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# Zone Modeling on parametric solid

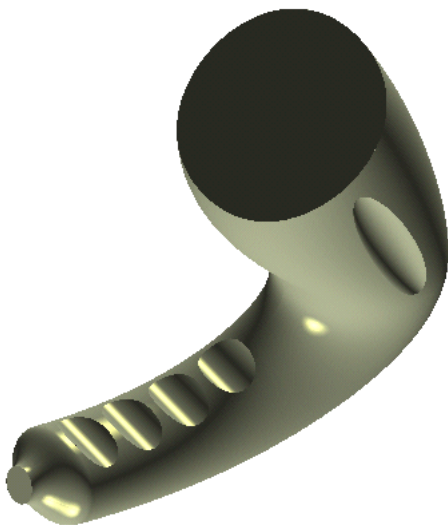
Zone Modeling is an extension of Global Modeling and enables you to apply all of the Global Modeling functionalities to selected zones (faces) of a solid. It is always associative and together with the Insert Feature Mode command, it can be used to modify a posterior zone of solid and propagate the controlled modification to subsequent features. This tutorial helps you to familiarize with the new command **Global Radial Bend** and also understand the capabilities of the **Zone Modeling** command.

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## 1. Step 1: Let's start...

Let us see how we can modify the features of given Hair drier model to suit our needs, using the concept of **Zone Modeling**. If you don't see the model file already on your thinkdesign window, open the model file from the downloaded folder which is typically the files folder inside the corresponding task folder wherever it is downloaded.



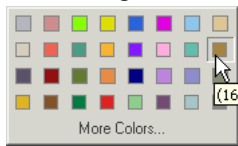
### NOTE:


With a double click on the exe file you can run the webtraining session. ThinkDesign will be open with the right model to start.

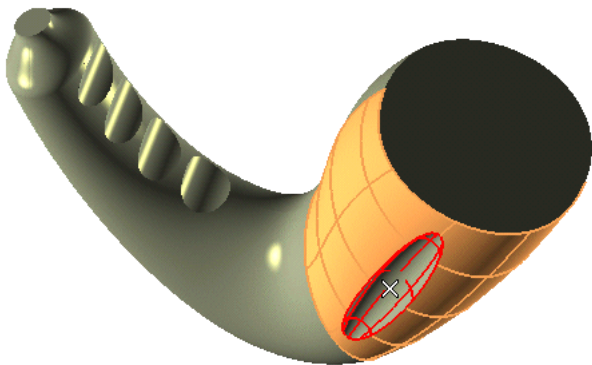
If request to open a file, you can find it in the C:\MyTraining path.

We just need to make some subtle changes to give them a better finish. For example, the grooves, provided for better grip, are on both sides of the handle. Let's try to remove the groove on one side.

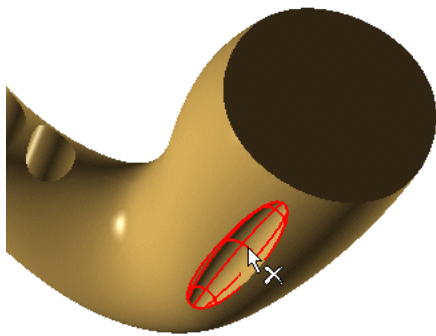
First, change the **Color** of the model. We select Color no.16, from the Color panel.




- Go to **Insert** → **Surface** → **Break Solid**
- Change the  Mode to Local from Global.
- Select the outer half face as well as inner conical surface.
- Click  OK to complete the command.

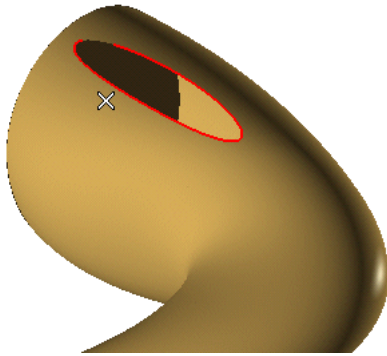


- **Delete** the inner conical surface as we do not need it now.



