

TABLE OF CONTENTS

Introduction 1

 Wireframe & Surfaces 2

 Pull Down Menus 3

 Edit 3

 Insert 4

 Tools 6

 Generative Shape Design Workbench 7

 Bottom Toolbars 10

 Volumes 12

 Advanced Surfaces 12

 Developed Shapes 13

 BiW Templates 13

 Advanced Operations 13

Wireframe 15

 Points 15

 Coordinate 15

 On curve 18

 On plane 23

 On surface 24

 Circle / Sphere / Ellipse center 26

 Tangent on curve 27

 Between 30

 Point Repetition 32

 Projecting points 37

 Intersection points 41

 Extremum 44

 Polar Extremum 47

 Lines 49

 Point-Point 49

 Point-Direction 52

 Angle/Normal to curve 53

 Tangent to curve 58

 Normal to surface 61

 Bisecting 63

 Intersection lines 65

 Projecting lines 66

 Axis 67

 Polyline 69

Planes	73
Offset from plane	73
Parallel through point	75
Angle/Normal to plane	76
Through three points	78
Through two lines	79
Through point and line	80
Through planar curve	81
Normal to curve	82
Tangent to surface	83
Equation	84
Mean through points	85
Plane Between	86
Circles	89
Center and radius	89
Center and point	91
Two points and radius	92
Three points	94
Center and axis	95
Bitangent and radius	96
Bitangent and point	97
Tritangent	98
Center and tangent	99
Corners	100
Curves	104
Connect Curves	104
Conics	109
Splines	115
Helixes	121
Spirals	127
Project Curves	130
Combine Curves	134
Reflect Line Curves	138
Silhouette Curves	140
Intersection Curves	143
Parallel Curves	147
Rolling Offset Curve	153
3D Curve Offset	155
Curve comparison	158
Supports	161
Work on Support	161
Creation on the fly	166
Modifying	169
Datums	171
Object repetition	173

Surfaces	175
Extruded	175
Revolution	177
Sphere	179
Cylinder	181
Offset	183
Variable Offset	186
Rough Offset	188
Offset with Multiple Sub-elements	189
Mid Surface	192
Sweep	202
Explicit	202
With reference surface	202
With two guide curves	210
With pulling direction	214
Linear	216
Two limits	216
Limit and middle	220
With reference surface	222
With reference curve	224
With tangency surface	226
With draft direction	228
With two tangency surfaces	232
Circular	234
Three guides	234
Two guides and radius	236
Center and two angles	238
Center and radius	240
Two guides and tangency surface	242
One guide and tangency surface	244
Limit curve and tangency surface	246
Conical	248
Two guide curves	248
Three guide curves	250
Four guide curves	252
Five guide curves	254
Adaptive Sweep	256
Fill Surfaces	261
Multi-Section Surfaces	267
Blend surfaces	277
Spines	283
Isoparametric Curves	288
Laws	290

Operations	301
Joining Elements	301
Healing Surfaces	308
Curve smoothing	312
Surface Simplification	313
Splitting Elements	316
Trimming Elements	322
Untrimming Elements	325
Disassembling Elements	327
Sew Surface	329
Remove Face	331
Extracting Boundaries and Faces	333
Multiple Extract	338
Fillet	341
Shape Fillet	341
Edge Fillet	347
Face to Face Fillet	352
Tritangent Fillet	354
Styling Fillet	356
Chamfer	362
Transformations	366
Translate	366
Rotate	368
Symmetry	370
Scale	371
Affinity	372
Axis to Axis	375
Patterns	376
Rectangular	376
Circular	379
User Defined	380
Extrapolating Curves and Surfaces	381
Inverting the Orientation	386
Creating the Nearest or Farthest Element	387
Multi-Selection	389
Analysis	395
Connect Checker Analysis	395
Light Distance Analysis	405
Draft Analysis	407
Surfacic Curvature Analysis	411
Porcupine Curvature Analysis	416
Geometric Information	425
Dress-Up	427

Geometrical Set Management	429
Inserting Sets	429
Changing Sets	430
Hiding and Showing Components	433
Activating and Deactivating Components	433
Reordering Components	434
Groups	436
Isolating Geometry	437
Copying and Pasting Geometry	438
Ordered Geometrical Set Management	441
Inserting an Ordered Geometrical Set	441
Modifying Children	443
Reordering Components	445
Scanning Ordered Sets	447
Inserting in an Ordered Set	448
Switching to a Regular Geometrical Set	449
Miscellaneous	450
Parents/Children	450
Historical Graph	451
Quick Select	453
Inserting Elements	455
Sets of planes	456
Regular	456
Irregular	457
Semi-Regular	458
Keep and No Keep Mode	460
Keep Mode	460
No Keep Mode	460
Current Body	464
Masks	466
2D Visualization Modes	470
Deleting Useless Elements	474
Review	475
Mouse Body	475
Buttons and Wheel	497

Problems	509
Problem #01	509
Problem #02	510
Problem #03	511
Problem #04	512
Problem #05	514
Problem #06	516
Problem #07	518
Problem #08	520
Problem #09	523
Problem #10	525
Appendix A	529
Shape - Generative Shape Design - General	529
Shape - Generative Shape Design - Work On Support	530
Appendix B	531
Part Design Using Surfaces	531
Split	531
Thick Surface	533
Close	534
Sew	535
Pad/Pocket	537
Boolean Operations	538
Appendix C	541
Generative Shape Optimizer	541
Bump Surfaces	542
Wrap Curve Surfaces	545
Wrap Surface Surfaces	547
Shape Morphing	549
Automatic Filletting	553
Appendix D	557
Developed Shapes	557
Unfold Surfaces	557
Transfer	561
Develop Wires	564
Appendix E	567
BiW Templates	567
Junction Surfaces	567
Diabolo	569
Hole	571
Hole Curve	575
Mating Flange	579
Bead	582
Blend Corner	584
3D Working Supports	587

Appendix F	589
Surface Machining Tools	589
Appendix G	593
Volumes	593

Introduction

CATIA Version 5 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 V5 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 into 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 should be reviewed and there are review exercises located in Appendix B. It is also important that you understand how to work with the 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 your non-solid geometry. When you create new wireframe or surface geometry, you will need to be sure that the correct geometrical set is current in order to have an organized tree.

Wireframe

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

Points

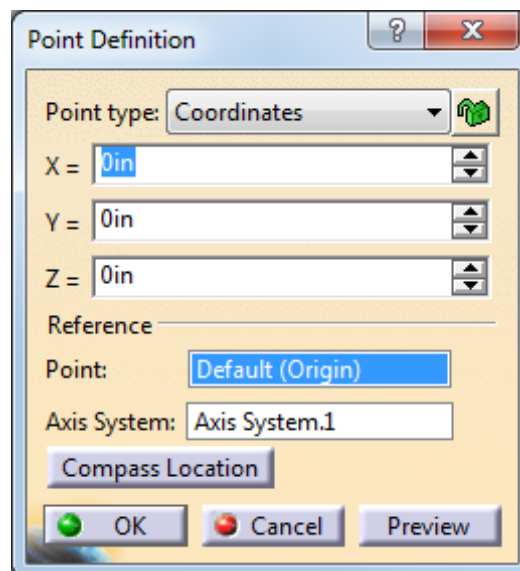
Points are useful to define specific locations and to assist in the creation of other wireframe geometry. You have a variety of options to define points. This exercise will explore those options.

Coordinate

Open the Points document. You should see a surface and some wireframe geometry.

Change to the Generative Shape Design workbench. If you are already in the workbench then you will not need to change. If not, to change workbenches you can select pull down menu *Start, Shape, Generative Shape Design*.

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



Point type Specifies what type of point you want to create, either *Coordinates*, *On curve*, *On plane*, *On surface*, *Circle/Sphere/Ellipse center*, *Tangent on curve* or *Between*

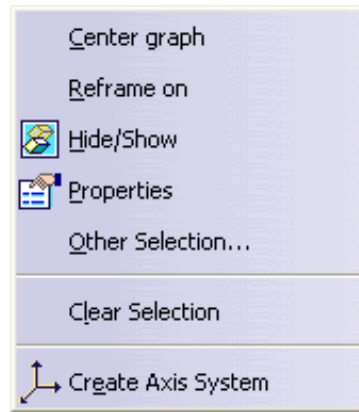
X=, Y=, Z= The coordinate values of the point that you want to create from the reference point

Reference

Point The point that the coordinates are based from. The default is the origin.

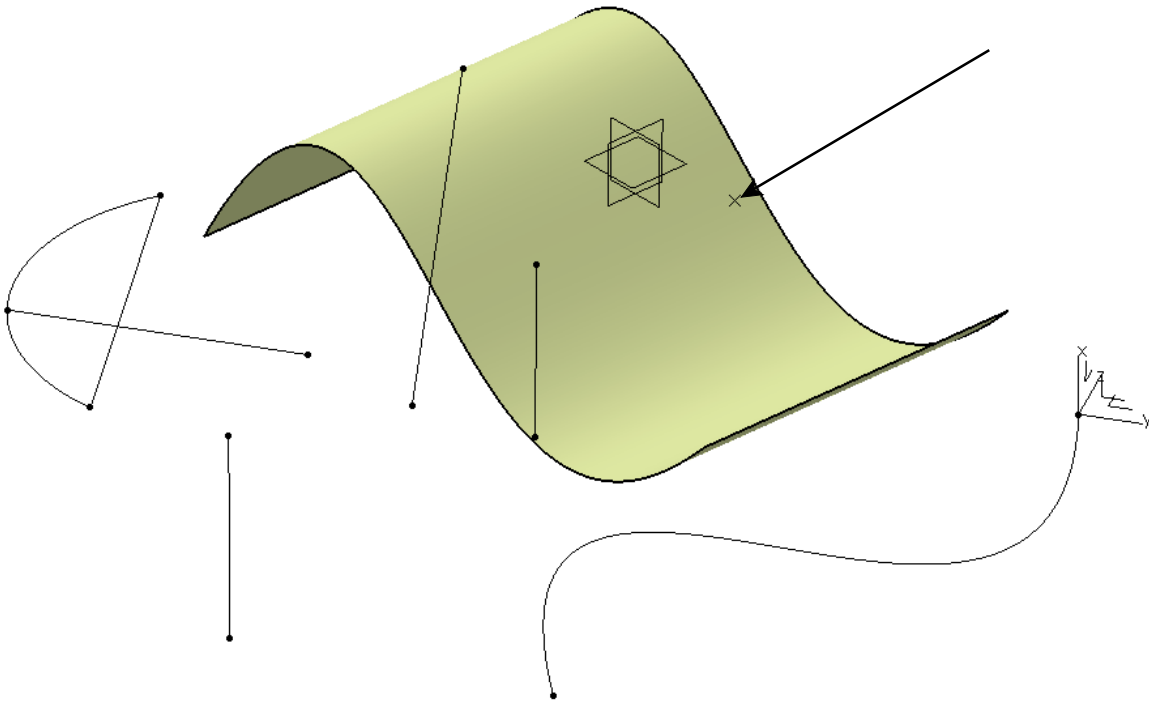
Axis System Defines the axis system that will be used to create the point

Right select in the *Axis System* selection box. A contextual window appears.



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


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



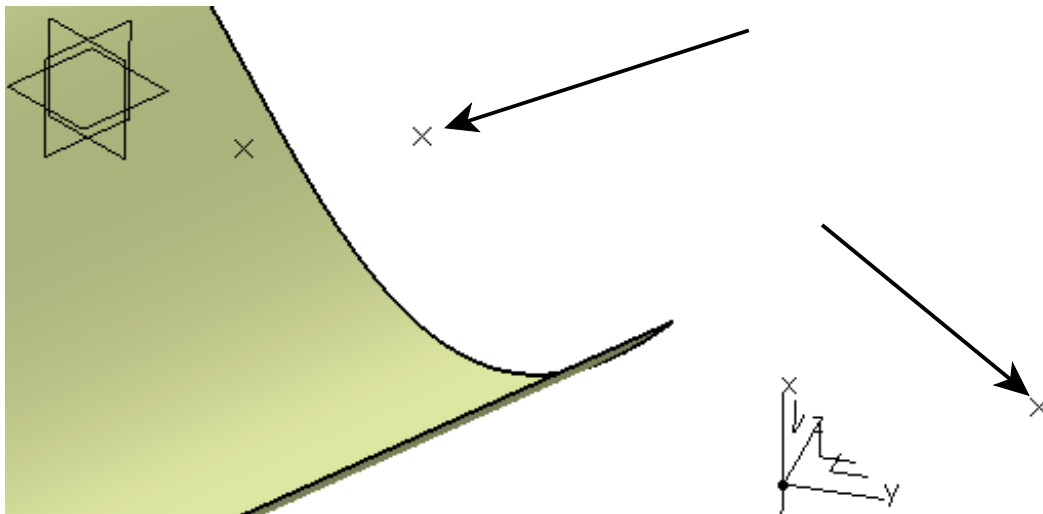
Select the Point icon again.  The *Point Definition* window appears. This time you are going to use a point as the reference instead of the origin.

