# *ZW3D*
# *Advanced Bottle Closure Training Guide*

# ZW3D™ V2014 Advanced Bottle Closure Training Guide

# *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 Closures 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:**

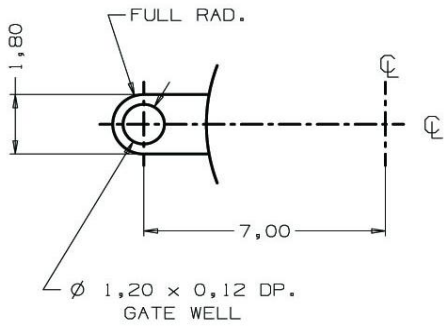
1. For best results, view and print the PDF version of the documentation using the Adobe Acrobat Reader.
2. Since the **Advanced Bottle Closures Training Guide** is only a portion of the complete documentation, it is highly recommended that you utilize the **HELP Manual**. You will find additional topics, 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.

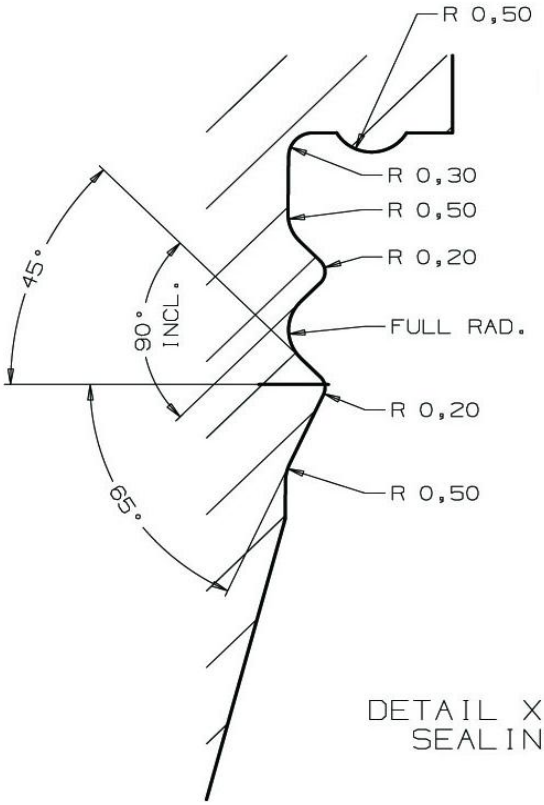
Thanks for being our customer!
Team ZW3D

# *Table of Contents*

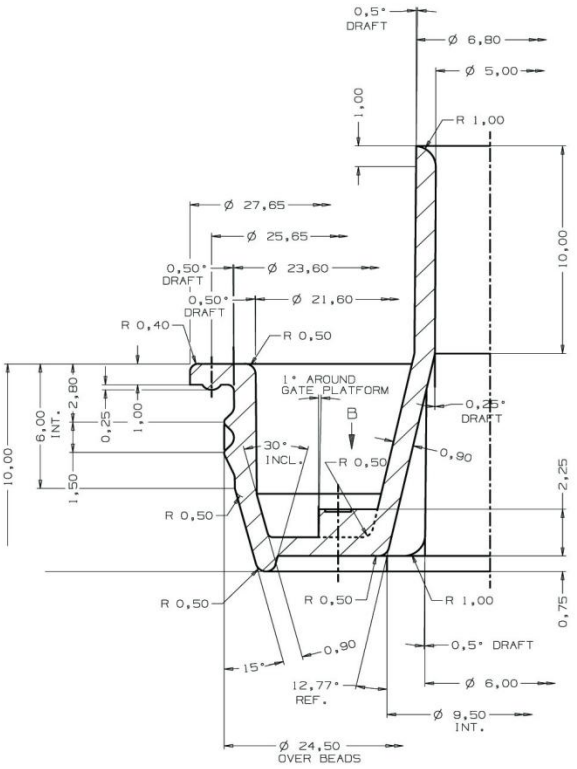
## Example one, The Plug



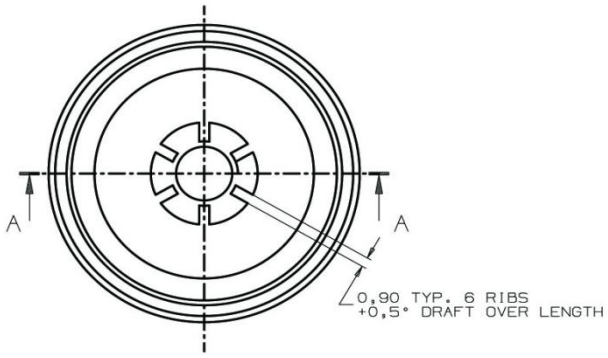

SCRAP VIEW ON ARROW B
(SCALE 10:1)

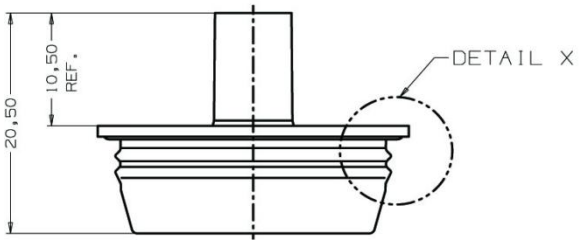



DETAIL X VIEW OF
SEALING BEADS




HALF SECTION AA



## Load the file “Bleach_cap.Z3”.

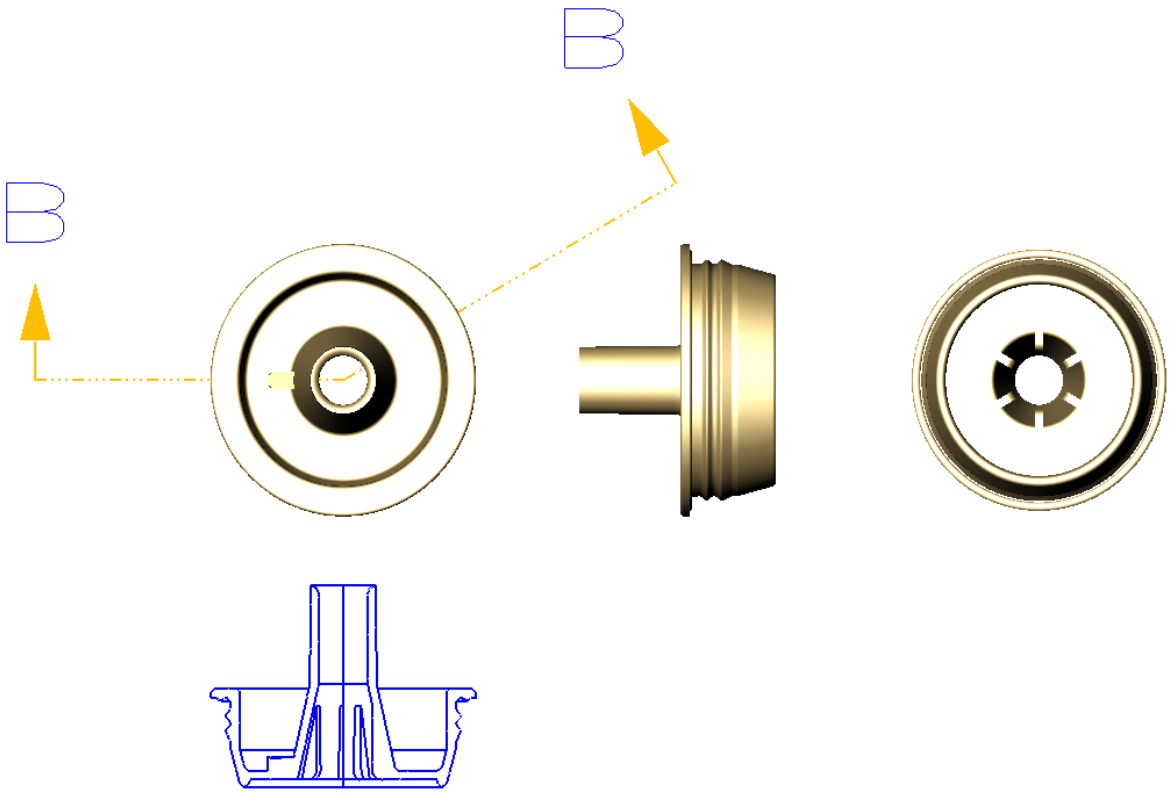
This file contains finished examples that you may refer to if you wish.

Create a new part called **“Plug”** using the **mm** template.

Edit the Dimension Attributes and select two places angular precision.

With the attributes set at the part level it will be in effect for all sketches we make in the Plug.

The laser plug is basically a revolved shape. If you wish you could create a single sketch that would describe most of the shape of the plug. (See the “Half Section AA” on the previous page.) However, it is sometimes easier to split the design into several simpler sketches. There is no correct way, use the method that best suits you. For our example we will use multiple sketches to create the part.

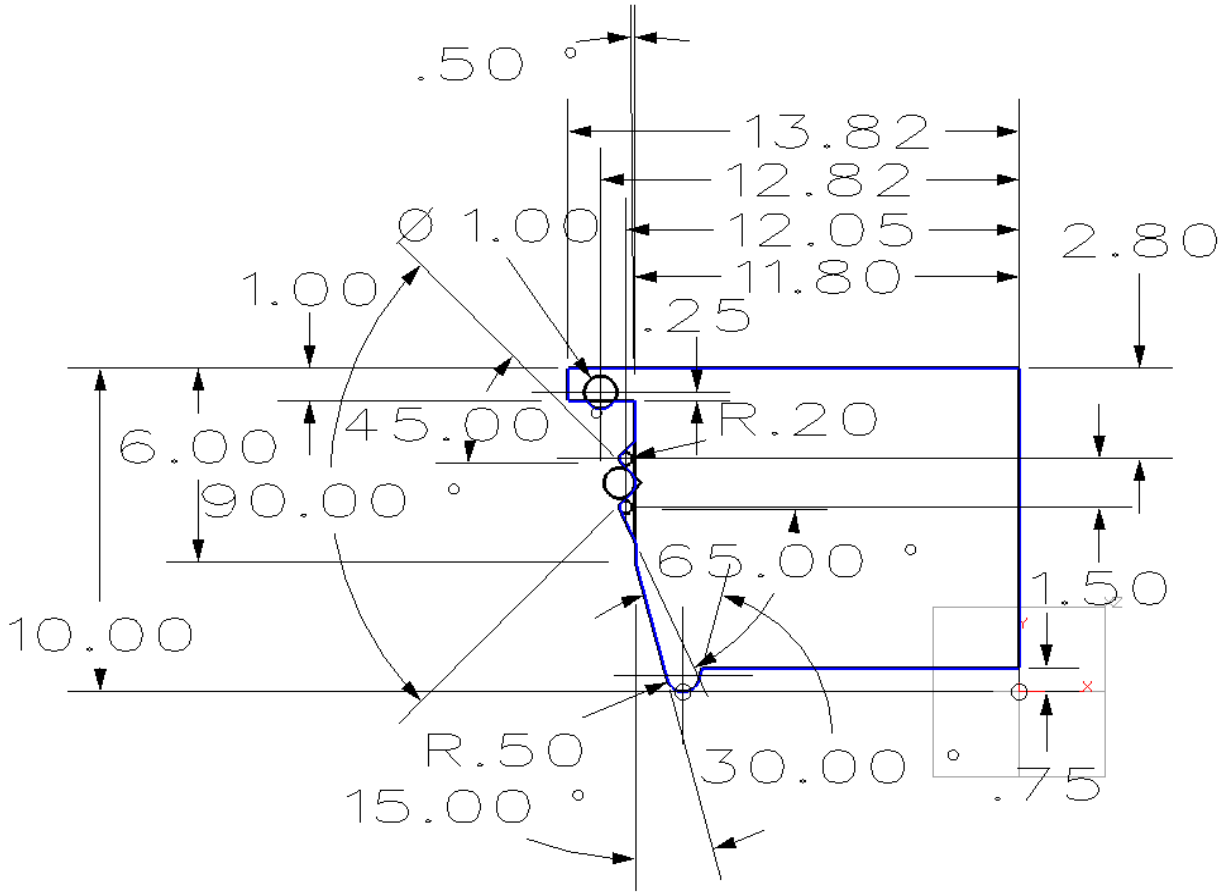


Let's start with a sketch that defines the basic shape (including the sealing beads) that will fit into the bottle neck.

Pick the **"Revolve"** command, and create a new sketch on the "**XZ**-Plane**"**.

- Once in the sketcher, select **Preferences** from the **EDIT** pulldown menu.
- Set the grid spacing to **1** mm.

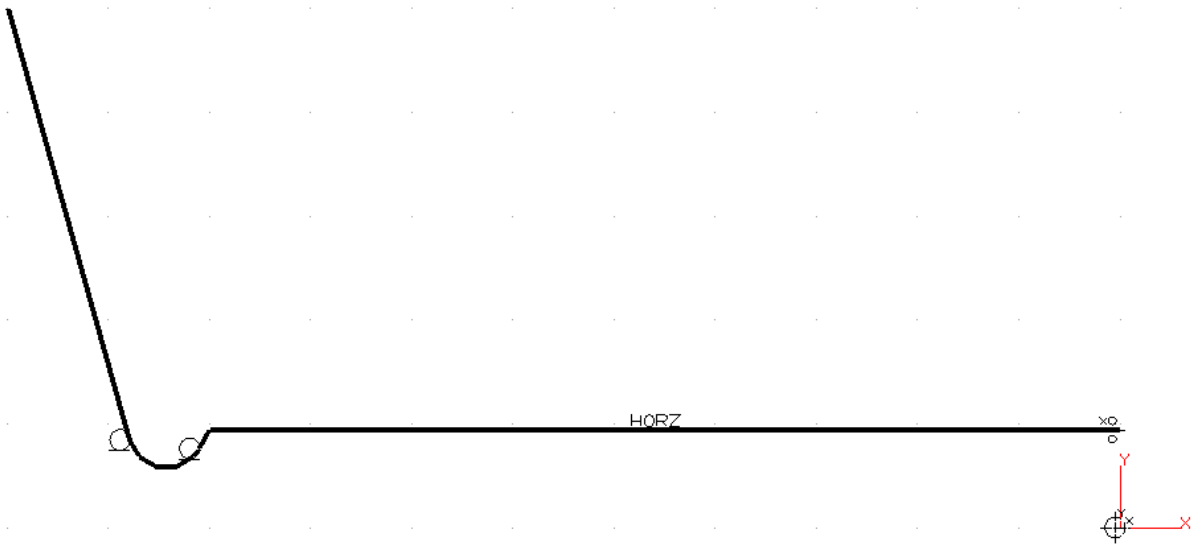
The completed fully constrained sketch with its "trace profile" is shown below, with the constraint symbols switched off for clarity. If you wish, you can try to create this sketch yourself. Remember also that in ZW3D the user is not forced to create fully constrained sketches.




If you prefer, or are have difficulty creating the sketch, go through the steps shown on the next pages.

Step 1.