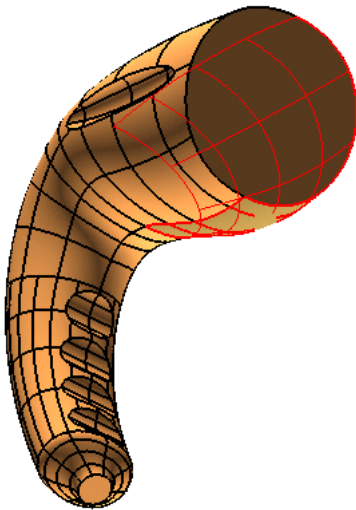
Now we need to get the original surface, which is in the untrimmed state.

- Go to **Modify** → **Surface** → **Untrim** command.
- Change the  Mode to Local from Global.
- Select the outer boundary of conical surface that we just deleted.
- Click  OK to complete the command. This will retrieve the original untrimmed surface.



Now, let's make a solid again.

- Go to **Insert**  $\rightarrow$  **Solid**  $\rightarrow$  **Make Solid**
- Select the exploded solid and the new untrimmed surface.
- Click  OK

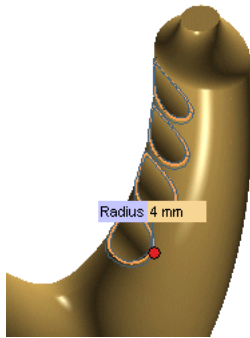


**NOTE:**

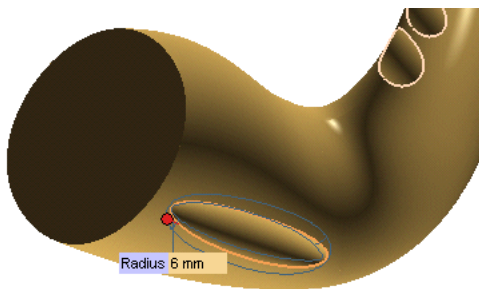
All these steps could be joined in one command. Load again the start model and use the **Modify**  $\rightarrow$  **Interactive Modeling**  $\rightarrow$  **Remove Faces** to remove the conical face. The command untrims automatically the face of the cover.

Let's create some fillets to have smooth edges on our model.

- Select all the edges on inner side of handle for a constant fillet. Now set the fillet with Radius4mm in the mini-dialog box. All these selected edges will form Group 1
- Select Continue from the context menu. This will allow you to select the  $\oplus$  Group 2 fillet.



- Select the conical edge on right side of handle as  $\oplus$  Group 2 fillet.
- Then, set the fillet with Radius6mm
- Click  OK



Now the model looks better. But, where do we put our thumb? The back edge of our model is round! Let's create a thumb rest area on our model.

## 2. Step 2: Keep your thumb.

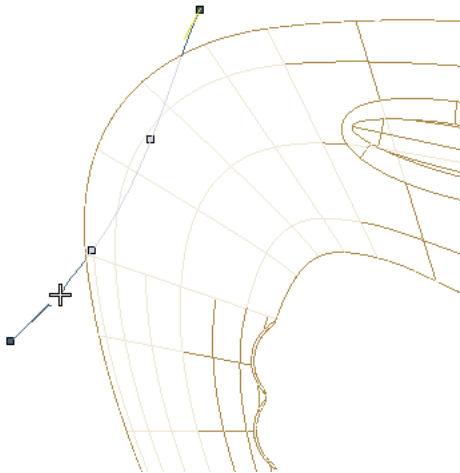
Let's make a place to keep the thumb. This will give you that extra grip to hold the Hair Drier properly.



Let us draw a curve which will use to cut the corner zone of this handle.

- Press  $\boxed{F8}$  to see Front View of model.

- Change the view to **Wireframe View**.
- Go to **Insert** ➤ **Curve** ➤ **Interpolation Points**
- Select 3 or 4 points along the round corner one by one to create the curve preview (as shown below).
- Click  OK to create the curve..

