
Surface Modeling - III

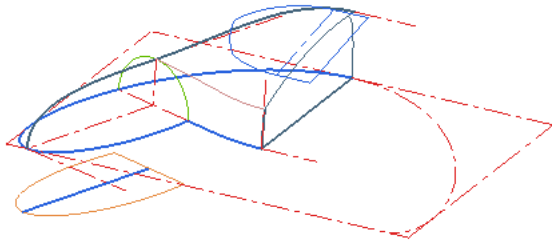
In this task, you'll learn about curves and surfaces by creating this video game controller. You'll also be exposed to Global Shape Modeling and Capping: two advanced tools used to change the entity's shape. To end this task we shall also use some of the solid modeling commands.


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1. Step 1: Creating the main surfaces

After downloading and installing this webtask, you'll find a model that contains all of the entities needed to follow this session. For better use, the curves have been placed on different layers.

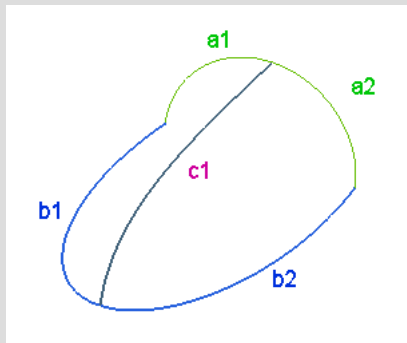


- Start the **Format**  **Layers** command and set current layer 0 and activate also layer 10.

They represent the first shape we will make. Sometimes, especially in this tutorial, we need to assign reference entities to describe the behavior of the new surfaces.

What surfaces?

There are different modes to make a surface starting from a series of curves.




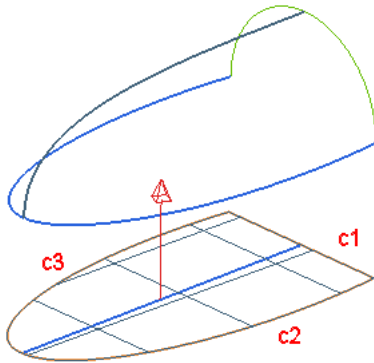
- Using b1 - c1 - b2 we can define a Grid surface.
- Joined a1 - a2 curves, we'll obtain a Proportional surface through b1 - c1 - b2 and a1a2.







Let's do the Global Shape Modeling! Now it's time for us to apply the **Advanced Modeling** (Global Shape Modeling) command. The purpose of this command is to change the shape of the entities defining a series of constraints, given by user. Let's use it.

While using Global Shape Modeling, we always need entities to change. This entities could be points, curves, surfaces or solids (simple faces or entire shape).

We want to change a planar surface and to modify it until we obtain the final shape.

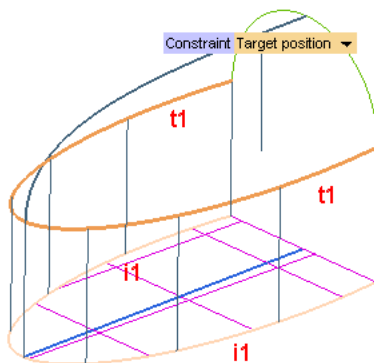
- Go to  **Plane Surface**.
- Select the three bottom curves (c1 - c2 - c3) that we have found in the start model.



- Activate the **Advanced Modeling** command.
- Pick  Initial curves under  Matching -  Matching Curves -  Group 1 in the Selection list.
- Select the i1 curves that have tangent continuity.
- Select now, under same  Group 1,  Target curves.
- Select the t1 curves keeping same order in the selection.

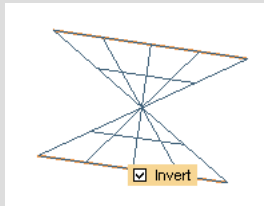
You'll see temporary lines that permit you to check the right parameterization of selected entities. These lines, shown as the initial entities will join the target.

- Make the ConstraintTarget position

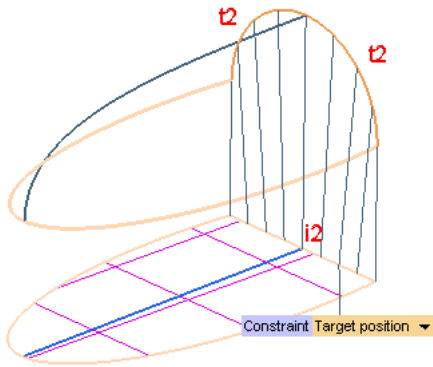


Curled parameterization!