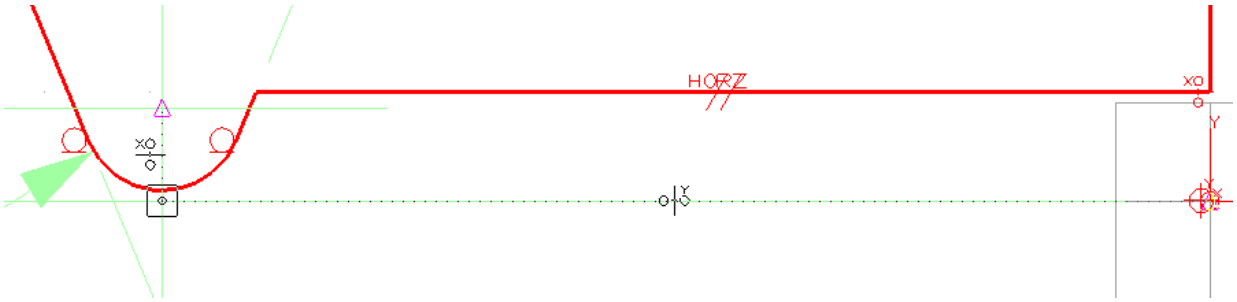
Zoom in closely (the part is really small). Make the number in the lower left corner of the screen, approximately 16.



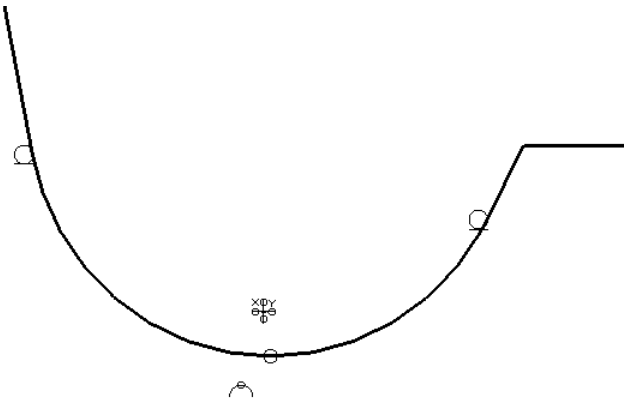

- Create all the above geometry using the "**Draw**" command.
- Start by creating the horizontal line shown.
- The start point is aligned in X with the zero point (notice the constraint).
- The Y value of the line is 1mm (snap to grid point).
- Use the read out option (**Ctrl+k**) and place the end point of the line at approximately minus 9mm in X.
- Create a short angled line down as shown.
- Toggle to tangent (re-pick the end point) and create the arc (make sure the arc is slightly above the zero point). You may want to temporarily turn off the smart pick so you don't pick up any unwanted constraints. Use Right mouse button options.
- Create the other angled line as shown.
- Continue with the sketch as show on the next page.

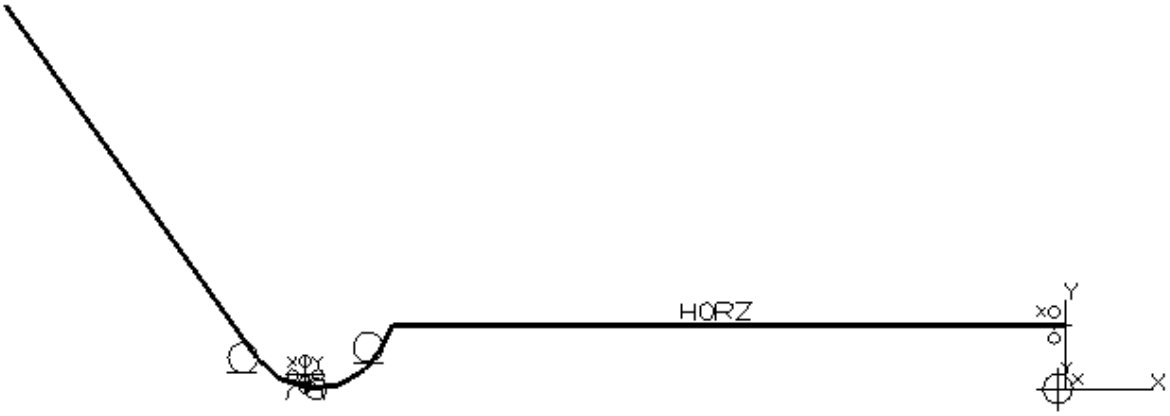
Next we need to align the bottom of the arc shown on the previous figure, with the point at zero (0,0).

To accomplish this first create a new point, which is aligned in Y with the point at zero (0,0) and aligned in X with the center of the arc as shown in the figure below.



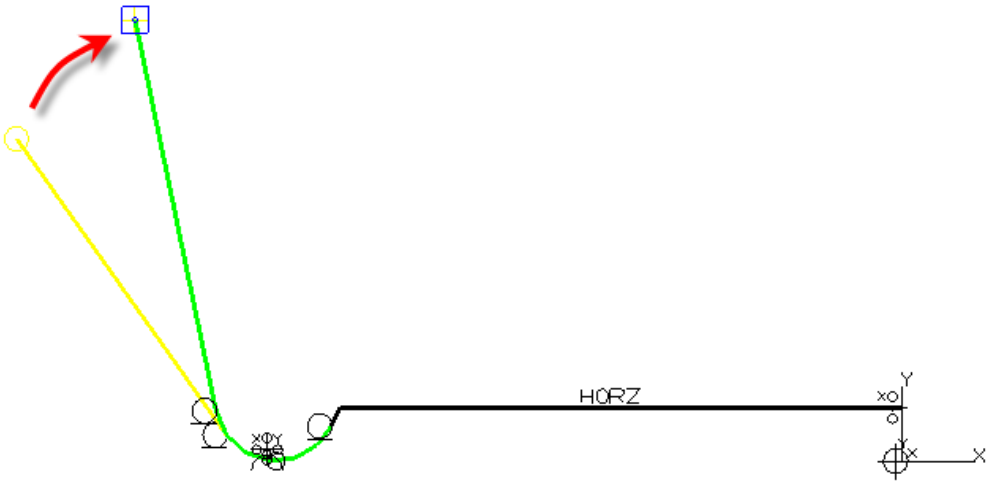

- Now we need to create a **point to curve constraint** to force the arc profile to pass through the newly created point.
- Select the "**point to curve constraint**" as shown on the icon above.
- Pick the arc.
- Pick the point and complete the command.
- The point and the constraints are show below.
- The constraints have been moved for clarity.
- Do not check the **Delay Solve** option



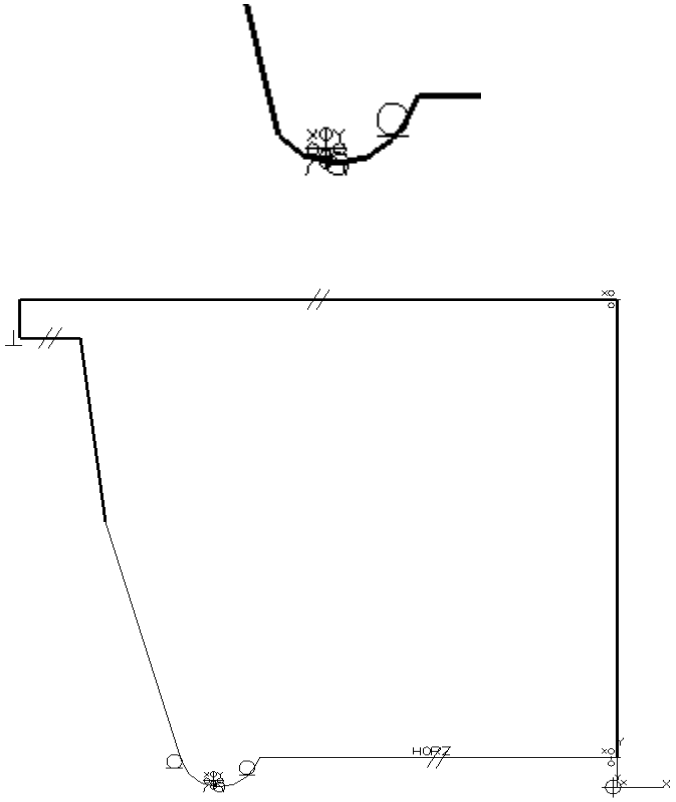



If your model looks like the above image, it may help to adjust the angle of the line to get it closer to what it should look like before continuing.

Use the **Drag** command found on the **Sketch Tab.** 

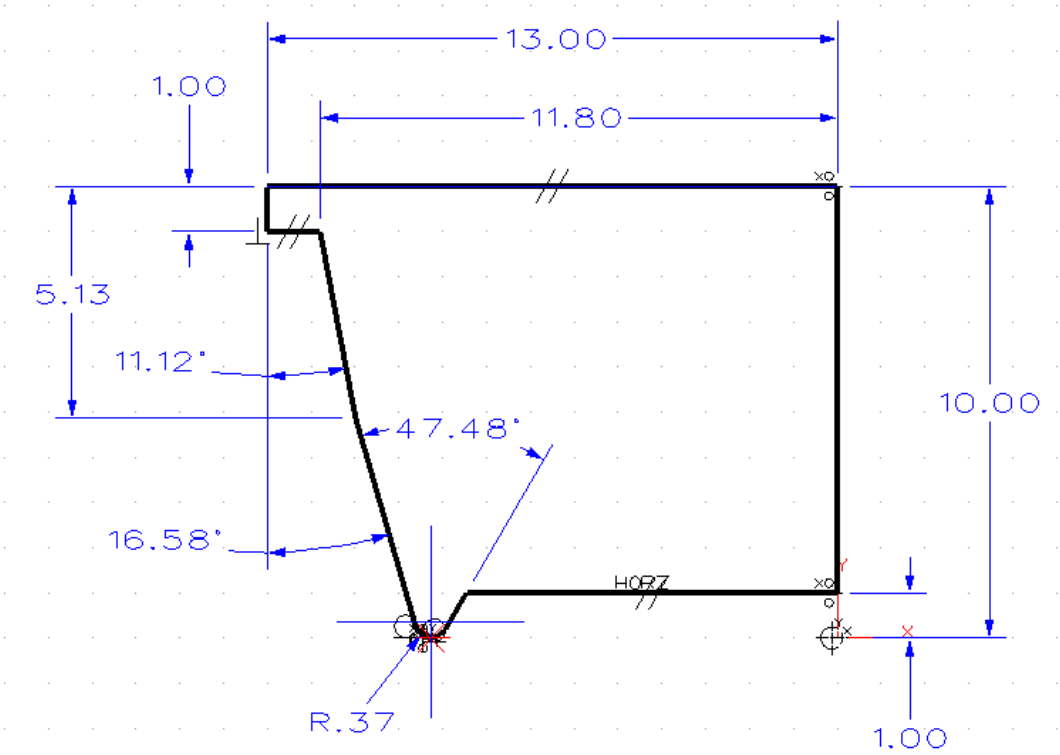



Click on the critical end point of your line and **Drag** to the new location.
The tangency with the arc is retained unlike the **Stretch** command that would violate the tangent constraint.

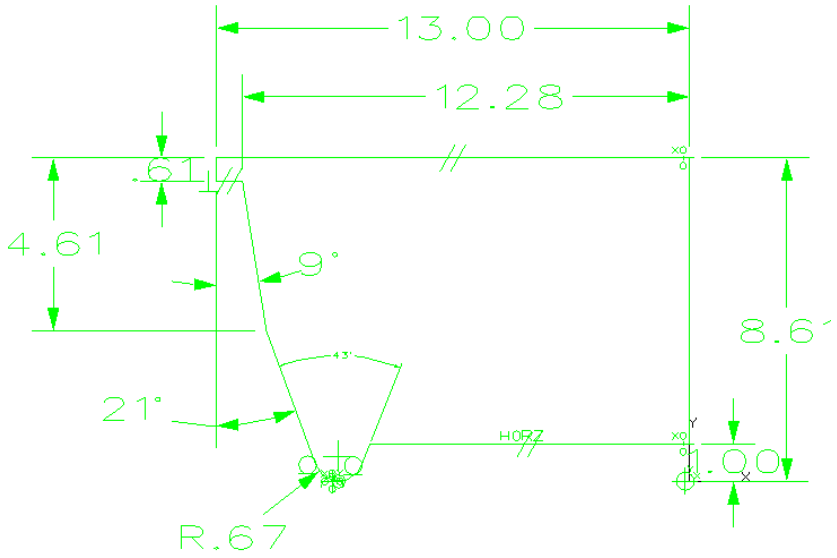


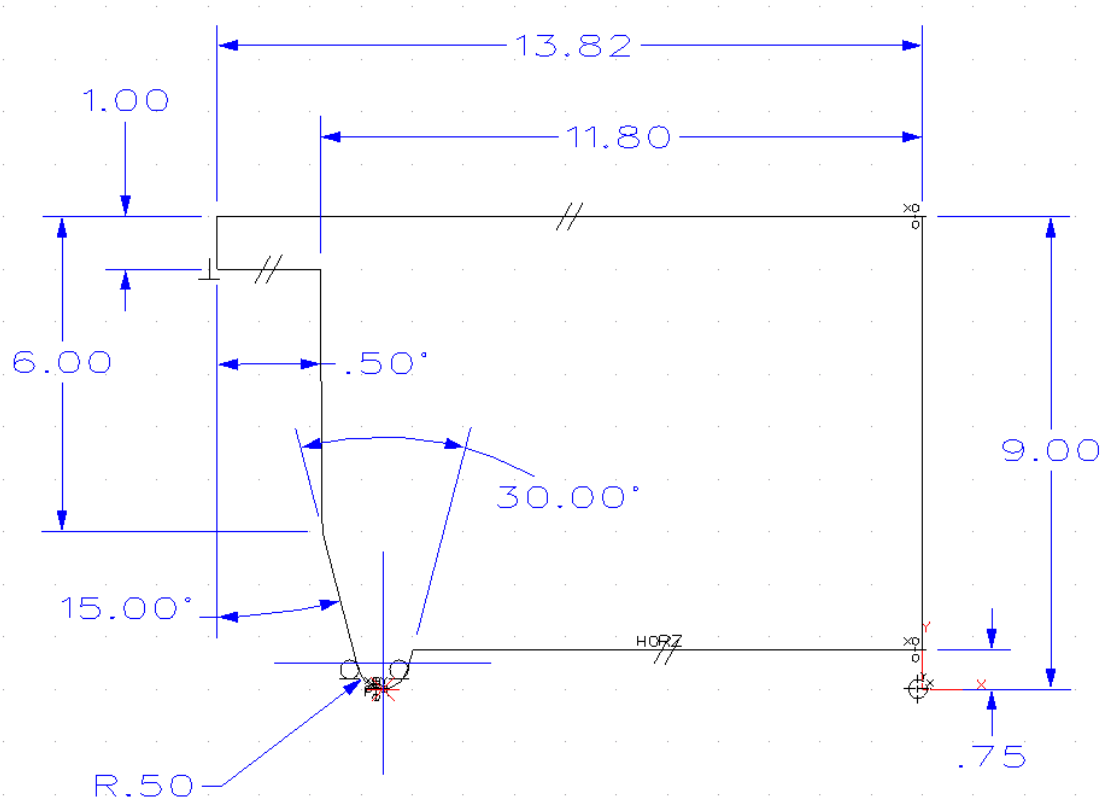

- On the Configuration dialog box continue with the smart draw command and create the five remaining lines shown above in bold.
- Notice the constraints.
- Next apply dimensions to the sketch as shown on the next page.

But first, let's check a setting in the configuration.
On the 2D tab make sure the **Auto prompt dimension value** is unchecked. This is a very valuable tool but when the sketch shape is not constrained or drawn to scale you will get unwanted results.



