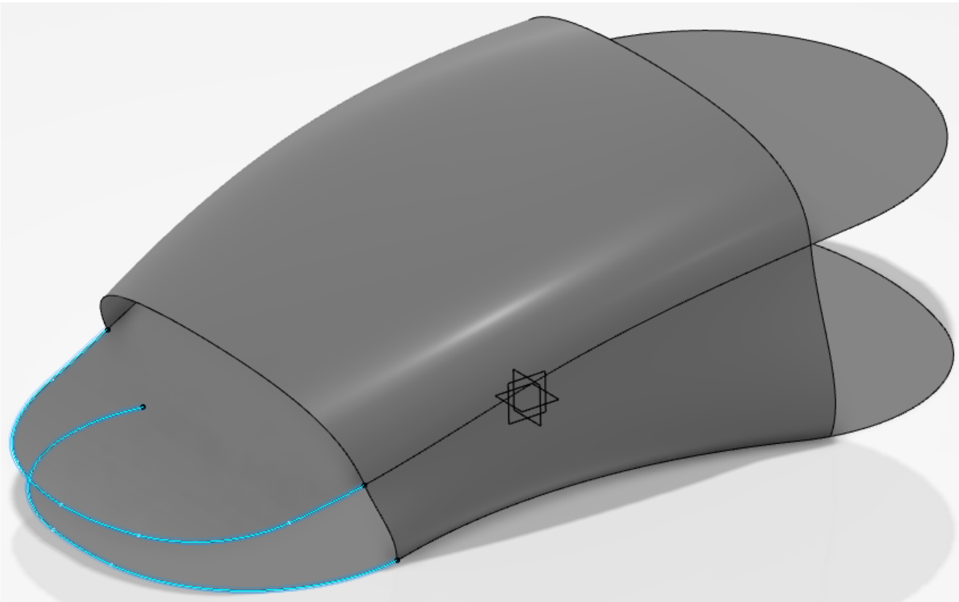
Split the upper spline at its normal planes, then hide all geometry except the boundary curves, the joined curves, and the new split. 



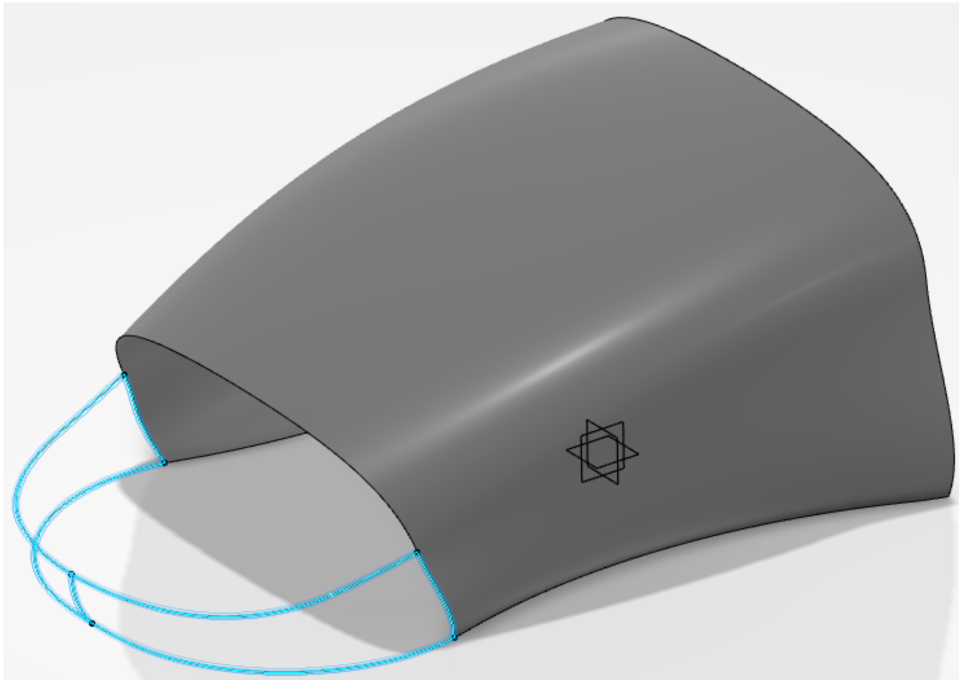
Create a multi-section surface using the joined curves as sections, and the boundary curves and split curve as guides. 



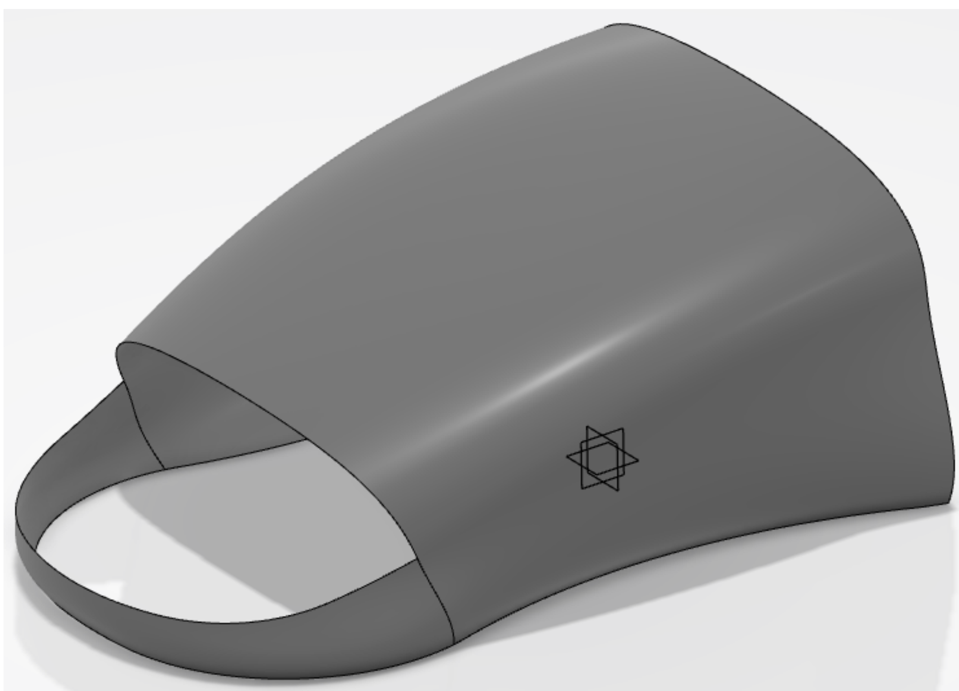
Next, create the two boundary curves shown below.  You'll have to show the two surfaces.



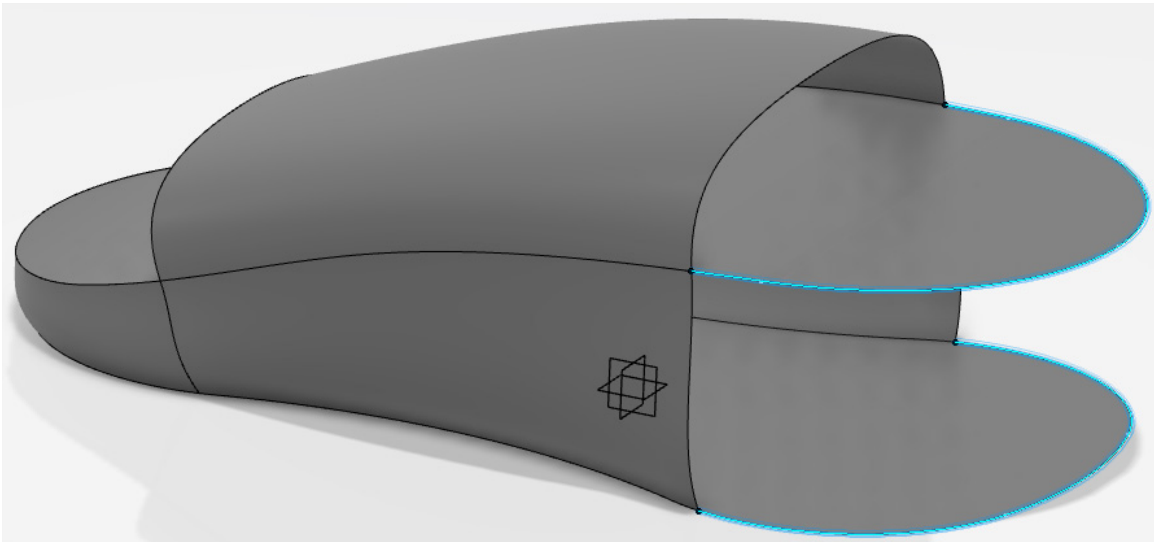
Create a multi-section surface using the geometry shown below.  The splines and the extracted arc are the sections, and the boundaries are the guides. Ensure the first and last sections are tangent continues to the surface shown below.