Open 'More Options' if the temporary lines are curled and activate the Invert minidualog.

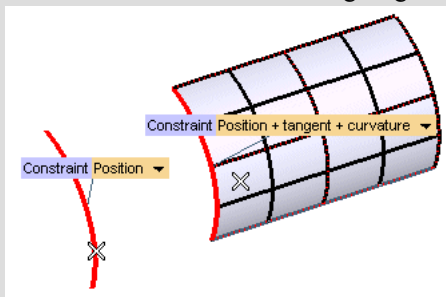


- Continue to assign constraints under \square Group 2.
- As \odot Initial curves select the i2 curve.
- As \odot Target curves select the relative t2 curves.
- Make the ConstraintTarget position



Curve or Surface Boundary!


There are different data selecting single curve or same shape but using the boundary surface.

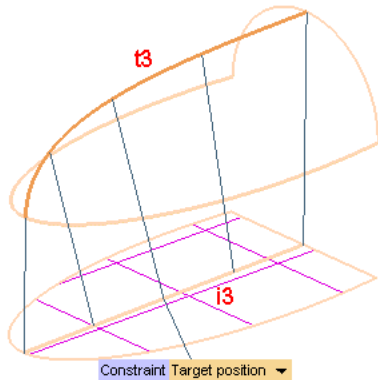


In the first case the maximum constraint will be ConstraintPosition instead in the second you can choose the surface data between Position -tangent -curvature and all their combination.




Now it is necessary to impose a crossing curve in the internal next shape.

- Opening under \square Group 3.
- As \odot Initial curves select the i3 curve.




- As  Target curves select the relative t3 curves.
- Make the ConstraintTarget position

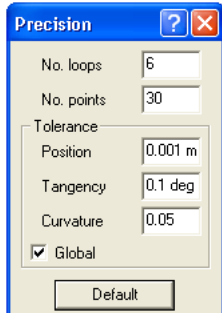
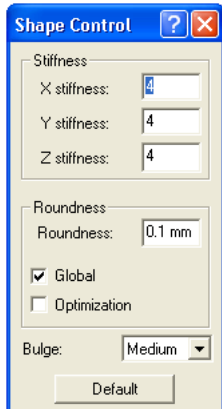


We need to select the area that we want to modify.

- Pick  Entities to be modified -  Entities Surfaces.
- Select the plane surface.
- Hits  Preview to show the result.



We have to set the parameters to define the better shape.

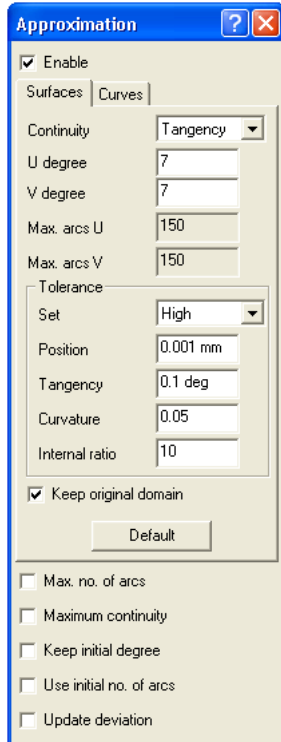
- Pick  More Options in the Selection list.
- Pick  Shape Control and  Precision; set the parameters as shown below.




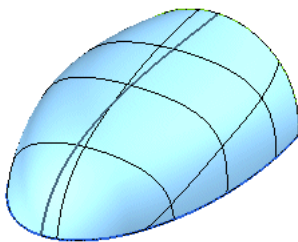
Global Modeling and NURBS surfaces; why use approximation?

Deepen these arguments through the GSM webtraining.

- Still in the the  More Options list select  Approximation option, check Enable and follow set parameters as shown below.

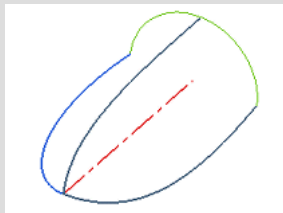


- Hits  Preview to show the result.
- Click OK to complete and exit the command.

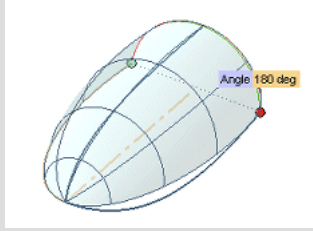


An other mode!