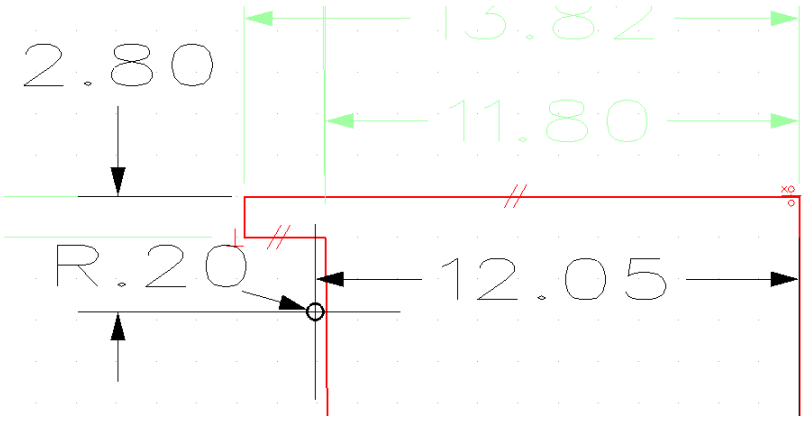

- The dimension values on your sketch will be different, but referring to the above sketch values.
- Add the 0.37mm radius Dimension.
- Add the 11.8mm and 13mm horizontal dimensions.
- Add the 5.13mm 10mm 1mm and 1mm vertical dimensions.
- Add the 11.12deg. 16.58deg. and 47.48deg. angular dimensions.
- All ten of the dimensions shown above were created using the smart dimension tool.
- The sketch is now fully constrained.
- Press the Inquire Status button
- Next edit all 10 of the sketch dimensions to the values shown in lower figure below. Use the “Delay Solve” option until the very last change.
- **DO NOT** exit the sketcher, we still have more geometry to create.


13.00
12.28
.61
4.61
9°
8.6
21°
HORZ
1.00
R.67

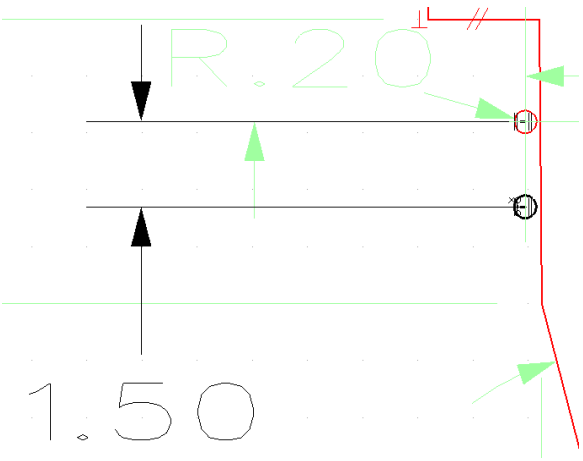

13.82
1.00
11.80
6.00
.50°
9.00
30.00°
15.00°
HORZ
R.50
.75

We now need to create the geometry for the sealing beads.

- Start by creating the circle shown below being careful not to apply any constraints when positioning the circle.
- Add the dimensions shown below in black.
- Edit the values to those shown.
- Note the 12.05mm dimension is derived as follows, **24.5/2-.2**

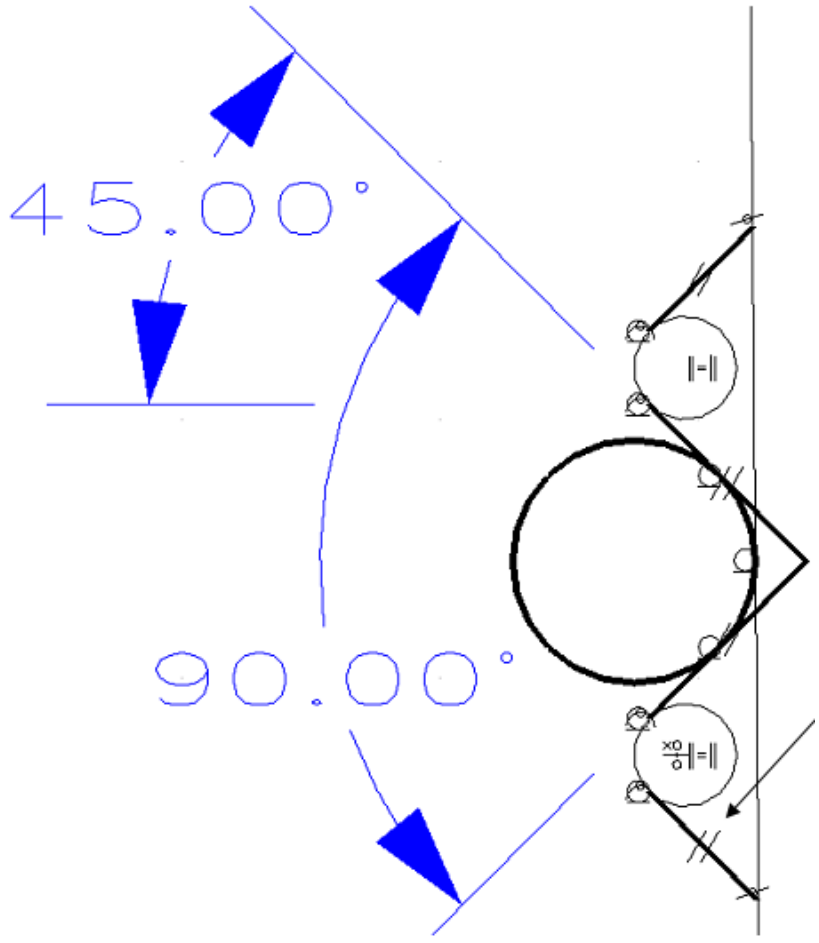



- Create a second circle below the first.
- Make sure you align in X the center of the new circle with the center of the previous circle. ‖=‖
- Add the “Constrain distance to be equal” constraint to fix the radius of the second circle to be the same as the first.
- Add and modify the value of the 1.5mm vertical dimension shown opposite in black.



Create the geometry shown in **bold.**

- Start by creating the four lines making sure they are tangent (right mouse button) to the small circles and also constrained where necessary to the line. Also note that the lines are parallel to each other (if necessary add the parallel constraint).
- Next create the 3pt circle; make sure the circle is tangent to all three lines shown.
- The next step is to add and where necessary edit the values of the angular dimensions shown in the figure below.



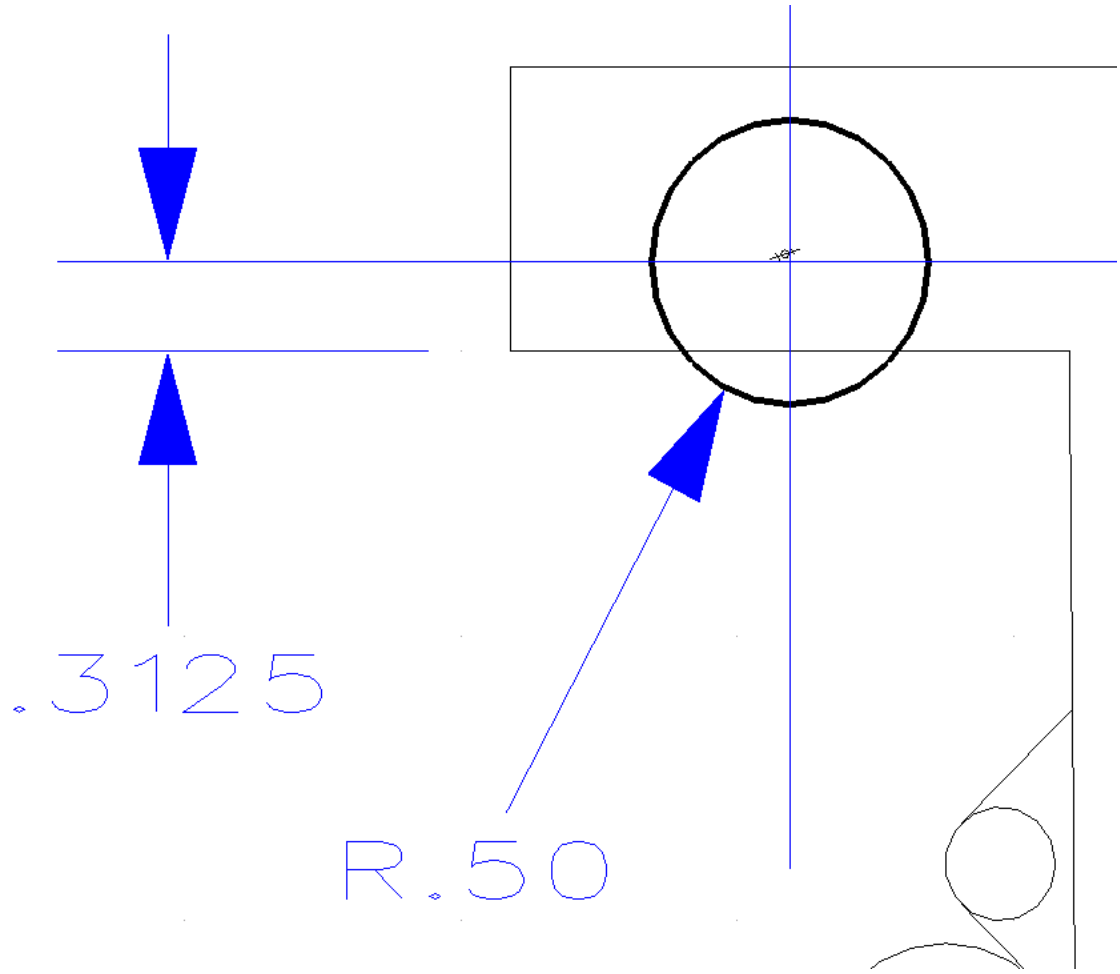

Note: You can optionally delete the lower parallel constraint and add a 65 degree angular dimension as indicated in the figure above.

Sketch should still be **Fully Constrained.**

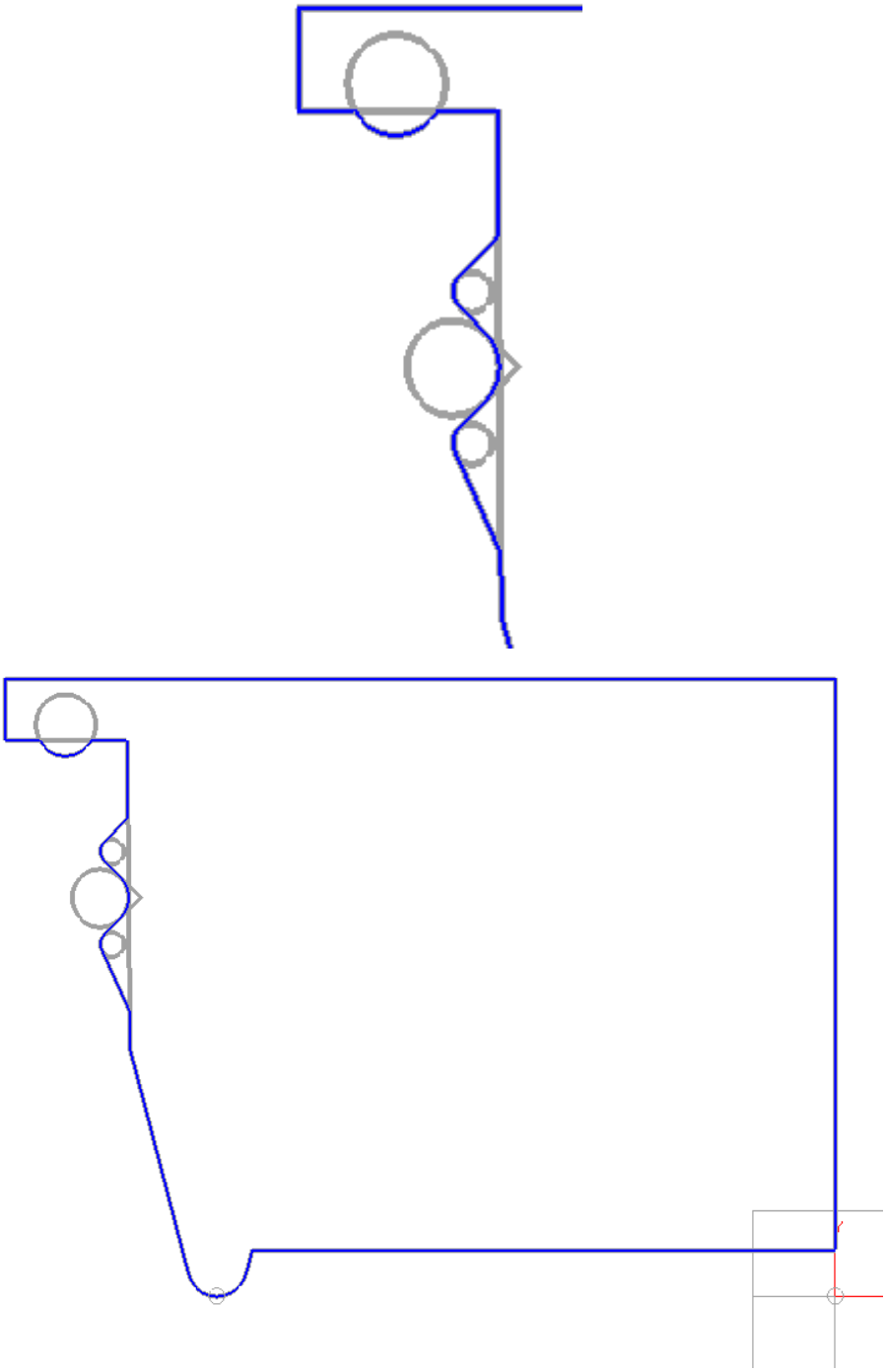
We need one more circle created

- Create the geometry and dimension as shown in **bold.**

-  Constrain the circle to the midpoint of the horizontal line but only in the X direction.