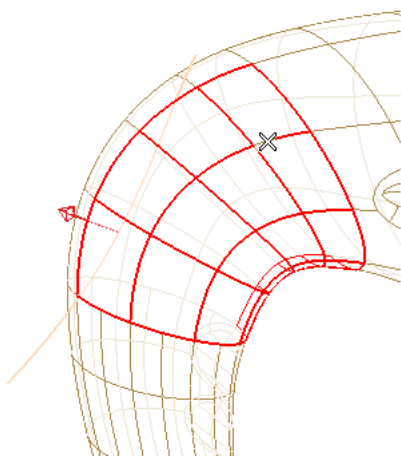


Let us edit the curve to make it smoother and continuous.

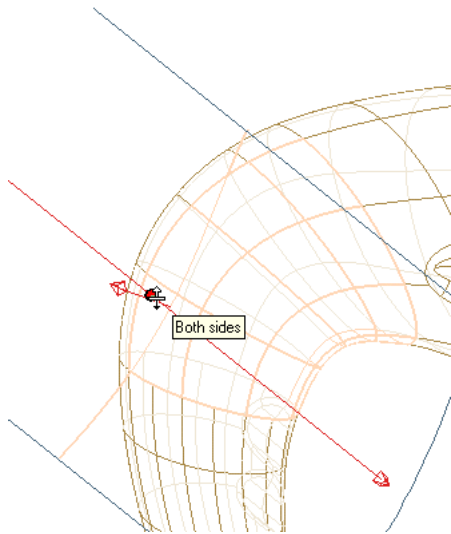
- Go to **Modify** ➤ **Curve** ➤ **Interpolate Points**.
- Drag and adjust the tangency vectors that extend from the end of the Interpolation curve to edit it.
- If needed, click and drag the interpolation points till you get a curve that would help you to rest the thumb.
- Click  OK.

We will be using this curve to cut the solid.

- Go to **Insert** ➤ **Solid** ➤ **Sweep** ➤ **Linear Slot**
- Select the curve as  Profile
- Select either of the corner faces as the  Face to cut.

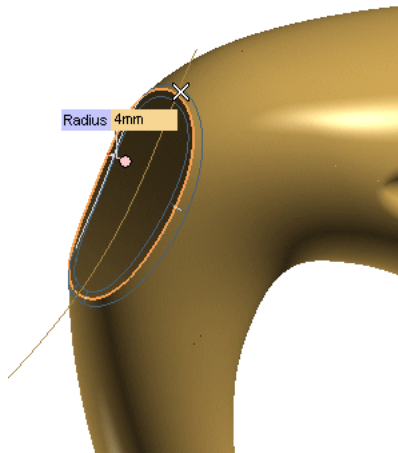


- Once the face is selected for slot operation, preview will be shown. Double click on the handle seen at the centre to change the option to Both Sides from One side.
- Make sure the material cut direction is away. Otherwise double click to invert the direction.



Now, let's provide a small fillet at the sharp edges of cut.

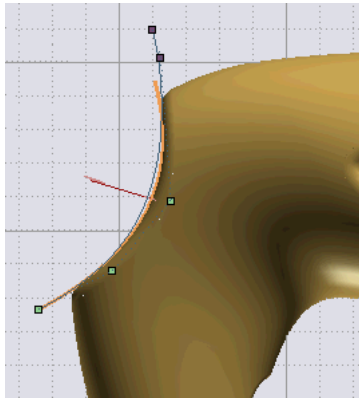
- Go to **Insert** → **Solid** → **Fillet** → **Edge**
- Select the sharp edge of corner cut.
- Specify the fillet with Radius 4mm and click  OK.





Hmm... want to change the profile a bit? No problem.

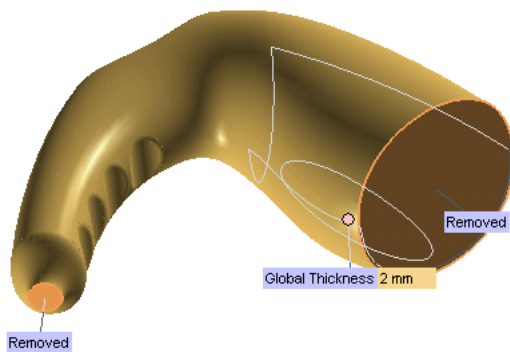
- Double click on the Profile curve to edit it.
- Once you are in profile mode, double click the curve again to edit the curve by control points or use **Modify** → **Curve** → **Interpolate Points**.
- Select the top 2 control points or others and then try to drag it forward or backwards. Once you are sure about the position, click  OK to adopt the changes.
- Double-Click in the background to exit Profile Mode and to **Rebuild Model**. The profile and the associated

Slot feature will be updated to the new position.



