TABLE OF CONTENTS

Introduction 1
Wireframe & Surfaces 2
Generative Shape Design Workbench 3

Wireframe 17
Points 17
Coordinate 17
On curve 20
On plane 24
On surface 26
Circle / Sphere / Ellipse center 28
Tangent on curve 29
Between 32
Point Repetition 34
Projecting points 39
Intersection Points 43
Extremum 46
Polar Extremum 49
Lines 51
Point-Point 51
Point-Direction 54
Angle/Normal to curve 56
Tangent to curve 61
Normal to surface 64
Bisecting 66
Intersection lines 68
Projecting lines 69
Axis 70
Polyline 72
Planes 76
Offset from plane 76
Parallel through point 78
Angle/Normal to plane 79
Through three points 81
Through two lines 82
Through point and line 83
Through planar curve 84
Normal to curve 85
Tangent to surface 87
Equation 88
Mean through points 90
Between 91
Plane Between 93

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Circles 95
Center and radius 95
Center and point 98
Two points and radius 99
Three points 102
Center and axis 104
Bitangent and radius 105
Bitangent and point 107
Tritangent 109
Center and tangent 111
Corners 112
Curves 117
Connect Curves 117
Conics 122
Splines 129
Helixes 135
Spirals 142
Project Curves 146
Combine Curves 149
Reflect Line Curves 153
Silhouette Curves 156
Intersection Curves 159
Parallel Curves 163
Rolling Offset Curve 169
3D Curve Offset 171
Curve Comparison 174
Supports 177
Work on Support 177
Creation on the fly 183
Modifying 185
Datums 187
Object repetition 189

Surfaces 191
Extruded 191
Revolution 193
Sphere 195
Cylinder 198
Offset 200
Mid Surface 203
Swept Surfaces 213
Explicit 213
With reference surface 214
With two guide curves 222
With pulling direction 227

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Linear . . . 229
Two limits . . . 230
Limit and middle . . . 235
With reference surface . . . 237
With reference curve . . . 240
With tangency surface . . . 242
With draft direction . . . 245
With two tangency surfaces . . . 249
Circular . . . 252
Three guides . . . 253
Two guides and radius . . . 255
Center and two angles . . . 257
Center and radius . . . 259
Two guides and tangency surface . . . 261
One guide and tangency surface . . . 263
Limit curve and tangency surface . . . 265
Conical . . . 268
Two guide curves . . . 269
Three guide curves . . . 271
Four guide curves . . . 273
Five guide curves . . . 275
Adaptive Sweep . . . 277
Fill Surfaces . . . 283
Multi-Section Surfaces . . . 290
Blend surfaces . . . 301
Spines . . . 308
Isoparametric Curves . . . 313
Laws . . . 315

Operations . . . 331
Joining Elements . . . 331
Healing Surfaces . . . 340
Curve smoothing . . . 345
Splitting Elements . . . 346
Trimming Elements . . . 353
Untrimming Elements . . . 357
Disassembling Elements . . . 359
Disconnect . . . 361
Sew Surface . . . 363
Remove Face . . . 365
Extracting Boundaries and Faces . . . 367
Multiple Extract . . . 372
Fillets . . . 376
Shape Fillet . . . 376
Edge Fillet . . . 383
Face-Face Fillet . . . 389
Tritangent Fillet . . . 391
Styling Fillet . . . 394
Automatic Filleting . . . 401

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Chamfer 406
Transformations 411
Translate 411
Rotate 414
Symmetry 417
Scale 418
Affinity 420
Axis to Axis 423
Patterns 424
Rectangular 424
Circular 427
User Defined 429
Extrapolating Curves and Surfaces 430
Inverting the Orientation 439
Creating the Nearest or Farthest Element 440
Multi-Selection 442

Analysis 447
Connect Checker Analysis 447
Light Distance Analysis 458
Draft Analysis 460
Surfacic Curvature Analysis 464
Porcupine Curvature Analysis 469
Geometric Information 479
Dress-Up 481

Geometrical Set Management 483
Inserting Sets 483
Changing Sets 485
Hiding and Showing Components 487
Activating and Deactivating Components 488
Reordering Components 489
Groups 491
Isolating Geometry 493
Copying and Pasting Geometry 494

Ordered Geometrical Set Management 497
Inserting an Ordered Geometrical Set 497
Modifying Children 499
Reordering Components 501
Scanning Ordered Sets 503
Inserting in an Ordered Set 504
Switching to a Regular Geometrical Set 505

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Miscellaneous 506
Parents/Children 506
Historical Graph 508
Quick Select 510
Inserting Elements 512
Sets of planes 513
Regular 513
Semi-Regular 515
Irregular 516
Keep and No Keep Mode 518
Keep Mode 518
No Keep Mode 518
Current Body 523
Masks 525
2D Visualization Modes 530
Deleting Useless Elements 534

Review 535
Mouse Body 535
Buttons and Wheel 557

Problems 569
Problem #01 569
Problem #02 570
Problem #03 571
Problem #04 572
Problem #05 574
Problem #06 576
Problem #07 578
Problem #08 580
Problem #09 583
Problem #10 585

Appendix A 589
App Preferences - 3D Modeling - 3D Modeling Core - Generative Shape Design - General 589
App Preferences - 3D Modeling - 3D Modeling Core - Generative Shape Design - Work On Support 591

Appendix B 593
Part Design Using Surfaces 593
Split 593
Thick Surface 595
Close 598
Sew 599
Pad/Pocket 601
Boolean Operations 602

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Appendix C 605
Developed Shapes 605
Unfold Surfaces 605
Transfer 610
Develop Wires 613

Appendix D 619
Volumes 619

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Introduction

CATIA 3DEXPERIENCE 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

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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.

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Wireframe

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

Points

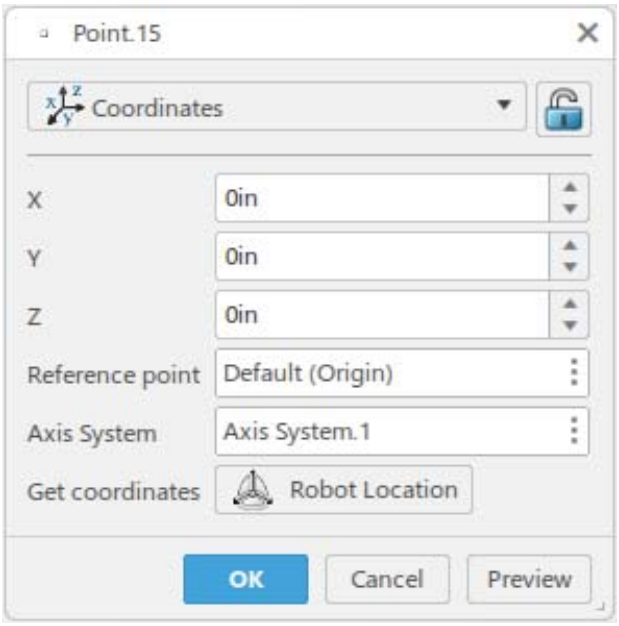
Points are useful to define specific locations and to assist in the creation of other wireframe geometry. You have a variety of options to define points that will be explored in the following exercises.

Coordinate

Open the WFAS - Points document. Remember, you will need to search for WFAS - Points 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*.

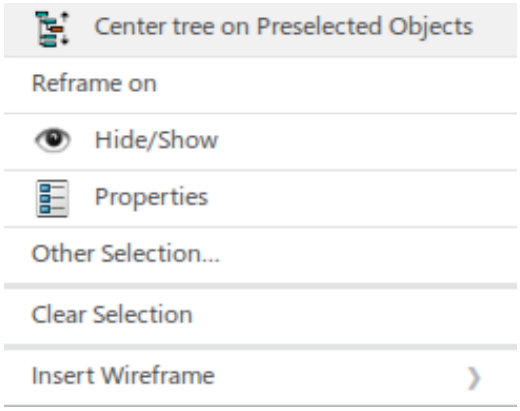
Select the Point icon. The *Point* window appears.



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

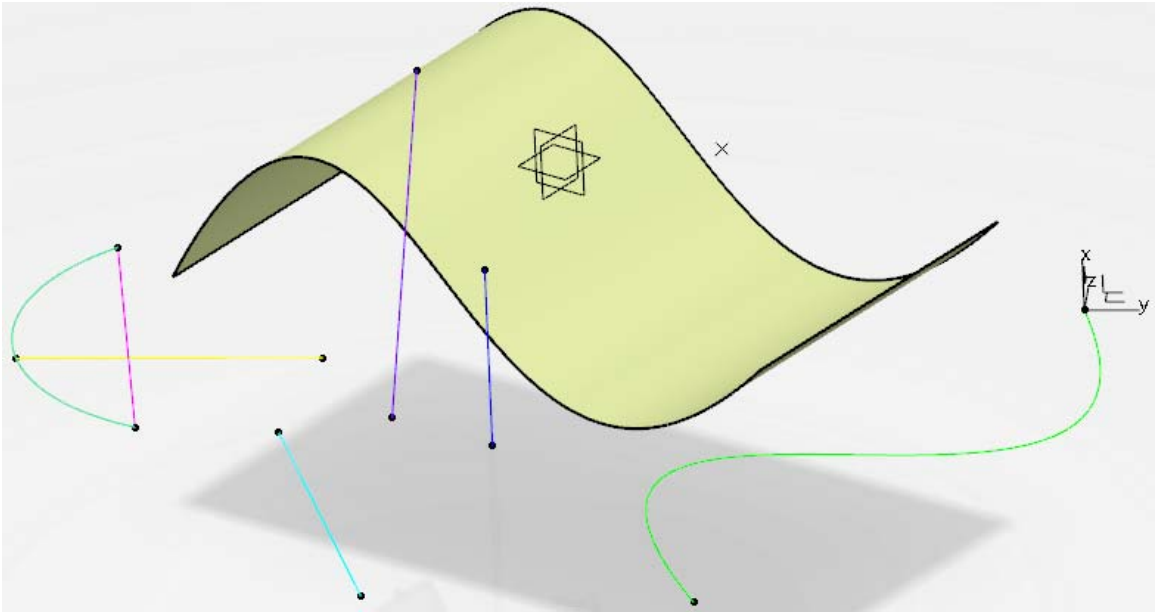
Reference point	The point that the coordinates are based from; the default is the origin
Axis System	Defines the axis system that the point will be based off of
Robot Location	Creates the point at the location of the robot in the display

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.



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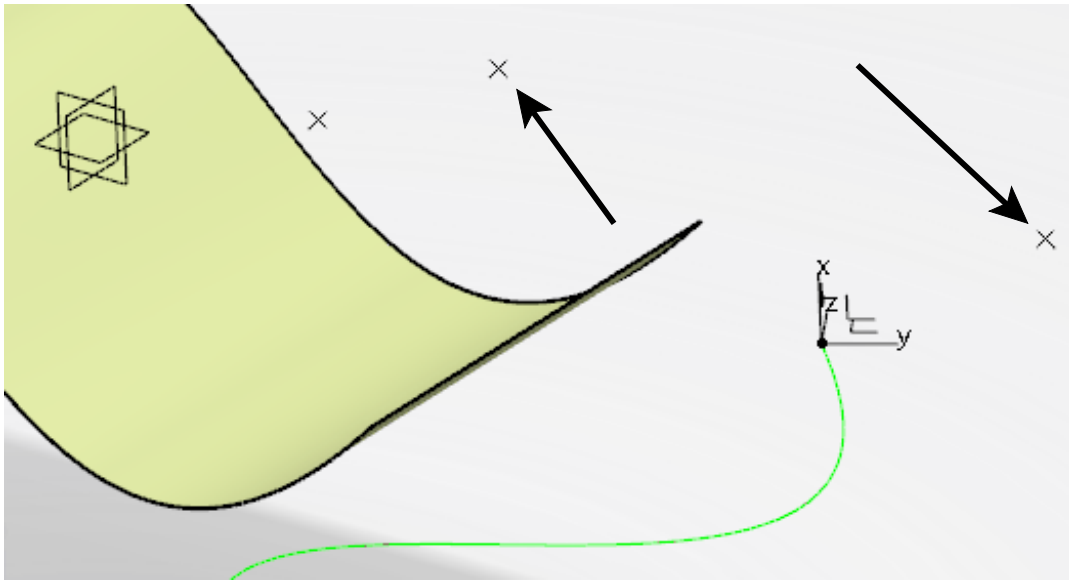
Select the Point icon again. The *Point* window appears. This time, you will use a point as the reference instead of the origin.

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* 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.