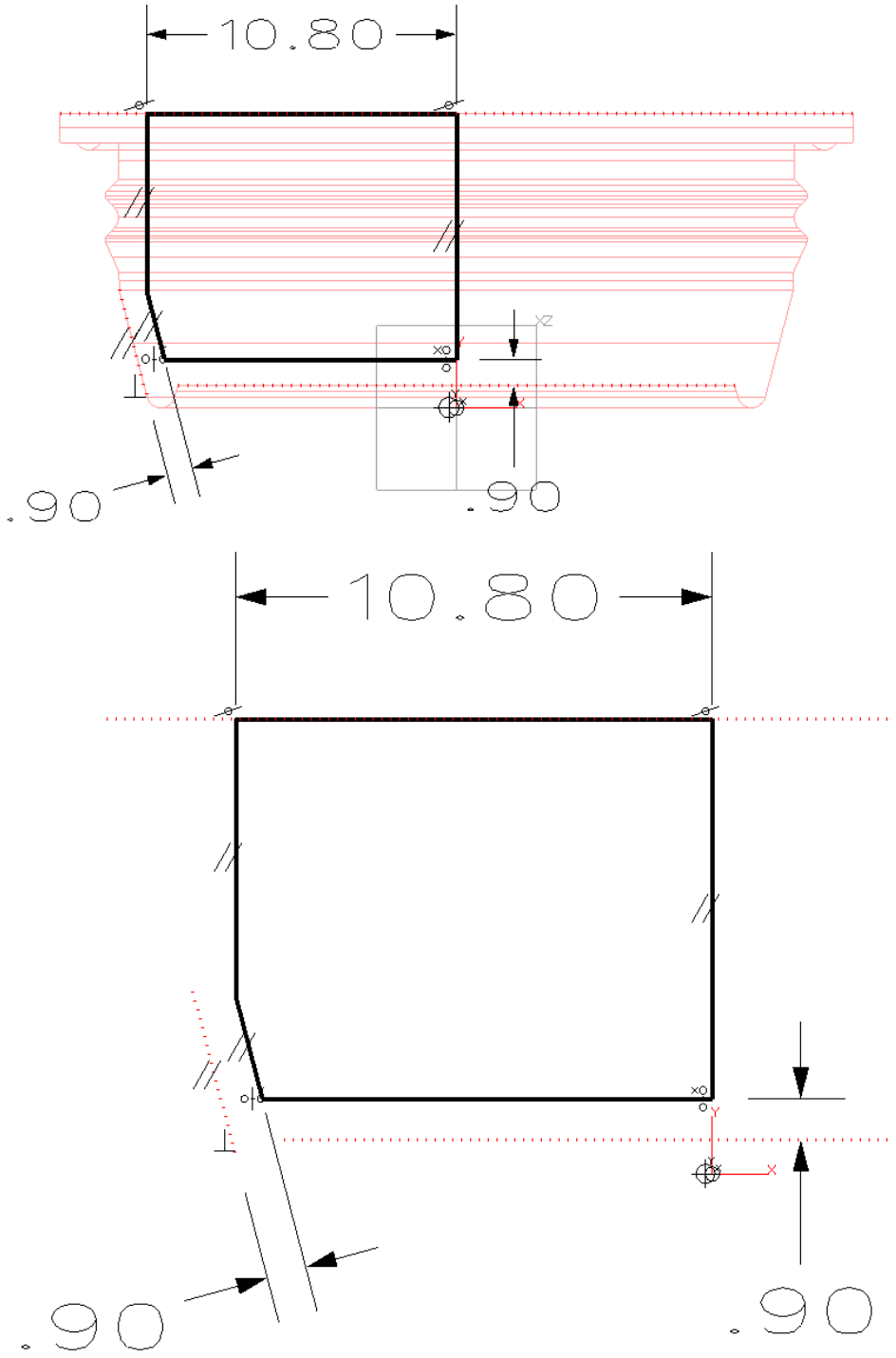

Sketch should still be **Fully Constrained.**
**Note:** This is the key to creating a Fully Constrained sketch. Break your design down into logical steps and constrain as you go. Use fillets where it makes sense.
Use the "**Trace profile**" command to create the trace profile shown in blue on figures shown. 



Exit the sketcher and then complete the revolved solid command, use the Z-axis as the revolve axis.

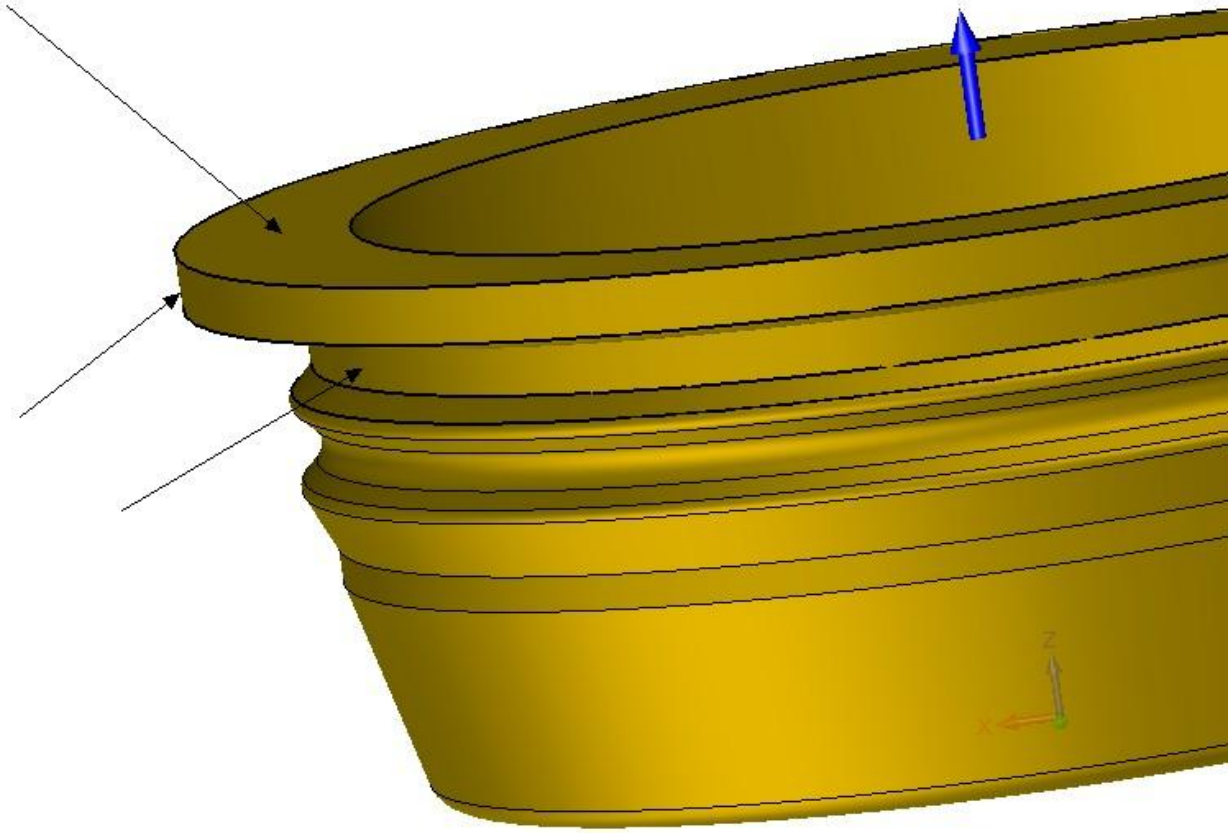
The revolved solid is shown in the figures below.

Use the **Revolve** **- Remove** command to produce the cut-out about the Z Axis as shown, note the sketch uses reference geometry from the existing solid.

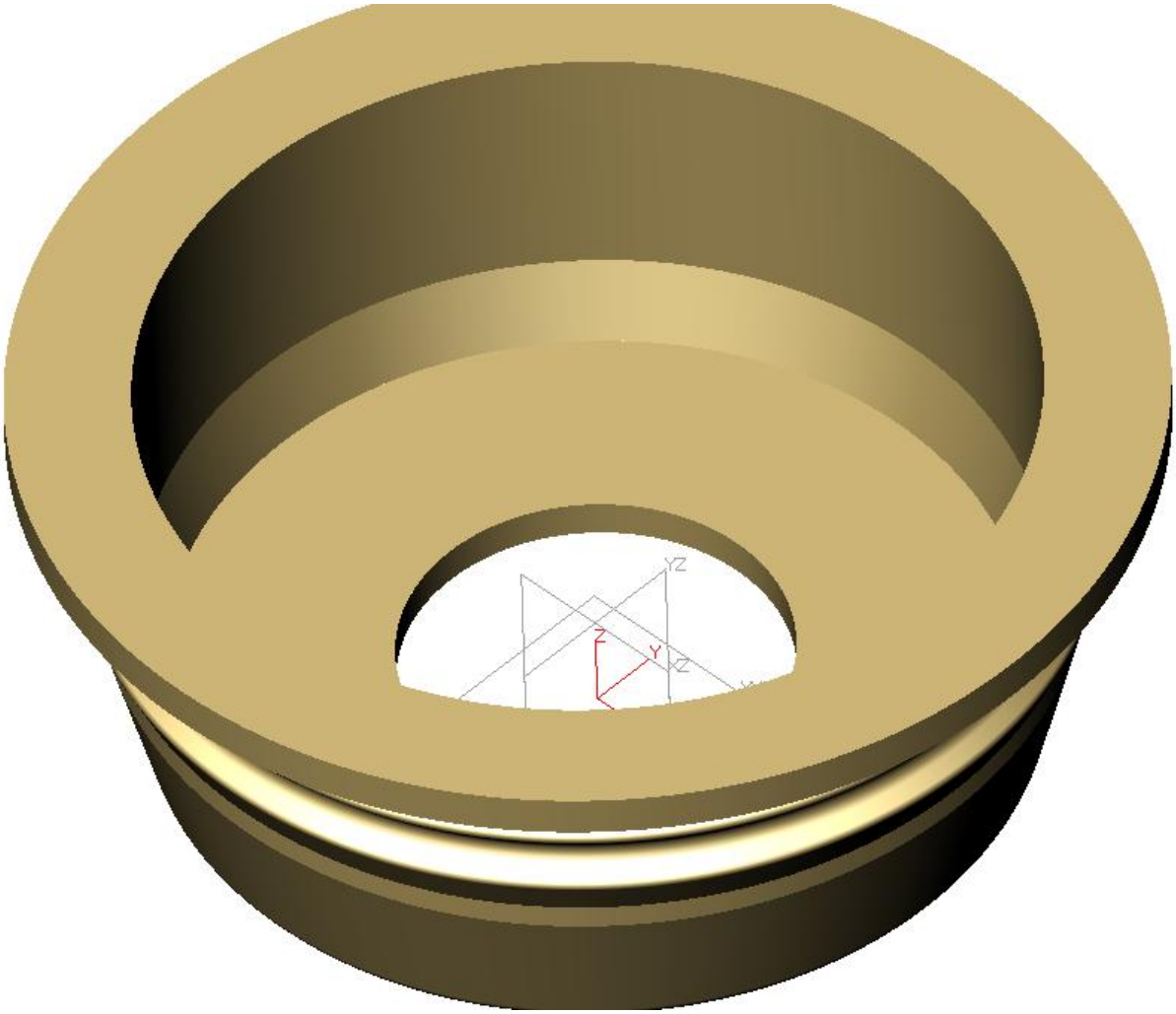


Use the “**draft**” command to create a draft angle of **+0.5** degrees as shown below.

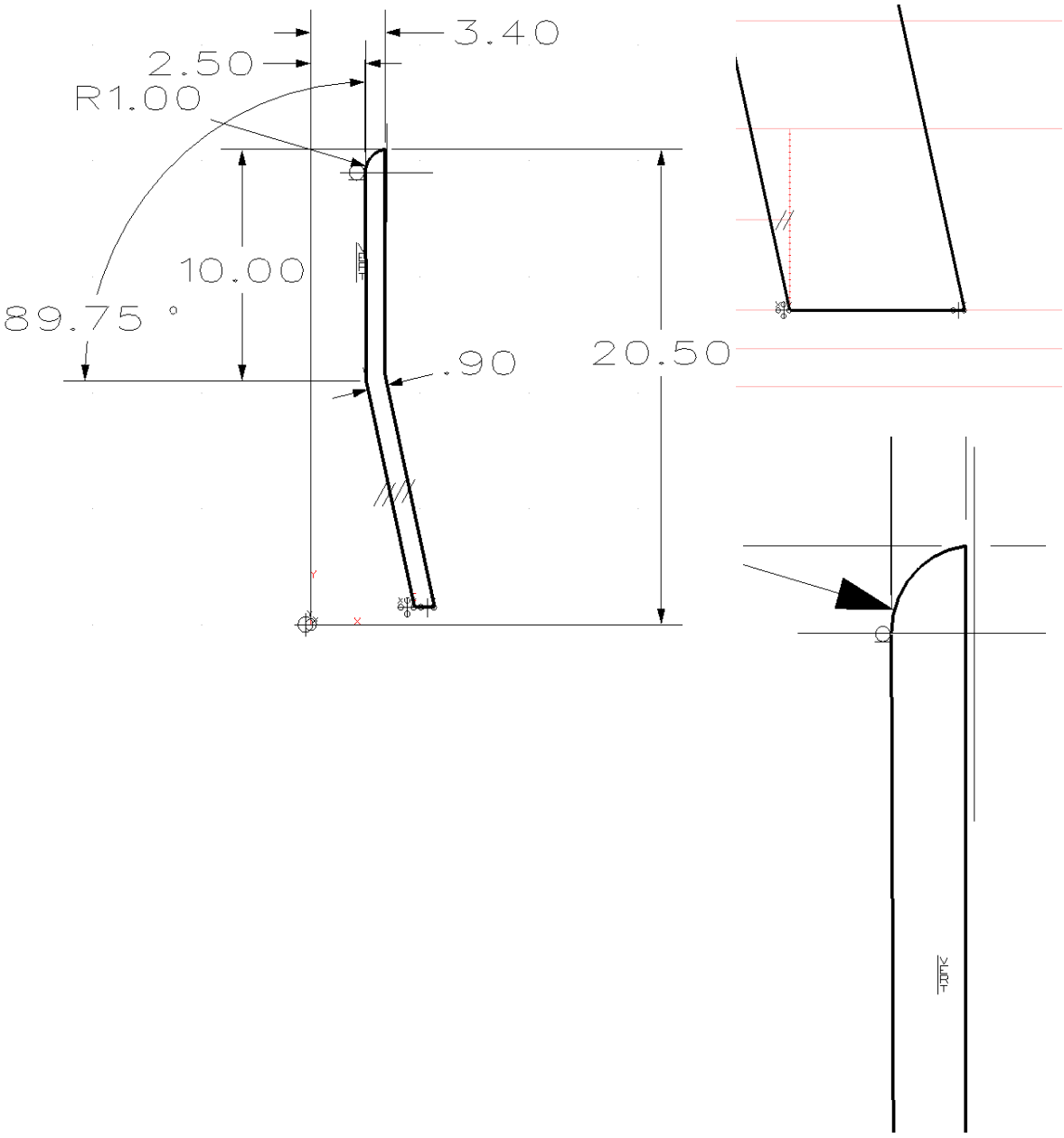
- The **Draft Entity D** (need to use the filter) select the top face.
  Pick the “**Faces**” option and select the two faces of outer cylinder indicated below



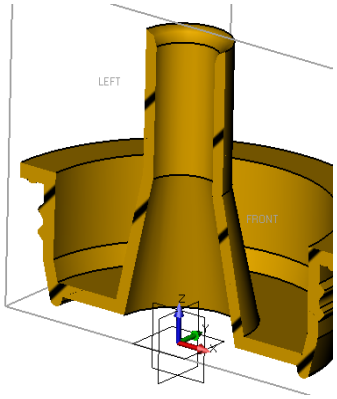
Create a **9.5**mm diameter hole through the center of the part as shown below.

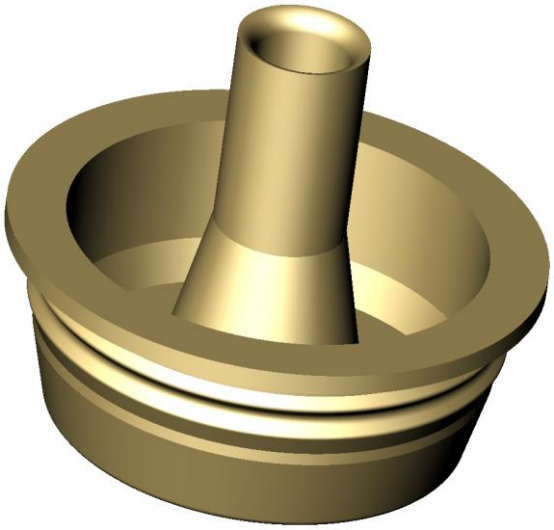



- Create a **revolved boss** on the **XZ** Plane. The sketch is shown below, note the right hand figure shows the reference edge in dotted red, which is used to anchor the sketch to the solid. Also note the point at zero (0,0).
- Note also that the vertical constraint is attached to the right hand vertical line. The left hand line is controlled by the 89.75 degree dimension.