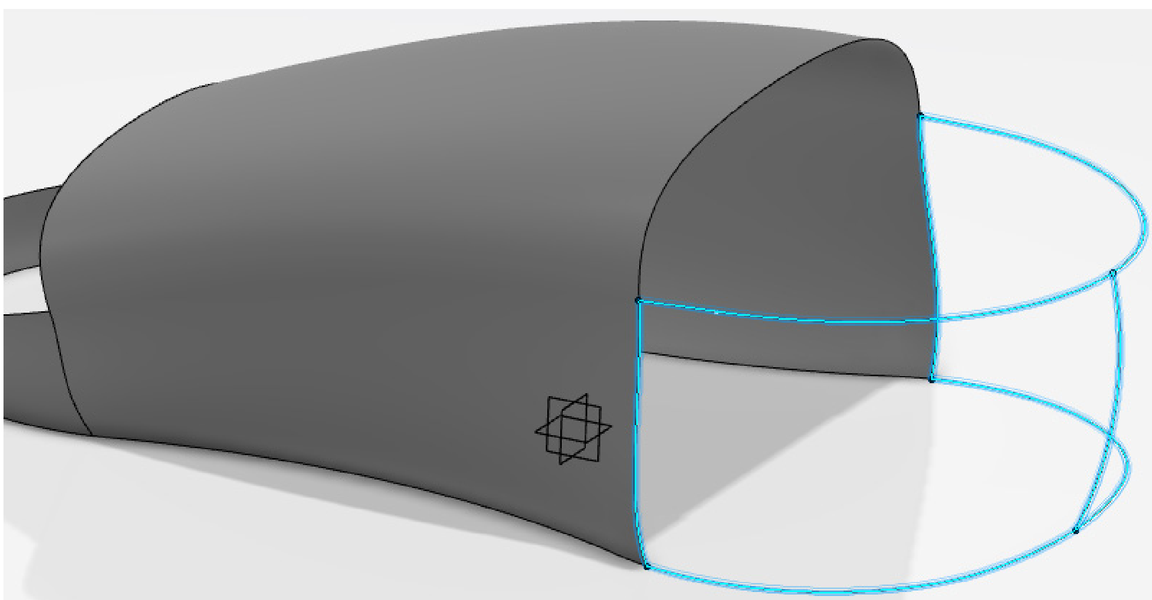
Your model should look like this.



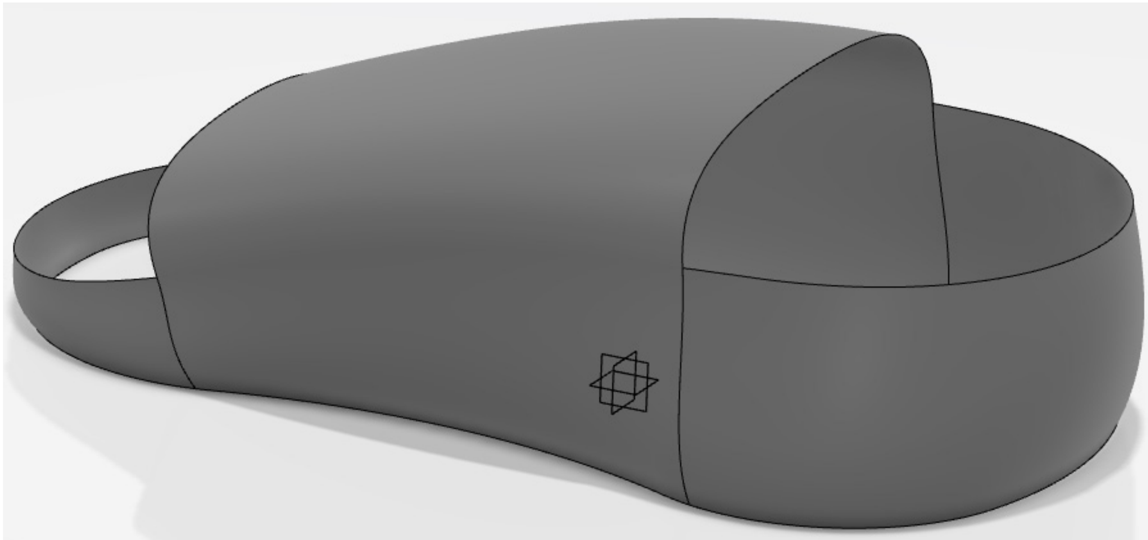
Create the boundaries shown below. 



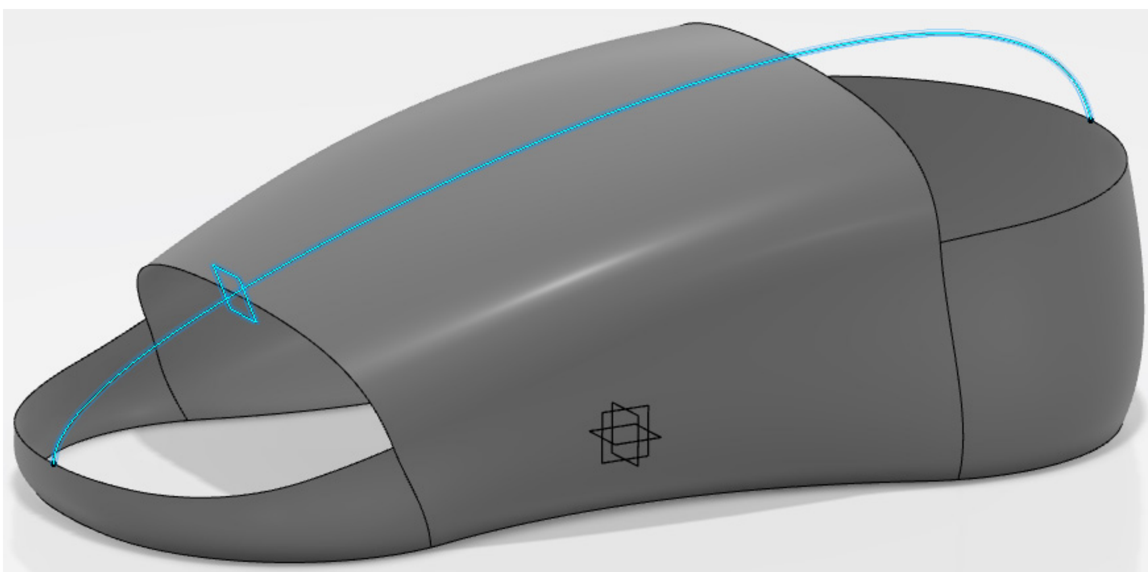
Create a multi-section surface using the geometry shown below.  The splines and the extracted arc are the sections, and the boundaries are the guides. Ensure the first and last sections are tangent continues to the surface shown below.



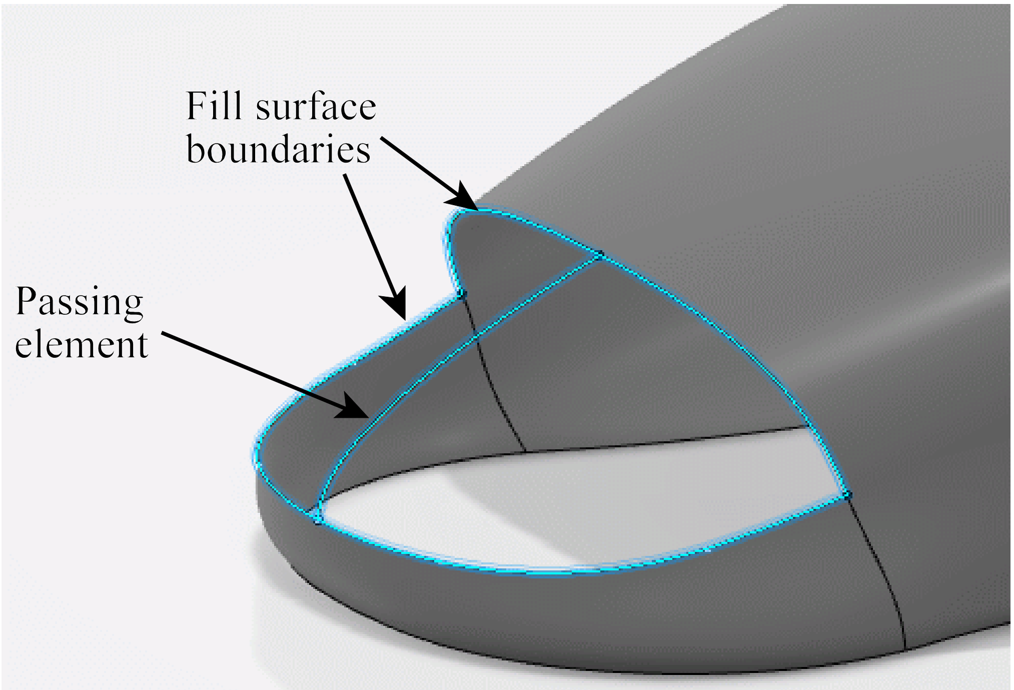
Your model should look like this. Only the three multi-section surfaces are shown below.