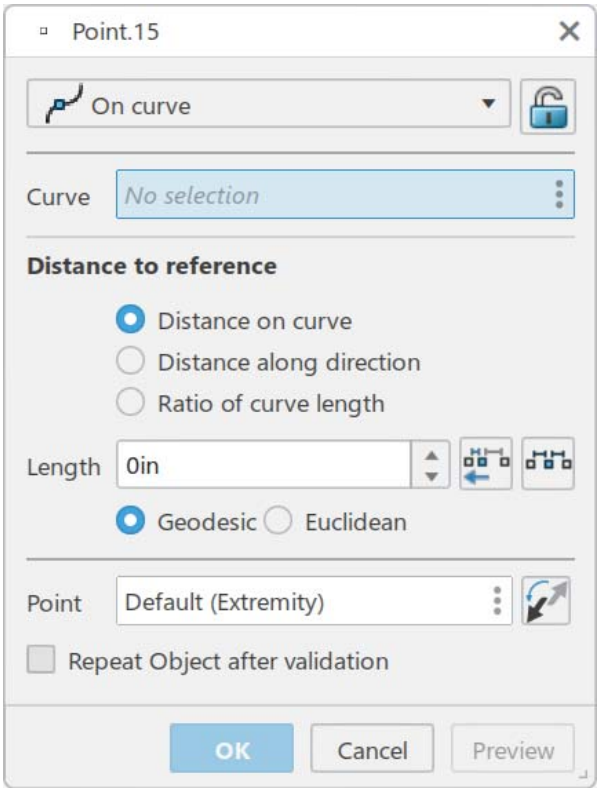


You can also create points on elements such as curves, planes, and surfaces.

On curve

Select the Point icon. The *Point* 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

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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
Point		Allows you to specify a reference point if you do not want to use an extremity for the reference
	Invert	Reverses which side of the reference the point is created on, or which extremity is used
Repeat object after validation		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 *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.

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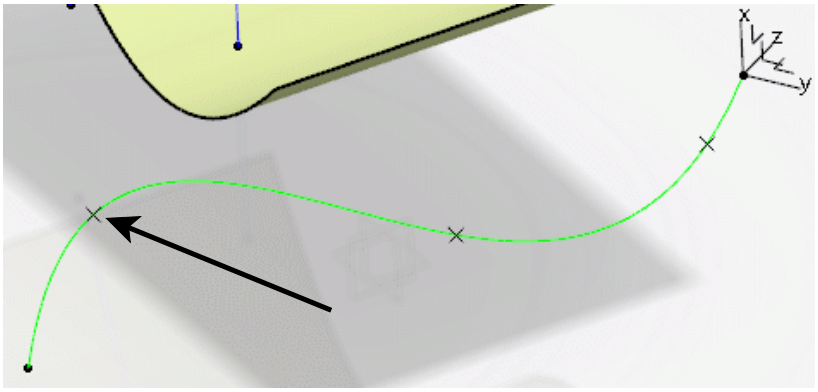
Select *OK*. The point is created and should appear similar to the diagram shown below.



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

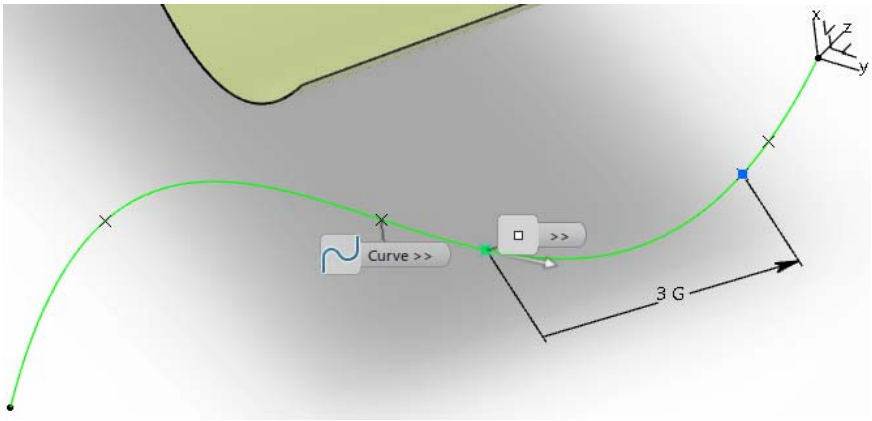
Select the *Distance on curve* option, then select the Nearest Extremity icon. The point appears at the nearest endpoint of the curve.

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



Select the Invert icon. 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.

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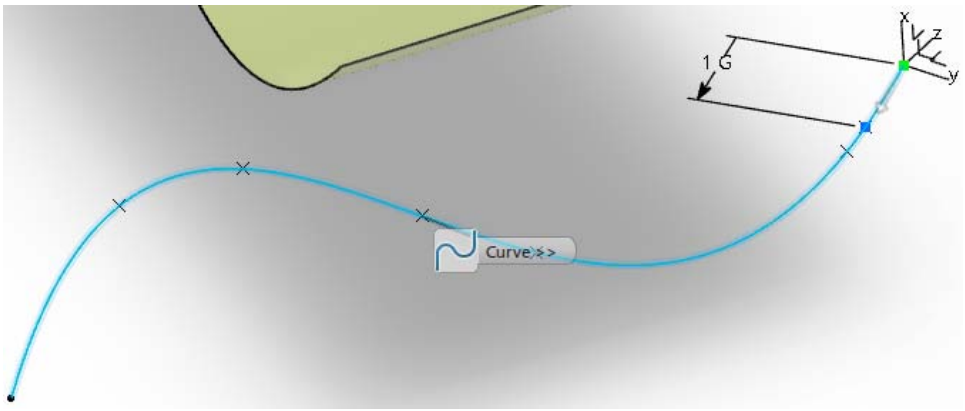
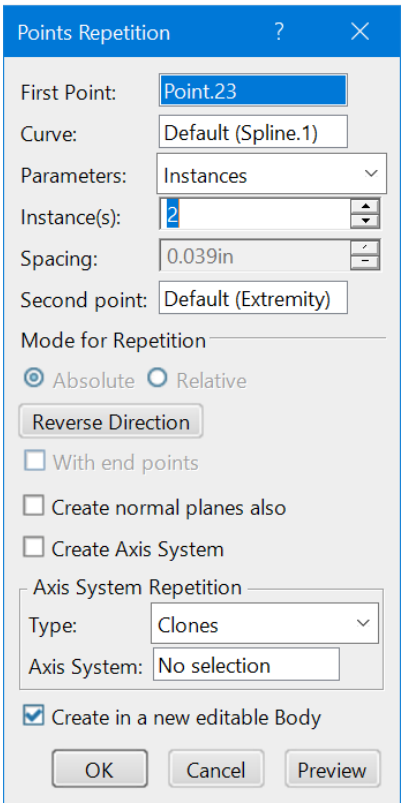
Select the Point icon again and ensure the *Point type* is set to *On curve*, then select the curve on the right.

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

Select the Invert icon. This moves the reference to the other end of the curve.



Turn on the *Repeat object after validation* 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

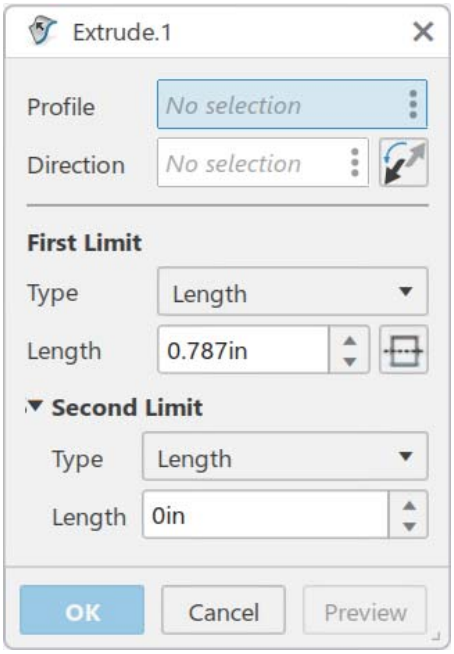
Surfaces are extremely important for defining contours. With the use of wireframe geometry, surfaces can be created to represent any contour needed. Once the surfaces are created, they can then be used in Part Design to make a solid model. There are a variety of options for creating surfaces. Some are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



Profile — Specifies the shape to be extruded

Direction — Defines the direction of the extrusion

Invert — Reverses the direction of the extrusion

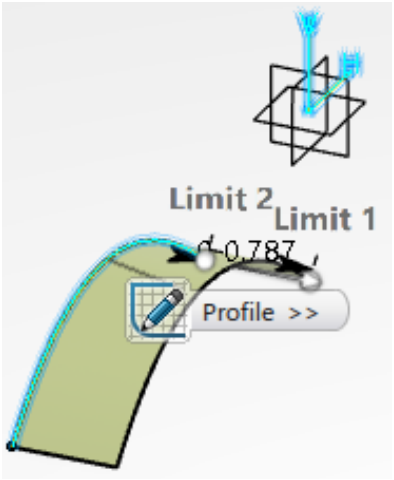
Type — Two options available: *Length* and *Up-to element*

Length — Specifies a distance for the extrusion to extend

Up-to element — Specifies an element that the extrusion will stop at

Length	Defines a distance or a limiting element
Mirror	If this option is selected, the extrude will extend the same distance in each direction; only available when the *Type* is set to *Length*

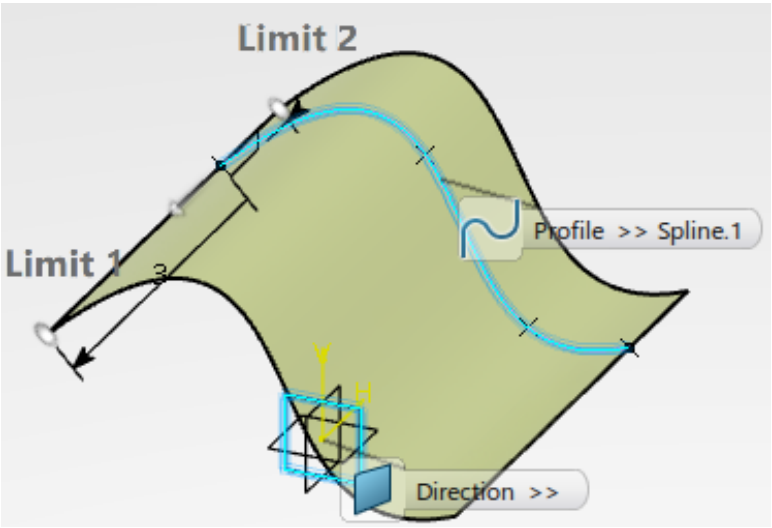
Select the curve shown below. This curve was created in a sketch. Whenever a sketch is selected for an extrusion, the direction will automatically be normal to the sketch plane.



Enter 3.0 for the *First Limit Length* and 1.0 for *Second Limit Length*, then select the Invert icon and click *OK*. The surface is created.

Select the Extrude icon again. The *Extrude* 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 *Length* fields to 1.0 and select *OK*. The surface is created.

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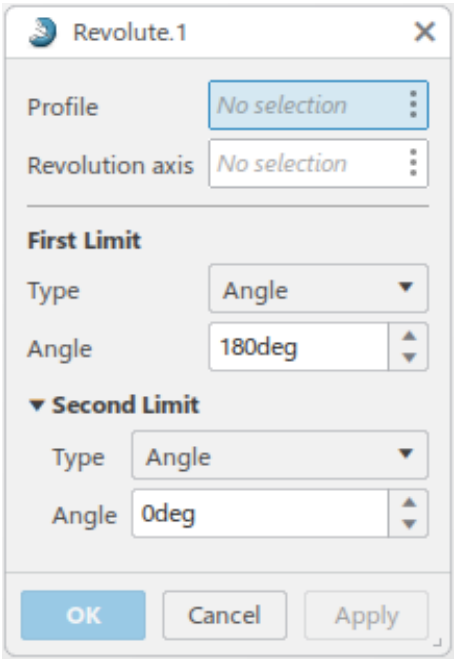
Revolution

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 *Revolute* window appears.



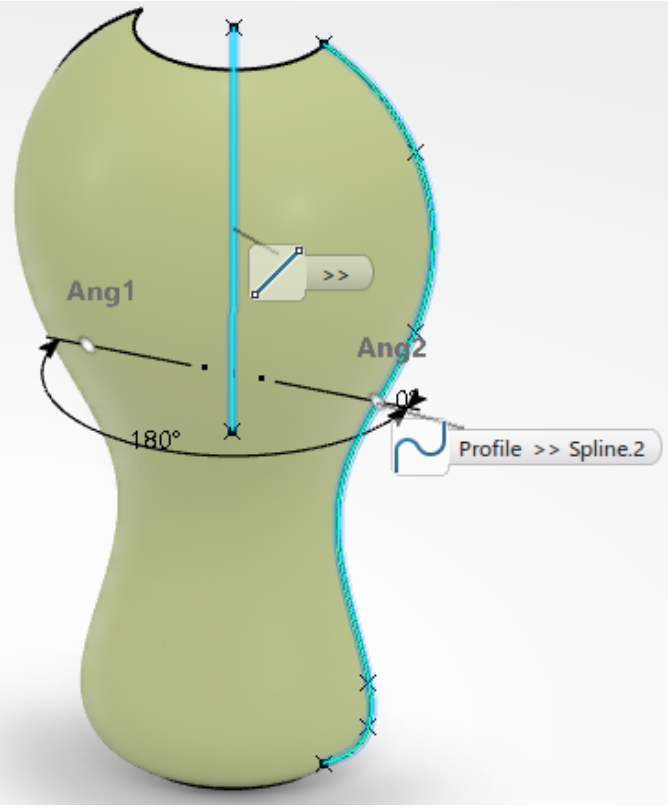
Profile — Specifies the shape that will be revolved

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

Type — Two options available: *Angle* and *Up-to element*

- *Angle* — Specifies an angle of rotation
- *Up-to element* — Specifies an element that the revolute will stop at

Select the profile and line as shown below.



