Table

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

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

Points

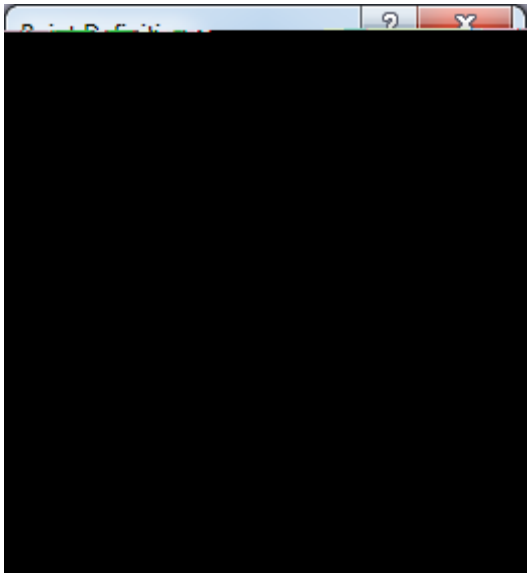
Points are useful to define specific locations and to assist in the creation of other wireframe geometry. You have a variety of options to define points. This exercise will explore those options.

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

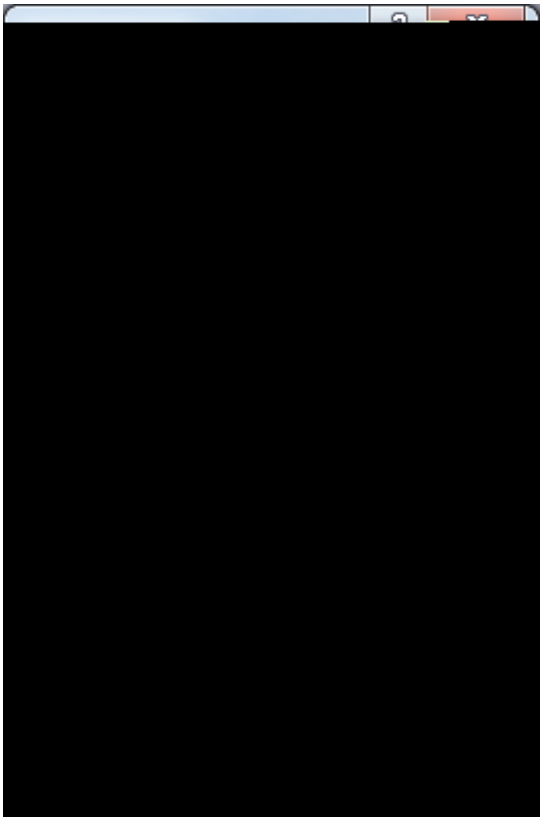
Select the Point icon again.  The *Point Definition* window appears. This time you are going to use a point as the reference instead of the origin.

Clear the *Axis System* selection box so that the abso

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

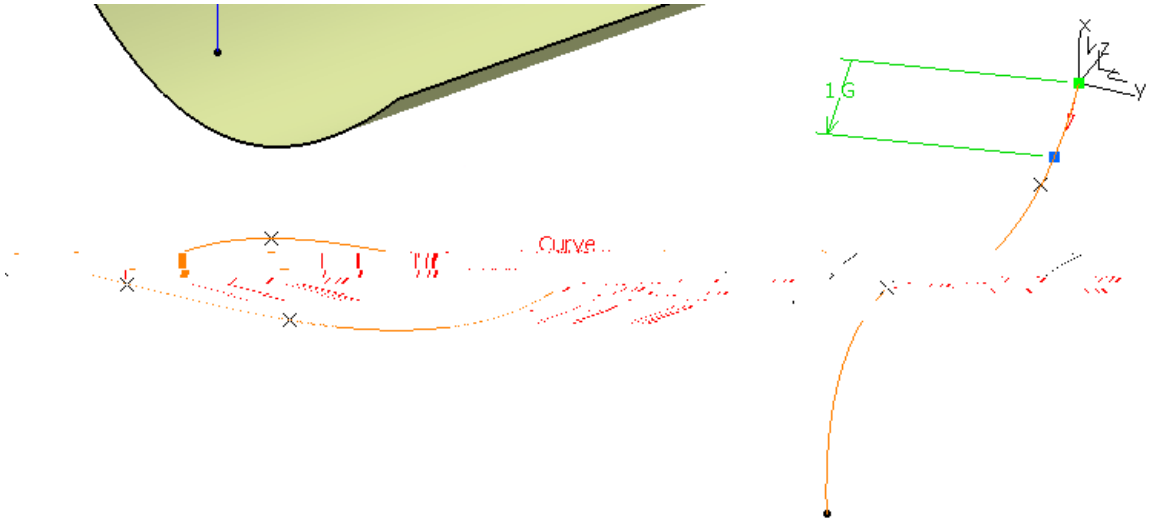
Select the *Reverse Direction* button. The arrow points the opposite direction. If you were using an extremity then reversing the direction would cause the reference point to switch to the other end of the spline.

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*.

