# *ZW3D*
# *Advanced Bottle Example Training Guide*

***Copyright and Trademarks***


Floor 4, NO.886, Tianhe North Road, Guangzhou 510635 P.R.China
(8620)38289780

# ZW3D™ V2014 Advanced Bottle Example Training Guide



Printed in the P. R. China.

# Forward

The full version of the ZW3D documentation is available by selecting **Help Browser** from the ZW3D **Help Menu**. Go to the **" Program Folder\doc"** directory on the ZW3D program directory and open the file **"ZW3D.chm".**

This **Advanced Bottle Example Training Guide** is available as a printed manual and in PDF format. To view or print the PDF version, install the Adobe Acrobat Reader and open the corresponding files in the **" Program Folder\PDF"** folder in the program directory. Refer to the **HELP Manual** for information relating to specific tasks.

**NOTES:**

1. For best results, view and print the PDF version of the documentation using the Adobe Acrobat Reader.
2. Since the **Advanced Bottle Example Training Guide** is only a portion of the complete documentation, it is highly recommended that you utilize the **HELP Manual**. You will find additional topics, more information on command options, input options, additional command notes, and cross-links to related tasks and topics.
3. It should be noted that the HELP Manual will always be more up-to-date than any printed or PDF material. If you find any discrepancies between the two, always refer to the HELP Manual.
4. While you are working with ZW3D, simply press **F1** to view the documentation about the command you select.

To help you customize your CAD/CAM requirements, please contact our Customer Support department on our website http://www.zwsoft.com/ or call at (8620) 38289780 ext 565. You can also contact your local sales representative or application engineer to discuss your requirements.

Thanks for being our customer!
Team ZW3D

# *Table of Contents*

## Example one, The Shampoo Bottle.

With this example, we will start with a basic design that includes both the bottle and the cap shape. From this we will export the geometry for the bottle and the cap into two separate parts. With this method the two parts have an import geometry operation which imports the relevant geometry from the basic design shape. If the user changes the basic shape, the cap and the bottle can be updated by performing a regen. Additional topics that will be covered are constraints in the sketcher, variational swept solids, and lofts with continuity attributes.

All of these examples are done in mm units.  You will want to set your configuration accordingly.

## Create the main bottle shape.

With our example the designer has the basic shape of the bottle and cap in mind.

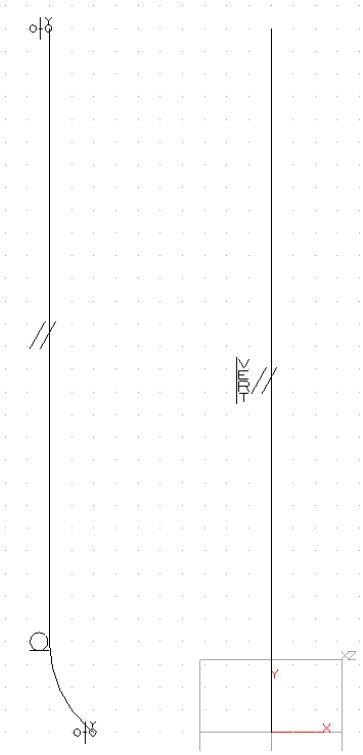
Create a new **multi-object file** called "**Shampoo.Z3**". 

Next create a part called "**01_Shampoo_bottle_main**", 

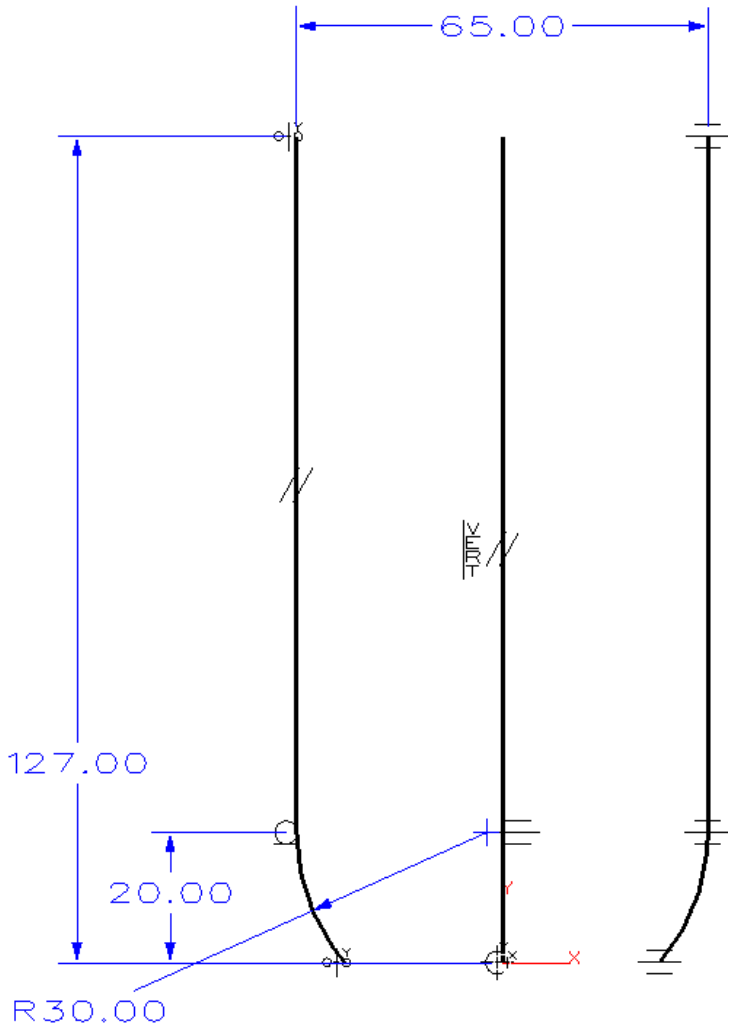
We will use the **Variational Sweep** command to create our main bottle shape, we will decide on the cap shape a little later.
First, we need to create three sketches.

**Create the first sketch on the XZ plane**  . This sketch will control the size of the bottle cross section.

- Right-click **Draw** a vertical line starting from zero.
- Then **Draw** a vertical line starting on the top left, aligned with the existing vertical line.
- Make the end point of the line about 20mm in Y from zero.
- While still in the quick draw command, re-pick the current end point to change the continuity option to tangent and create the arc, as shown on the left. Note that the end point of the arc is aligned in Y with the end point of the first line.
- Add horizontal constraint between the end of the arc and the origin. 
- The sketch should now be as shown opposite.

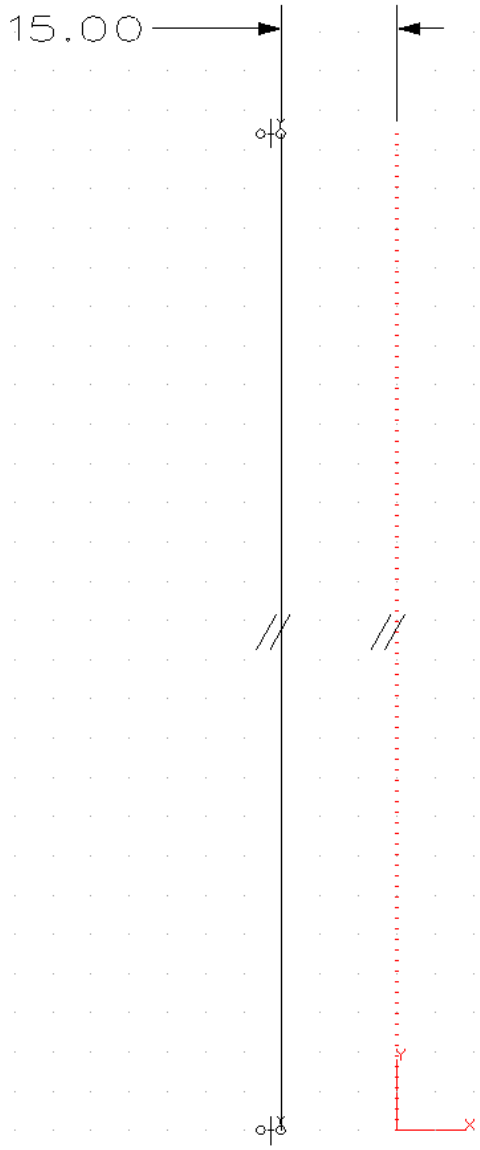


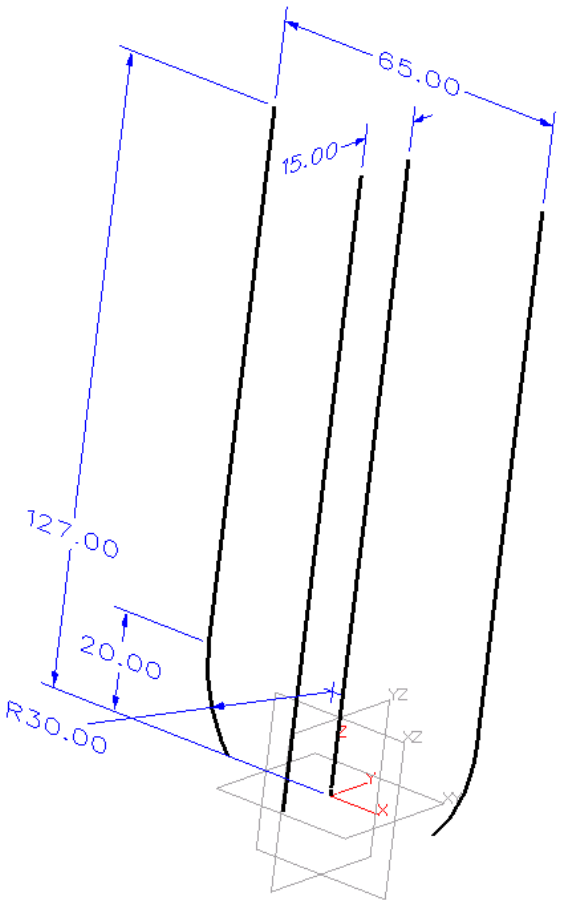
- **Right-click Dimension** and add the two vertical and the radial dimensions as shown.
- Use the **Mirror Entities** command to mirror the left hand line and arc about the Vertical line as shown opposite.
- Use the Quick **Dimension** command again to add the horizontal dimension.
- Edit the values of the dimensions to be the same as those shown opposite.
- Check your constraints
-



Next **create the second sketch on the YZ plane**. This sketch will control the thickness of the variational sweep bottle shape.

- We need to create the line shown opposite.
- The line must be parallel to, and have the same length as the middle vertical line from the first sketch.
- First, using the reference curve command, create a reference curve; pick the vertical middle line from the first sketch as shown.
- Now create the line.
- Next dimension the line, and edit the value to **15**mm as shown opposite and below.
- 

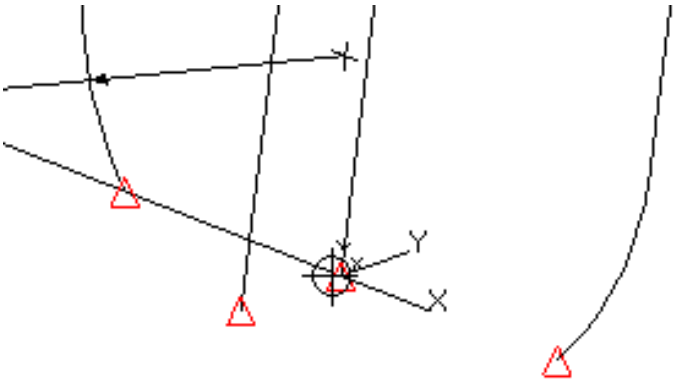


The Third sketch defines the cross sectional shape of the bottle as it sweeps along the drive curve. The two previous sketches control the size of the cross section; this sketch defines the shape. In our example the designer has chosen an elliptical cross section. We will create the ellipse in our next sketch and constrain it to the geometry created in sketch one and two.

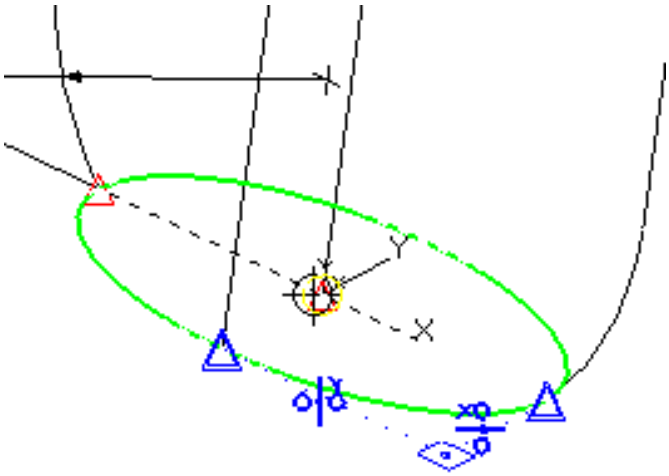
**Create the third sketch on the XY plane.**

- We need to create reference points using the geometry from the first two sketches. This will force the ellipse to be attached to the sketch one and two curves, as it sweeps along the drive curve.
-  Make sure you pick the correct command. Choose the **Curve Intersection Reference** command, do **not** use the point reference command.
- The Project reference point command will Project points that remain fixed in position on the sketch plane.
- The correct command, **Curve Intersection Reference**, also creates points, which are used to constrain the sweep profile sketch. This sketch is then regenerated as it sweeps along the drive curve.
- The correct use of the **pick filter** is very important. Make sure the filter is set to "**sketch**". If for example the filter was set to the curve option, and the user picked one of the arcs from sketch one, the sketch profile would only be constrained to this arc and not to the entire length of the arc plus its tangent vertical line.
- **Repeat** for the second sketch.
- The points are shown opposite as small red triangles.

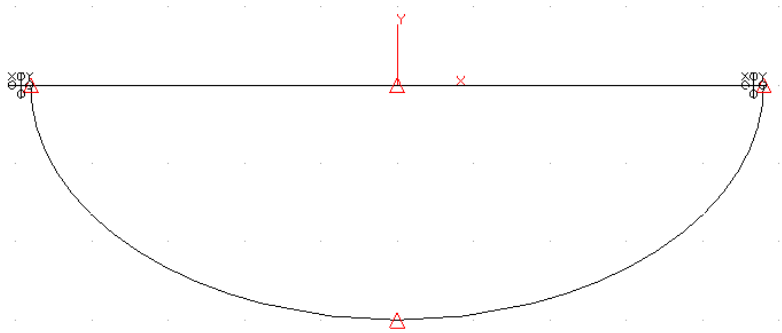



- Create an ellipse (Center), through the center point, and using the smart pick options align the "corner point" of the ellipse to three of the reference points as shown opposite.
- Erase the dimensions and all the constrains except those at (0,0)

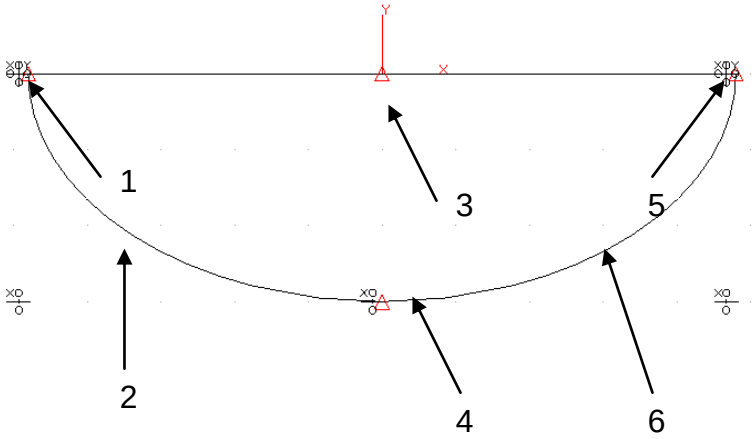
We don't want a constant size ellipse. It needs to follow our previous sketches.



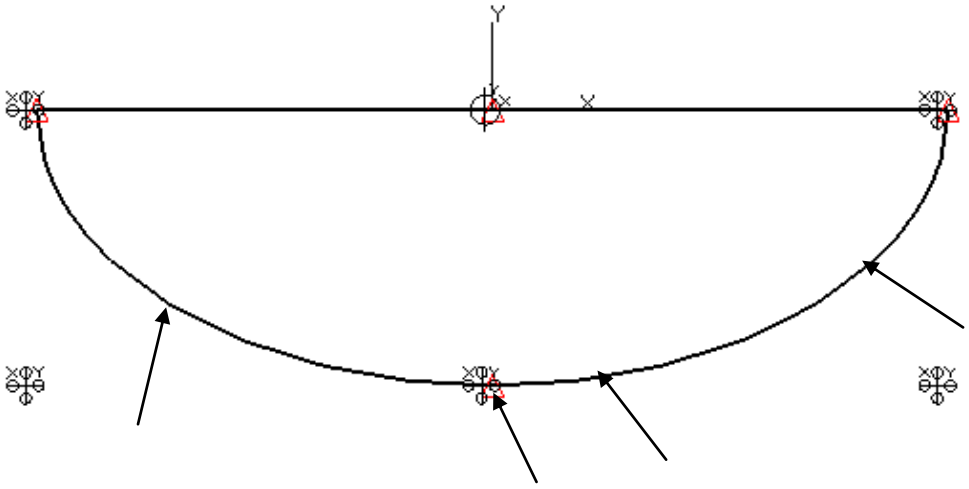
- Next create a horizontal line through two of the reference points.
- **Trim** the ellipse as shown.



- Use the **points vertical constraint** command and constrain the points in X as shown as follows.
- Select the first base point, by picking the reference point.
- Select the point to be constrained to the base point, by picking the point on the curve **(not the reference point)** opposite in Y to the base point.
- Repeat this command twice, for the other two sets of points, as shown opposite.



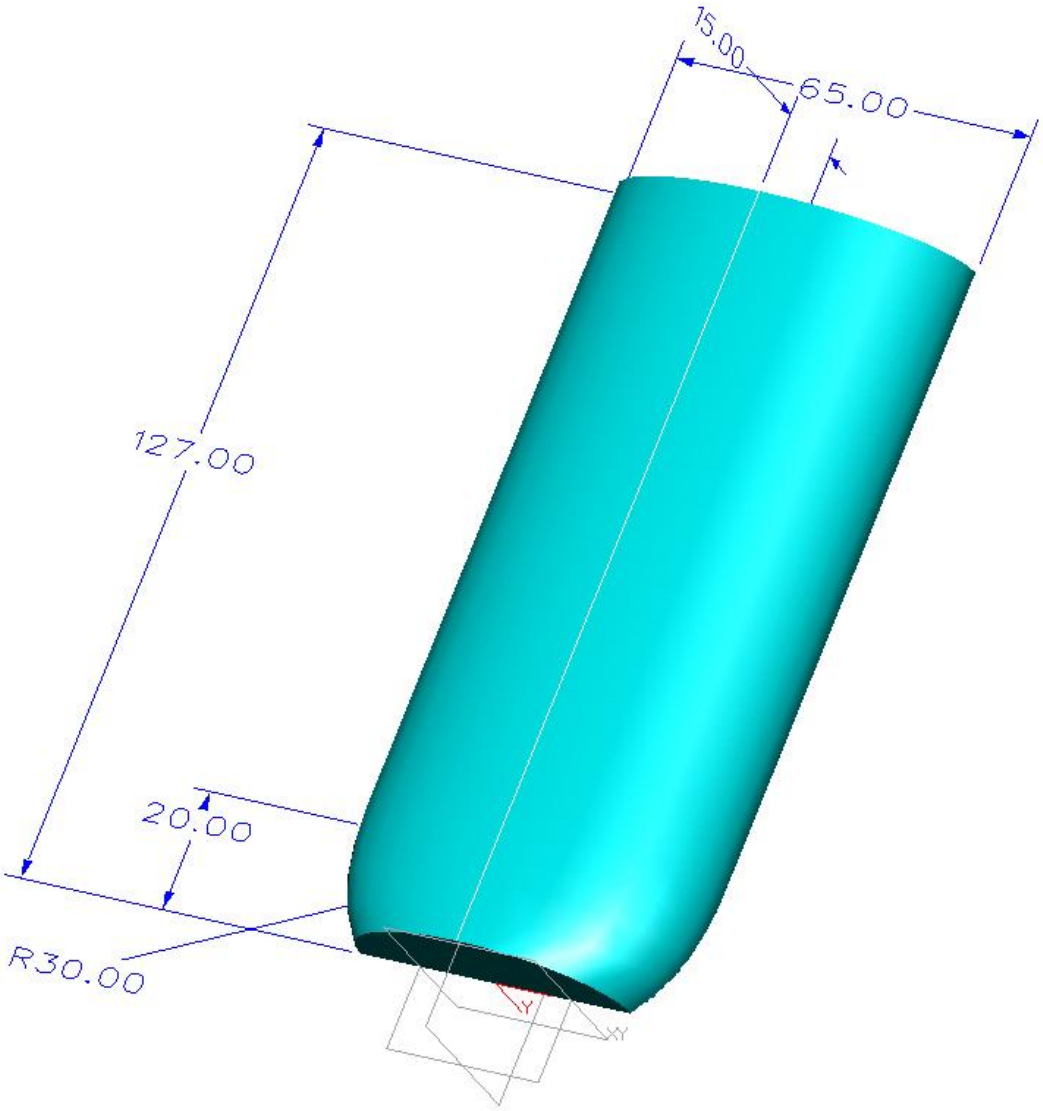
- Finally use the **points horizontal constraint** command, to constrain three of the ellipse points to the reference point in the middle and on the ellipse curve, opposite the zero point, as follows.
- First select the command.
- Next pick the reference point opposite the zero point in Y. this is the base point. Make sure you pick the reference point and not the ellipse point.
- Then pick the ellipse point (not the reference point) opposite the zero point in Y. To correctly pick the ellipse point, place the mouse pointer on the ellipse curve near to, but not on, the reference point.
- While still in the command pick the two “corner” points of the ellipse.
- The sketch and its constraints are shown below.



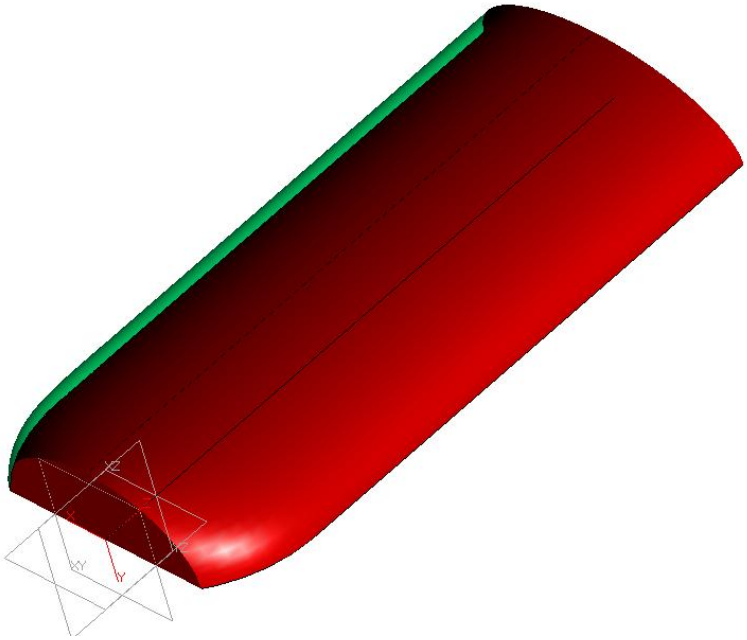

-  Exit the sketcher.

Using the “**Variational sweep**” command, we will next create the main bottle shape.

- Pick **Variational sweep**
- Select the ellipse sketch as the Profile.
- With the **pick filter set to curve**, select the middle vertical line as the Path
  The pick filter is set to curve because for the drive curve we need the middle vertical line only and not the entire sketch1 geometry.
- The elliptical shape from sketch3 will now be swept along the vertical line (drive curve) from sketch1, continually regenerating the sketch3 shape, keeping it attached to the “rail” curves of both sketch1 and sketch2, as shown below.



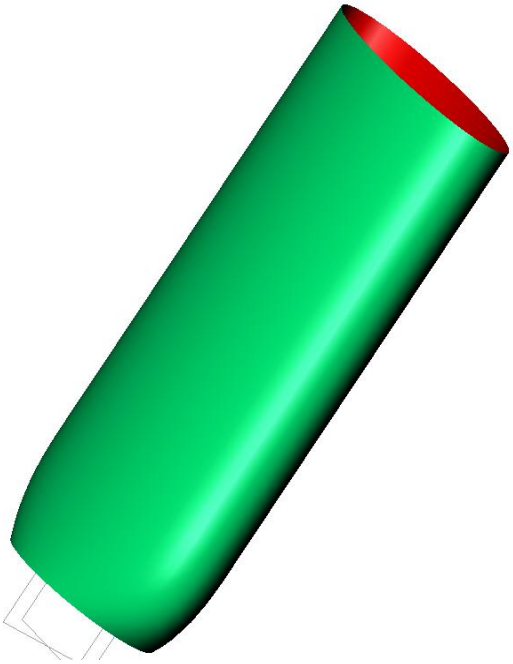

- We now need to mirror the one half of the bottle and join the two halves into one.
- First let's **erase the faces that are not needed**.
- With the pick filter set to "Face" erase the two faces as shown opposite.



- Select the **Mirror** command, and mirror the existing shape (**Pick filter set to shape**) about the XZ plane.

- Next use the "**Combine shapes add**" command and join the two shapes together as shown opposite.

Or **Sew** the two open shapes. The result is the same.
Or you could have used the Mirror option "Add"



Shortly, we will be creating the cap shape, but first let's create a new layer and assign the sketch geometry to it.

Sketch geometry is usually “blanked” automatically when a command such as “Extrude” is used. But sometimes, as in our case, the sketch geometry stays visible. Sketch geometry can be erased. We can also blank the sketch geometry, but I find that using a layer solves the problem nicely.

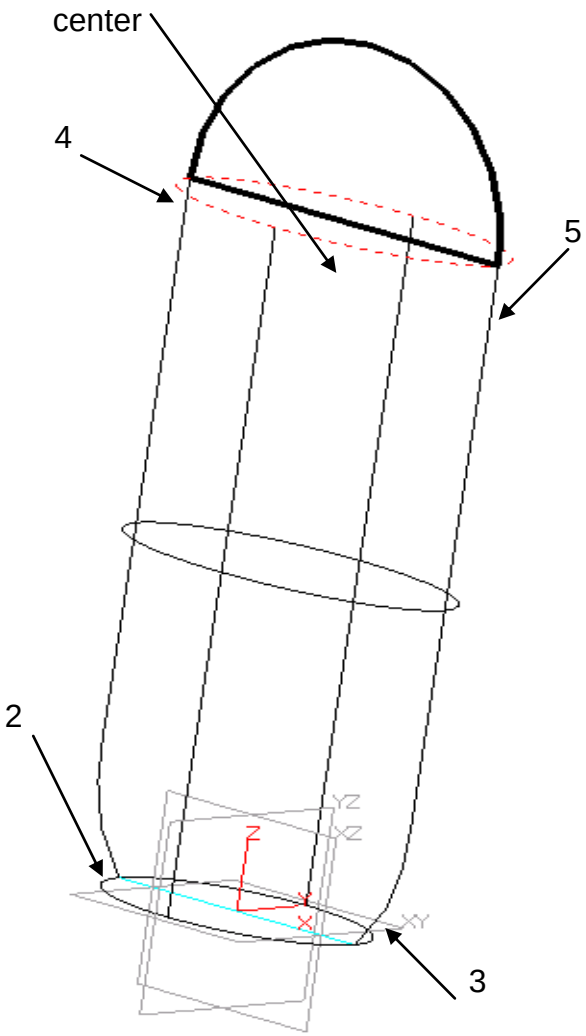
- Pick the **Layer Manager** from the Document Aware Toolbar. 
- Select “New” button from the bottom of layer manager to create a new layer which named “Construction Geometry”, and then pick “OK” .
- Assign the Sketch to “Construction Geometry” layer. Select the Sketch, and then select “Construction Geometry” from the layer list located in Document Aware Toolbar.
- Now , hide the Sketch. Open the Layer Manager, click **On** cell to turn off the “Construction Geometry” layer (the label turns to  ).

Since your new layer is turned off, the sketch will turn off.

Pick the History Manager tab to continue modeling.

## Let’s create the shape for the cap.

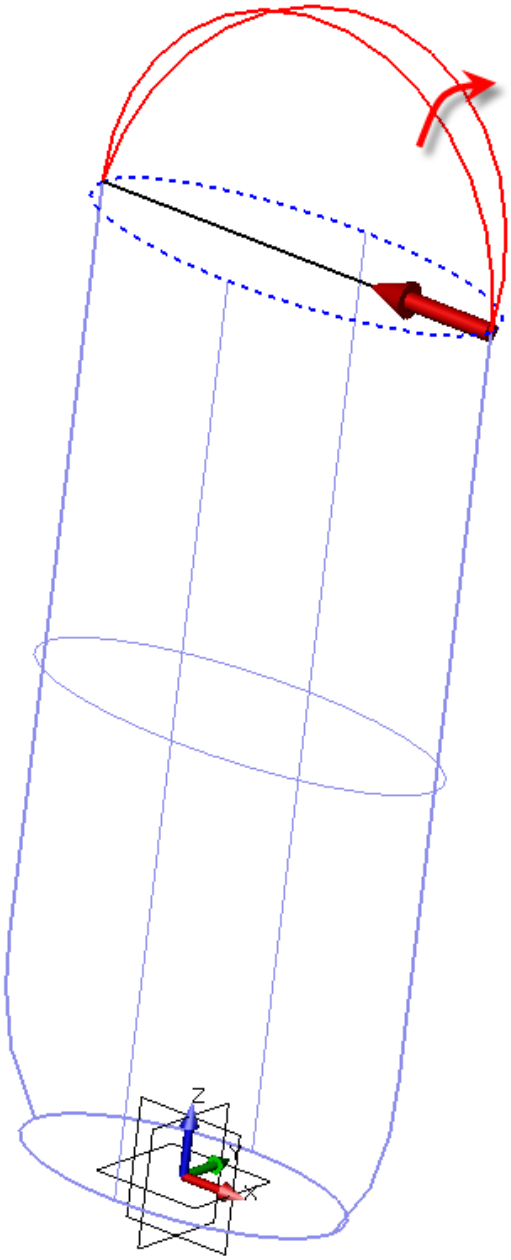
- As shown, **create a 2pt line** through the endpoints of the face edges on the far end of the bottle. Use the wire-frame geometry this time instead of creating a sketch.
- Next, again using wire-frame geometry, **create an arc (center)**.
- Because we are using wire-frame geometry instead of sketch geometry, the arc needs to be “fixed” to some plane. Sketch geometry is fixed to the sketch plane. We will use three points to define an implied plane, which is the same as the XZ-Plane.
- The center point (point one of our implied plane) will be the center of the newly created line, pick the middle of the line.
- The next two points (together with the center point) locate and fix the arc to our implied plane.
- For the two plane points pick the end points of the edges (on the XZ-Plane) at the other end of the bottle. The next two points are the start and end points of the arc.
- Middle-click to skip Angle
- Pick the left-hand point (as shown on the sketch above) first and the right-hand point second.




- Next we need to rotate the newly created arc 20 degrees about the line.
- Press **Crtl+A** on the keyboard to set the view as shown below.
- From the **EDIT > move entities** suite of commands, select the "**along a direction**" option. 

This command allows for movement of a distance as well as rotating about a direction vector.

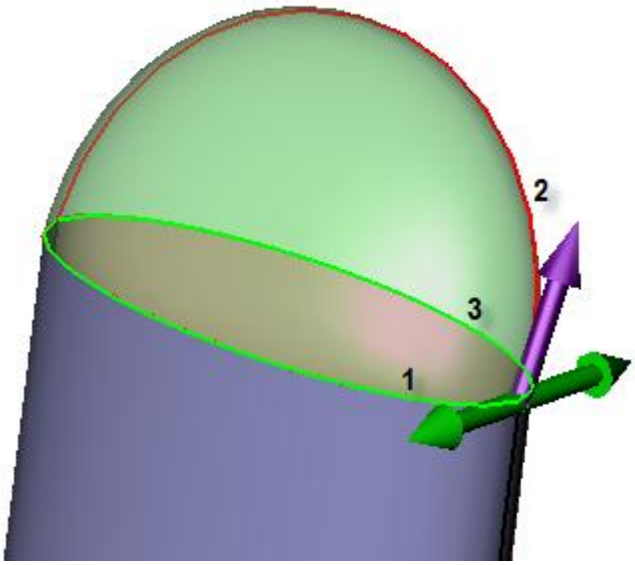
- Pick the arc as the entity to move. (Tip: Check the pick filter)
- Pick the right-hand side of the previously created line, as the direction to move. Other options are found under the right mouse button.
- Enter **0** for the distance amount.
- Enter **20** for the rotation angle.
- Erase the line.

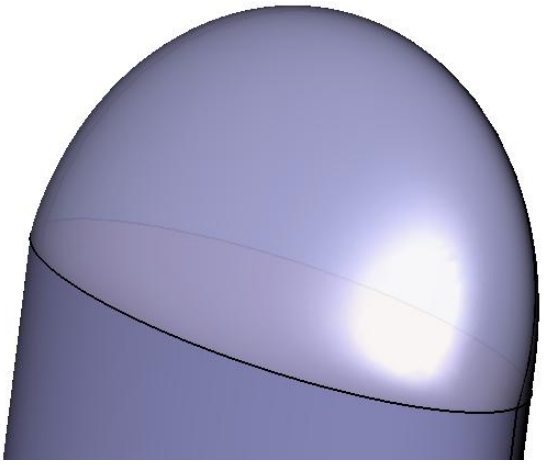


The next step is to create a loft face through the two open edges and the arc. We need to create the loft with tangent continuity with respect to the edges, and to "weight" the continuity differently for each edge to create the shape we need.

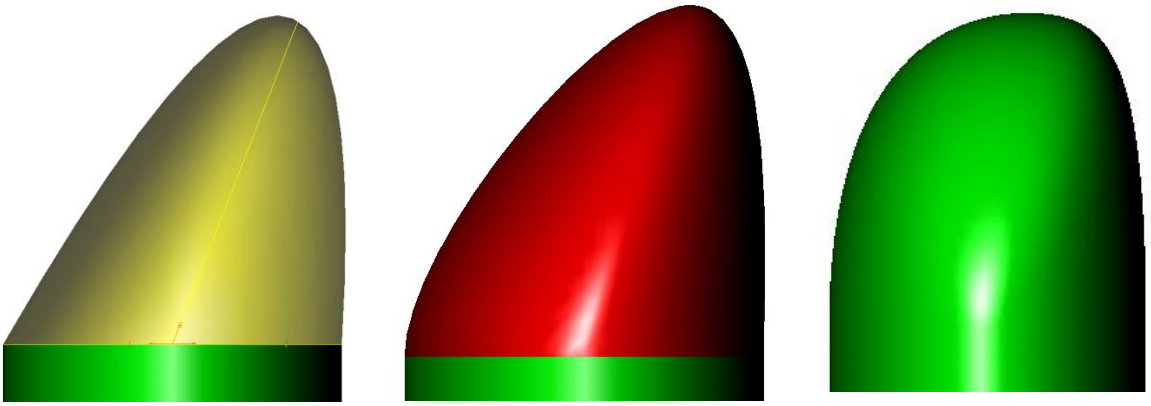
- Press **Crtl+A** to set up the same view as before.
- Select the "**Loft > Base**" command. 
- Pick Filter should be set to "All"
- Pick the nearest open edge towards the right side end point.
- Next pick the arc, again towards the right side end point.
- Next pick the furthest open edge, once again towards the right side end point.
- The pick points are shown in the upper figure opposite.
- Press the middle mouse button to terminate edge/curve selection.
- The graphical preview of the loft should now be as show in the lower figure opposite.

- Note that if this loft were accepted, the continuity would be tangent.
- There are three levels of continuity that can be applied to lofts, they are
- **None** (not tangent).
- **Tangent**, to enforce the position of the edge and tangency to the face at the edge.
- **Curvature**, to enforce the position of the edge, tangency to the face at the edge and curvature across the face.
- We wish to attach the default tangency continuity attributes.



- So that we can see what the other continuity settings give us, pick the pulldown arrow next to Continuity – Tangent and select **None**.
- See the preview image below left, rotated to the right orientation.
- Set it back to **tangent**
- Click on the Start tab and slide the **weight** indicator to about 0.5
- Click on the End tab and slide the weight indicator to about 0.4
- Pick **OK**.
- Erase the arc

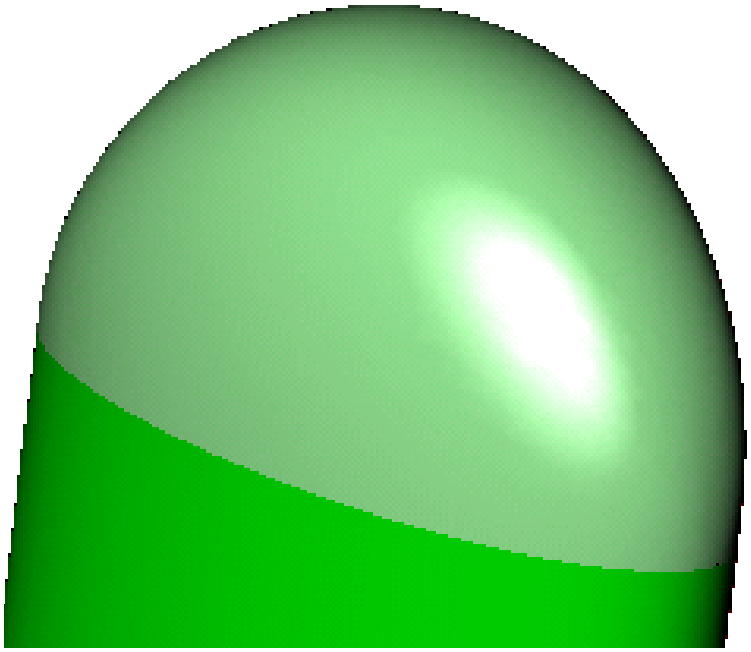


If we had picked the **Add** option the cap geometry would have automatically sewn to the bottle and the surface normals would have matched the bottle. We would then have one shape. Our goal however is to separate the bottle and cap into separate objects.

Now that we have the bottle and cap shape we need to make them both solid shapes.

- Blank the cap shape. Notice the bottle is open. The quickest way to put a flat surface between the two open edges is to use the Fill gaps command from the HEAL tool bar.
- Now swap entities and do the same thing to the cap..If you want, change the Face attributes of the cap as shown below.
- Unblank all.

Now we are ready to separate the two closed shapes into two separate objects.



## EDIT > Copy > Geometry to Part

Now that we have our main bottle and cap shape defined, we need to copy the bottle and the cap into two different parts. To accomplish this we will use the "**EDIT > Copy > Geometry to Part**" command. First however, here is an explanation of the command taken from ZW3D's HELP Manual.

Use this command to copy geometry from the active part to a destination part (new or existing). If a new part is initialized, you are asked whether you want the geometry instanced into the part as a component.

An "import" feature is logged in the destination part that "pulls" user-selected geometry from the active part if you use the "Backup" option. Future regenerations of the destination part will re-import the geometry. If the destination part is new, its default geometry tolerance is changed to match the geometry tolerance of the source part.

Let's copy the cap and the bottle geometry into two parts.

- From the **Edit** pull-down menu **> Copy** suite of tools select "**Geometry to part**"
- Pick the cap shape.
- Next you are asked to specify a file and part name.
- We are going to create a new part in our current file, so the file name is not needed.
- Enter the name "**02_Shampoo_Cap**" for our new part.
- Make sure that Delete original entities and Extract history is not selected.
- For more information, click on the  icon.
- We do not want to instance the exported geometry back into the active part, so do not check the option to "Extract as a component".
- Repeat the above method for the bottle and call the new part "**03_Shampoo_Bottle**".

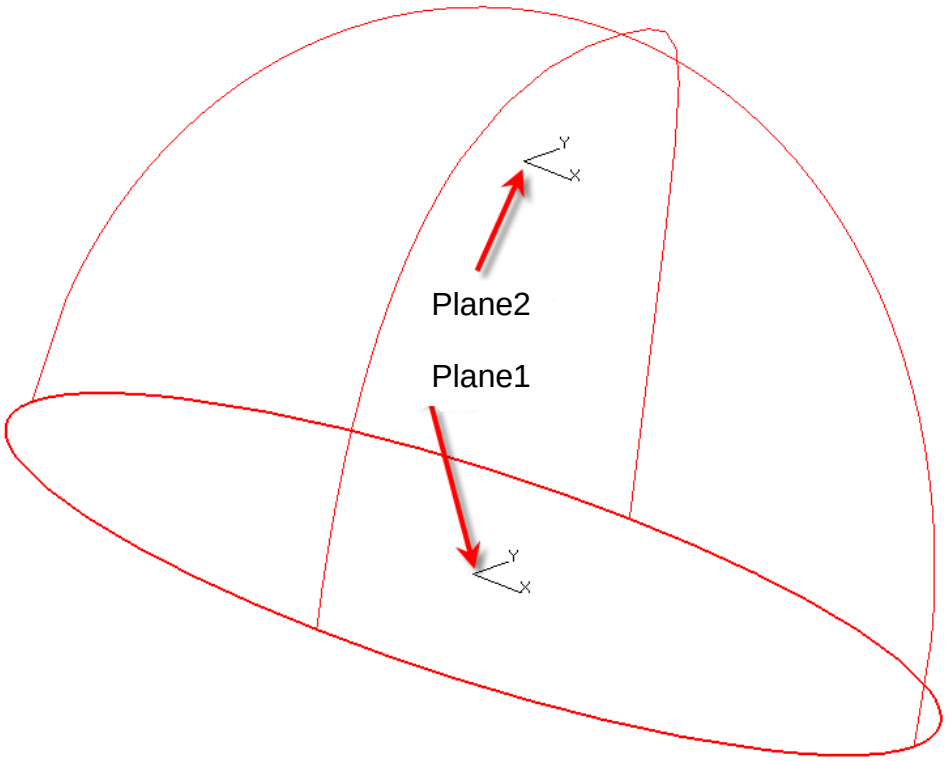
## Example two, Shampoo Cap.

Now that we have our main controlling part shape designed, we can continue with the design of the Bottle and Cap parts.

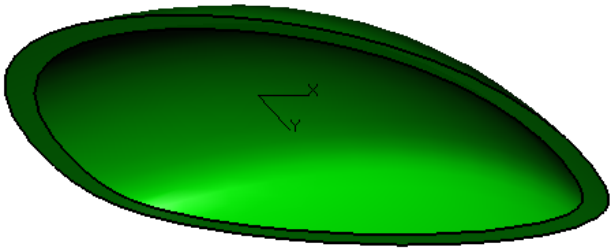
-  and at the object level, edit the part "**02_Shampoo_Cap**".

Notice the encapsulated part history does not include the three default datums. But if you zoom out you will see them.

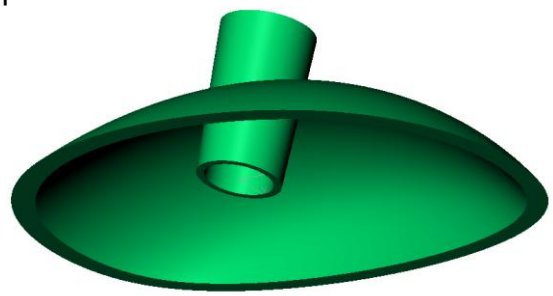
- Create a new plane offset from the "XY-Plane", **for the distance use the right mouse button option "To point"** and select an "edge" end point on the planar face of the cap.
- Next create another plane parallel to the just created plane, offset **25**mm in the **plus Z** direction, as shown below.
- From the **Combine Shapes** suite of commands, use the **Divide shapes** command  , to split the cap into two separate shapes, as shown opposite.
- We will refer to the bottom shape as "cap".
- We will refer to the top shape as the "lid"



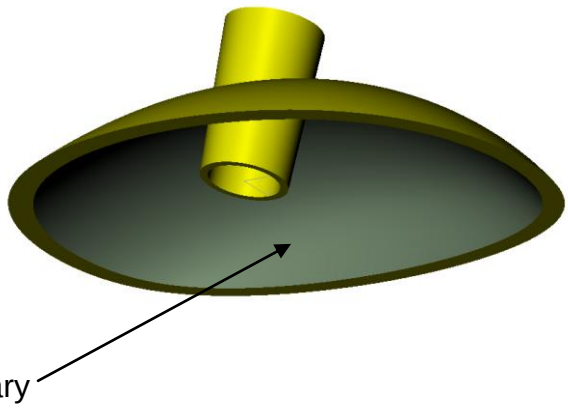
- **Blank** the lower half of the cap.
- **Shell** the top half inwards (minus) -**1**mm, select the planar face as the face to be left open, as shown opposite. 

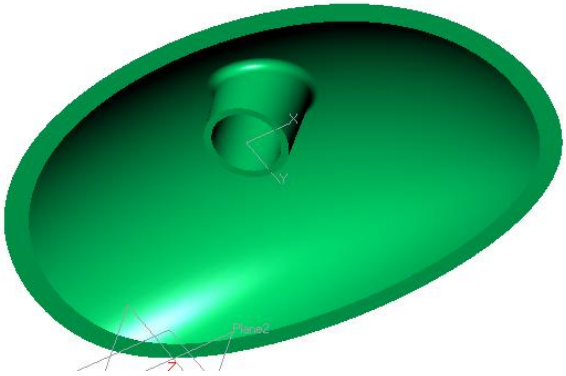


- We are now going to attach a boss to the top half of the cap.
- Select **Extruded Base.** 
- The sketch plane is the same plane that we used to divide the cap.
- In the sketcher create two circles at zero, diameter one = **4.5**mm, diameter two = **5.5**mm.
- Extrude the shape starting at zero and finishing at **20**mm, with a **draft** angle of **2** degrees, as shown opposite.