Change *First Limit Angle* to 0.0 and *Second Limit Angle* to 180, then select *OK*. The surface is created.

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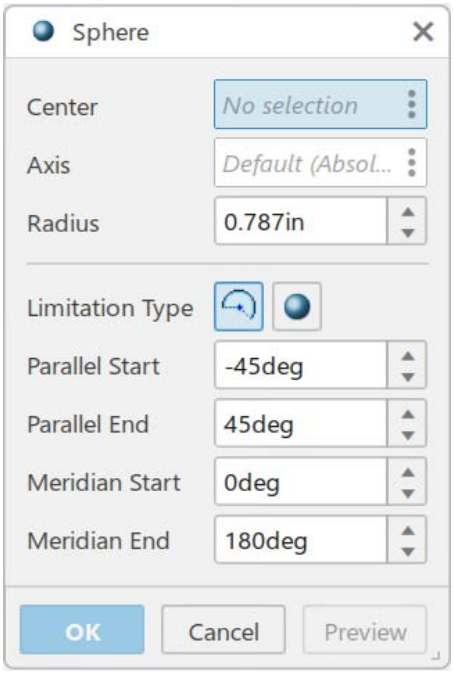
Sphere

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

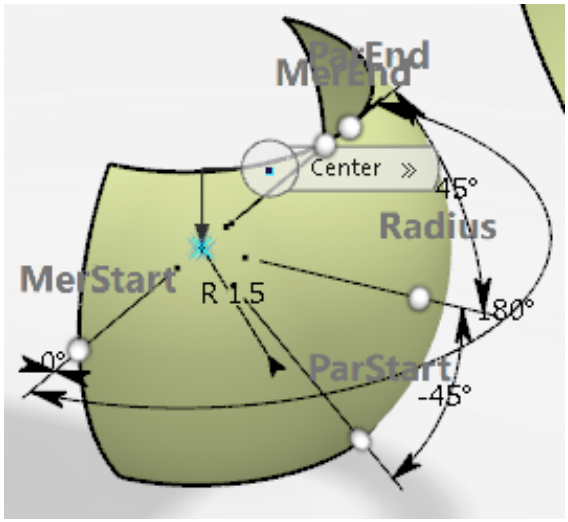
Select the Sphere icon. It is located within the sub-toolbar of the Extrude icon. The *Sphere Surface Definition* window appears.



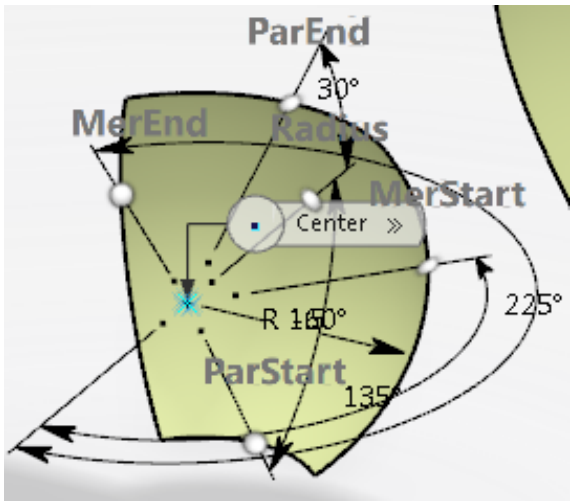
Center	Specifies the center point of the sphere
Axis	Determines the orientation of the *Parallel* and *Meridian* curves
Sphere radius	Defines the radius of the sphere
Limitation Type	
Angles	Creates a partial sphere
Whole Sphere	Creates a full sphere
Parallel Start Angle	Defines the starting angle in 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*.



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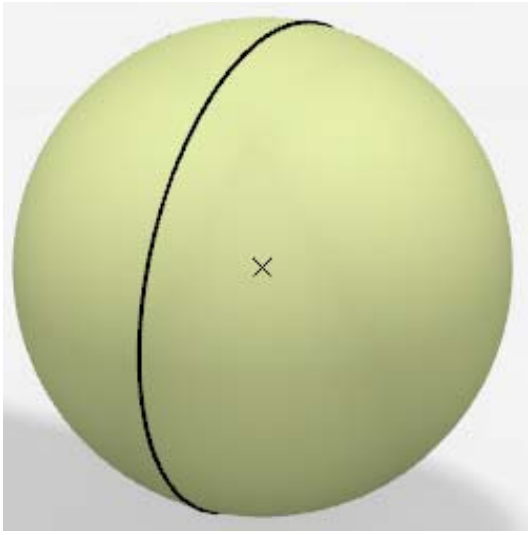
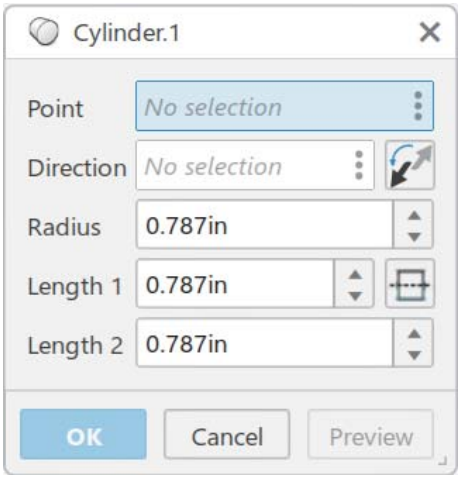
Select the Whole Sphere icon from the window and click *OK.* All the limit options became unavailable.



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* window appears.



Point	Specifies the center point of the cylinder
Direction	Specifies the direction the cylinder will extrude
Invert	Reverses the direction of the extrusion
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
Mirror	Changes *Length 2* to be the same as *Length 1*

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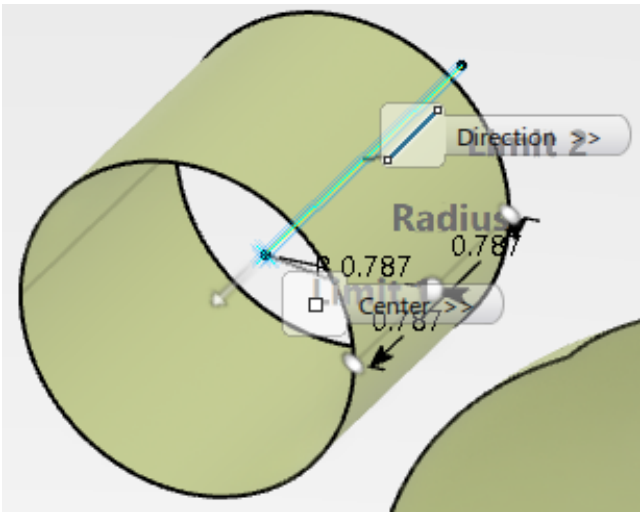
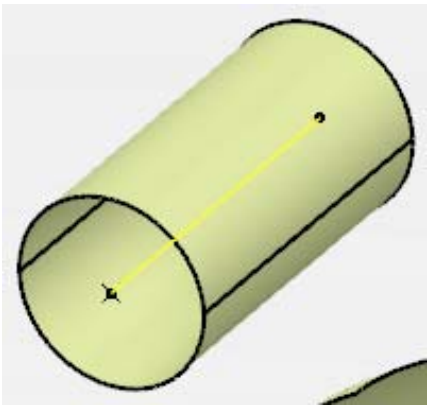
Select the point and the line as shown below.



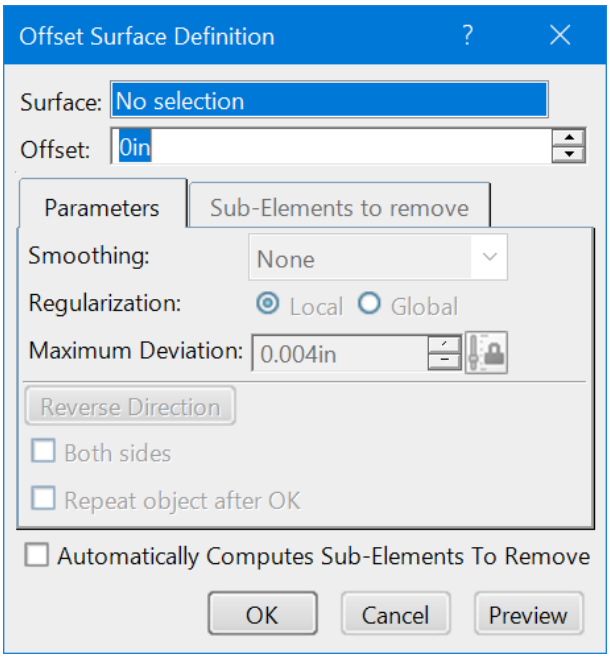
Change the *Radius* to 0.5, *Length 1* to 2.0, and *Length 2* to 0.0, then select the Invert icon and click *OK*. A cylindrical surface is created.



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.

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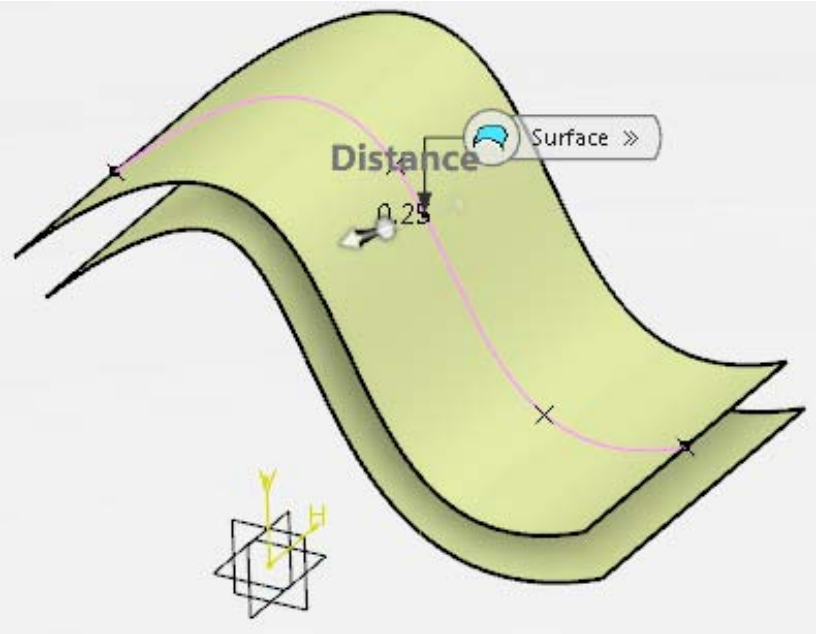
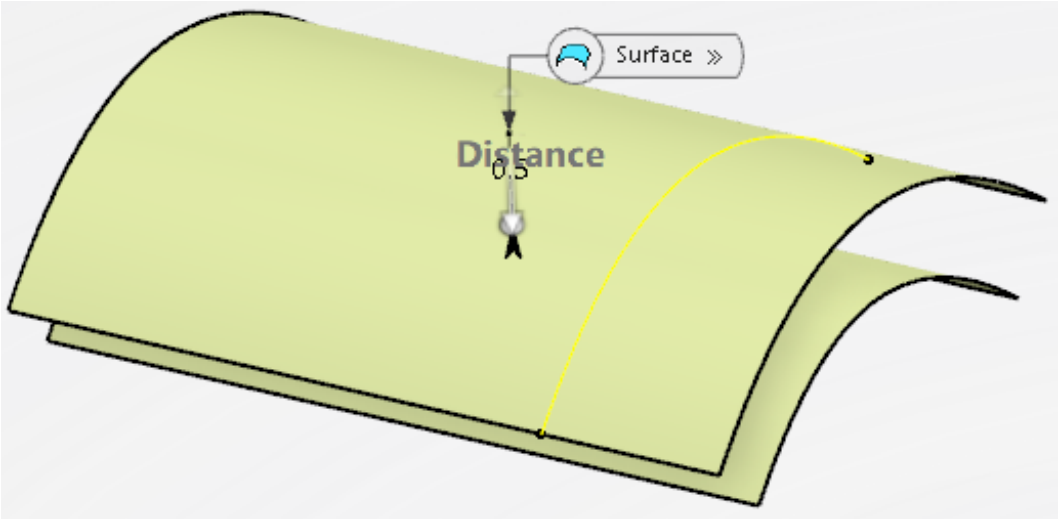
Select the surface as shown below, then enter 0.25 for the *Offset* and select *Preview*. An offset surface appears.