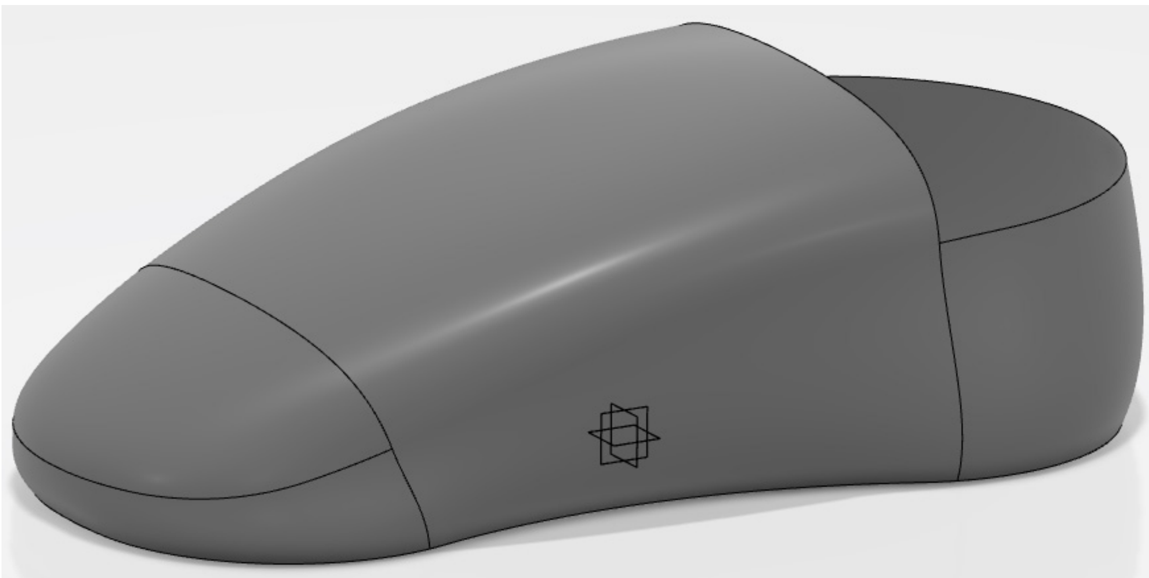
Split the upper spline at the plane shown below, keeping the left side. 



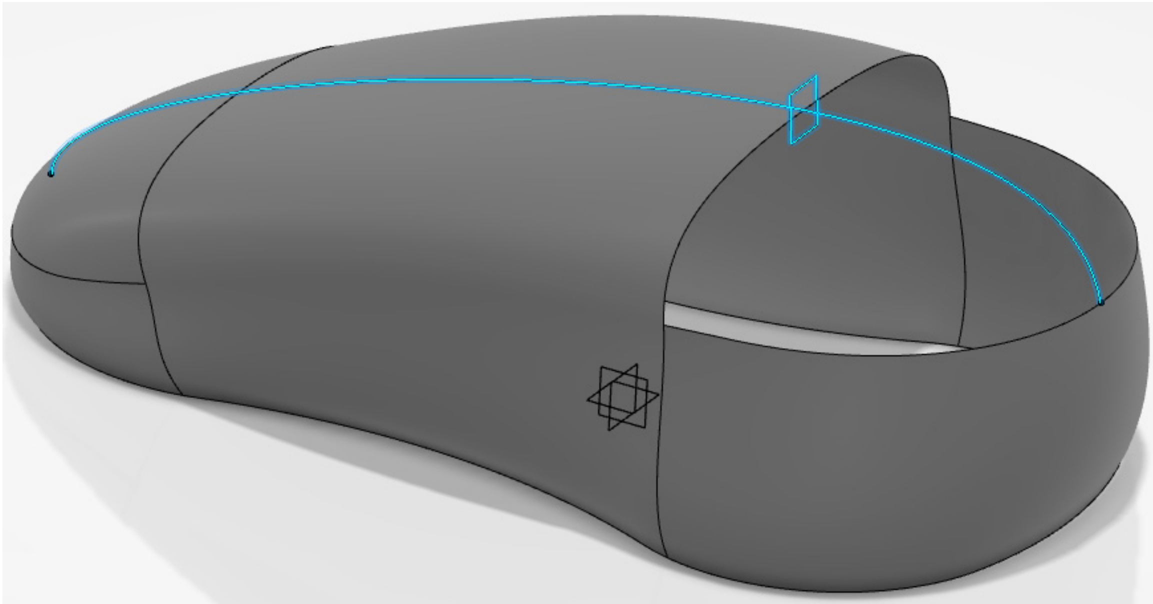
Create a fill surface from the curves shown below that passes through the split curve you just created.  Ensure the fill surface is tangent continuous to the existing surfaces.



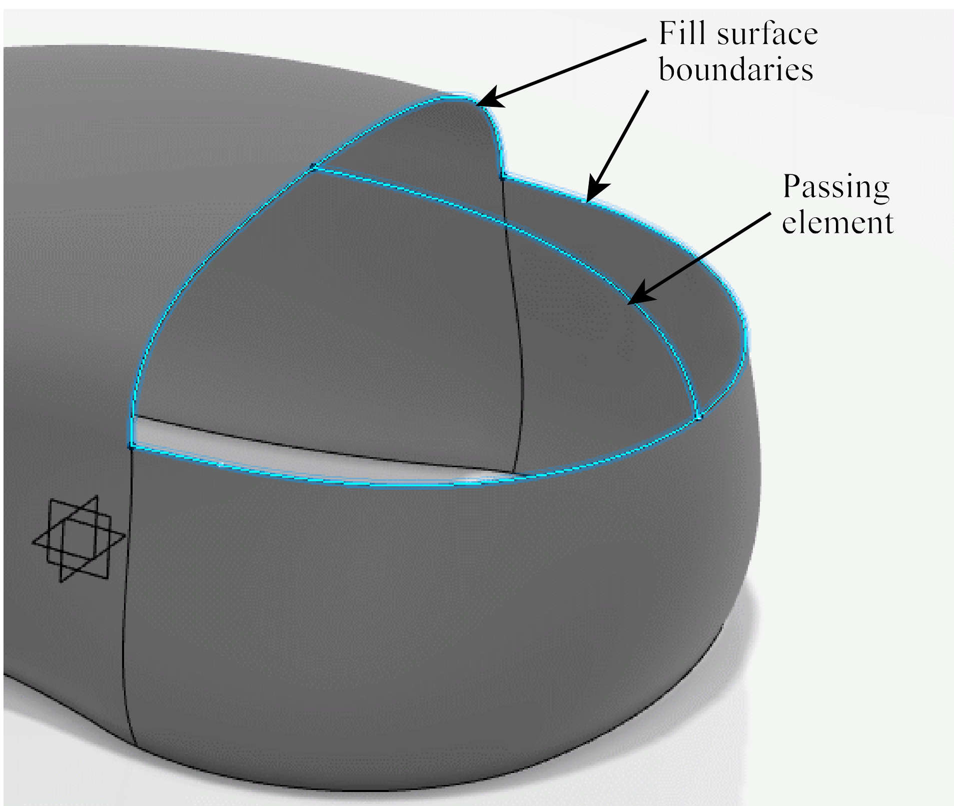
Your model should look like this. Only the multi-section surfaces and the fill surface are shown below.