- We want the boss to join to the lid on the internal face only, and **not** to protrude through.
- We will use the **Combine shape > add** command with the boundary face option
- Pick the lid as the base shape, and the boss as the shape to be added.
- Pick the internal face (shown opposite in grey) as the bounding face.
- Next create a **0.5**mm fillet as shown opposite.

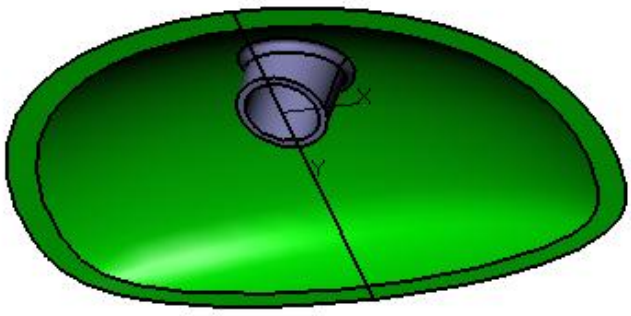




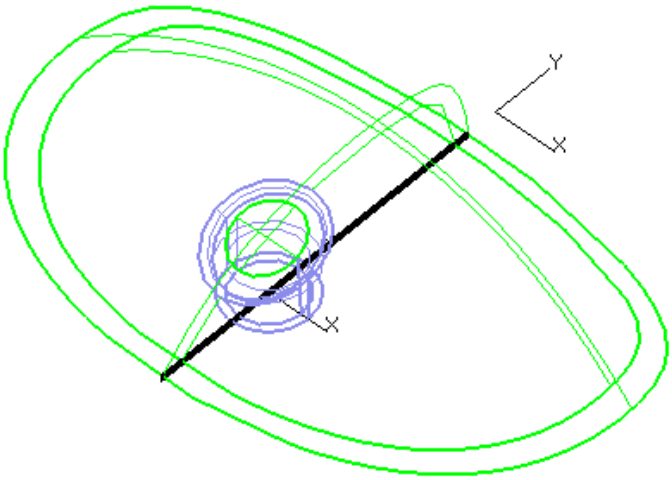
Next we need to create some geometry to enable us to rotate the lid 180 degrees.

From the wire-frame tab, use the "**Intersect Curves**" command  to create a curve between the planar top face of the cap and the plane on the YZ-axis, as follows. At this stage the cap is still blanked.

- Make sure you pick the correct geometry, use the pick filter as necessary.
- Pick the command.
- Pick the **YZ**-Plane for the first Shape.
- The second face set is the top planar face of the **cap**, which is blanked at the moment.
- Pick the "Swap entity visibility" button, to toggle the visibility of the lid and the cap.
- Pick the top face (pick filter) of the cap.
- Blank the line (pick filter).
- Pick the "Swap entity visibility" button once again, to toggle the visibility of the lid and the cap.
- The curve is shown below.

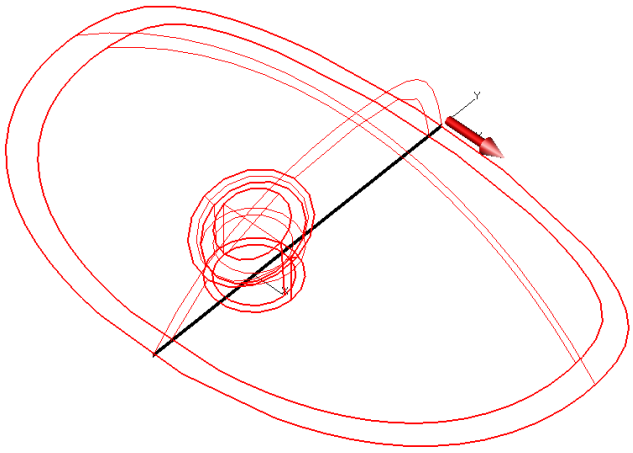


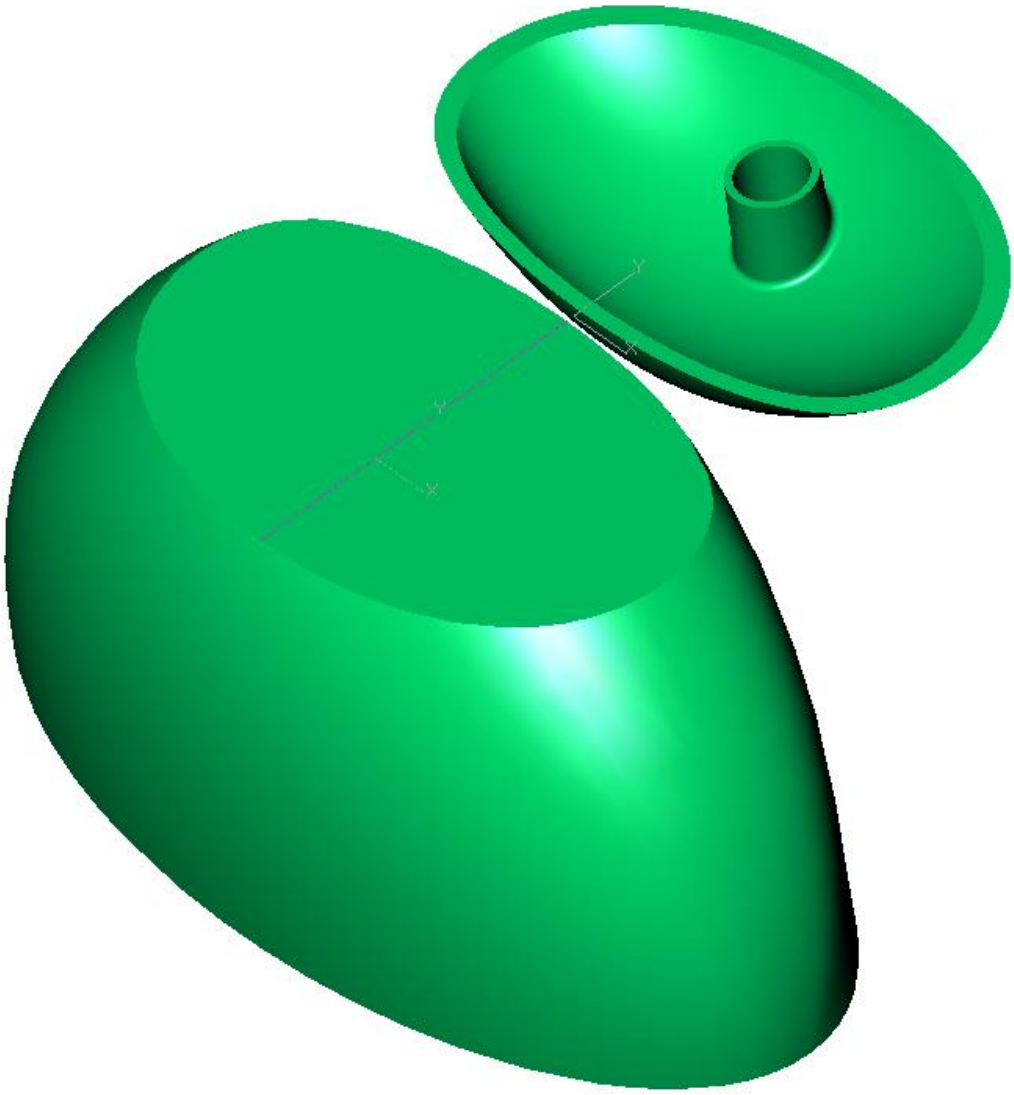
- Next we will create a reference plane, 0.5mm away from the end point of the newly created line.
- First select the "Insert Datum" command.
- Pick the **XY**-Plane.
- The origin is offset from the far end point 0.5 Right mouse Critical Right mouse Offset. See the menu below.
- This places the new plane precisely 0.5 in plus "Y" from the end point of the line.




We will now rotate the lid 180 degrees around the X-axis of the newly created plane.

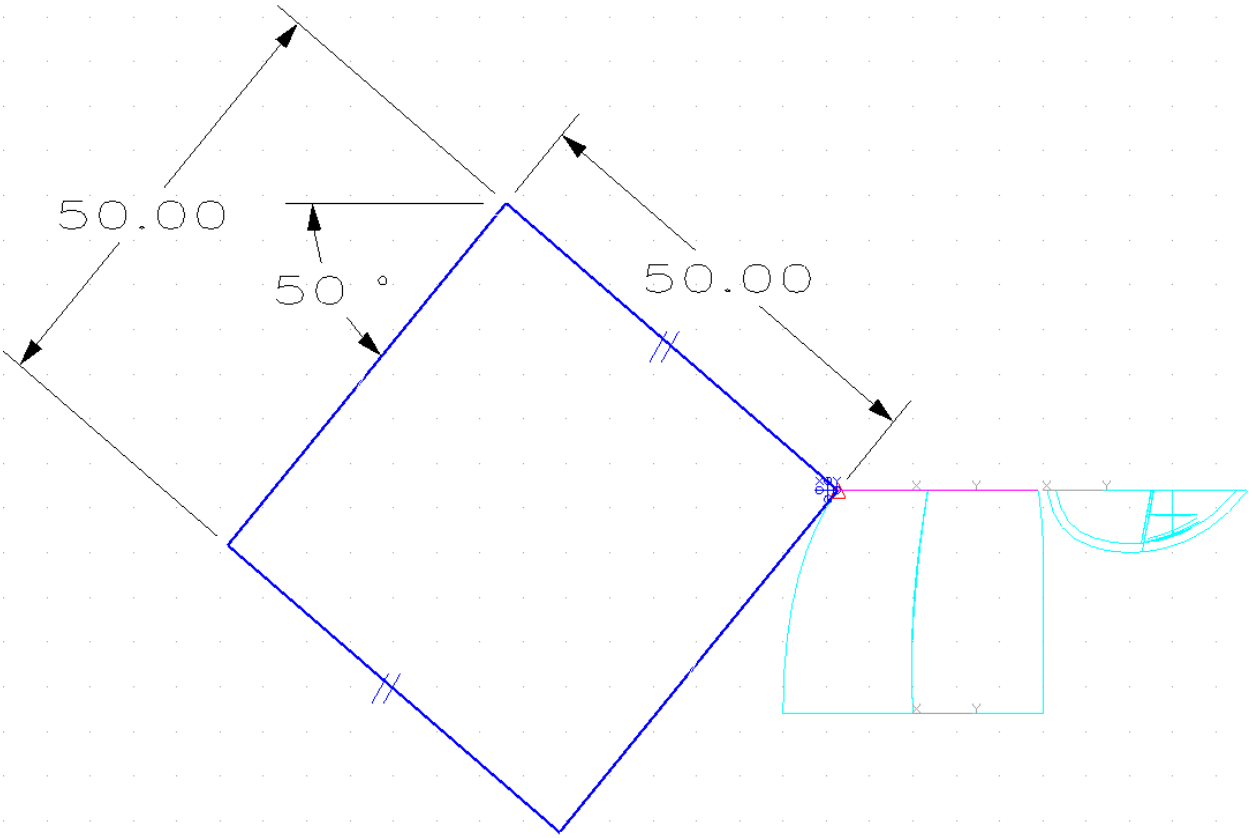
- Select “Move along a direction”
- Pick the lid shape as the entity to move.
- The move direction is the X-axis of the new plane. See the image.
- Enter 0 for the distance amount.
- Enter 180 for the rotation angle.
- Unblank the cap.





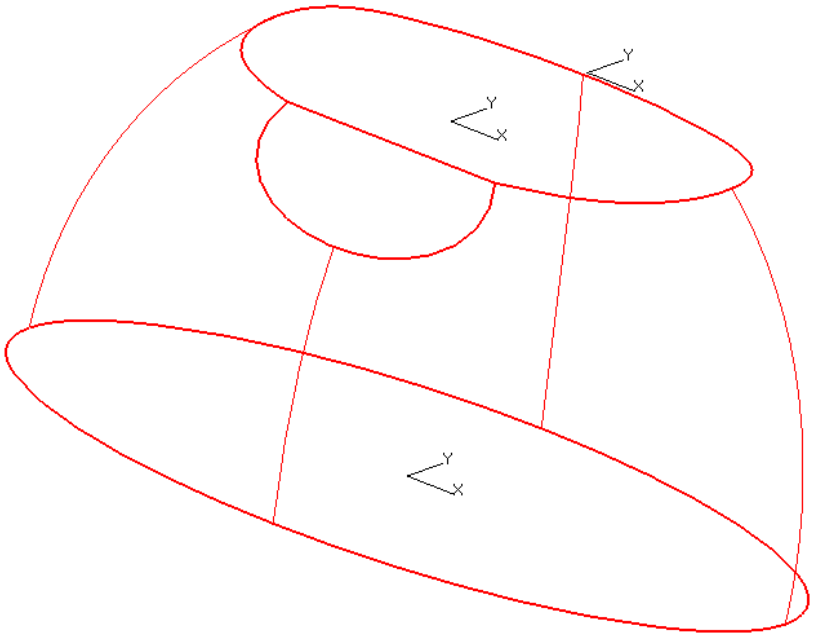
We will now continue work on the cap, so if you wish you can blank the lid.
Next we will create an angled cut on the curved front face of the cap.

- Select the "**Extrude** > **Remove**" command.
- Pick the YZ-Plane , as the sketch plane.
- Create the sketch shown below.
- Note the reference point (the red triangle), is to the end point of our previously defined line.
- If the "Create rectangle (three points)" option is used to create the box and the user places the box corner through the reference point, then the constraint will be added automatically.

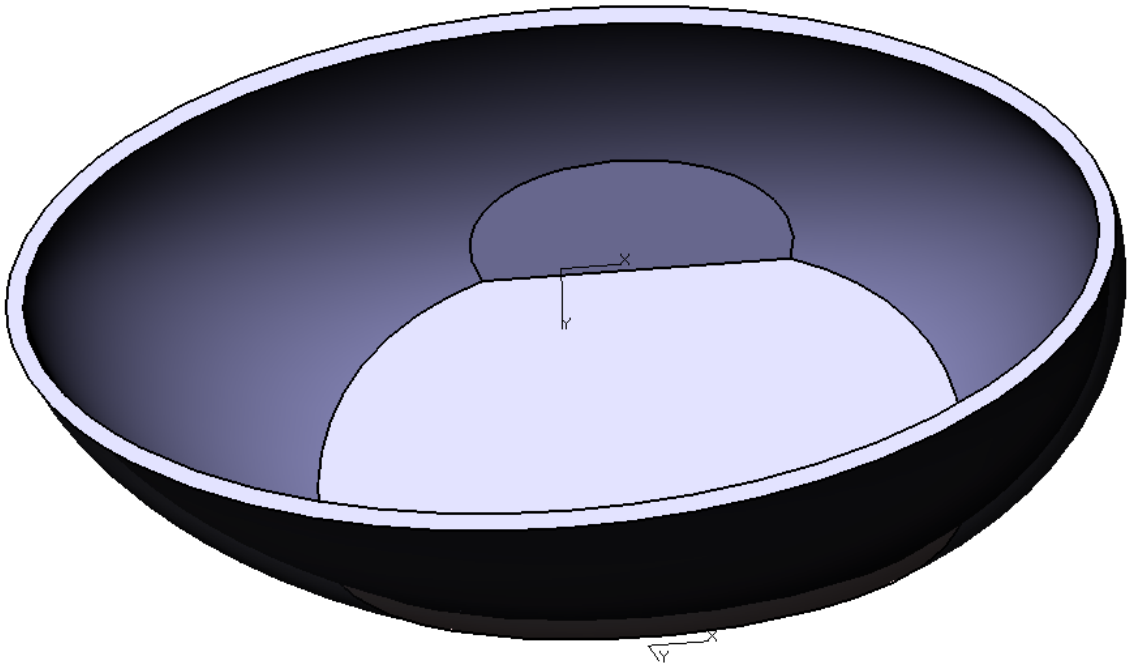



Exit the sketcher.

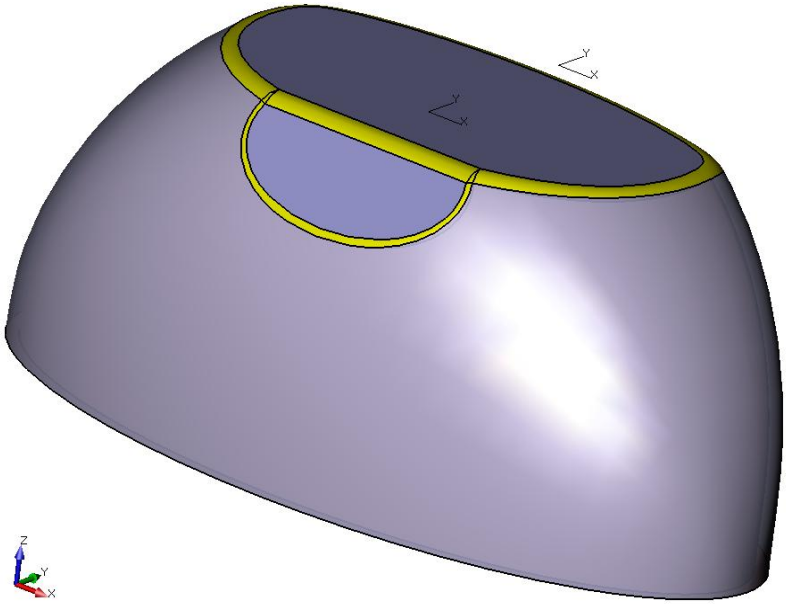
- Enter 50 and –50 for the start and end of the extrude > remove.
- From the options buttons, select the offset option.
- Select “Shrink/Expand”.
- Enter 1 for the offset value.
- Complete the extrude > remove.
- Erase the wireframe line.
- The cap should now look as shown below.



Shell the cap inwards (minus) -1mm, with the bottom face open, as below.



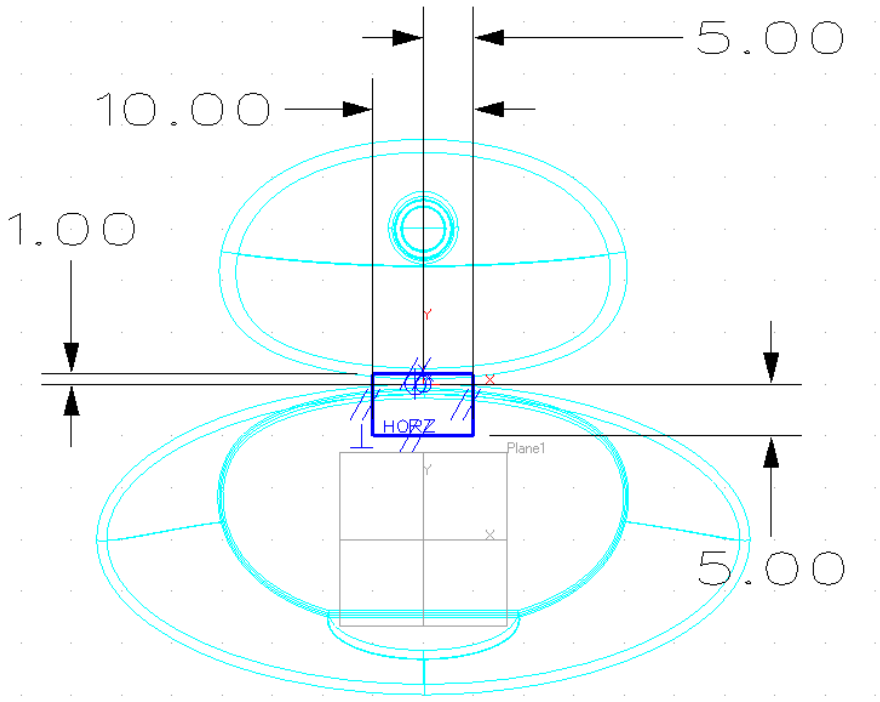
Next fillet the outside edges (shown below in yellow) .5mm.
(Illustration shows fillets larger for clarity)

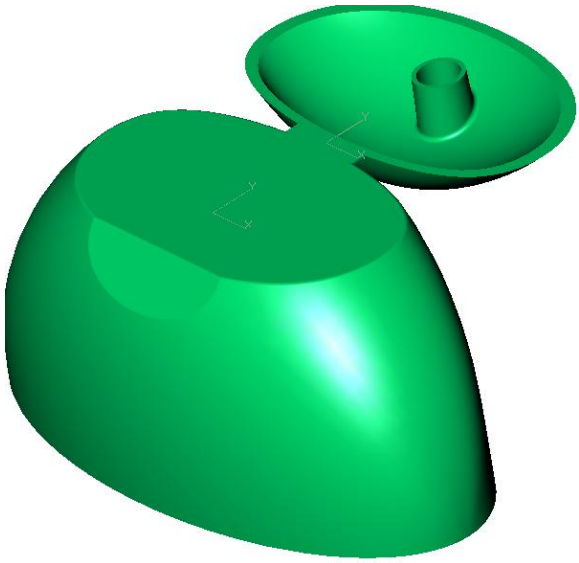


The next step is to join the lid and the cap together with the extrude > add command. The resultant hinge is clearly an approximation of the true hinge shape that would be required.
Time permitting and if you wish, you may decide to construct the correct hinge shape.

- For our example however, first unblank the lid.

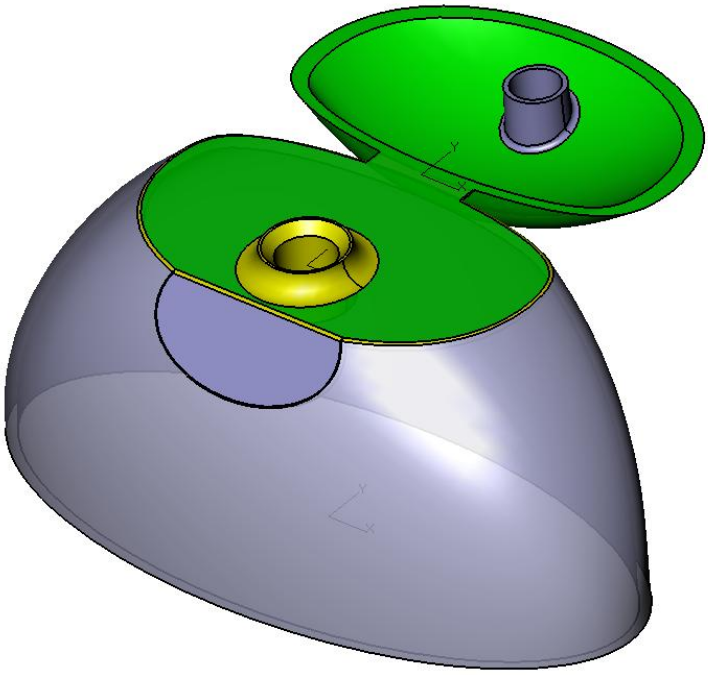
- Select the **Extrude** **> Add**, command, and create the boss shown below.
- The sketch plane is the same plane that we used to rotate the lid.
- The size of the boss is shown below 10mm x 5mm, and -0.2mm thick.





Continue with the cap design as follows.

- Select the "**Extrude** **> Add** " command.
- Pick the datum plane on the top face of the cap as shown below, for the sketch plane.
- In the sketcher create a circle of diameter 8mm at the zero position of the sketch plane.
- Extrude the boss +2mm and –3mm.
- Next create a round extrude > remove of diameter 5.5mm, through the newly create boss.
- Next create a 1mm chamfer on the edge of the hole, as shown below.
- Finally create a 2mm-radius fillet on the boss, as shown below.



Continue with the cap design.

- **Select the "Extrude > Base" command. (not ADD)**
- Use the plane parallel to the lower planar face of the cap (not the top face), as the sketch plane.
- In the sketcher create two circles with origin zero, and the diameter 14mm and 15mm.

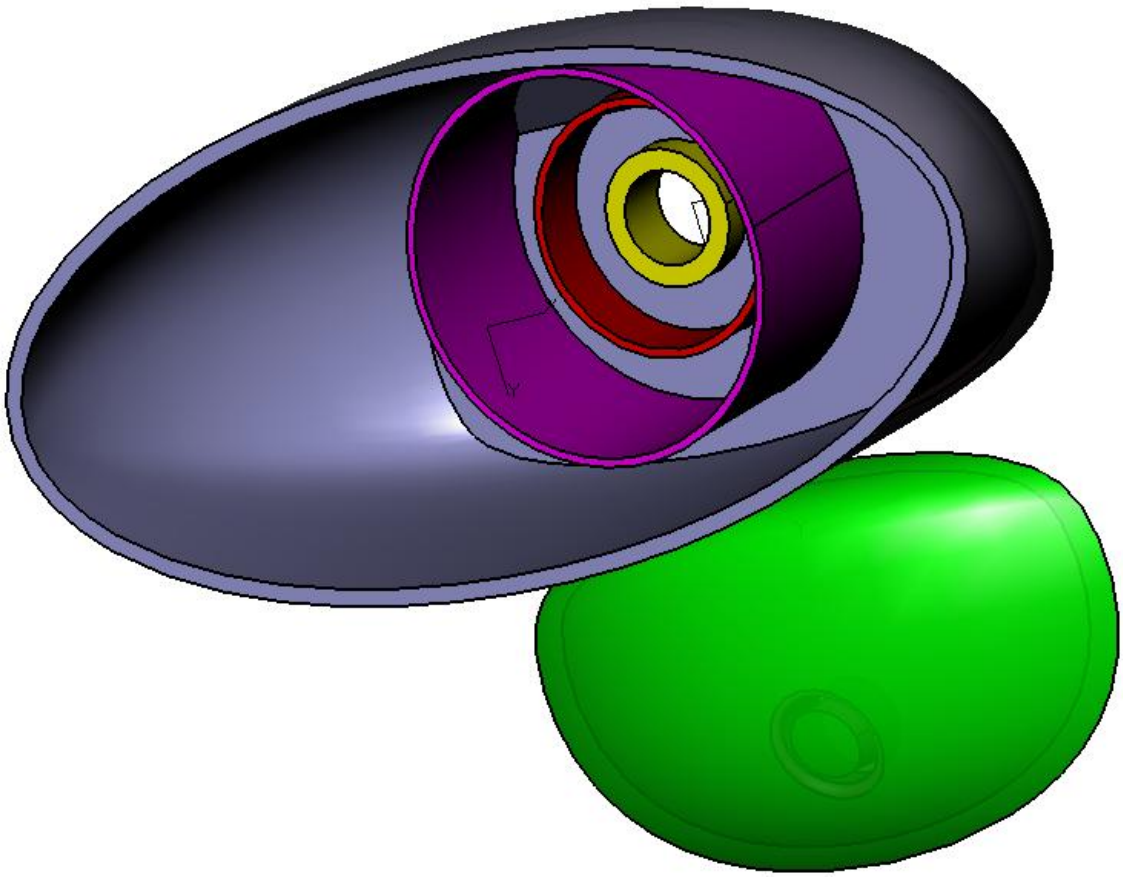
- Exit the sketcher.

- Extrude the solid, start 20mm end 40mm.

Using the **Combine Shapes > Add** command, join the new solid to the cap on the inside only.

- The above add command must be used with the "**Bounded face**" option.

- Repeat the above method to create and attach a second solid to the cap using the bounded Boolean option.
- Once again this consists of two circles in the sketcher of diameter 22mm and 23mm.
- The start height of the extruded solid is 12mm; the end height is 40mm.
- The cap and the two "Bosses" are shown below, with the bosses shown in a different color for clarity.

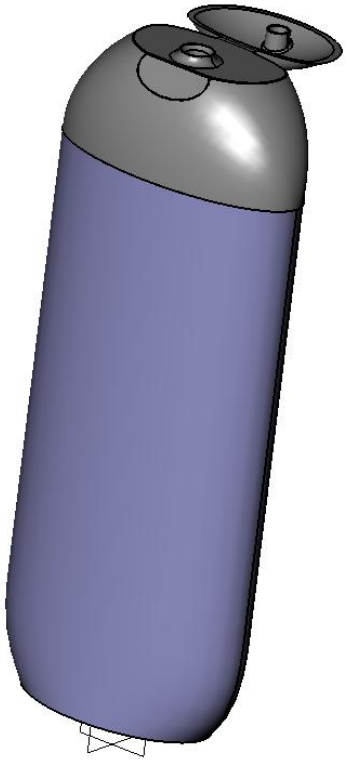


## Creating an assembly to design the rest of the Bottle.

We will now continue with the design of the bottle. We need to reference some of the cap geometry in the bottle, to ensure that everything fits together nicely. To accomplish this we will use assembly modeling.

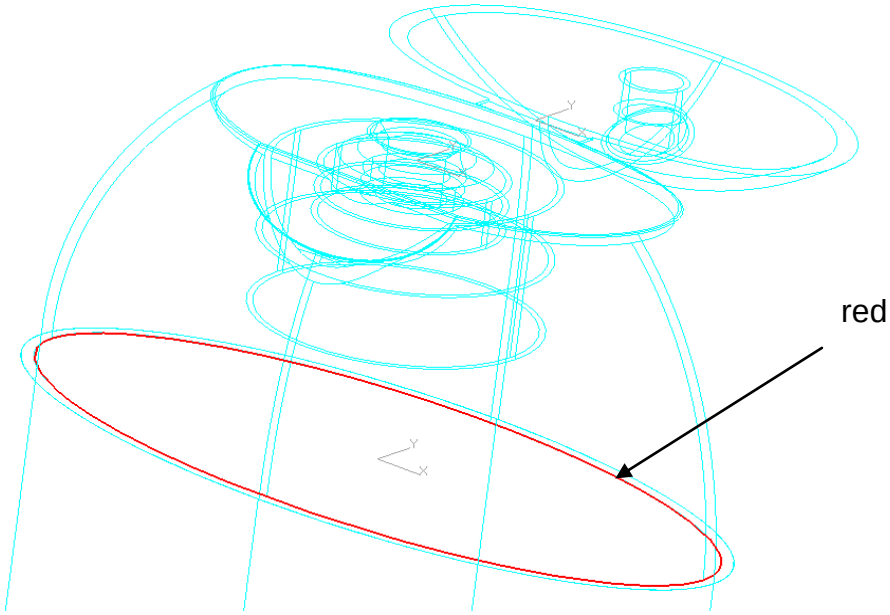

Exit the “02_Shampoo_Cap” part, up to the object level, and create a new part called “**04_Shampoo_Bottle_Assembly**”. Then instance the two components as follows.

- Place the mouse pointer in the graphics area of the screen and with the right mouse button select “Insert Component”.
- Select the “02_Shampoo_Cap” part from the object browser.
- We need to bring in the cap into the assembly first, as this will be used to create geometry that will later be used in the bottle part. Loading the cap part first ensures we keep a “Parent/Child” relationship between the parts when we need to regenerate the assembly (and its components). Notice the Regen option button. This will be explained later.
- Place it at (0,0,0).
- We have now instanced our cap part into our current assembly.
- Repeat the above method to instance our second component, the 03_Shampoo_Bottle.
- Place it at (0,0,0).



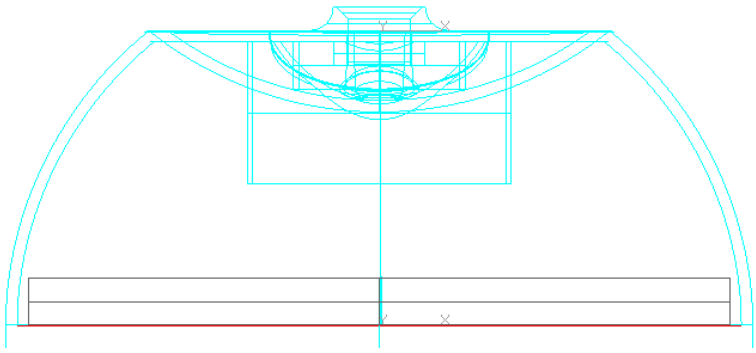
## Edit the bottle part.

Place the mouse pointer over the bottle component and right click > Edit part to edit the bottle part.
Notice the top banner in ZW3D shows you that you have entered the bottle part.
We are now going to create an extruded add shape using geometry from the cap component.
Create two reference curves from the inside edges of the cap as shown in red below.




- Use the extrude > add command to create the boss.
- When you are asked to pick sketch, press the right mouse button and pick “Insert Curve List”.
- Pick the two reference curves.
- The start of the extrude > add is 0, and the end 4 in the Plus “Z” direction.
- Use the offset options to shrink the boss in (minus) -1mm.

As shown in black opposite

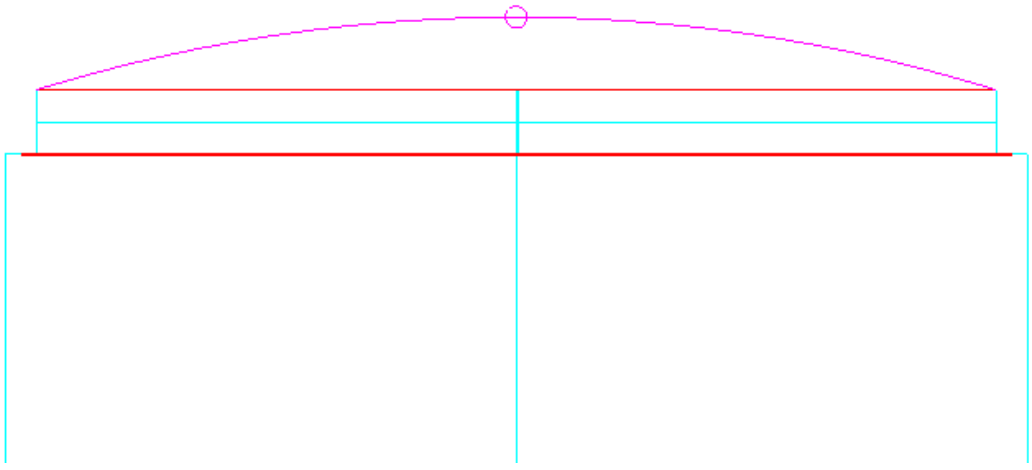


## Change the top face of the part.

Use the “show target” button to display the bottle part only. This is a convenient way of displaying the bottle without having to see the cap part.

The current planar top face of our bottle needs a more pleasing shape. We will accomplish this by using a 3pt. Arc and a lofted face.

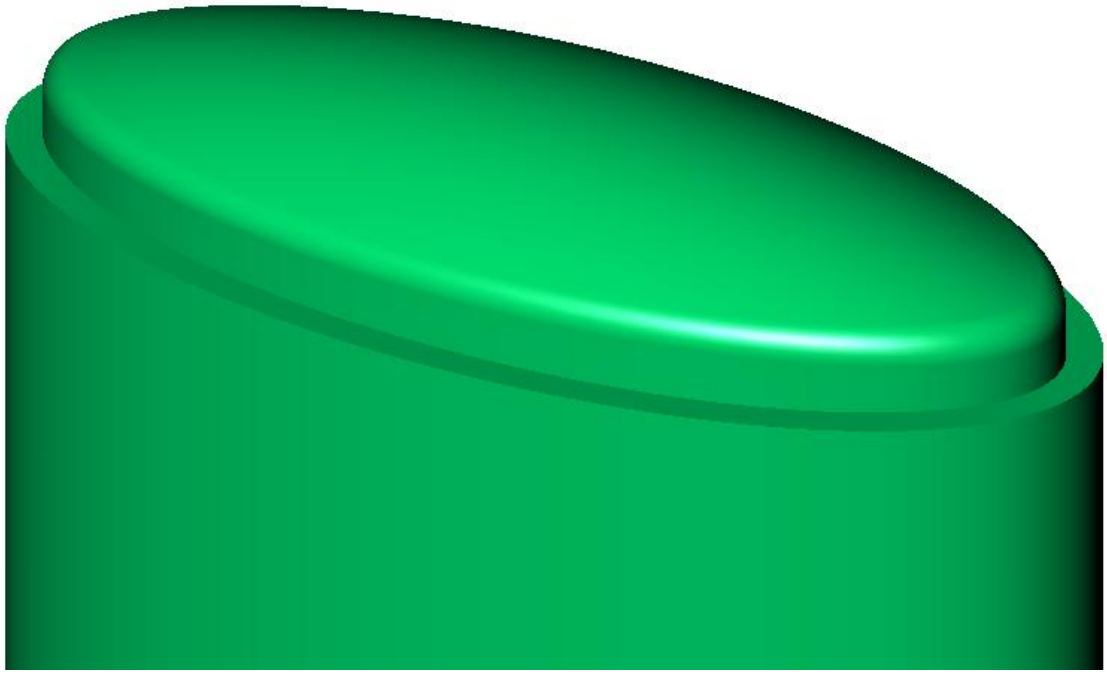
- First, let’s remove the unwanted top face. We could also join the soon to be created face with the bottle and the existing planar face would “disappear”. Or we could also use the replace face command to replace the planar face with the curved loft face.
- Pick the erase command, and with the pick filter set to face, erase the top planar face of the bottle.
- Now let’s create the arc middle circumference point. This point needs to be offset from a point halfway between the two open edges (the middle of the “hole”).
- Select the “Point” command from the Wireframe Tab
- Press the right mouse button again, and select offset.
- Press the right mouse button again, and select between.
- Pick the two end points of one of the open edges.
- Accept the default percentage distance (50).
- Select the **XY** plane for the reference plane.
- Enter zero (0) for both the X and the Y values, and 4.5 for the Z offset value.
- Now we can create a 3pt. Arc from the both end points of one of the open edges, with the previously created point as the middle or circumference point of the arc.
-



## Creating and adding the Loft face to the bottle.

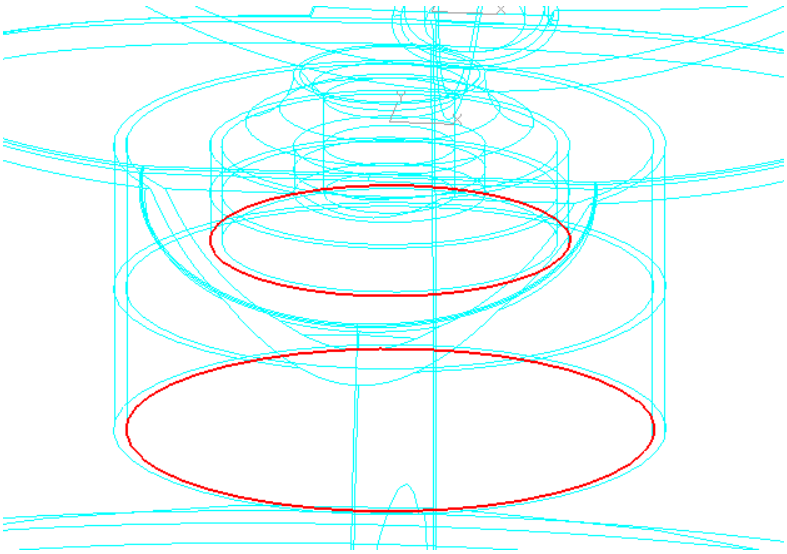
Use the "**Loft**" command to create a lofted face from the top open edges of the bottle and the arc, as follows.

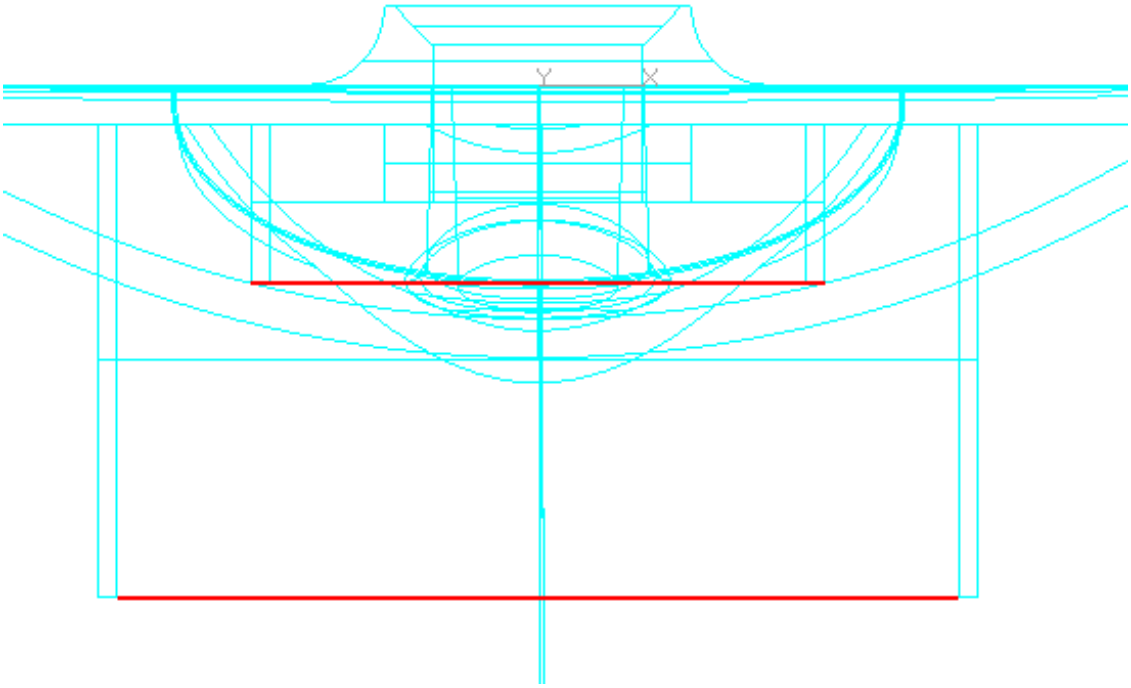
- Select the **"Loft**  **> Base**  **"** command.
- With the pick filter set to all, pick one of the open edges (blue dotted) of the bottle near the end point of either the left or the right side.
- Pick the arc near the same end point of the previous pick.
- Pick the other open edge, again with the pick point near the end point of the previous picks.
- It is important to pick near the same end points, or the loft will twist.
- Change the continuity flag to "None"
- As stated previously you could have chosen either the left or right end points.
- If you chose the left points, the lofted face would have its normal vector pointing outwards, and because "auto sew" is on, the loft automatically sews to the bottle to make a valid solid.
- If however you chose the end points on the right, the face normal are pointing inwards, depicted by the pink (Back) face color.
- You can simply "Sew" and the normals will adjust.
- Or you could use the "Reverse Normals" command to reverse or flip the face.
- Then you would have had to join the face to the bottle using either the "Combine Shapes > add" command or the "Sew" command.
- Erase the arc and the point and fillet the edge 2mm as shown below.



## Create the bottle neck.

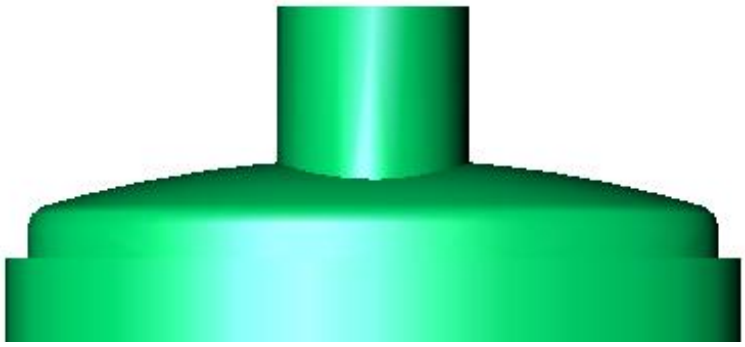
- Toggle the visibility of the cap component using the "**Show All**" command. 
- Create two reference curves from the cap geometry, as shown below in red.





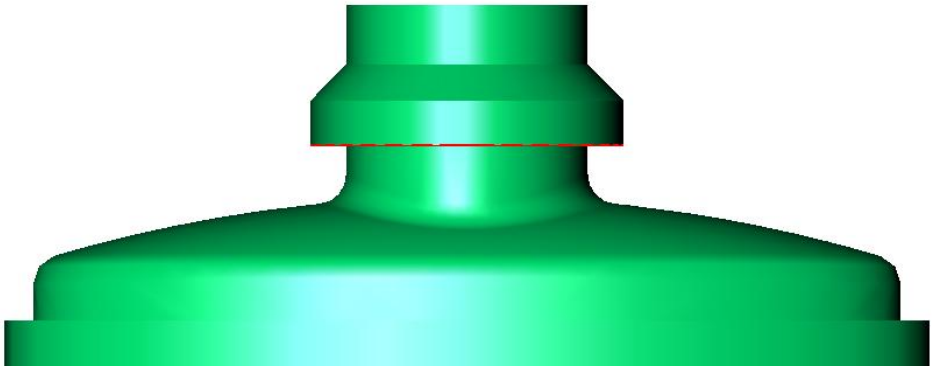
Use the “Extrude > Add” command, to extrude a boss for the neck shape as follows.

- Select the “**Extrude** **> Add**” command.
- With the right mouse button, pick “**Insert Curve List**”.
- Pick the smaller diameter previously defined reference curve.
- For the start, enter **3**mm, for the end, enter **–50**mm.
- Use the option button “**offset**” and pick the **shrink/expand** option, to apply **1**mm offset.
- The above described, boss is shown below.



Next using the “Extrude > Base” command, we will create the next shape that makes up the bottle neck as follows.

- Select the “**Extrude** **> Base**” (not add) command.
- With the right mouse button, pick “**Insert Curve List**”.
- Pick the larger diameter previously defined reference curve.
- For the start, enter **0**, for the end, enter **–6**mm
- This time do not apply any offset.
- Now create a **3**mm **chamfer** on the top edges of the previously created solid.
- Use the “**Combine shape** **> Add**” command to add the solid to the bottle.
- Create a **2**mm-radius **fillet** to the edge between the neck and the bottle.
- As shown below.



**Next shell the bottle and add some fillets as follows.**

- Use the "**Shell**" command  to shell the bottle (minus) **-1**mm inwards. Pick the top face of the neck as the open face.
- Create **0.5**mm-radius **fillets** using chain select on the edges shown below. The fillets are shown colored blue for clarification. 
-

- Pick “**Show All**” and then **exit the part** bottle.