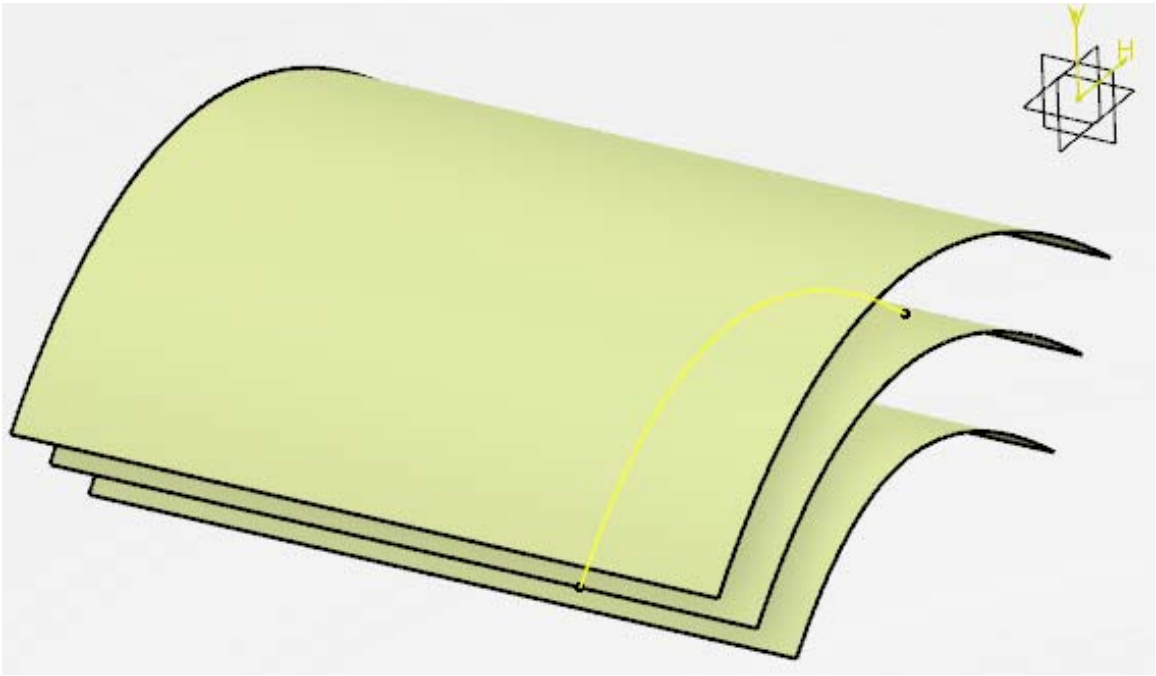
Select the *Reverse Direction* button and click *OK*. The offset surface appears above the original surface instead of below it.

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.

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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.

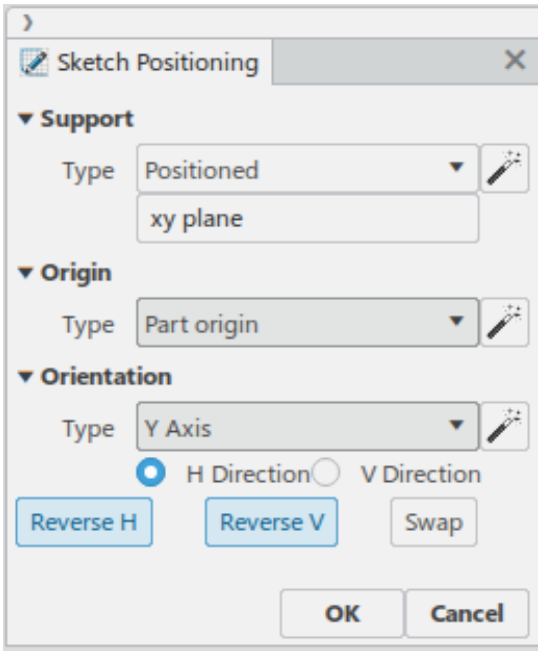
Note: Set your view mode to Shading With Sharp Edge in order to obtain the same results shown in the following images.

Mouse Body

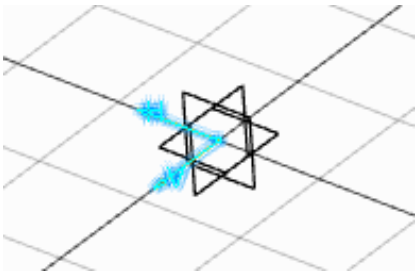
You will first create the mouse body, followed by the buttons and wheel.

Create a new 3D part.

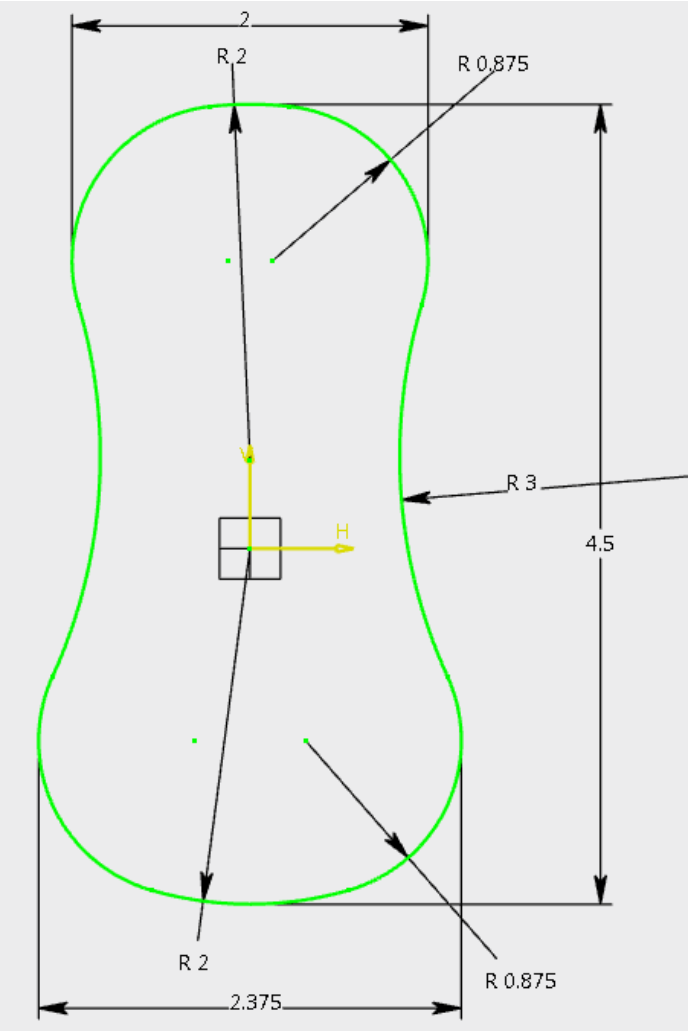
Insert a geometrical set named Mouse Body, then select the Positioned Sketch icon and set the options as shown below.



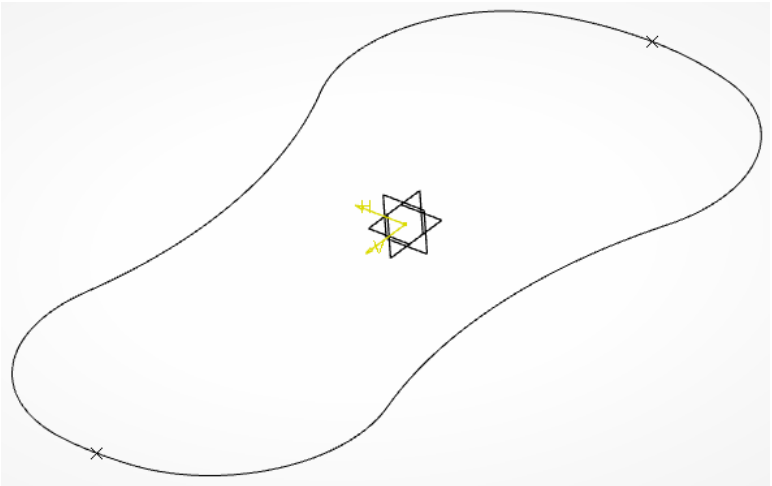
The sketch axis should be oriented as shown.



Create the following sketch. All curves are tangent continuous. The geometrical constraints have been hidden for clarity. Keep in mind you may have to rotate the sketch around so the Horizontal and Vertical axes are oriented correctly.



Create an extremum point at each end of the sketch.



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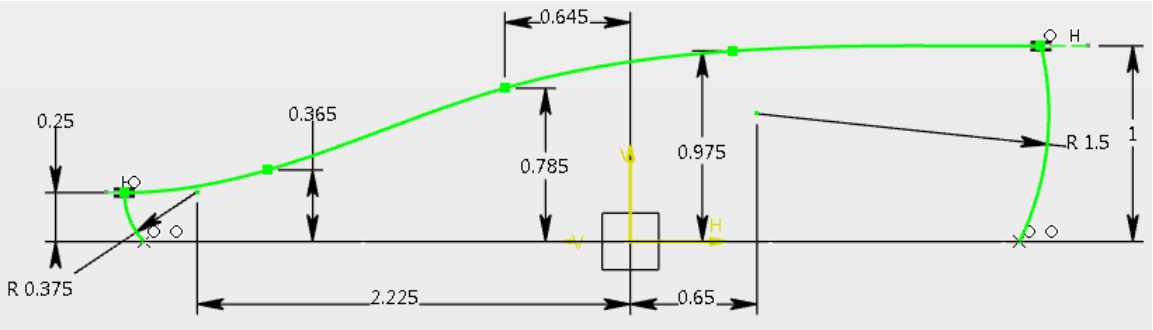
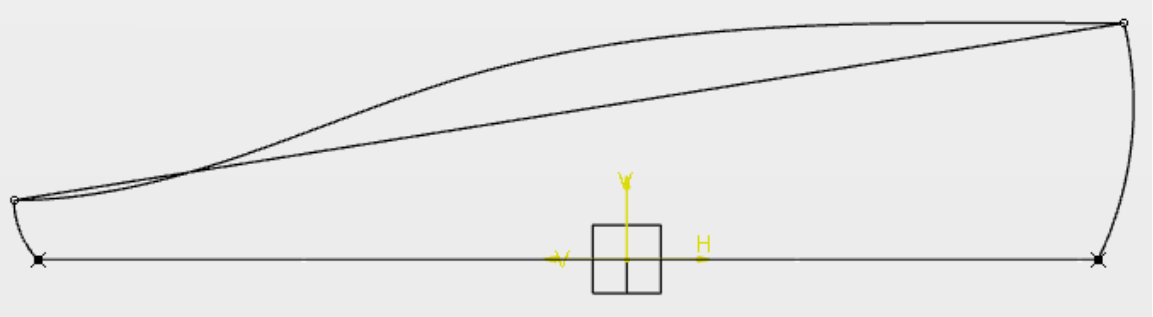
Create the following sketch on the zx plane. The bottom of the arcs are coincident to the extremum points. Be sure the taller end of this sketch is towards the wider end of the first sketch.

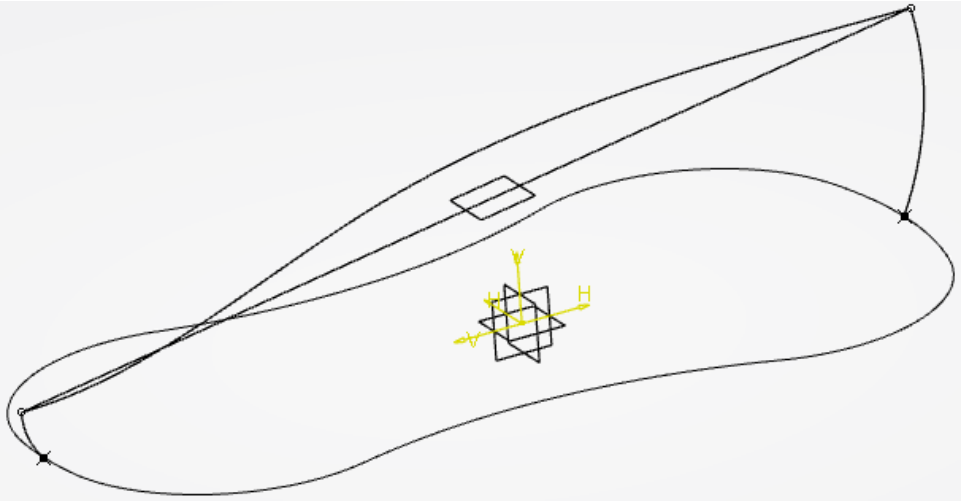


Extract each curve from the sketch.