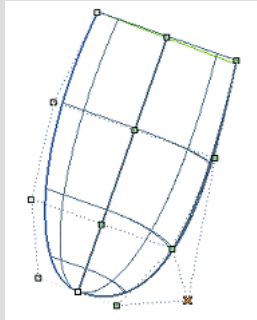
An other mode to obtain the similar shape of the last surfaces, could be changing the control points, in 3D environment, of one or more reference surfaces.



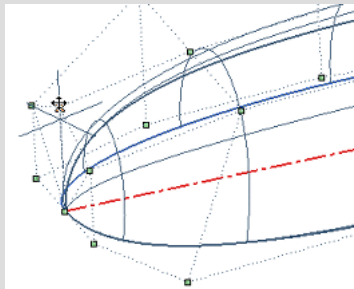
One rotational surface through some curve in our model.



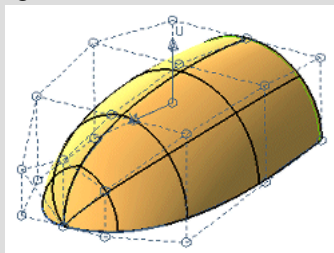
Editing the control points we have different points to drag. Using Horizontal-Vertical orientation with respect to the current view. Refer the image below.





Using XYZ vector to assign fixed directions in the space.

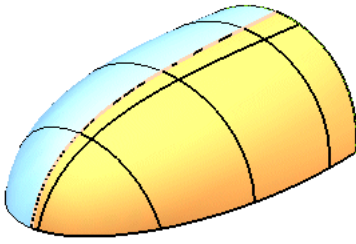


And obtain the result. The surface now presents few control points, arranged in ordinary mode and easy to manage.



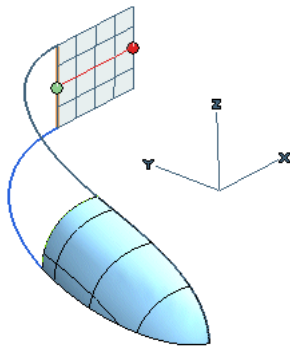
Obviously, we have taken a simple example compared to all options present in the **Modify Surface Control Points**.

- Go to **Modify**  **Surface**  **Trim with Limits**.
- Select internal curve as Limits.
- Select the GSM surface and keep both sides.

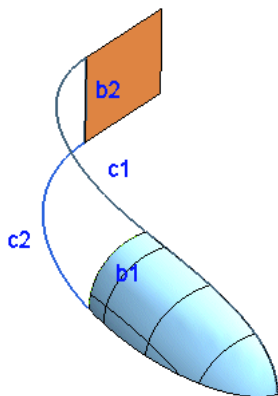


Let's continue to define other shapes. In some cases, especially with symmetry parts, we need to make reference surfaces to assign the better tangent or curvature continuity without the unwanted cuspid conditions.

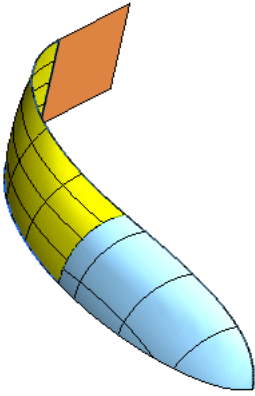
- Activate layer 30.
- Go to **Insert** → **Surface** → **Linear**.
- As in below image, use Direction option to X to make a reference surface respect YZ plane.




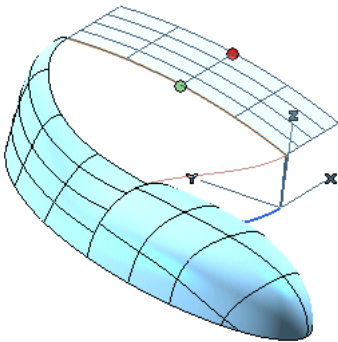
- Go to **Lofted Surface** command.
- As **Boundary Set A** select the curves c1 and c2.
- As **Boundary Set B** select the boundaries b1 and b2.
- Pick **More Options** and assign Proportional in the Type.
- Click **OK** to complete and exit the command.





Got that, we'll use other surface command to complete the part.