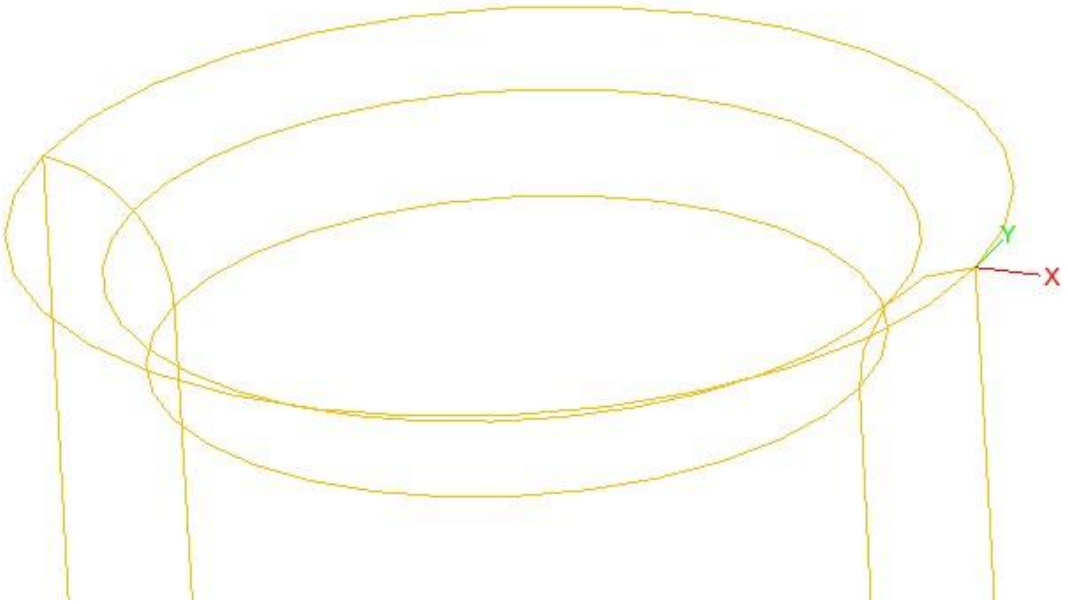

Exit the sketcher and complete the Revolve - Add command as shown below.



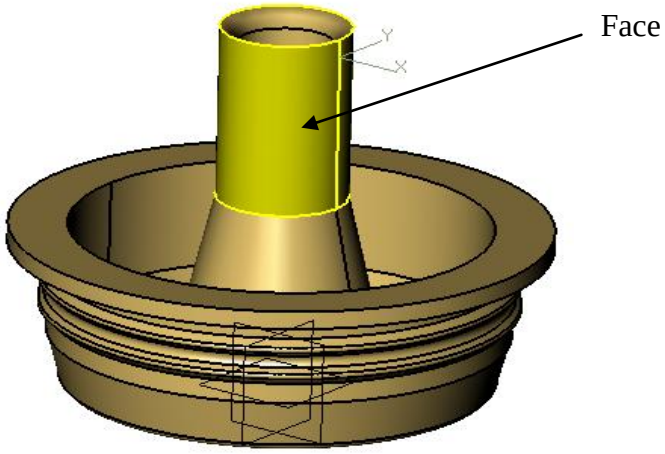



Next we will create a 3 degree draft on the outer face of the previously created boss.

- Right click > Insert Datum to Create a **datum** plane through a point, use the right hand mouse button to select critical and locate the base plane as shown below.

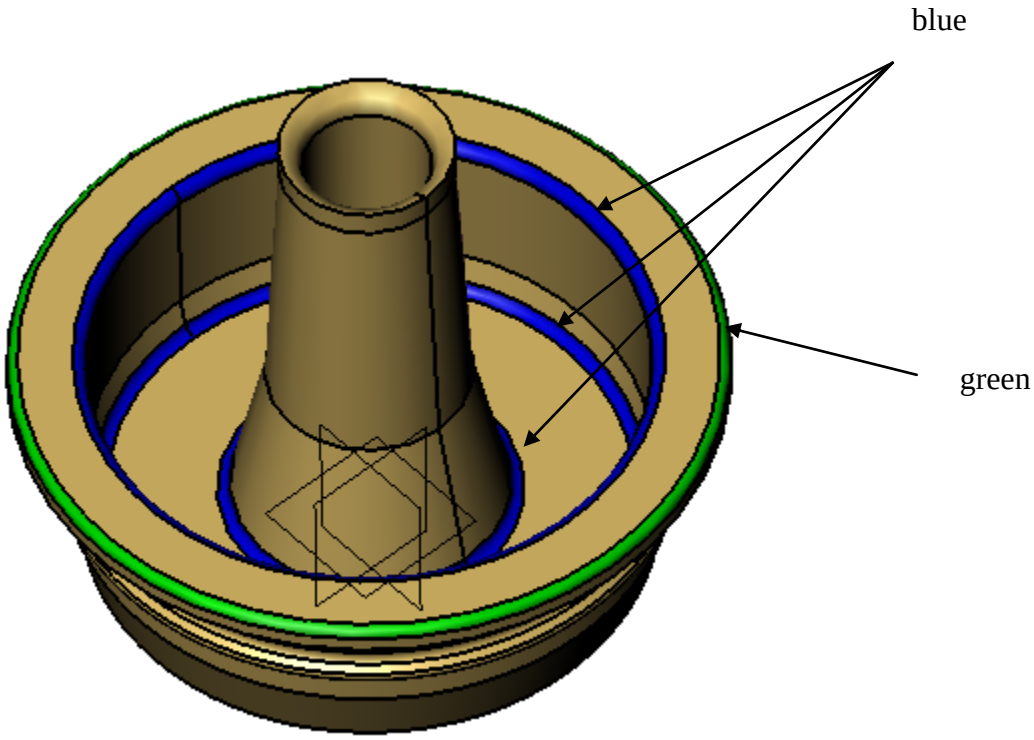


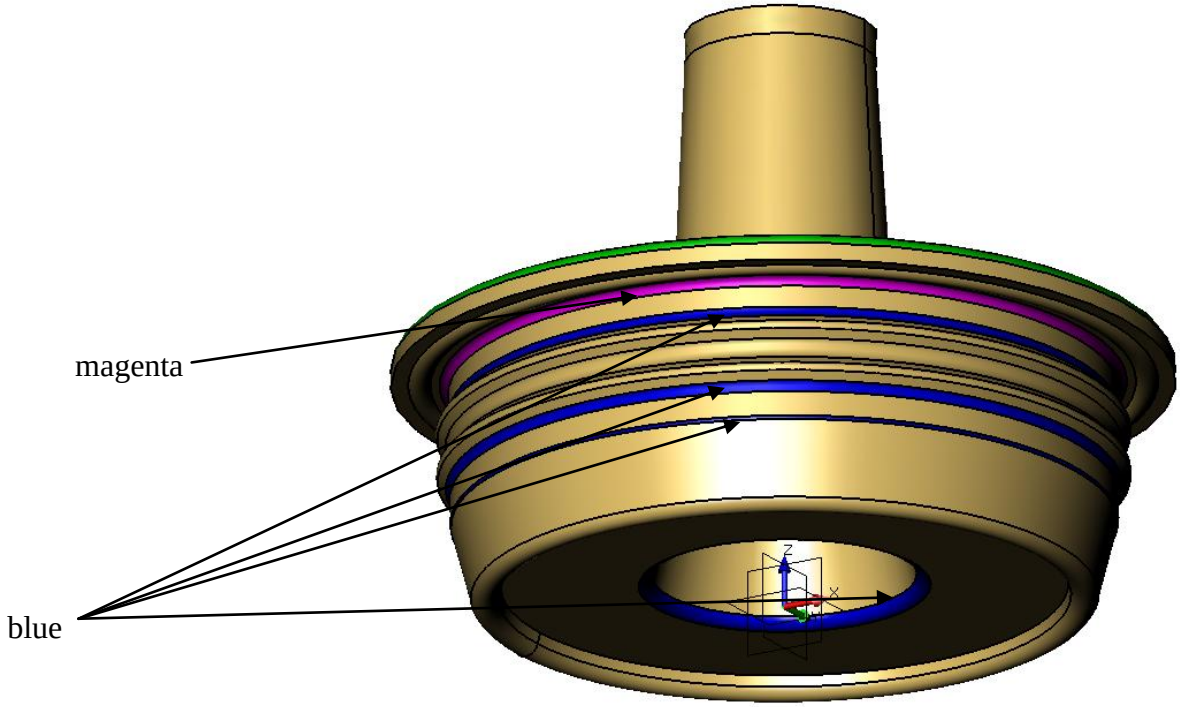
- Select the “**Draft**” command.
- “”Pick the new datum plane.
- The draft angle is **minus 0.5** degrees.
- Pick the “**Faces**” option and select the face indicated below. We definitely don’t want to draft the entire solid; the inside wall already has draft.
- The reference plane in this example does not need to be selected; it is the same as our datum. If we had elected to draft about a parting line then we would have to pick the reference plane.




Create the fillets shown below.

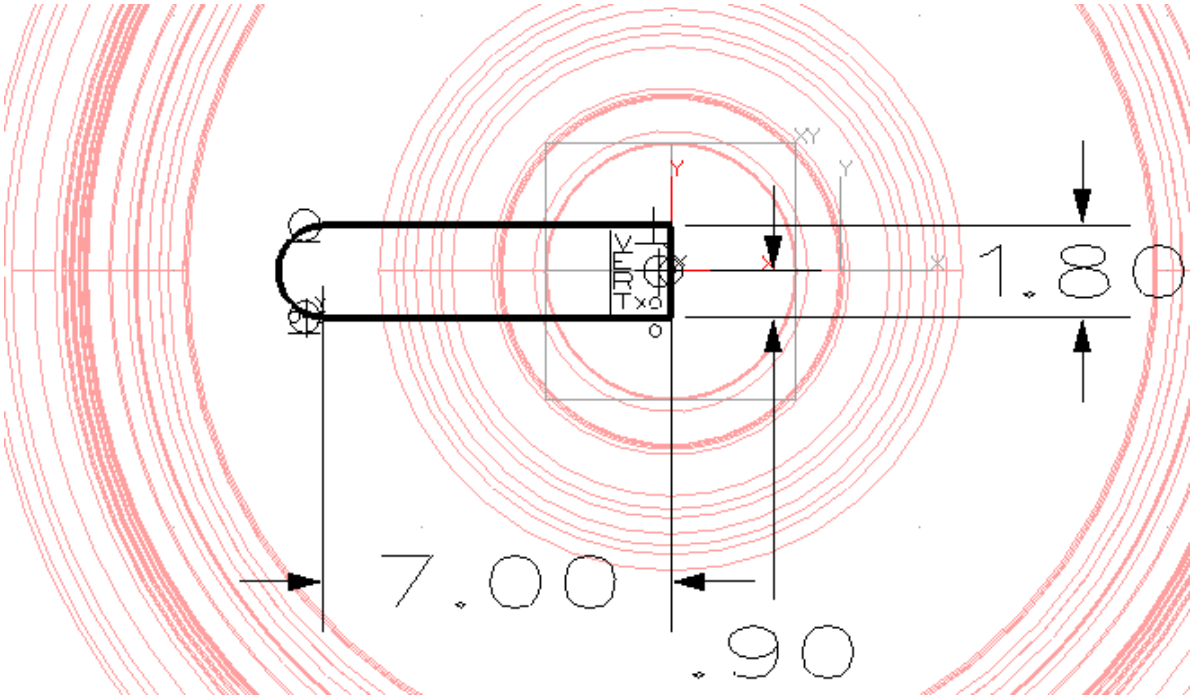
- Seven filets radius **0.5**mm as shown in **blue**.
- One fillet radius **0.3**mm as shown in **magenta**.
- One fillet radius **0.4**mm as shown in **green**.


blue
green


magenta
blue

Next we will create the **plug** feature shown in the upper left view as shown in the original drawing on page 3.

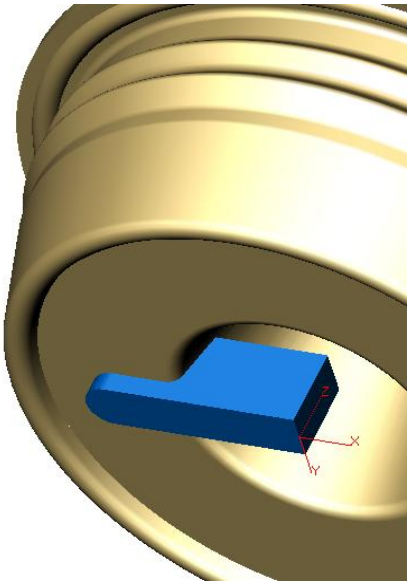
- Ues the **Extrude** command and pick the **Add** option .
- Create the sketch on the **XY** Plane as shown below.
- Notice the dimension 2.25mm for the height of the feature is actually given from the step on the underneath of the part. However as an exercise lets create the feature from the XY Plane instead so we can use the bounded Boolean command.
- Exit the sketcher and extrude the feature, start=**0** end=**3.0**mm. **Don't hit OK!**
- While still in the command, shade the model and look underneath.

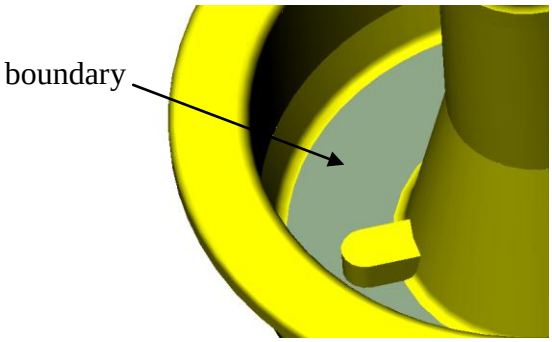



Right now the boss is extending into the center of the hole (see the left view below) as well as where we want it (see the right view). We must limit or set a boundary.

We wish to add the solid to the top side of the part only.

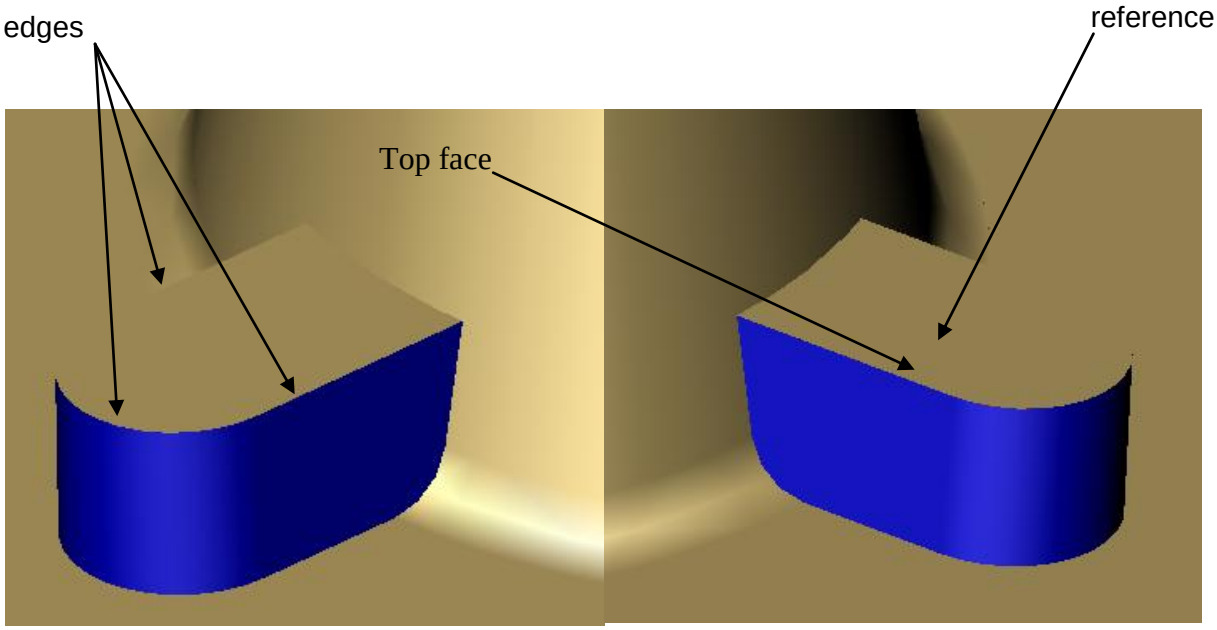
- Pick the boundary define button, select the face shown below in grey and the solid will be added to this side only.
- You only have to pick one of the faces for each complete "trim loop".