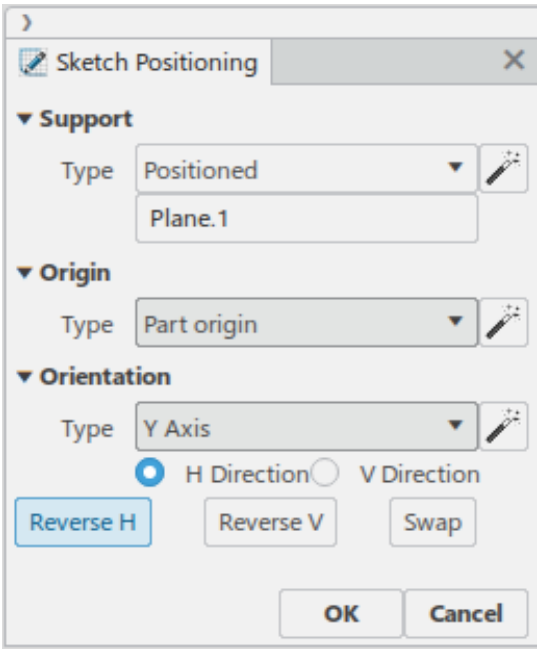
Create a *Point-Point* line between the top points of the arcs.



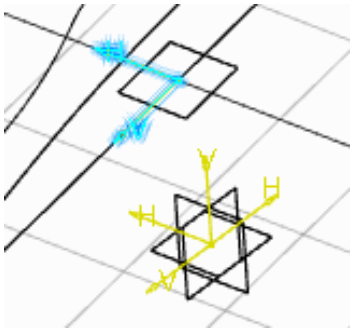
Create a plane through the line. Use the line for the *Rotation axis* and the zx plane for the *Reference*. The plane should be normal to the reference.



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



The sketch axis should be oriented as shown.



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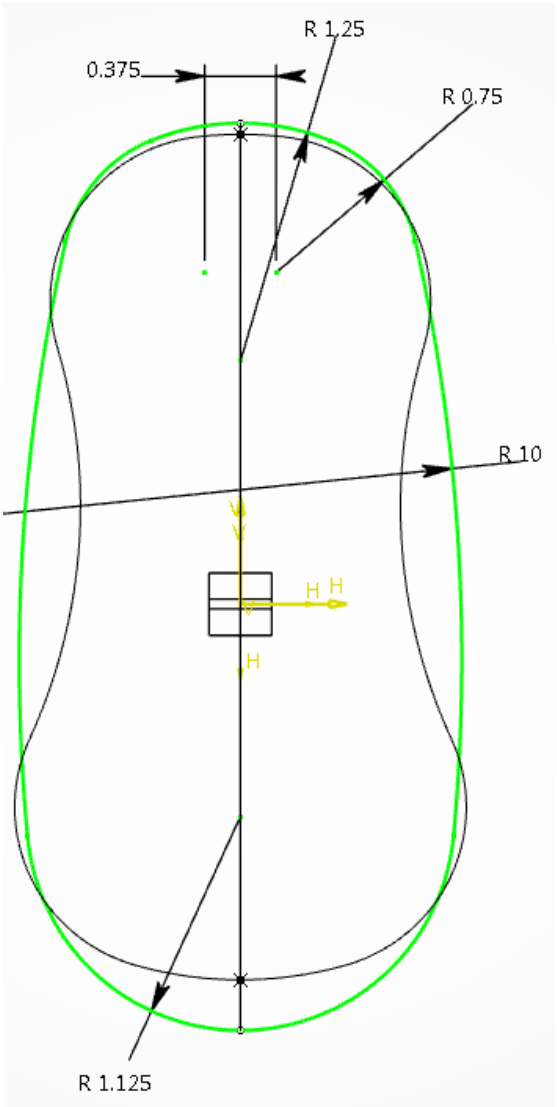
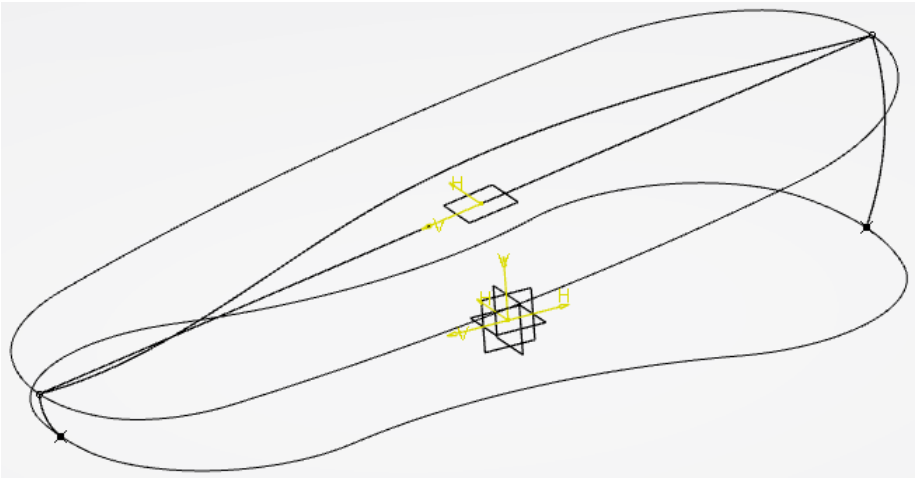
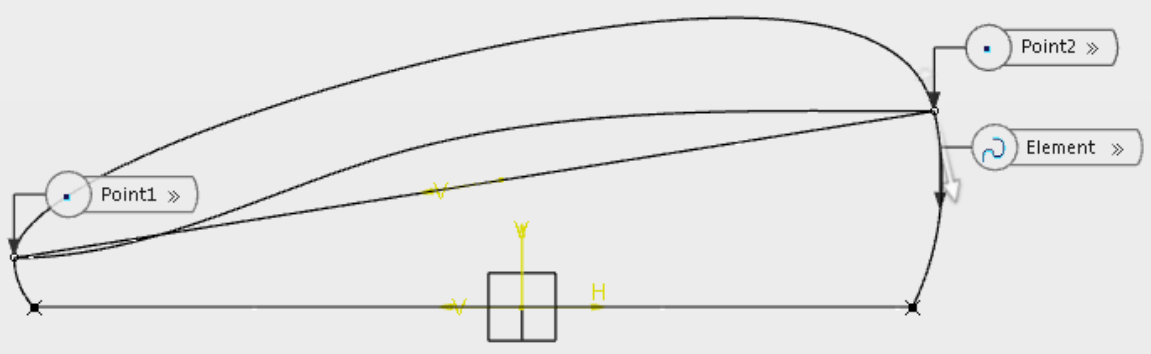
Create the following sketch. The top and bottom arcs in this sketch are coincident to the upper end points of the extracted arcs. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.



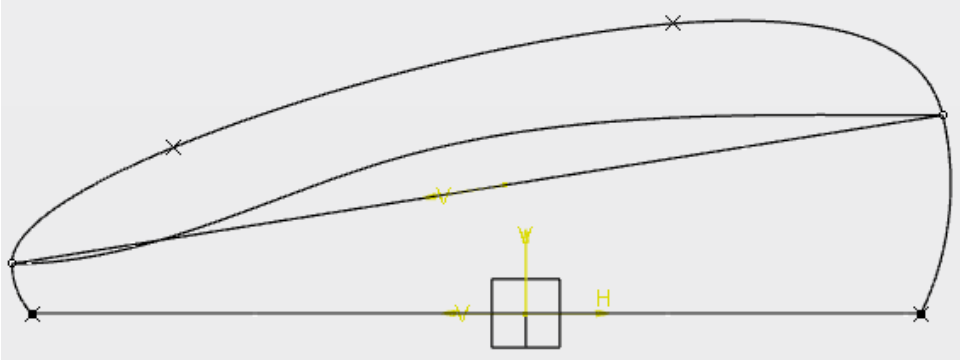
Your model should look like this.



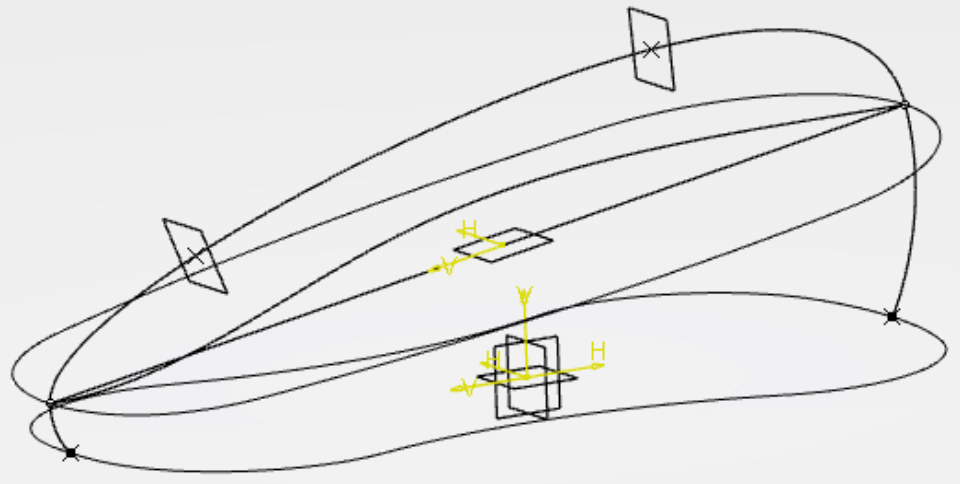
Create a spline between the two points at the top of each extracted arc. The spline will be tangent continuous to both arcs with a tension of 0.375 at the first point, and 0.75 at the second point.



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.



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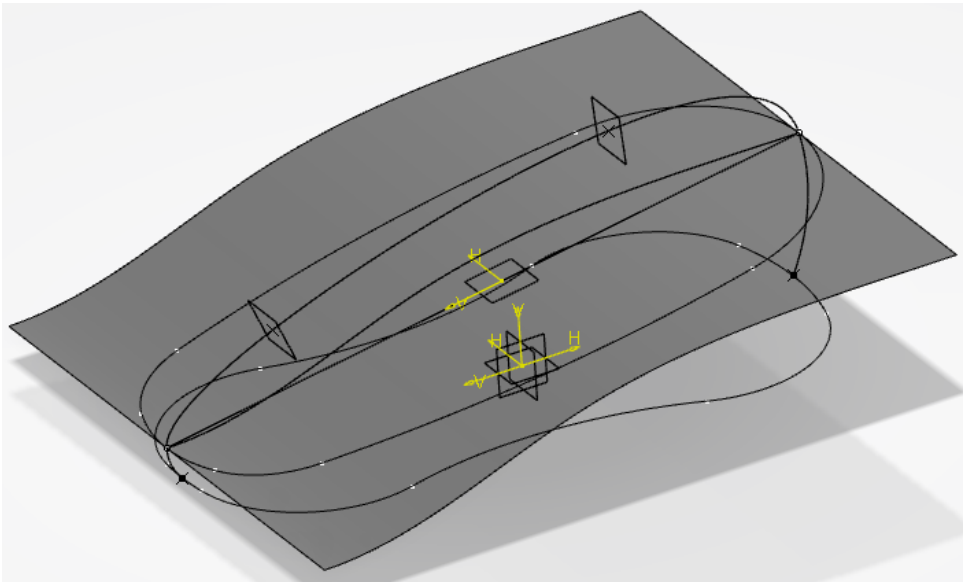
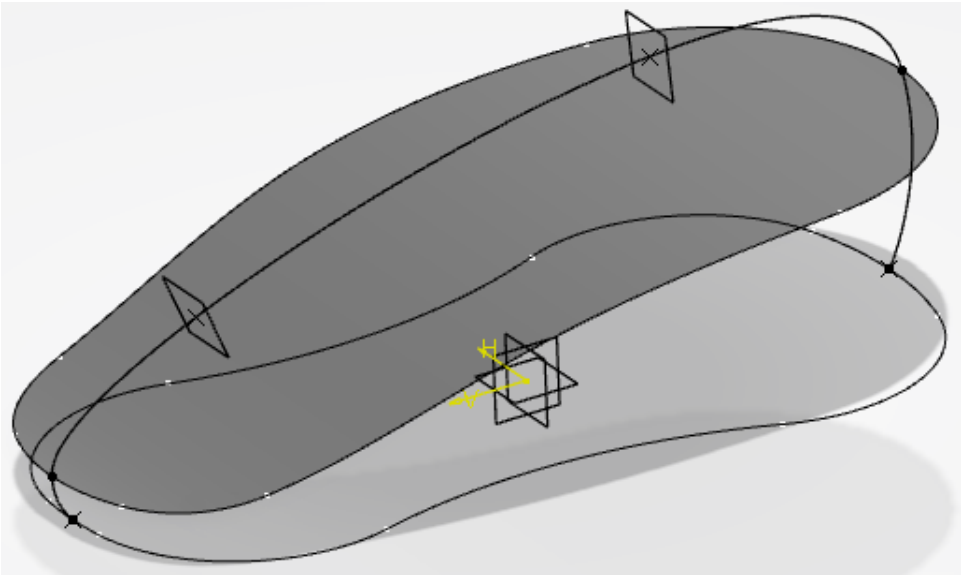
Extrude the extracted spline two inches in both directions normal to the zx plane, then change the name of the extrude to CHANNEL SURFACE in the specification tree.