Clear the *Axis System* selection box so that the absolute axis system will be used and 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* area of the window.

Key in 0.0, 2.0, 1.0 for the X, Y and Z values respectively and select *OK*. The new point is created based from the original point instead of the origin.

Select the Point icon again.  The *Point Definition* window appears. This time you are going to create a point using the axis system that has been created instead of using the absolute axis system.

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

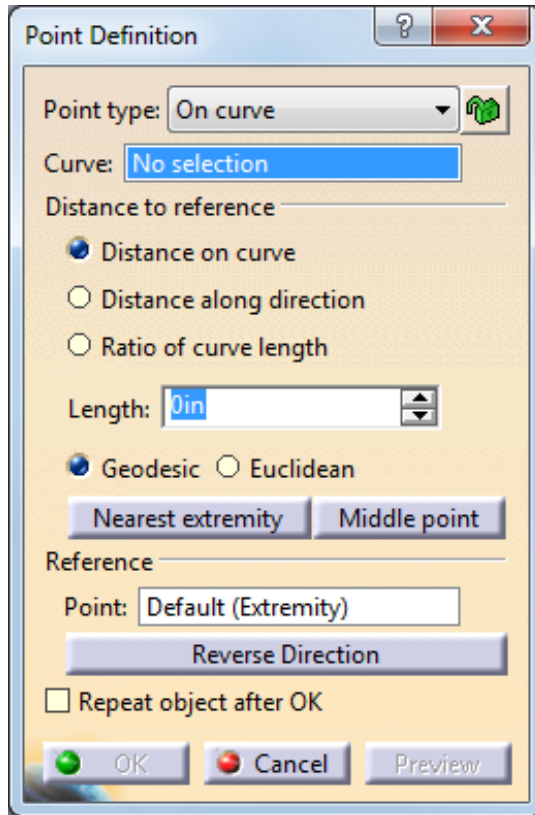


You can also create points on elements such as curves, planes and surfaces.

On curve

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

Change the *Point type* to **On curve**. The options in the window change.



Curve Specifies the curve on which you are going to create a point on

Distance to reference

Distance on curve Allows you to specify a distance along the curve from the reference point

Distance along direction Allows you to specify a distance along the curve in a specific direction

Ratio of curve length Allows you to specify a ratio between the reference point and the extremity

Length/Offset/Ratio You can specify a *Length* for the *Distance on curve* option, an *Offset* for the *Distance along direction* option or a *Ratio* for the *Ratio of curve length* option

<i>Geodesic</i>	The length is measured along the curve
<i>Euclidean</i>	Corresponds to <i>Distance on curve</i> only, the length is the absolute value from the reference point
<i>Nearest extremity</i>	Creates the point at the nearest extremity
<i>Middle point</i>	Creates a point at the midpoint of the curve
<i>Reference</i>	
<i>Point</i>	Allows you to specify a reference point if you do not want to use an extremity as the reference
<i>Reverse Direction</i>	Reverses which side of the reference is used or which extremity is used when creating the point
<i>Repeat object after OK</i>	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 an arrow.

Turn the *Distance on curve* option on, change the *Length* to be 5.0 and make sure it is set to *Geodesic*. The point appears 5 inches along the curve.

Turn the *Euclidean* option on instead of the *Geodesic* option. The point is still on the curve but it is 5 inches from the reference point instead of being 5 inches along the curve.

Select *OK*. The point is created.

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

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

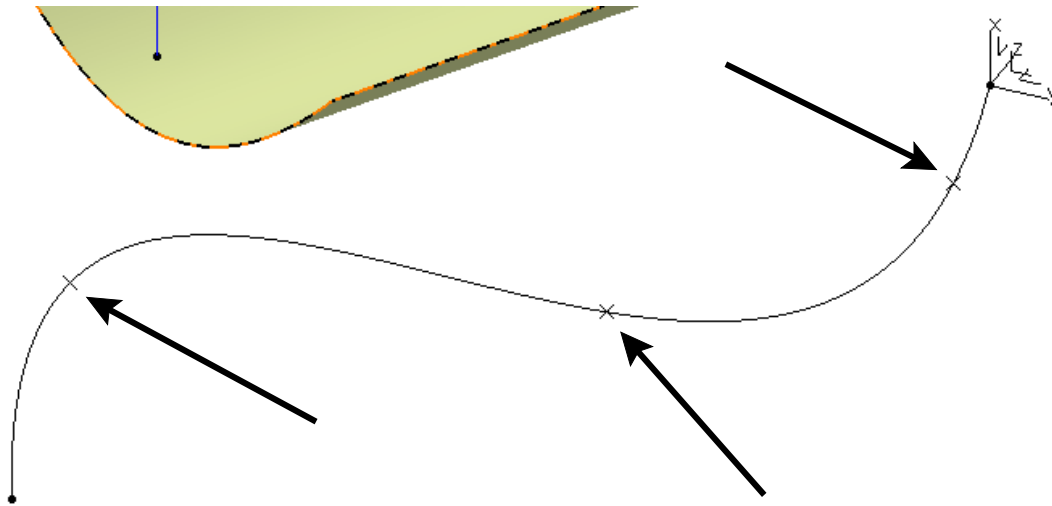
Change the *Offset* to 1.0. Notice that the dimension is measured normal from the *yz plane* along the curve.


Select *OK*. The point is created.

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

Turn on 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. You should note that only the *Distance on curve* option can use the *Euclidean* option.


Select the **Reverse Direction** button, then select **OK**. The point is created and should appear similar to the diagram shown below.



Select the **Point** icon again and make sure the **Point type** is set to **On curve**. Select the curve on the right. 

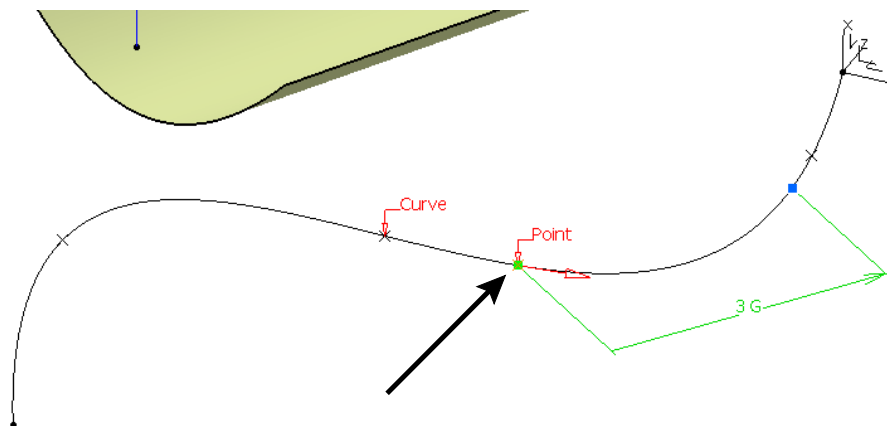
Select the **Nearest extremity** button. The point appears at the nearest endpoint of the curve.

Select the **Middle point** button and select **OK**. The point appears at the midpoint of the curve.

Select the **Point** icon again and make sure the **Point type** is set to **On curve**. Select the curve on the right.  This time you are going to use a reference point other than an extremity. All the other options work the same except the distance and ratio, which are based from the reference point.

Turn on **Distance on curve**, **Geodesic** and change the **Length** to **3.0**.

Select in the **Reference Point** box and select the point shown below. Notice the direction of the arrow.



Select the **Reverse Direction** button. The arrow points the opposite direction. If you were using an extremity then reversing the direction would cause the reference point to switch to the other end of the spline.

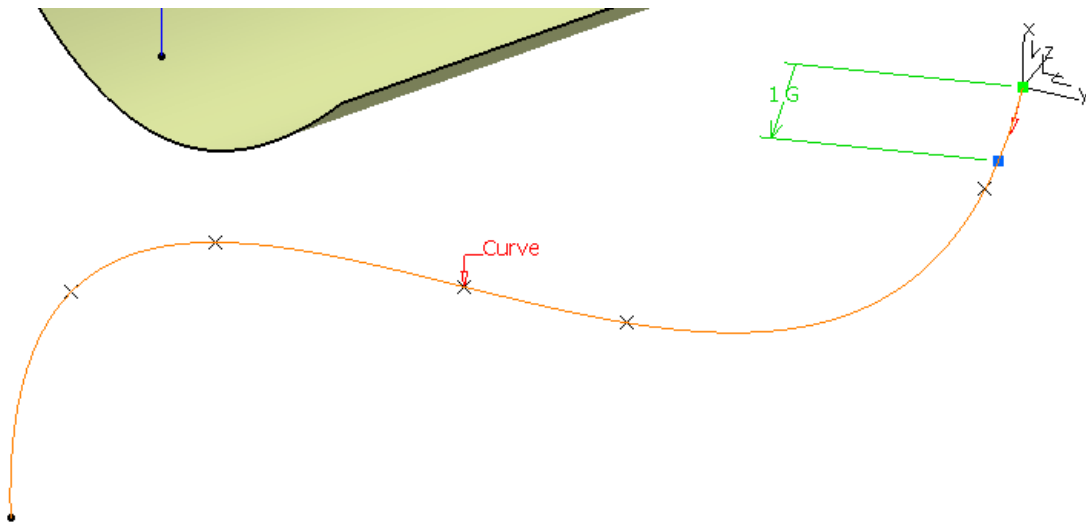
Select **OK**. The point is created.

Select the **Point** icon again and make sure the *Point type* is set to *On curve* and select the curve on the right.



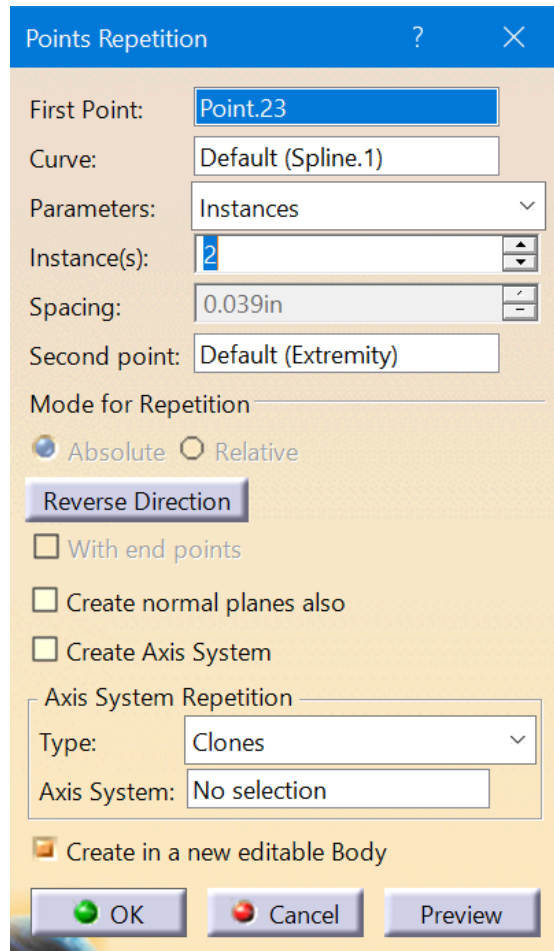
Turn on *Distance on curve, Geodesic* and change the *Length* to 1.0.

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



Turn on *Repeat object after OK* and select **OK**.

The *Points & Planes Repetition* window appears.



These options will be covered in detail a little later in the book. This was just meant to introduce a method for creating multiple points.

Select *Cancel*. Only the one point is created. This completes the options for creating a point on a curve.

Select off of the point in order to release it.