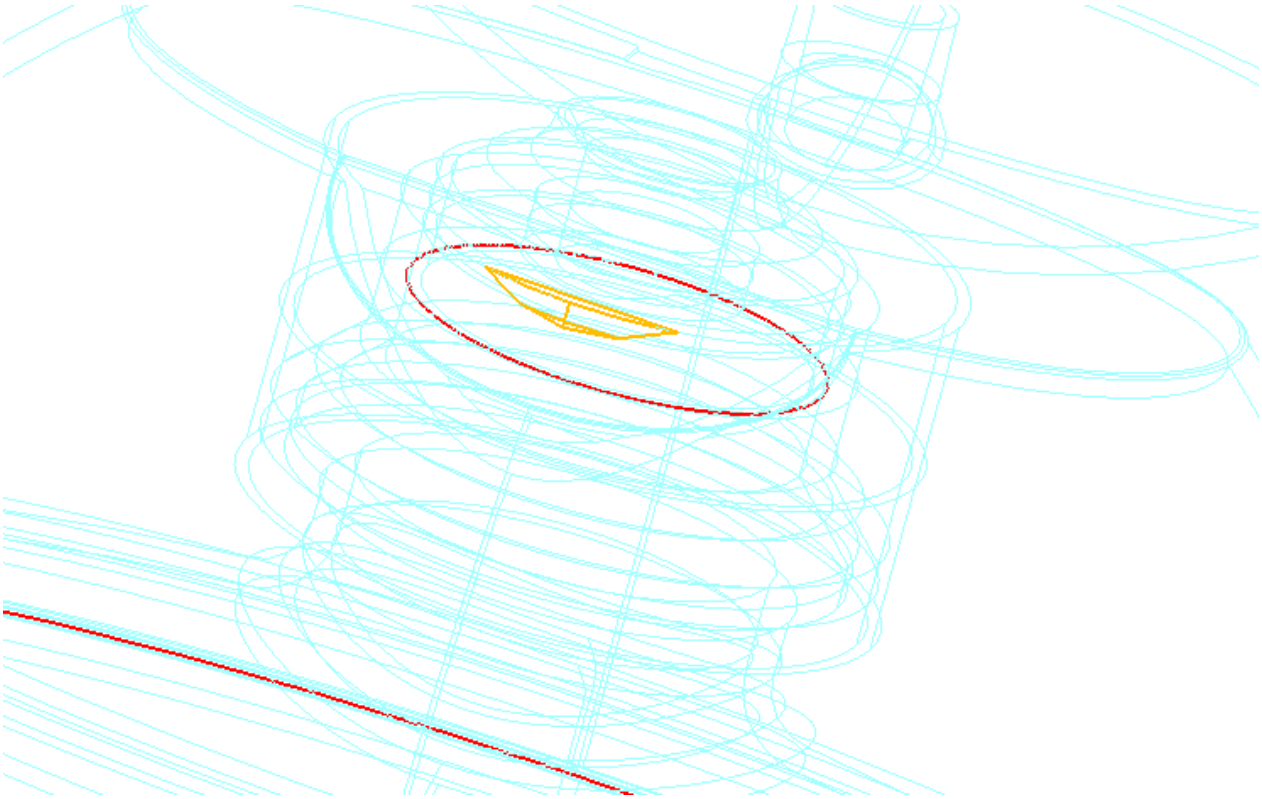
This will take us back up to the Bottle Assembly top tree level, where the cap and the bottle are components. The cap and the bottle are clearly not completely designed parts, but hopefully you will have found some of the techniques used useful.

## Interference Checking.

There is in fact an error on our bottle part. Part of the bottle neck is interfering with the cap.

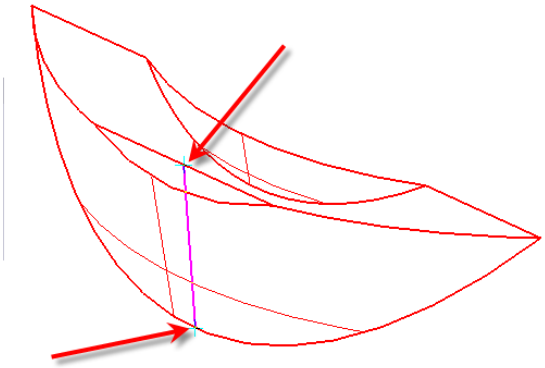
From the assembly tab perform an interference check between the components.

- The message shown opposite, and the Shape shown below in orange, indicates that there is interference between the neck and the cap.



## Edit the bottle to remove the interference problem.

- First thing is to figure out how much interference you have. Everyone's part is going to be slightly different because of the way we lofted the cap back on page 16, adding weight to influence the shape. (We can shorten the top of the bottle or adjust the loft of the cap)
- I am going to use 2.0mm to adjust the height of . the neck.



- With the **pick filter** set to "**All**", right click > Edit part to **edit the Shampoo Bottle** part.
- Press the "**Show Target**" button to view the bottle only. 
- The height of the neck needs to be reduced slightly. (2.0)
- Double click on the neck "**Extrude**  **> add**  " feature, to display its dimensions.
- Change the value of the 3mm dimension to 1mm.
- The history will automatically regenerate. Make sure that no history operation was suppressed.
- Press the "**Show All**" button to view the cap. 
- Exit part to go back up into the assembly.
- Perform an **interference check** and no interference will be detected.

Save your file Shampoo.Z3**.**

Example two, The Conditioner Bottle.

With this example we will create both the bottle and the cap in one part.
Some of the topics covered include:
Mesh Faces
Loft Faces with and without continuity
Referencing external dimensions in the sketcher
Global sketches
Twisted swept features

## Create the main bottle shape.

With our example the designer has the basic shape of the bottle and cap in mind.

Create a new multi-object file called “**Conditioner.Z3**”.

Next create a part called "**01_Conditioner_Bottle**". This time we are going to create the bottle and the cap in one file.

## Global (or Root) sketches)

With the previous example the main shape of the bottle was elliptical. You were required to create and constrain an ellipse in the sketcher. Let's assume that an elliptical shape may be required on several of your bottle designs. It would be inconvenient to have to create a new constrained ellipse every time you wished to use an elliptical shape in your designs. We could use the "Clipboard" to copy an existing sketch from some model, and then paste it into your new design. However, in this case a much better option is to use "Global or Root" sketches. Global sketches can be saved in Object files (just like parts) and are available to anyone on your systems. Once the global sketch is copied into your part you can change the dimension values, or even modify the sketch geometry, without altering the original global sketch. We have supplied you with a file called Conditioner_Sketches, which contain Global sketches that will speed up the time it takes to go through the Conditioner bottle example. Also in this file is a Global sketch of an ellipse, but in this case an ellipse is so common you could also use Ready Sketch.

## Insert a Global sketch into our new design.

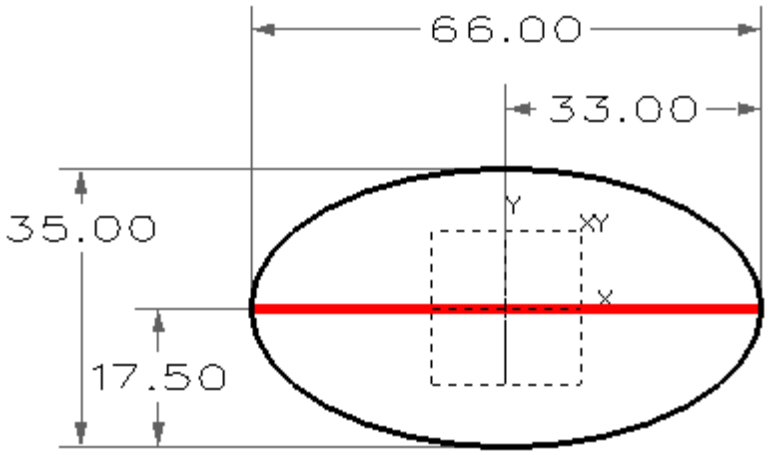
- From the "**Edit**" menu, select **Copy** then **External Sketch**.
- Next from the Objects menu, use the yellow folder icon to browse for the correct file.
- Pick the Conditioner sketches from the Object Files menu.
- Pick **ellipse** from the Objects menu.
- Middle-click on each of the prompts.
- This will place our ellipse onto the default XY-Plane.

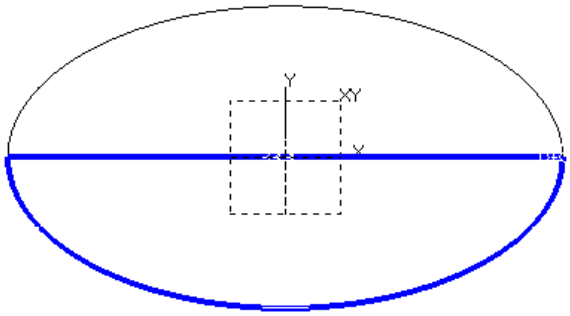
## Edit the sketch.

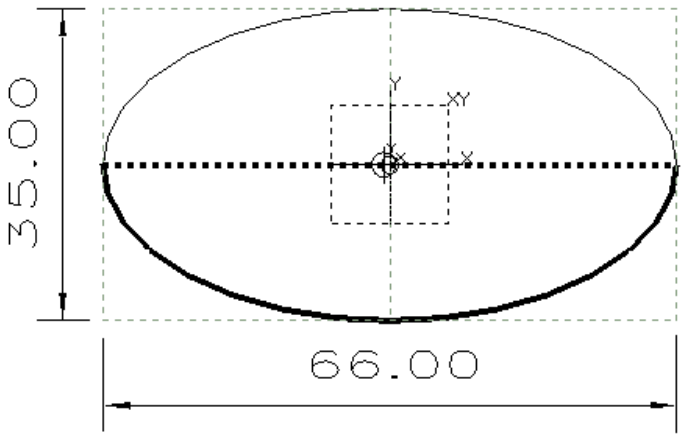
The ellipse has now been placed onto the XY-Plane. But the sizes are not what we want, and also for our design we wish to use one half of the ellipse only.
Place the mouse pointer over the sketch and double click to edit the sketch.

- Change the value of the 60mm dimension to 66mm.
- Change the value of the 140mm dimension to 35mm.
- We wish to use the lower half of the ellipse only.
- Trimming the ellipse would affect the constraints.
- We will use the trace profile command.
- But first create the two point line shown in red opposite.
- Next use the trace profile command to create the profile of the lower half of the ellipse, shown in blue opposite.
- 
- If you used Ready Sketch just left-click to position at (0,0)
- Change the dimension values as instructed above.

- There is already a horizontal line that can be used for the Trace Profile. See the bold geometry opposite. This is much easier for common shapes.



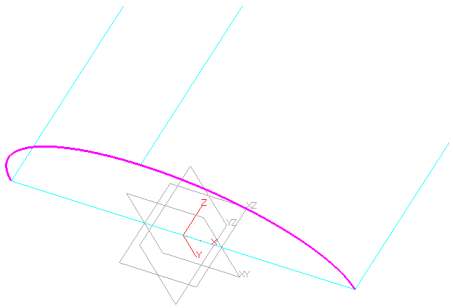

Ready Sketch

## Extrude the sketch.

Use the “Extrude > Base” command to extrude the sketch.

- Start = 0
- End  = 130.

- Use the “Curve from edge” command to create the curve shown opposite in magenta. This curve will be used shortly to create an extruded boss which forms part of the bottle design.

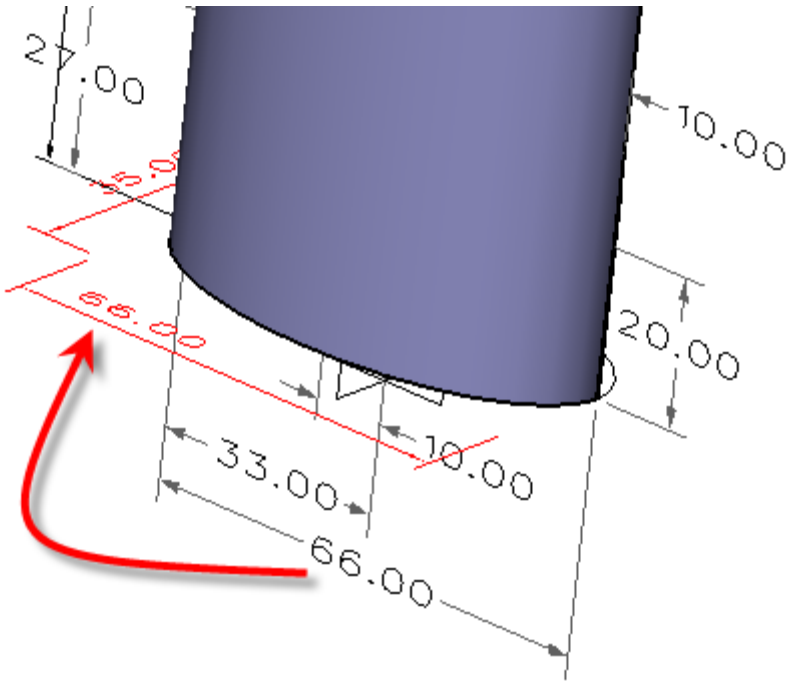


## Insert another “Global” sketch, to cut the bottle.

The bottle design requires that the end of the bottle towards the cap should have a curved shape.
The shape in this case is made up of a number of lines and tangent arcs, which were all drawn using the “Quick Draw” command in the sketcher.
The sketch is available in the “Conditioner sketches” file and is called “Cap Bottle split”.

As previously described insert the global sketch.
This time however place the sketch on the **XZ-Plane.**
If the width of the original elliptical shape changes, you must remember to change the corresponding values in the new sketch also.
Clearly it would be an advantage for the new sketch to update automatically when the elliptical sketch width changes. To accomplish this, please do the following.

- First toggle the existing ellipse sketch dimensions on.
- Next edit the newly inserted sketch.
- We need to associate the 66mm dimension from the active sketch with the 66mm dimension from the ellipse.
- Continued on next page...

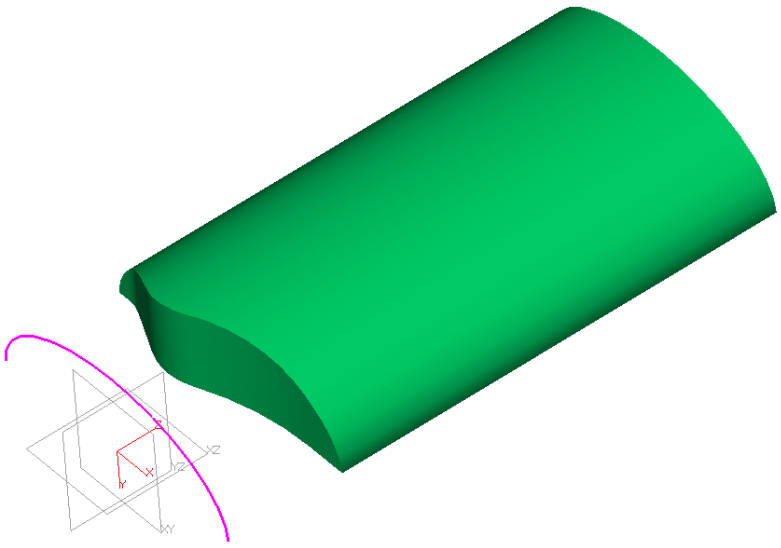



### Link the Dimensions to the first sketch and then cut the part.
**Input Dimension Value:[Conditioner_Bottle:ellipse_d0]**

- Double click on the black 66mm dimension.
- Press the “Select Dimension” button in the Input Dimension Value form.

- Select the red 66mm dimension shown in the figure on the previous page. Note in order to select the dimensions which are from a different sketch, “External dimensions”, you will have to press and hold down the “F7” key.
- Repeat for the black 33mm dimension and “link” it to the red 33mm dimension.
- Note: Both dimensions in the Input Dimension Value form have the prefix “Conditioner Bottle:” attached to them. Please be aware that if you were to copy the part (family of parts for example) into a different part, the “linked” dimension would still point to the relevant dimension in the Conditioner part, and not to the relevant dimension in the copied part. This is the default. If you wish to refer to the dimension in the “local” part, you will need to delete the reference “Conditioner Bottle:” and refer only to the label name of the dimension. For example “ellipse_d0.
- Any changes to the width of the ellipse (from sketch1) will propagate to the dimensions in sketch2.
- 
- Use the “Extrude > remove” command to cut the part as shown opposite.
- Start = 0.

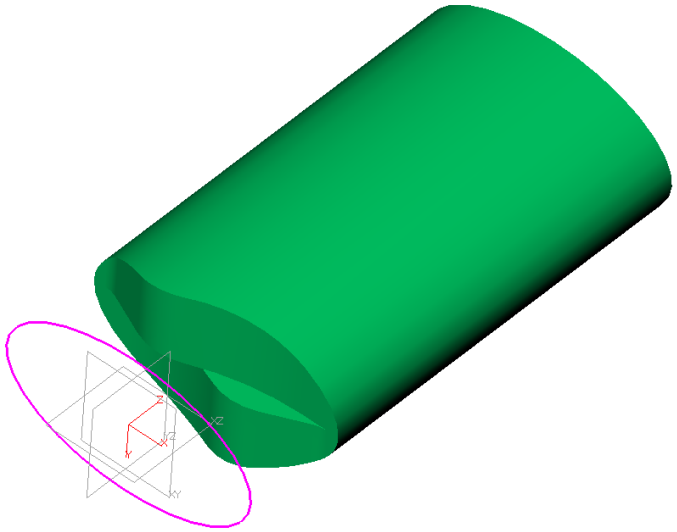
End = 50.



## Copy/rotate the bottle shape and the magenta curve.

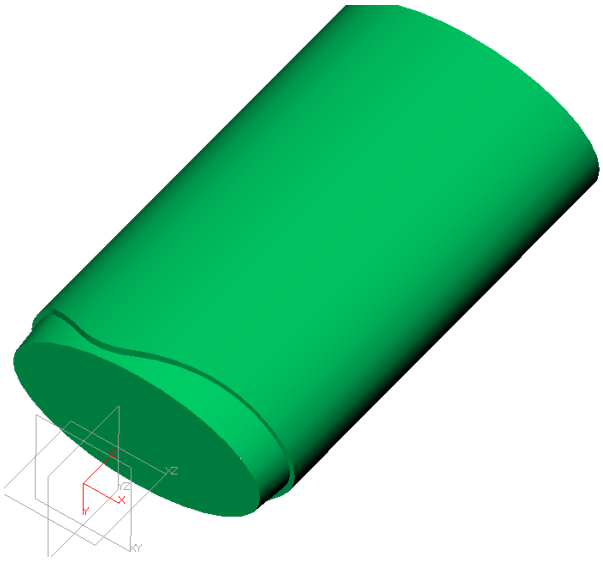
Use the “**Edit > Copy> Copy Entities**”, “ **along a direction**”  , to copy the **shape** and the magenta **curve**, 180 degrees about the Z-axis.

- Select the command.
- Pick the shape and the curve as the entities to copy.
- With the right mouse button select Z-axis as the copy direction.
- Distance to copy =0.
- Rotation angle =180.
- As shown opposite.



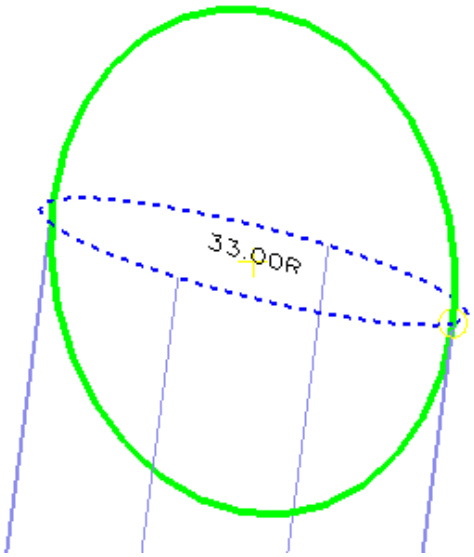
- Pick the "**Extrude** **> Add** " command.
- Right mouse button, select "**Insert Curve List**".
- Pick the two magenta curves shown in the previous figure.
- Turn on the display of dimensions.
- The start value of the extrude > add is **2mm less then the 20mm** dimension from sketch2. Use the right mouse button, select "**expression**" then from the Input Dimension Value form pick **Select Dimension** and select the **20**mm.
- **Append "-2"** to the name in the field. As shown.
- The end value is **1mm more than 27mm** dimension from sketch2.
- Use the same method as before but this time **append "+1"** to the name in the field.
- Draft angle = **5** degrees.
- Offset option "**shrink/expand**" **-2mm** (minus).
- Then erase the two curves.
- Turn off dimension display.

**Create a tapered boss as shown below.**



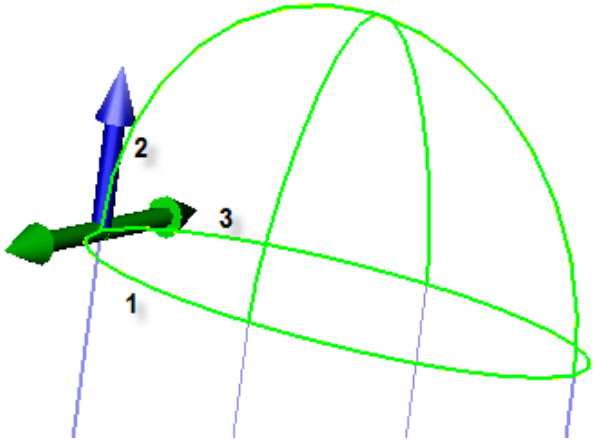
**Create the end wire-frame geometry and tangent loft.**

- Erase the top planar face of the bottle.
- Create a circle (center, boundary).
- The center point is 50% between the two end points of one of the open edges.
- The boundary point or radius is one of those same end points.
- The Align Plane is XZ-Plane.
- Use the “One touch trim” command  to trim the circle as shown below.

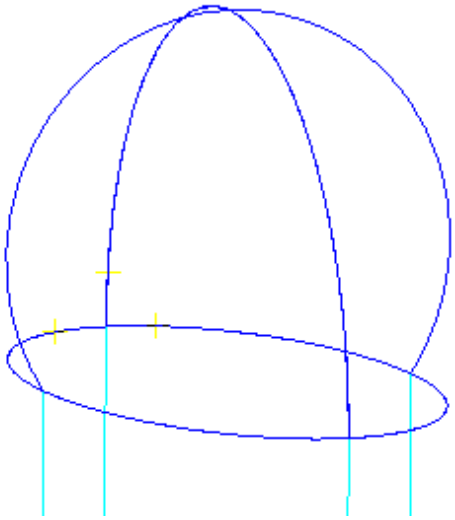


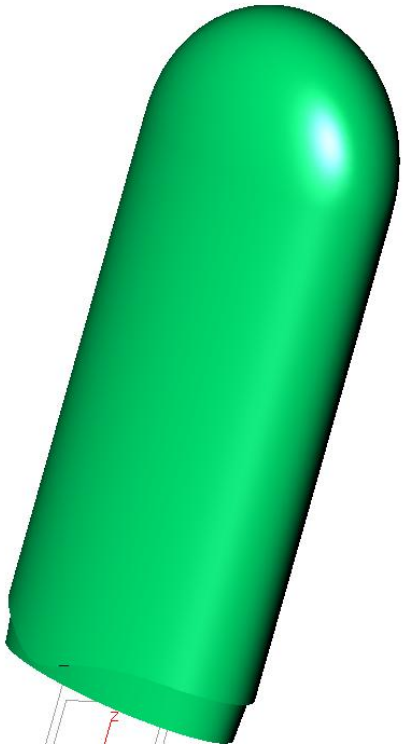
## Create a lofted face through the open edges and the trimmed circle

- Press **Ctrl-A** to set up an auxiliary view.
- Use the **Loft** **– Add** command to create a loft through the open edges and through the trimmed circle.
- Pick one edge, then the circle and finally the other edge.
- 
- The rest of the loft options are described on the next page.



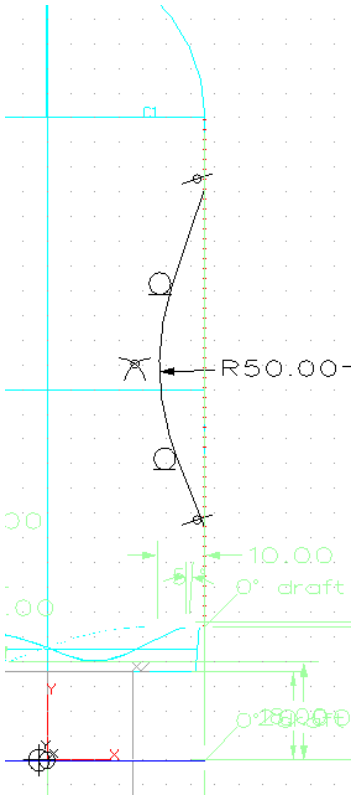

- We need the continuity set to TANGENT.
- We need to apply a weight on both of the open edges to change the shape of the default tangent loft.
- On the previous Shampoo Bottle example, we applied a different weight to each side. Here we can apply the same tangent weight to both edges.
- Adjust the weight to a value of 0.4 on the “both ends” tab.
- Complete the loft command.
- The loft is automatically sewn to the existing shape.
- Erase the trimmed circle.





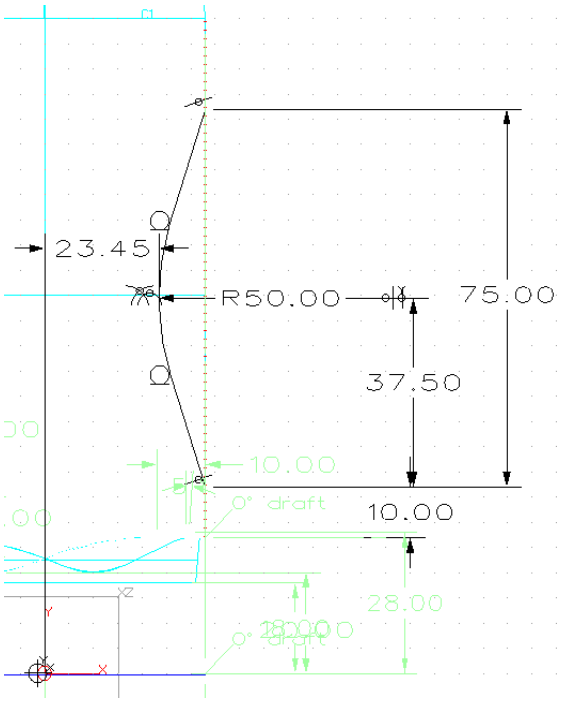
## Created a twisted cut feature on the side of the bottle

- We need to create a sketch for the “Sweep path” of the twisted cut.
- Switch on the display of dimensions.
- Create a new sketch on the **XZ**-Plane.
- Create a “curve reference” on the right hand edge of the main bottle body.
- Place the mouse pointer in the graphics area away from geometry.
- Right mouse click and select “Draw”.
- Right mouse click and select “Default”
- Create two lines as shown, make sure the line end points are on the reference line.
- Escape out of the “Draw” command.
- Right mouse click and select “Fillet”.
- Place the mouse pointer and pick at the intersection of the two previously created lines.
- Enter a radius value of 50mm.
- The sketch so far is shown opposite.
- Notice the constraints.
- Continue with the sketch on the next page.

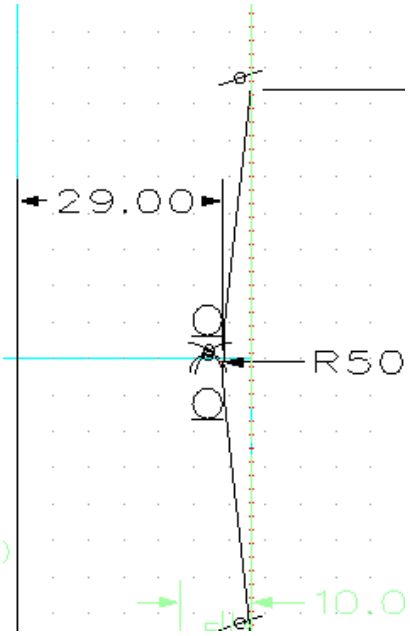



- Right mouse click and select dimensions.
- Dimension the part and change the values of your dimensions to those shown opposite.
- Note the 37.50 dimension is to the fillet center point.

Note: the 23.45 dimension is from the zero point to a point, which is on the fillet circumference, and the dimension is horizontal. This was all done with the quick dimension command. The on curve constraint was added automatically.




- Double click on the 23.45mm dimension, to change the value.
- From the Input Dimension Value form, pick the “Select Dimension” option.
- Using the “F7” key (external dimensions), pick the 33mm dimension from sketch1 (the original ellipse sketch).
- Append “-4” to the end of the Input Dimension Value text field.
- Press “Ok” and the dimension value should now be changed to 29mm as shown opposite.

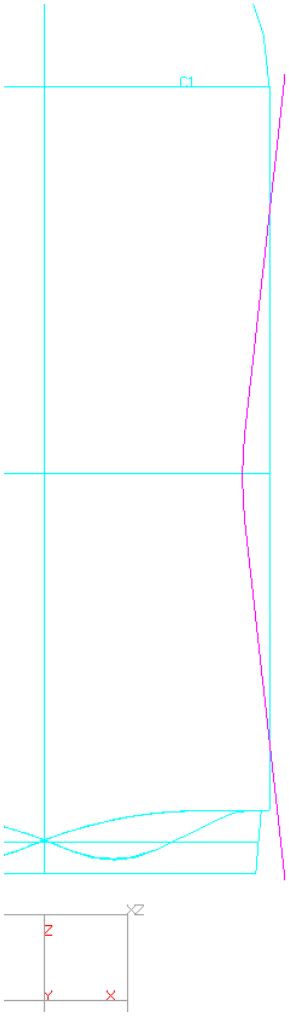


## Concatenate and extend the curves

-  and then switch off the display of the dimensions.
- Next we will concatenate the sketch curves into one wire-frame curve and extend both ends 20mm.

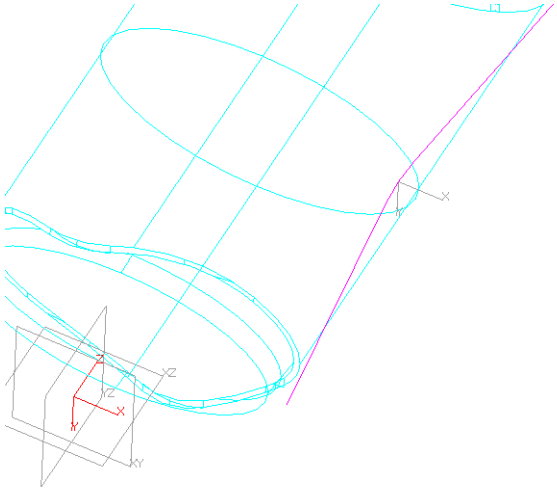
- Select "**Concatenate curves**" command. 
- Pick the sketch curves in sequence to make a new curve.
- We will now extend both ends of the curve.
- Note: Sketch curves cannot be extended or trimmed when in the Modeler. To make picking of the wire-frame curve easier, first "Blank" the sketch (pick filter set to sketch).
- Select "Trim/Extend curve".
- Pick the concatenated curve.
- Enter a value of 20mm for the extension.
- From the option buttons select "Linear"

Check the box to extend "Both" ends.

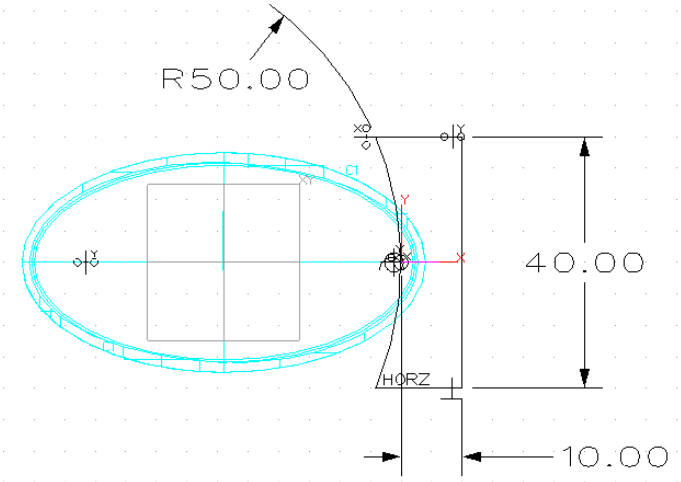


Next we will create another sketch that will be used for the cross sectional shape of the twisted cut. But first, create a new plane as follows.

- Create a “Datum Plane” located at a point” as follows.
- Pick the **XY**-Plane.
- For the **origin**, press the right mouse button and select **middle**.
- Pick the middle of the wire-frame curve as shown opposite.
- We will create our new sketch on this plane.
-

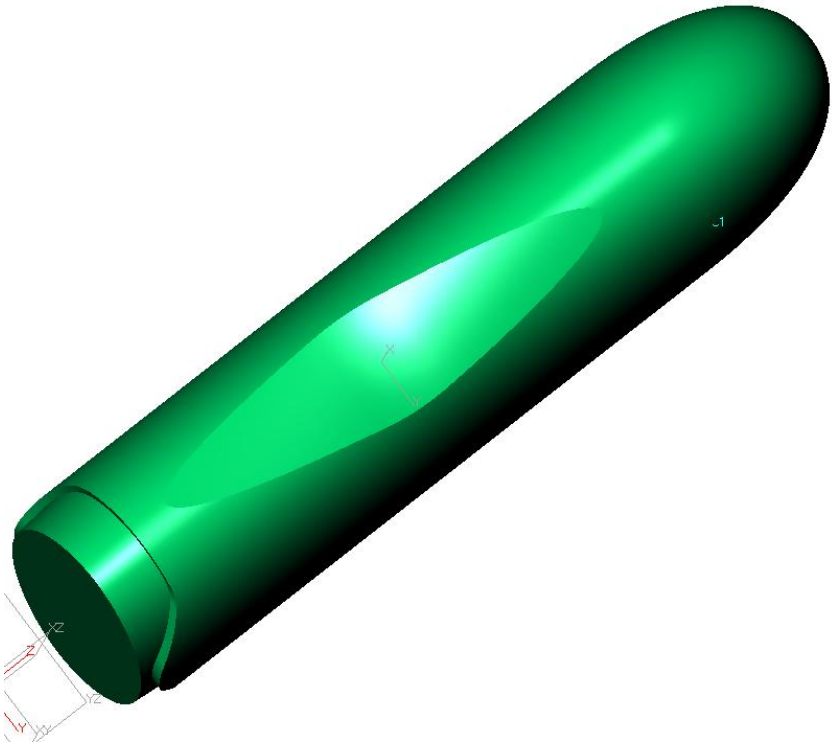


- The new sketch with the correct sizes is shown opposite.
- If you wish, using the same technique as used on the Shampoo Bottle example, you can insert a sketch copy from the “Conditioner sketches” file. The name of the sketch is “Cross_section”.
- Or if you prefer you can create the sketch shown. Note that all the geometry was created with quick draw. Create the arc with quick draw (hold down the “Alt” key after the first end point pick. Make sure the center point of the arc, is aligned in “Y” with the zero point, note the relevant constraints.



## Now we will create the twisted sweep cut

- Pick the "**Sweep** **> Remove**" command.
- Pick the newly created cross sectional sketch.
- For the sweep path pick the lower end of the wire-frame curve.
- For the sweep frame method, select "Natural".
- From the "twist" options, select "Linear".
- Enter –20 and 20 in the twist text fields, as shown opposite.
- Press ok.
- **Do not** erase the wire-frame curve at this stage.

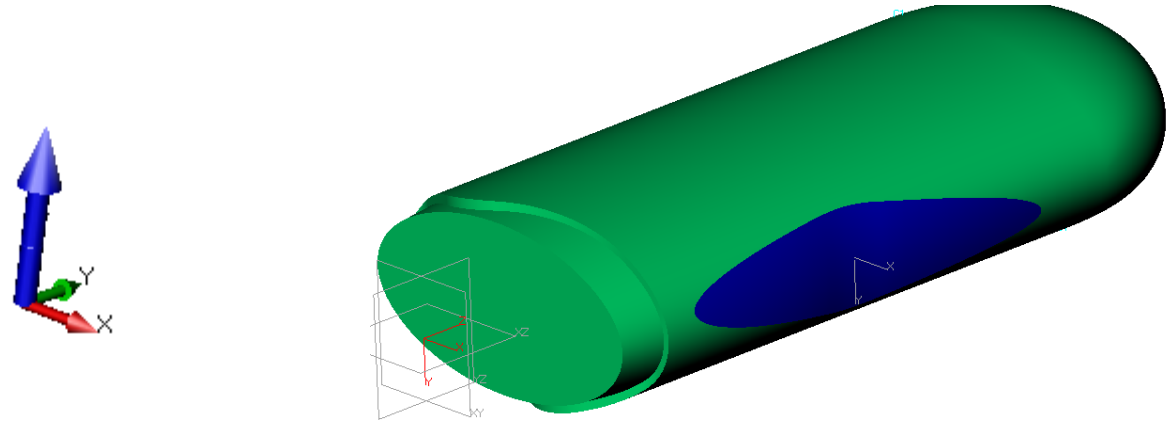


## Using the "Pattern" command.

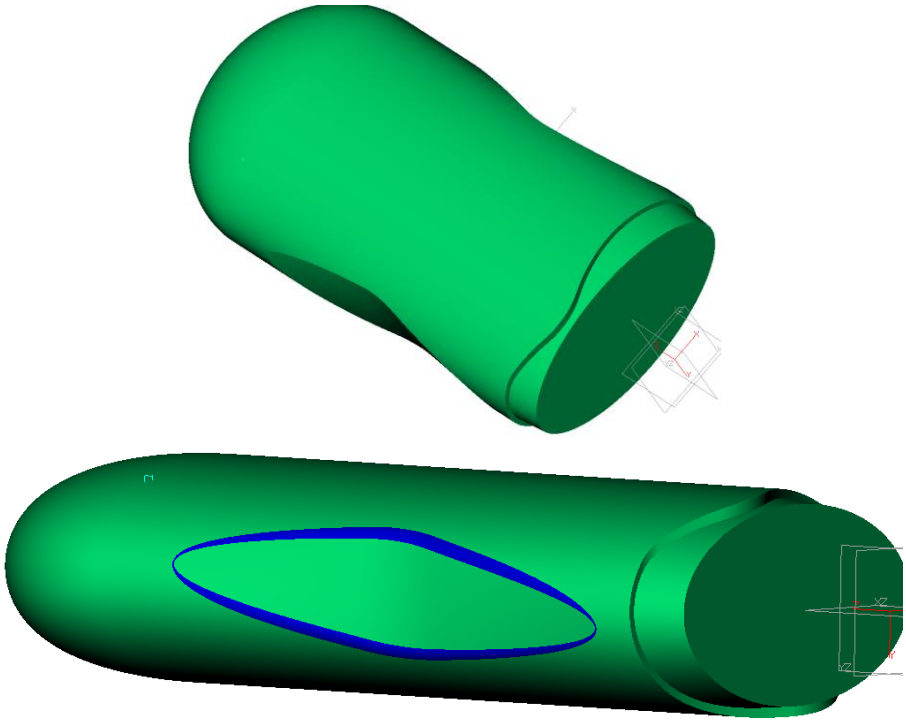
Now that we have created the cut on one side of the bottle, we need to copy the cut feature to the other side.

- Pick "**Edit > Pattern**" and select "**Circular**".
- Pick the twisted cut feature. (Pick filter set to feature)
- For the axis of rotation, select **Z-axis**.
- Total number of features = **2**.
- Enter spacing in degrees = **180**.

Now that we have completed the feature command, we can erase the wire-frame curve.
Why could we not erase the wire-frame curve before the copy feature command?



- Create **3**mm radius **fillet**s on all the edges of both twisted cuts. 
- Use chain select to pick the edges.
- The fillets on one of the cuts is shown opposite in blue.

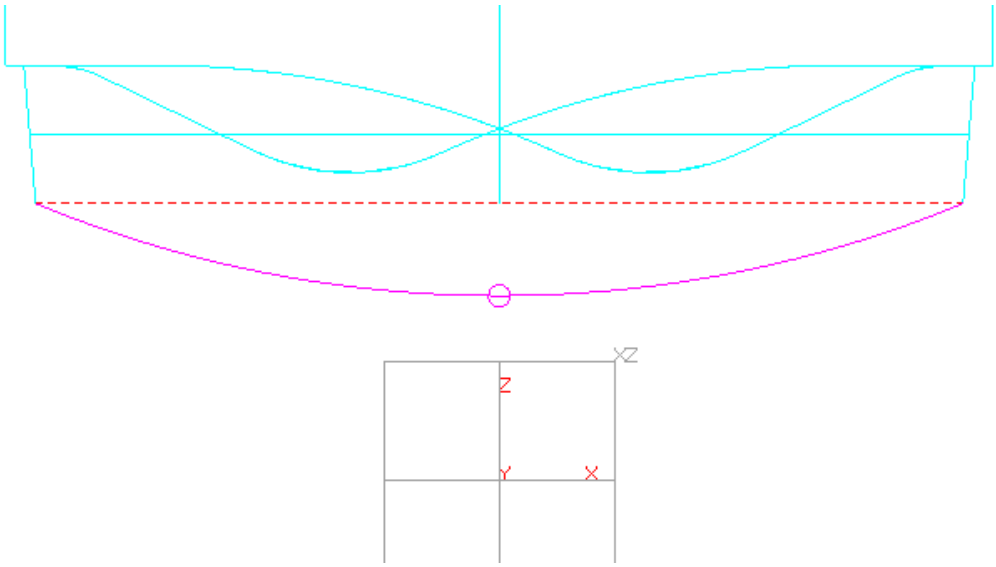


## Create the neck loft shape and the neck of the bottle.

We will be shortly creating the neck of the bottle, but first set the pick filter to “Face” and erase the existing planar face at the “neck” end of the bottle. Next we will create a 3pt arc. The end points of the arc will be at the edge endpoints at both ends of one of the open edges. The middle circumference point will be the point described below.

Let’s create the arc middle circumference point. This point needs to be offset from a point halfway between the two open edges (the middle of the “hole”).

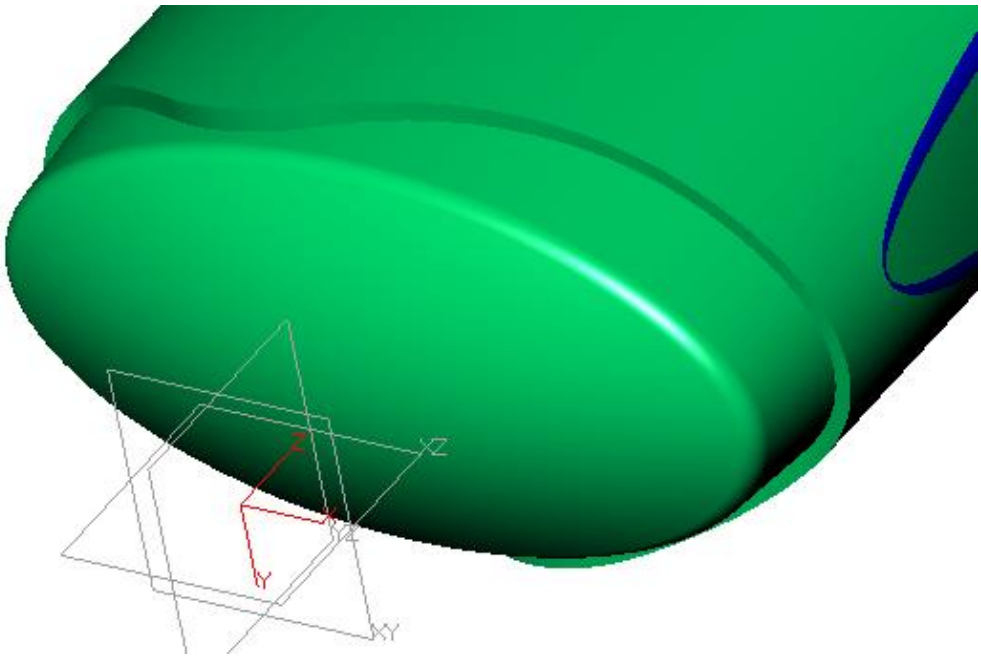
- Pick the “**Point**” command from the wireframe tab.
- Press the right mouse button, and select offset.
- Press the right mouse button again, and select between.
- Press the right mouse button again, and select critical.
- Pick the two end points of one of the open edges.
- Accept the default percentage distance (50).
- Enter zero (0) for both the X and the Y values, and -6 for the Z offset value.
- Now we can create a 3pt. Arc from both end points of one of the open edges, with the previously created point as the middle or circumference point of the arc as shown.
- If you had wanted, you could have skipped the point command and directly defined the mid point of the arc using the same technique.