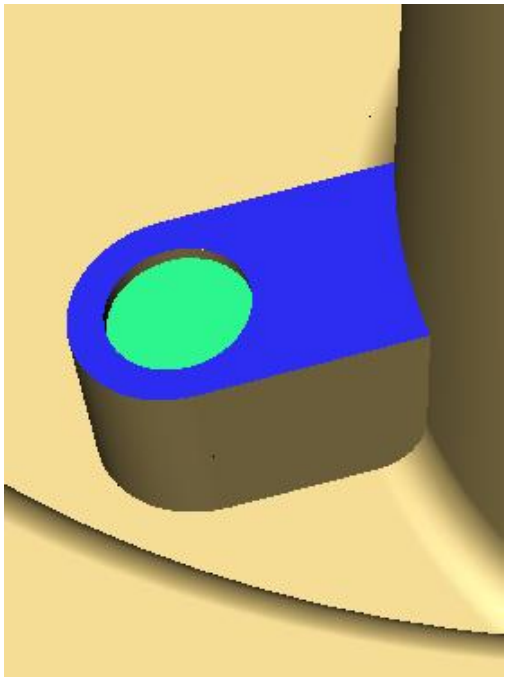





Use the "**draft**" command  to apply a **1** degree draft to the three vertical faces of the previous feature as shown below in Blue.

- "Draft Entity  D" select the top face.
- Enter a draft angle of **minus 1** degree.
- Pick the "Faces" option and select the blue faces indicated below





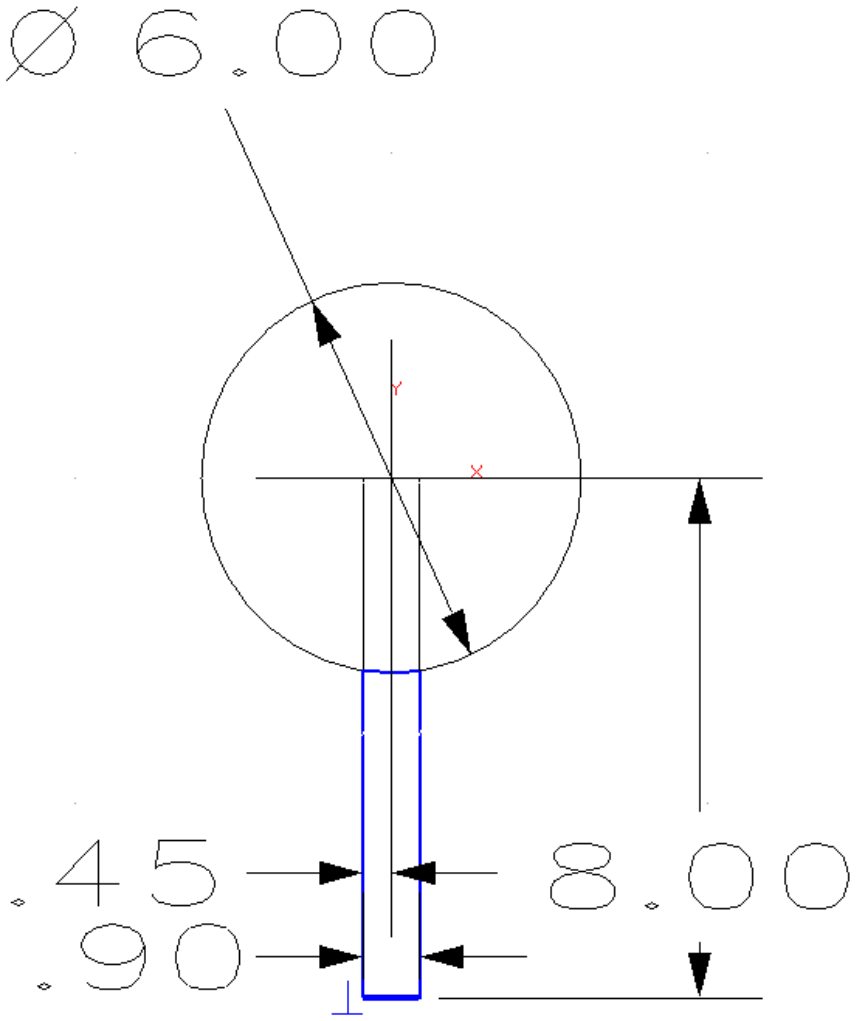
Use the **Extrude** **- Remove** command to create the 1.2 dia x 0.12 deep. Gate well, as shown opposite.

- When creating the sketch pick the top face as shown in blue. Make sure you have the “edges on” option set to automatically create reference curves from the face edges.
- Locate the sketch circle to the reference edges.

Exit the sketcher and complete the extrude.

We will now create 6 ribs with correct draft angles on the inside conical hole.

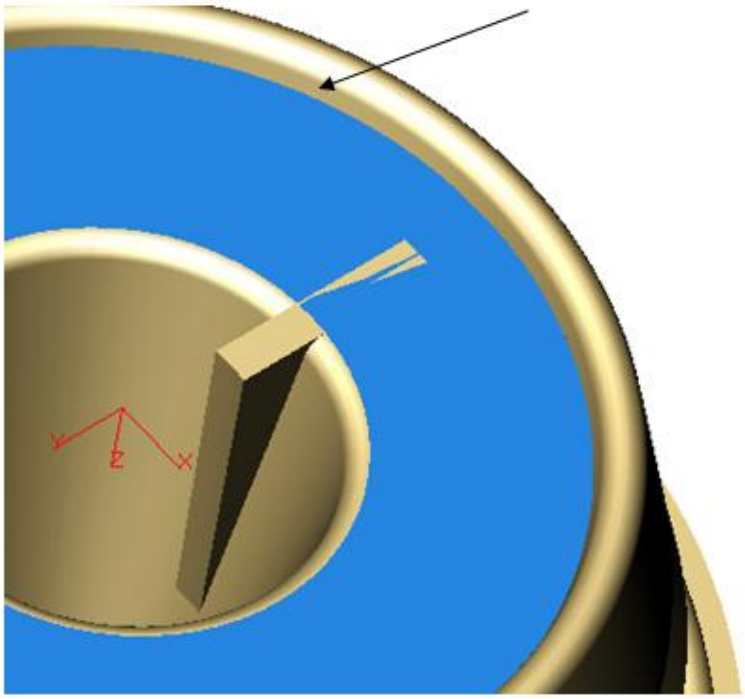
- Select the **Extrude** **- Base** command.
- Create a sketch on the **XY** Plane as shown below.
- Use the **trace profile** command to select the correct profile as shown below in blue.
- Exit the sketcher

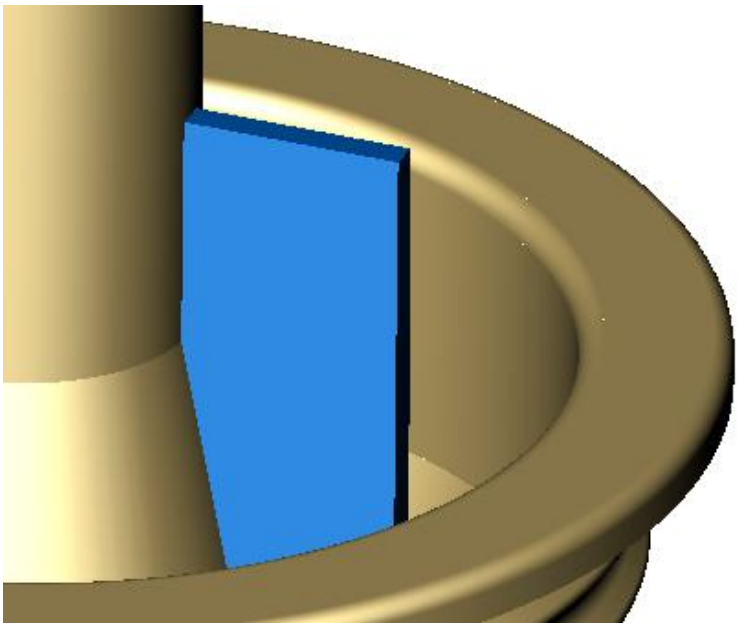



Complete the extruded command.

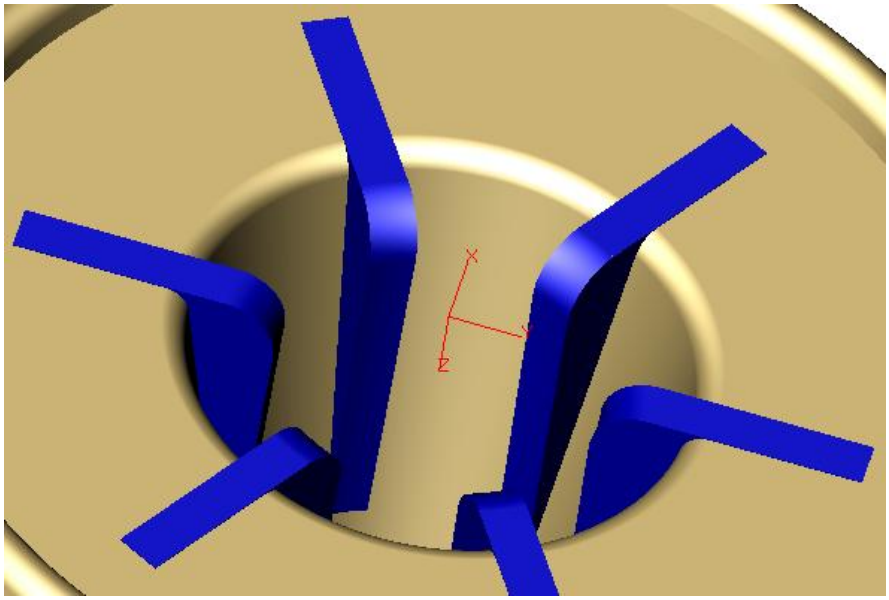
- Start : Use the **right mouse** option and select "**To Point**" . Then **right click again** and pick "**On Entity**" and pick the face shown below in blue.
- End: **15** mm.
- Draft angle **–0.5**.

The first rib is shown opposite and below. Note that the rib protrudes through the part at this stage.





- Create a **1**mm **fillet** on one of the edges of the newly created rib solid.

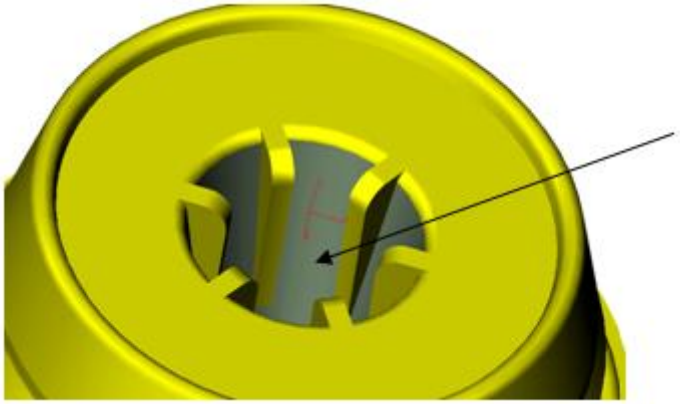


- Pick **EDIT > Pattern**  from the pulldown menu, and choose **Circular** option.

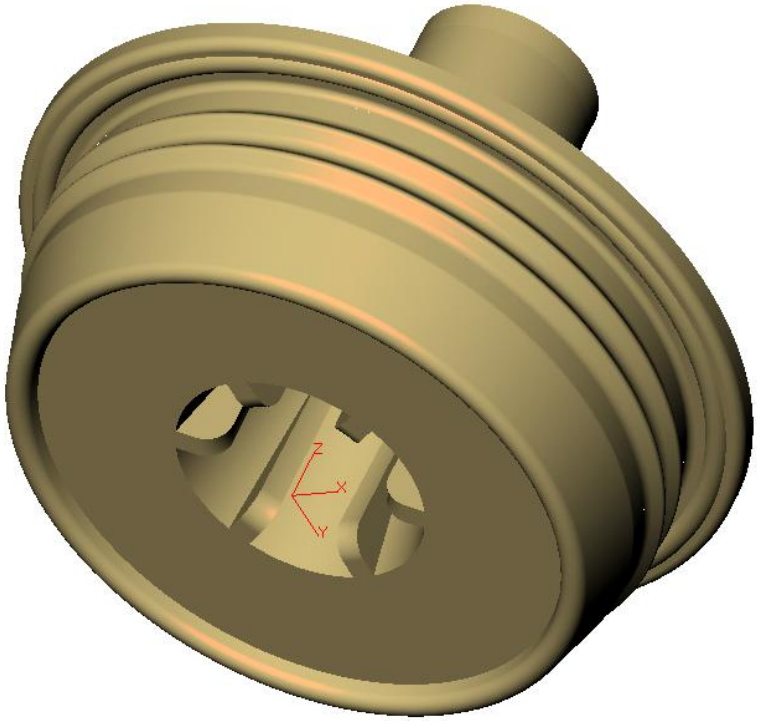
- Pattern the rest of the ribs as shown below.
- Put your **filter on shape** before picking the rib
- about the **Z** Axis.
- Number of copies = **6**
- Angle = **60**

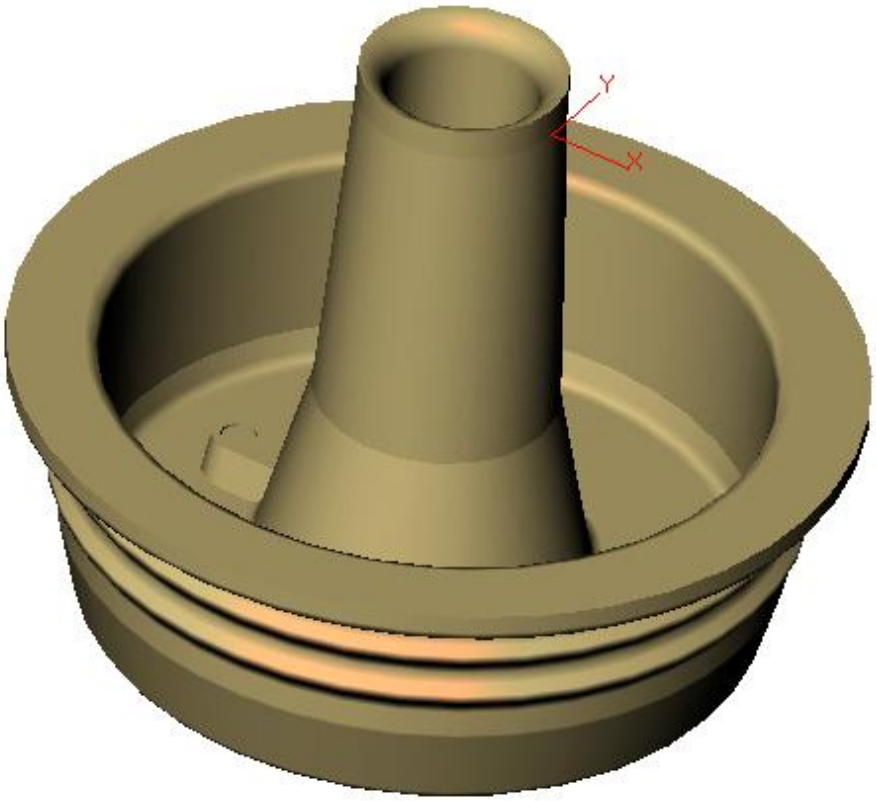
Next we need to add the six ribs to the plug.