Surfaces

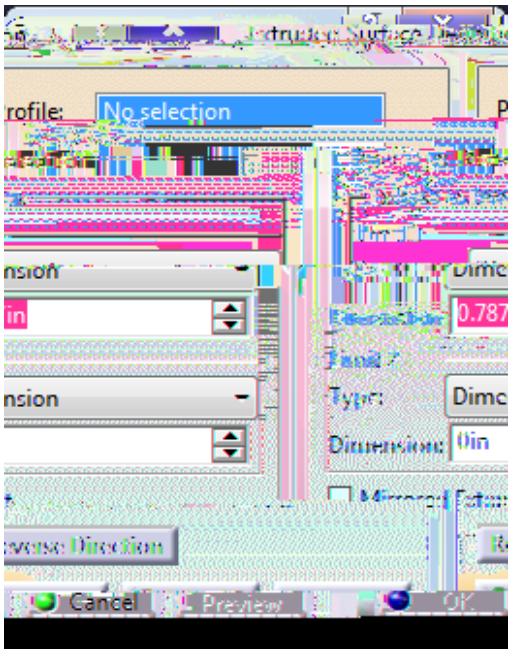
Surfaces are extremely important for defining any type of contour. Using wireframe geometry, you can create surfaces to represent any contour that you need. Once you have created the surface(s) that you need you can then use them in Part Design to contour your solid model. You have a variety of options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

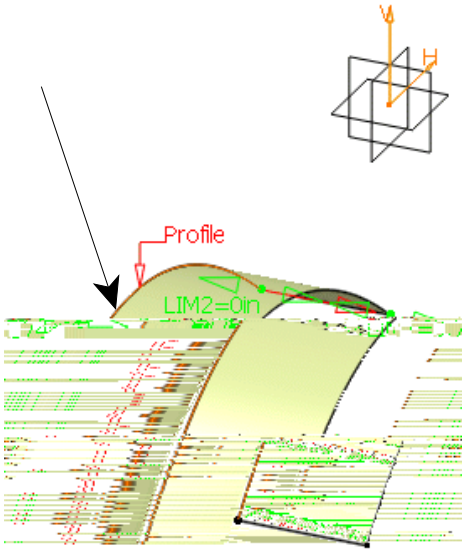
Type Specifies either a constant dimension or up to a selection

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

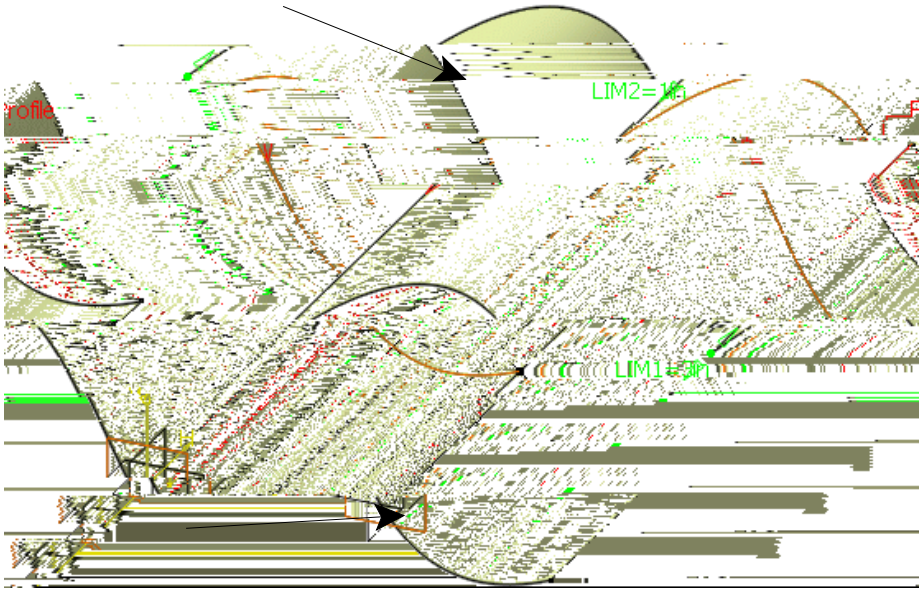
Select the curve as shown below. Since this curve was created in a sketch, the extrude option automatically assumes you want to go normal to the sketch.



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 again.  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.

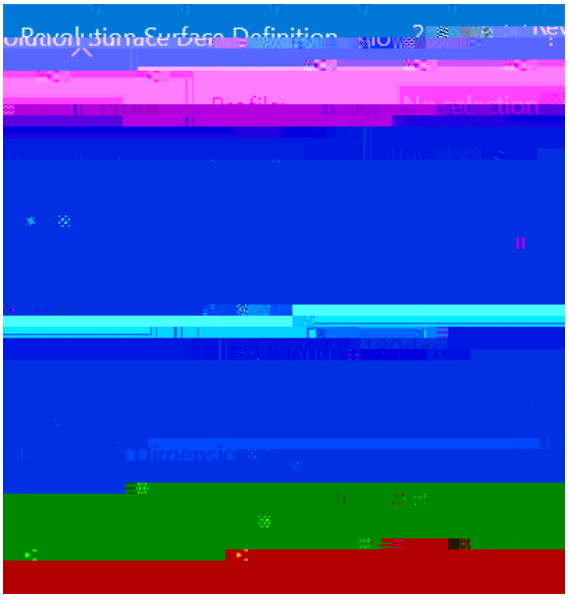


Change the limits so that both are 1.0 inch and select *OK*. The surface is created.