## Creating and adding the Loft face to the bottle

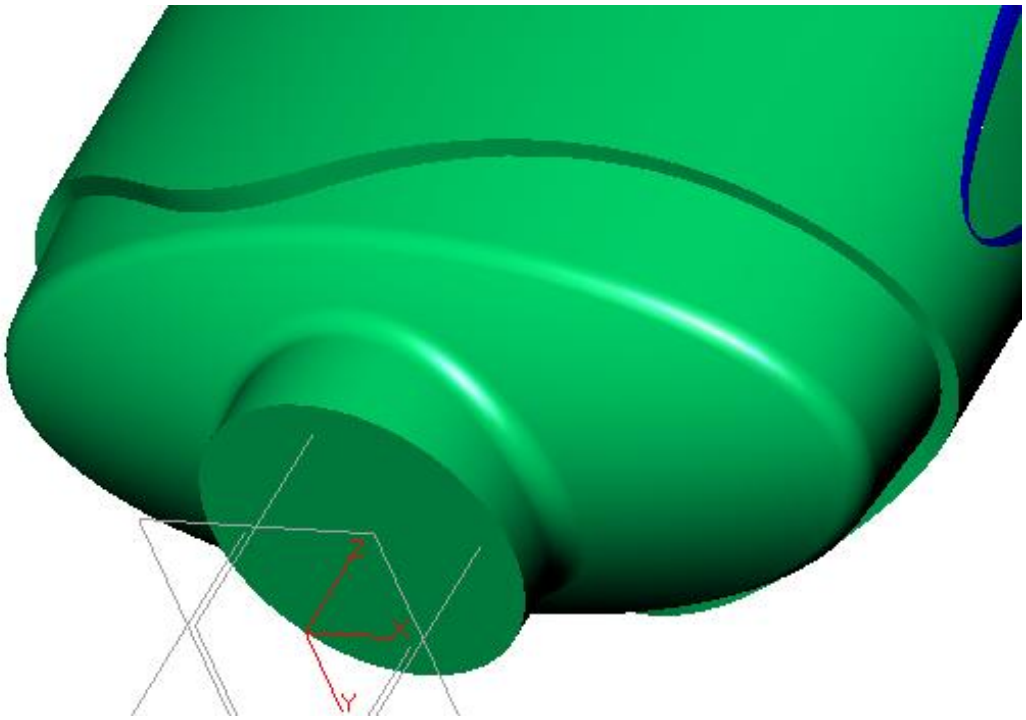
Use the "**Loft**" command to create a lofted face from the bottom open edges of the bottle and the arc, as follows:

- Press **Ctrl-A** to set up an auxiliary view.
- Select the "**Loft**  **> Add**  " command.
- With the pick filter set to all, pick one of the open edges (dotted blue) of the bottle near the end point of either the left or the right side.
- Pick the arc near the same end point of the previous pick.
- Pick the other open edge, again with the pick point near the end point of the previous picks.
- It is important to pick near the same end points, or the loft will twist.
- As stated previously you could have chosen either the left or right end points.
- Change the continuity to "**None**".
- Erase the arc and the point and fillet the edge 2mm as shown below.




## Create the neck as shown below,

- Use an "**Extrude**  **> Add**  " command.
- The sketch plane is the **XY**-Plane.
- The diameter of the boss is **22**mm.
- Start **8**mm.
- End **20**mm.
- Then **fillet 2mm** between the neck and the bottle.



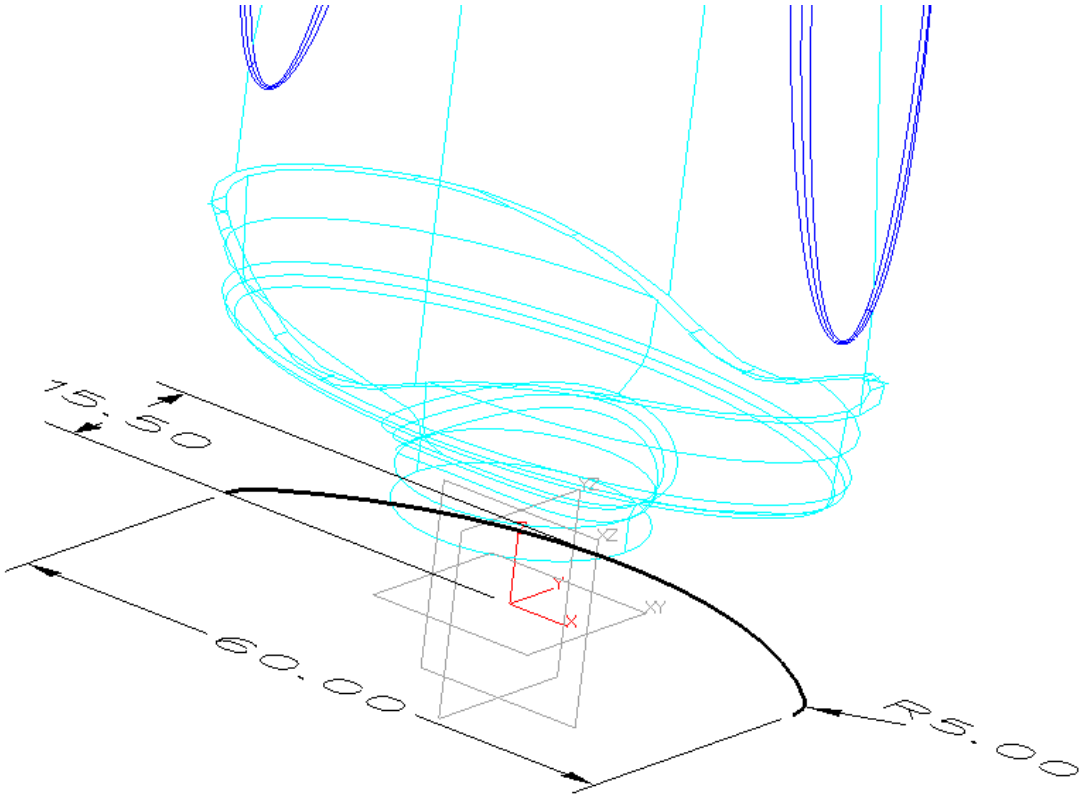
You are saving the file periodically ……aren't you?

## Create the Cap shape.

We can now create the bottle cap. We could create a separate part (the cap) and using assembly tab, instance the geometry from the bottle that we need in the cap. In this example however, we will create the cap shape in the same part as the bottle shape. If we later needed to have two separate parts, for creating drawings for example, we could use the "**Edit > Copy > geometry to part**" with the "Backup" option or "**Extract shape to Component**" commands to export the bottle and the cap into two separate new parts.

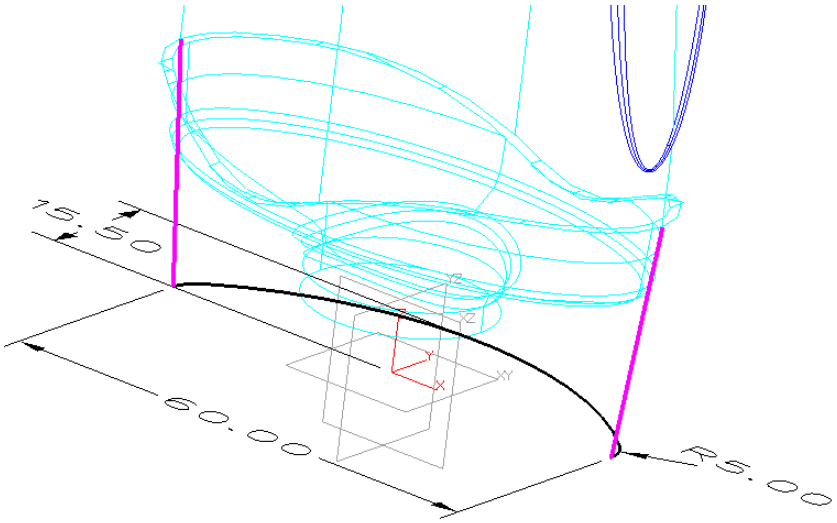
## Creating a Mesh surface.

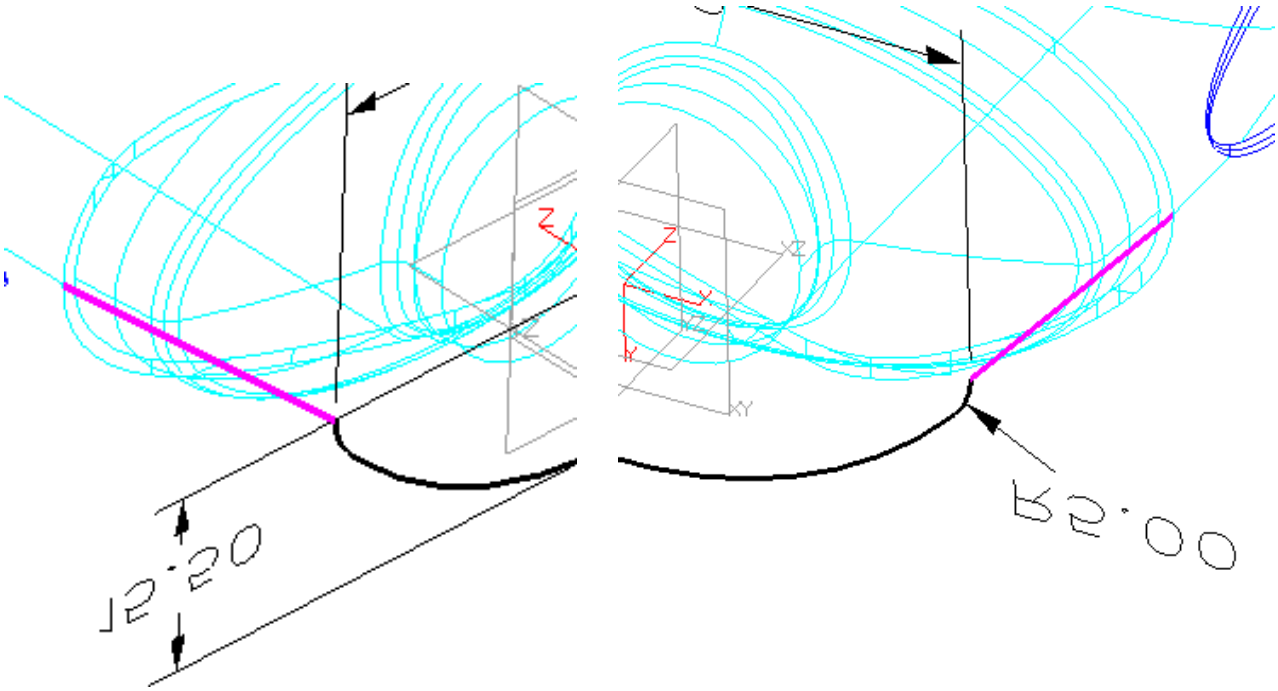
- The end shape of the cap is not quite elliptical. The shape required will be created from three circular shapes. If you wish, you could create the sketch shape required. However, we have saved a fully constrained sketch that you can use if you prefer.
- Use the "**Edit > Copy > External Sketch**" command to insert the sketch "**End_Cap_shape**" from the "Conditioner_sketches" file.
- Place the sketch on the **XY**-Plane as shown below.



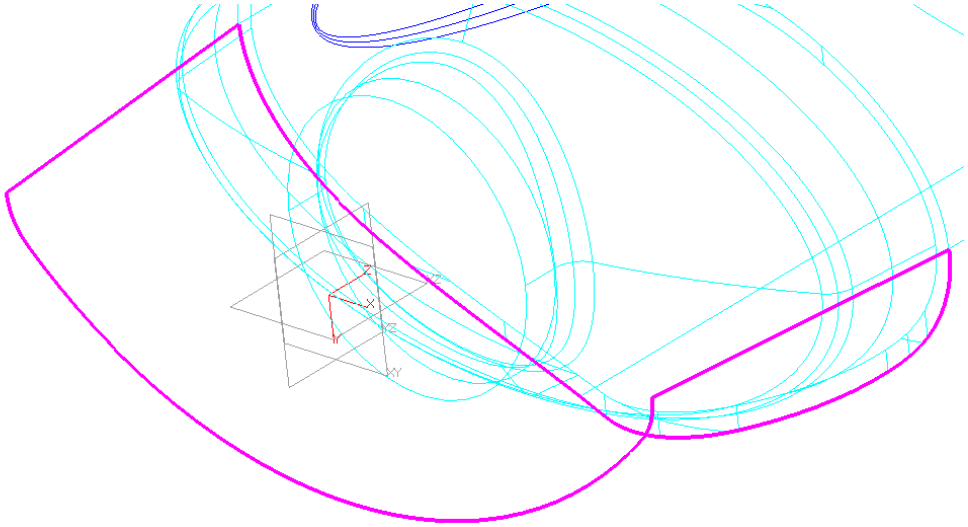

Next we need to create two lines that will be used in our soon to be created mesh face.

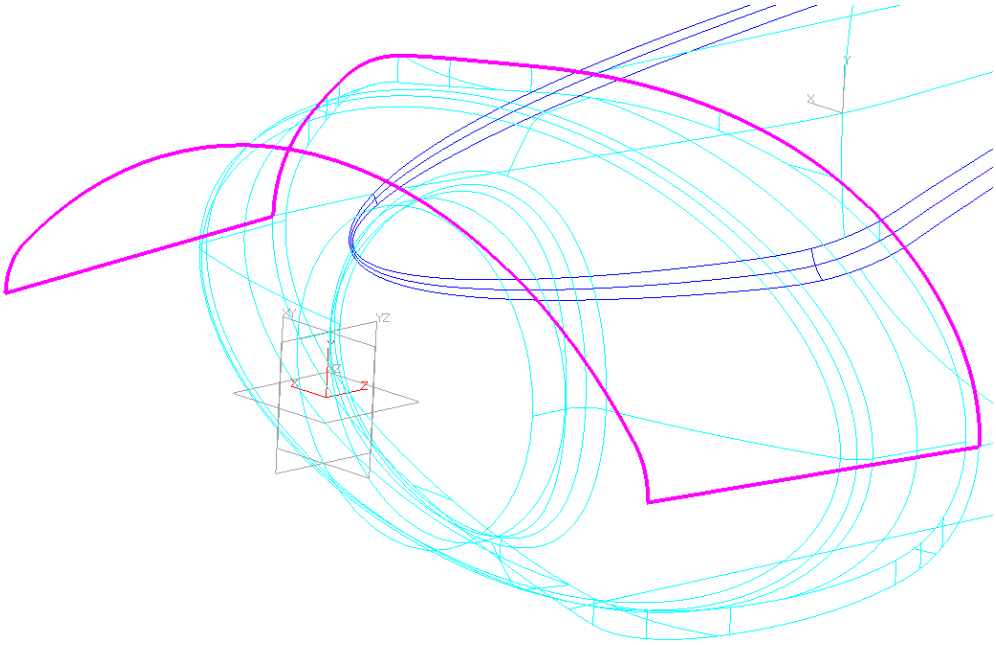
- From the wire-frame options, select "**2 points line**" and created the lines shown below. 
- Use the right mouse button when creating the lines to set the pick option to "**critical**". Set the pick filter to "**all**".
- As shown in the following figures below.

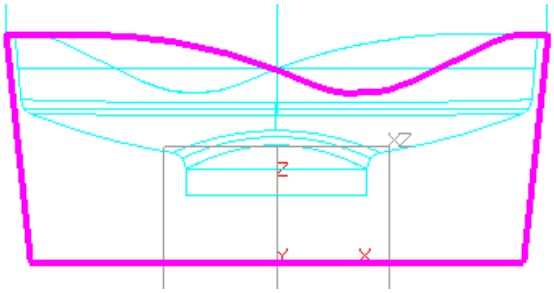





- Now use the wire-frame tab to concatenate the sketch curves into one wire-frame curve. **Do not** use chain select here, rather pick the curves individually.
- Next we need to create a curve from the edges of the bottle as shown below.
- Pick the “create curve from edges” command, and pick six **“outer”** edges on the lower half of the bottle as shown in magenta below.
- Concatenate these curves into one curve, again **Do Not** use chain select.

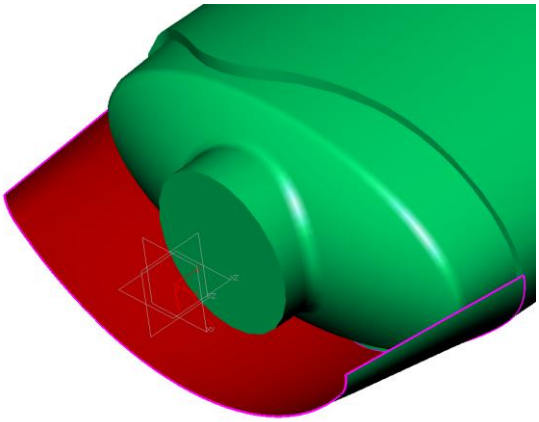




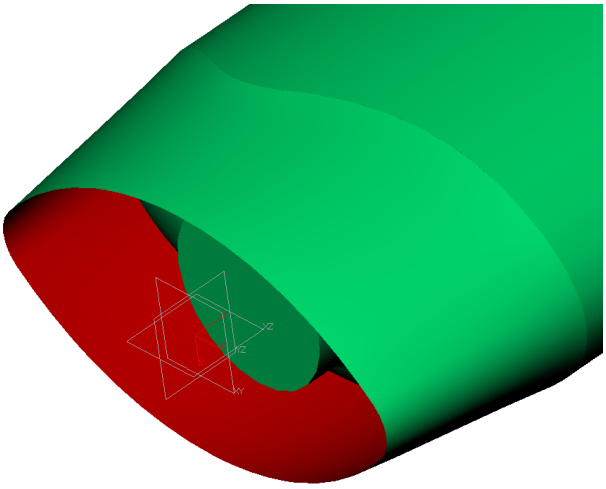


## Create the Mesh face as follows

- Select the "**Curve Mesh"** command. 
- Set the pick filter to "**curve**".
- Select the first set of curves by picking the two lines.
- Select the second set of curves by picking the two curves.
- As shown opposite.

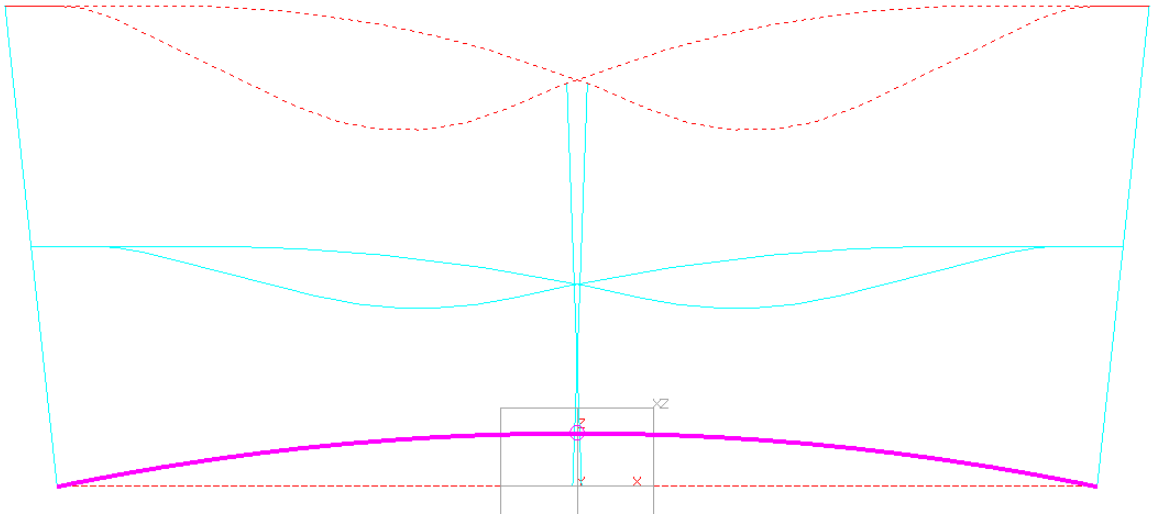


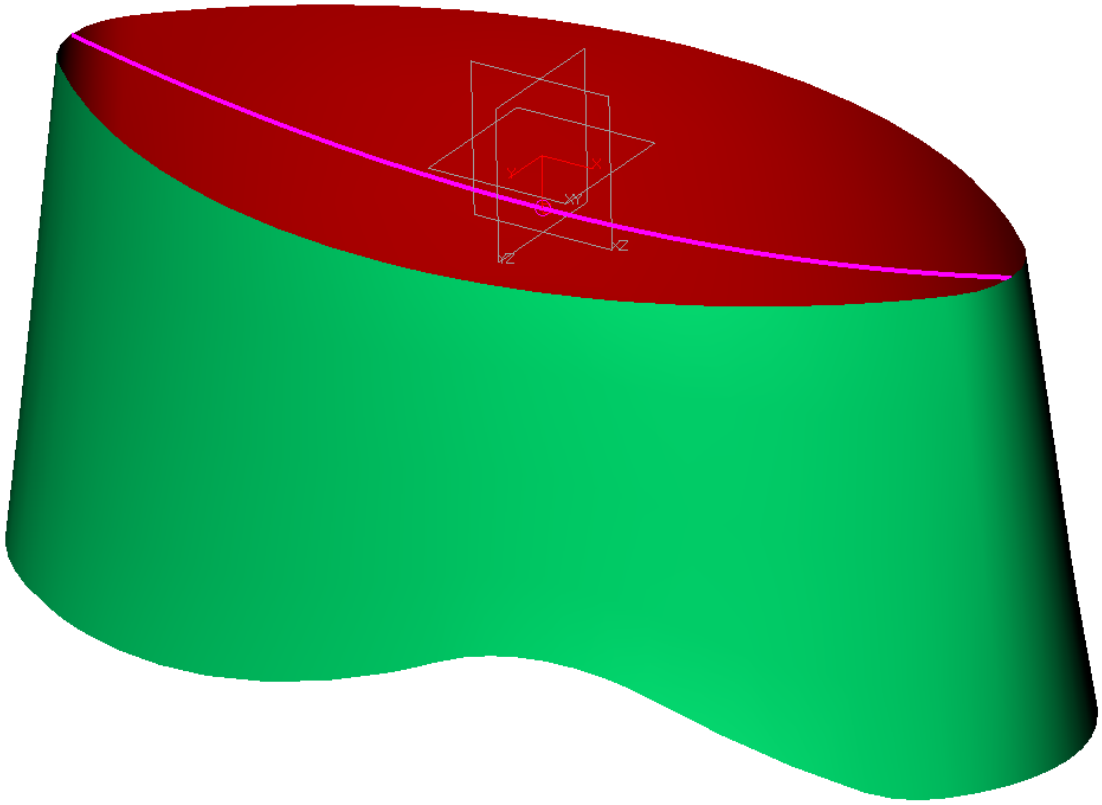
- Next with the pick filter set to "curve", erase the four wire-frame curvesif you pick Keep curves when you create the Mesh face.
- Select "**Edit > Copy > Copy Entities**" **,** "**along a direction**" command. 
- Pick the mesh face.
- Set the move direction to be the **Z**-axis.
- Distance to move =**0**
- Rotation about the Z-axis =**180**.
- Next use the "**Combine shape**  **> Add**  " command to add the two mesh faces together.
- As shown opposite.



## Create the end face on the cap.

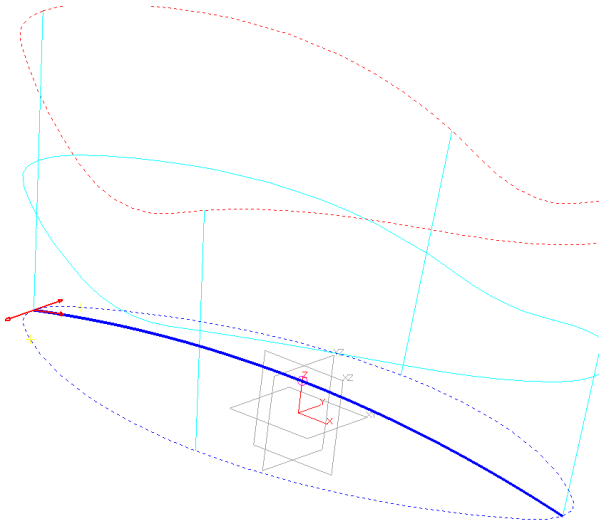
First **blank** the bottle; set the pick filter to **shape**.
Next create the offset **point** and **3pt arc** shown below.
The offset distance = **Z 2.5mm**.





## Create the lofted face as follows

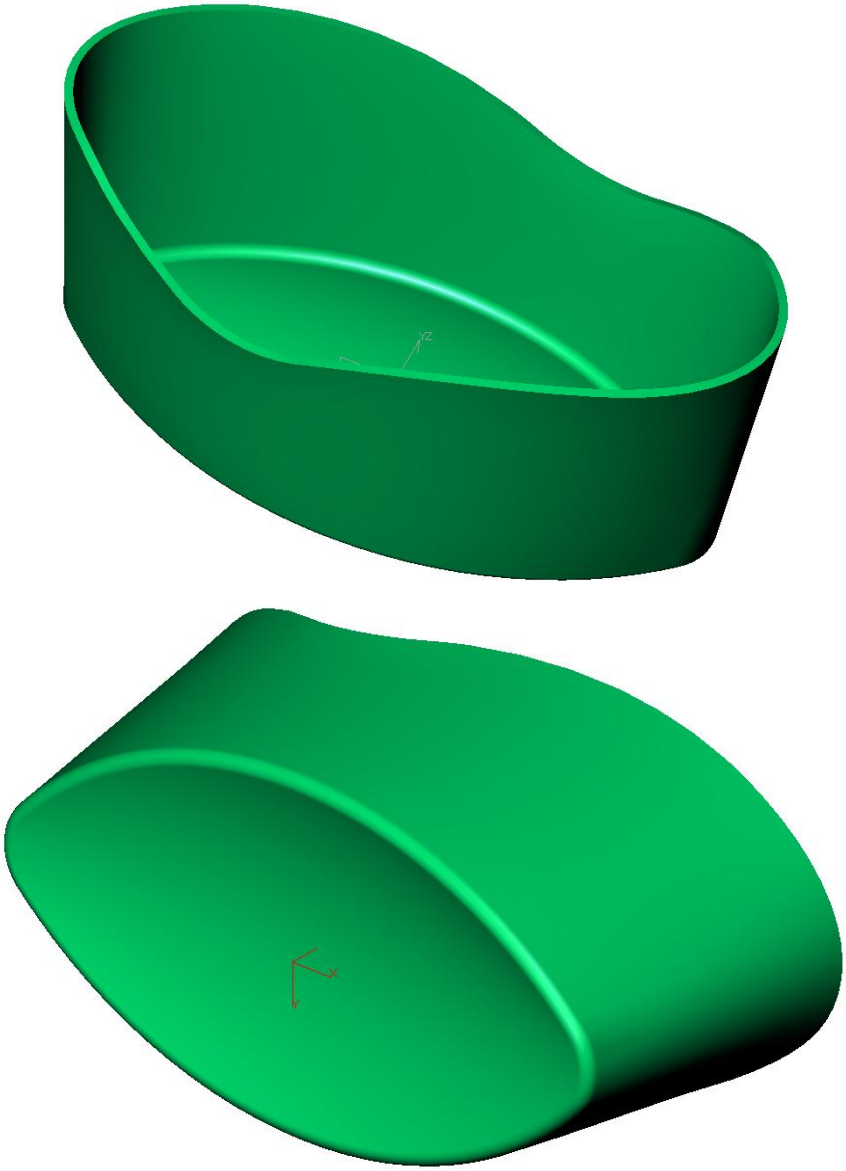
- Press “**Ctrl-A**” to set up the view shown opposite.
- Created a lofted face using **Loft** **> Add** from the two open edges and the arc as shown opposite.
- Continuity is **None**
- Erase the arc and the point and sketch (pick filter set to curve and then to point then to sketch). Right-click > Pick All each time.



## Shell the cap inwards (minus) -0.8mm

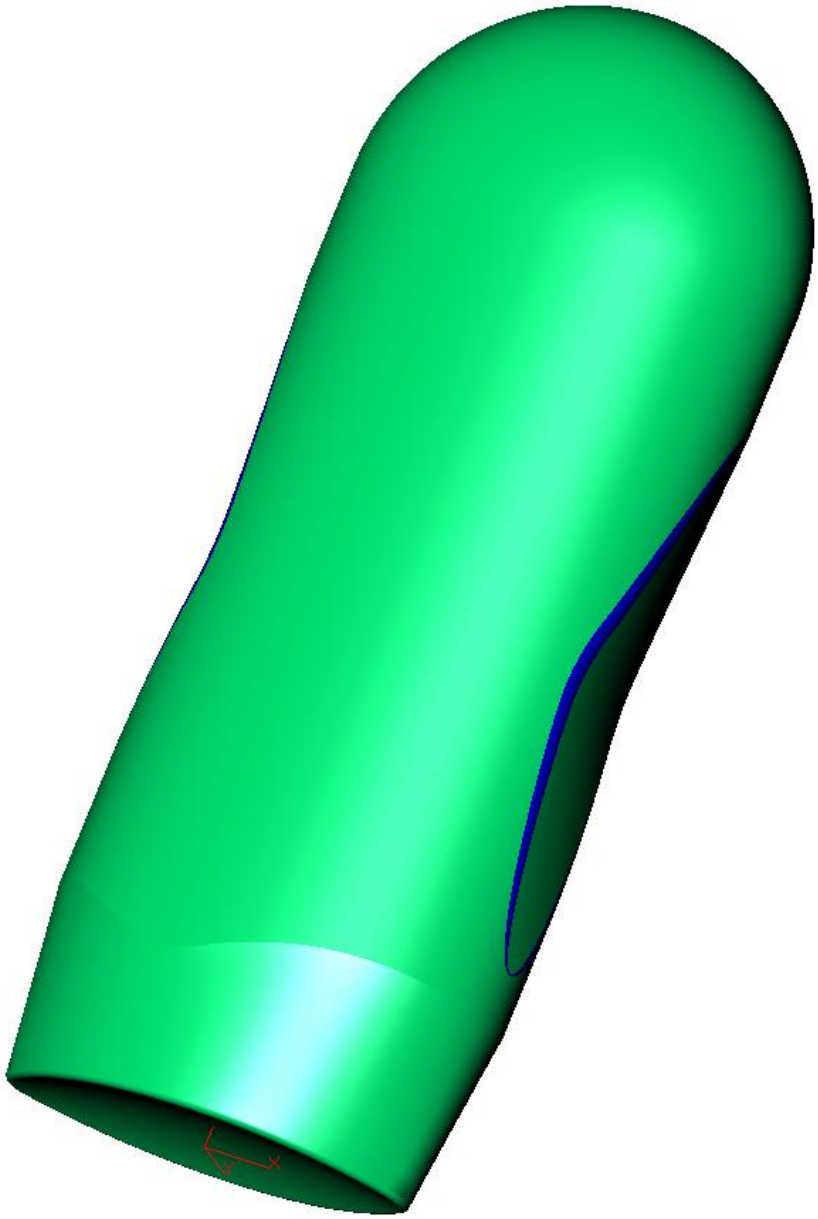
- Fillet the outer edges of the cap 2mm.
- Select the "**Shell**" command. 
  Shell the cap (note the cap is not a solid at this stage).
  Amount to shell = **0.8mm** but whether it is plus or minus will depend on the direction of the surface. If your model is pink on the outside you will want to use +0.8 so the part extends toward the blue or positive direction.
  Faces to leave open, none, press accept (Middle mouse button or Enter key).

NOW, it's a solid.



**Swap entity visibility.**

**Shell the bottle –0.8mm leave the top face of the neck open.**

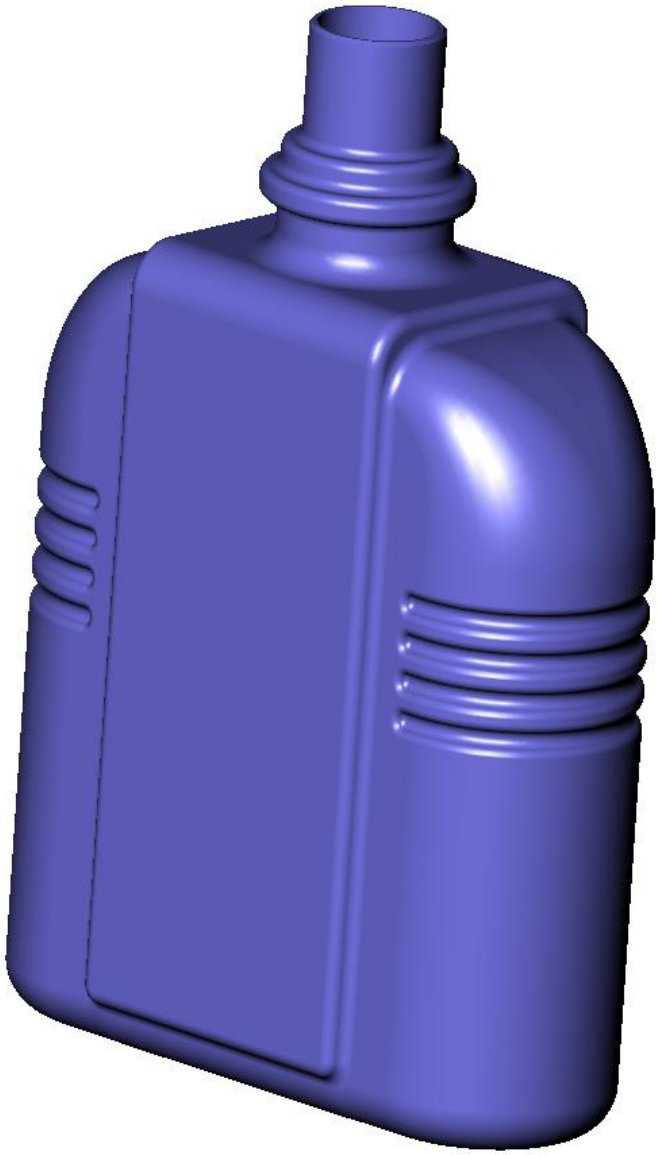


**Unblank all.**

If desired you can now export the two shapes into two separate parts.

Save your file “Conditioner.Z3”

## Example three, The Glass After-Shave Bottle.



Some of the topics covered include:

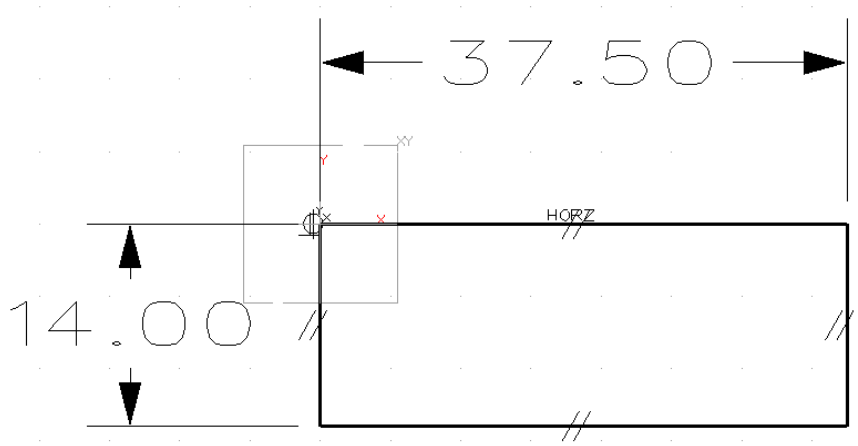
Constant and variable fillets.
Creating solids from open profiles by using the “interior” and “offset” options.
Bounded Boolean.

Create a new multi-object **file** called “**After_Shave.Z3**”.
Next create a new **part** called “**01_After_Shave_Bottle**”

This bottle is symmetrical and sometimes it is convenient, as in the previous examples, to model one half of the part. In this example however we will first model one quarter only.

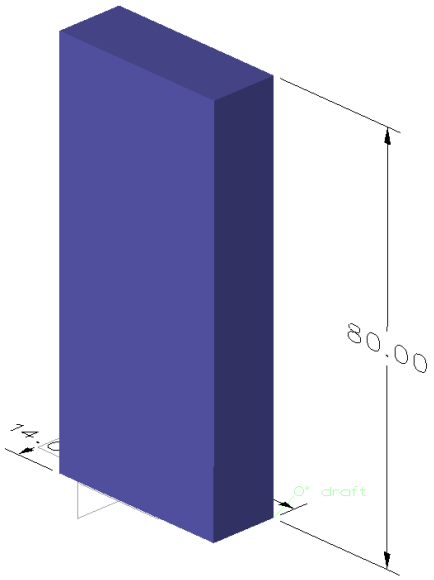
Let's start.

- Select the "**base extrude**" command. 
- Create the sketch on the **XY**-Plane as below.




- 
- Extrude the solid.
- Start = **0**
- End = **80**

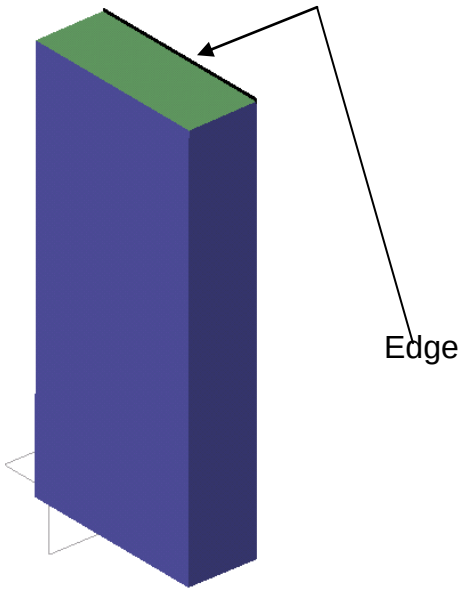
As shown opposite.



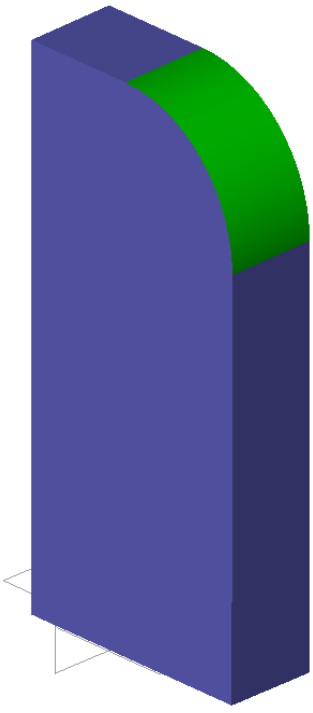

## Create a "draft" angle on the top face.

- Select the "**Draft**" command. 

- Pick the **edge** (filter)shown opposite.
- Draft angle = **1**
- Pick Y axis.
- Middle mouse click to finish



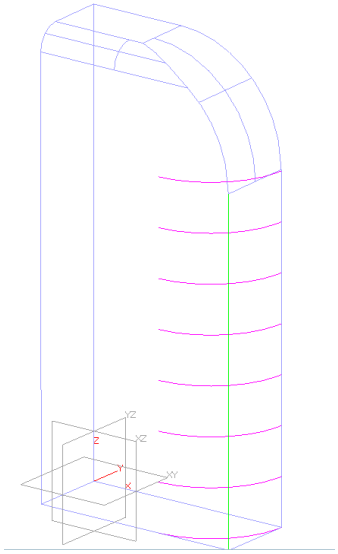

- Create a **20**mm **fillet** as shown in green opposite.

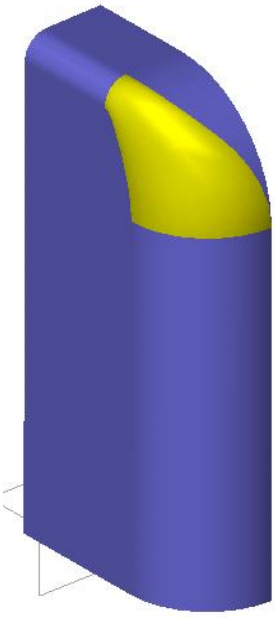


## Add fillet radius attributes and create a variable fillet

We will shortly be creating a variable fillet on the bottle edges, but first let's experiment with Constant filleting.

- First create a constant 4mm fillet on the edge shown opposite.
- Now create a 14mm fillet on the vertical edge shown in the bottom left figure (do not chain select).
- The finished fillet is shown below.
- Notice how the transition from the 14mm fillet to the 4 mm fillet occurs on the edge that we did not select for filleting, i.e. the edge of the previously created 20mm fillet.
- The transition near the 4mm fillet is a little severe. Let's try a variable fillet instead.
- **UNDO** both the 14mm and the 4mm fillet.



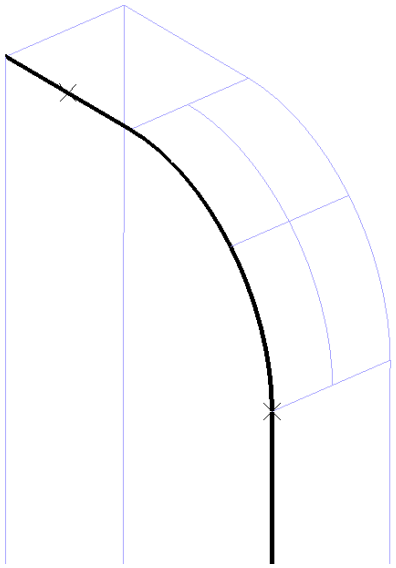


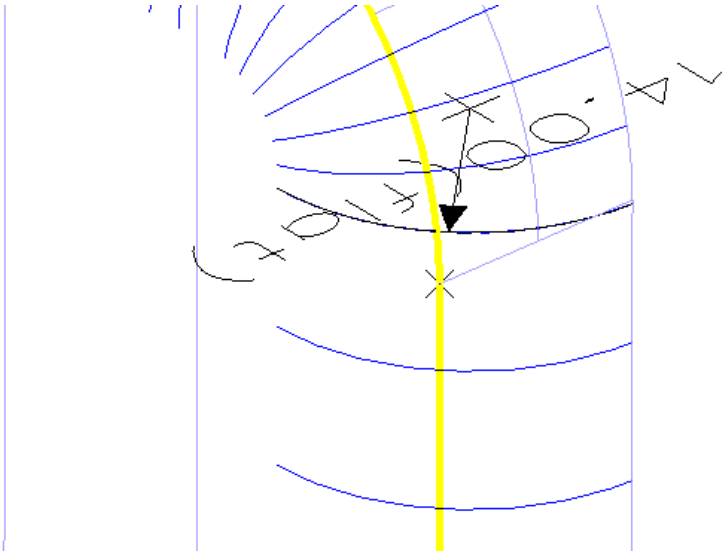
We will next create a variable fillet along three of the edges of our part.

Review the Variable radius commands in the HELP Manual.

- First, select the **fillet** command. 
- Pick the **three edges** shown at the right.
- Set the radius to **4**mm and hit **Enter**.
- Pick the **Advanced** tab
- Pick **Add** in the **Variable Radius**.
- Pick the lower critical point shown.
- Enter a Value of 14 for the first fillet radius.(Don't hit Enter, yet)
- Check the box for "locally flat"
- Now hit Enter.

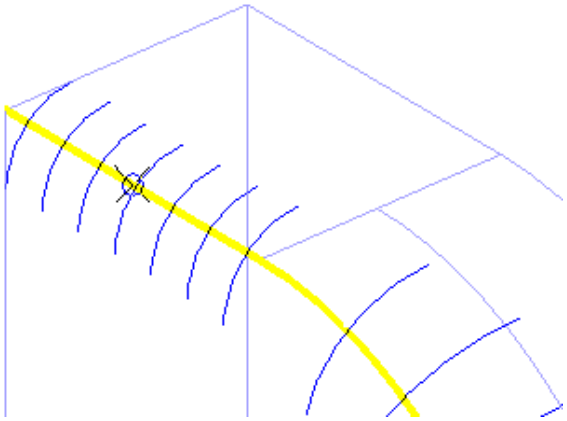
Continued……..

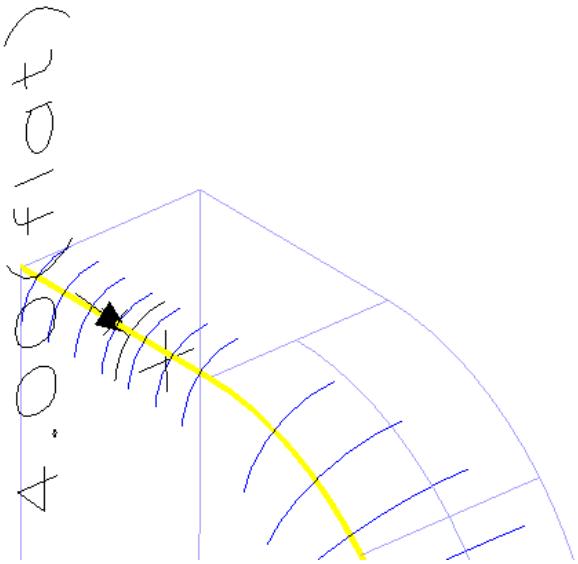


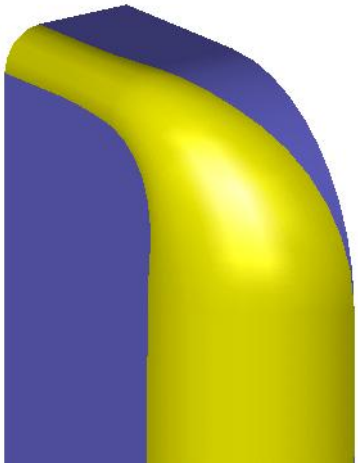


- Attach the next fillet radius attribute to the edge middle point, shown opposite

- The radius = 4
- Locally flat should still be selected.

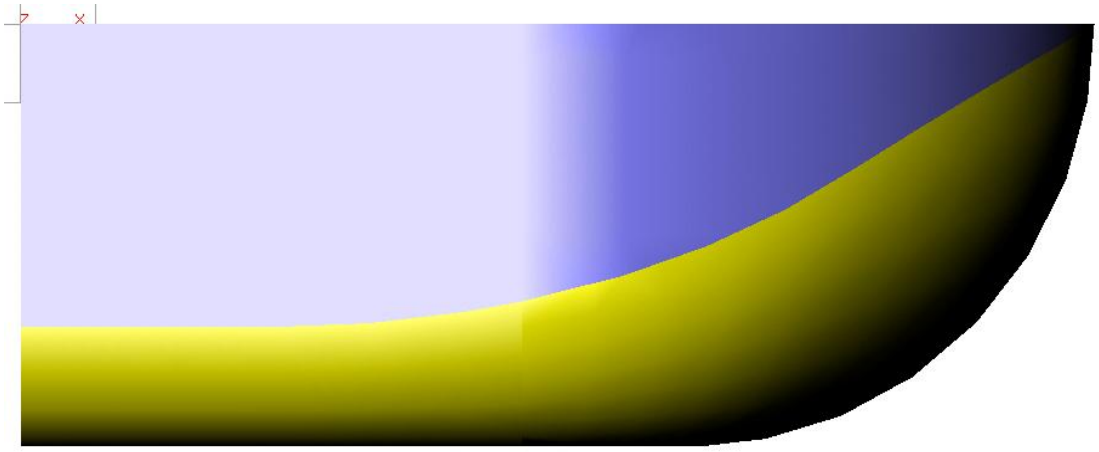






The shape of the transition doesn't look very much different than what is on page 70, until you look at the top view. Now you can see that the transition from 14 to 4 now goes to the mid point of the top edge.