Surfaces

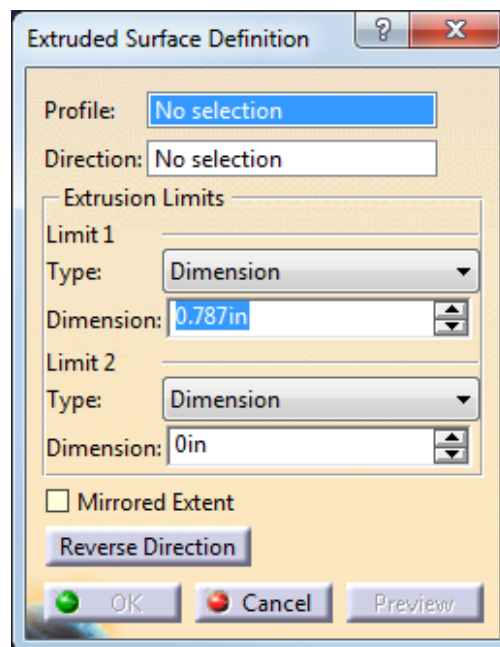
Surfaces are extremely important for defining any type of contour. Using wireframe geometry, you can create surfaces to represent any contour that you need. Once you have created the surface(s) that you need you can then use them in Part Design to contour your solid model. You have a variety of options to create surfaces. Some options are straightforward while others are much more involved.

Extruded

Extruded surfaces are created by extruding an element in a linear direction. The resulting object is called *Extrude*.

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

Select the **Extrude** icon.  An *Extruded Surface Definition* window appears.



Profile Specifies the shape that will be extruded

Direction Defines the direction of the extrusion

Extrusion Limits

Limit 1/2 Defines the direction and limits for the extrusion

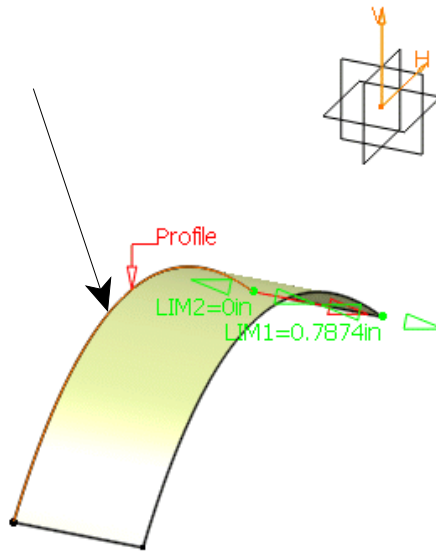
Type Specifies either a constant dimension or up to a selection

Dimension Specifies the limit distance

Mirrored Extent Extrudes the *Limit 2* dimension the same length as *Limit 1*

Reverse Direction Reverses the direction of the extrusion

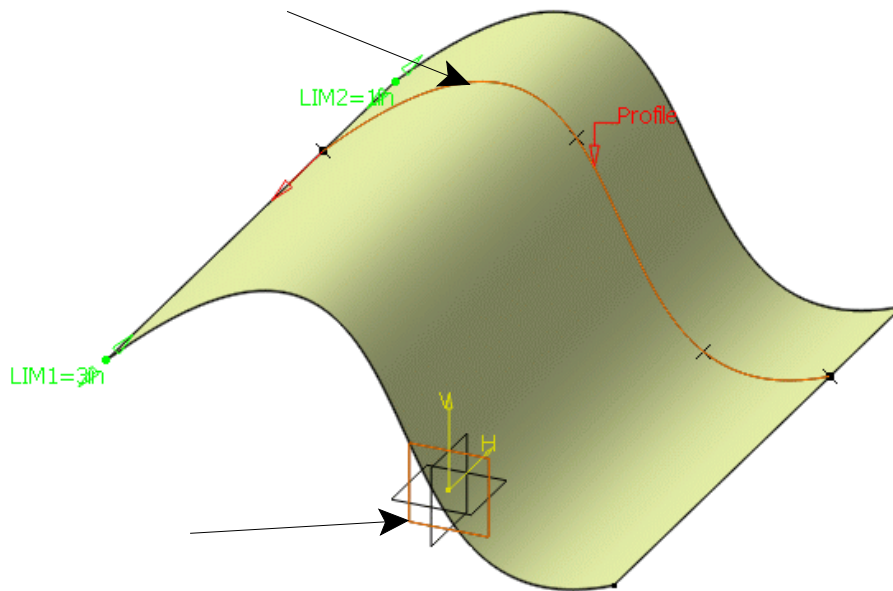
Select the curve as shown below. Since this curve was created in a sketch, the extrude option automatically assumes you want to go normal to the sketch.



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

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


Select the curve and plane as shown below. The plane defines the direction to be normal to the plane.

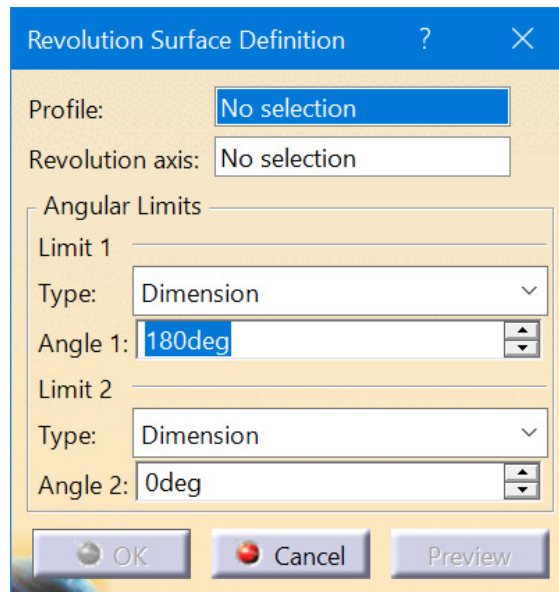


Change the limits so that both are 1.0 inch and select *OK*. The surface is created.

Revolution

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

Select the **Revolve** icon.  The *Revolution Surface Definition* window appears. The icon is located under the **Extrude** icon.



Profile Specifies the shape that will be revolved

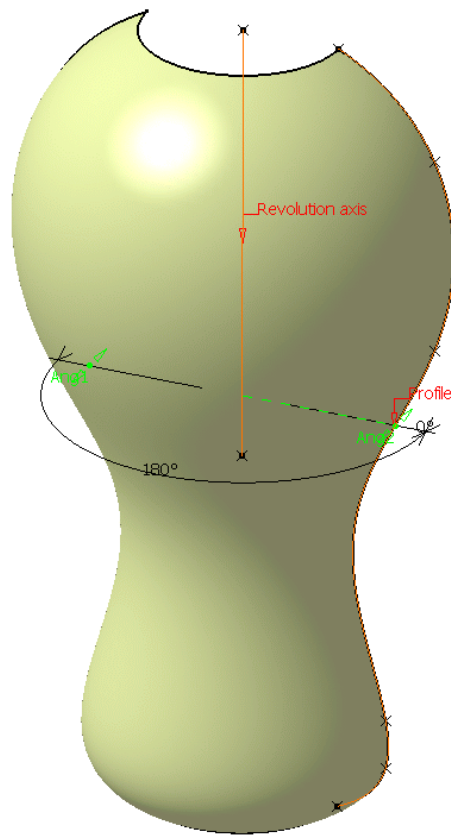
Revolution axis Defines the axis around which the profile will revolve. If your profile is a sketch and has an axis defined in it then that will be the default revolution axis.

Angular Limits

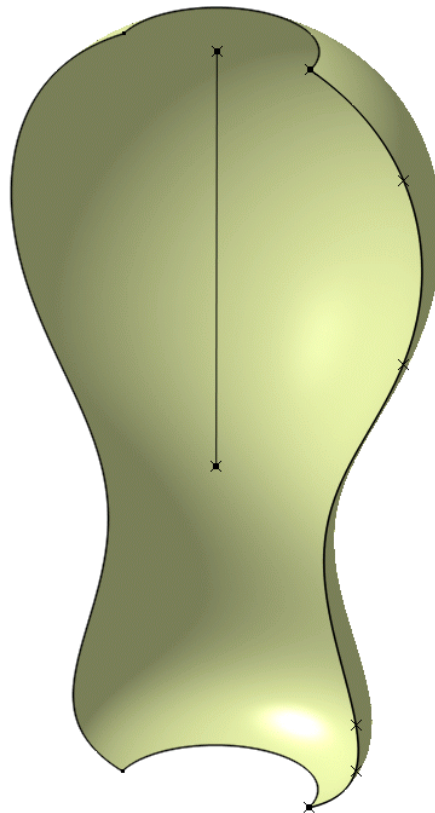
Limit 1 Specifies either a *Dimension* and starting angle for the revolution or an *Up-to element*

Limit 2 Specifies either a *Dimension* and ending angle for the revolution or an *Up-to element*

Select the profile and line as shown below.




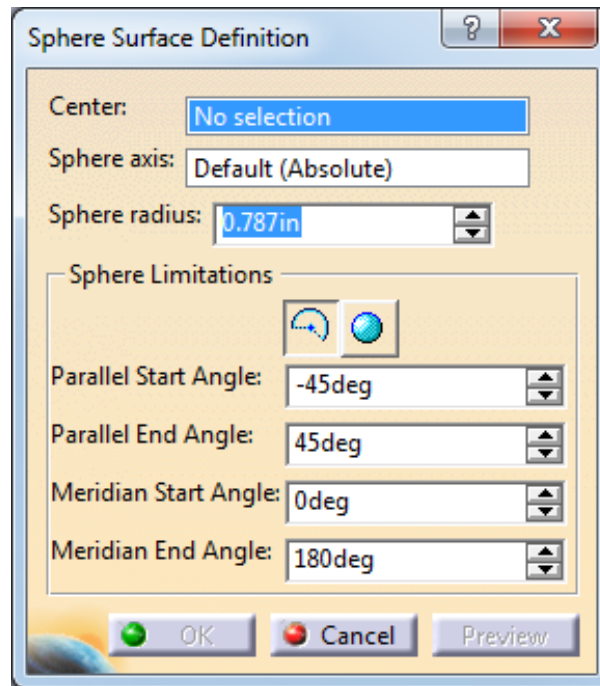
Change *Angle 1* to be 0.0 and *Angle 2* to be 180 and select *OK*. The surface is created.



Sphere

Sphere surfaces are created by defining a center point and a radius. The resulting object is called a *Sphere*.

Select the **Sphere icon**.  The *Sphere Surface Definition* window appears. The icon is located under the **Extrude** or **Revolve** icon.



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



With limits



Whole sphere

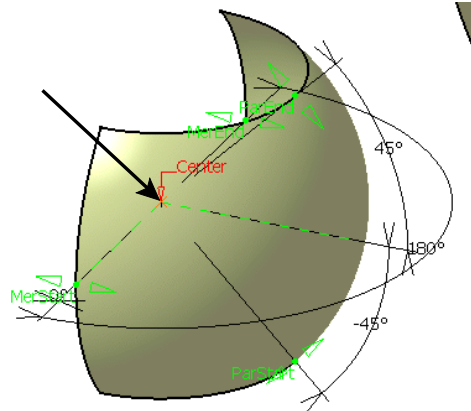
Parallel Start Angle Defines the starting angle in the parallel direction

Parallel End Angle Defines the ending angle in the parallel direction

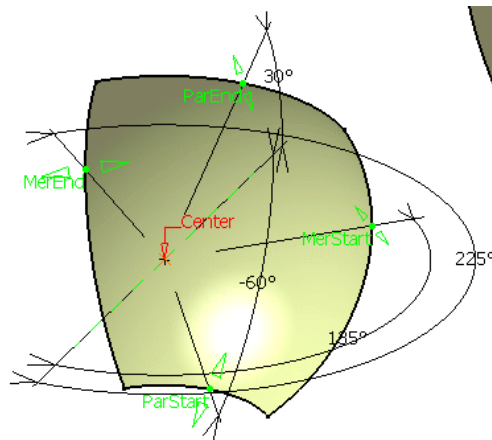
Meridian Start Angle Defines the starting angle in the meridian direction

Meridian End Angle Defines the ending angle in the meridian direction

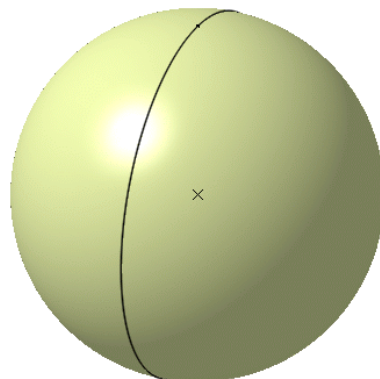
Select the point as shown below, key **1.5** for the *Sphere radius* and select *Preview*. Since you do not have any other axis to select, you will use the default. The *Parallel* limits have a range of -90 to 90 while the *Meridian* limits have a range of -360 to 360. Basically, the *Parallel* limits are the up and down limits and the *Meridian* limits are the left and right limits. Of course, this depends on your axis.



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, and select *Preview*.