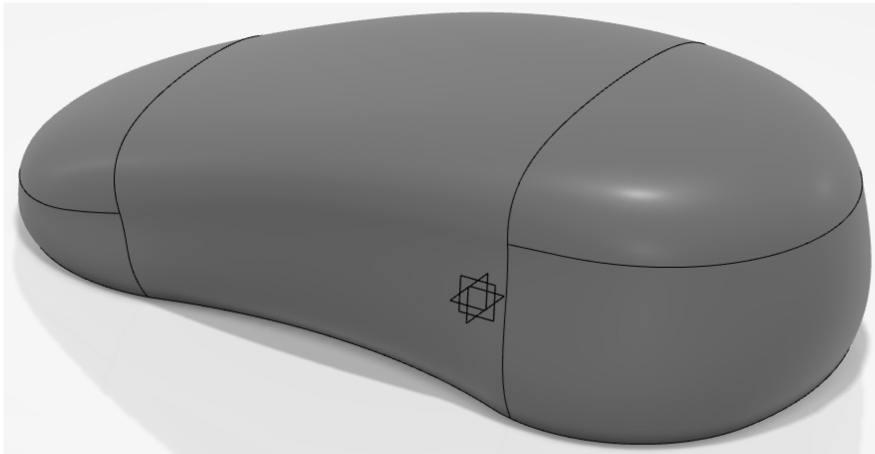
Split the upper spline at the plane shown below, keeping the right side. 



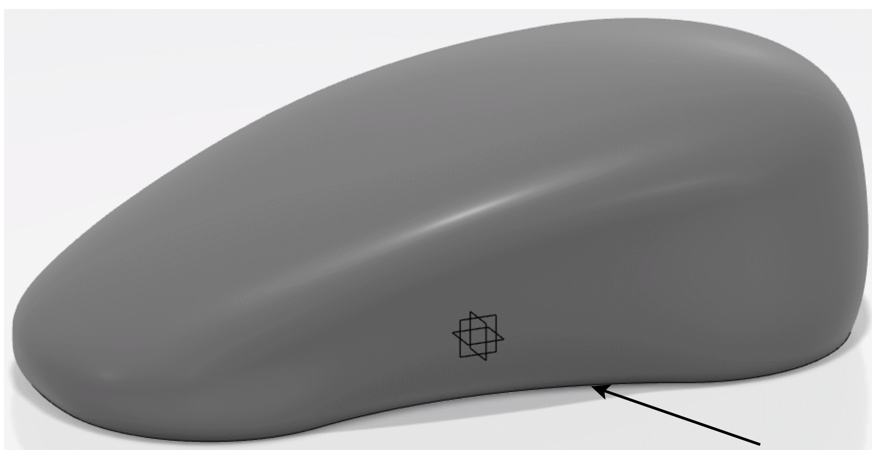
Create a fill surface from the curves shown below that passes through the split curve you just created.  Ensure the fill surface is tangent continuous to the adjacent surfaces.



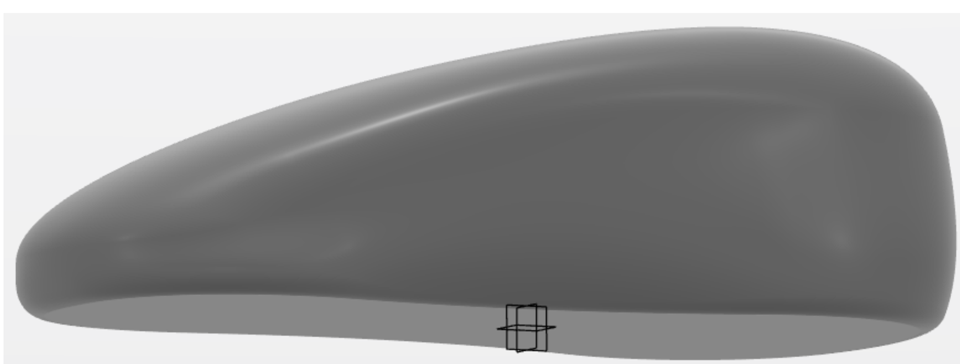
Your model should look like this. Only the multi-section and fill surfaces are shown.



Join all of the multi-section and fill surfaces together.  Ensure that the normal arrow is pointing to the inside of the join. Do not forget to include the first fill surface you created for the bottom profile.



Create a 0.125 inch fillet along the bottom edge, then change the name of the fillet to <u>OUTER SURFACE</u> in the specification tree. 



Thicken *OUTER SURFACE* 0.0625 inches to the inside, then hide the *PartBody*. 

Buttons and Wheel

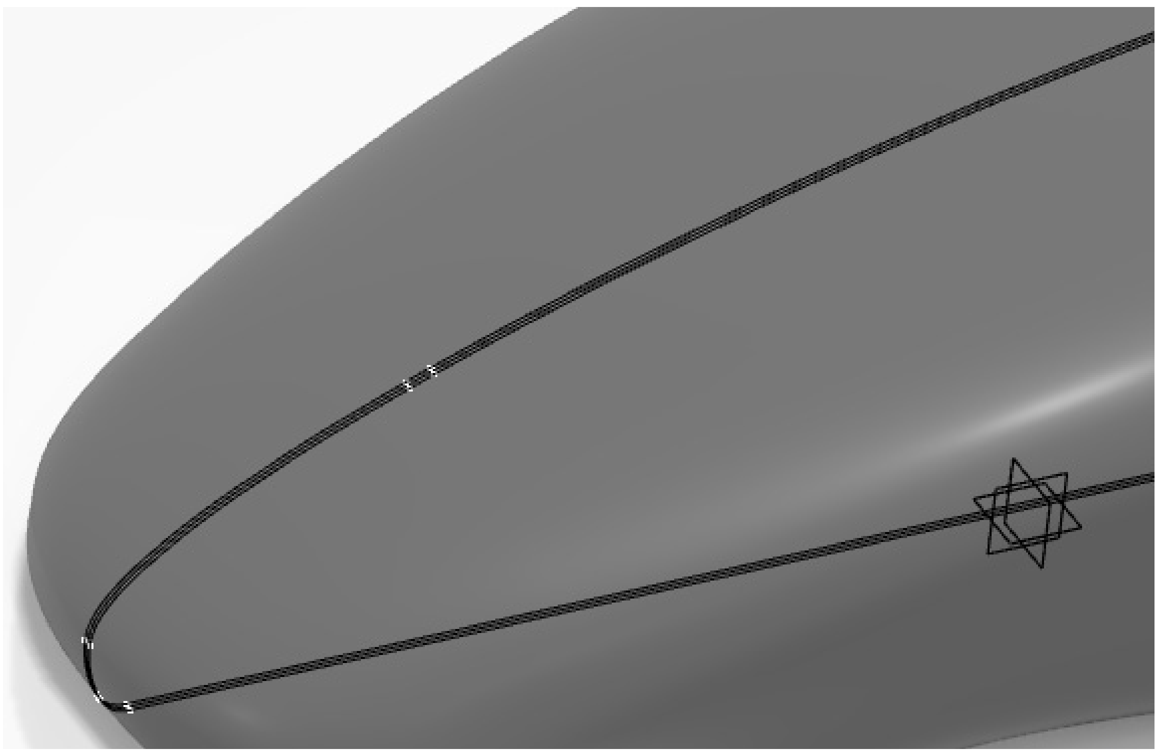
Insert a new geometrical set named <u>Buttons and Wheel</u>. For organizational purposes, the remaining reference geometry will be created in this new set.

Offset *OUTER SURFACE* inward 0.025 inches, then change the name of the new offset to <u>INNER SURFACE</u> in the specification tree and hide *OUTER SURFACE*. 

Intersect *INNER SURFACE* with the zx plane. 

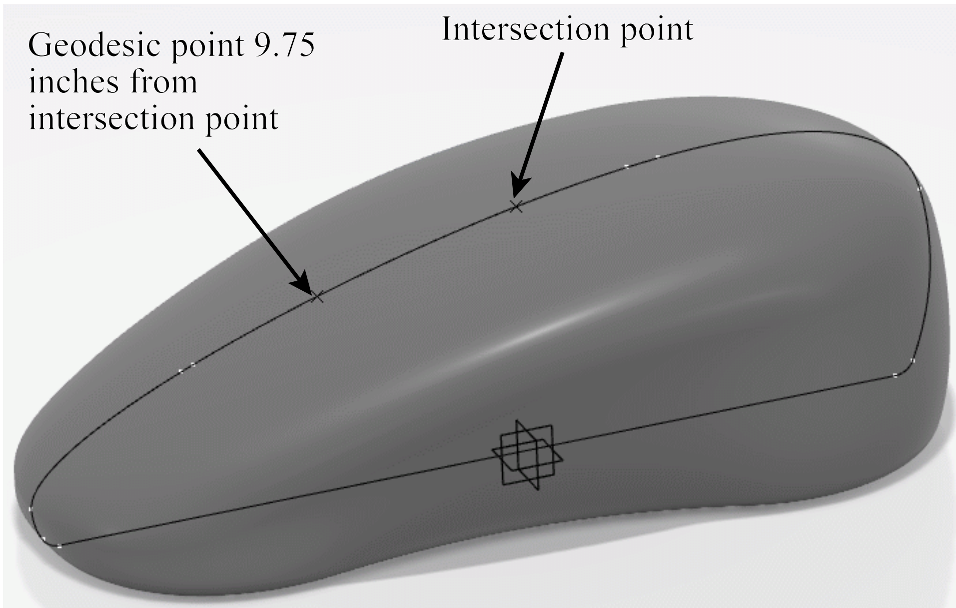
Create a parallel curve from the intersection curve 0.0125 inches away on both sides.