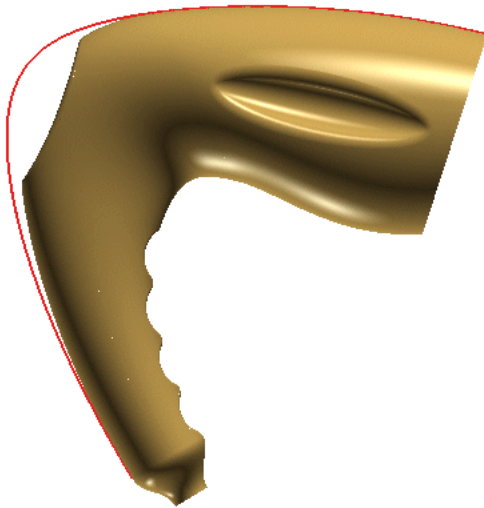
Let's Shell the solid to give it a consistent wall thickness.

- Go to **Insert** → **Solid** → **Shell**
- Select the flat faces at the exhaust and at the electrical cord ends of the hair dryer as the  Faces to be Removed
- Define a Global Thickness 2mm for the shell.
- Click  OK to adopt the changes.



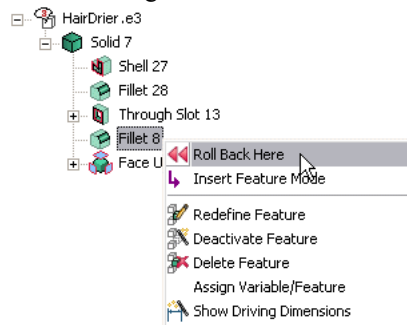
This will create a hollow solid with specified shell thickness. What if we wanted to change the outer curvature shape just a little bit. How would we do this? Check the next step and see.

### 3. Step 3: Want to change the curvature?



Let's draw the initial boundaries of the solid.

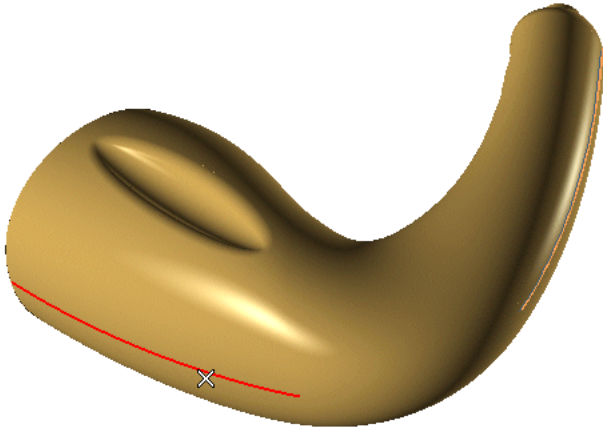
- **Roll Back** the History tree. Right click on Through Slot operation and select Roll Back Here. This will give us the original curvature of the handle.



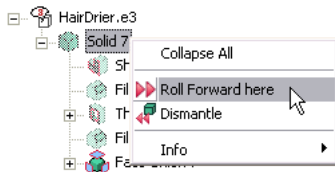
Change the **Line Width** to 2 and **Color** to red(28).

- Go to **Insert** → **Curve** → **Boundaries**
- Associative Mode needs to be OFF if you don't want lost the feature just done.
- Set  Boundaries option to Select to draw only selected boundaries.
- Select one boundary at the upper-back edge and the other boundary at the bottom-rear edge of the part as shown.
- Click  OK to create the boundary curves.







Now that we have our boundary curves, go back to the History tree and right click on the top solid and select **Roll Forward** to restore our model.