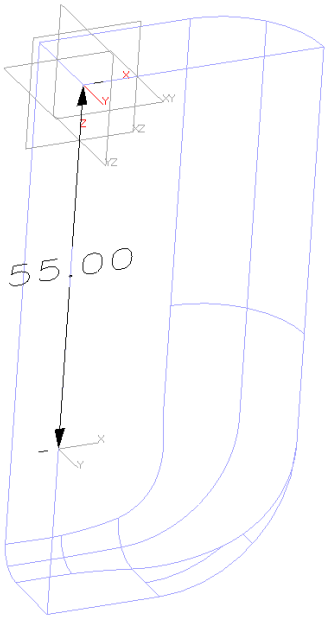
Notice also that the entire side face has been eliminated by the 14 mm fillet.



## Create a "Groove" shape feature.

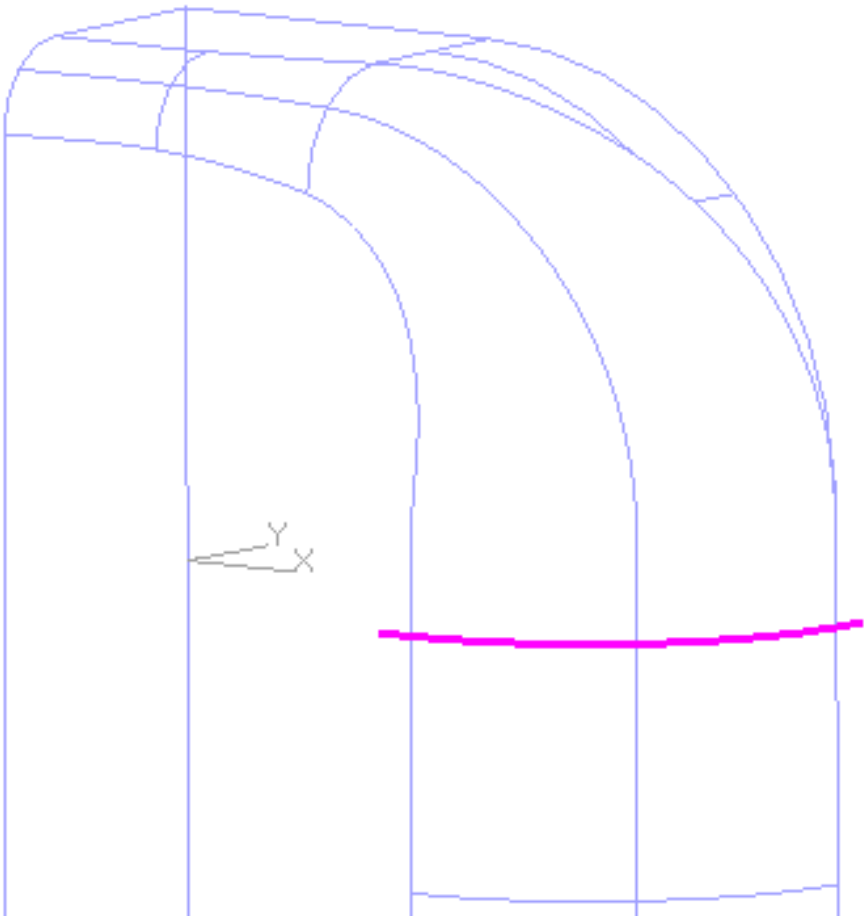
First we will create a datum plane that will be used to create an intersection curve between the plane and the "14mm" fillet face.

- Select the " **Datum plane**" command.
- Pick the **XY**-Plane.
- Offset distance = **55**mm.



## Create an intersect curve and extend the curve both ends 2mm.

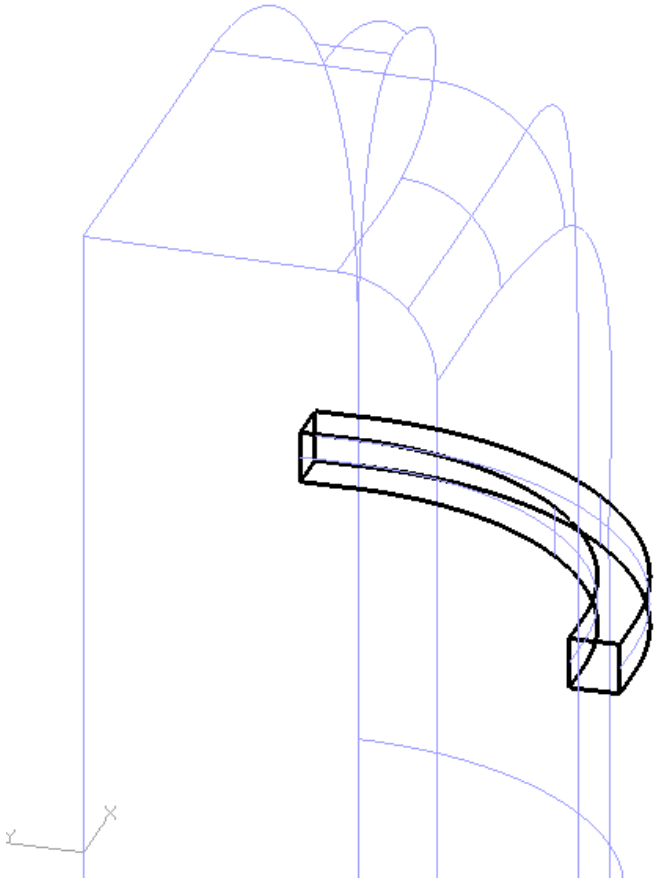
- From the wire-frame tab select the "**Intersection curve**" command. 
- Pick the previously defined plane.
- With the pick filter set to "face" pick the "14mm" fillet face.
- Next from the wire-frame tab select the "Trim/Extend curve" command.
- Use the relevant options to extend the curve **linearly** both ends 2mm as shown below.



## Create an extruded solid from the previously created curve.

- Select the "**Extrude > Base**" command.
- Pick the curve
- Start value = **1**mm.
- End value = **-1**mm.
- From the option buttons, select "**Offset**".
- Select the "**Thicken Same**" option.
- Enter a value of **1**mm, which will "thicken" the solid by 2mm.

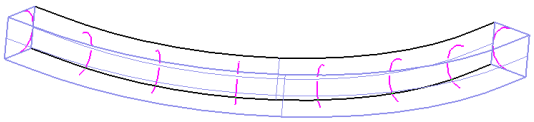
**Erase** the curve, as shown below, if you have picked **Keep profile**.

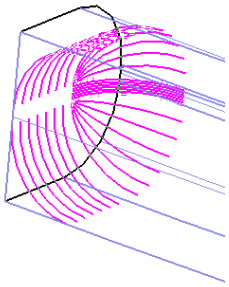


## Create fillets on the new solid.

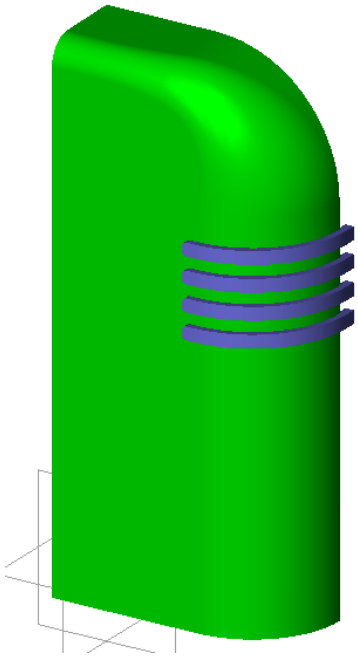
- Create **0.9mm fillets** on the two inside edges.

Then create **0.9 mm fillets** on the edges on the left side only (Chain pick), as shown.



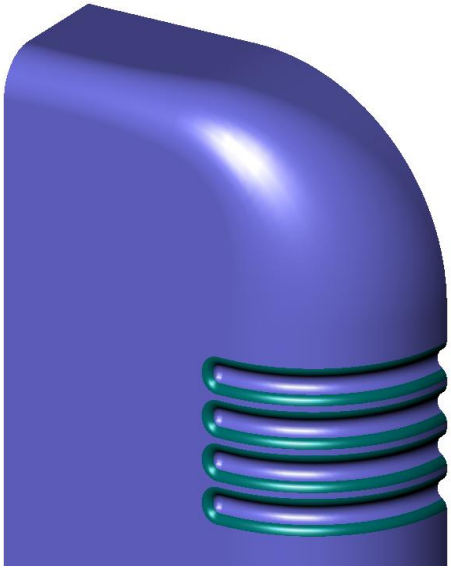


- Next copy the solid as shown.
- Select "**Edit > Copy> Copy Entities**", pick "**along a direction**" command.
- Set the pick filter to "shape".
- Pick the new solid.
- The copy direction is **–Z**.
- The distance between is **4**mm.
- Number of copies is **3.**



## Create the grooves and fillet the edges

- Use the "**Combine shape > remove**" command to remove the four "Groove" solids from the bottle.
- Then using chain pick, (hold down the Shift key) create 1mm fillets on all the edges shown opposite.

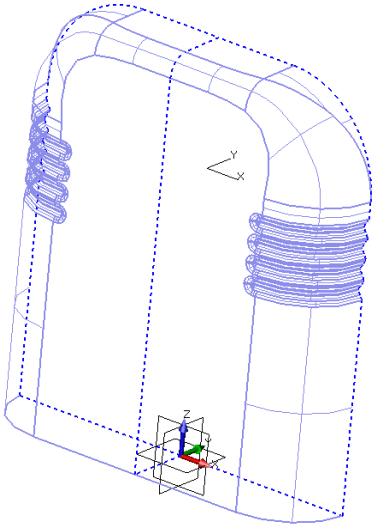


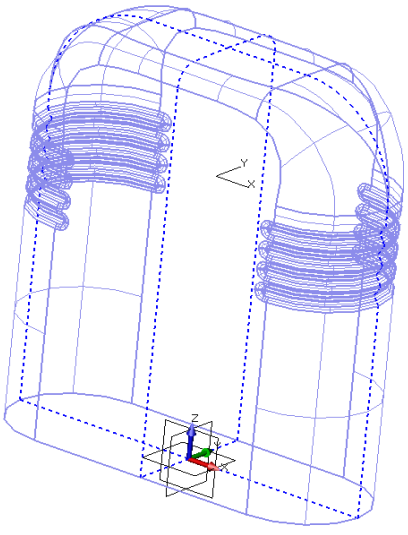
## Copy the shape three times to make the "full" bottle shape.

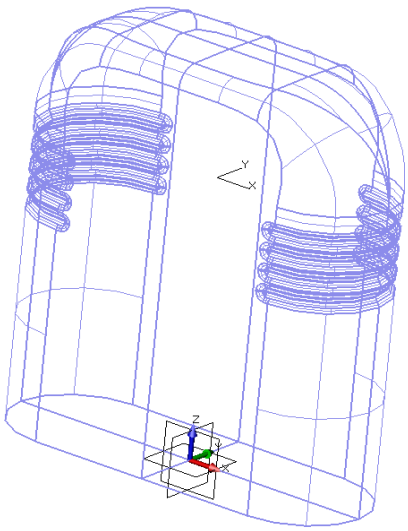
- First, erase the "back" and "middle" faces as shown opposite.

- Select “**Mirror**” command. 
- Set the pick filter to “shape”.
- Pick the existing shape.
- The mirror plane = **YZ**-Plane.
- As shown top left.
- Repeat this procedure to mirror the two existing shapes about the XZ-Plane as shown bottom left.
- Notice the “Blue Dotted” open edges.

- We will now sew the edges together with the “**Sew**” command. 
- Pick “**OK**” to sew everything.
- Notice in the bottom right figure that all the edges are now joined successfully. The message window also indicates there are zero open edges.



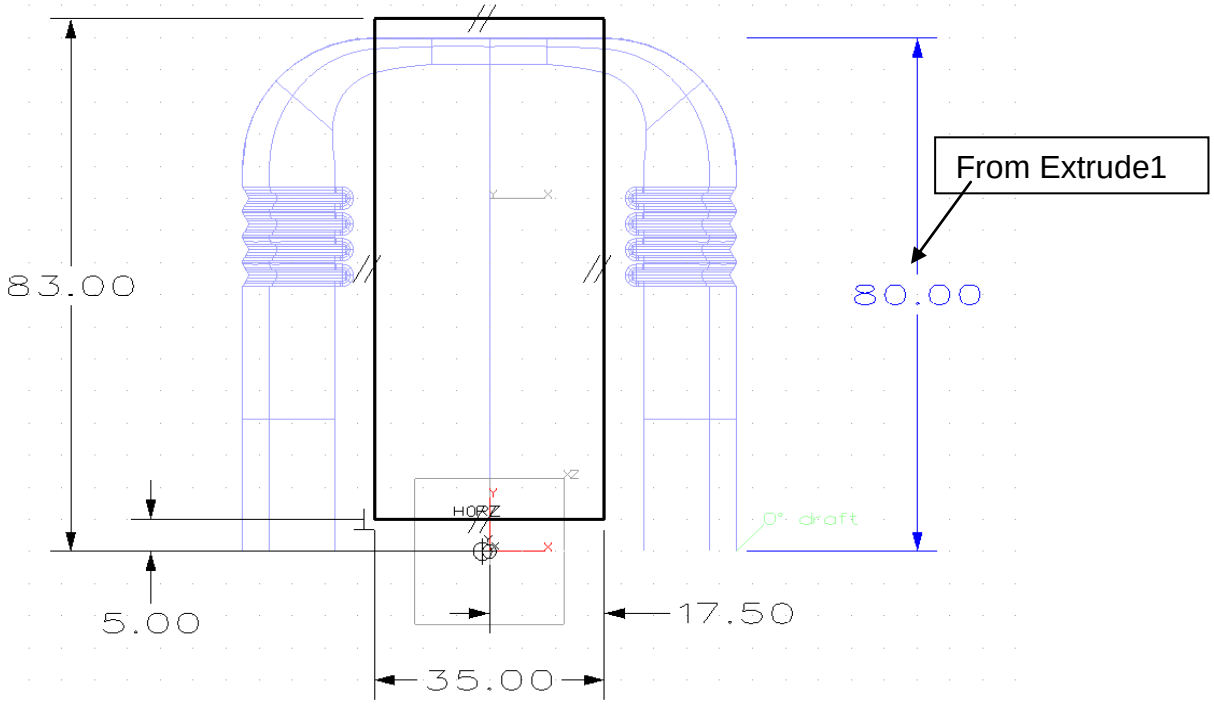




## Create a Boss Feature.

The planar faces on the front and back of the bottle consist of several faces each. Although it is not strictly necessary, let's say we want these 4 faces replaced with 2 planar faces. Notice that when we create the next "Boss", the common planar faces are merged automatically together.

- Switch on display of dimensions.
- Select the "**Extrude**  **> Add**  " command.
- Create a new sketch on the **XZ**-Plane, as shown below.
- Notice the point at zero.
- Using the "Input Dimension Value form" make the 17.5 dimension, half the 35mm dimension.
- Another way of constraining the sketch to keep it centered is to erase the 17.5 dimension and add a mid-point constraint. Use the option "Coordinate > X". This will center the shape in the X direction only.
- Again using the "Input Dimension Value form" and the "F7" key make the 83mm dimension 3mm bigger than the external 80mm dimension.Input Dimension Value：[01_After_Shave_Bottle:Extrude1_Base_d3]+3
- Temporarily change the 35 dimension to test your constraints.

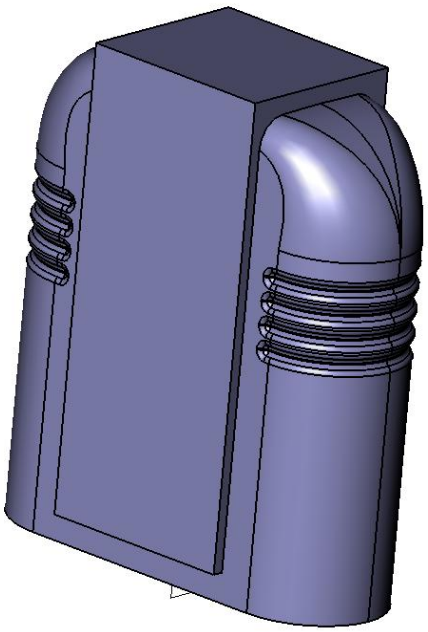



- The start of the extrude = **-16mm**
- The end = **16mm.**
- The planar faces will now have "merged" automatically as shown opposite.

Let's also merge the bottom faces

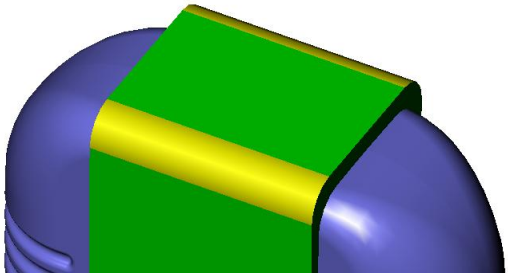
- Select "**Merge**" from the freeform menu. 

- Pick the 4 bottom faces

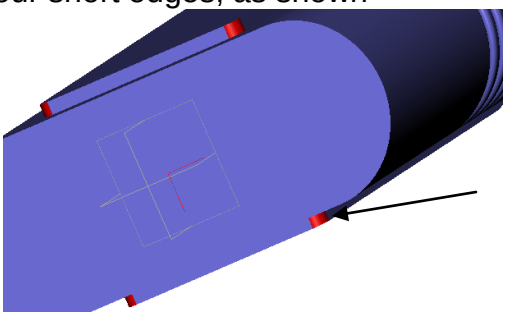


## Create fillets on the edges of the boss.

- Create 5mm fillets on two edges, as shown.

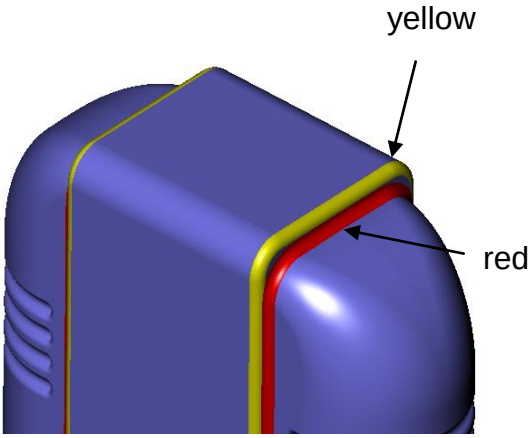


- Create 2mm fillets on four short edges, as shown



- Finally create 1.5&1mm fillets as follows.
- First “chain select” the outer edges and create the 1.5mm fillet, as shown opposite in yellow.
- Secondly using “Chain select”, pick the inner edges. Then create the 1mm fillet, as shown opposite in red.

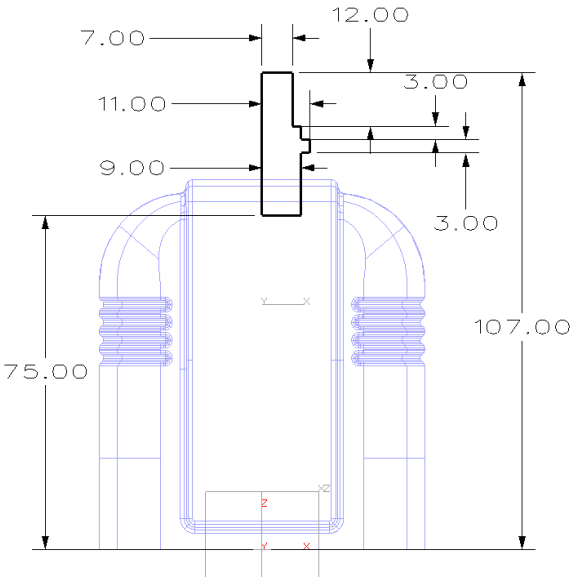
- Because we created the “yellow” fillet first, the “red” fillet can roll over onto the faces of the “yellow” fillet.

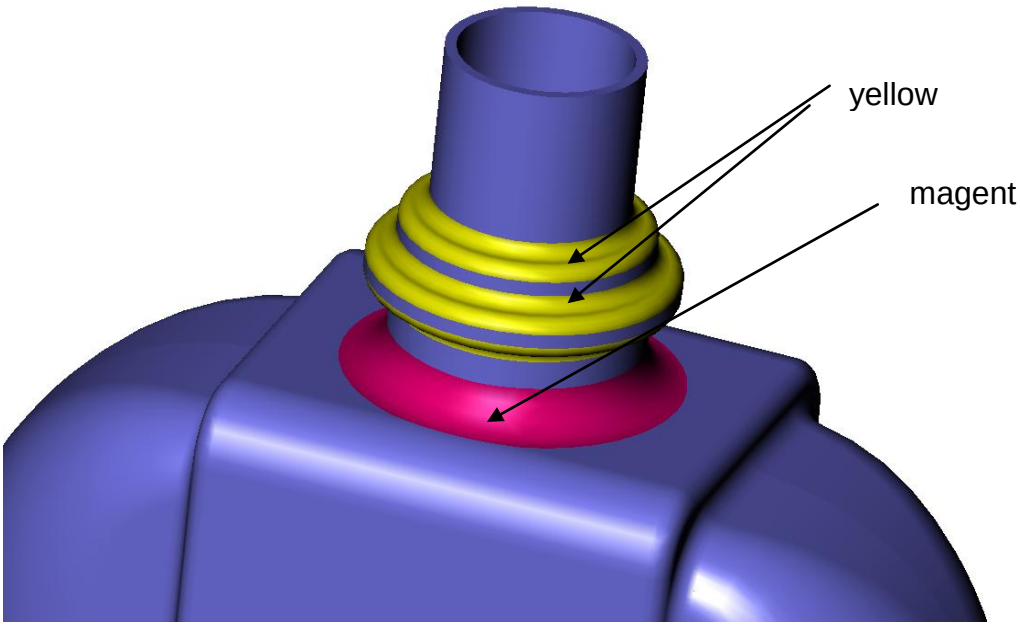



## Create the Neck

To create the neck shape we will use a revolved > add.

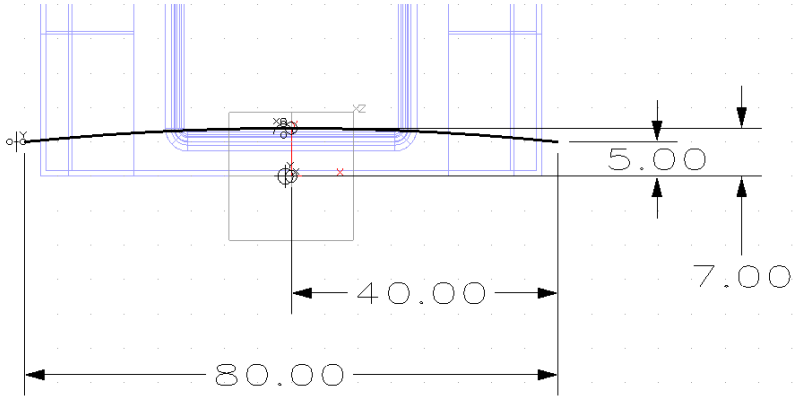
- Use “**Edit > Copy > External Sketch**” insert the sketch shown onto the **XZ**-Plane.
- The sketch is in the object file “After_Shave_Sketches” and the sketch is called “After_ Shave_Neck”.
- Select the **Revolve**  **> Add**  command.
- Revolve the neck as shown below.
- Create a 3mm fillet, shown in magenta.
- Create 1mm fillets on six edges, shown in yellow.
- Shell the part (minus) –0.8mm, top face open, as below.



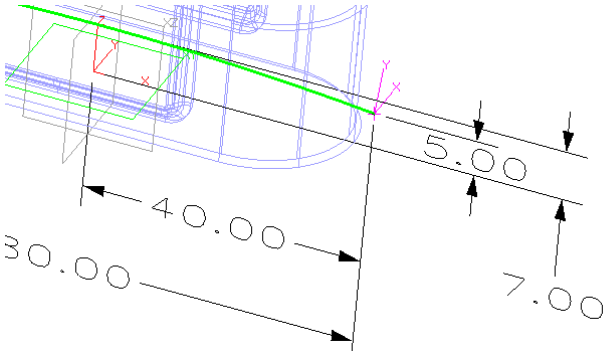



## Create the internal “pocket” bottom shape

- Insert the sketch copy shown on the XZ-Plane.
- The sketch is in the object file “After_Shave_Sketches” and is called “After_Shave_Sweep1”.

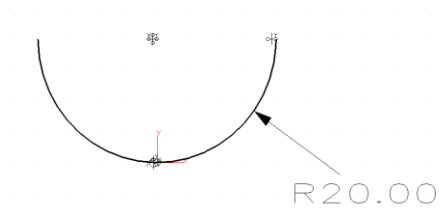



- Create the plane shown in magenta opposite.
- Place the new plane on the right end point of the curve, right mouse “critical”.



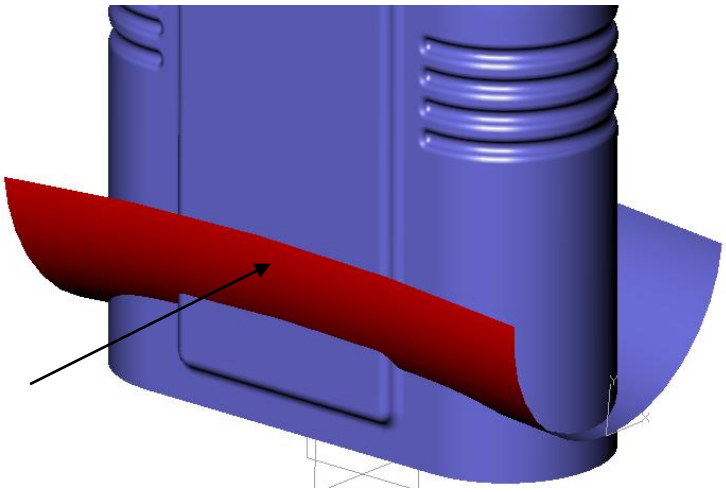

- Create a new sketch on the previously defined plane.
- Select the “Show target” button.
- Create a 3pt arc as shown, make sure the middle circumference point goes through the point at zero. Also make sure that the end points of the arc are aligned in Y.
- Create a radius dimension as shown.

- Use the “**Auto constrain**” command, pick the point at zero as the base point.
- Change the dimension value to 20mm if necessary.




## Create a swept face

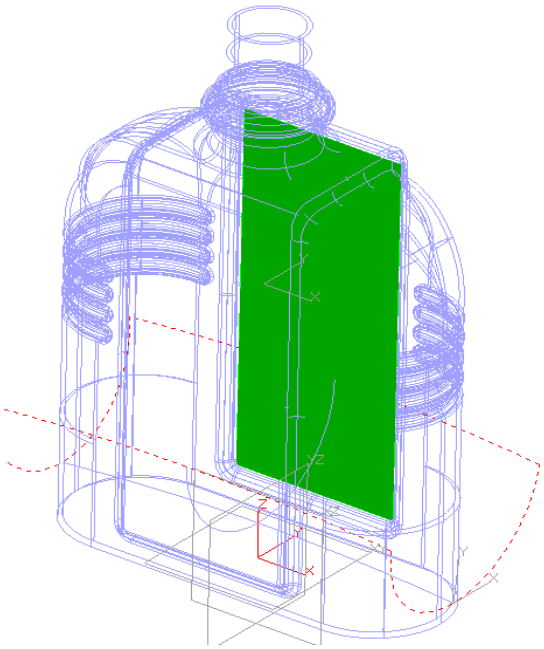
- Using the “**Sweep**” command , sweep the previously defined 20mm arc, along the sketch copy arc as shown opposite.
- Before proceeding, we need the **red** side of the surface as shown.
- If necessary Reverse the normals.



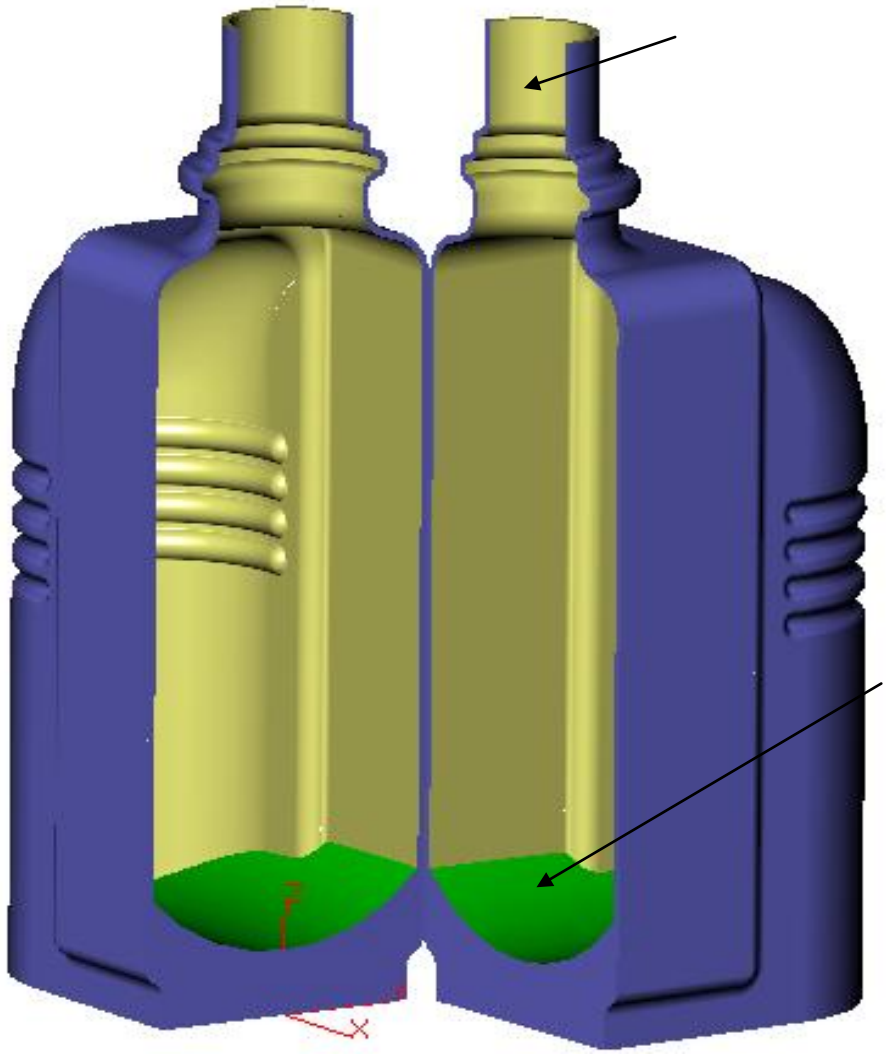
## Use a “Bounded Boolean” to add the face to the bottle.

We need to replace the inside bottom faces of the bottle with the swept surface shown. Traditionally this type of task has always been quite difficult to achieve. It would usually involve lots of surface trimming. Fortunately we can use a Combine shape > add command, with the Bounded face option, to simply add the swept face to the existing solid.

- Select the “**Combine shape** **> Add** ” command.
- Pick the **bottle** as the base shape.
- Pick the **swept face** as the face to be added.
- Pick the **inside face** of the bottle as shown opposite in green, **as the bounded face**. Tip: in order to pick the correct face, zoom in closely and using the “Alt” key, pick the face at the edge of the small internal radii on the inside of the shell.
- If you don’t get the correct surface the first time, use Right click “Redefine Last”
- Complete the rest of the command.
- The swept face is now “added” to the bottle as shown on the next page.

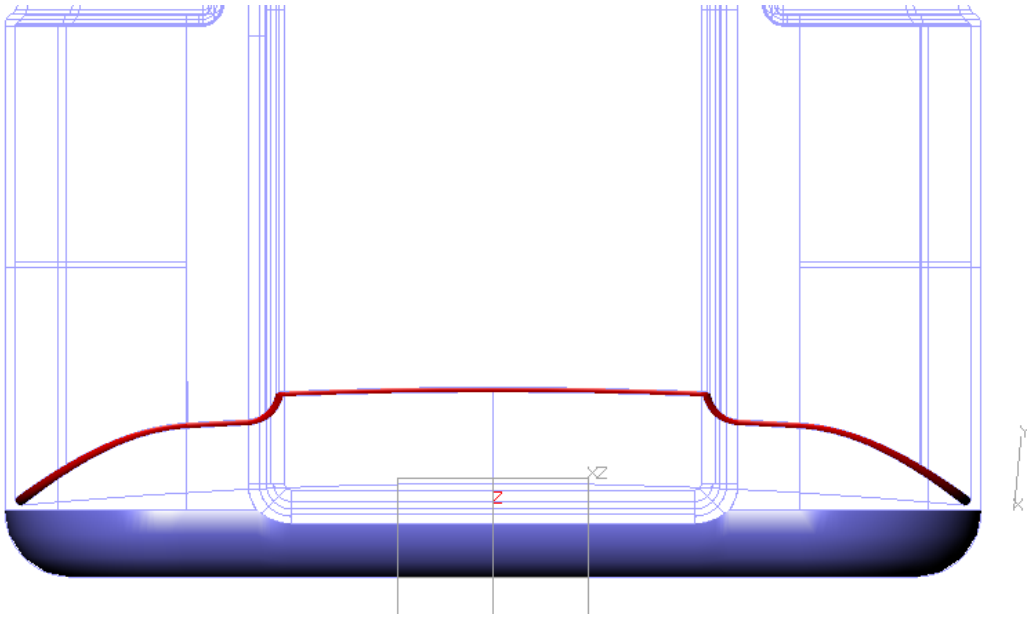


The Figure below, which has been divided in two for clarity, shows the bounded face (in green) which has been added to the inside faces (in yellow) only.

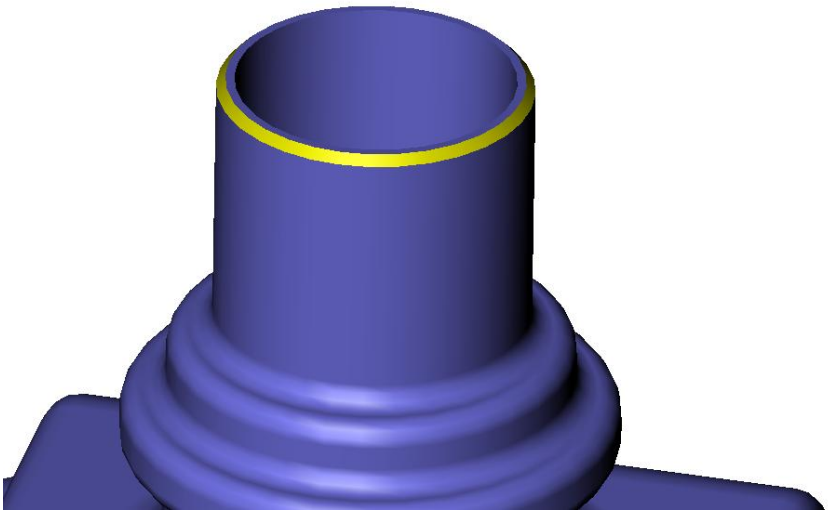


## Create fillets and a chamfer as follows.

- Create a 0.5mm fillet on the bottom inside edges between the swept face and the inside bottle faces, as shown below in red. Use chain pick to select all the edges.
- Create a 5mm fillet on the bottom external edges, as shown below in blue.



- Create a 0.4mm chamfer on the edge shown opposite in yellow.

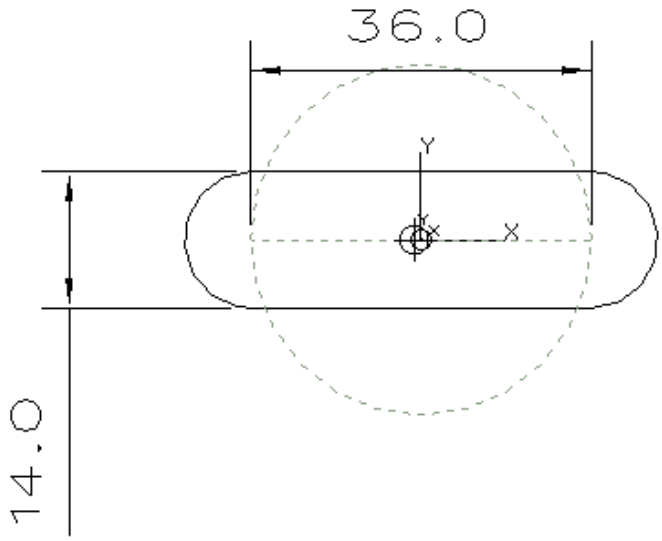


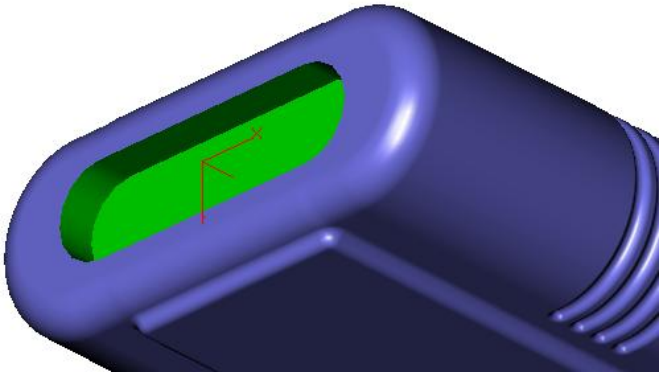
**Create an extrude > remove on the bottom face of the bottle.**

- Select the “Extrude > Remove” command.
- Create a sketch on the XY-Plane as shown opposite.
- Next using the **slot** command in **Ready Sketch**, create the shape opposite.
- Change the dimension values as shown.
- 
- Complete the **extrude > remove**.

- Start = **0**.
- End = **6**.

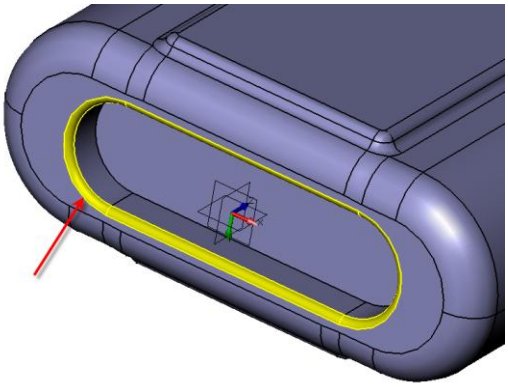
As shown opposite in green.



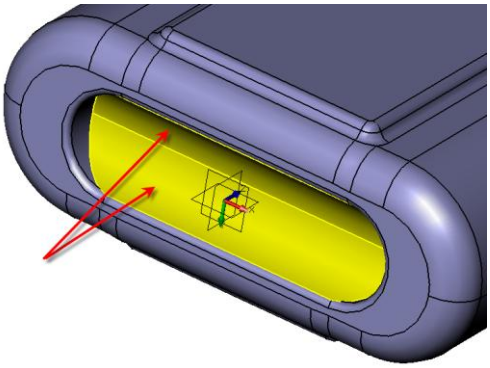



## Create Fillets on the extruded bottom cut of the bottle

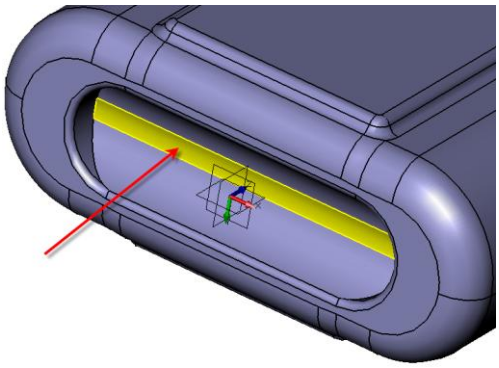
- Start with a 1mm fillet as shown opposite in green. 
- Using “Chain” select to pick the edges.



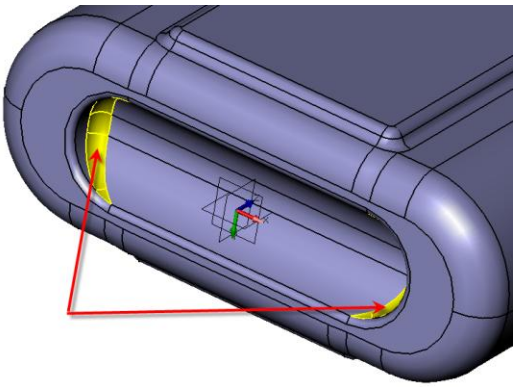
- Next create a 9mm fillet as shown opposite in green.
- Pick the two straight edges on either side of the base of the extrude > remove.
- Notice that the fillet produces a sharp edge in the middle, because the radius is too wide for the slot.



- Create a 5mm fillet on the “sharp edge” as shown opposite in green.



- Finally create 2mm fillets on both sides, as shown opposite in green.
- Make sure you use **chain** select to select all the relevant edges.



Save your file “After_Shave.Z3”.

## Example four, The Softener Bottle

## Some of the topics covered.

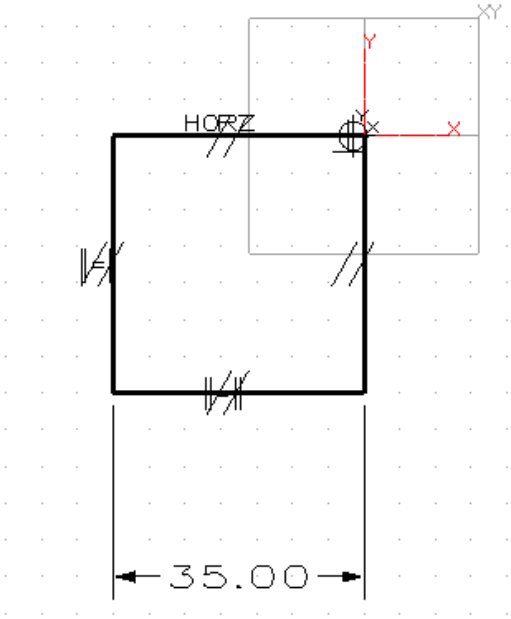
- Sweep.
- Surface Mesh.
- Equation sets.
- Local Variables.
- Variables from the result of the Inquire command.
- Copy Shapes and the Combine option.
- Creating threads and thread run-outs.

## Create a multi object file called “Softener.Z3”

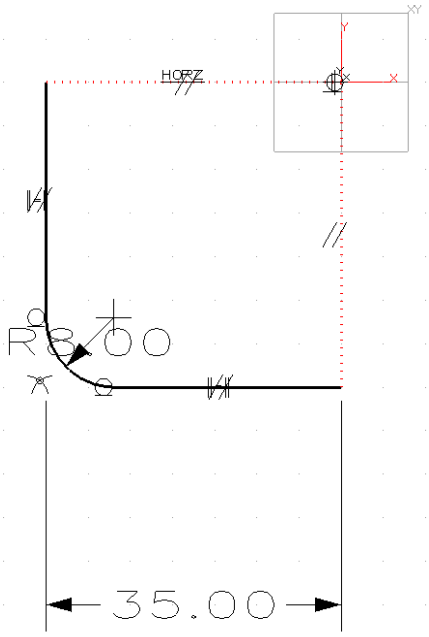
## Create a new part called “01_Softener_Bottle”

This bottle is symmetrical and sometimes it is convenient, as in the previous examples, to model one half of the part. In this example as in the previous example, we will first model one quarter only.

- Create a new sketch on the **XY**-Plane.
- Dimension and constrain the sketch as shown opposite.
- Note the “**Equal length Constraint**”  applied to the length of two lines. (You have to erase one of the 35mm dimensions first)
  This can be on either the two lines that join in the lower left corner or the two lines in the upper right corner, because of the fillet added below.
- The 35mm dimension will now control both sides of the box.

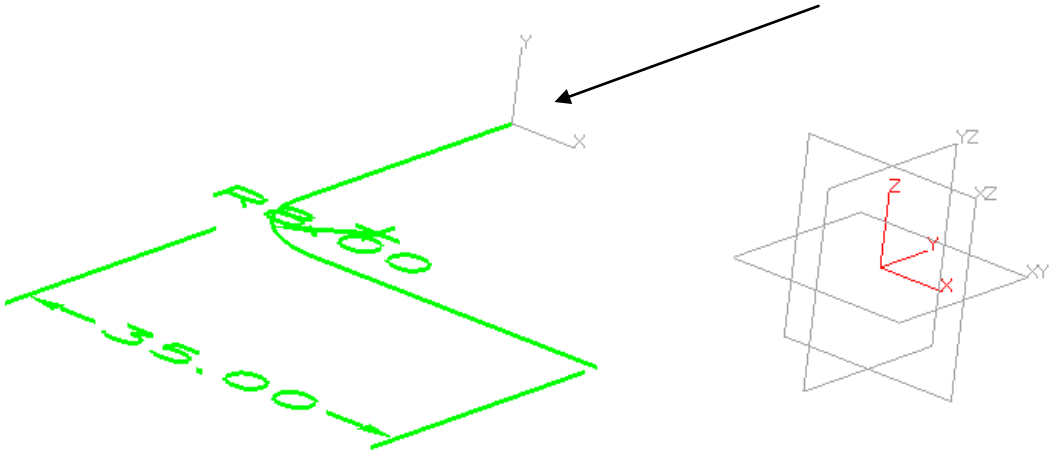



- Create an 8mm fillet as shown.
- We only need the two lines and the fillet arc when in the Modeler.
- Toggle the two other lines into reference geometry, as shown in dotted red opposite. (Pick the two lines then **Right-Click > Toggle type**)
-  Rengen and exit  .