 Ensure that *INNER SURFACE* is the support.

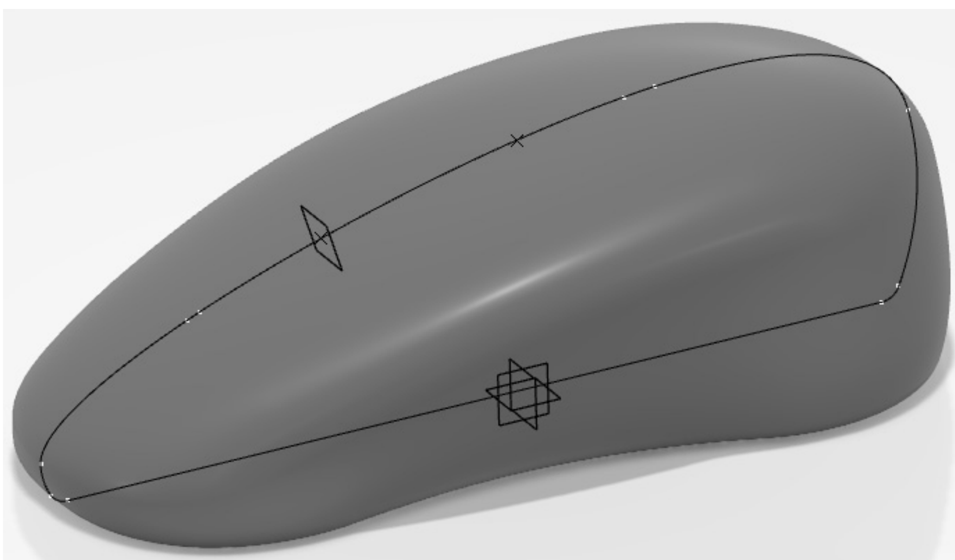


Intersect the yz plane with the intersection curve shown above, keeping only the top point. 

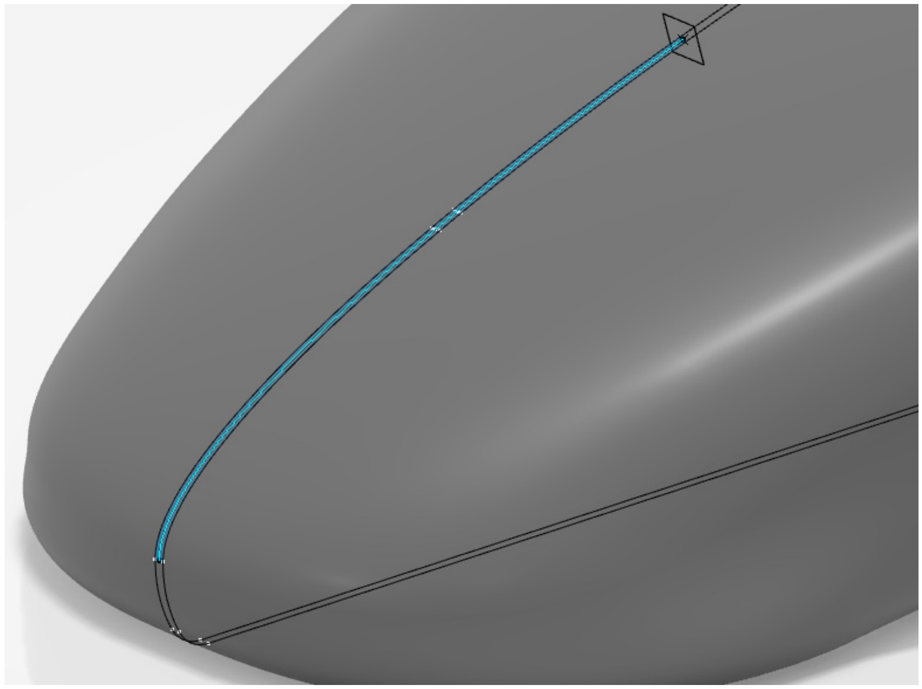
Create a geodesic point on the intersection curve that is 9.75 inches away from the intersection point you just created. Ensure that the point is between the nose of the mouse body and the intersection point. The parallel curves have been hidden in the following image.



Create a plane normal to the intersection curve at the last point created, then change the name to <u>CHANNEL PLANE</u> in the specification tree.

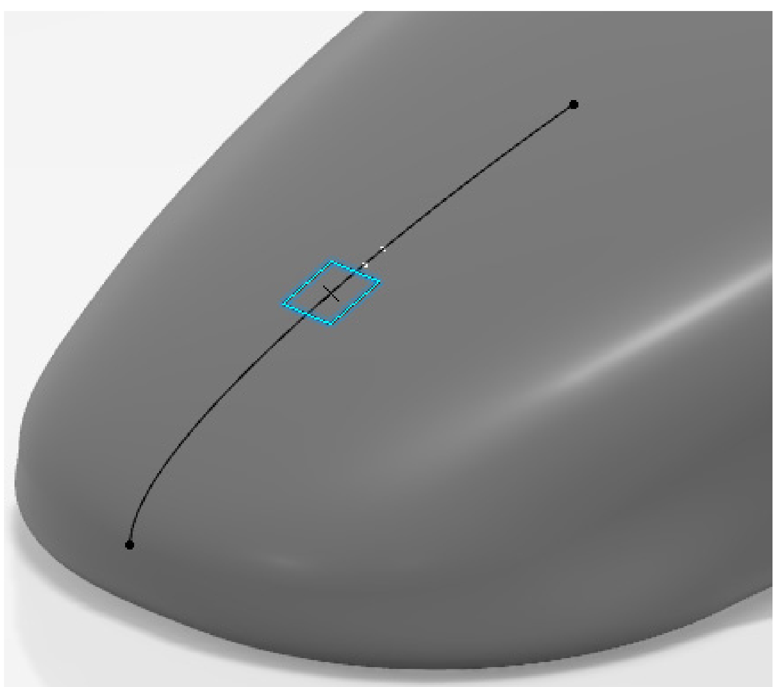


Split the intersection curve with *CHANNEL PLANE* and *CHANNEL SURFACE*, then change the name of the split to <u>CHANNEL CURVE</u> in the specification tree. The split curve is shown below.

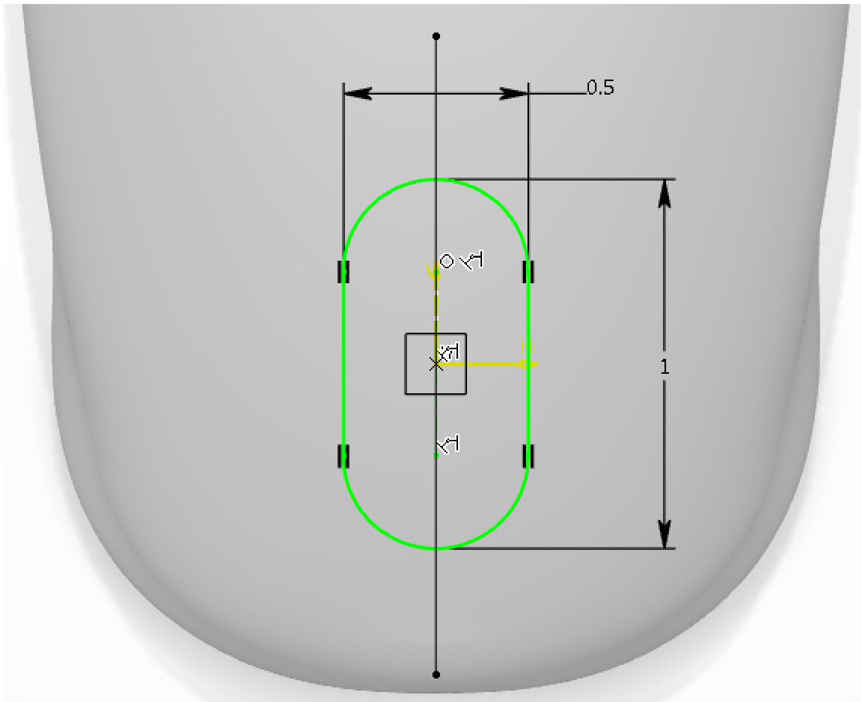


Create a midpoint on *CHANNEL CURVE*.