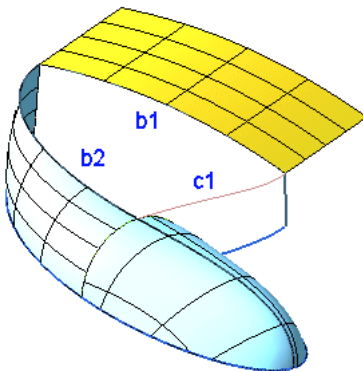


- Now activate layer 20 and layer 40.
- Insert  **Linear Surface** using the curve that lies on YZ plane and Direction normal to it.




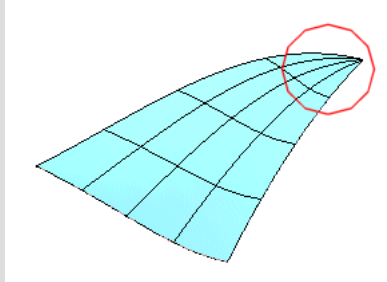
Let's do the Capping! Now it's time for us to apply the Capping command. The purpose of this command, as word 'Capping' say, is to make one surface about the selection of constraint entities defined from user. The 'GSM' engine will be applied to define a better shape through the selected entities. We'll also redefine the Capping to change with Associative parameter. Let's use it.




- Go to  **Capping** command.
- As  Constraint Boundary Curves, use different groups to add each entity (b1 - b2 - c1).
- For c1 and b2 curves use ConstraintPosition while for b1 select the boundary surface to assign ConstraintPosition + tangent.

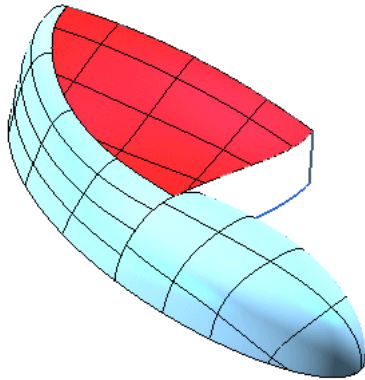






What Surface?

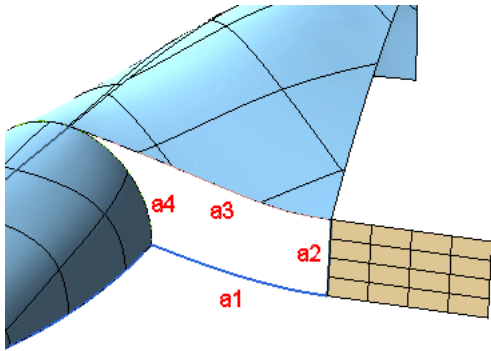
Using  **Lofted Surface** between b1 and b2 curves we obtain non-homogeneous parameterization. In fact the internal isoparmeter curve tend to finish at a generic point.



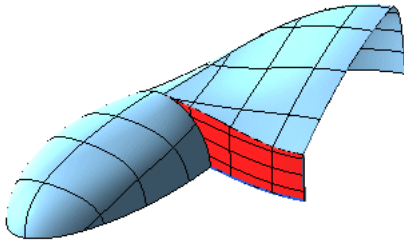
- Pick  More Options and assign same parameters used for the last GSM command for Precision, Shape Control and Approximation.
- Hits  Preview to show the result.
- Click  OK to create the ruled surface.



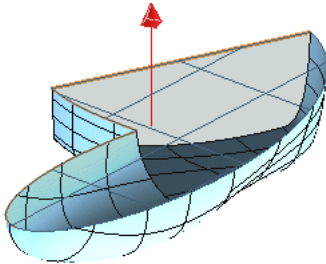
- Use the  **Capping** command again.
- As  Constraint Boundary Curves, use different groups to add each entity.
- For a1 and a3 and a4 curves use ConstraintPosition.
- For a2 select the boundary surface to assign ConstraintPosition + tangent.
- Pick  More Options and assign same parameters used for the previous Capping command for Precision, Shape Control and Approximation.
- Click  OK to create the capping surface.




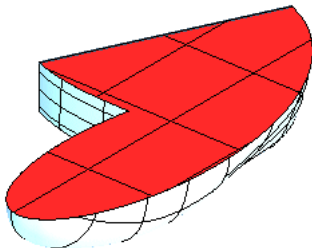
Here is the result. We still need two more surfaces to make the volume.




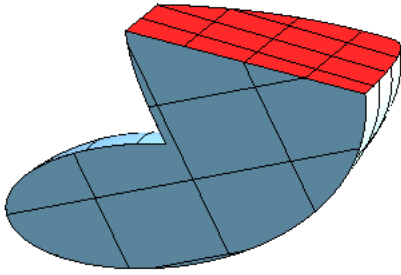
- Pick  **Plane Surface** and select the bottom surface's edges or curves where need.