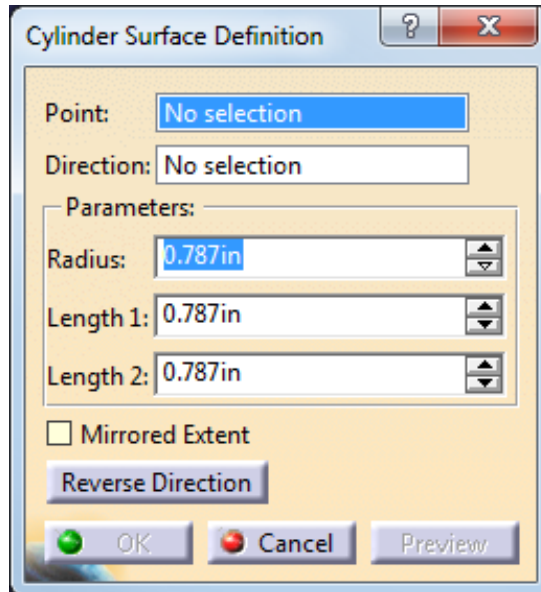
Select the **Create the Whole Sphere** icon from the window, then click **OK**.  All the limit options were made unavailable.



Cylinder

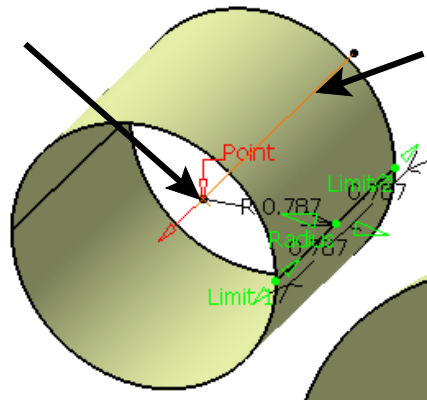
Cylindrical surfaces are created by defining a point and a direction, then you can specify a length and radius. The resulting object is called a *Cylinder*.

Select the **Cylinder icon**.  The *Cylinder Surface Definition* window appears. The icon is located under the **Extrude** or **Sphere** icon.



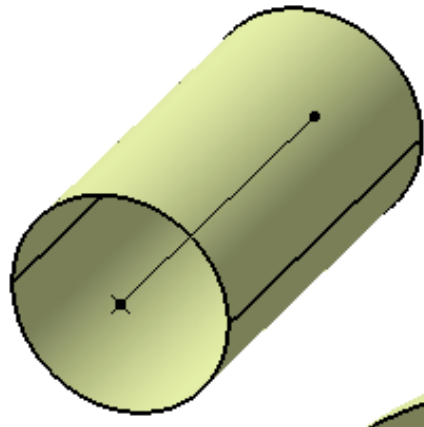
<i>Point</i>	Specifies the center point of the cylinder
<i>Direction</i>	Specifies the direction the cylinder will extrude
<i>Parameters</i>	
<i>Radius</i>	Defines the radius of the cylinder
<i>Length 1,2</i>	Defines the length of the cylinder in both directions
<i>Mirrored Extent</i>	Extrudes the <i>Length 2</i> dimension the same length as <i>Length 1</i>
<i>Reverse Direction</i>	Reverses the direction of the cylinder

Select the point and line as shown below.



Change the *Radius* to 0.5, *Length 1* to 2.0, *Length 2* to 0.0. The cylinder changes size.

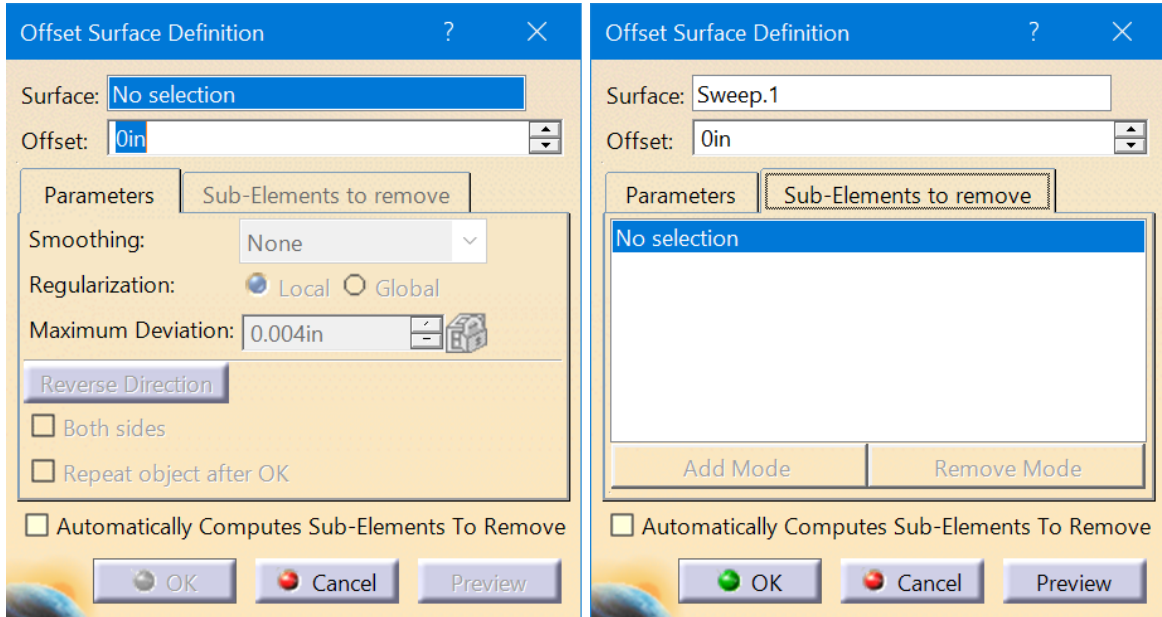
Select the *Reverse Direction* button and select *OK*. The surface is created.



Offset

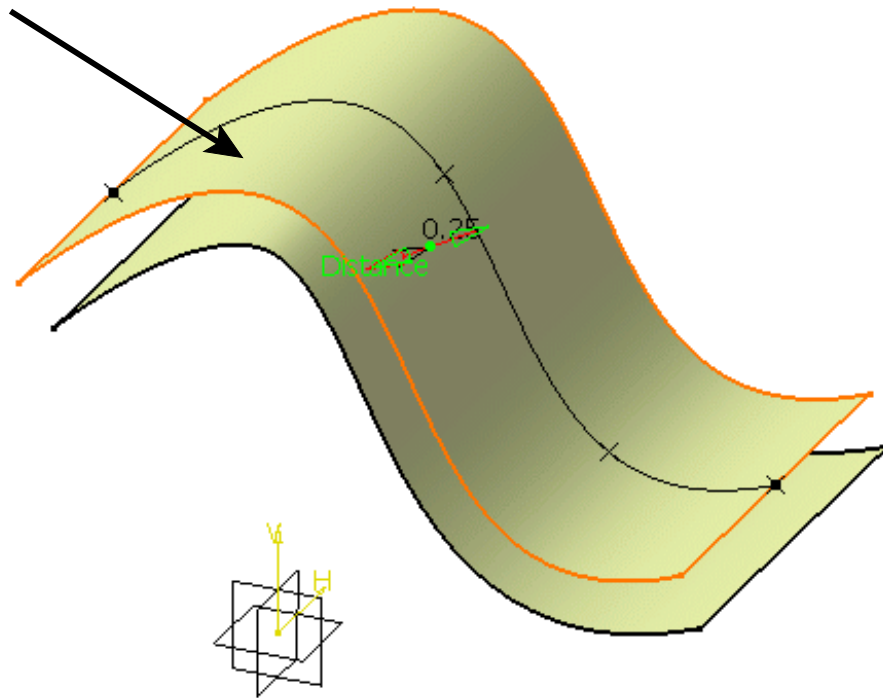
Offset surfaces are created by offsetting an existing surface a specified distance. The resulting object is called an *Offset*.

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



- Surface* Specifies the surface that will be offset
- Offset* Defines the distance of the offset
- Parameters*
 - Smoothing* A smoothing can be applied if a constant offset will not work. It will introduce deviation into 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 both directions
 - Repeat object after OK* Allows you to repeat the offset numerous times
- Sub-Elements To Remove* If an offset has problems, you can perform the offset without the sub-elements that have errors. The sub-elements will be listed in the *Sub-Elements To Remove* list. This is useful when trying to determine why an offset fails. You can *Add* or *Remove* sub-elements to the list.
- Automatically Computes...* Automatically computes sub elements to remove

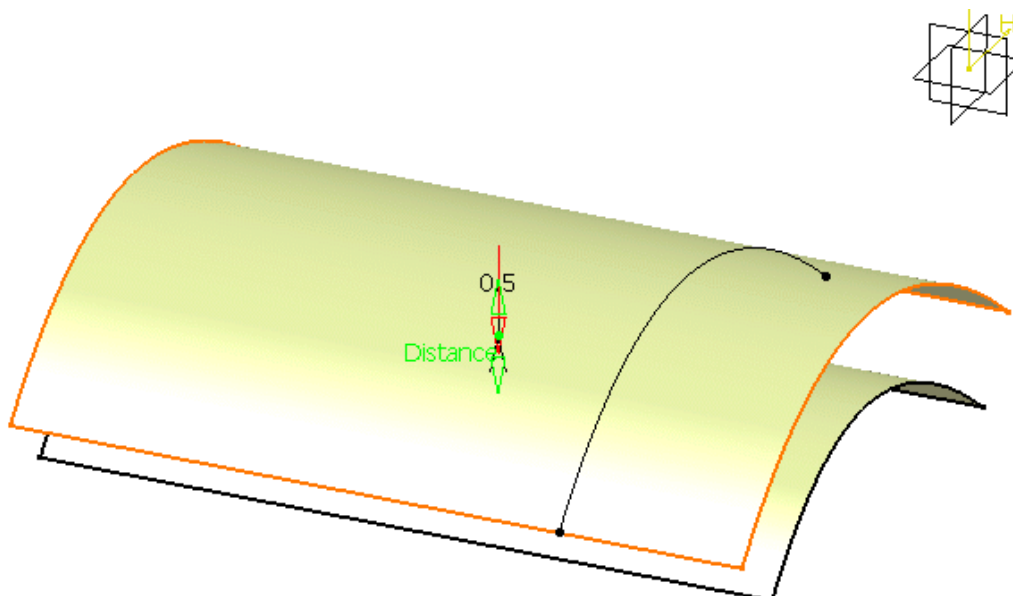
Select the surface as shown below, key **0.25** for the **Offset** and select **Preview**. An offset surface appears.



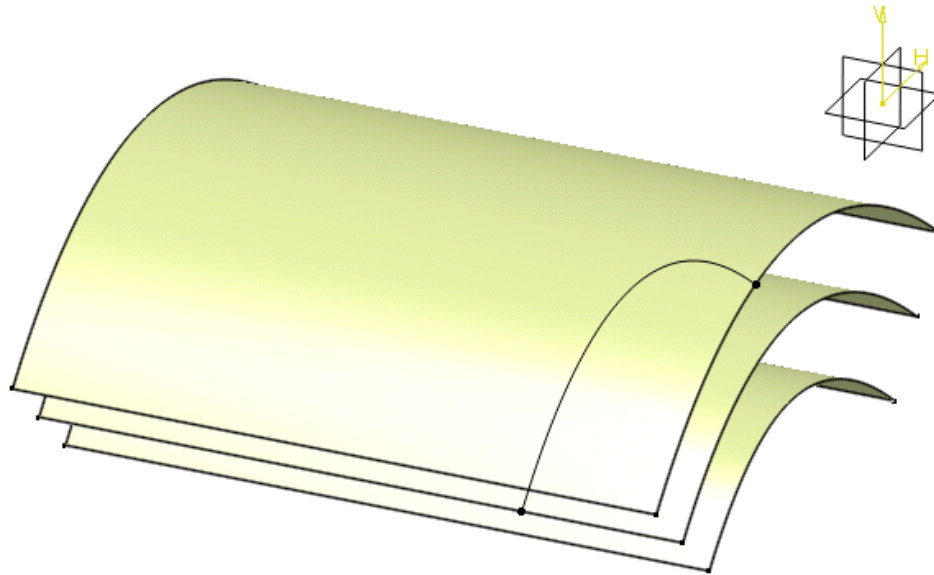
Select the **Reverse Direction** button and select **OK**. The offset surface appears above the original surface instead of below it.

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

Select the surface as shown below, key **0.5** for the **Offset** and 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, you can use the Object Repetition icon on offset surfaces.



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.