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*

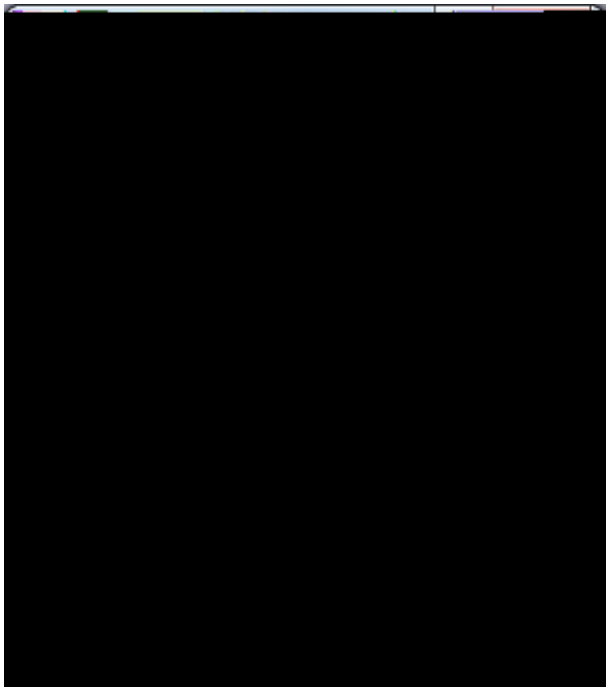
Limit 2 Specifies either a *Dimension* and ending angle for the revolution or an *Up-to element*

Select the profile and

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.



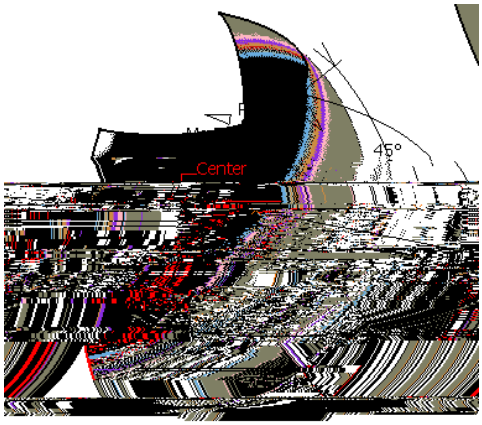
Center	Specifies the center point of the sphere
Sphere axis	Determines the orientation of the *Parallel* and *Meridian* curves
Sphere radius	Defines the radius of the sphere

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

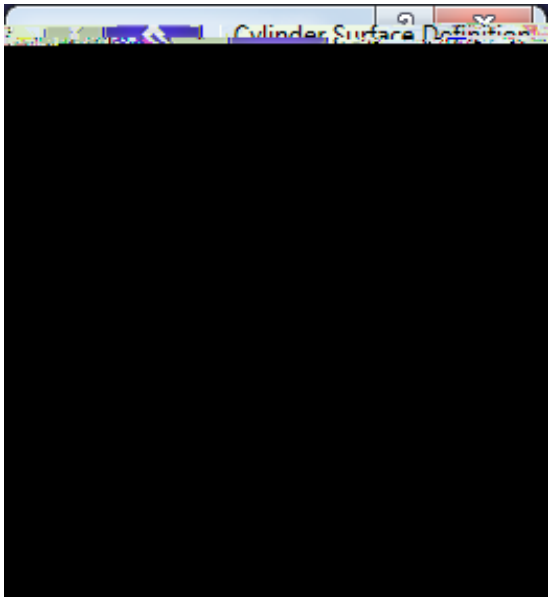
Select the point as shown below, key 1.5 for the *Sphere radius* and select *Preview*. Since you do not have any other axis to select, you will use the default. The *Parallel* limits have a range of -90 to 90 while the *Meridian* limits have a range of -360 to 360. Basically, the *Parallel* limits are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



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.