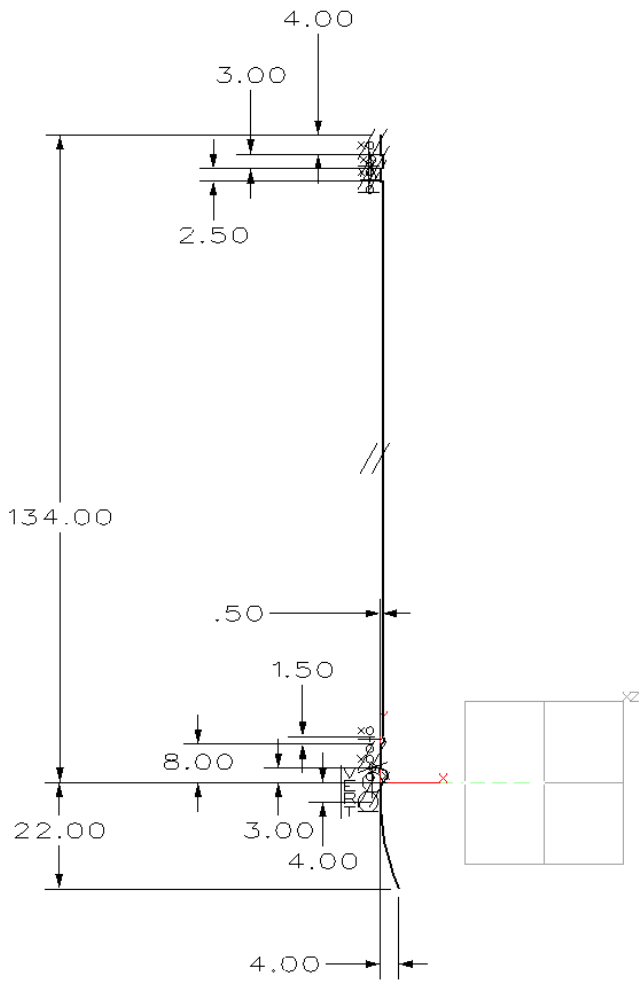



**Create a new plane at the end of one of the sketch lines.**

- Select the "**Insert Datum**" command,
- Pick the **XZ**-Plane.
- Locate the **origin** at the end of one of the sketch lines as shown below.

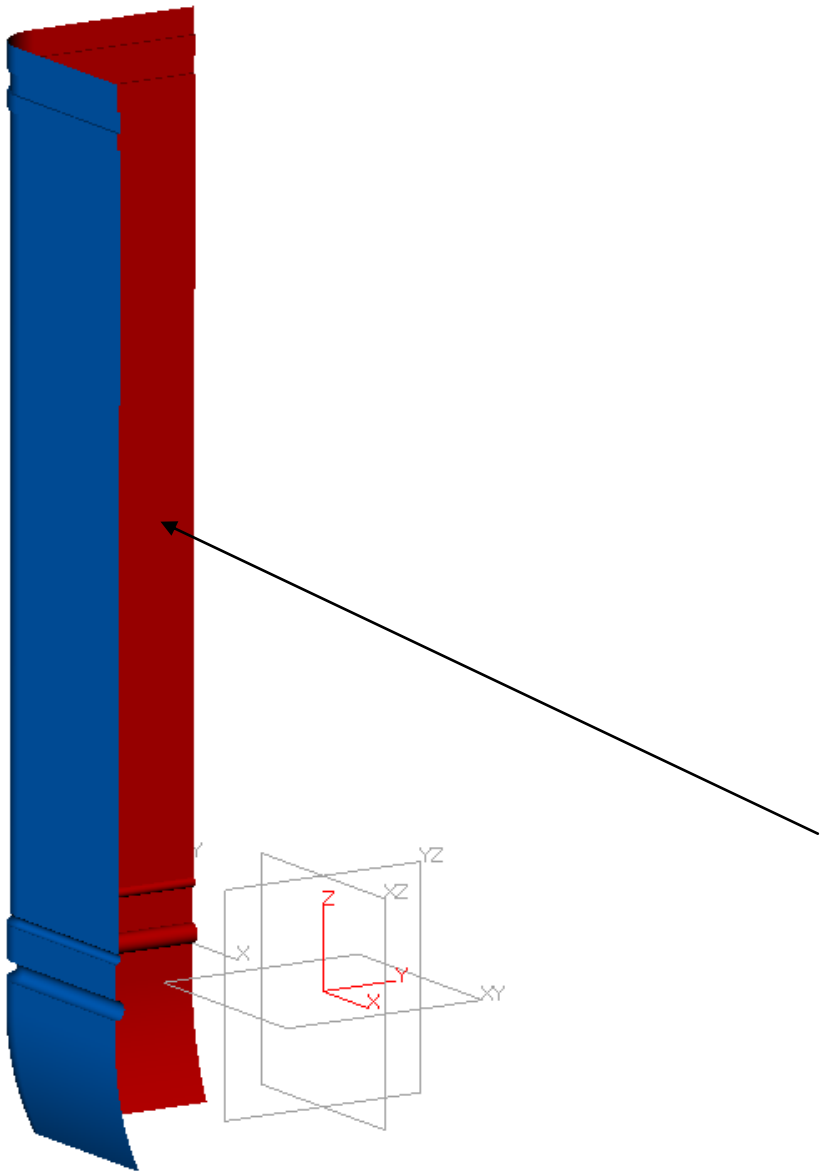



- Insert a sketch copy from the file "Softener_Sketches", the name of the global sketch is "Softener_one".
- Locate the origin of the sketch copy at the zero position of the previously created plane, as shown opposite.




## Create a “quarter” bottle shape using a surface sweep.

- Select the “**Sweep**” command and sweep the sketch copy along the first sketch.
- Make sure you have the pick filter set to “sketch” so that the entire sketch geometry (with exception of the reference curves) is used for the sweep path.
- When the sweep is complete you may notice that the face normals are facing the wrong way (pink color towards the outside”.
- If so, select the “Reverse Normals” command to flip the face normals as shown opposite.




## Create two additional datum planes

- Select the "**Insert Datum**" command.

Note the part has been rotated for clarity.

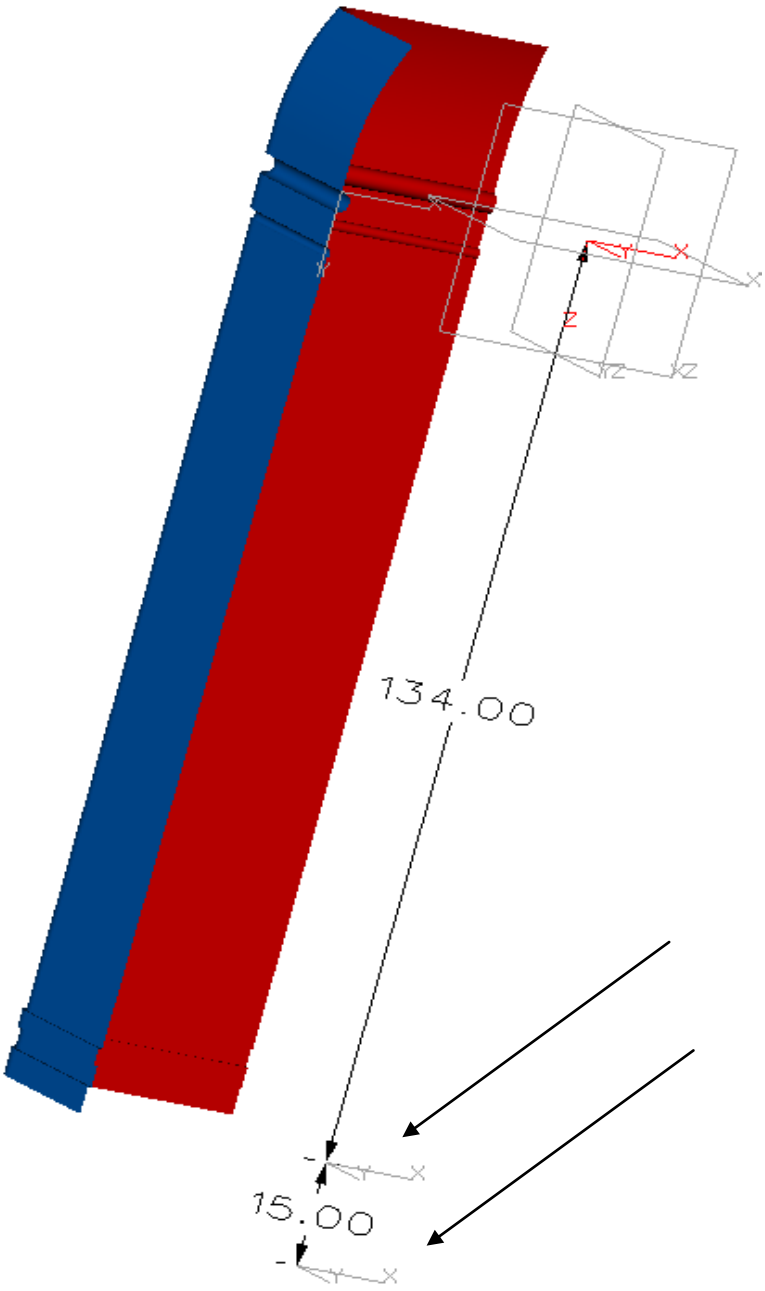
Create the first plane.

- Pick the **XY**-Plane.
- The end distance is to an edge end point at the top of the swept shape. Select the "**To Point**" option; with the right mouse and pick the end point as shown (**134**mm dim.)

Create the second plane using the same command.

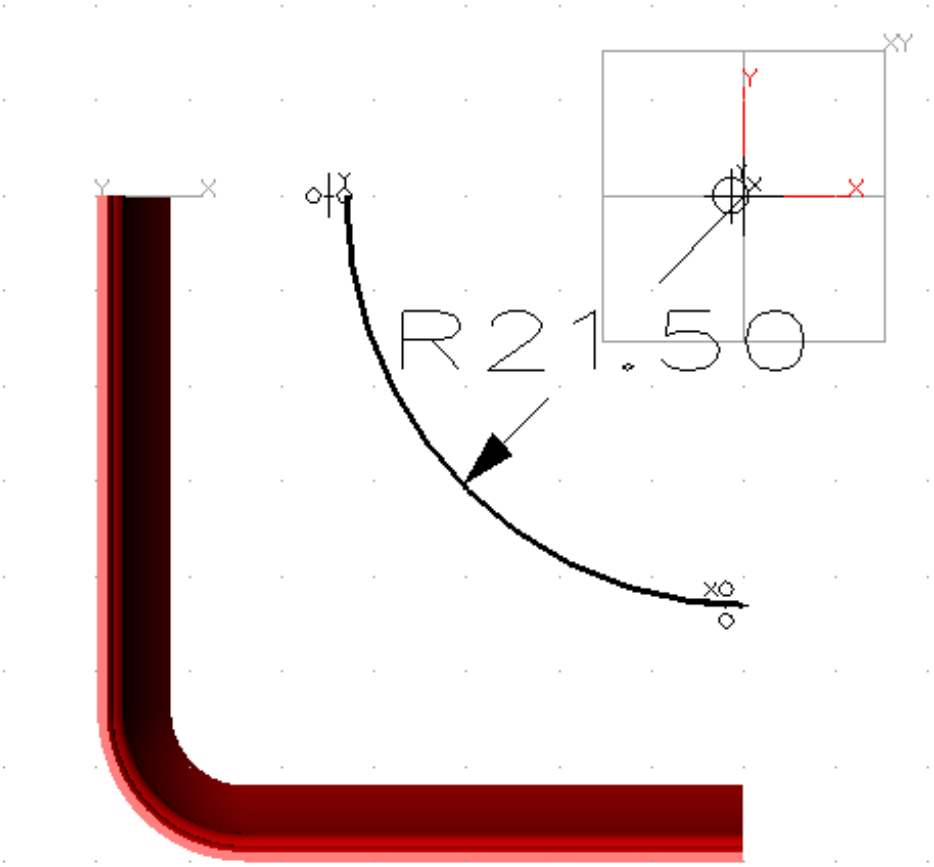
- Pick the previously created plane.
- Enter **15** as the distance.

Both planes are shown opposite.



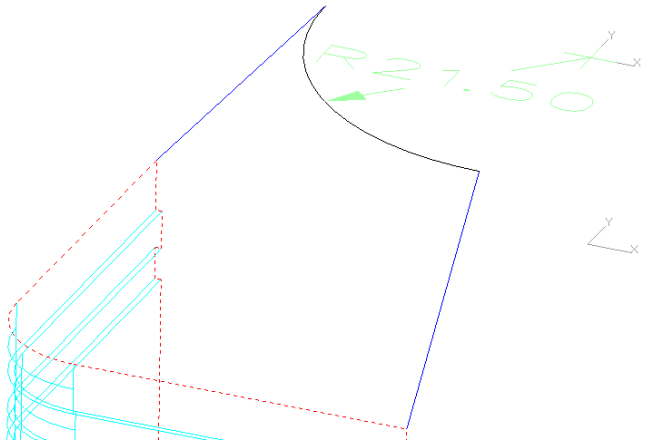
## Create a sketch for one quarter of the neck shape.

- Create a new sketch on the second plane previously created, shown with a 15mm dimension on the previous figure.
- Create an arc “center”, with the center point at zero.
- Add the radial dimension to finish constraining the sketch.
- Modify the radius value as shown below.
- 

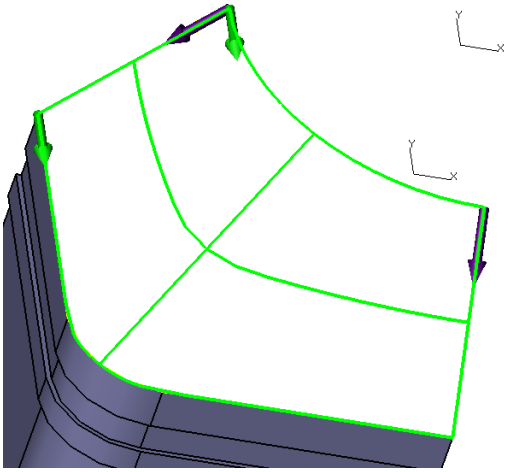


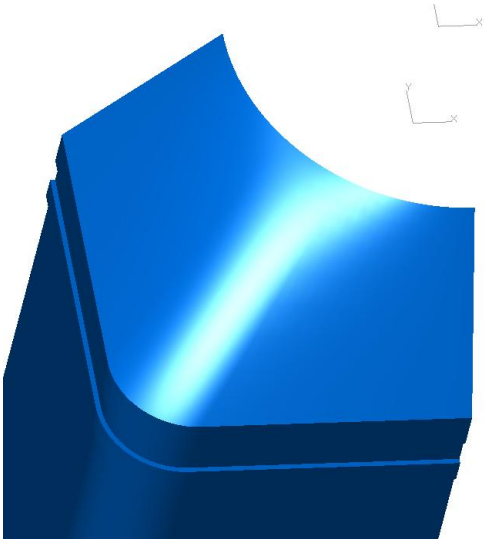

## Continuing with the neck shape.

- Create **two wire-frame lines**, from the end points of the last sketch and the end points of the face edges shown opposite.




- Select the "**Curve Mesh**" command. 
- Create the mesh face shown below.
- For the Ucurves pick the two lines on the same side relative to the arc.
- For the Vcurves, pick filter set to curve, pick the arc then Right-click Insert Curve List, pick the three edges of the model.
- Make sure that you don't pick **Keep curve**, if you picked, erase the wire-frame geometry (pick filter set to curve).
- Notice the mesh face is joined automatically to the existing swept shape and that the surface normal orientation is maintained.



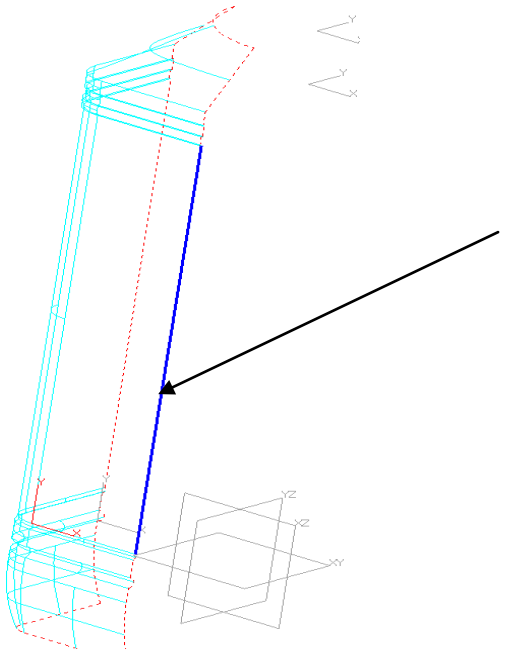


## Creating the cut out features on the 8mm-radius part of the bottle

We wish to create seven slot type features equally distanced on the 8mm-radius part of the bottle. Also if the height of the bottle were to change, the distance or pitch between the slots must change automatically. To accomplish this we will use a **local** variable and an equation set. Why this variable is considered "local" will shortly be explained. First, let's define a variable that is associated to the length of one of the edge curves on the bottle.

ZW3D will let the user attach an optional variable to the result of some of the "Inquire" commands. This will force the inquire operation to be logged into the history. So if the part geometry that was used in the "Inquire" command changes parametrically, the variable(s) will also update accordingly.

- Select the "**Inquire > Distance**  **> Point to Point**  " command.
- Select the "**Variable Name**" option and enter the name "**edgelen**".
- With the right mouse option set to critical and the pick filter set to edge, pick the edge end points on the longest vertical edge of the bottle, as shown in blue below. The distance of the two end points is currently 115mm.
- The operation "InquireDist1" will now be inserted into the history.



## Create an equation set to define the pitch between slots

The next step is to define the pitch. In our example we require seven slots, so we need to divide the “edgelen” local variable by eight. We could easily accomplish this using another local variable, however we will take this opportunity to use an equation set and also to discuss the term “local variable”.

- A complete description of equation sets can be found on the HELP Manual.
- Create a new Equation set.
- All equation sets are shown in the equation set form, which is shown automatically each time you pick “Insert > Equation set”.
- If you wish to view or modify an existing equation, use the same command as before, picking the relevant equation set from the equation sets form.
- The equation set form will now be displayed.
- Place the mouse pointer in the “Name” input field and “click”, to activate it.
- Enter the name “Pitch”, do not press enter at this stage.
- Activate the “Expression field”.
- We need to refer to our local variable “edgelen”.
- Press the “Get Variable” button.
- The Variables form will now appear.
- We can use this form to browse.
- It should already be set to “Softener Bottle”
- We can now see all the “edgelen” variables that were previously created with the “Inquire” command.
- Select the variable “edgelen_D”.
- The variable name will now be entered into the expression field in the equation set form.
- Append “/8” to the end of the expression as shown below, **do not** press enter at this stage.

## Local variables and expressions with part names, in equation sets.

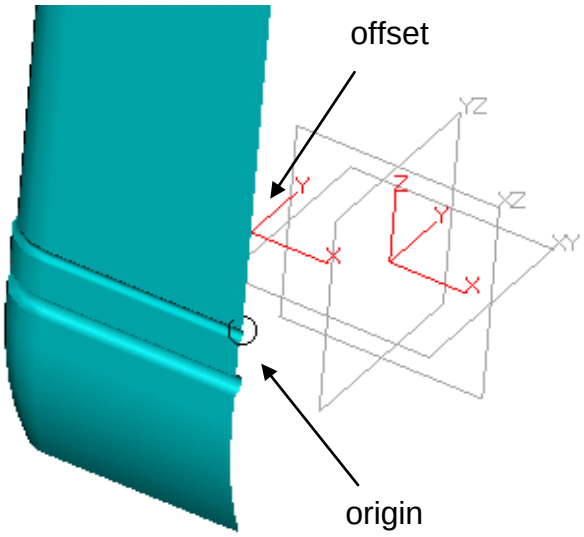
The following text is currently in the expression field. “edgelen_D/8”.

**Please at this stage take the opportunity to review the equation set form from the on line help.**

- Press the “**Add Equation to List**” button to enter the variable “Pitch”.
- Notice the value has not yet been calculated.
- Press the ”**Apply**” button.The equation will be saved and the value of “Pitch” will be displayed. (see below)
- Press **OK.**

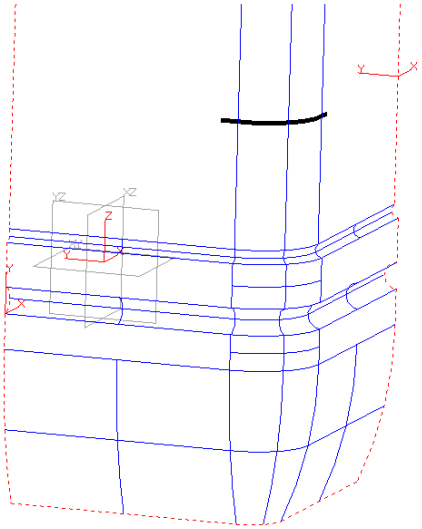
## Continue with the seven slots.

- **Create a plane** located at a point.
- Pick the **XY**-Plane as the reference plane.
- Pick “**Origin**”
- Pick an edge end point at the lower end of the same vertical edge that was used for the “edgelen” variable, as shown opposite.
- For the **Offset** distance we want to use the variable “Pitch”.
- Right mouse, select “**Expression**”.
- From the “Input Dimension Value” form, select “**Select Variable**”
- From the “Variables” form select “**Softener Bottle**”.
- Notice that the “edgelen” variables are accessible here.
- Select the **EquationSet1**
- Select the variable “**Pitch**”.
- The variable is shown below in the “Input Dimension Value” form.
- Press the “**OK**” button to accept the expression as the distance value.
- Press **OK and Cancel the Browser**



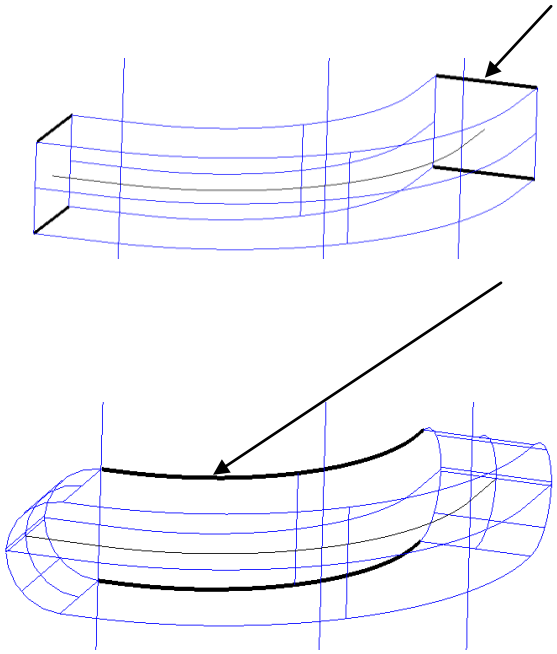

## Create an intersection curve.

- Create an intersection curve between the previously created plane and the “fillet” face as shown opposite. Make use of the correct pick filters.
- Extend the curve linearly both ends 1.5mm.

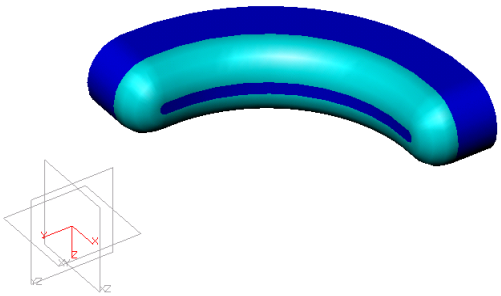


- Select the “extruded base” command.
- Right mouse select “Insert curve list”, pick the curve.
- Select a point on the profile to indicate the “interior”.
- Start value = 1.5mm.
- End value = -1.5mm.
- Draft angle = 0.
- Select the “Offset” option from the option buttons.
- Select “Thicken same”.

- Enter 1.5 for the offset distance.
- Accept the defaults for the rest of the command.
- The new solid is shown opposite.
- Create 1.5mm fillets on the four  short edges as shown
- Create 1.25mm fillets (chain select) on the “inside” edges shown in magenta.



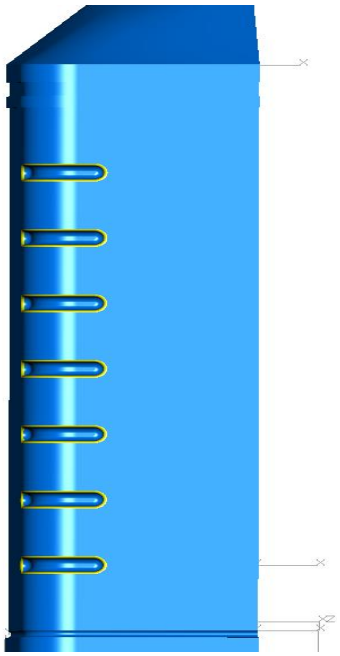
## Create the slot “solid”.



## Create the seven slots.

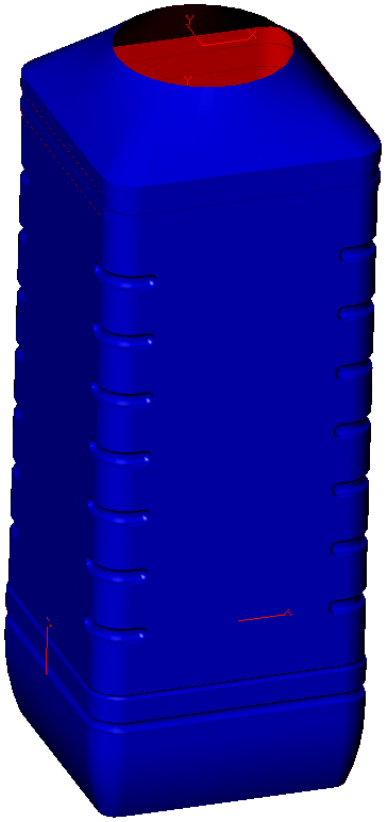
- Select “**Edit > Copy > Copy Entities**” and pick “**along a direction**.”
- Pick the new shape.
- The copy direction is the **Z** Axis (right mouse button).
- The distance between copies is the “**Pitch**” variable.

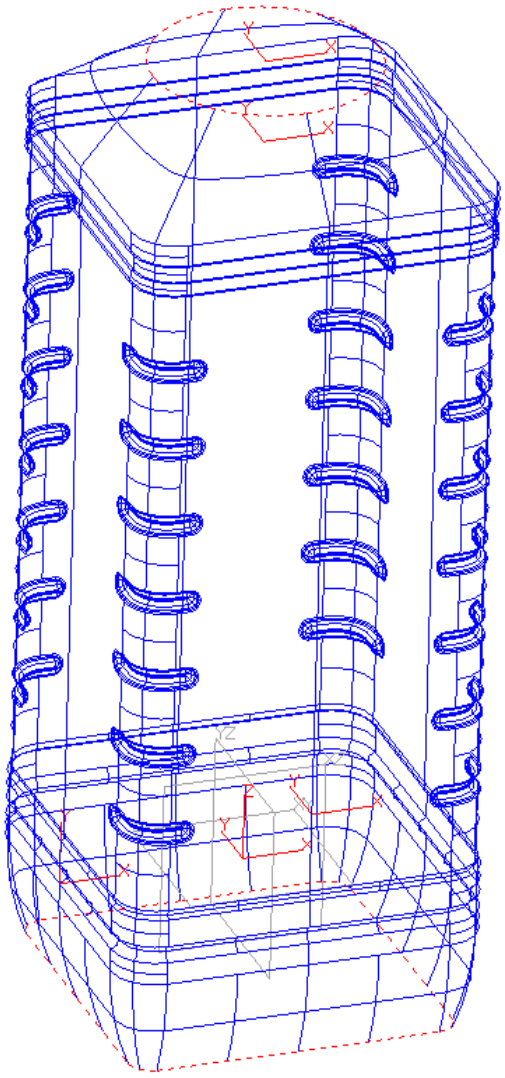
- To enter this use the right mouse button and choose “**Expression**” as before, or simply type “Pitch” in the normal text input window. (not the menu form), Hit <Enter>.
- Rotation angle = **0**.
- From the option button “**Copies**” enter “**6**”.
- Next erase the wire-frame curve.
- Next using the “**Combine Shapes > remove**” command remove the seven shapes from the original swept shape.
- Now create **0.5mm fillets** on the edges between the original bottle shape and the slots (chain select) .



## Copy the Shapes to make the complete bottle shape.

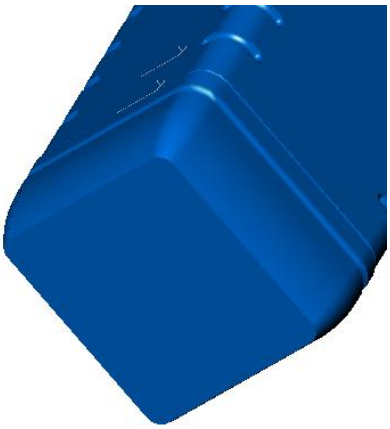
- Pick “**Edit > Copy> Copy Entities**” and pick “ **Along a direction**”.
- Set the pick filter to **shape** and pick the existing bottle shape.
- Direction = **Z** axis (right mouse button).
- Distance = **0**
- Angle in degrees = **90**.
- Copies = **3**
- Pick “**Sew**” to close the seams.





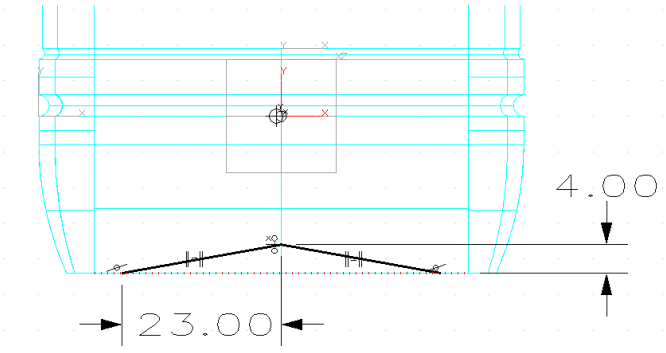
## Close the top and bottom faces of the bottle.

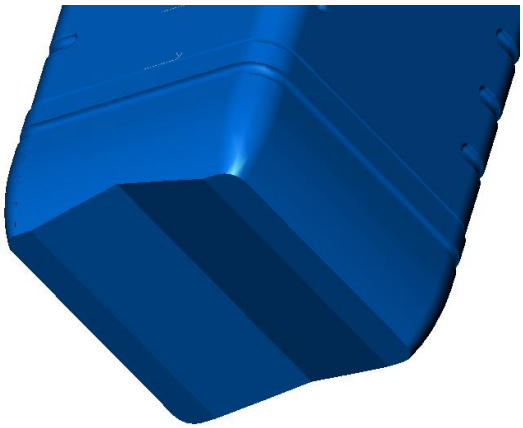
- Create a **planar face** on both ends (chain select) of the bottle.
- Suggestion: "**Heal > Fill gaps**" and just pick "**OK**"



## Create a "Vee" shaped cut-out on the bottom face

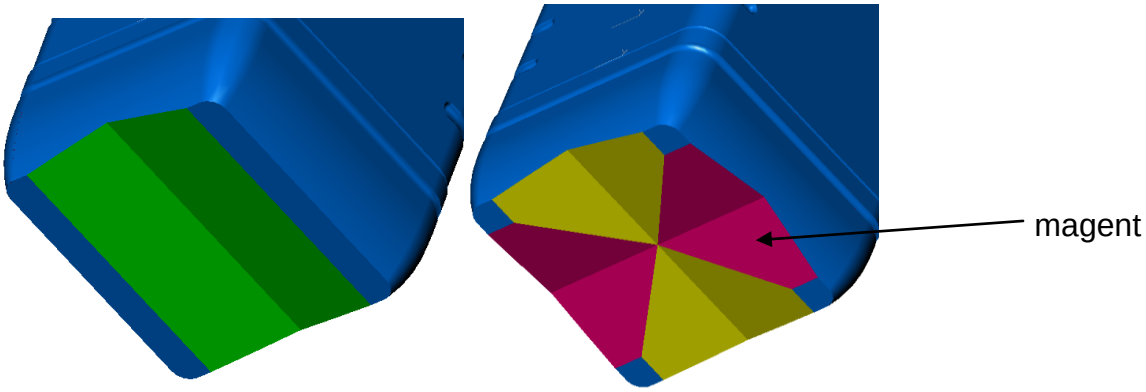
- Select the "**Extrude > remove**" command.
- Start the sketch on the **XZ**-Plane.
- Create a reference curve from the bottom edges as shown in black opposite.
- Create the two lines shown; align one end point in "X" with the zero point. The other end points of the two lines are "on" the reference curve (right mouse option "Default").
- Dimension the part as shown; notice the equal distance constraint, which controls the length of one of the lines.
- 
- Start of cut = **50**.
- End of cut = **-50**, as shown opposite.

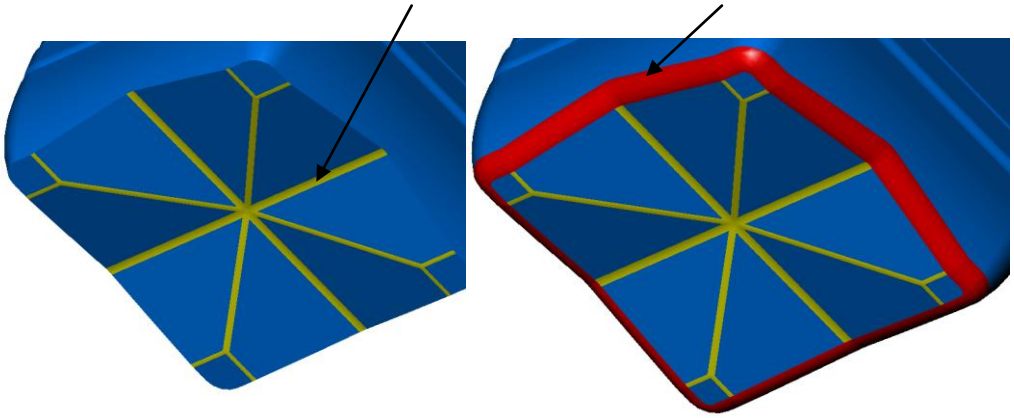




**Create a circular pattern feature and fillet the edges, to complete the bottom cut-out.**

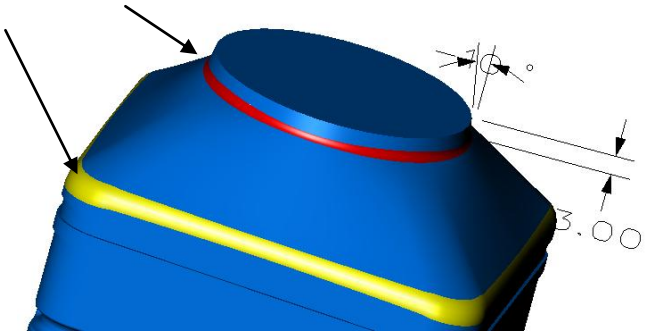
- Pick **Pattern** **> Circular** from the Shape tab.
- Pick the cut-out **feature** as shown opposite.
- Rotation axis (right mouse) = **Z** axis.
- Total number of feature = **2**.
- Spacing in degrees = **90**.
- The result is shown opposite in magenta.
- Create a **5mm fillet** to 16 edges as shown below left.
- Create a **4mm fillet** on the outer edges (chain select) as shown below right.





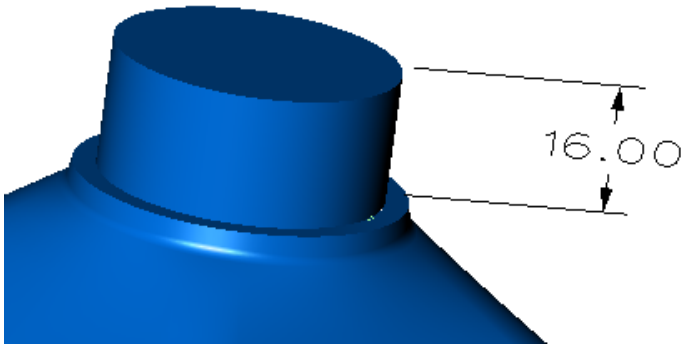
## Create a tapered boss at the neck end of the bottle.

- Create a **10** degree tapered boss, **3mm high** as shown opposite. Note the 10 degree boss tapers in (in other words the boss is smaller at the top.)
- Right mouse, "**Insert Curve List**".
- Create a **4mm fillet**, as shown.
- Create a **2mm fillet**, as shown.



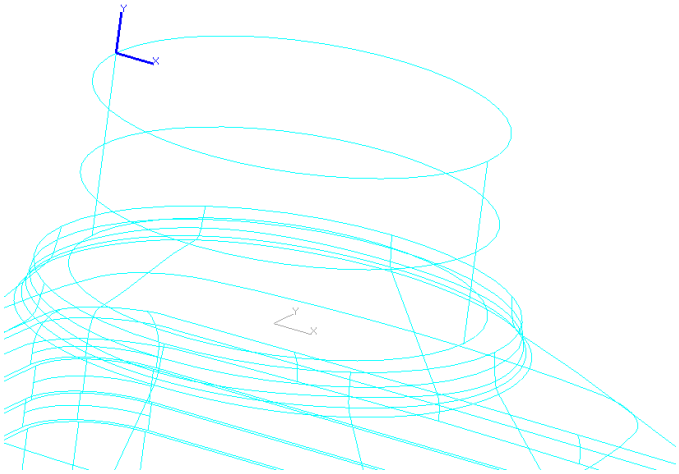

## Create a parallel boss at the neck of the bottle

- Create a boss **16mm high by 36mm Diameter.**
- Use the **Primitive cylinder** shape.

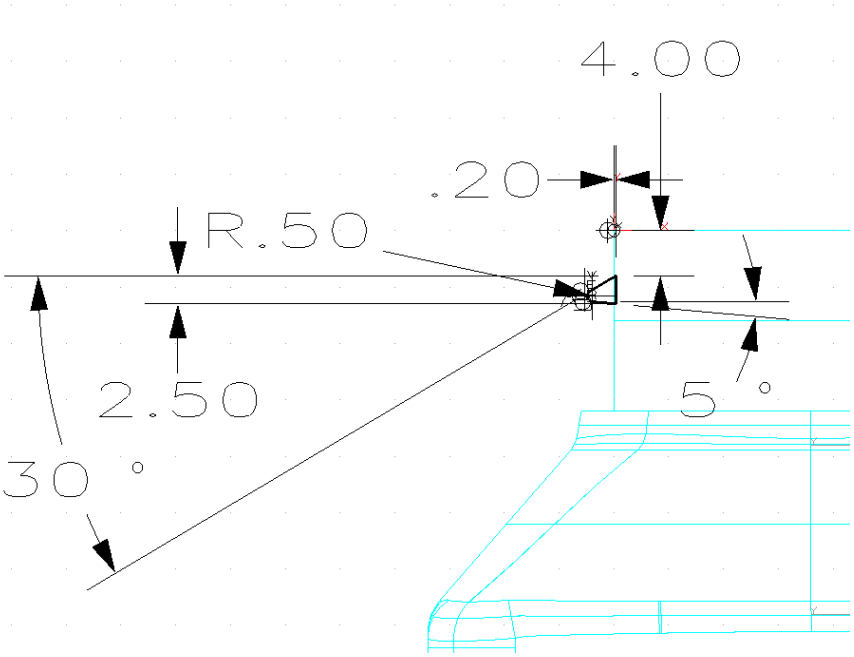



**Create a plane and import a sketch copy, which will be used for creating a thread on the neck.**

- Create a **plane** located at a point.
- Select the **XZ**-Plane.
- Pick "**Origin**".
- With the right mouse option set to "middle", pick the top edge of the bottle neck to locate the new plane, as shown opposite.



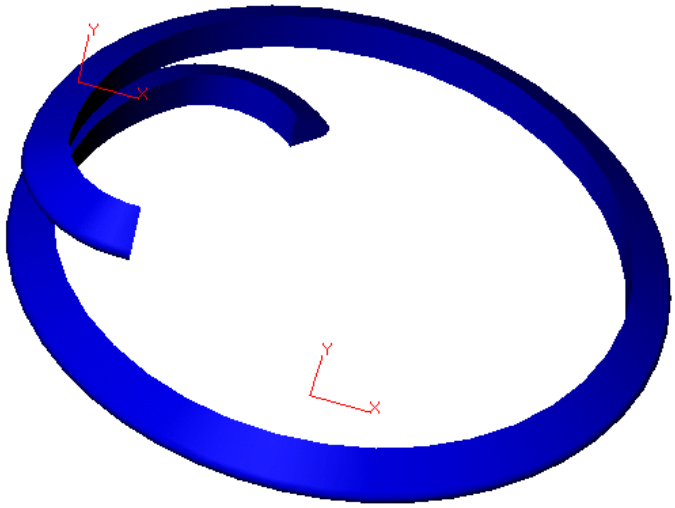
- Insert a sketch copy from the "Softener_Sketches" file. The name of the sketch is "Softener_two".
- Locate the sketch copy on the previously defined plane, as shown opposite.




## Create the basic thread form as a Boss.

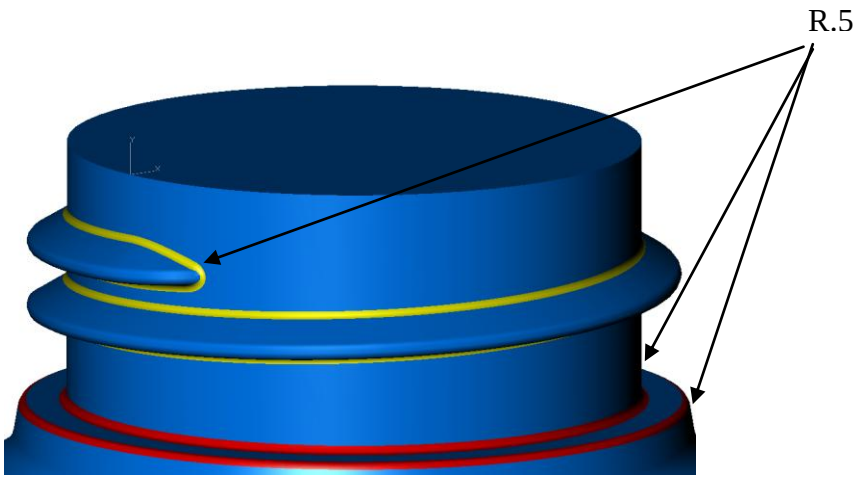
Create a new solid boss.

- Select the "**Spiral sweep > add"** command.
- Pick the new sketch copy profile.
- Revolve axis = -Z axis (note **minus Z**).
- Number of turns = 1.125
- Distance between turns =4.
- The "Lead" option is "In".
- Select "Both" so that we get a lead in at the start and end of the thread.

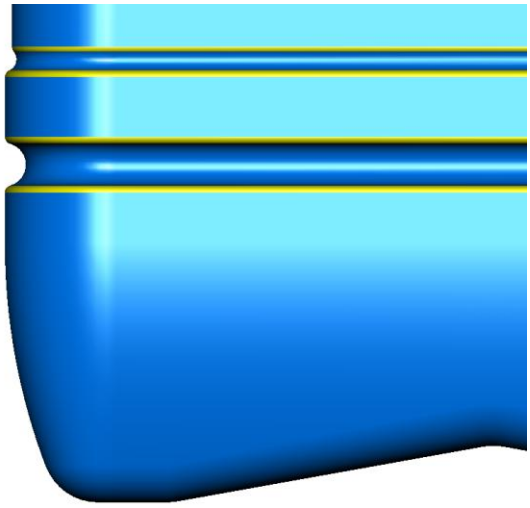


## Add fillets to various edges.

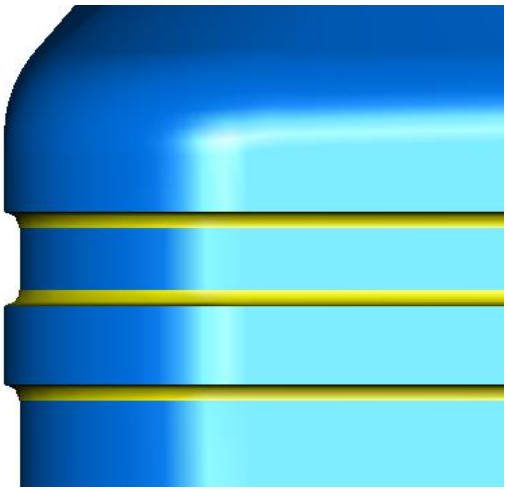
- Create the **0.5mm fillets.**



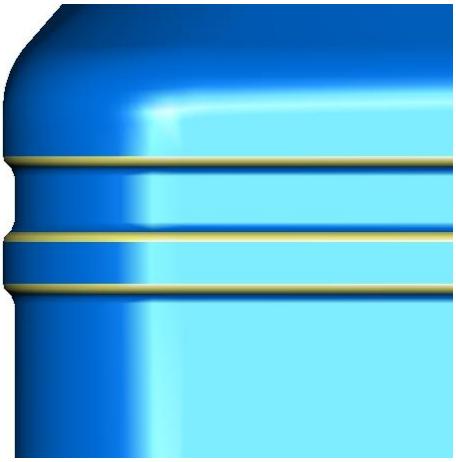

- Chain select the lower edges shown, and create a **0.4mm fillet**, as shown opposite.



- Chain select the upper concave edges shown, and create a **0.4mm fillet** as shown below.



- Chain select the upper convex edges shown, and create a **0.4mm fillet** as shown below.



**Create a shell (minus) –0.25mm, leave the top face open, as below**

Save the file as Softener.Z3.