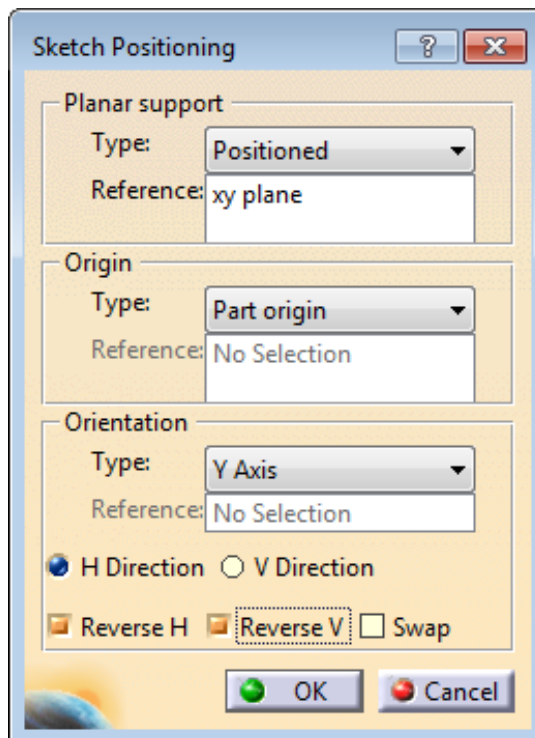
Note: Set your view mode to Shading With Edges Without Smooth Edges in order to obtain the same results shown in the following images.



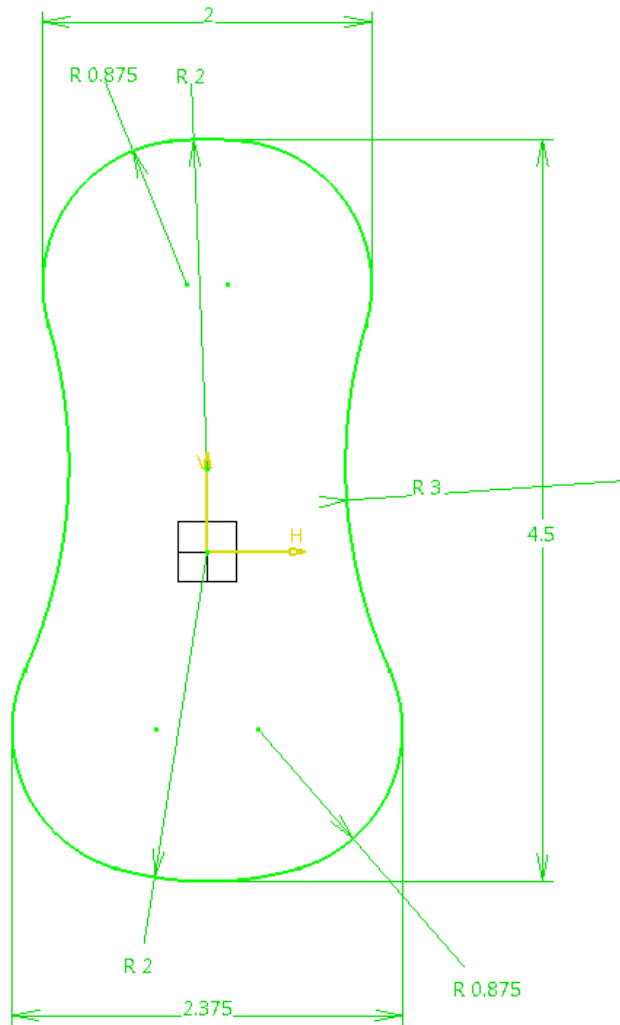
Mouse Body

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

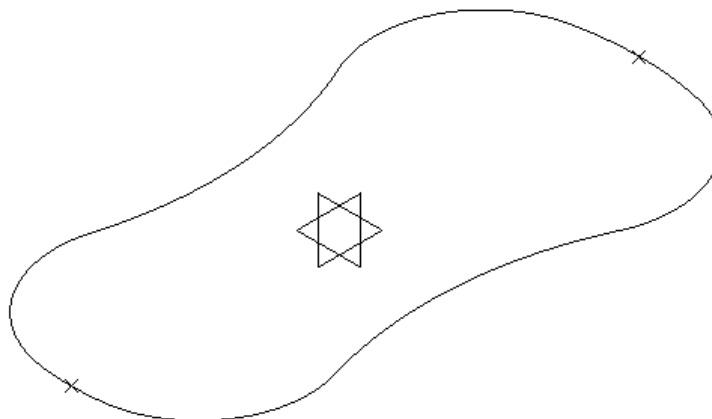
Insert a geometrical set named **Mouse Body**, then select the **Positioned Sketch** icon and set the options as shown below.



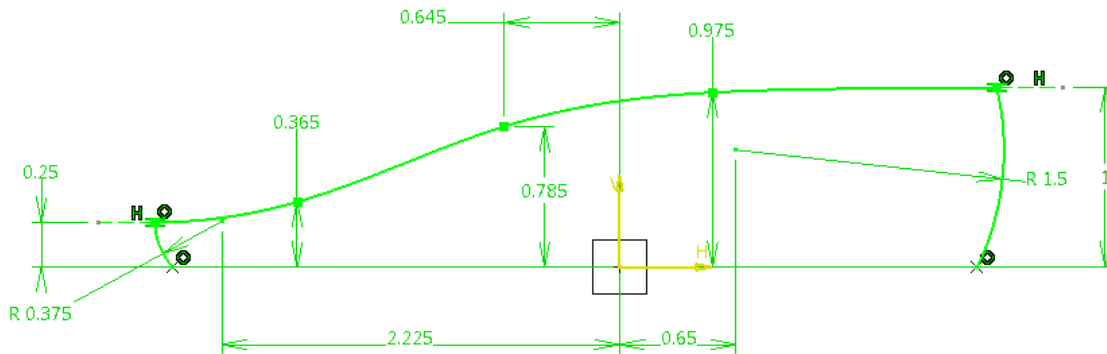
Create the following sketch. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.



Create an extremum point at each end of the sketch in the yz direction.

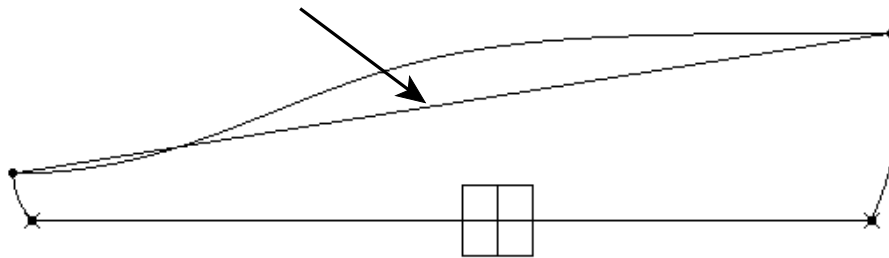



Create the following sketch on the zx plane.  The bottom of the arcs are coincident to the extremum points.

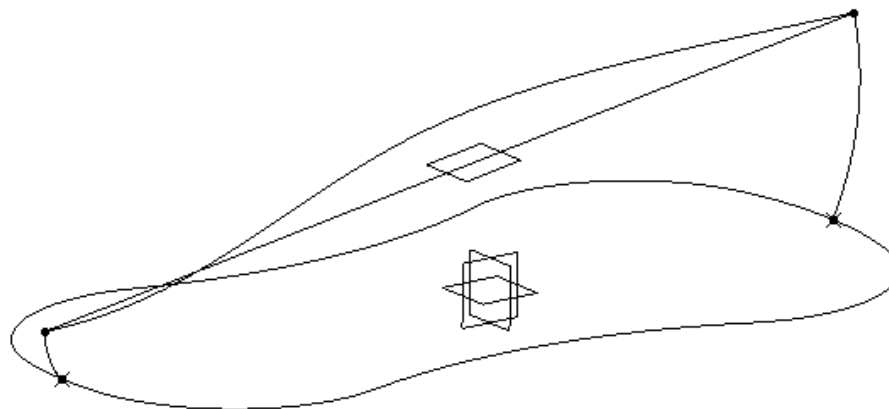


Extract each curve from the sketch. 

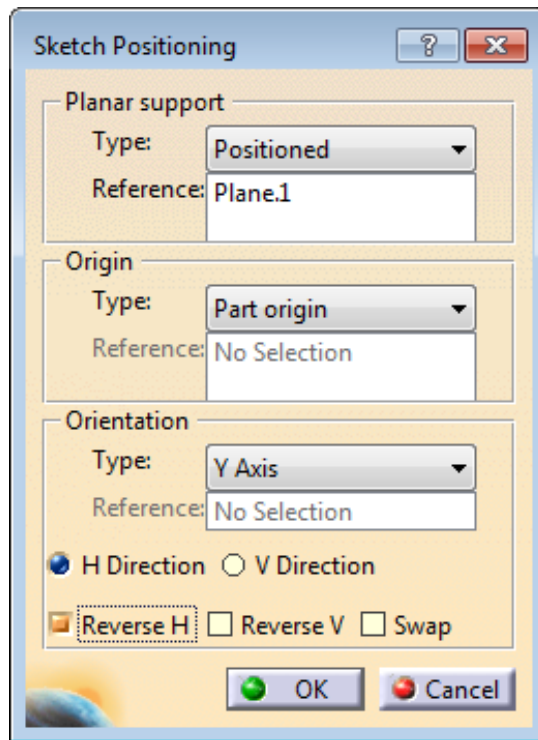
Create a *Point-Point* line between the top points of the arcs. 



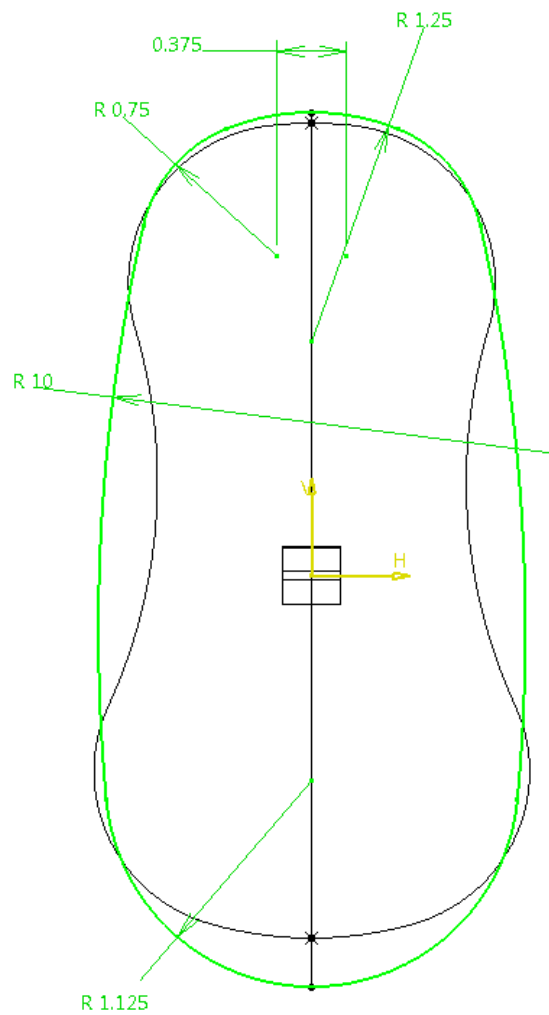
Create a plane through the line.  Use the line for the *Rotation axis* and the zx plane for the *Reference*. The plane should be normal to the reference.



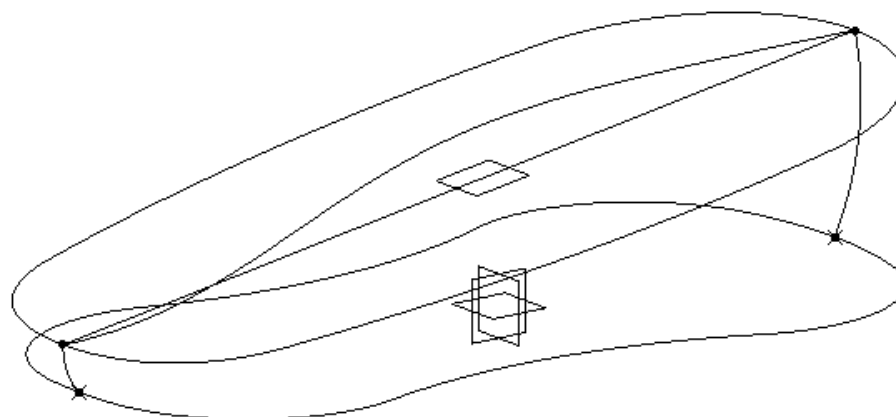
Select the **Positioned Sketch** icon and set the options as shown below.