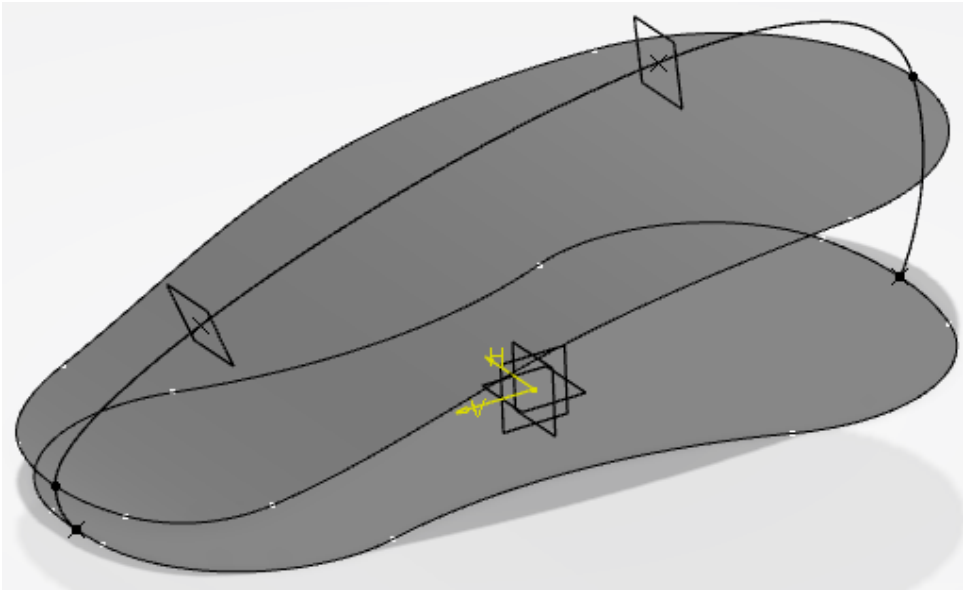


Project the last sketch you created to *CHANNEL SURFACE* along the normal direction of the first plane created.

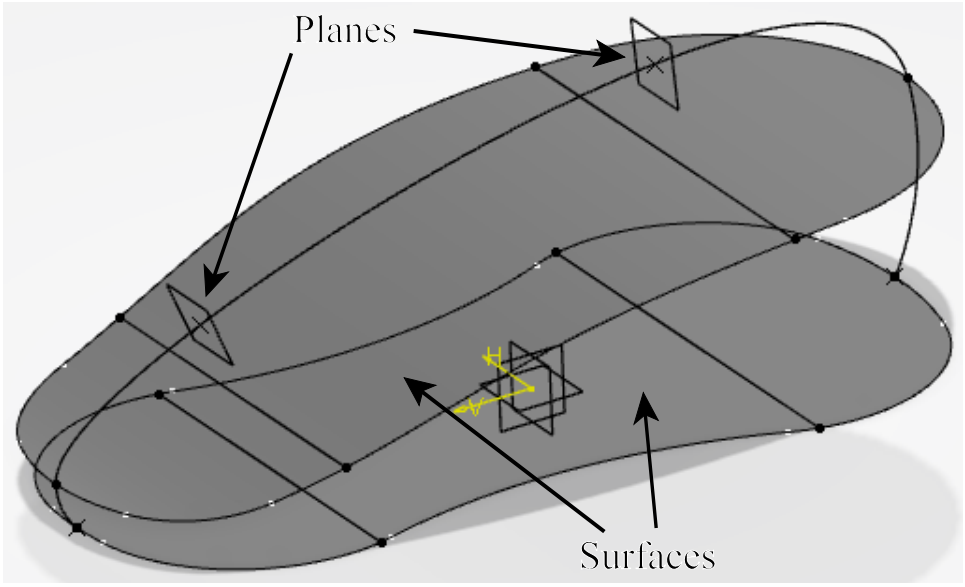
Split *CHANNEL SURFACE* with the projected curve, keeping the inner portion.
The split surface is shown below. Much of the geometry has been hidden. Feel free to hide your geometry as necessary to reduce clutter.



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.



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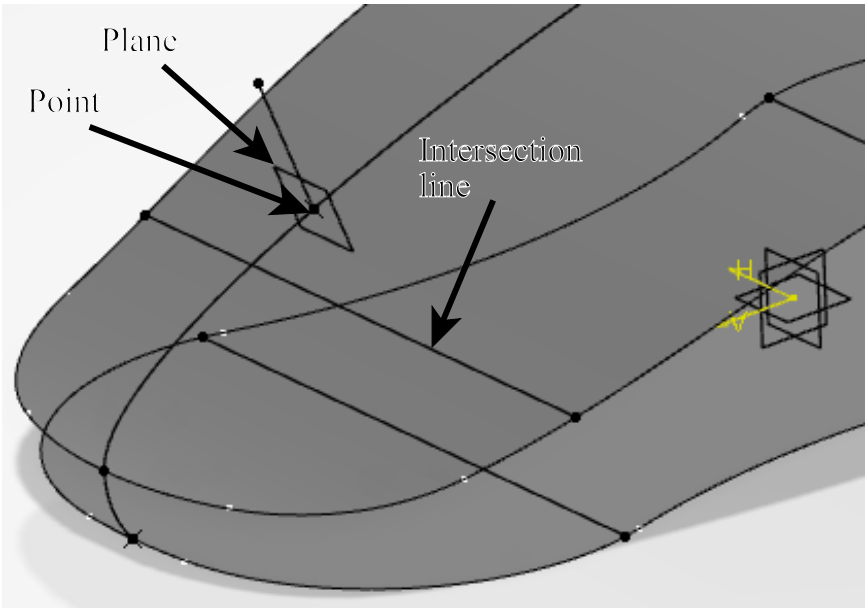
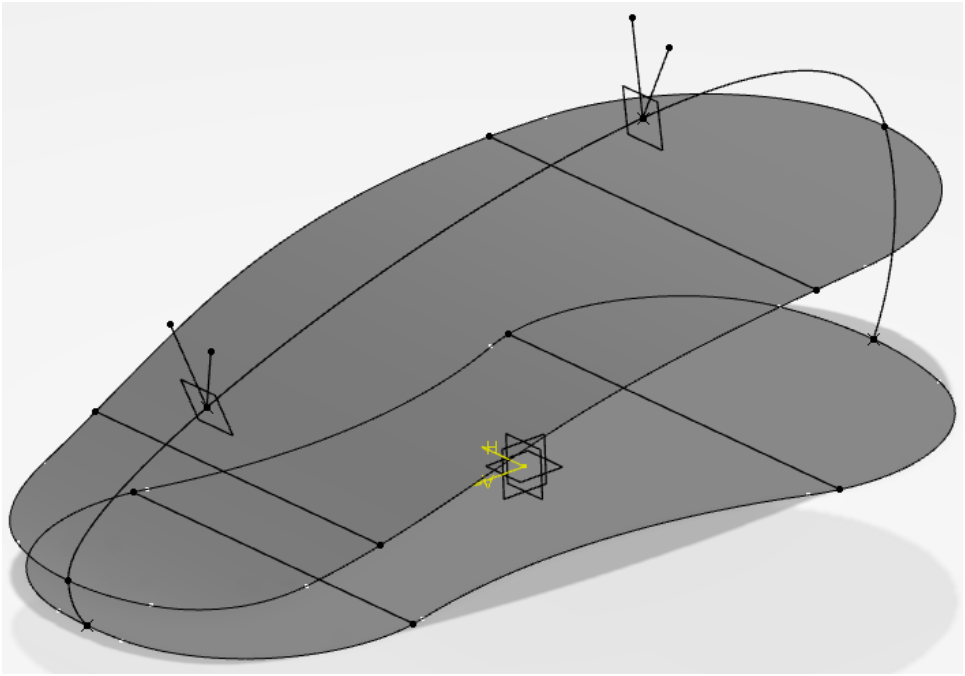
Create a 0.5 inch line normal to the intersection line indicated below that starts at the point and uses the plane as its support.

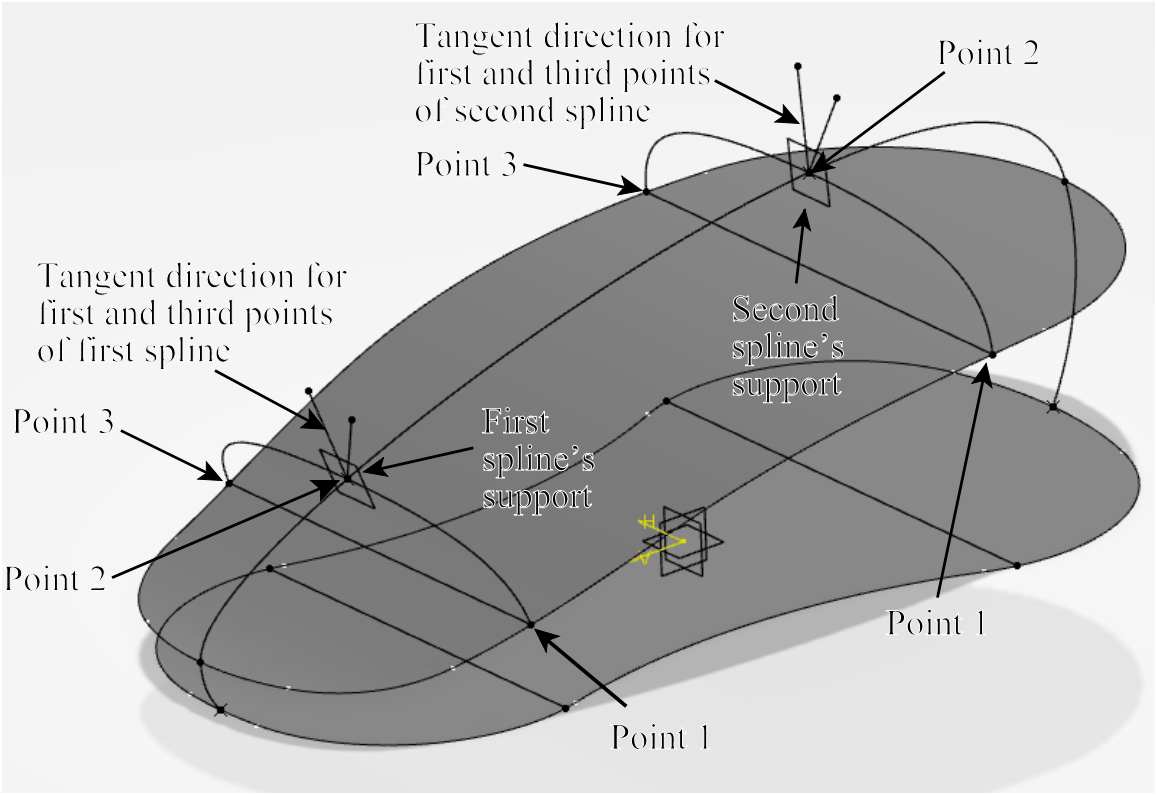


Create another line using the same geometry, but this time specify an *Angle* of -60. Ensure the line extends upward.

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.