- Click  **Apply** to create the first plane and ...




- select the symmetrical entities with the active  **Plane Surface** to close all parts.

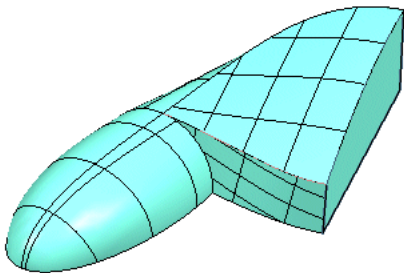




That will take care of the surfacing of the plastic part. Let's convert everything into a solid then we'll look at some fillets.

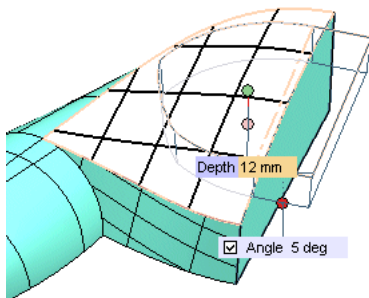
2. Step 2: Let's continue to modeling!

Let's complete our model using the hybrid modeling

- Pick  **Make Solid** command and select all surfaces.
- Click **OK** to complete and exit the command. Your solid will be shown in the history tree as a static solid.





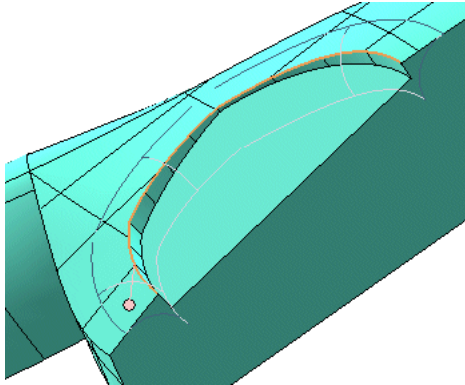
- Activate layer 60 to show one profile.
- Go to  **Linear Slot**.
- Select the profile and the top face as reference.
- Pick  More Options and assign Depth12 and angle5 removing material.




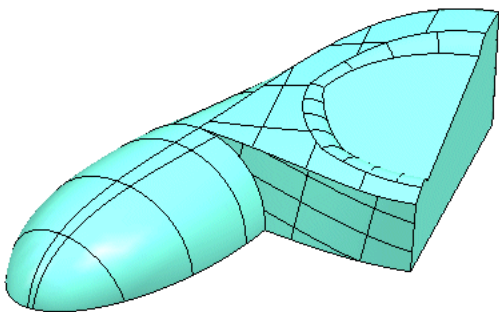
- Click **OK** to complete and exit the command.

Let's use other solid features to make the final shape.


- Start the  **Fillet Edges** command.
- Pick  More Options.
- Uncheck Continuous curvature.
- Check Tangency Chain.
- Uncheck Keep Edges.
- Select the top edges of the last slot feature.

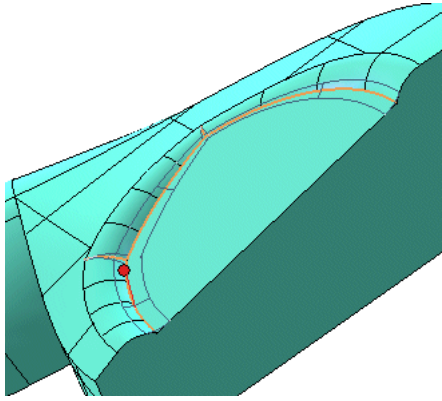


- Set the Radius to 8 mm.
- Set as Constant Radius mode.
- Click  Apply to create the fillet and stay in the command

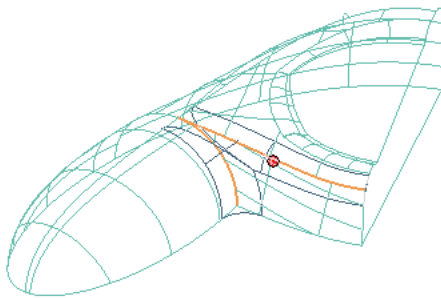



After this, we will have to remove the tangent discontinuity done in the last command. Select all of them as shown in the image below.