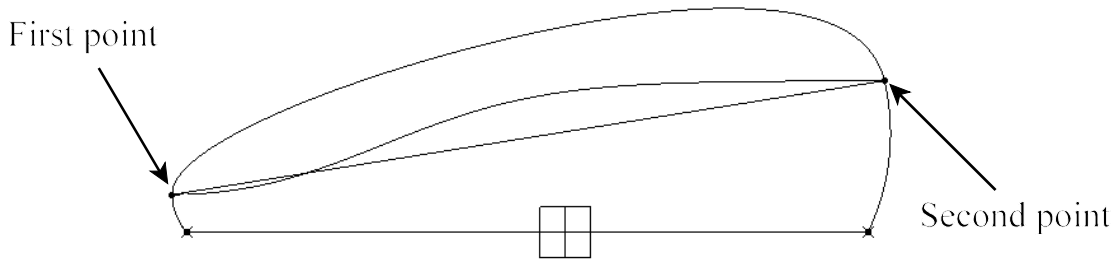
Create the following sketch. The top and bottom arcs in this sketch are coincident to the upper end points of the extracted arcs. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.




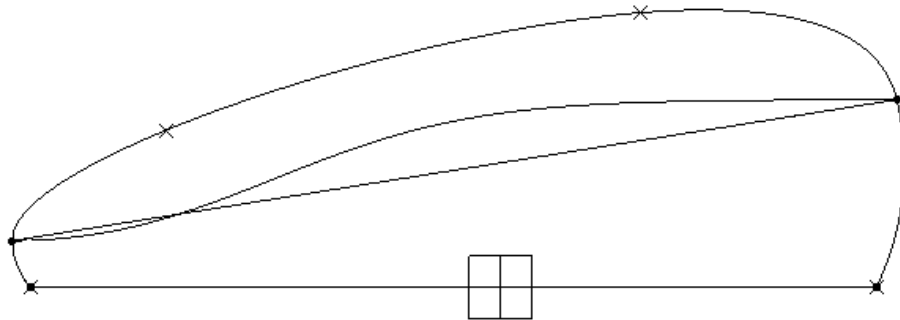
Your model should look like this.



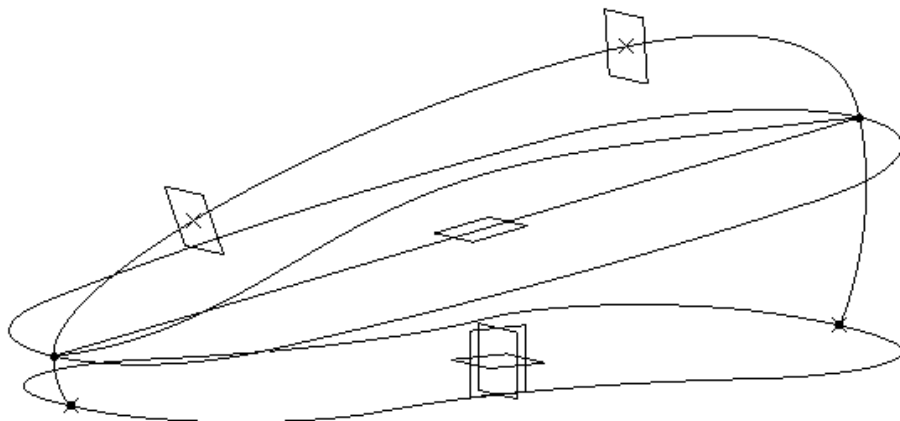
Create a spline between the two points at the top of each extracted arc.  The spline will be tangent continuous to both arcs with a tension of 0.375 at the first point, and 0.75 at the second point.



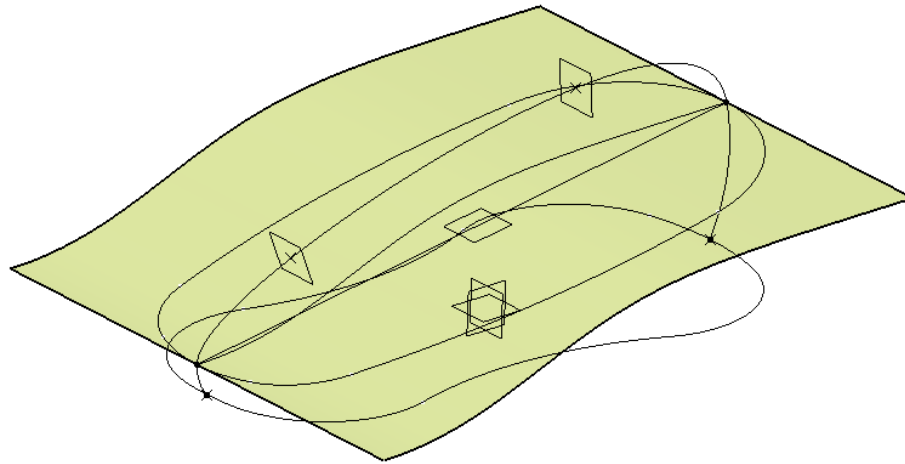
Create two geodesic points on the new spline.  The left point will have a ratio value of 0.2 from the left end of the spline, and the right point will have a ratio value of 0.3 from the right end of the spline.



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 CHANNEL SURFACE in the specification tree.



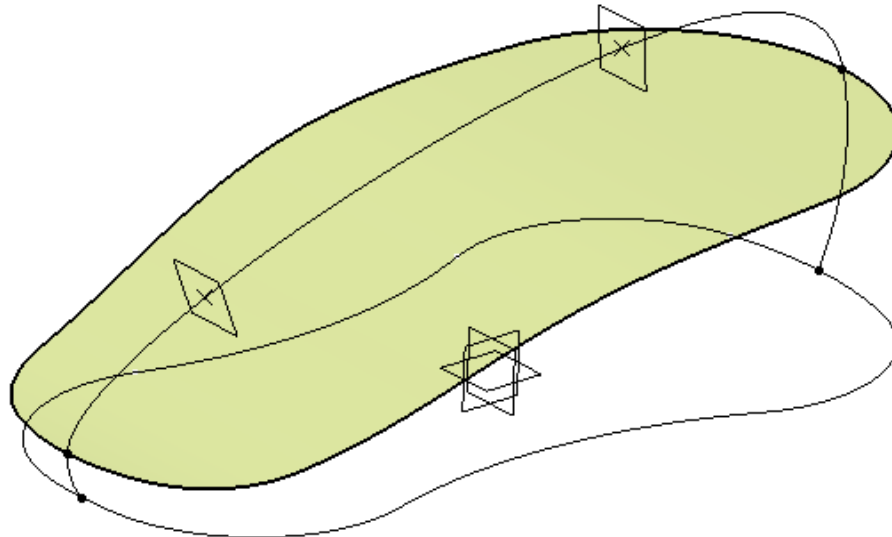
Project the last sketch you created to *CHANNEL SURFACE* along the normal direction of the first plane created.



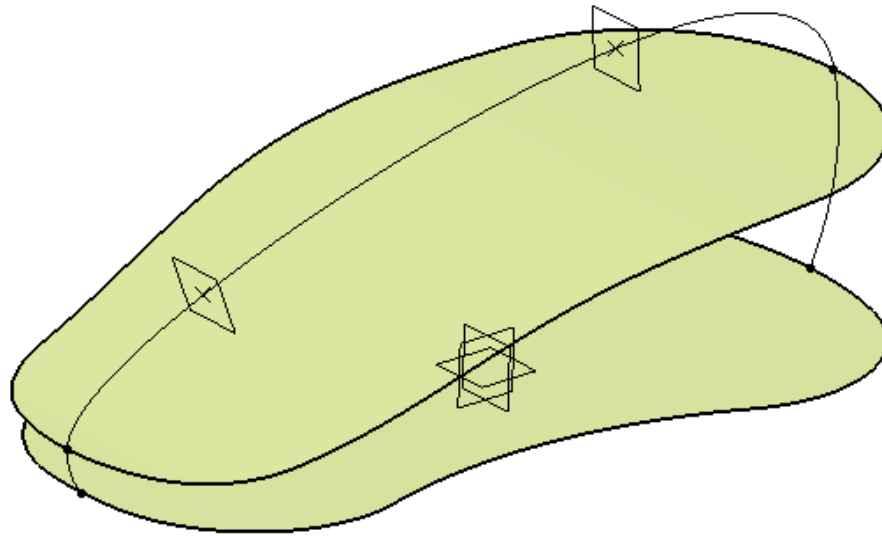
Split *CHANNEL SURFACE* with the projected curve, keeping the inner portion.



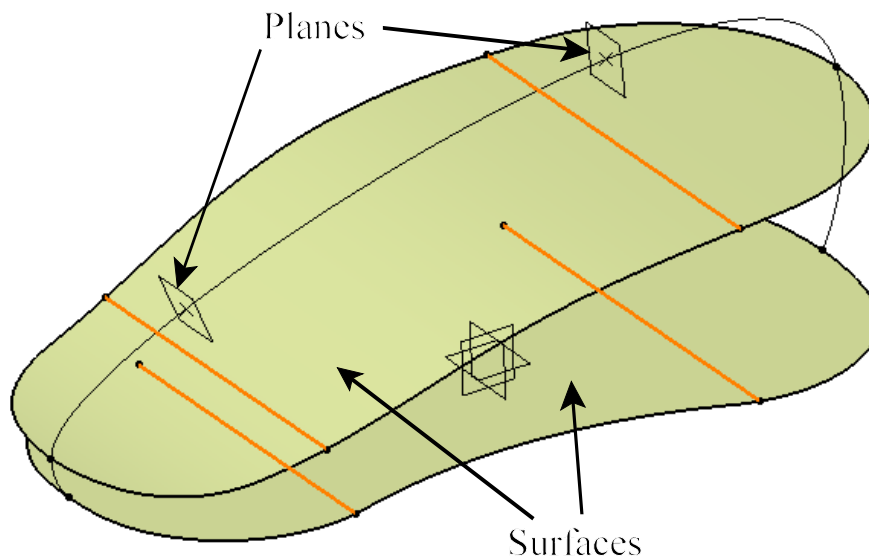
The split surface is shown below. Much of the geometry has been hidden. Feel free to hide your geometry as necessary to reduce clutter.



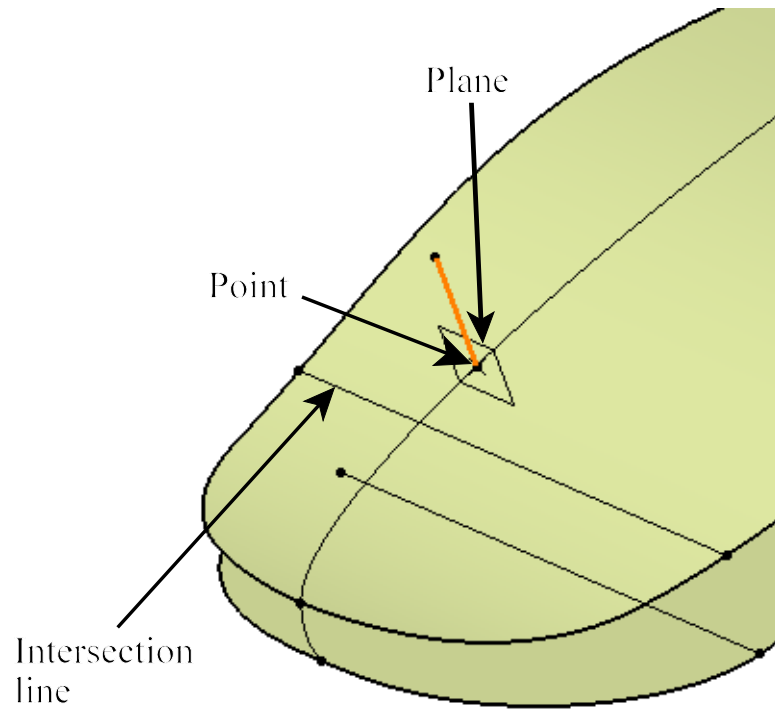
Fill your first sketch with a surface.  This is the bottom profile of the mouse.