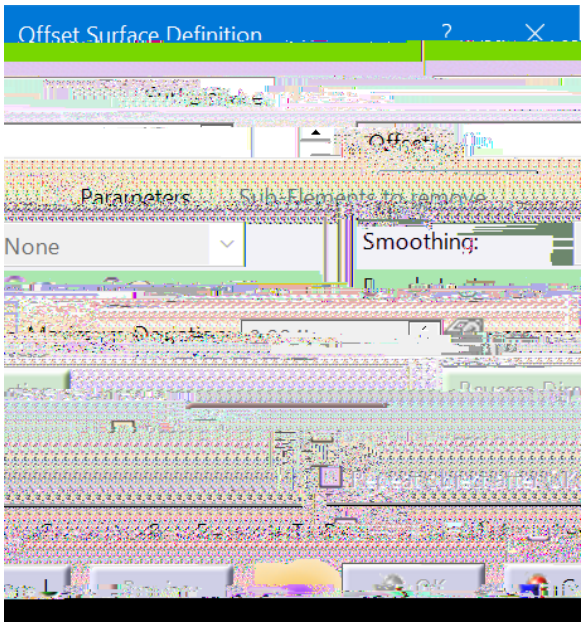


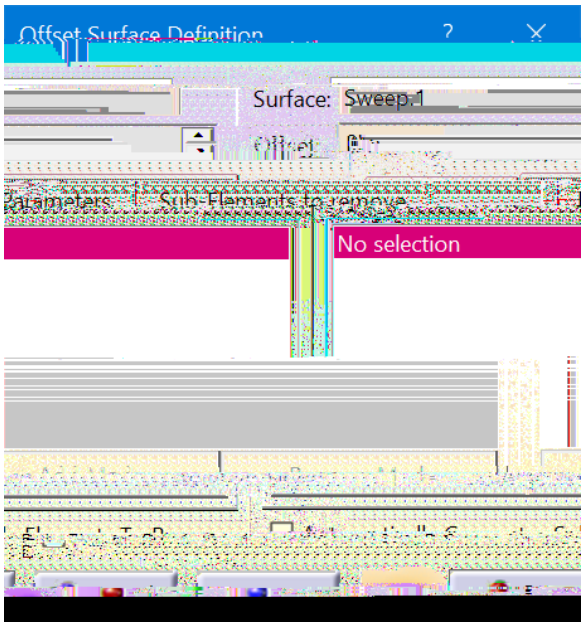
Point	Specifies the center point of the cylinder
Direction	Specifies the direction the cylinder will extrude
Parameters	
Radius	Defines the radius of the cylinder
Length 1,2	Defines the length of the cylinder in both directions
Mirrored Extent	Extrudes the *Length 2* dimension the same length as *Length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

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

7XUQ RQ Both sides WKH RSWLRQ DQG 2IIVHW VXUIDFHV DSSHDU DER
RULJLQDO VXUIDFH

Note: Since the offset surface has a Repeat object after OK option, you can use the Object Repetition icon on offset surfaces.

6DYH DQG FORVH \RXU GRFXPHQW

Review

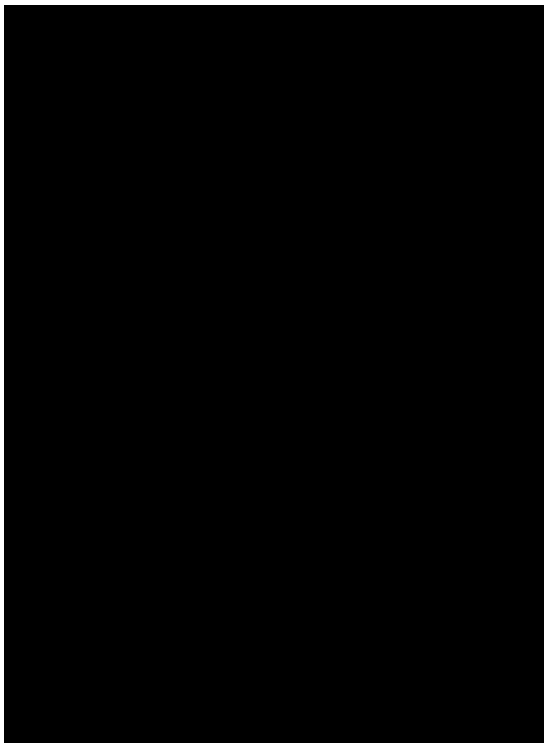
For this review exercise, you will create a computer mouse. The intention of the exercise is to demonstrate the process of building a solid model by utilizing wireframe and surface geometry.