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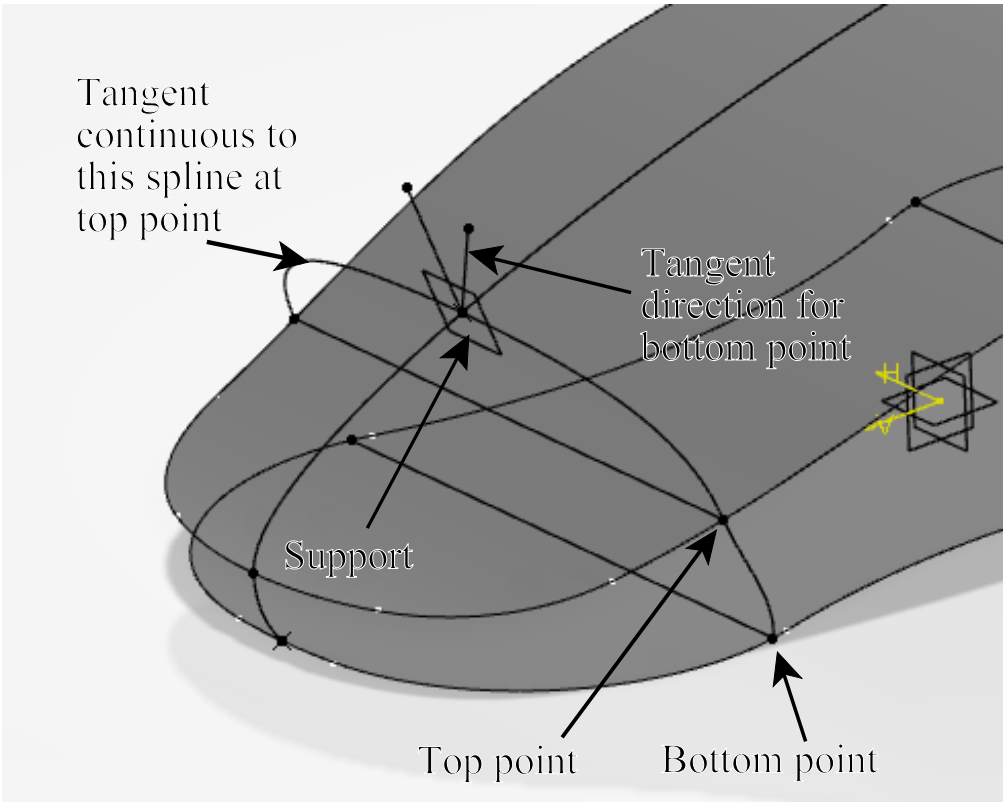
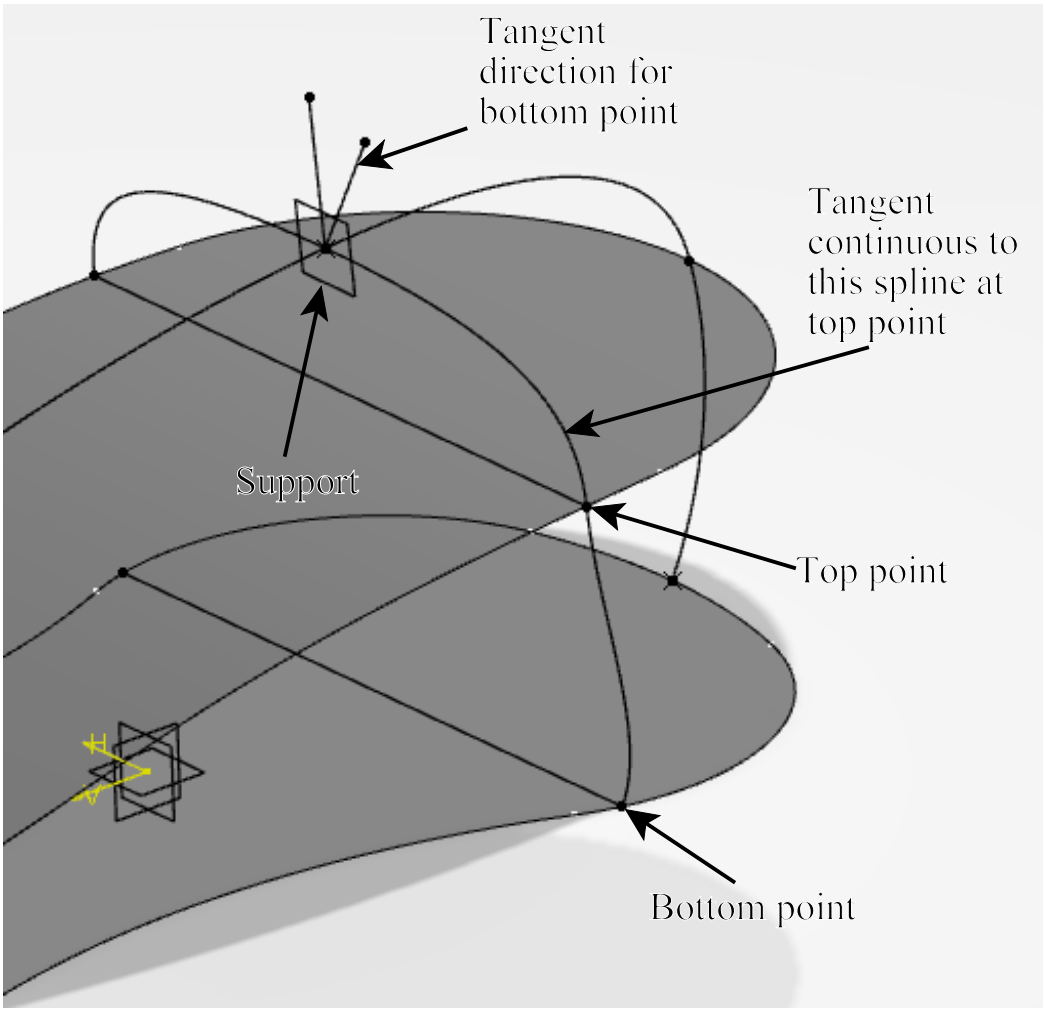
Next, create the spline shown below. This spline is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.5 at the bottom point. Ensure the spline lies on the support plane indicated below.



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.



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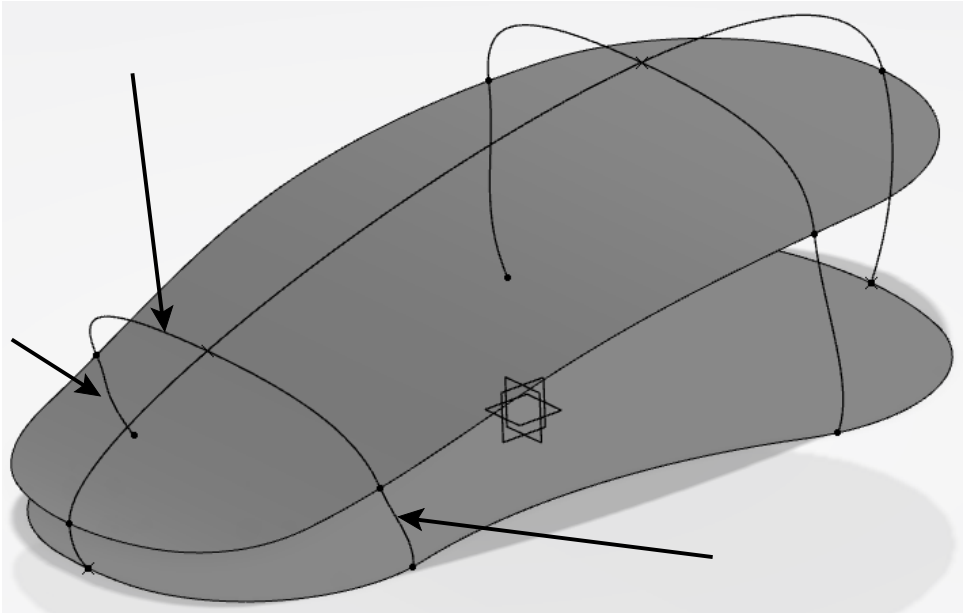
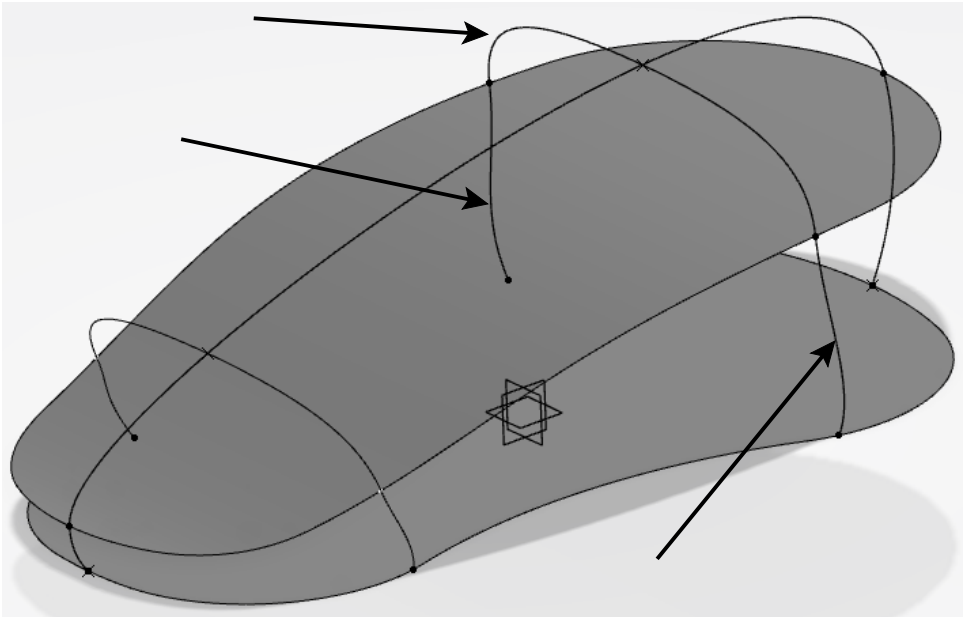
Mirror each of the last two splines created across the zx plane.

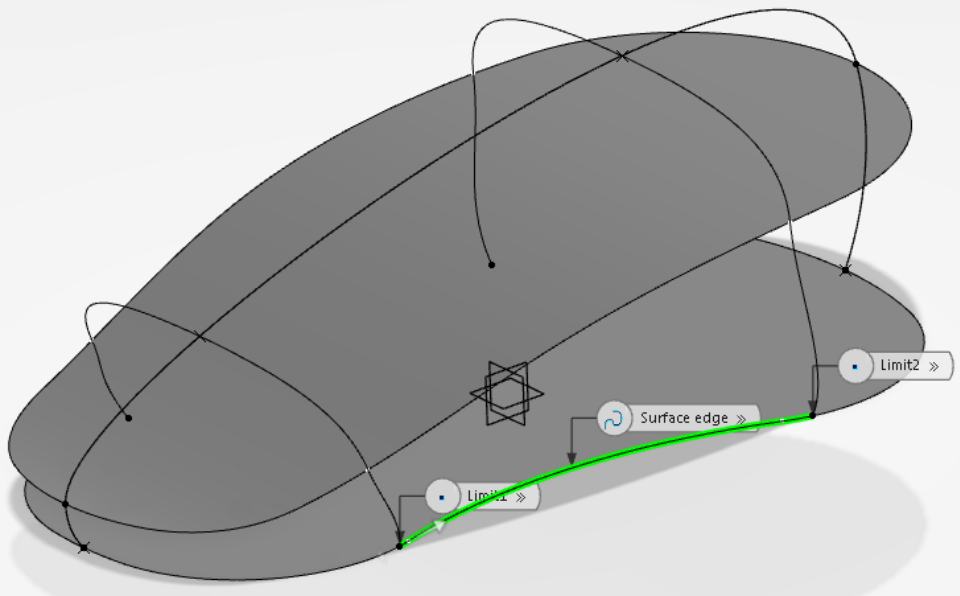
Join the three curves indicated below. Ensure they are tangent continuous.



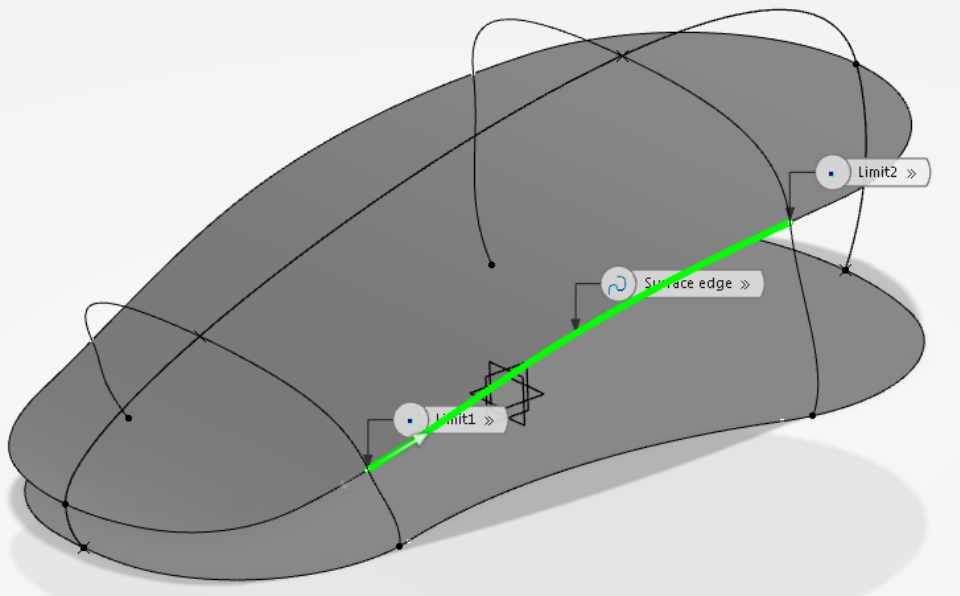
Create another join for the three curves shown below. Ensure they are tangent continuous.



Create a boundary curve on the following edge.



Create another boundary curve on the edge shown below.