- Select the **Combine Shape**  **> Add** command  .
- Pick the main body of the plug as the base
- Pick the six ribs as the shapes to be added. (There are 3 ways to pick them?)
  Try right-click **pick all**.
- Pick the internal conical face as the "**boundary"** as shown above

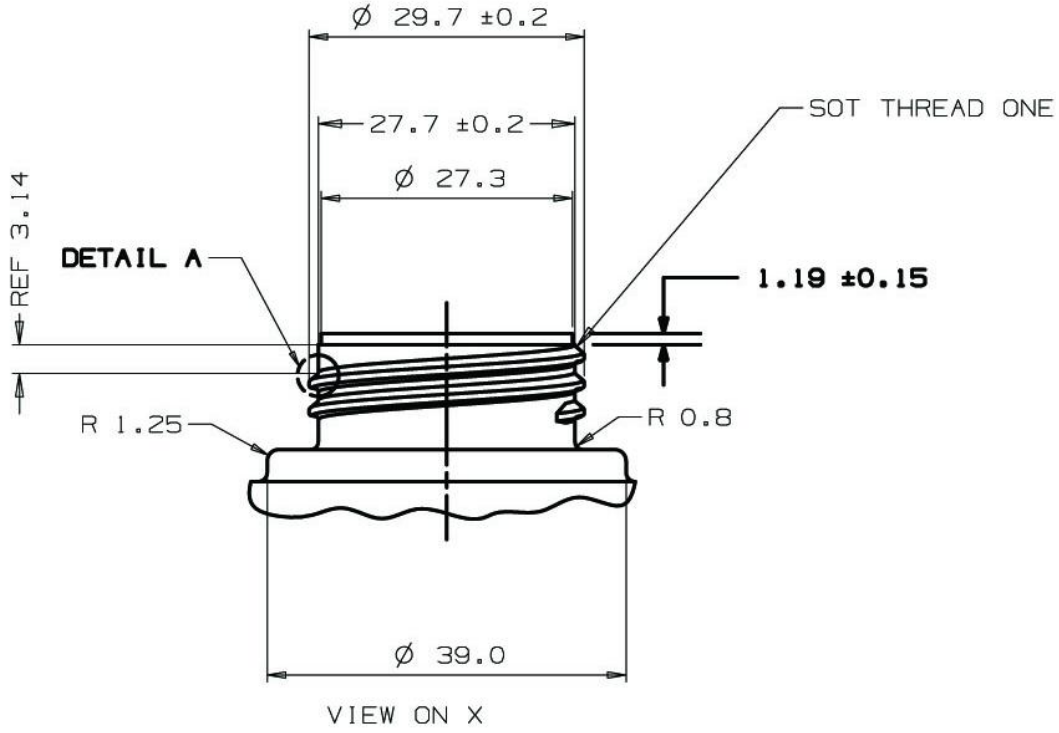


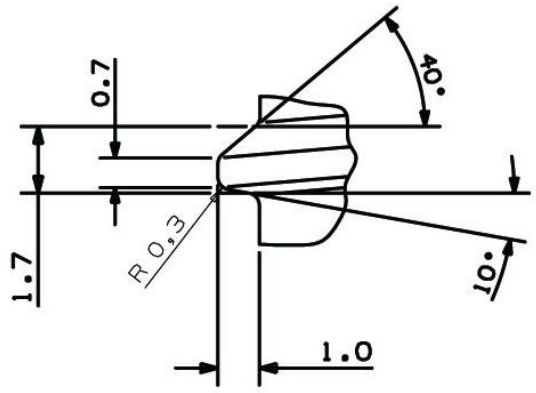
The finished plug is shown below.




Y
X

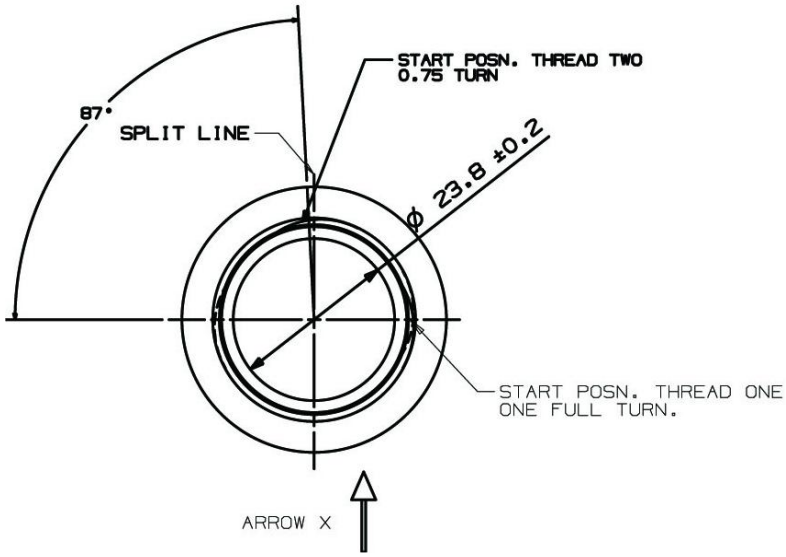
## Example two, The Neck.

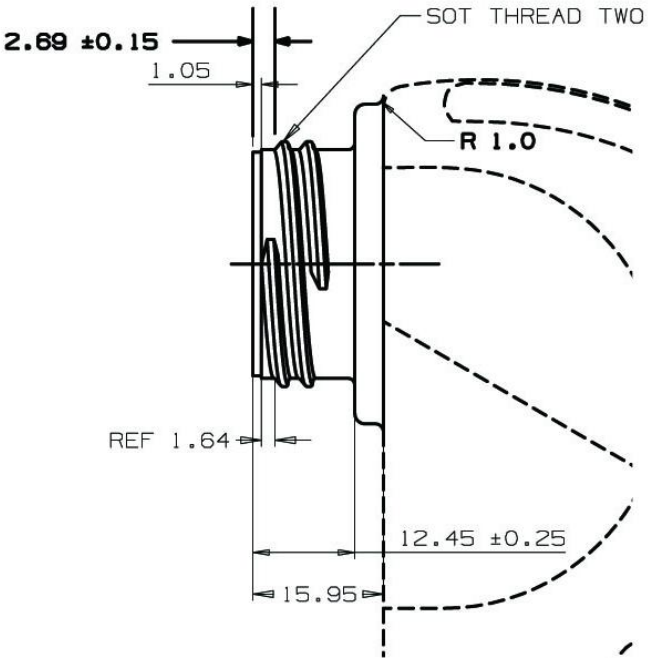



SHOWING THREAD DETAILs

THREAD: 2 START R.H.
PITCH 3.0 LEAD 6.0



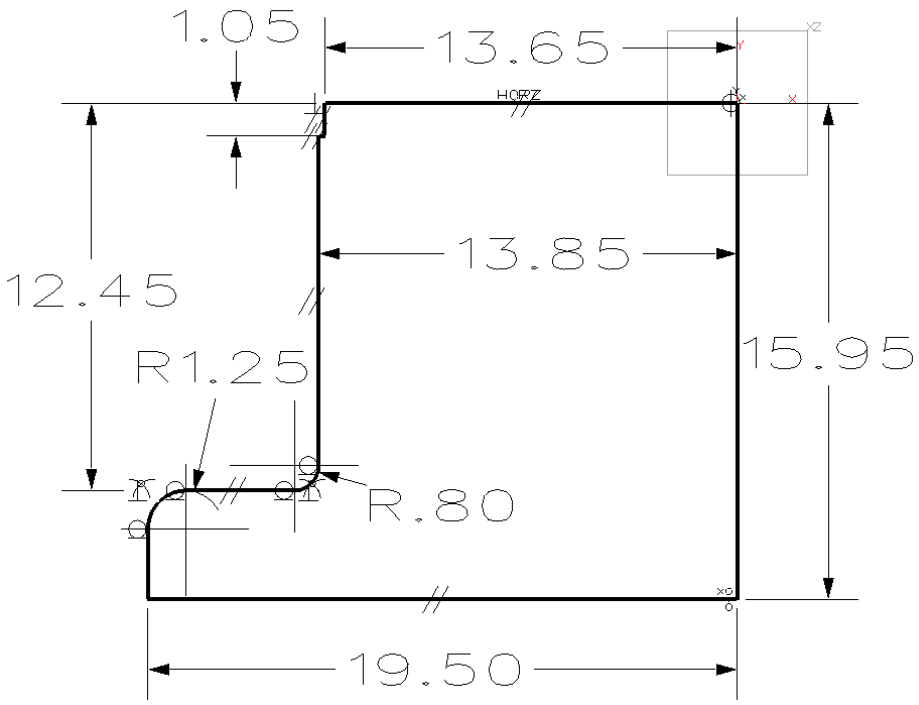



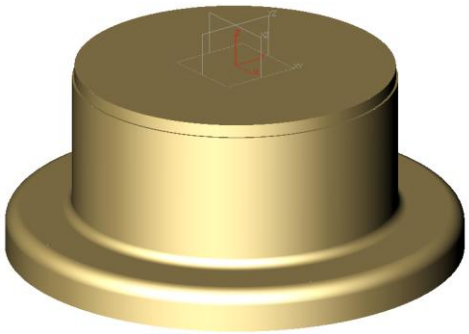

## Create a new part called **“Neck”**

This part is small so when using the sketcher you may find it helpful to set the **grid spacing to 1mm**.

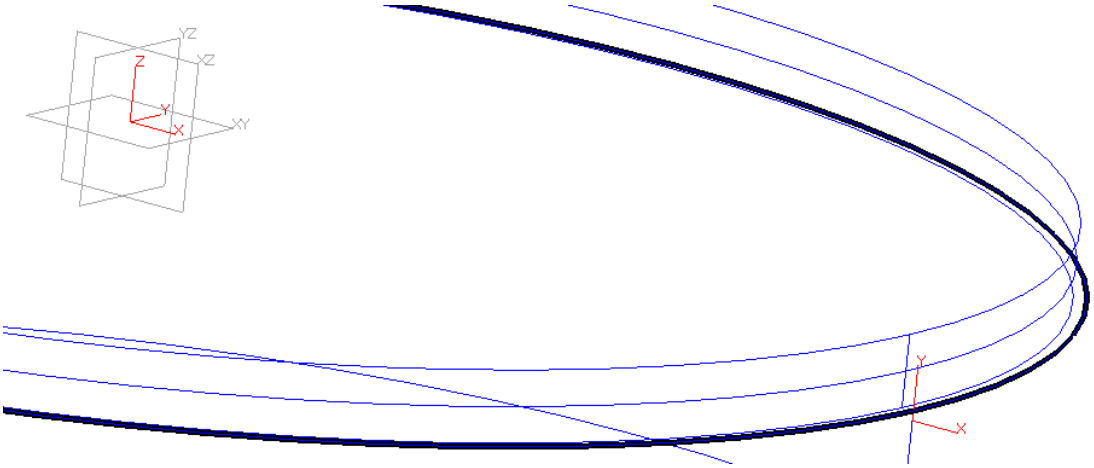
Let’s start by creating the main shape of the neck. This will be a **revolved solid**.

- Select the **Revolve** command.
- **Create the sketch** shown below on the **XZ** Plane.
- Zoom in closely (the part is really small). Make the number in the lower left corner of the screen, approximately 20. This helps to sketch to scale.
- **Exit the sketcher** and complete the revolve command.
- **Revolve** about the **Z Axis** (right mouse button).

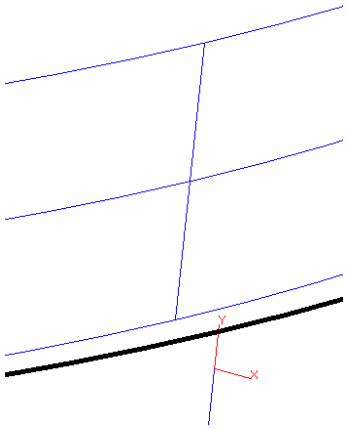


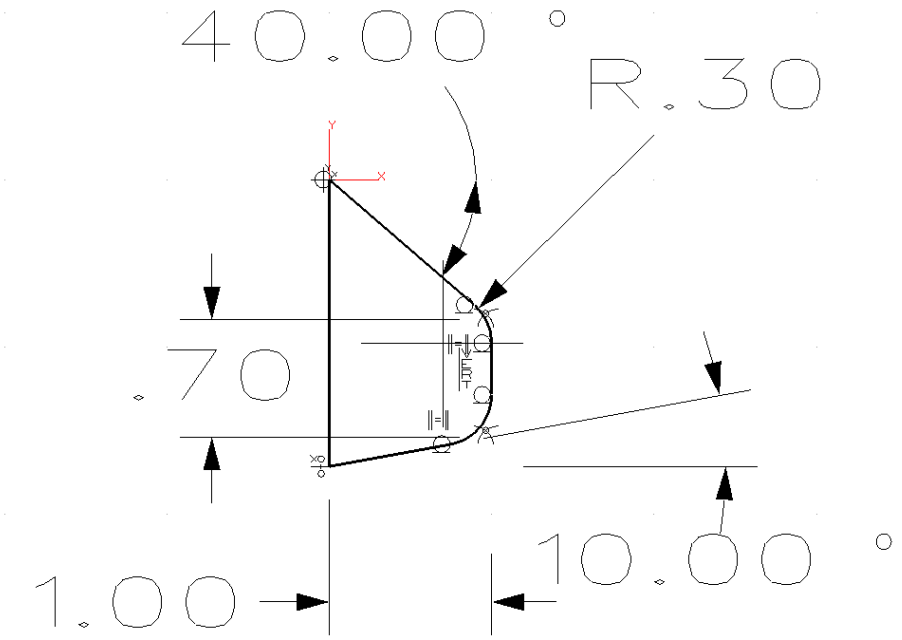
We now need to consider the two-start thread. We will create a datum plane in the correct position. This plane will eventually be used to locate the sketch of the thread profile shown in detail A of the original drawing.



- Right Click **Insert Datum**
- Pick the “Align to XZ of the geometry frame” option. 
- Offset **0**
- Pick the **Origin** Define button
- Follow closely; we are going to do this in one command. See the position of the datum in the images to the right. There is no critical point on this side of the model.
- Right-click **Critical**
- Right-click **Offset**
- Right-click **Along**
- Pick the edge shown
- Pick **OK** to accept the **50%** setting
- For the plane pick the **XY**
- X=**0**;Y=**0**;Z=**-0.18**
- **OK**



Create a **new sketch** as shown below. Locate the sketch on the previously defined plane.