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

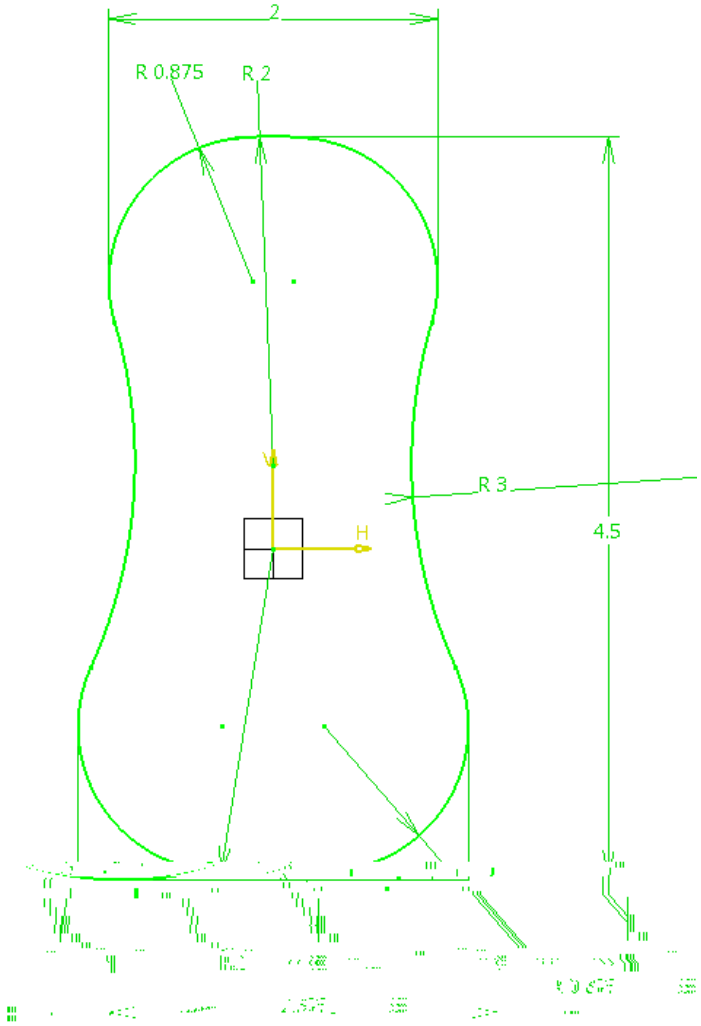
Mouse Body

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

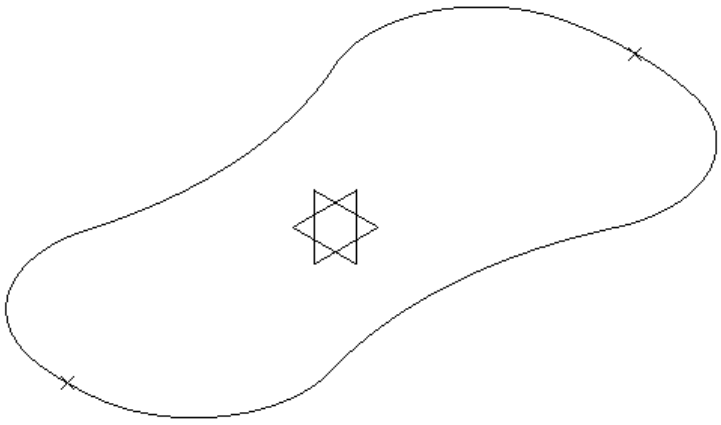
Insert a geometrical set named Mouse Body, then select the Positioned Sketch icon and set the options as shown below. 



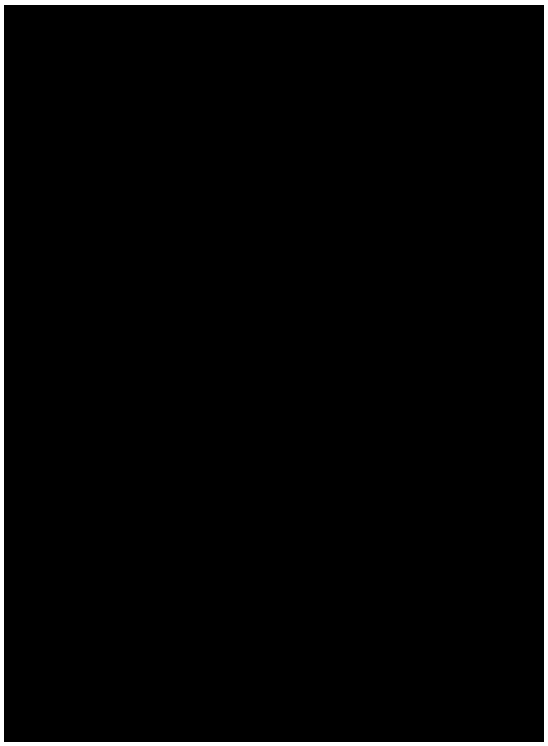
Create the following sketch. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.



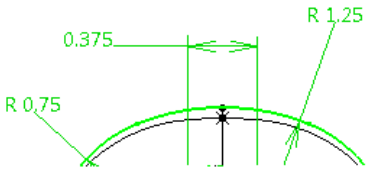
Create an extremum point at each end of the sketch in the yz direction. 



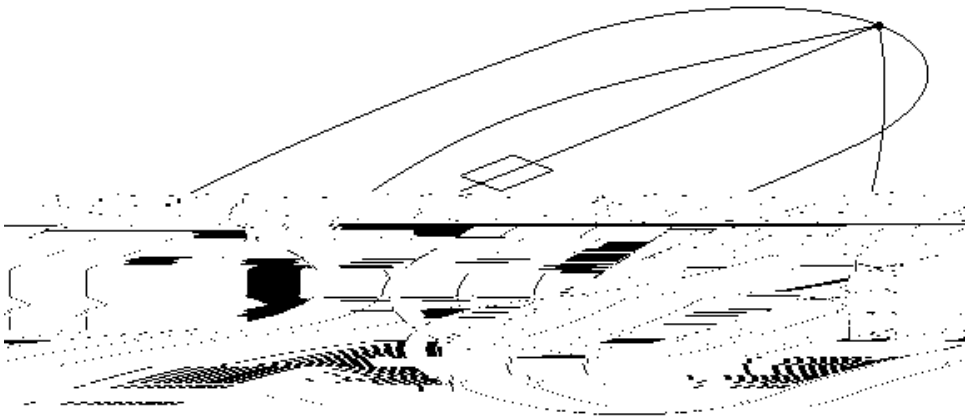
Select the Positioned Sketch icon and set the options as shown below. 