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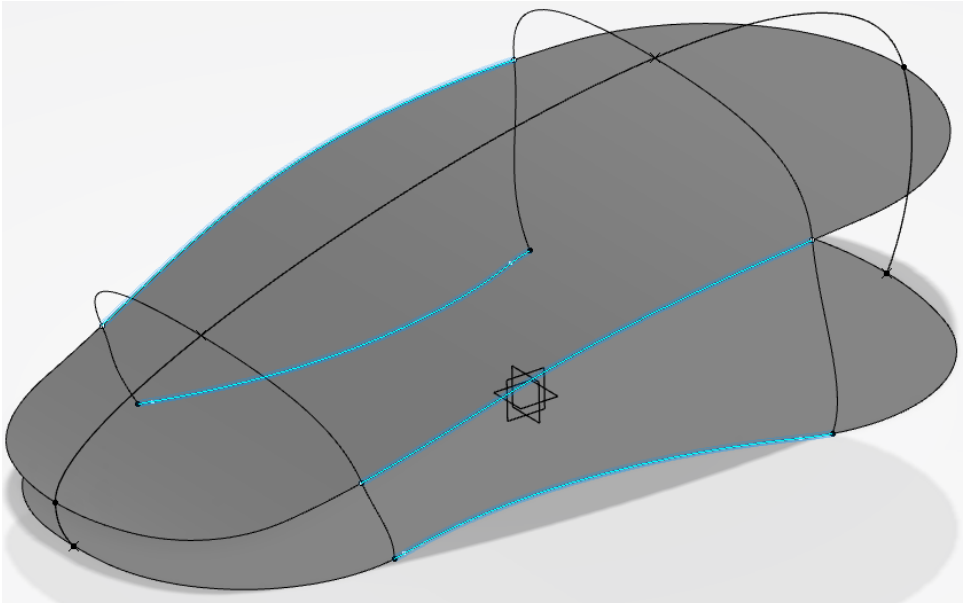
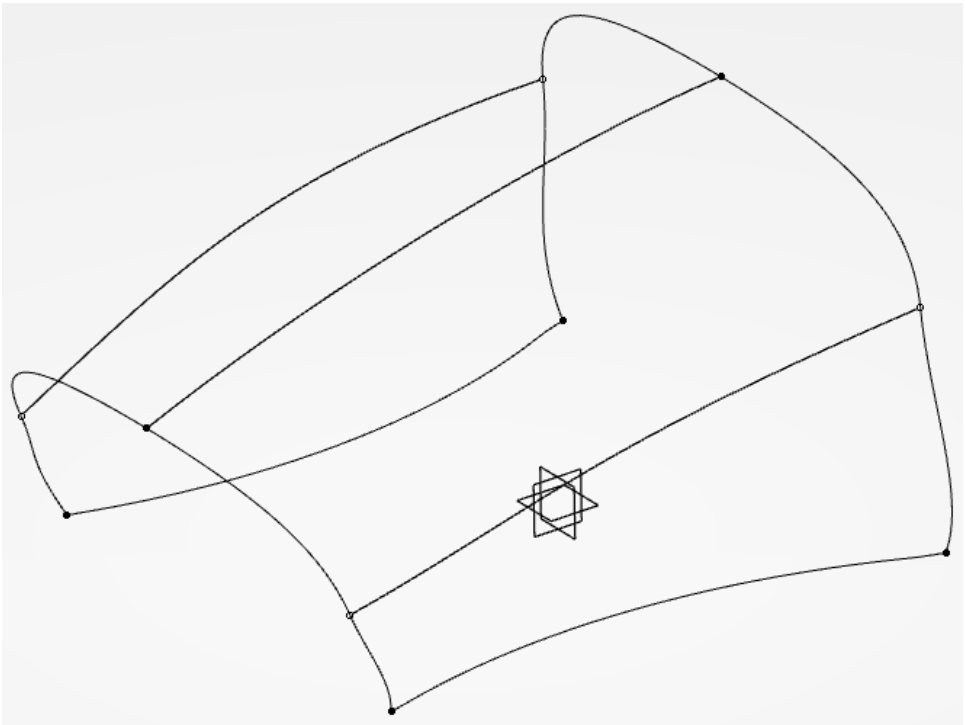
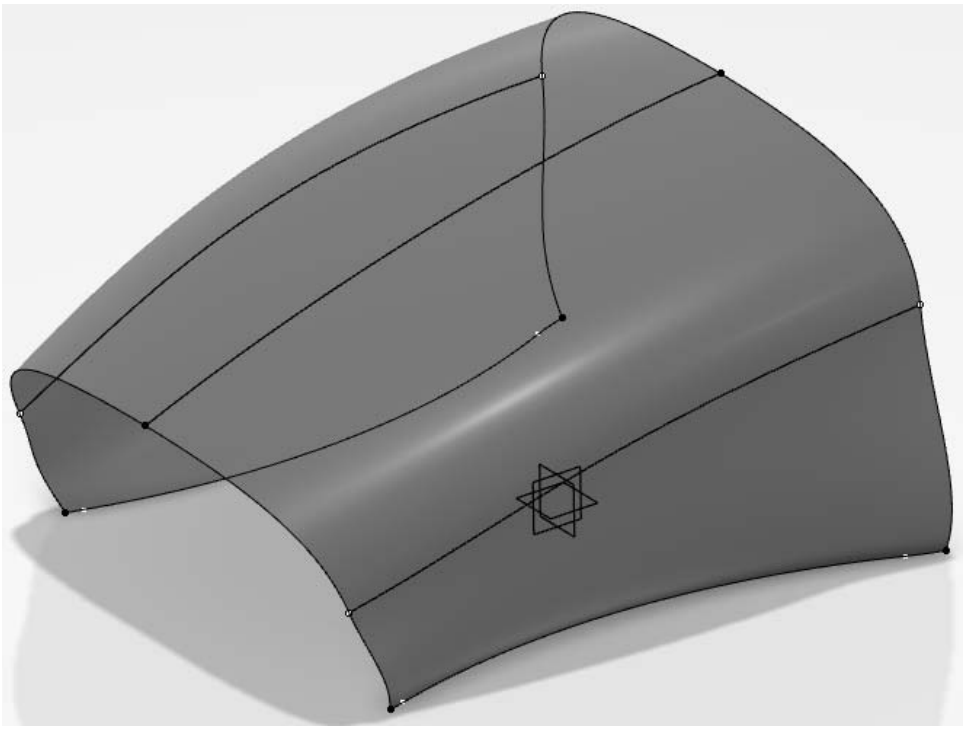
Create the same two boundaries on the opposite side of the surfaces. Only the boundary curves, the joined curves, the split surface, and the filled surface are shown below.



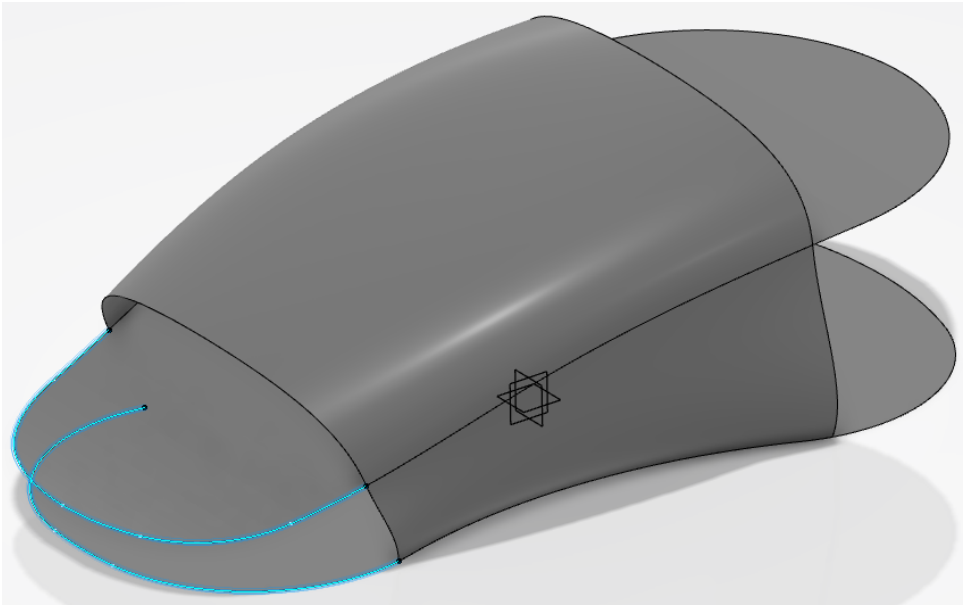
Split the upper spline at its normal planes, then hide all geometry except the boundary curves, the joined curves, and the new split.



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.



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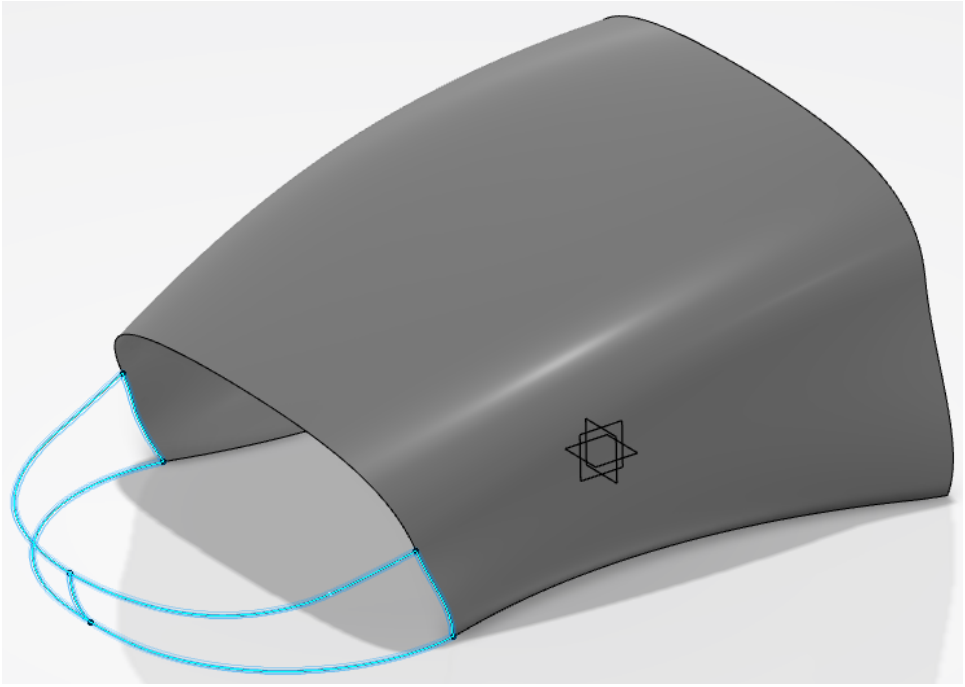
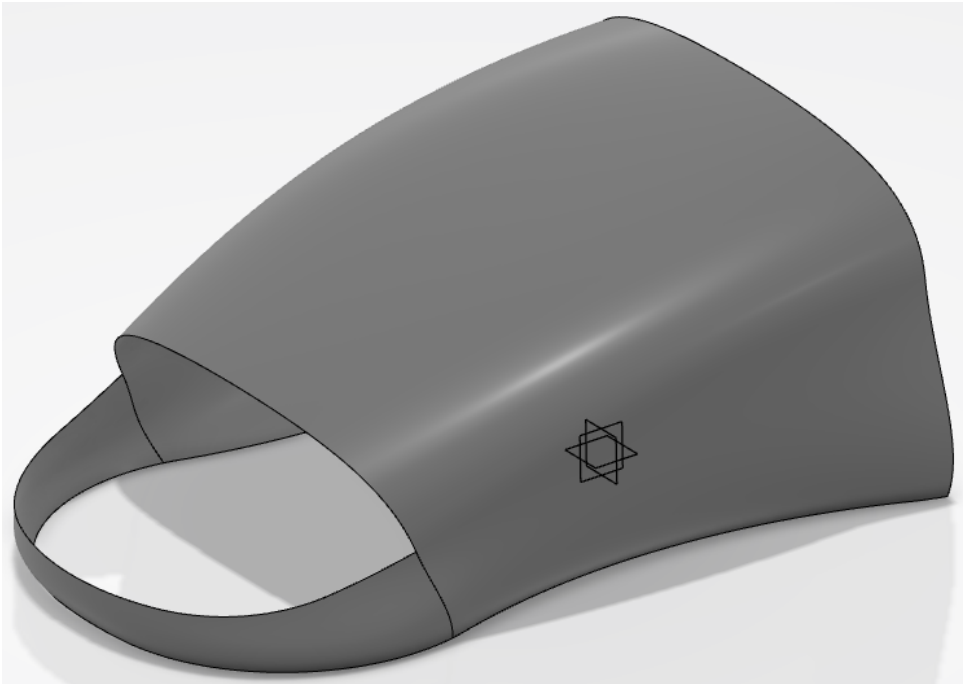
Create a multi-section surface using the geometry shown below. The splines and the extracted arc are the sections, and the boundaries are the guides. Ensure the first and last sections are tangent continues to the surface shown below.



Your model should look like this.