Create a plane that is tangent to *INNER SURFACE* at the midpoint on *CHANNEL CURVE*. The tangent plane, the midpoint, INNER SURFACE, and CHANNEL CURVE are shown below.



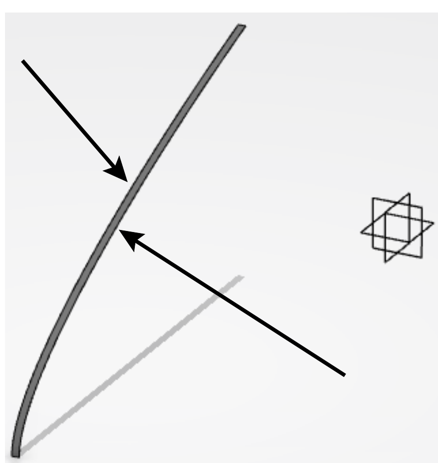
Make the *PartBody* the in-work object, then create the following positioned sketch on the new plane.  Use the midpoint for the *Origin* and the y axis for the *Orientation*, then reverse the *V Direction*.



Pad the sketch upward 0.0625 inches and use *Up to next* for the *Second Limit*, then hide the *PartBody*. 

Activate the *Buttons and Wheel* geometrical set.

Split *INNER SURFACE* with both parallel curves, *CHANNEL PLANE*, and *CHANNEL SURFACE*, keeping the smaller, inner portion.  The result is shown below.



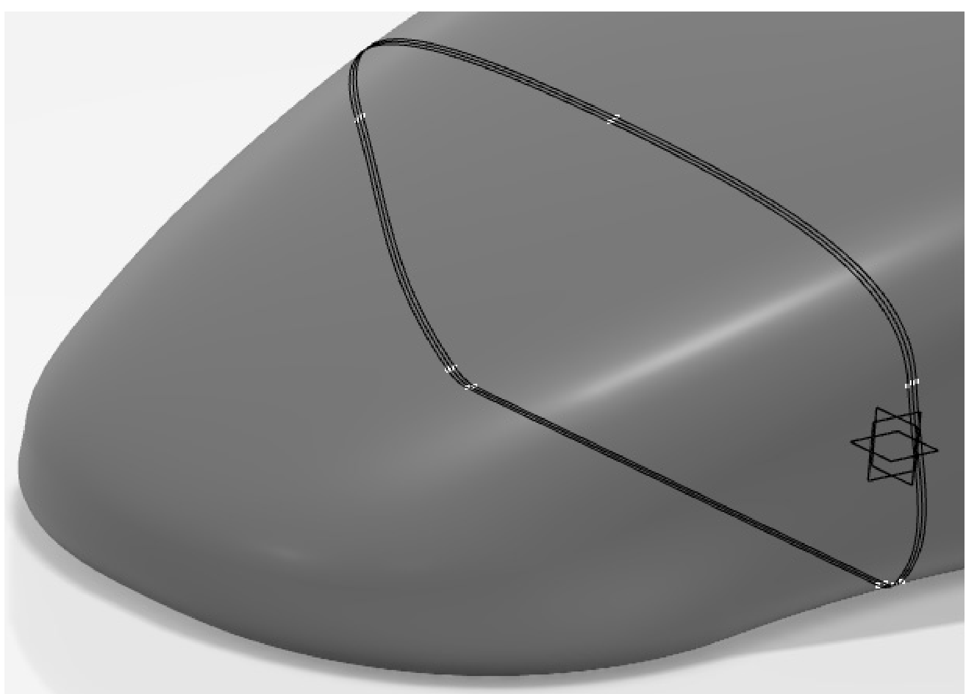
Create a boundary element on both edges indicated above. 

Create a linear sweep with a zero degree angle that is 0.05 inches long on both boundary elements, using the zx plane as the reference surface. Ensure the direction of the sweep is outward. The split surface and both linear sweeps are shown below.

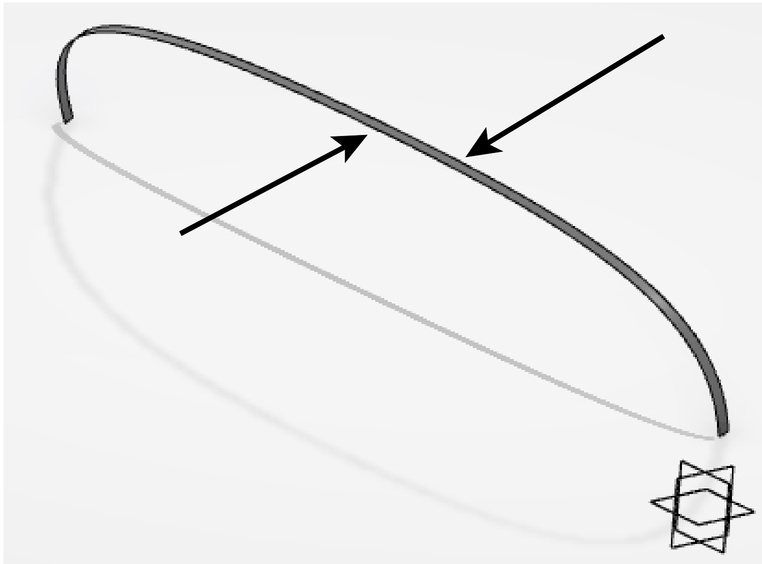
Join the split surface and the linear sweeps shown above, then change the name of the join to <u>TOP CHANNEL</u> in the specification tree.

Hide *TOP CHANNEL*, then show *INNER SURFACE* and intersect it with *CHANNEL PLANE*.

Create a parallel curve from the intersection curve that is 0.0125 inches away on both sides. Ensure that *INNER SURFACE* is the support. The intersection curve, both parallel curves, and *INNER SURFACE* are shown below.



Split *INNER SURFACE* with the new parallel curves and *CHANNEL SURFACE*, keeping the top portion.  The result is shown here.



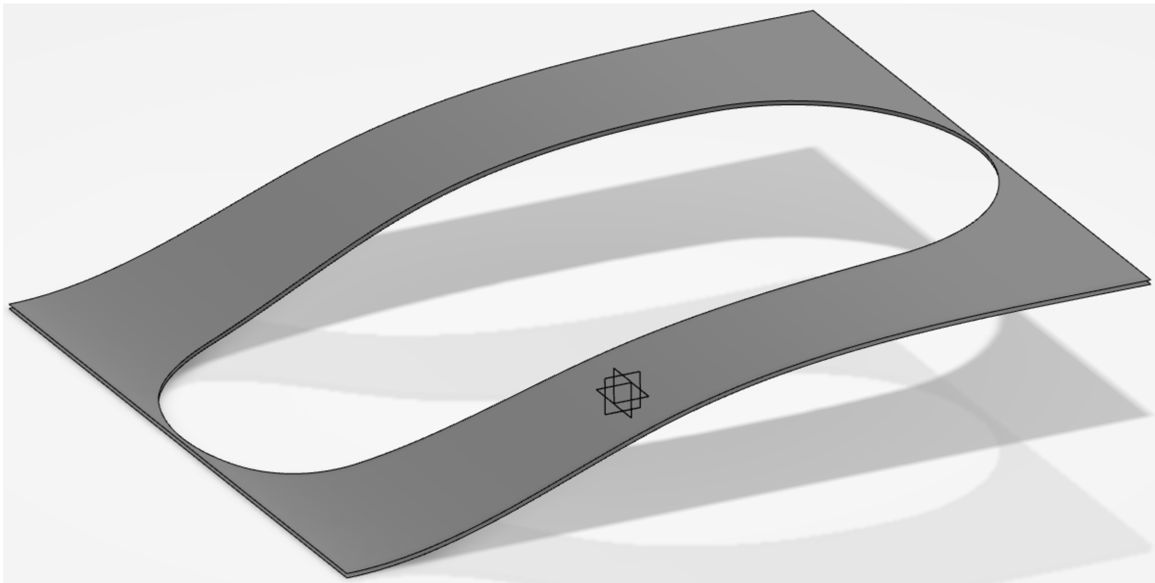
Create a boundary curve on both edges indicated above. 

Create a linear sweep with a zero degree angle that is 0.05 inches long on both boundary elements, using *CHANNEL PLANE* as the reference surface.  Ensure the direction of the sweep is outward. The split surface and both linear sweeps are shown below.