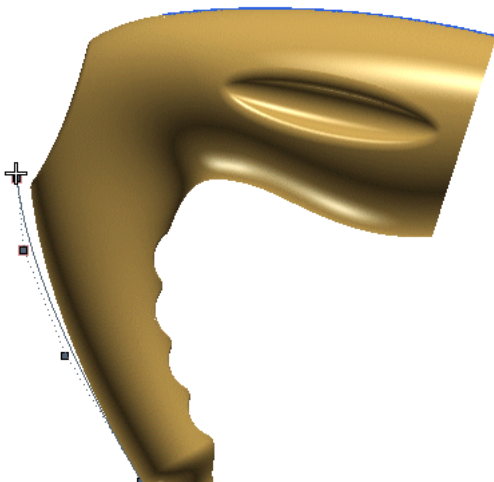


### Roll back behavior

When using the Roll-Back command, there are some important factors to be considered. If you roll back and insert a History based event, all of the rolled back history will be lost. Instead, use Insert Feature Mode to insert a history based event. The reason we could do it here is that creating boundary curves is not a history based and therefore, does not adversely affect our History tree.

Let's edit the profile shape of the hair dryer.

- Start the **Modify** ⇨ **Curve** ⇨ **Control Points** command and click on bottom curve as  Curves to edit. Select the top 3 control points of the curve using a selection window.
- Using your mouse, pull the control points away from the solid and click  OK to complete the command.

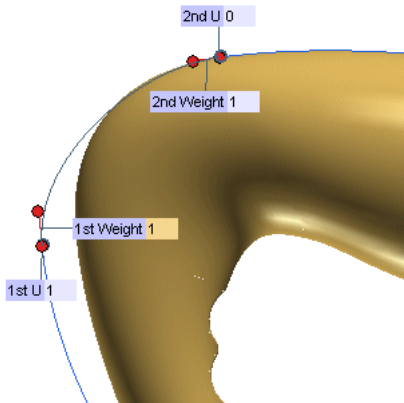


We need to connect this modified bottom curve, with the top curve.

- Go to **Insert** → **Curve** → **Connect**
- Select the modified bottom curve as **First Curve** and the unchanged top curve as **Second Curve**.
- Change the Continuity option to Smooth Curvature.
- If you want analyze the result, hit right click on the workarea and select Info and Curvature. Leave the Curvature Scale at 1000 and Density at 5 which are default values, in the selection list.
- Make sure that the U parameter is at the two ends of the curve. They will have values of 0 or 1 .
- Drag the 1st Weight and 2nd Weight Handles to get the desired curvature.

#### Why should I set U parameter to zero or one?

Setting the U parameter to a value of 0 or 1 will force the new connecting curve to be at the absolute end of our two initial boundary curves. These initial points could be anywhere on the initial curves but in this case, they have to be at the ends.



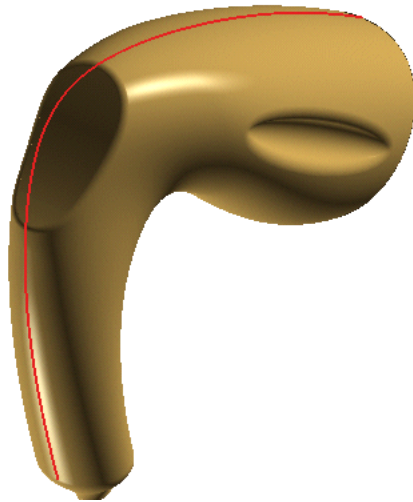
Click  OK to complete the command.

#### Warning.


Make sure that there is no overlap between the connect curve and the initial curves. This can cause problems in the Zone Modeling operation. If there is any overlap, use the **Trim/Extend Curves** to remove overlaps.

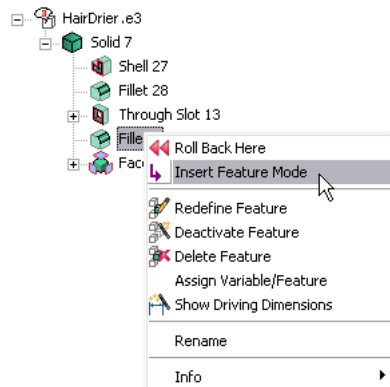
## 4. Step 4: Time to use Zone Modeling.

Now it's time to use **Zone Modeling** to change the current curvature. Let's see how we can achieve this.