## Example five, The Bleach Bottle.

Some of the topics covered are:

- Splitting faces & the offset face command.
- Using faces to Boolean solids.
- More surface operations such as lofts, N-blends, revolve etc.
- Using the draft command with the split face command to create special chamfers.
- Using components in your bottle design.

## Create a multi object file called “Bleach.Z3”.

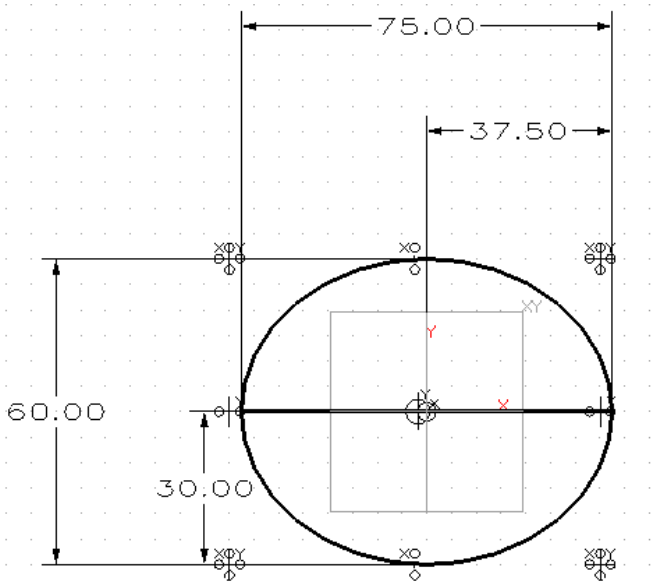
## Create a new part called “01_Bleach_Bottle”.

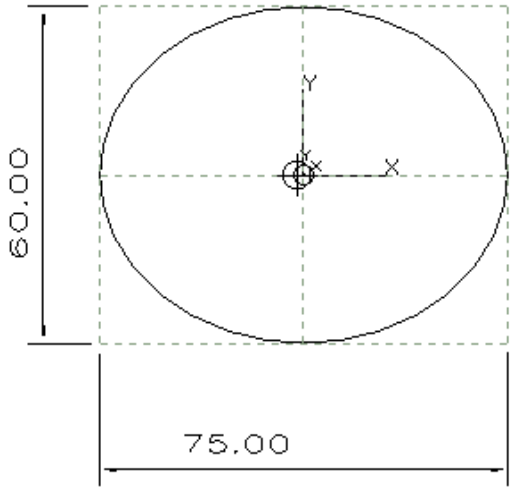
This bottle is symmetrical and we will first model one half of the bottle only.
The main body of the bottle shape is elliptical. We already have a global sketch saved in another file. We can use this sketch in our new bottle.

- Select “**Edit** pull-down menu **> Copy > External Sketch**”.
- From the “Conditioner sketches” file, select the “ellipse” sketch.
- Locate this sketch on the XY-Plane.
- Edit the sketch.

- As an alternative, if you didn’t pick this up in one of the previous classes, ZW3D also has a sketch clipboard. It is located in the user\resource directory and should be defined in your configuration file. There is one for mm and inch units. Sorry, we did not create one in miles!
- To access it, open a sketch and right click “**Paste**”.
- Another solution for accessing predefined sketches is to use “**Ready Sketch**” that can be obtained from the tab. It contains a wealth of “well defined” sketches. See below left. Most of the constraints are blanked for clarity.

- Change the dimension values of the major X and Y dimensions to 75mm and 60mm.
- **Create a “two point” line** on the X-axis and on the ellipse at each of the end points of the line, as shown opposite.
- We require one half of the ellipse only, so use the trace profile command to create the profile on the lower half.

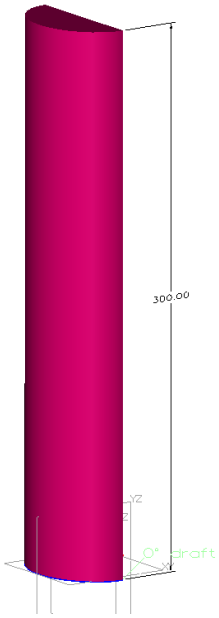


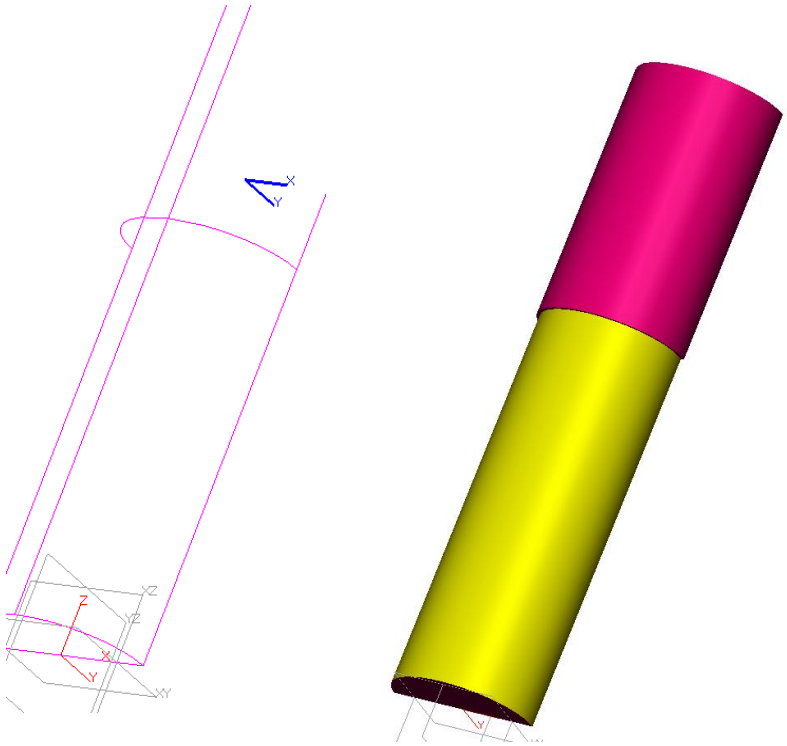



## Create the first extruded shape, split and offset the curved face.

- **Extrude** the trace profile as shown. 
- Start =**0**
- End =**300**.
- Create a **plane** offset from the **XY**-Plane **180**mm.

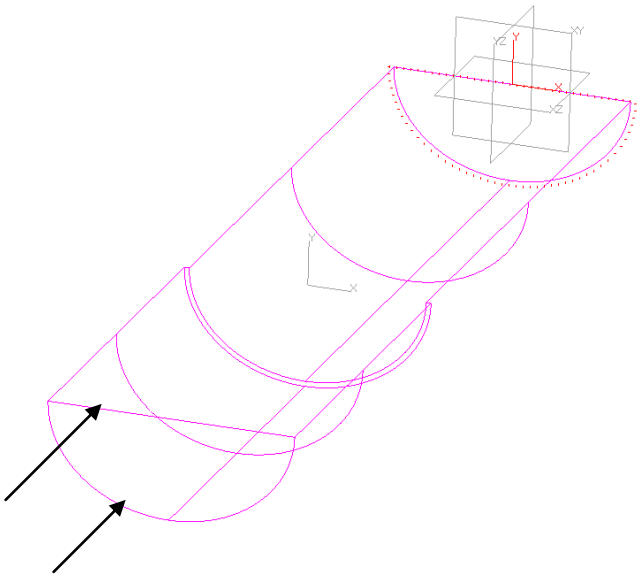
- Use the "**Split with Faces**" command  , to **split** the curved face (not the back planar face) with the new plane. Make sure you use the correct pick filters.
- Select the "**Face Offset**" command and offset the lower curved face (not the back planar face) inwards (minus) –1.5mm as shown below in yellow.



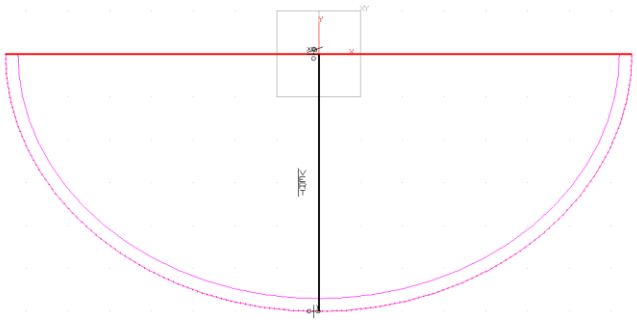


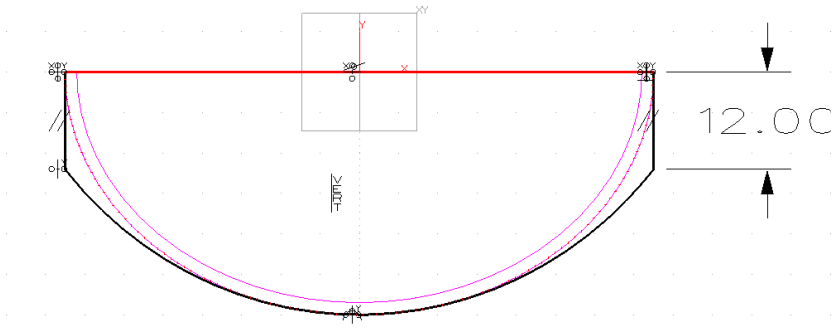
## Create an extruded boss at the end nearest to the XY-Plane.

- Select the **Extrude** > **Add** command.
- The sketch plane is the **XY**-Plane.
- Create two reference curves from the elliptical edges at the “large” end of the bottle (the face furthest from the XY-Plane).
- These two curves will be projected onto the **XY** sketch plane as shown in red dotted opposite.

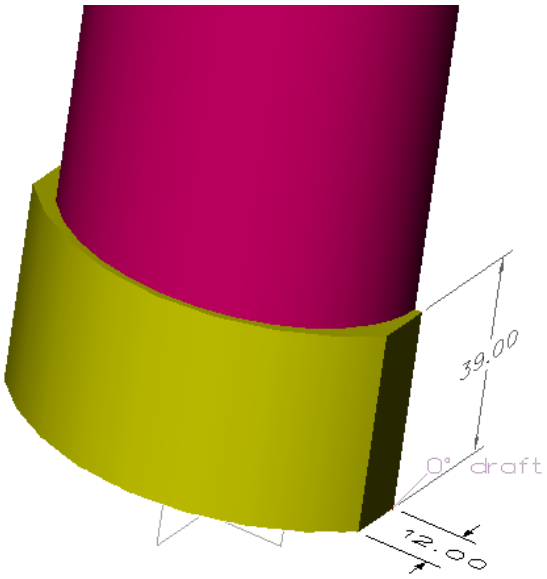


- Toggle the horizontal reference line into normal geometry, as shown opposite in red. (not Unlink)
- Create a vertical line from zero to a point on the elliptical reference curve (right mouse default), as shown opposite in black.
- Add the two constraints shown opposite.
- Toggle the vertical line from normal to reference geometry.
- Create two vertical lines of the same length as shown.
- Next create a three point arc (quick draw), through the two end points of the previously created vertical lines and the end point of the vertical reference line as shown opposite in black.
- Change the value of the vertical dimension to 12mm.
- 






- Complete the extrude > add command.
- Start = 0.
- End = 39.
- As shown opposite in yellow.




## Create an angled face on the step between the boss and the ellipse shape

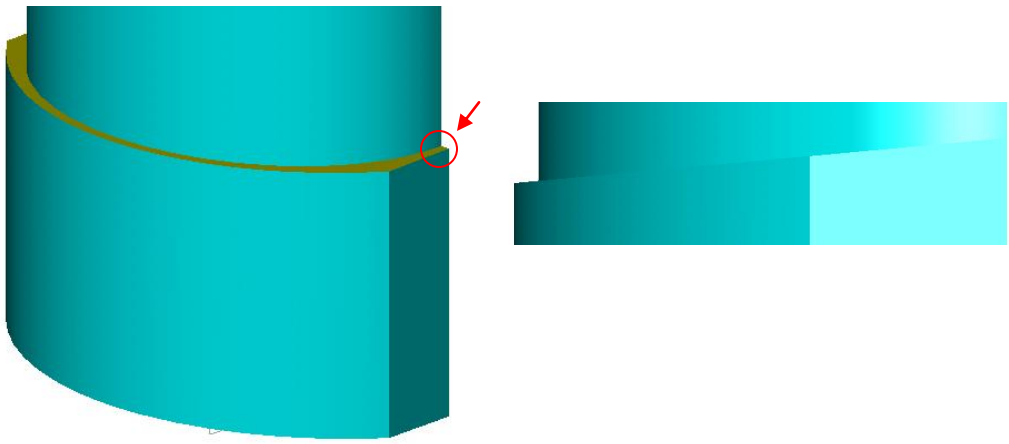
We are going to show you two methods. One easy, the other is to show some other commands that you will need in other applications.

## METHOD 1

Use the **Draft** command from the Shapes tab. 

To define what we are going to draft about, change the pick filter to EDGE and pick one of the short edges (which is shown below )created by the extrude.

The angle is 1.0 degree and pick Y axis.

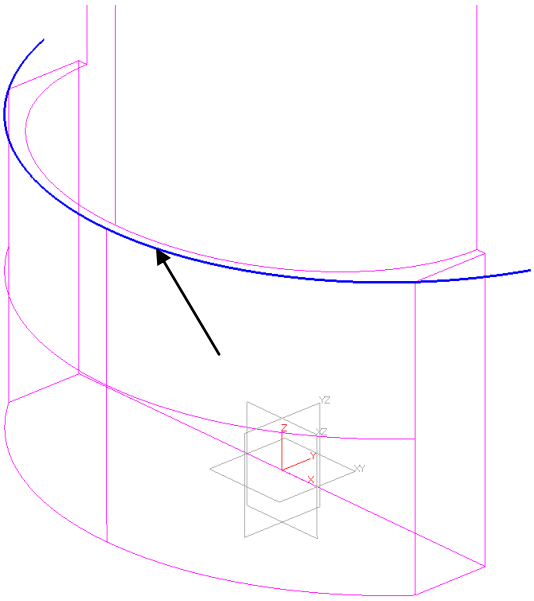


UNDO this and we'll proceed with an alternative method.

## METHOD 2

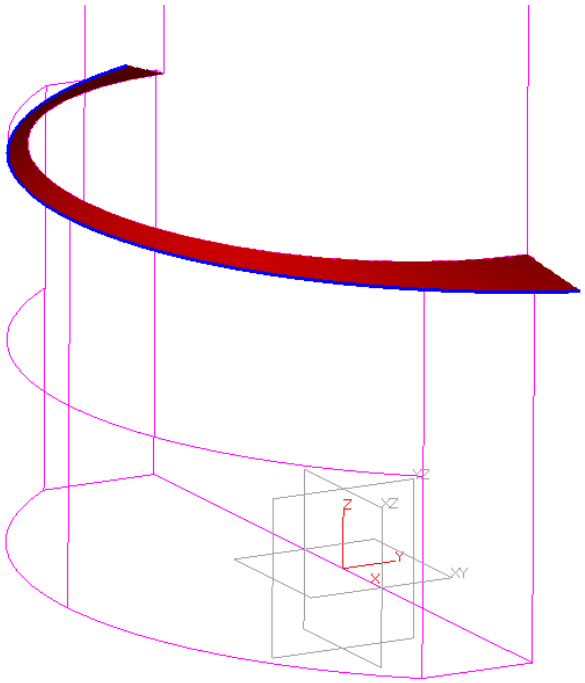
We need to create an angled face with two edges of the face remaining parallel to the XY-Plane.

- Select the "**Trim/Extend curve**" command .
- Pick the front edge shown
- From the options buttons pick "**Arc**" and check "**Extend Both ends**".
- Pick the edge shown, enter a value of **16**mm and the curve will be created as shown below in blue.
- Notice the curve was created automatically, we did not first have to create an edge curve.



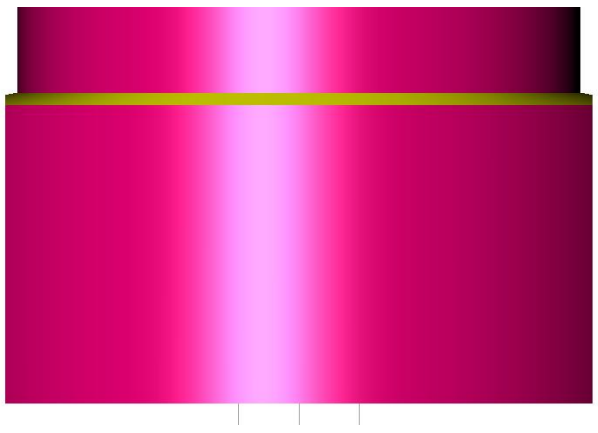
## Create the angled loft face.

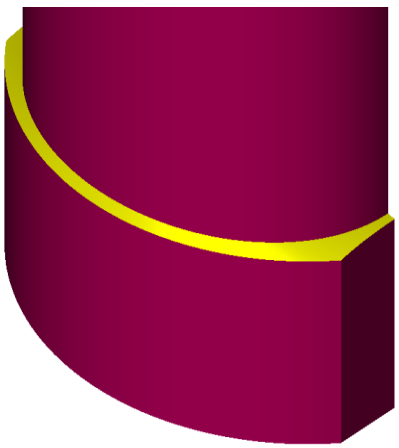
- Use the "**move entities along a direction**" command and move the curve **1.5**mm in a minus **Z** direction.
- Create a loft face between the curve and the inside edge created by the extrude, with continuity = none,  as shown opposite.
- We are going to use this face in a Boolean remove command.
- It is important which side the normal is pointing. In the figure opposite the negative side of the face (shown in red) is pointing up. The positive downward pointing face is the part that will be removed from the solid.
- If necessary use the "Reverse normals" command to reverse the face so that the red side points up.



## Remove the face from the solid and erase the wire-frame curve.

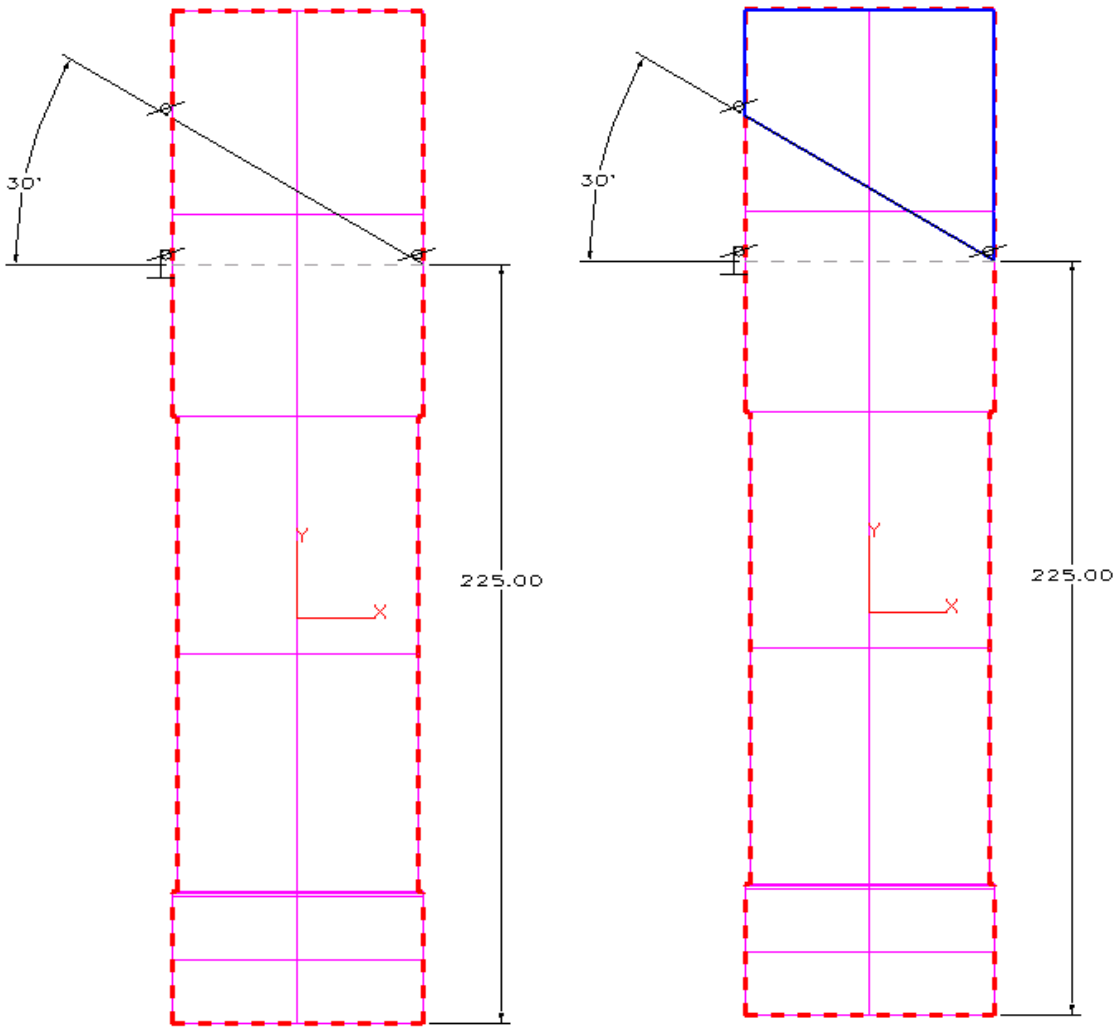
- Select the “Combine shape > remove” command and remove the loft face from the solid as shown below.
- This will keep the two edges parallel to the XY-Plane and keeps the edges a constant 1.5mm apart in the Z direction.





## Create an angled cut at the top (neck) end of the bottle.

- Select the "**Extrude**  **> Remove**  " command.
- Middle click for a new sketch
- Make sure the option "Reference face edges" is checked.
- Pick the back planar face of the bottle as the sketch plane.
- Because we toggled edges on, the edges of the planar back face will be Projected onto the sketch plane.
- Create the angled line to the sizes shown below left, note the line is "on" the reference edges (right mouse, default option).
- Next create a trace profile as shown below in blue.

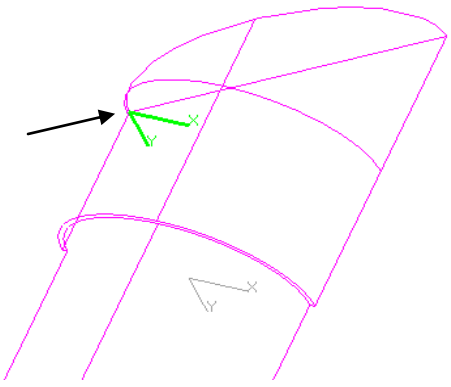


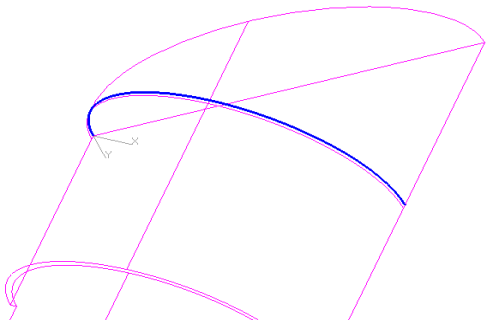
## Create the cut.

- 
- Create the cut as shown.
- Start = 0
- End = -100 (minus).
- **Draft angle = (minus) -1 degree.**

## Creating the revolved and other faces at the neck end of the bottle.

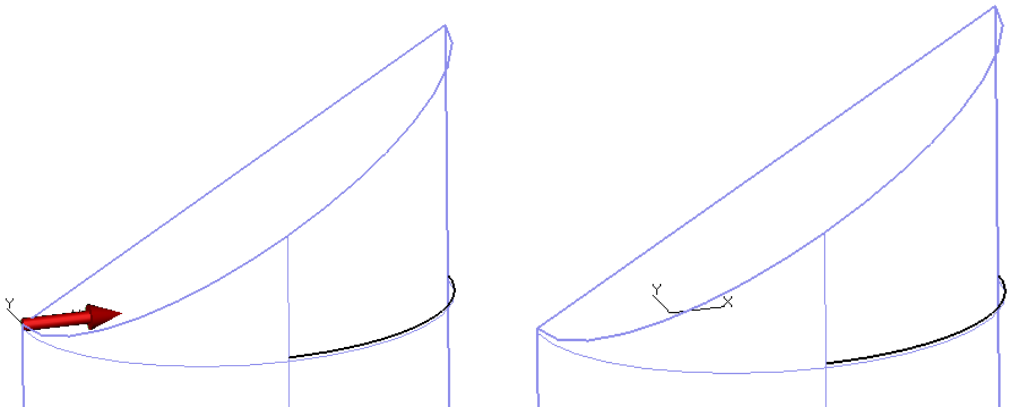
- Create a datum plane parallel to the **XY**-Plane with the origin at the point shown below left.
- Create an intersection curve  between the new plane and the top elliptical face, as shown below right in blue.





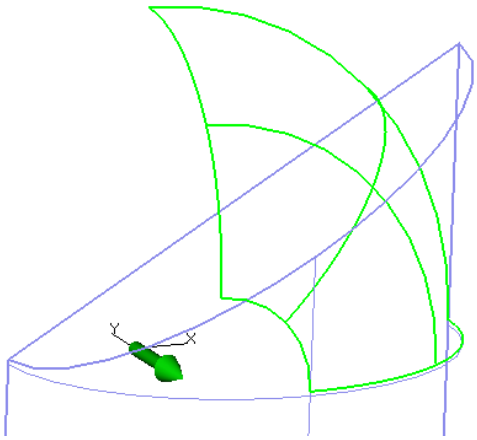
## Trim the curve and move the previous plane.

- From the WireframeTab, select "**Trim to faces**". 
- Trim the new curve with the **YZ**-Plane; side option should be set to "**Keep**".
- We will be using this curve to create a revolved face about the Y-axis of the previously created plane. But first we need to move the plane to a more suitable location.
- Select the **Edit** pull down >Move Entities, use the "Move entities along a direction" Command.
- Pick the plane to move and move this plane 21.6mm along it's own X-axis, as shown below.



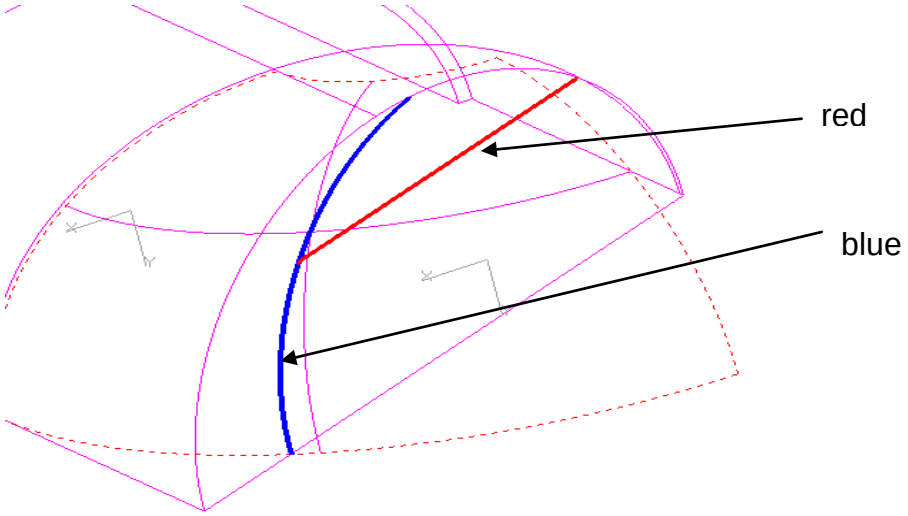
## Create a revolved face

- Pick the "**Revolve**" command.
- Revolve the curve around the minus Y axis of the "blue" plane.
- Why minus and not plus?
- Start = 0.
- End = 90.



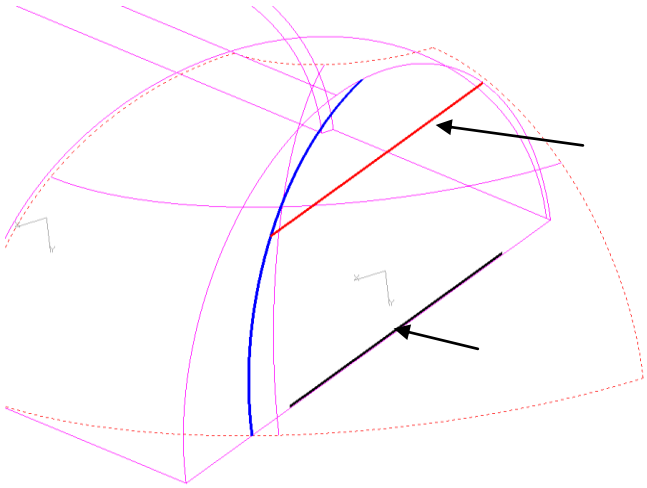
## Create the end shape geometry.

- Create a plane parallel to the XZ-Plane offset 20mm.
- Create an intersect curve between the new plane and the end angled face, as shown in red.
- Create an intersect curve between the revolved face and the end angled face, as shown in blue.
- Select “One touch trim” and trim the red curve to the blue curve as shown below.

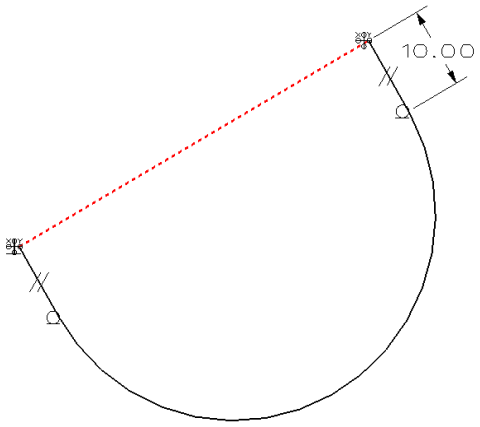



## Continue with the end shape geometry

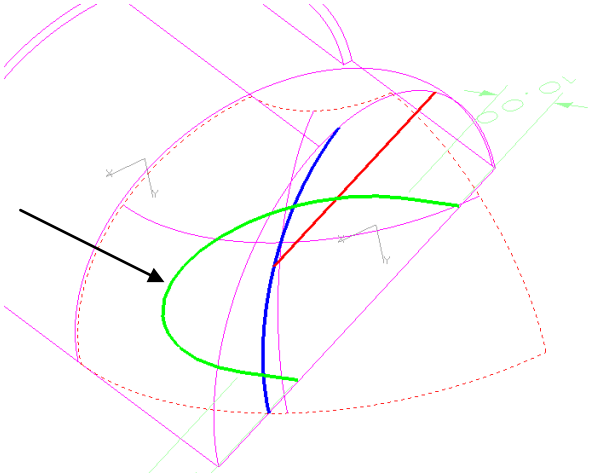
- Create a new sketch on the **XZ-Plane.**
- Create a Projected reference curve.
- Pick the curve shown in red opposite.
- This curve is then Projected onto the sketch plane, as shown opposite in black. Your line will be red dotted if you are using default colors.



- Continue with the sketch and create the sketch entities, as shown opposite.
- Use Show Target for clarity.
- Change the value of the dimensions to 10mm.
- 

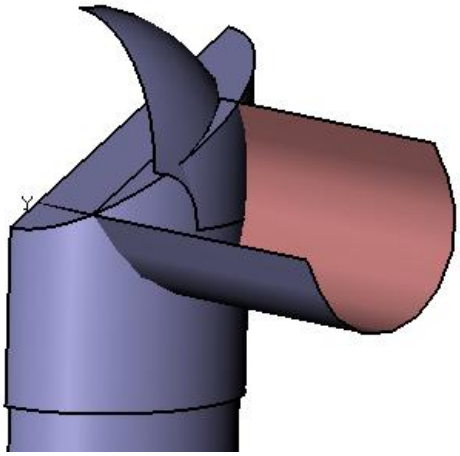



- Next create a concatenated curve from the three sketch entities, as shown opposite in green.

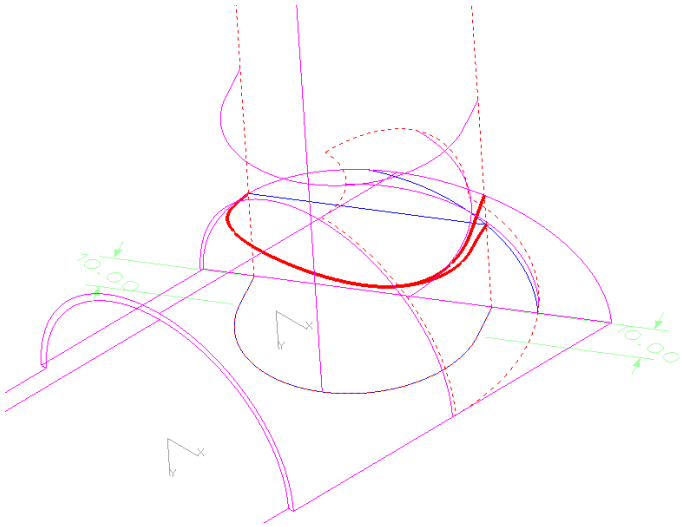


## Continue with the end shape geometry.

- Create an extruded face from the concatenated curve.
- Use the “Alt” key when picking to ensure you pick the curve and not the sketch.
- Extrude the face as shown opposite.
- Start = 0.
- End = -100.

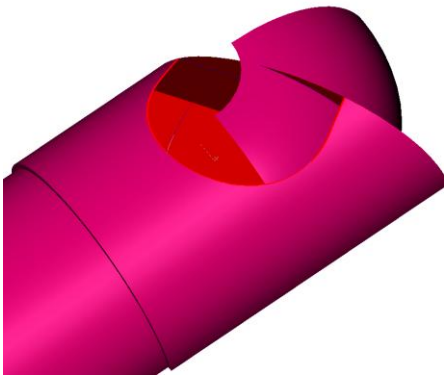


- Create an intersection curve, between both the elliptical face and the revolved face at the “neck” end of the bottle and the previously created extruded face, as shown in red opposite.
- It may be a good idea to change the default line color first.
- Pick the elliptical and the revolved faces as the first shape.
- Pick the extruded face as the second shape.

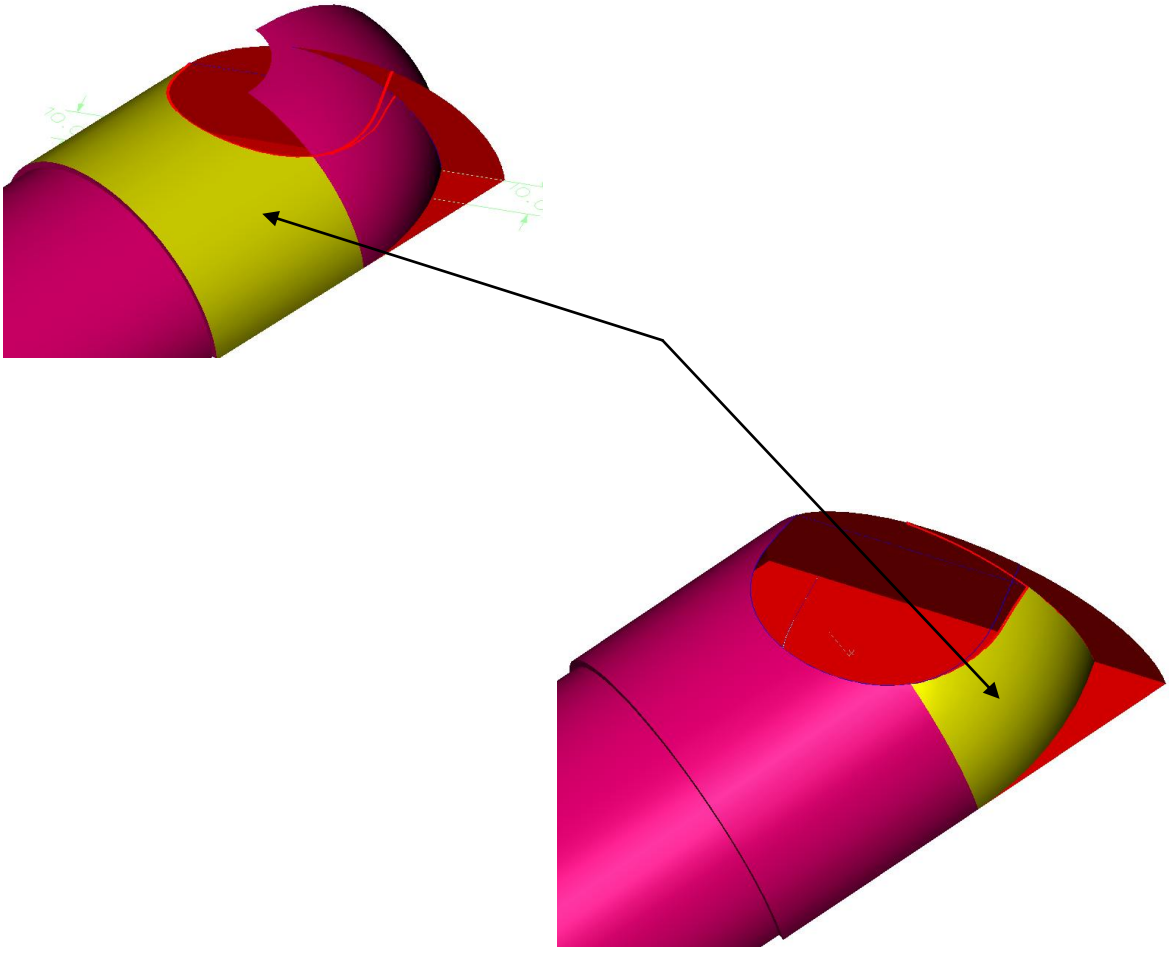


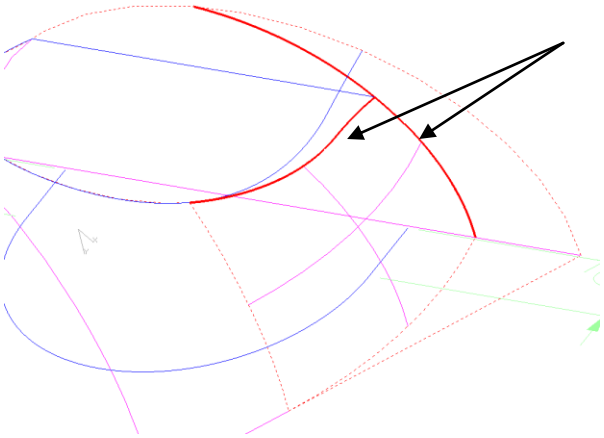
## Trim the end geometry faces.

- Erase the previously created extruded face.
- In the following trimming operations, use the pick filter to select the correct entities (edges or curves).
- Trim the elliptical face to the curve that was created from the intersection of the **elliptical face** and the extruded face with the “**Trim to faces**” , as shown opposite.



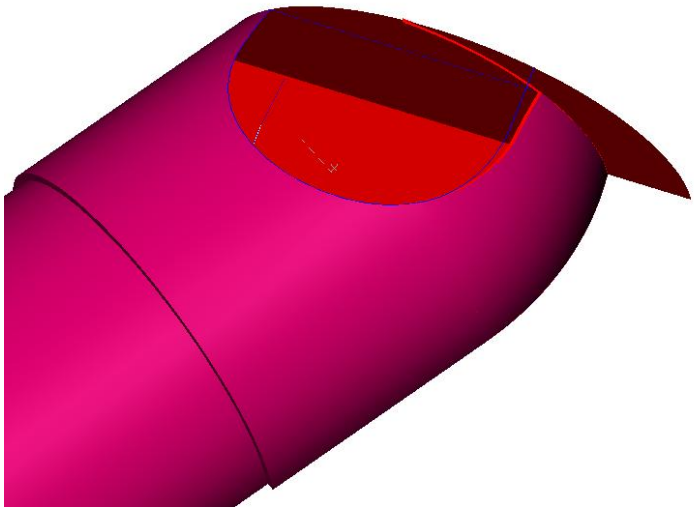
- Next trim the elliptical face once again as before, but this time to the edge of the revolved bottom face. Pick the Extensive trimmed face. Then use the end angled face to cut the revolved face as shown right below figure, in yellow, keeping the lower side.
- Trim the revolved face to the two curves shown red in the lower left diagram.
- The trimmed revolved face is shown in yellow in the figure below.

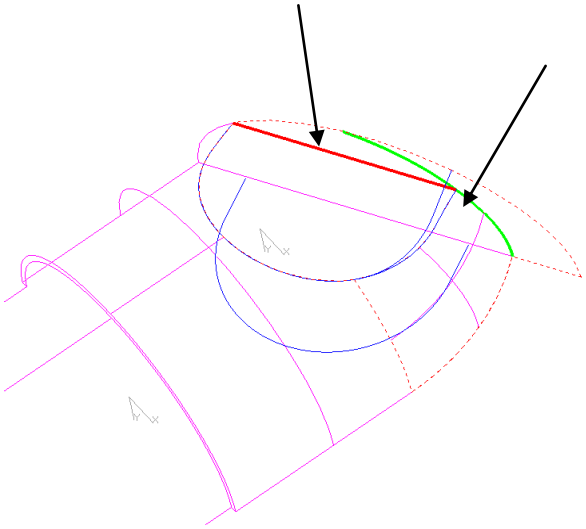


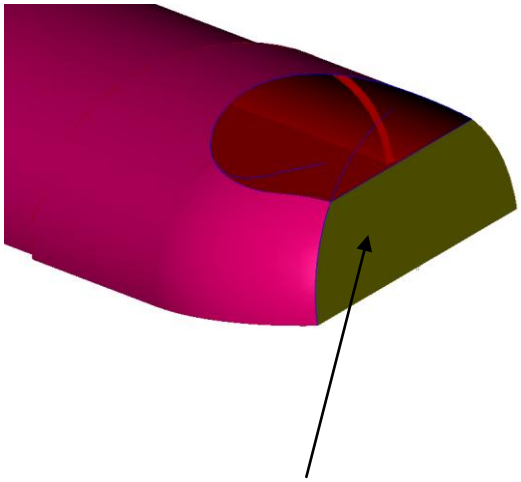


## Trim the end geometry faces.

- Trim the back planar face, to the edge of the revolved face as shown.
- Next trim the end angled face.
- You can trim the angled face “in one hit” if you use the following method.
- Trim the face to the curve shown in red below left and the edge of the revolved face. Do not trim the face to the green curve.
- The trimmed face is shown below in yellow.
- Use “**Sew**” and pick “**OK**” to add the revolved face to the rest of the bottle.

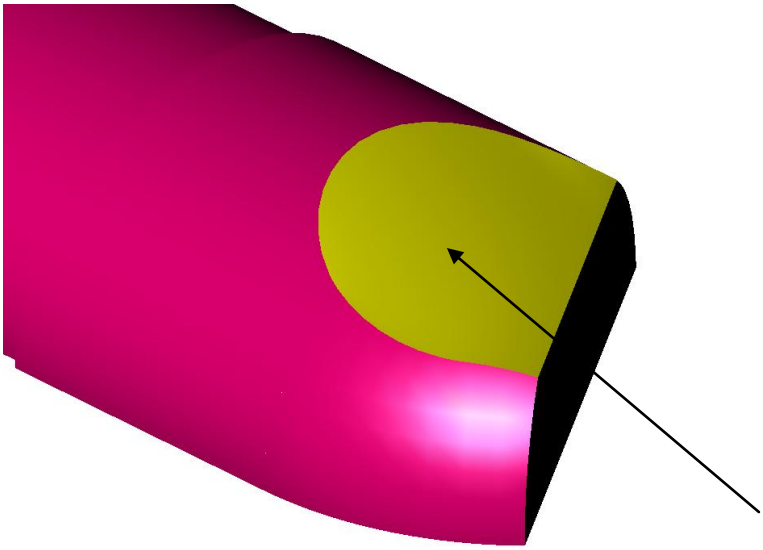






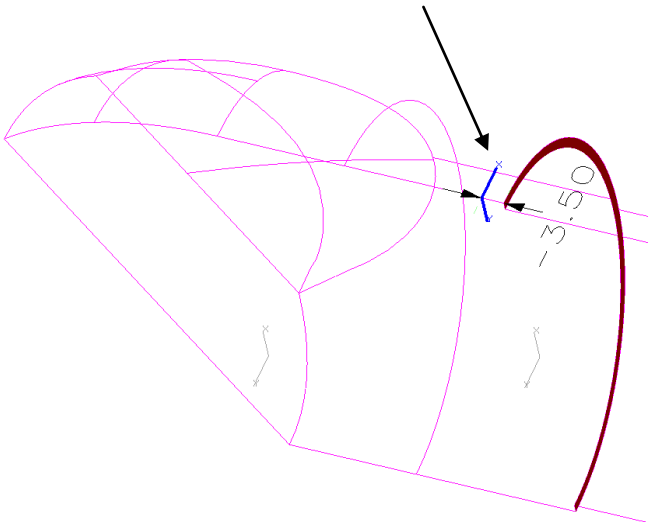
## Create an N-Blend face to close off the bottle shape

- Set the pick filter to “curve”.
- Erase the wire-frame curves.
- Select the “N-Blend” command  and create the N-Blend shown opposite in yellow, as follows.
- Pick the edge of the revolved face.
- Next pick the edge of the angled face.
- Finally pick the edge of the elliptical face.



## Create five grooved slots in the elliptical face.

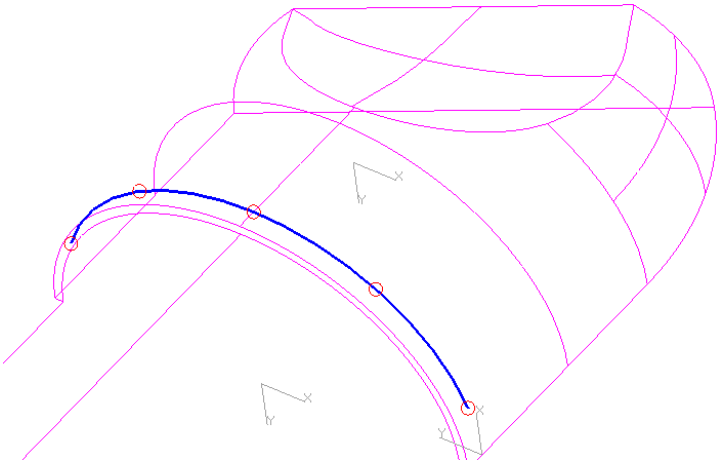
- Create a datum plane offset from a plane.
- Pick the face shown shaded opposite.
- Enter a value of (minus)  –3.5mm, as shown opposite in blue.