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.

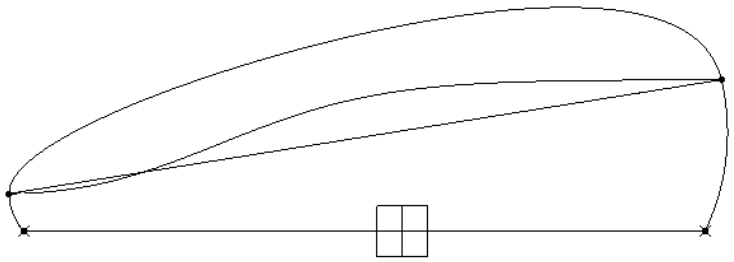


Your model should look like this.

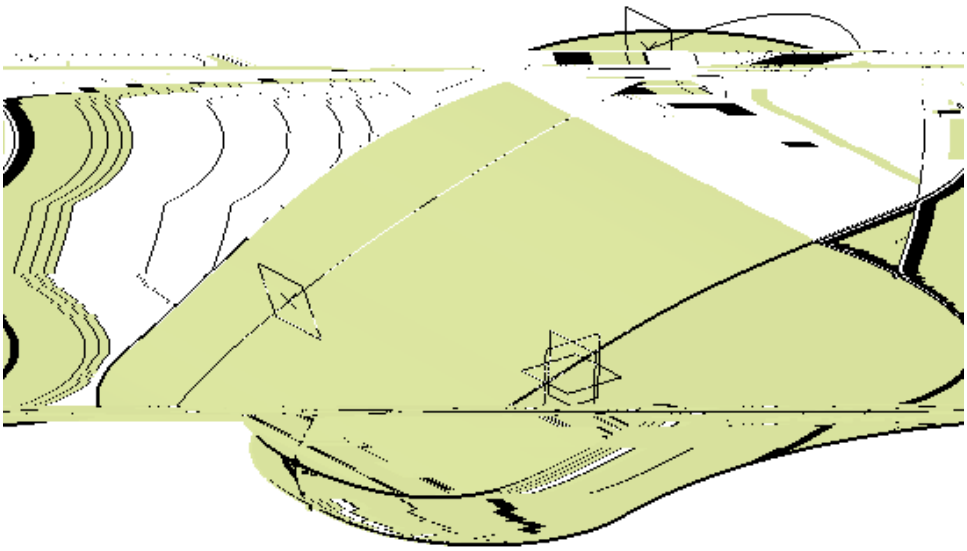


Create a spline between the two points at the top of each extracted arc.  The spline will be tangent continuous to both arcs with a tension of 0.375 at the first point, and 0.75 at the second point.

Fir

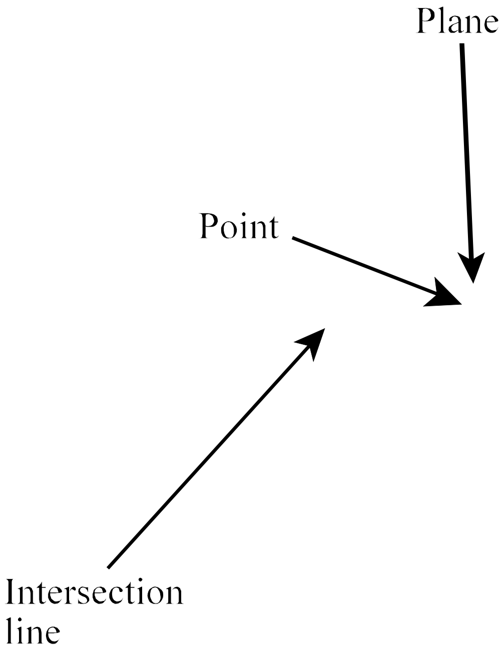


Fill your first sketch with a surface.  This is the bottom profile of the mouse.