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



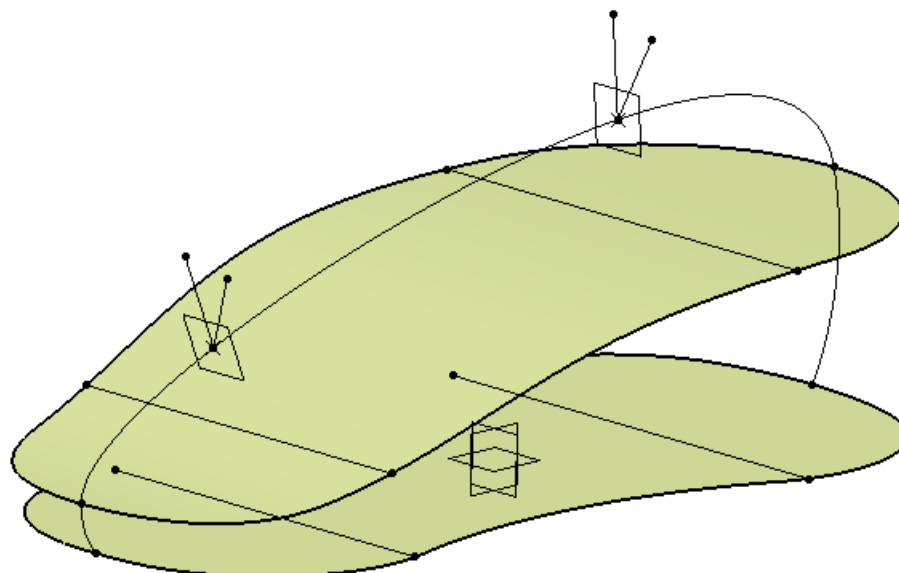
Create a 0.5 inch line normal to the intersection line indicated below that starts at the point and uses the plane as its support. 




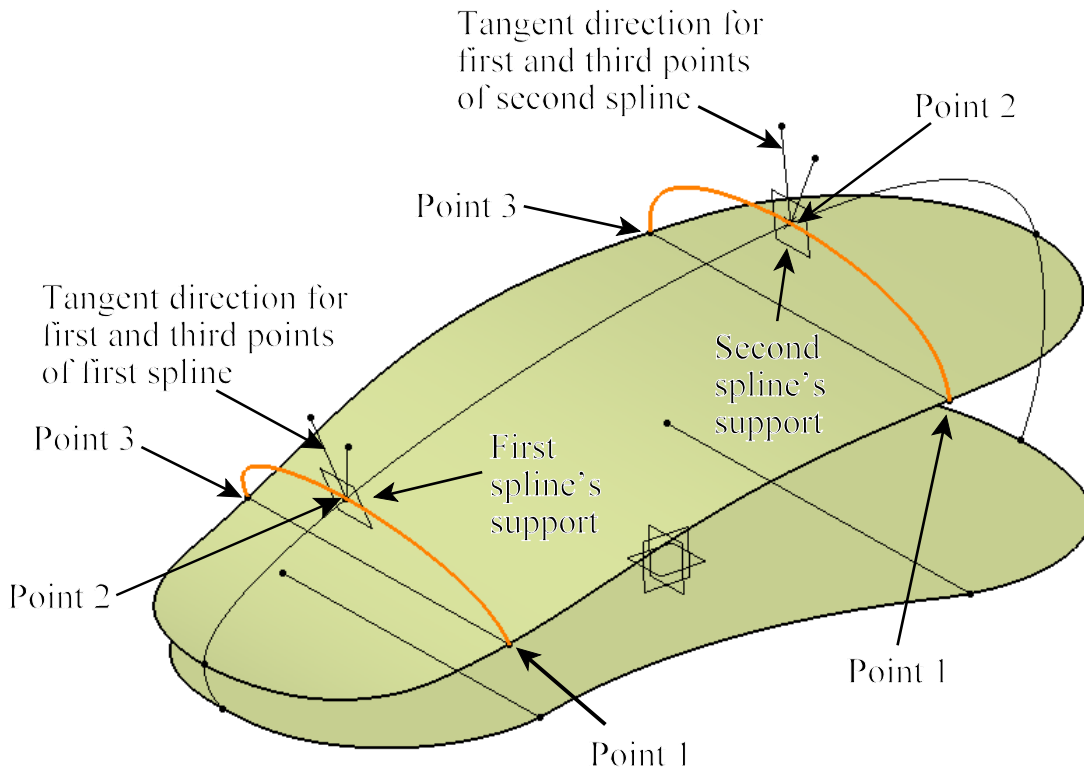
Create another line using the same geometry, but this time specify an *Angle* of -60. 


Ensure the line extends upward.

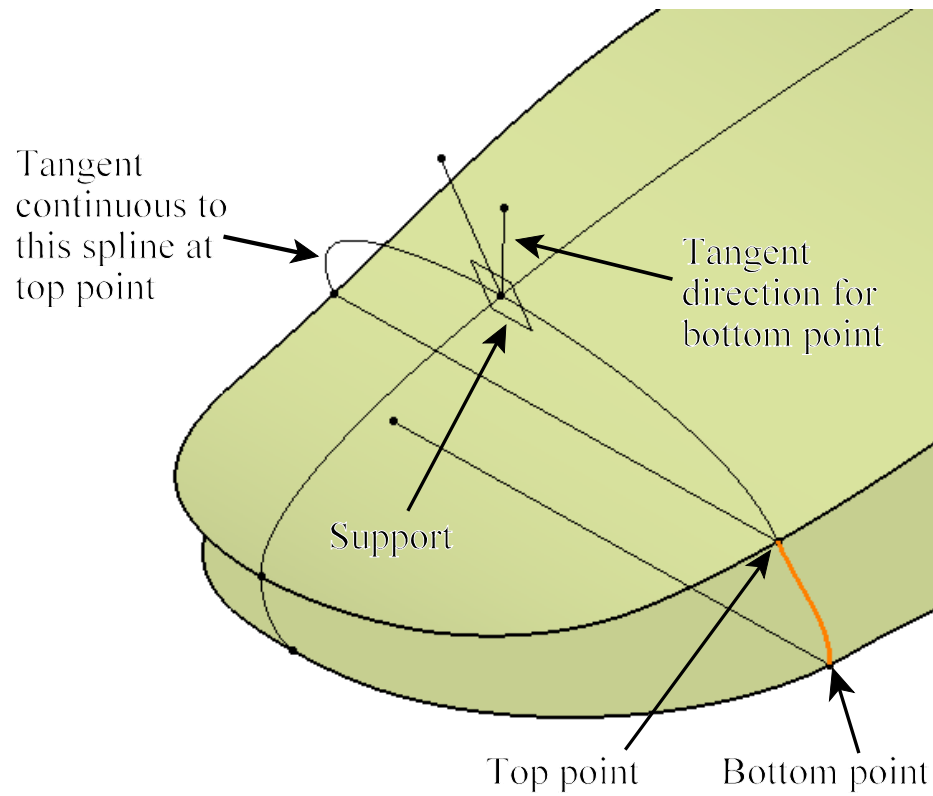
Repeat this process at the other point so that your model looks the same as below.



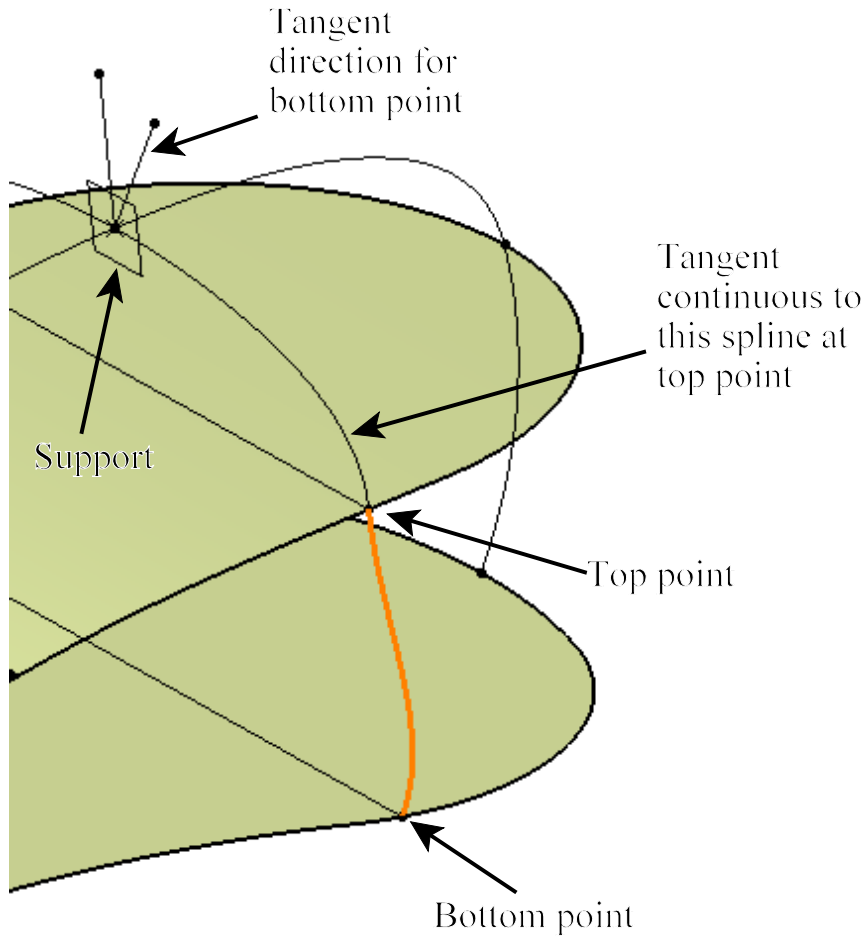
Create the two splines shown below.  The tension at all three points for both splines should be 1.0. Use the normal lines you just created for the tangent direction of the first and third points, and the zx plane for the tangent direction of the second points. Ensure each spline lies on the appropriate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



Next, create the orange 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.

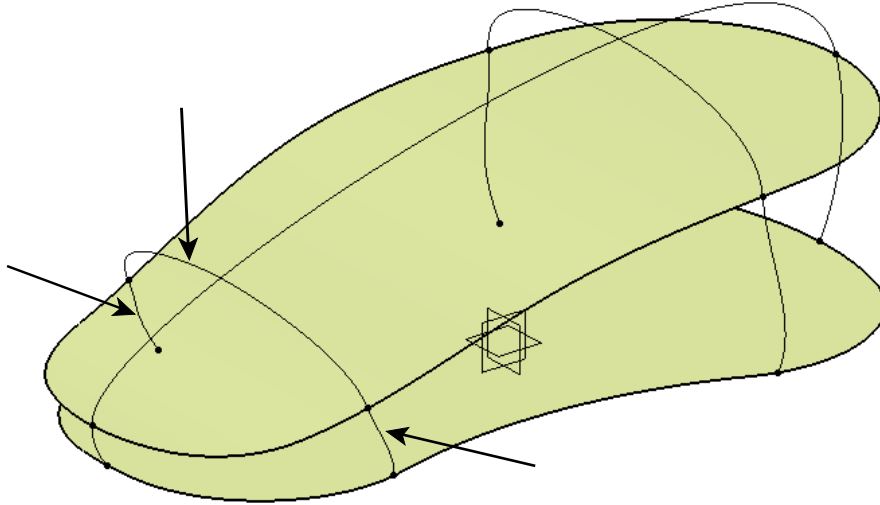



Create the orange spline shown below using the same method as the previous spline. It is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.25 at the bottom point. Ensure the spline lies on the support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

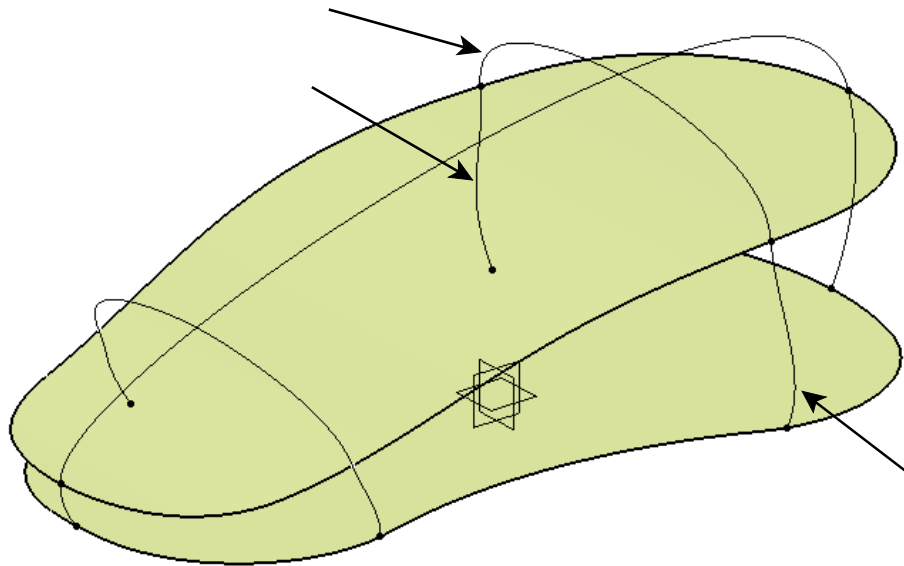


Mirror each of the last two splines created across the zx plane. 