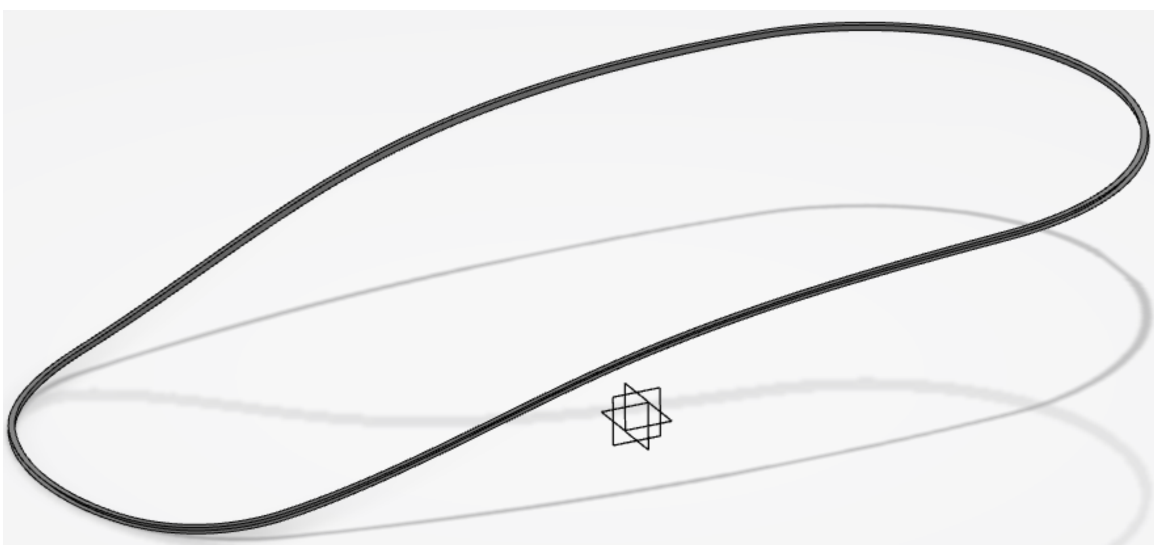
Join the split surface and the linear sweeps shown above, then change the name of the join to <u>CROSS CHANNEL</u> in the specification tree and hide it. 

Show *CHANNEL SURFACE*, then offset it downward 0.025 inches. 

Show *INNER SURFACE*, then trim CHANNEL SURFACE and its offset with *INNER SURFACE*, keeping only the inside portion of *INNER SURFACE*. The result is shown below.

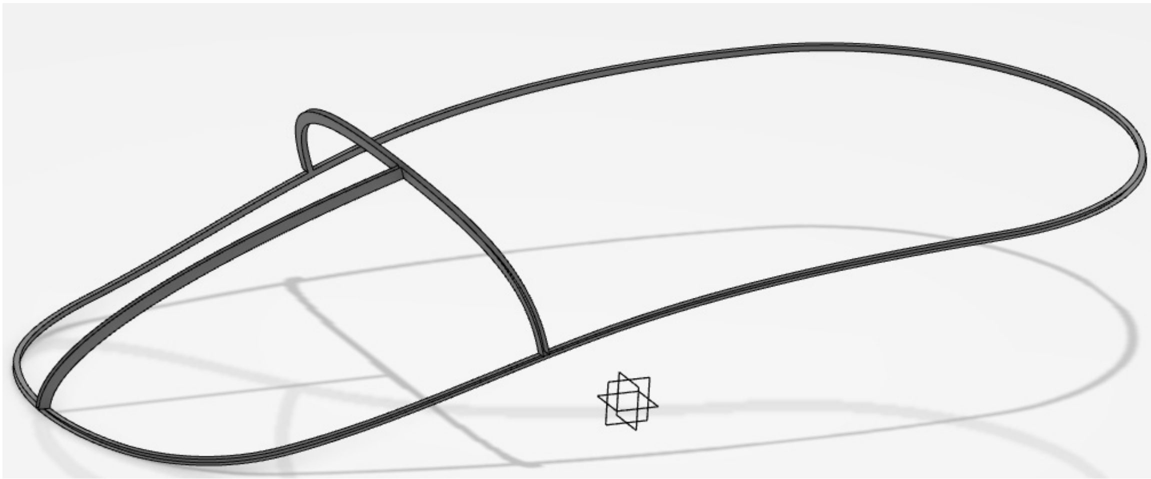


Split the newly created trim with *OUTER SURFACE*, keeping the inside portion, then change the name of the split to <u>SIDE CHANNEL</u> in the specification tree.

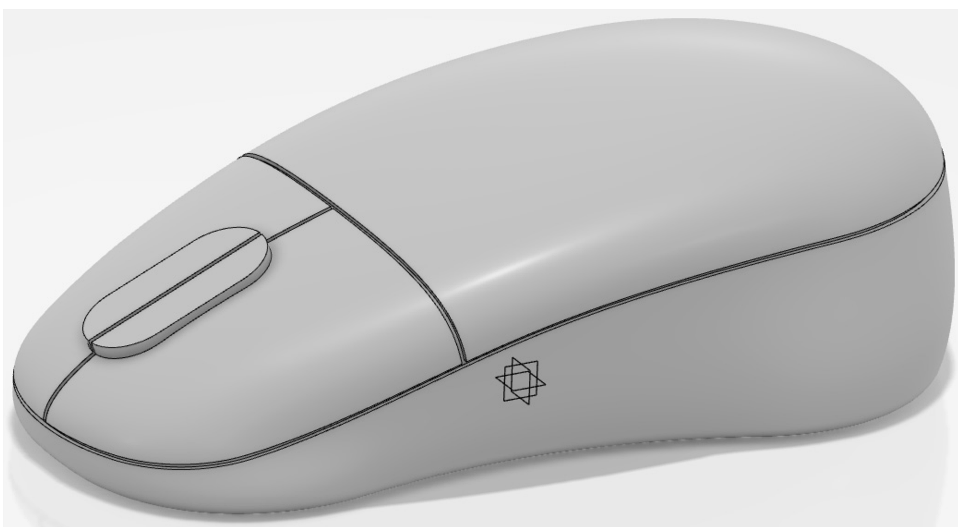


Show *TOP CHANNEL* and *CROSS CHANNEL* from the tree.

Trim *SIDE CHANNEL*, *TOP CHANNEL*, and *CROSS CHANNEL* together, then change the name of the trim to <u>COMBINED CHANNELS</u> in the specification tree. 



Split the *PartBody* with *COMBINED CHANNELS*, keeping the inside portion of the solid.  Ensure the *Extrapolation type* is set to *Tangent*. The *PartBody* is shown here.



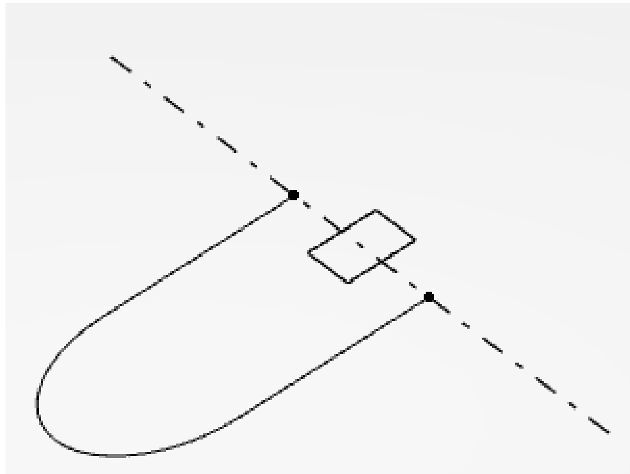
Create a plane that is offset downward from the last plane by 0.15 inches.  Be sure to activate the *Buttons and Wheel* geometrical set.

Project the pad's sketch to the new plane. 

Create an axis line through the minor axis of the projected, elongated hole. 

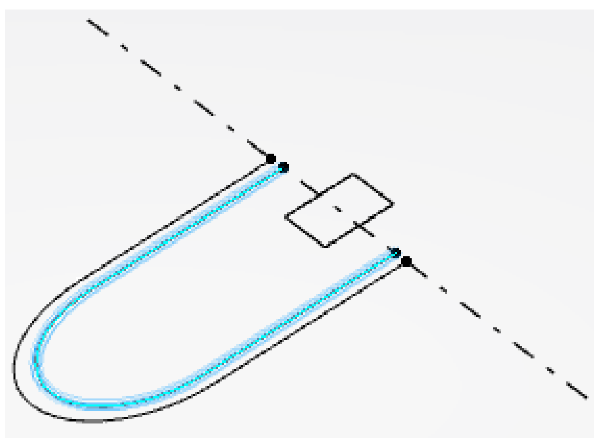
Create a parallel curve to the inside of the projected, elongated hole that is 0.1 inches away.  Use the offset plane as the support.