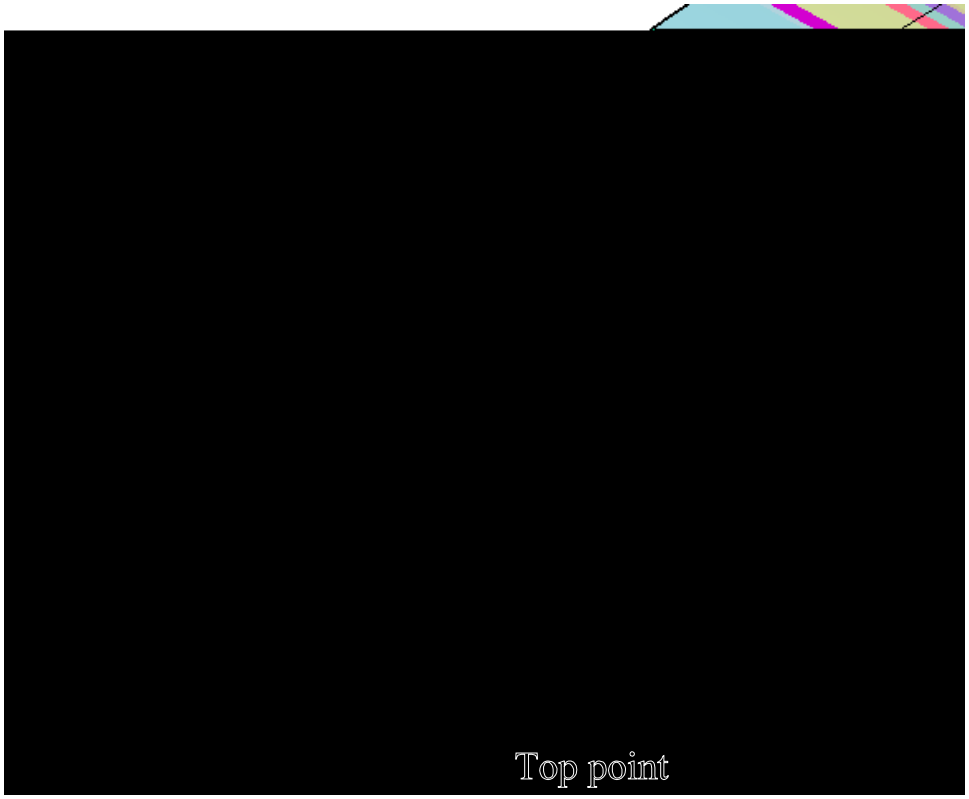
Create an intersection line between both planes and surfaces shown below.  You should have four, separate intersection lines.

&UHDWH D LQFK KHLJKW OLQH DW WKH SRLQW RQ WKH LQWHUVHFWLRQ FDWDHU W
SRLQW DQG XVHV WKH SODQH DV LWV VXSSRUW



&UHDWH DQRWKHU OLQH XVLQJ WKH VDP

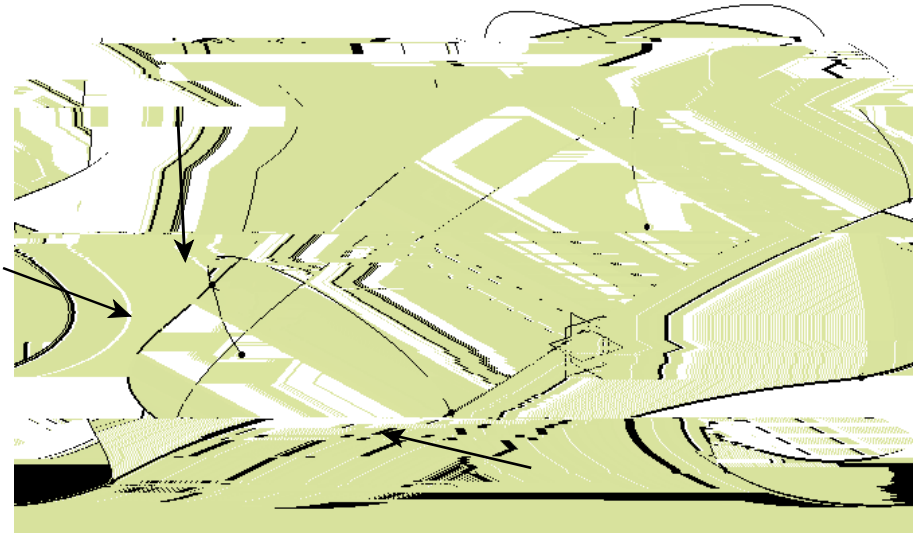
Next, create the orange spline shown below.  This spline is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.5 at the bottom point. Ensure the spline lies on the support plane indicated below.



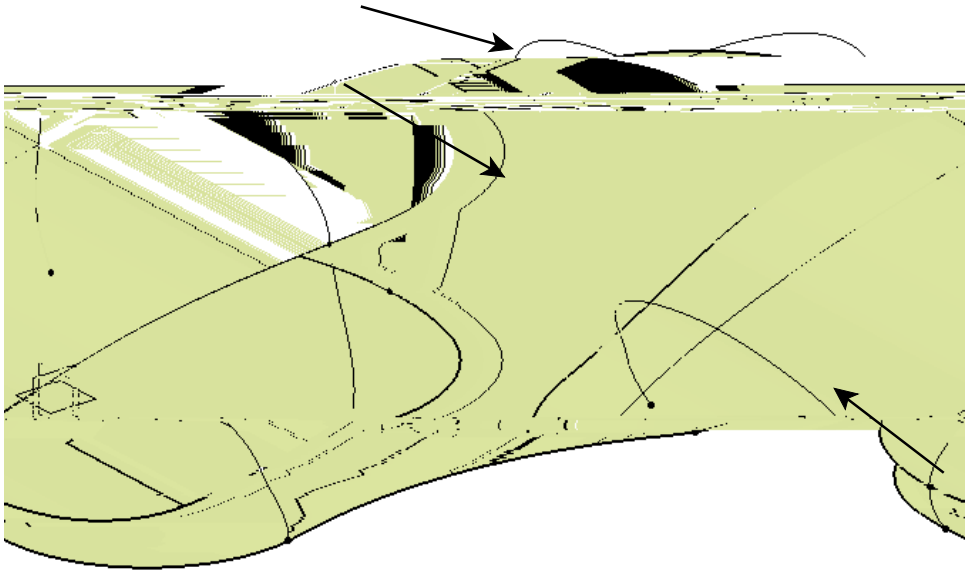
Create the orange spline shown below using the same method as the previous spline. It is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.25 at the bottom point. Ensure