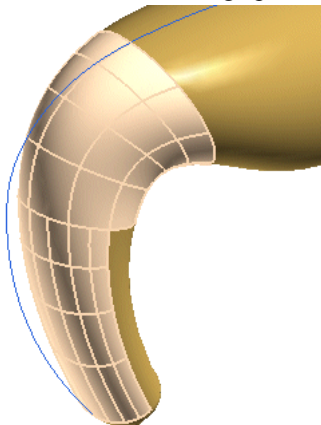


We will go back in the history tree to the point, where we added the fillet onto the elliptical groove on the side.

- Go to History tree and right click on the Fillet feature that was applied to the elliptical groove, and select  **Insert Feature Mode**.




- Go to **Zone Modeling**.
- Select all the four faces adjacent to the central curve.
- Click on AutoPreserving option to preserve the selected face's boundaries automatically.

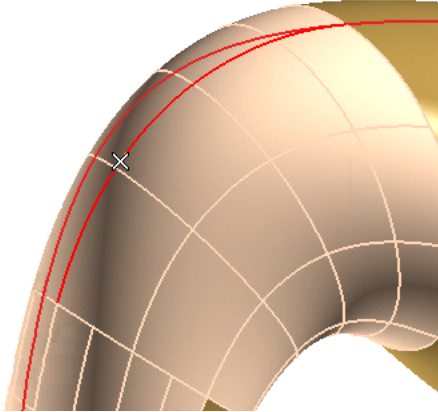



After selecting AutoPreserving option, all curves to keep will have a continuity based in data constraint.

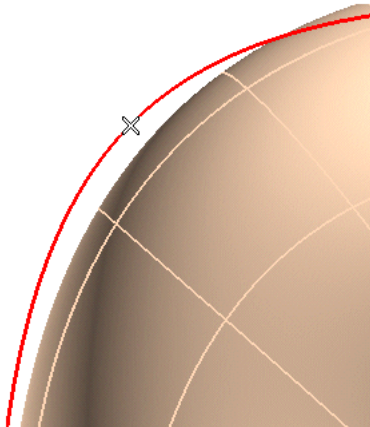
- Next, for all curves (Group 1, 2, 3 and so on), set the constraint to: ConstraintKeep position + tangent.

Now, we will select the Initial Curves and Target Curves for the Zone modeling command.




- Click on  Initial Curves. Select the boundaries that run up the back of the hair dryer. Select the top boundary first and then the bottom one sequentially.



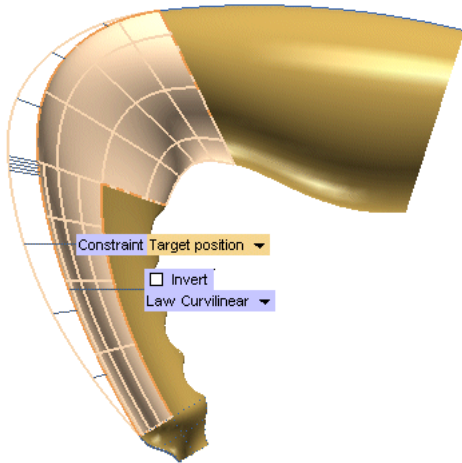
- Click on  Target Curves. Select the Connect curve and then the modified bottom curve. The selection sequence for Initial and Target curves should be same.



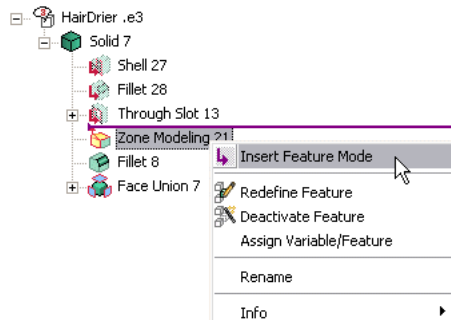
#### ZM selection: How to get it right??

How do you know you have made the right selections? How do you know you are ready to apply the command? There are two easy clues that tell us if we are ready to apply the ZM command. The first is that we will see the  OK,  Apply and  Preview buttons at the top of the selection list. The second clue is the preview vectors (or lines) that extend from the initial curves to the target curves.