Split the projected, elongated hole with the axis line.  The offset plane, the axis line, and the split projection are shown below.



Create a 360 degree groove with the split projection curve and the axis line. 

Create a parallel curve to the inside of the split projection that is 0.025 inches away, then change the name of the parallel curve to <u>WHEEL PROFILE</u> in the specification tree.  Use the offset plane as the support. The parallel curve is highlighted below.

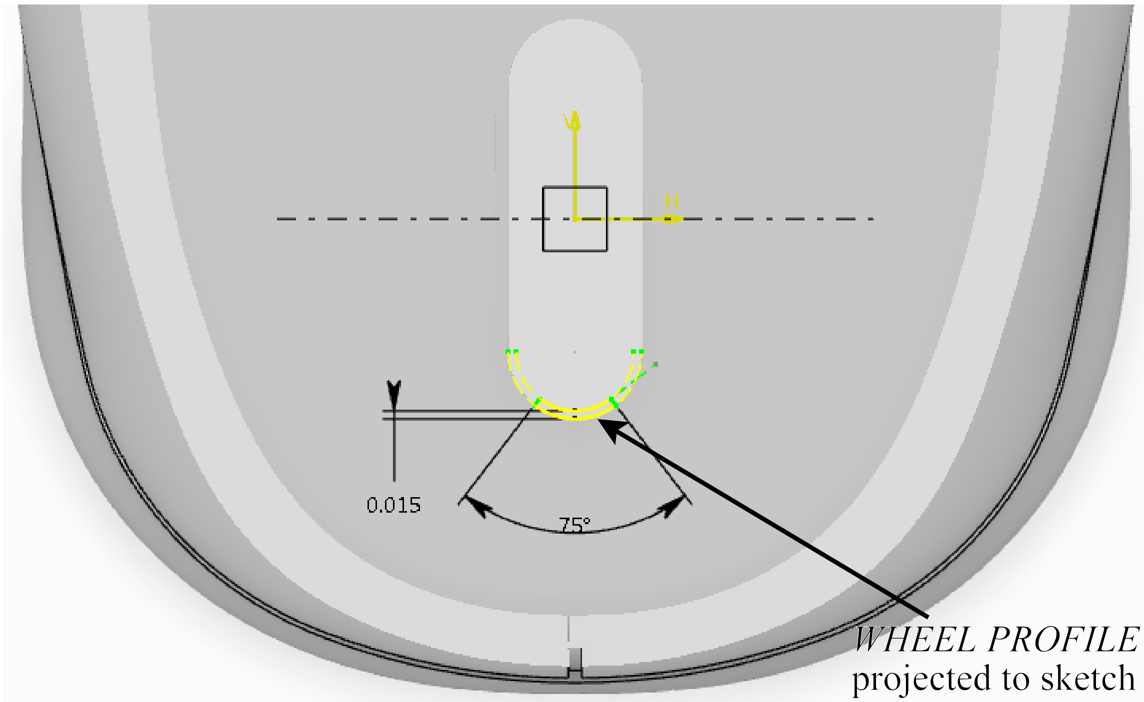


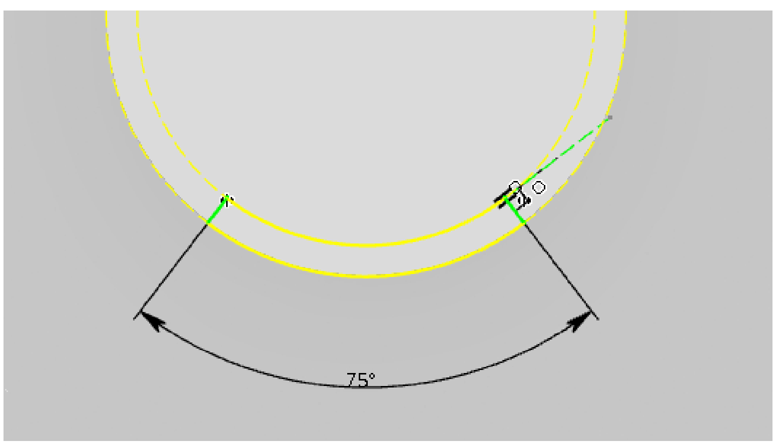
Create a 360 degree shaft from *WHEEL PROFILE* and the axis line. 

Use *WHEEL PROFILE* to create the following positioned sketch on the offset plane.

 The offset plane is the *Planar support*, the midpoint previously created is the *Origin*, and the y axis is used for the *Orientation*. Reverse the directions as necessary. In the image below, the *PartBody* is cut at the sketch plane for clarity.

Note: Show the PartBody if the sketch is not visible.

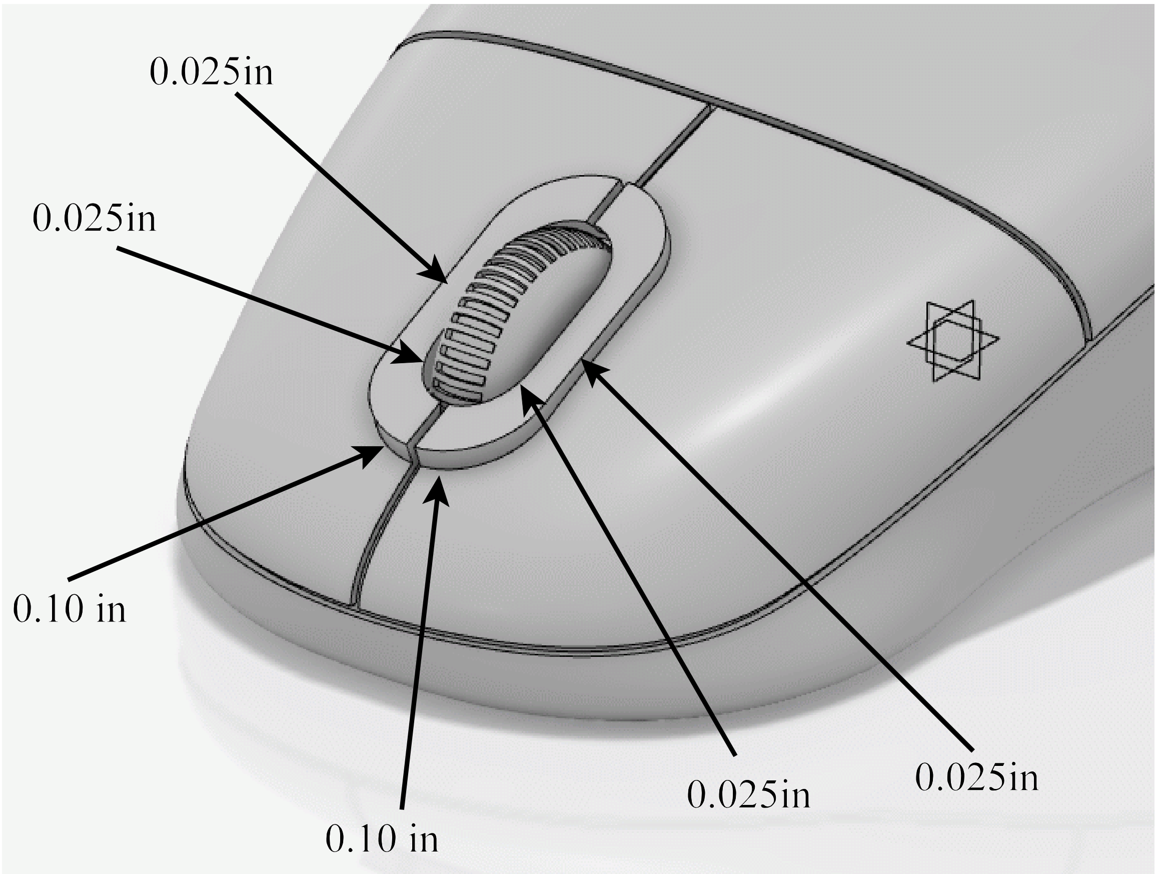




Create a groove that is two degrees in both directions using the new sketch and the axis line. 

Create a 360 degree circular pattern of the groove with 50 instances.  Use the axis line for the *Reference Direction*.

Fillet the edges indicated below with the given values.



Your finished model should look like this.