## Create an intersection curve and trim both ends, then create five points

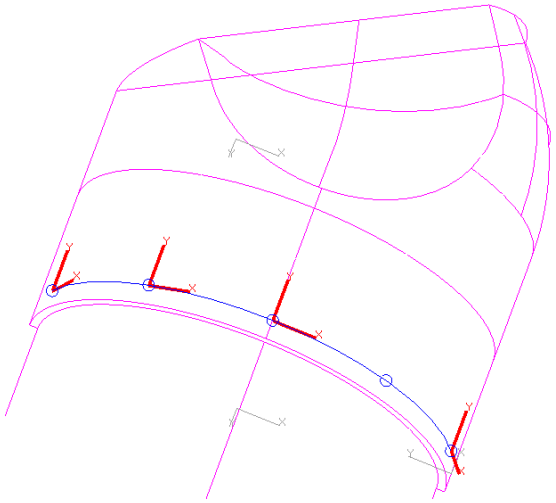
Create an intersection curve between the previously created plane and the elliptical bottle face as shown.

- Select the “trim/extend curve” command.
- Check the “Extend both ends” option.
- Pick the curve and trim both ends –10mm.
- Create “N” equally spaced points on curve”.
- Select the **Wireframe Tab** and pick the **Points on Curve** command.
- Pick the curve and enter 5, to create the five points shown opposite.



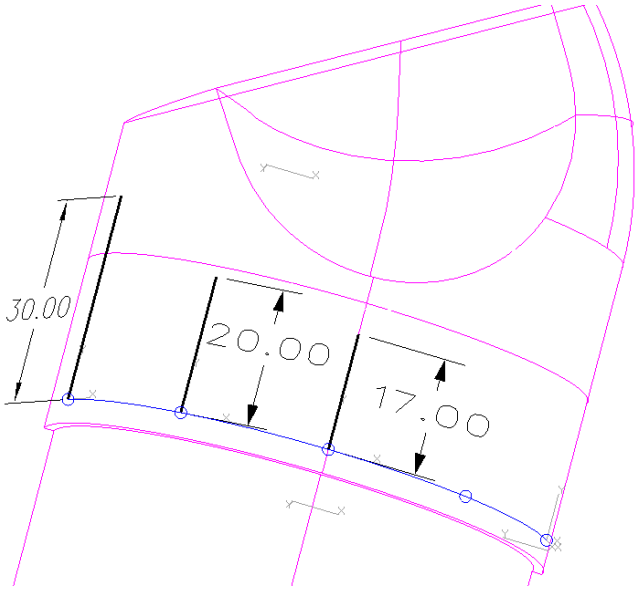
## Create four planes tangent to the elliptical face and through the points.

- Right click “Insert Datum”.
- Set the pick filter to **face**, and pick the upper elliptical face
- Pick the “Origin” option
- Set the entity filter to “3D point”, and pick the points shown.
- The Offset is 0.
- Note pick four points and not five, as two of the grooves are identical and therefore the second groove can be copied.



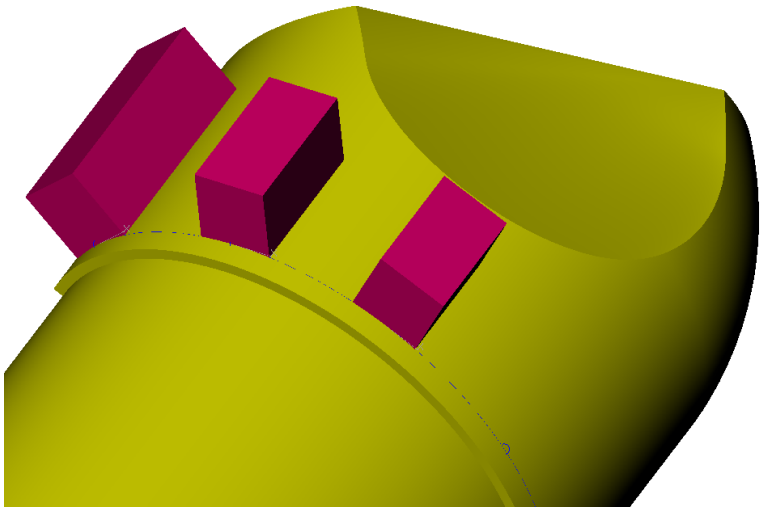
## Create three sketches on three of the tangent planes.

- Create a new sketch on the left-hand tangent plane.
- Create a vertical line.
- Add the dimension.
- Modify the value to 30mm.
- Copy the sketch onto the clipboard.
- 
- Create two more sketches on the planes shown opposite.
- In each sketch paste from the clipboard, then change the value accordingly as shown.



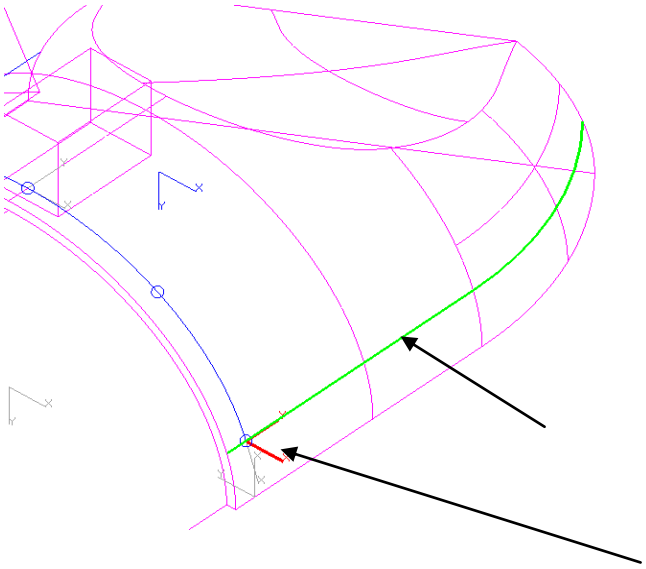

## Create three extruded solids.

- Create an extruded solid.
- Select the command.
- Pick the 30mm long sketch.
- Start = -2mm.
- End = 10mm.
- From the “Offset” options, select “Thicken Same”.
- Enter 5mm for the offset value.
- Repeat for the other two sketches as shown opposite.



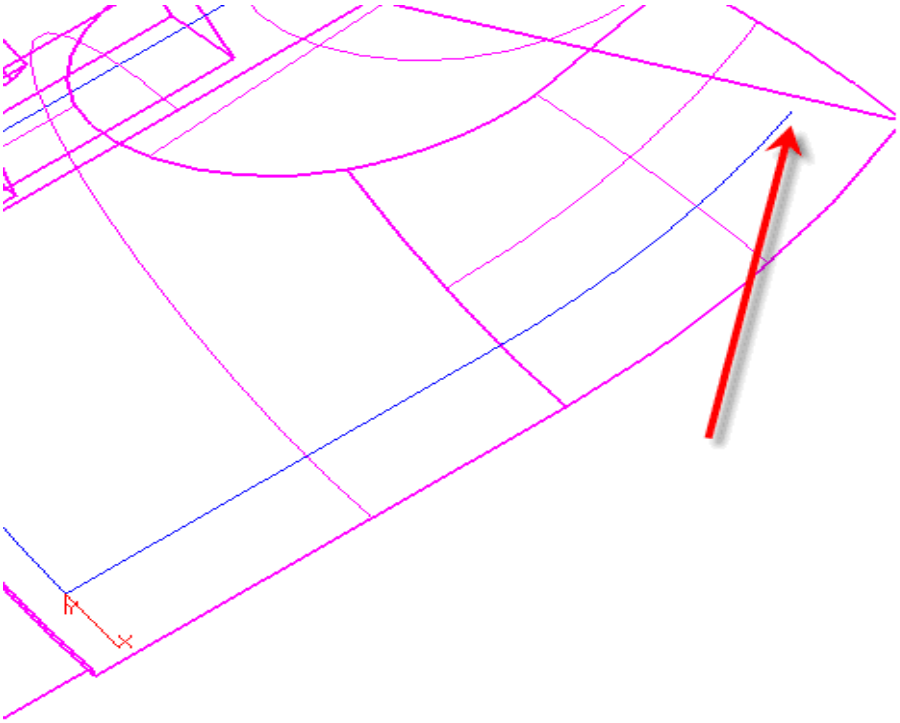
## Create the geometry required for the curved (swept) groove

- Create a “Datum Plane”.
- Pick the XZ-Plane
- Set the Origin at the right-hand point as shown opposite in red (set the pick filter to point).
- Create an intersection curve between the new plane and both the elliptical face and the revolved face, as shown in green.
- The plane intersects the elliptical face twice. We do not require the left curve; we will erase it later.
- Erase this datum



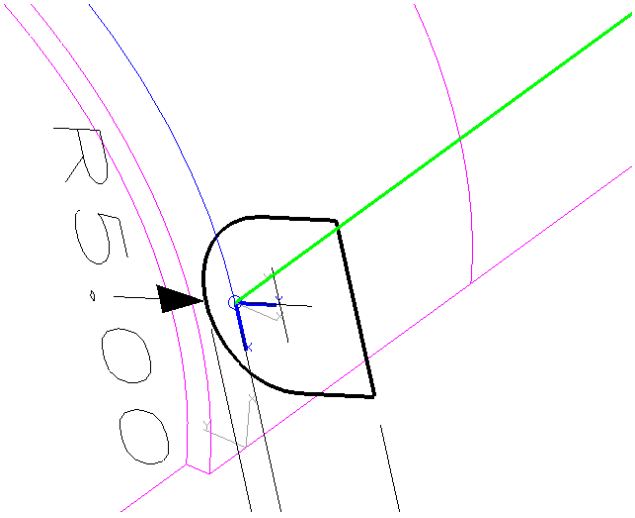
## Concatenate the right-hand curves and trim to the correct length

- Concatenate the two right-curves.
- Trim one end of the curve as follows.
- Select the “Trim/Extend curves” command.
- Uncheck the “Extend both end” option
- Pick the end of the concatenated curve opposite to the “red plane” end.
- Enter –5mm for the trim value.
- Next trim the end nearest to the “red plane” using the one touch trim command, to trim the small part of the curve as shown opposite in green.



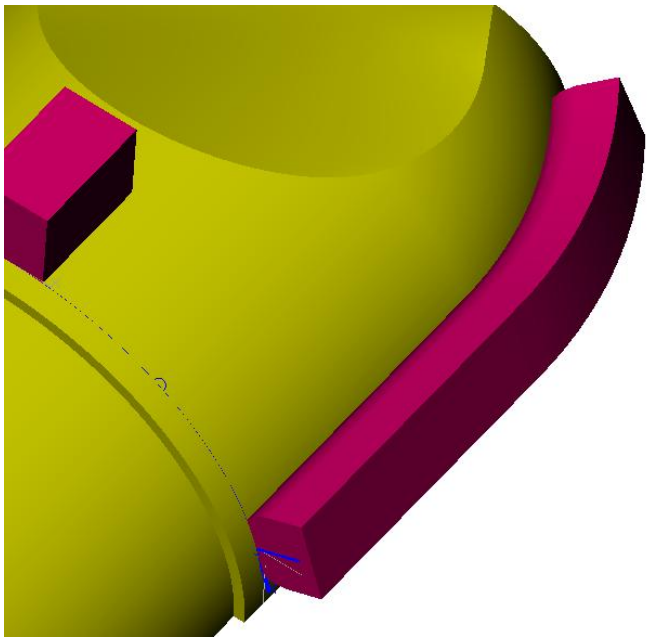
## Insert a sketch copy to be used in the sweep command

- Next we will insert a sketch copy that will be used to create a swept solid.
- First we need to move the right-hand “Tangent face” plane 90 degrees about its X-axis, as follows.
- Select the **move entities along a direction** command.
- Pick the “Tangent face” plane shown opposite.
- Move this plane 90 degrees about its own X-axis, as shown.
- Insert a sketch copy onto the moved plane. The file name = “Bleach_Sketches”.
- The global sketch is called “Bleach_Groove”.



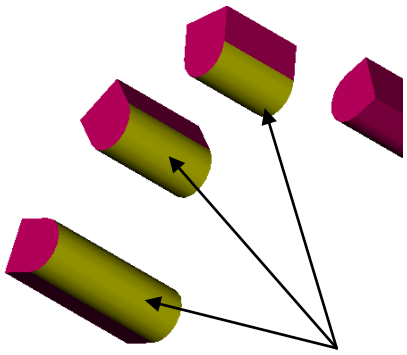
## Create a swept solid

- Create the swept solid shown opposite.
- Next erase the wireframe entities and the 5 points, set the pick filter to curve and then to points.

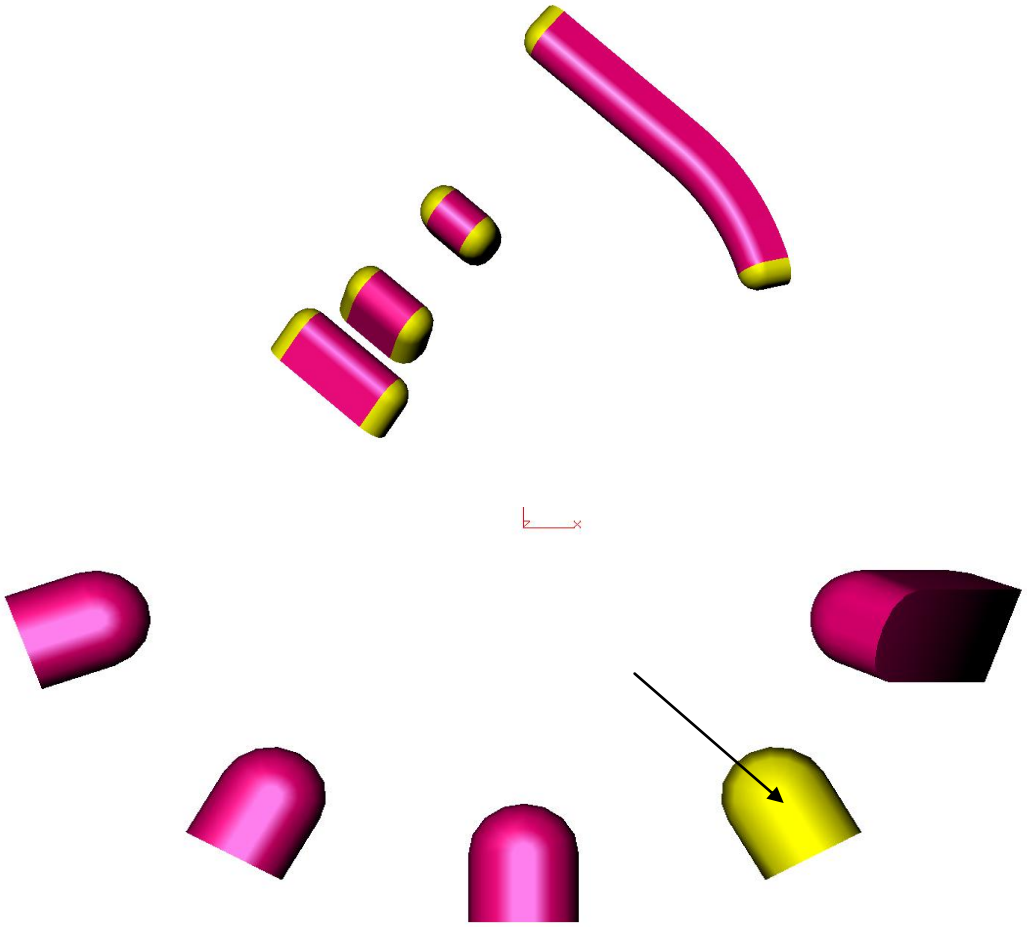


## Fillet the edges of the “groove” solids

- Fillet the six edges 5mm as shown in yellow on the three solids opposite.



- Fillet the 30 edges 4mm as shown in yellow on the four solids opposite.
- Finally from the copy entities suite of commands, mirror the “missing” fifth grooved solid about the YZ-Plane as shown below in yellow (ensure the pick filter is set to shape.

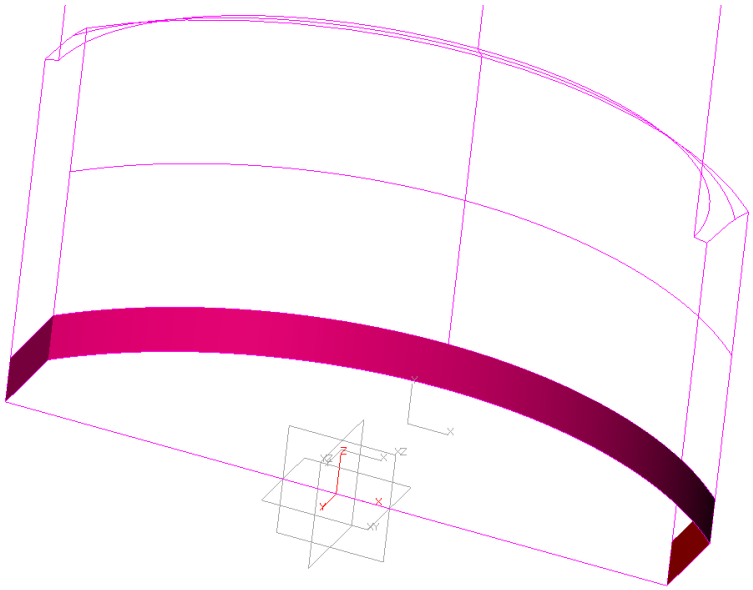


## Use the “groove” solids to create five grooves in the bottle

- Subtract (remove) the 5 solids from the bottle use combine > remove command  .

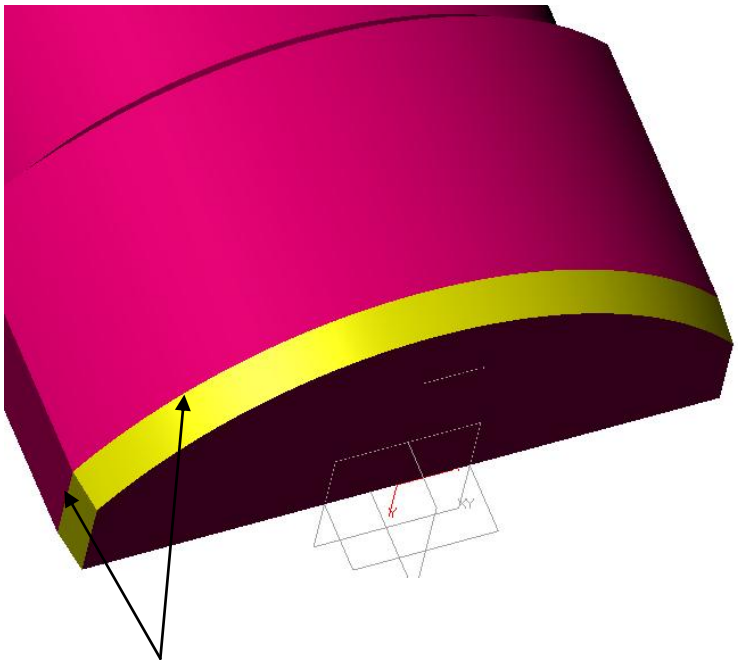
## Create draft angle “Chamfers” at the base end of the bottle

- Create a plane offset from a plane.
- Pick the XY-Plane.
- Offset this plane 5mm towards the neck end of the bottle.
- Use this plane to split the two side faces and the elliptical face, as shown opposite. Use the “Split with Faces” command and not the divide command, as we want the faces to remain attached to the rest of the bottle shape.

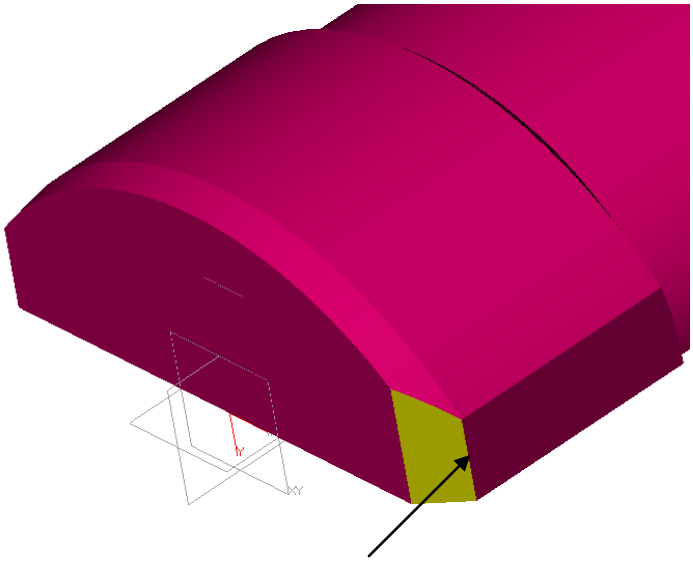


## Create 10 and 50 degree draft faces

- Using the draft command.
- Apply a draft angle of 10 degrees to the two faces (pick filter set to edge), shown opposite in yellow.
- Set “Face S” to “Bottom”
- Set “Intersections” to “Do not remove”
- The “ref plane” is the same plane that was used to “split” the faces or you can pick XY, they both indicate the same direction of pull.

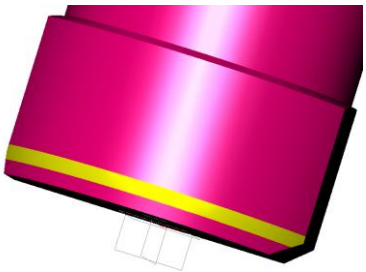


- Now, using the same technique, apply a 50 degree draft to the face shown opposite in yellow.

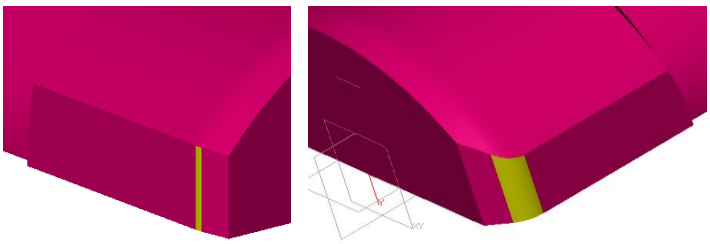


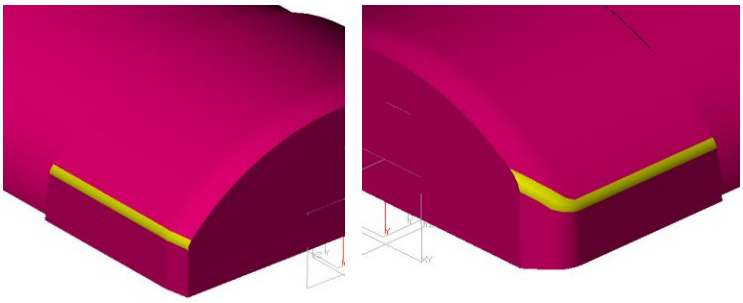
## Create various fillets as shown

- Create a 20mm fillet on the front edge as shown opposite in yellow.

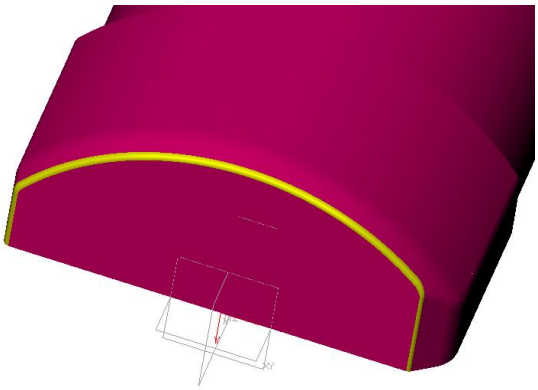


- Create 6mm fillets on the other two edges that we just used for the 10 and 50 draft angles.
- Create the two sets of 3mm fillets on the edges shown opposite in yellow.
- Use chain select to pick the edges on both sides.



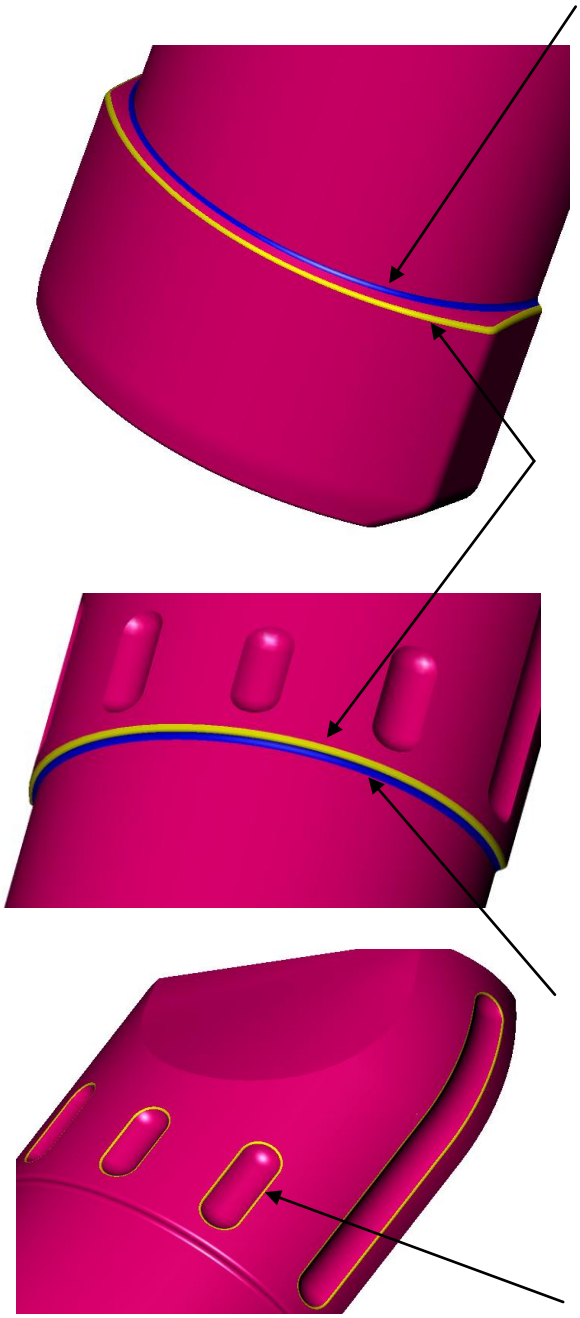


- Using chain select, create 1mm fillets along the bottom edges of the bottle as shown opposite in yellow.



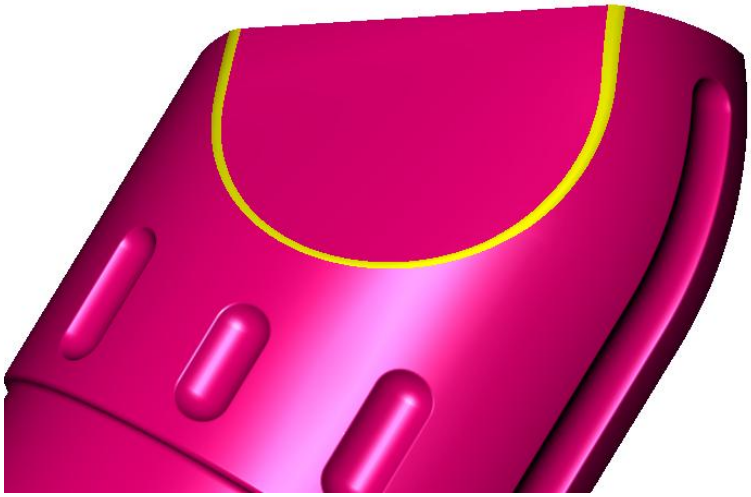
## Create more fillets as shown

- Chain select the edges to produce the 1mm fillet shown opposite in yellow.
- Next Fillet the edge to produce the 1mm fillet shown opposite in blue.
- Notice how the blue fillet runs into the yellow fillet.
- Create a 1mm fillet on the edge shown opposite in yellow.
- Create a 1mm fillet on the edge shown opposite in blue.
- Create 0.5mm fillets on the five slots of the bottle as shown opposite in yellow.



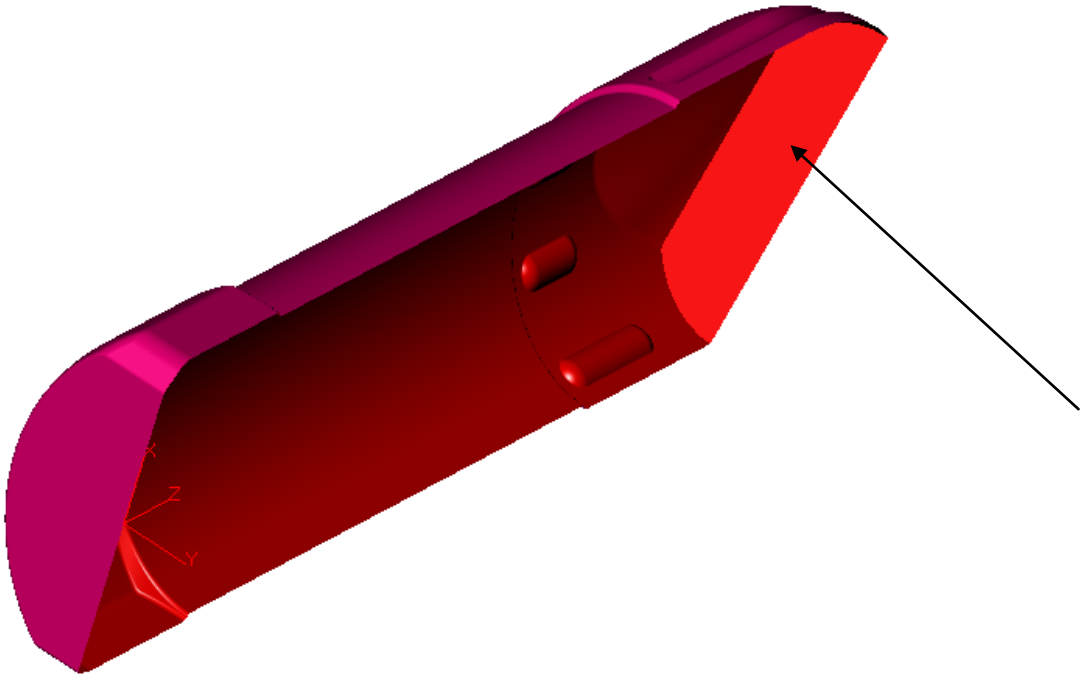
## Create more fillets as shown

- Chain select the edges and create a 3mm fillet as shown opposite in yellow.



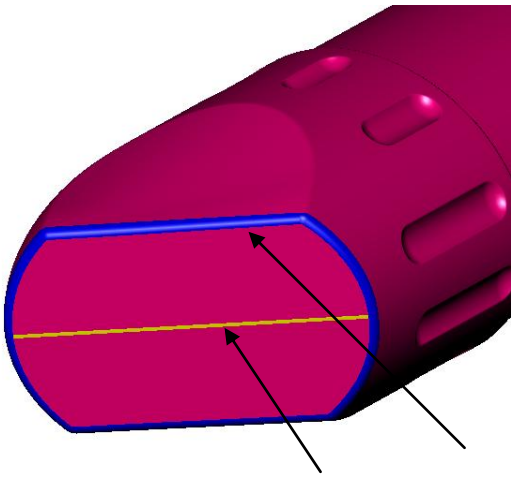
## Create the second half of the Bottle

- Erase the back planar face as shown below.



## Create the second half (continued)

- Select the "Mirror" command.
- Set the entity filter to "shape" and mirror the existing shape about the XZ-Plane.
- We now have two shapes. Use the "Sew" command, to join the two shapes together.
- Fillet the edge shown opposite in yellow 20mm.
- Chain select the outer edges shown in blue 2mm.



 Save the current file .

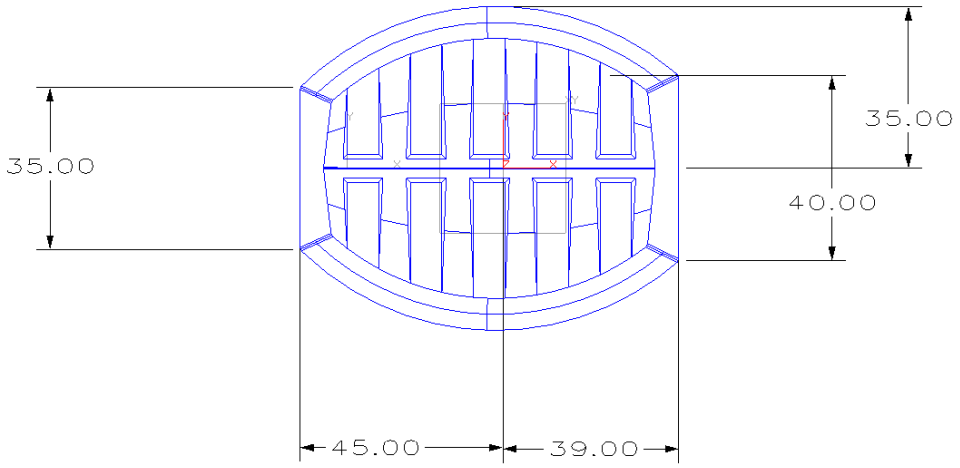
The base cut out that will be applied to the bleach bottle could also be used on different bottle sizes. Let's copy an existing "cut out" part and modify it to suit our requirements.

Open the file "Five slot base.Z3".

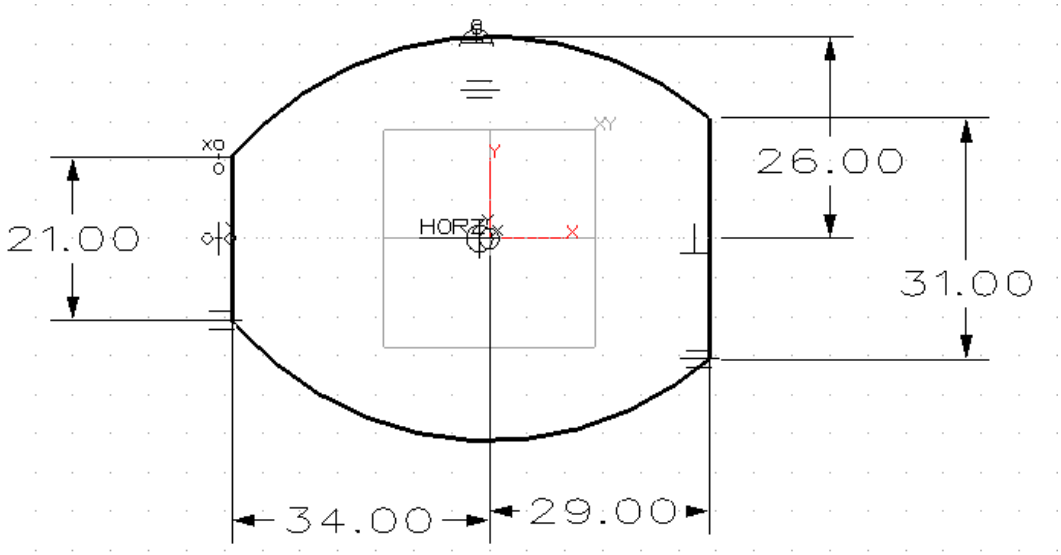
- Click on part name "master" from the Object list.
- Right-Click and select the "Copy" command . 
- You can place the copied part in any file using the browse option.
- For our example we will place our copy in the "Bleach" file.
- Re-open the file named "Bleach.Z3 ".
- Exit part to the object level.
- Right-Click on part named "01_Bleach_Bottle" and select the "Paste" command  .
- Right-Click on the new "master" object.
- Select the "Rename" command  .
- Name the new part "bleach_cutout" as shown.
- Click OK

## Edit the size of the new part

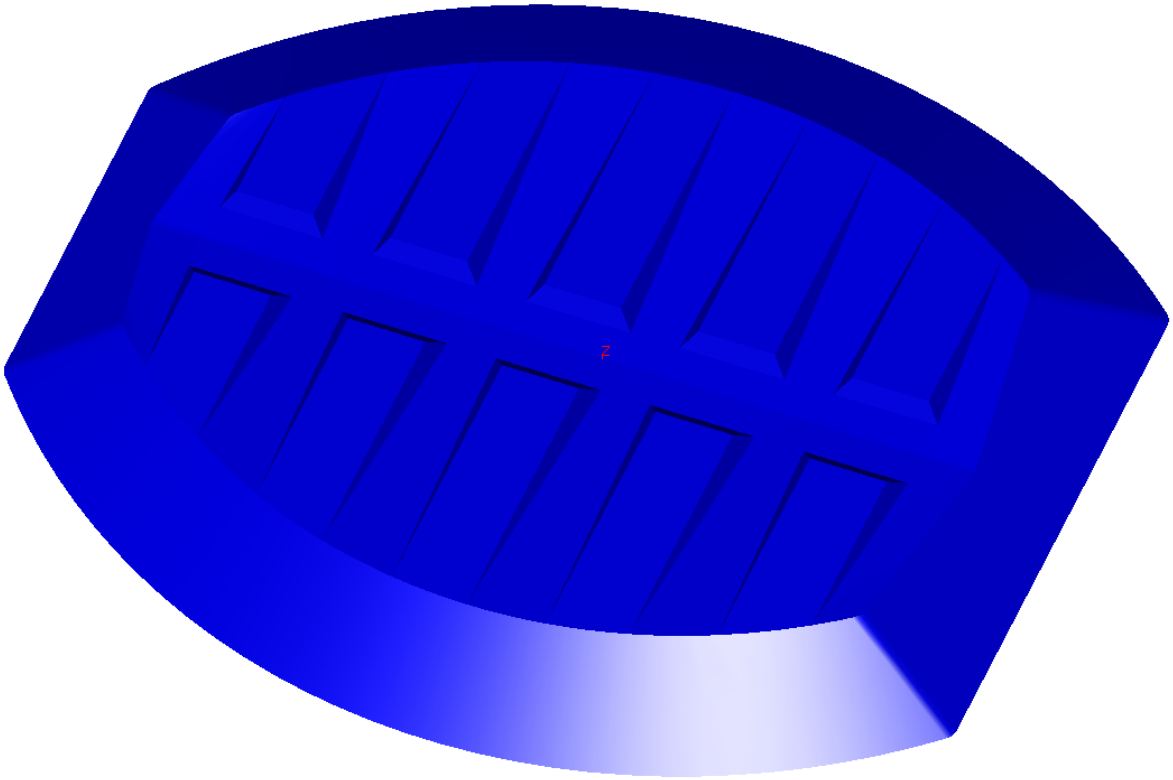
- Edit the part “bleach_cutout”.
- Open the dimensions showed in the sketch 1 followed below.
- Starting from right to left; change the dimension values as follows.
- 35mm to 26mm.
- 40mm to 31mm.
- 39mm to 29mm.
- 45mm to 34mm.
- 35mm to 21mm.




Turn on dimensions as shown below.



**Regen the part as shown below**

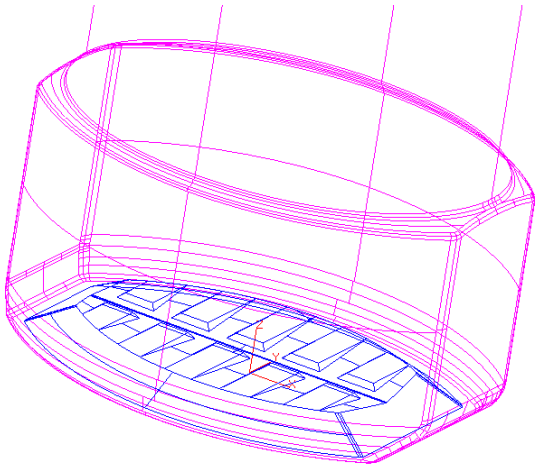


Save the part to disk.

**Create the cut out on the base of the bottle.**

**Load the "01_Bleach_Bottle" part.**

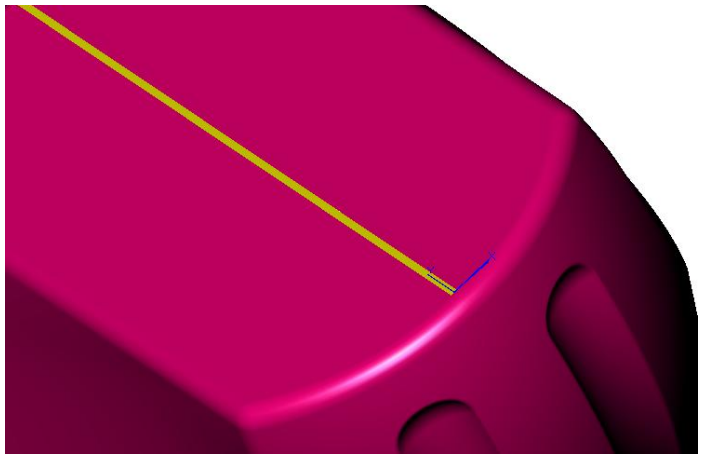
- Place the mouse pointer in the graphics area away from any geometry, press the right mouse button and select "Insert Component".
- Check the option to "Anchor" the component.
- Pick the part "bleach_cutout".
- Locate the part onto the zero point of the XY-Plane by accepting the defaults.
- As shown below.

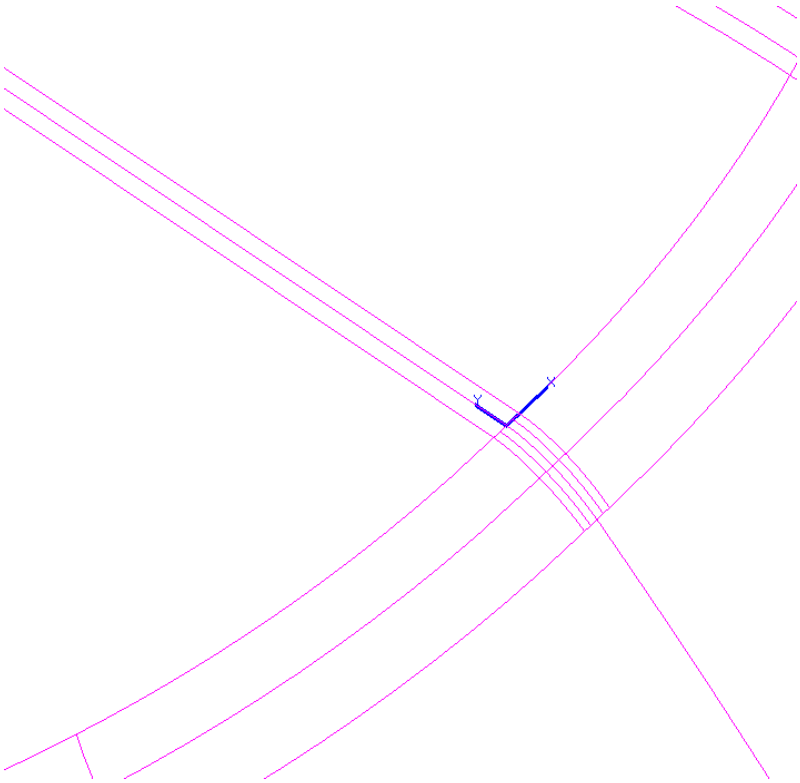


- The “bleach_cutout” part now resides in our bottle as a component.
- We need to convert this component into a cut in our bottle.
- Select the “Component Merge” command  with the “Remove” option. 
- Pick the “bleach_cutout” component.
- Accept the defaults for the rest of the command.
- As shown opposite.

## Attach the neck component to the bottle

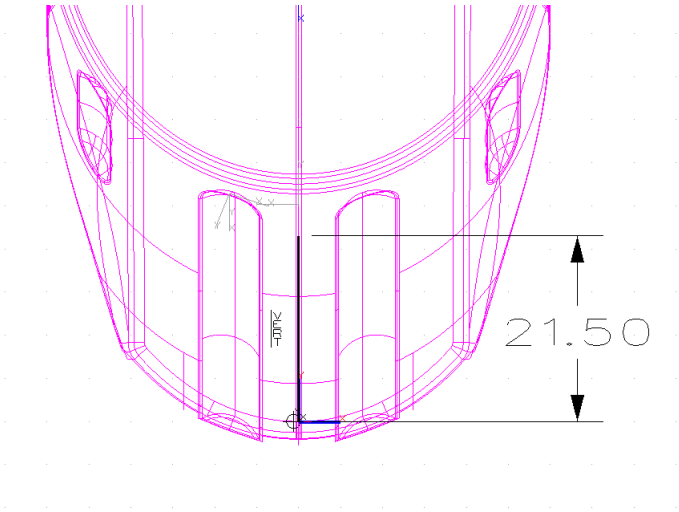
- Create a plane tangent to a face as shown.
- The tangent face is the fillet face at the neck end of the bottle as shown opposite in yellow.
- The origin of the plane is end edge point of one of the fillets shown below in wire-frame.
- The orientation is not important.  See the next step.





## Attaching the neck, continued

- Create a new sketch on the previously created plane.
- Draw a vertical line from zero and dimension it as shown opposite.
- We will use the end point of this line to locate the neck component.
- 




- Insert a component (right mouse button).
- Use the browse option to locate and select the object file “Bleach cap.Z3”.
- Pick the part “Laser neck”.
- Place the component out in space.
- Right Click and select the Align command, make the component bottom coincide with the datum inserted in the last step. At the same time, make them downward shift 1mm.
- Right Click and select the Align command again. Make the origin of the component coincide with the end point of the sketch created above
- Use the “Component Merge” with the “Add” option to attach the neck to the bottle.
- Create a 1mm fillet between the bottle and the neck as shown opposite in yellow.

Save the file “Bleach.Z3”