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

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

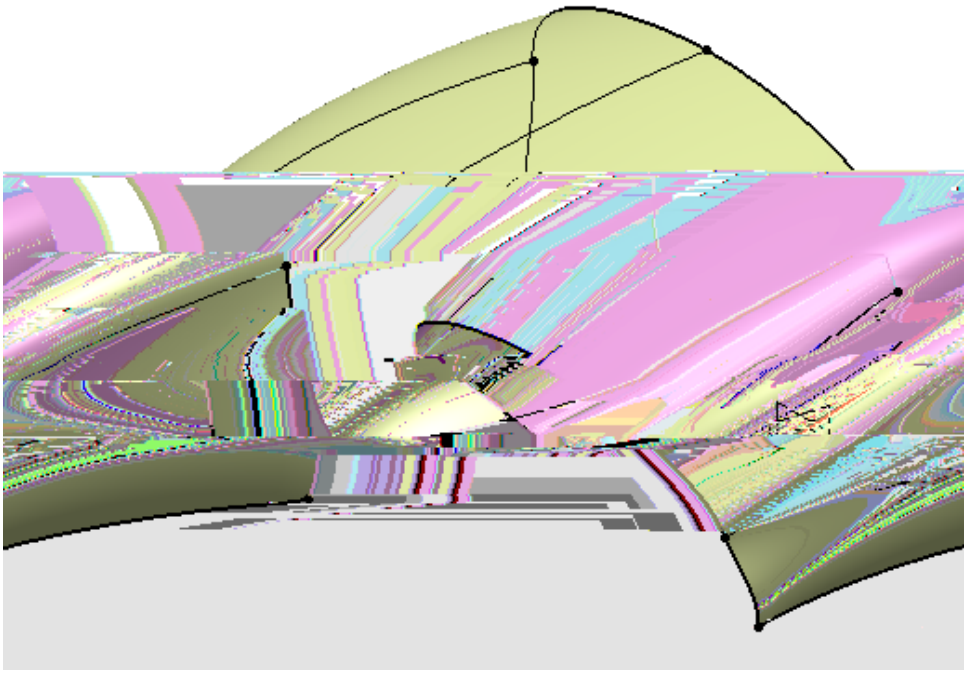


Create another join for the three curves shown below. Ensure they are tangent continuous.

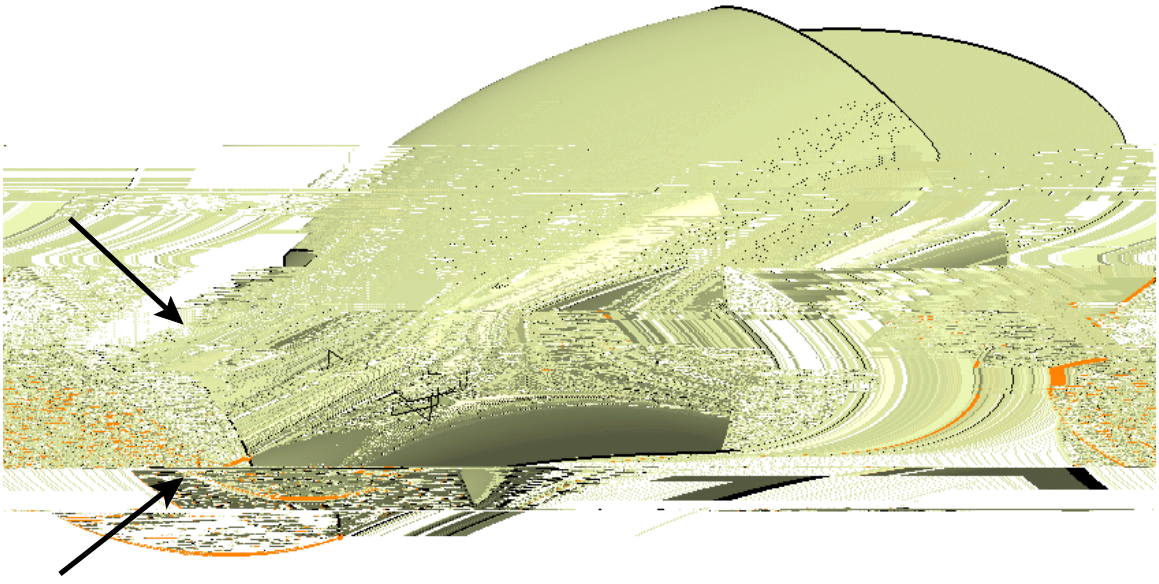


&UHDWH D ERXQGDU\ FXUYH RQ WKH IROORZLQJ HGJH

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.