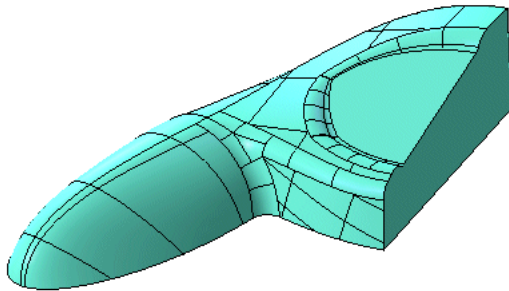
- Set the Radius to 3 mm.
- Set as Constant Radius mode.
- Click  Apply to create the fillet and stay in the fillet command




Now let's give a better shape in the front area. Select both edges as shown below.

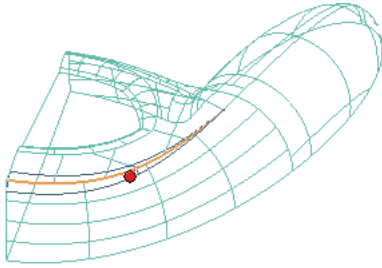


- Set the Radius to 5 mm.
- Set to Constant Radius mode.
- Click  Apply to create the fillet and stay in the fillet command

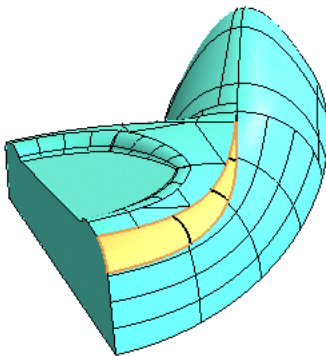


Now remove the tangent discontinuity on the rear side.

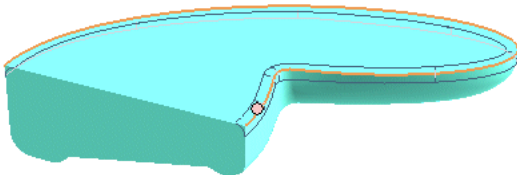
- Start the  **Fillet Edges** command.
- Uncheck Continuous curvature.
- Uncheck Tangency Chain.
- Uncheck Keep Edges.
- Select the top edges of the last slot feature.



- Set the Radius to 8 mm.
- Set as Constant Radius mode.
- Click OK to create the fillet.

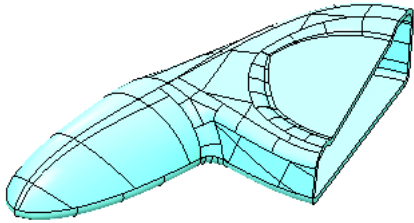



- Start the **Fillet Edges** command and select the bottom edges of the entire model.
- Check Tangency Chain.
- Set the Radius to 4 mm.
- Set as Constant Arc Length mode.
- Click OK to create it.




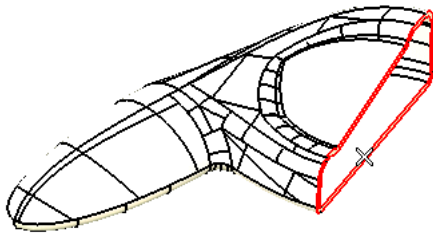
- Start the **Solid Shell** command
- Select the symmetrical planar face to indicate the part to remove.
- Assign Global Thickness to 2 mm



- Click  OK.

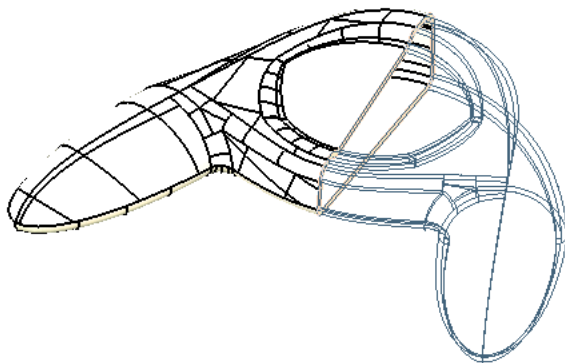



Use  **Mirror Solid** to mirror the solid.

- Start the  **Mirror Solid** command.
- Select SolidXX in the History Tree where XX represent the internal identifier number. So you have the possibility to select all global features assigned to the solid.



- Pick  First Symmetry Plane, choose On Face and select the closer planar face between external and internal side.
- Click  OK.



In the history tree you'll find a new static solid; it will be linked at first. Using  **Solid Union** and selecting both, you'll obtain the final object. It will appear a message that identify this solid like multishell; it's right because inside to the solid there will be a closed air cavity.