- Once, all the essential inputs are provided, the  Preview of the model will show that all the old curves are drawn to new position.



- Set Law as Curvilinear. If the minidiialog doesn't appear, open the More Options list.
- Select the More Options in the Selection list.
- Select the Precision option in the Selection list.
- Change No. Loops to 3
- Click OK to complete the operation.
- Now, again go back to History tree and you will see that, the **Zone Modeling** feature is added in the list just above the fillet.
- Right click on Zone Modeling feature in History tree and click on **Insert Feature Mode**. This will bring back the model to latest phase.





- Once you come out of the **Insert Feature Mode**, the Rebuild button will be active.
- Click on **Rebuild Model** button to rebuild the whole model. After completing the operation, you'll see that, the curvature of model is changed and it is on the lines of the new curve.

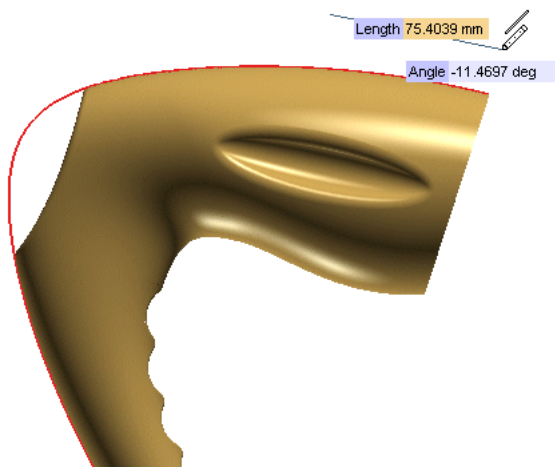
## 5. Step 5: Get surprised !!!

Now, it is time to change the front end of the Hair Drier. The mouth looks too wide. How do we make it more elegant?

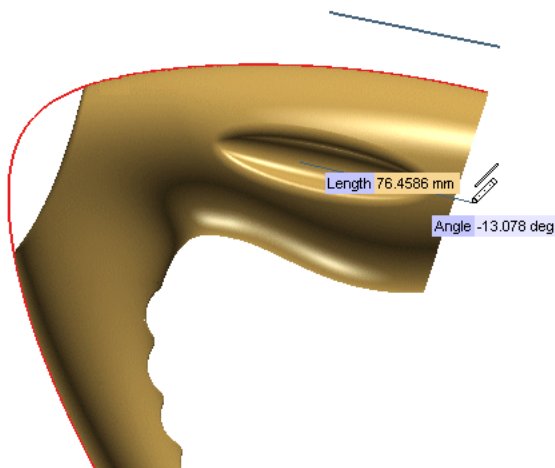


We need to have some basic curves so that we can modify the front face of a Hair drier. So, let's draw some base profiles.

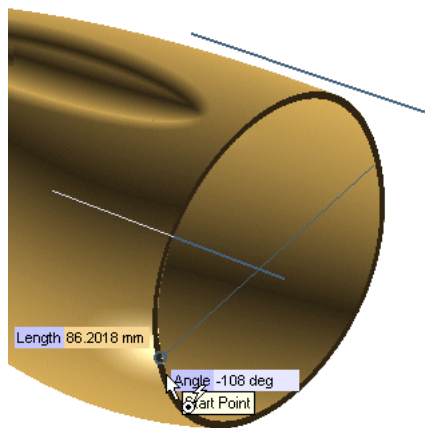
- Go to  2D Profile mode from  Model mode.
- Start the **Two-point Line** command and set the Type to Limited.
- Draw a two point line that is approximately parallel to the centerline of the front hole. We will use this line as a bend line later.



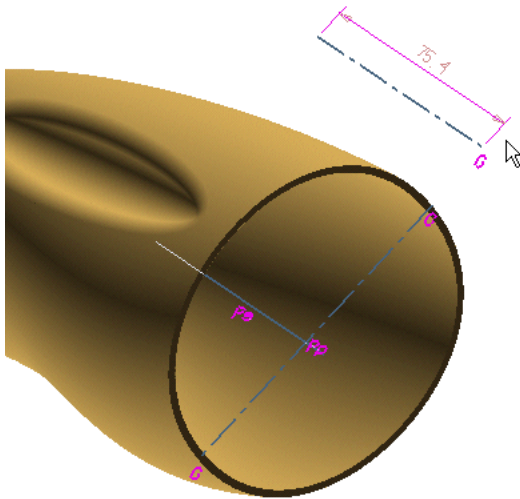
- While still in the **Two-point Line** command, draw one more line, approximately passing through the centre of hole. We will use this line as Axis of Revolution later.