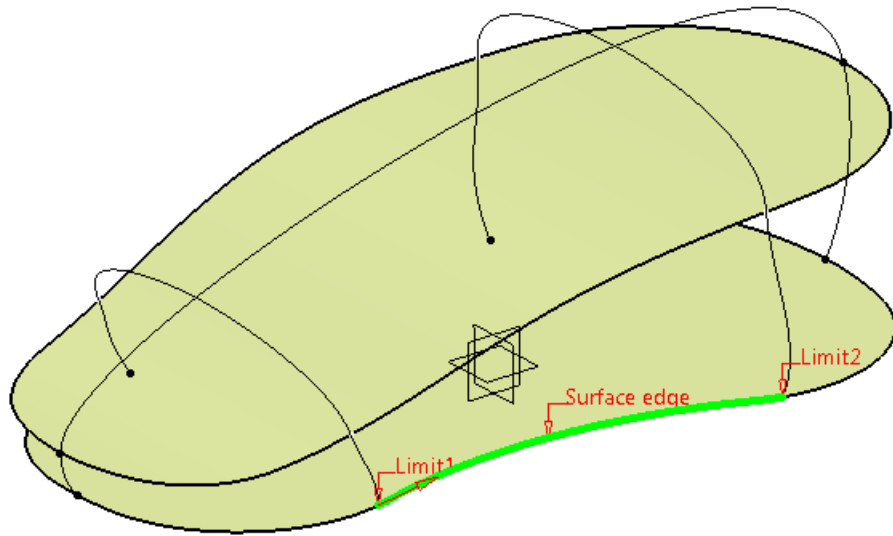
Join the three curves indicated below.  Ensure they are tangent continuous.




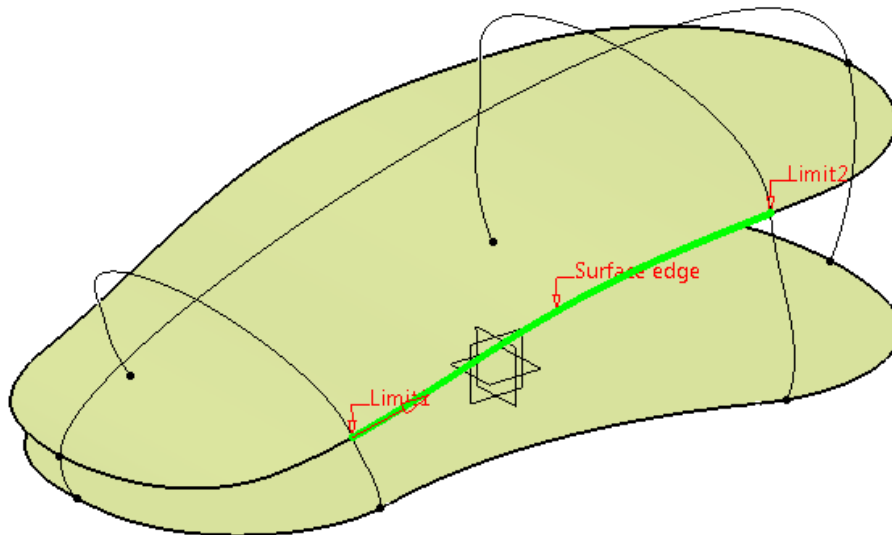
Create another join for the three curves shown below.  Ensure they are tangent continuous.



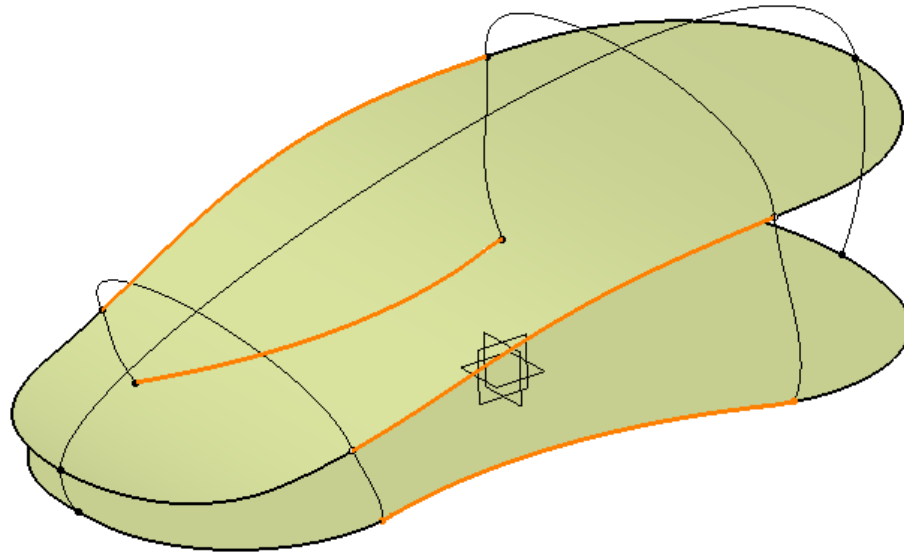
Create a boundary curve on the following edge. 




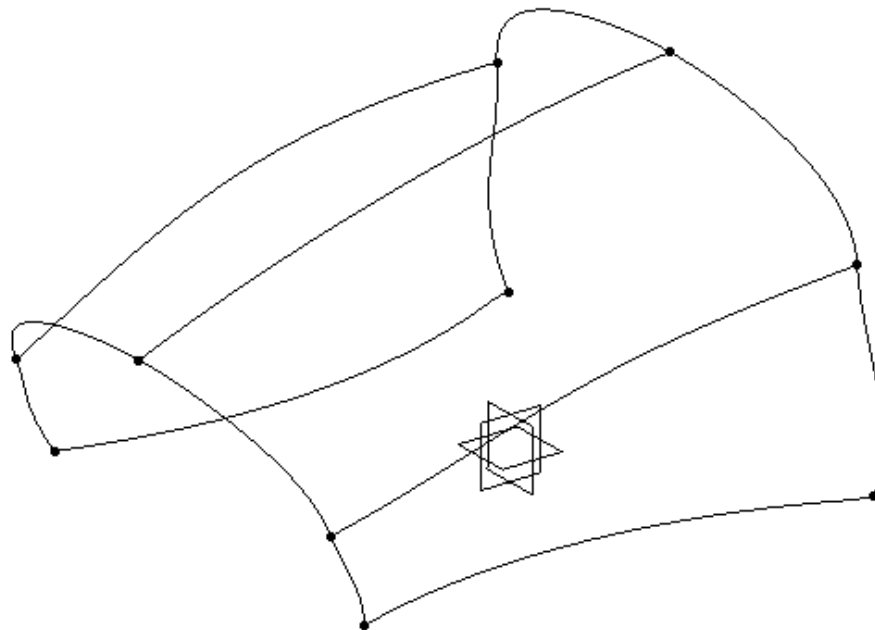
Create another boundary curve on the edge shown below. 




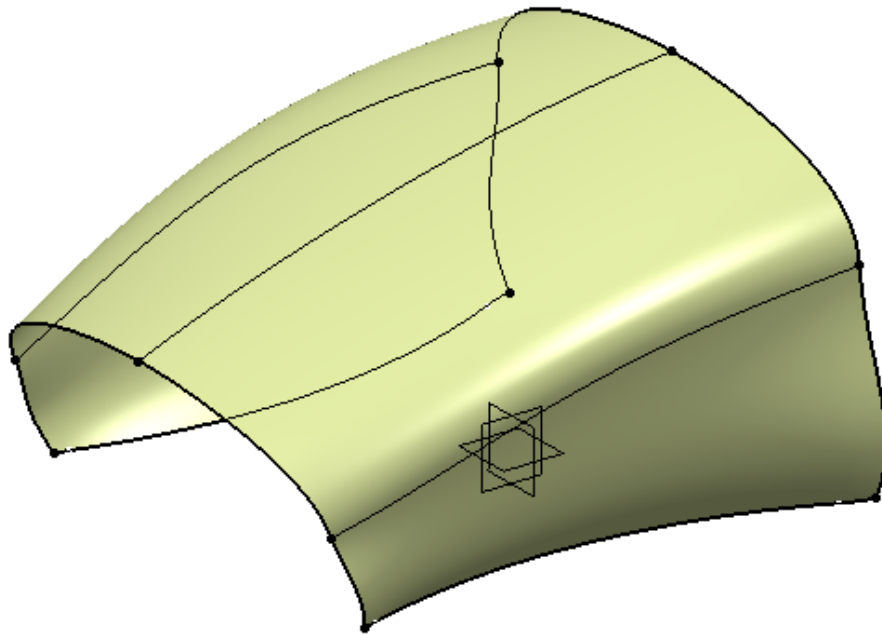
Create the same two boundaries on the opposite side of the surfaces.  Only the boundary curves, the joined curves, the split surface, and the filled surface are shown below.



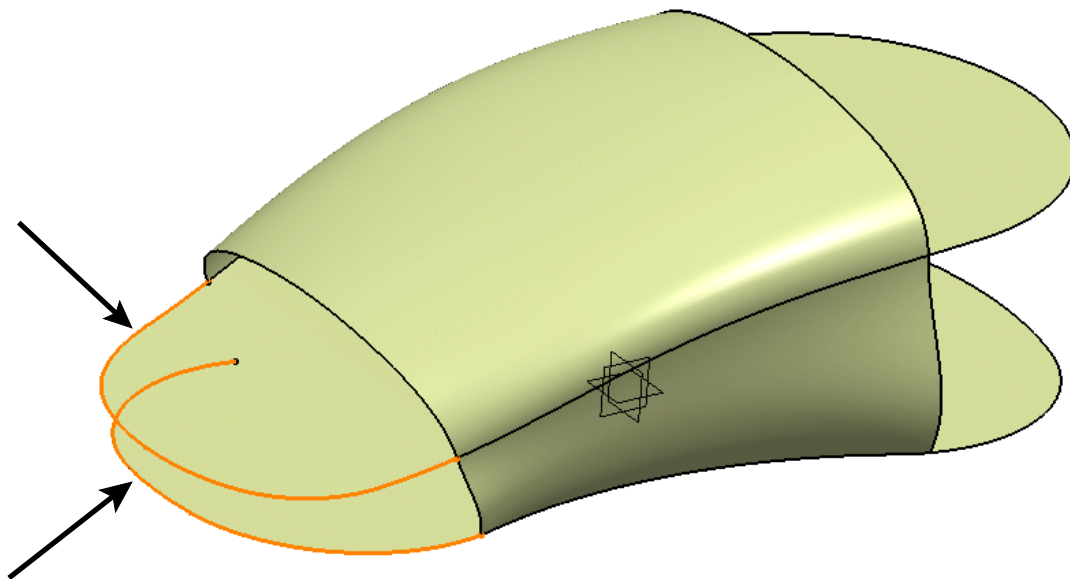
Split the upper spline at its normal planes, then hide all geometry except the boundary curves, the joined curves, and the new split. 




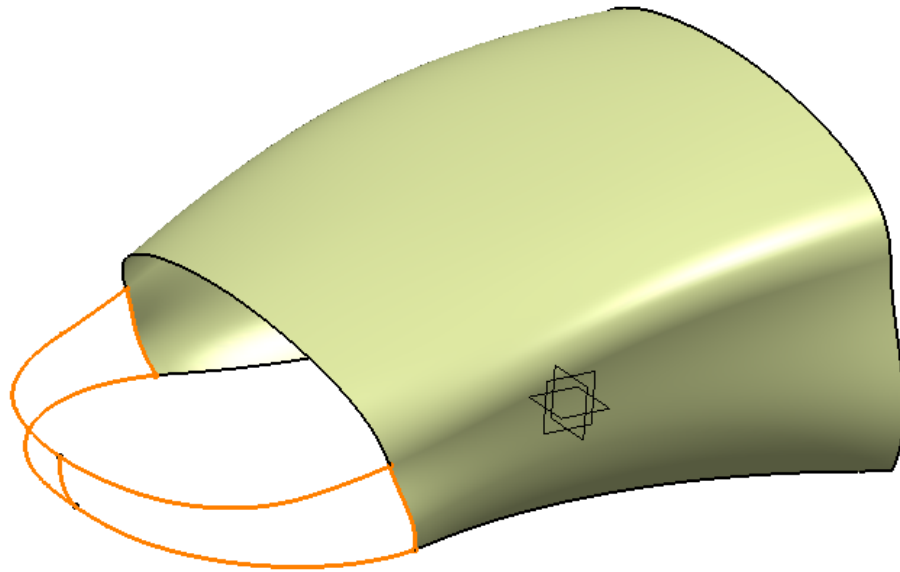
Create a multi-section surface using the joined curves as sections, and the boundary curves and split curve as guides. 



Next, create the two boundary curves shown below. 



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.