- Now, draw a Chord line from the top to the bottom of the front opening of the model. You may use the **Two-point Line** command for this. Make sure your Grid is turned off so that the line snaps to the model and not the grid.

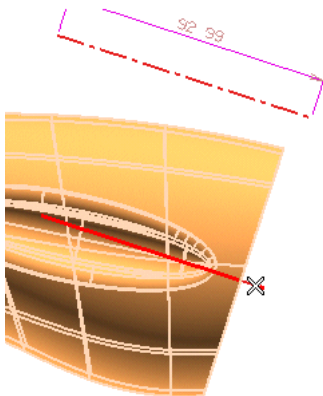


- Select the Bend Line and the Chord line. Right click to show the context menu and select **Make/Unmake Reference** to turn these lines into Reference lines.
- Put a **Ground Constraint** constraint to both ends of the Chord line. Put one on the right end of the Bend Line.
- Put a **Parallel Constraint** for Bend Line and Axis of Revolution Line.
- Insert **Perpendicular Constraint** for Chord and Axis of Revolution Line.
- Click on the **Smart Dimension** command and select the Bend Line to place a dimension on it.



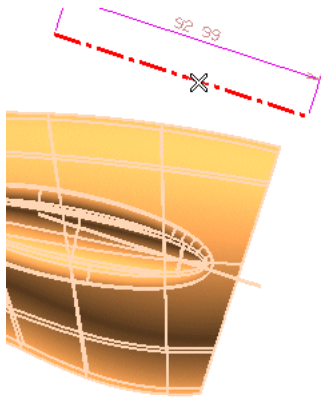
Now, we will use the **Global Radial Bend** command to modify the front face of our model.

- Change the default **Color** to Grey.
- Go to **Modify** → **GSM** → **Global Radial Bend**
- Check the  **Associative Mode**.
- Select as  **Entities to Be modified the complete solid** .
- Hit  **Axis of Revolution** and select the Centre line drawn.

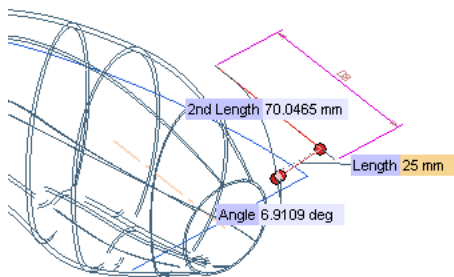


- Select the outer line drawn as  **Bend Line**.

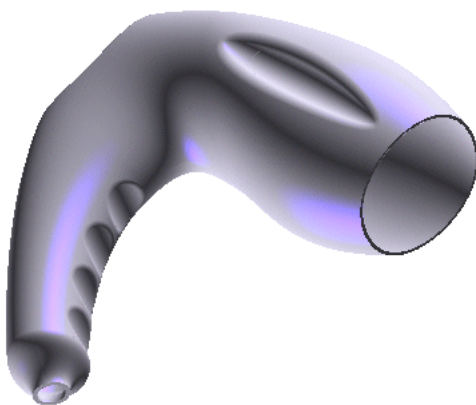




- Now, by changing Angle and Length and 2nd Length parameters, you can dynamically change and view the changes in model.
- Try changing the Angle and Length parameters to see the effect of each parameter. Take some time here to understand how each one of these parameters functions.
- Once you are satisfied with the final shape, click  OK to complete the command.



This will create a new solid containing the original solid and the Global Radial Bend feature. In the History Tree, right click on the **Global Radial Bend** feature and select redefine to see how easy it is to play with this feature.



This ends our web task on Zone Modeling. While running through this task we also happen to touch upon topics

like curve creation and editing. We also tried our hand at another new function, the Global Radial Bend command. We hope it was an enriching learning experience for you.