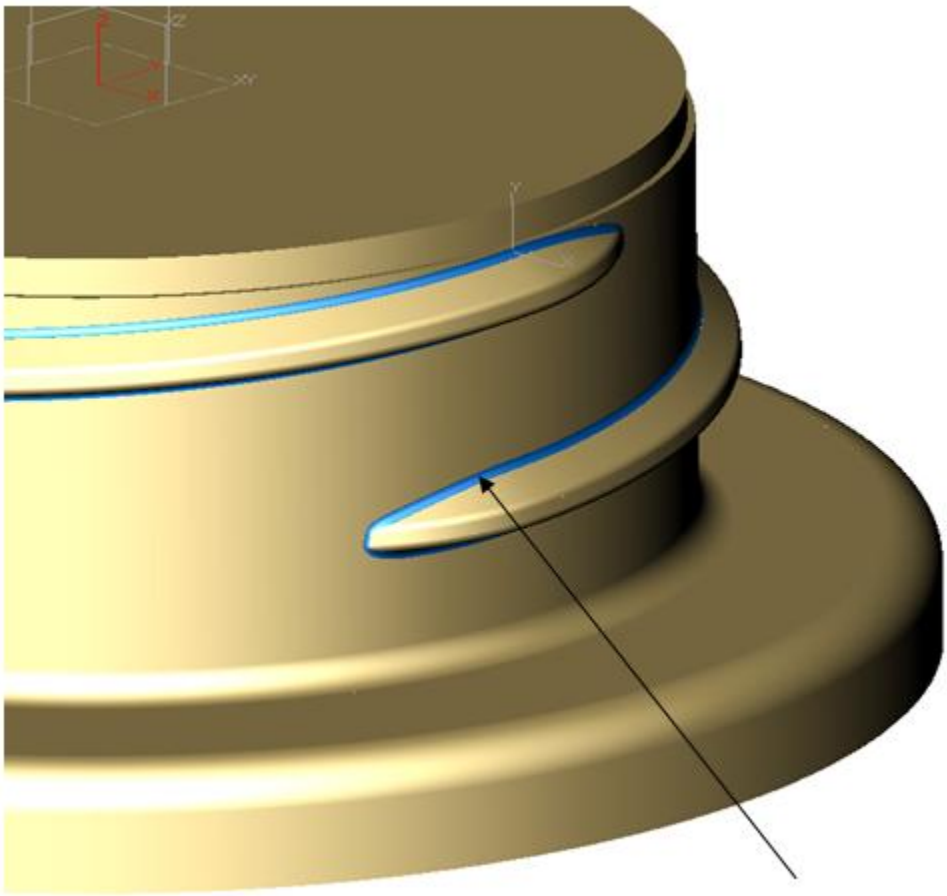

- We will shortly use this sketch again, but located on a different plane as this is a two-start thread.
- If a particular thread form is used often, a global sketch could be created and used on multiple parts.
- However, for our example we will copy the sketch into the sketcher's clipboard which we can then later paste onto the new correct plane.

- Select “**Copy**” from the edit menu and window select the complete current sketch.
- Locate the clipboard copy at the current zero point.
- Exit the sketcher.

Using the “**Spiral Sweep**” command, create the first thread. 

- Pick the add option
- Pick the sketch profile.
- With the right mouse button select the spiral axis, (minus) **–Z**.
- Number of turns = **1**.
- Distance along axis per turn = **6**. (The lead value, not the pitch, because this is a two-start thread).
- Taper angle =**0**.
- Select the lead in/out option as “**in**”.
- Specify lead in/out radius =**5**.
- Specify the lead in/out angle =**90**.
- From the option buttons ensure that the Lead in/out option “**End**” is set to **both**.
- Complete the command

Use chain-select to apply a **0.3**mm **fillet** at the intersection of the thread and the neck as shown below in blue.



Create and add the second thread to the neck.

We have our thread form sketch copied to the clipboard (or perhaps saved as a global sketch).
The location of the sketch needs to be positioned correctly. We will accomplish this by copying the datum plane used to locate the first thread. The new plane will need to be rotated around the Z Axis and also translated down the Z Axis as follows.

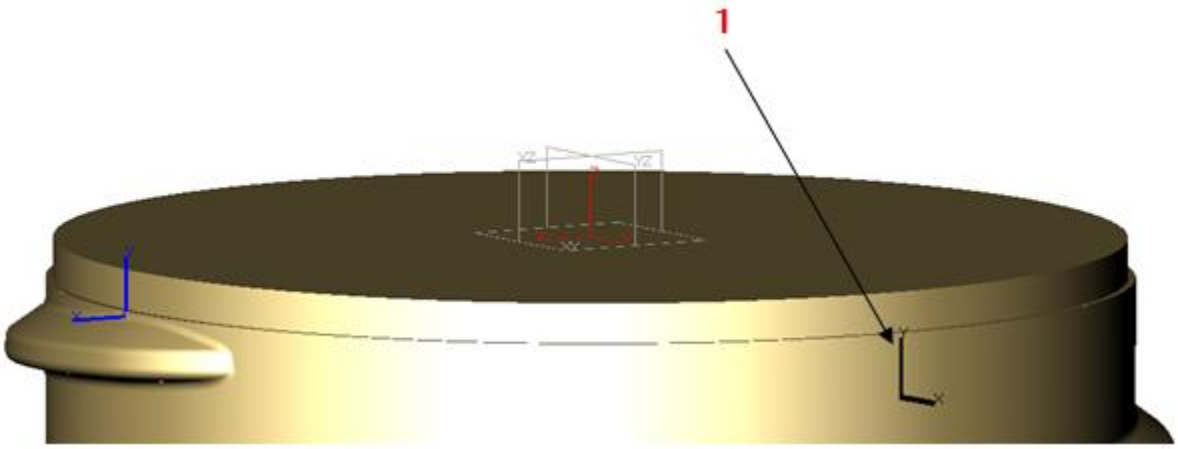
Either create variables in ZW3D or use a calculator to establish the correct amount of translation, as follows.

Translation=Lead/(360/angle)
The lead in our example = 6mm.
The angle (from the original drawing) is 93 degrees counter-clockwise from X. (180-87)
Therefore the translation down in Z for a rotation of 93 degrees based on a 6mm lead is:-

6/(360/93) = 1.55mm

**Copy** the existing plane as follows, using **copy entities along a direction** commamd within the **Copy** command under the **Shape Tab**. 

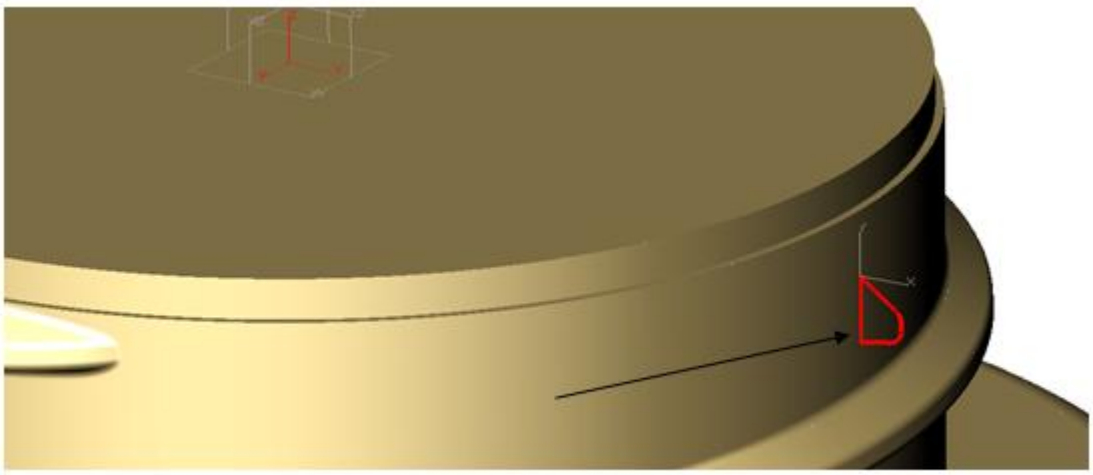
- Number of copies **1**
- Copy around the **Z** Axis.
- Distance to copy = **-1.55**mm (note the minus value).
- Angle to rotate =**93** degrees.




1: New datum

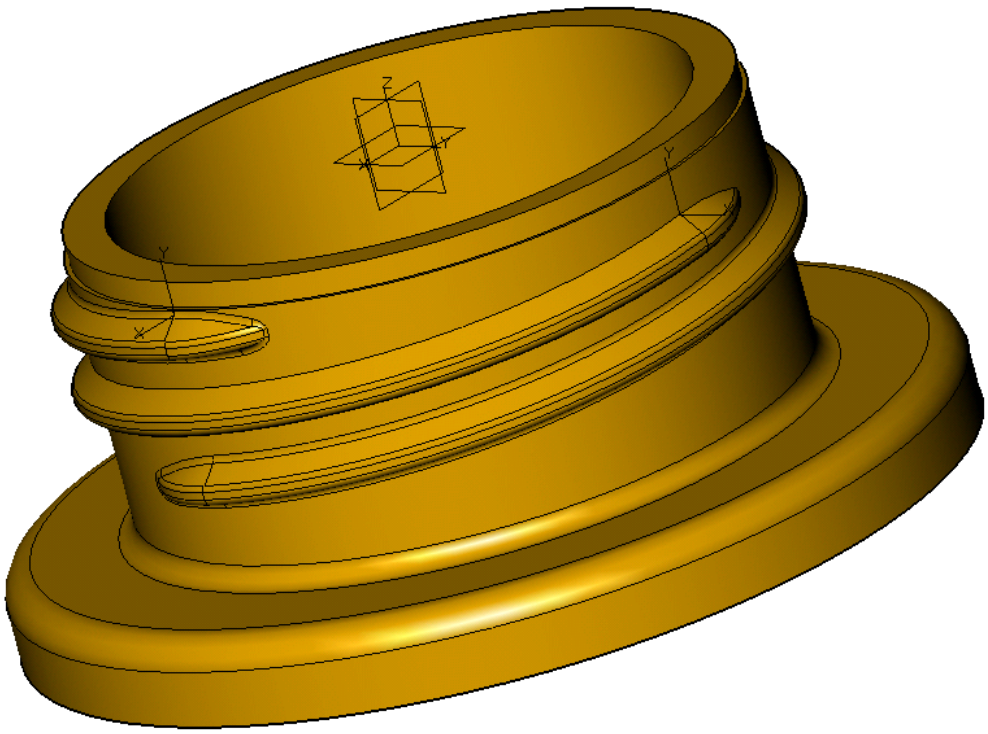
- Use the right mouse button and **insert** a new **sketch** onto the newly created plane.
- Once in the sketcher, use the right mouse button and select "**Paste**".
- Paste the thread form sketch (enter **0,0** for the origin value) as shown in red below.

Note the dimensions and constraints have been blanked.

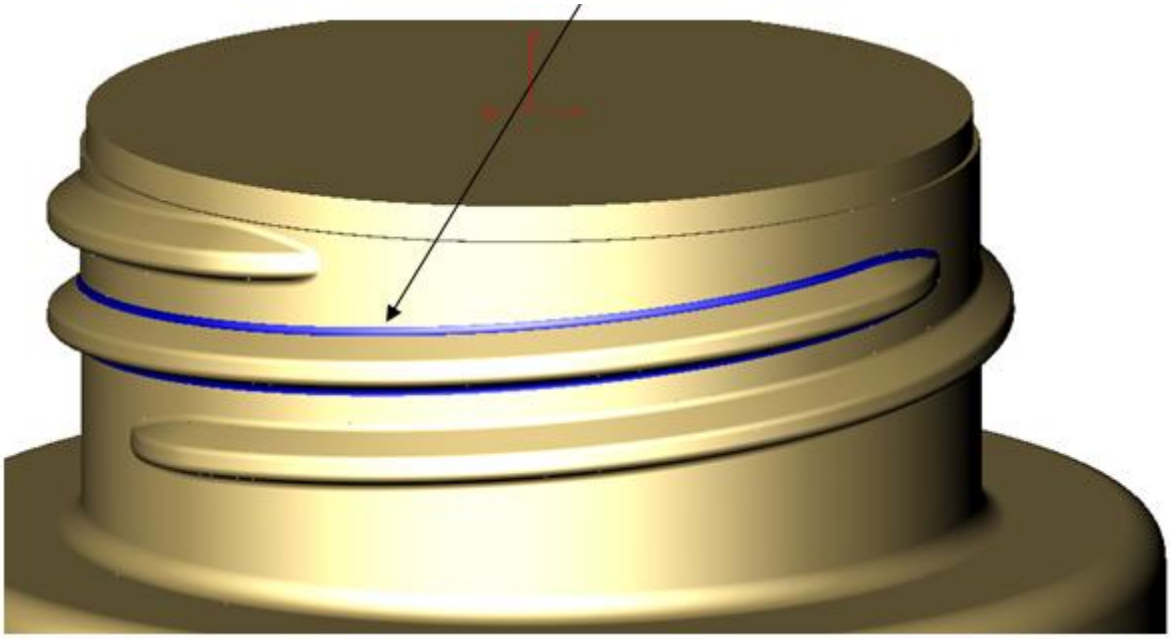


Using the "**Spiral Sweep**" command, create the second thread, as before but notice the different value for the number of turns. Note the number of turns does not include the lead in/out.

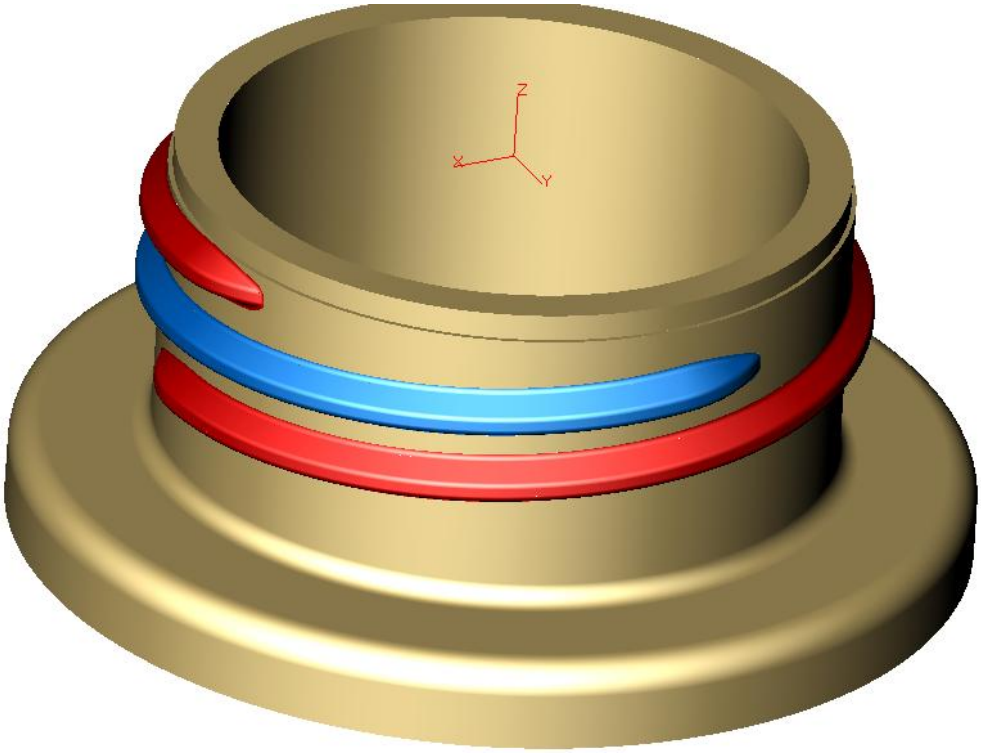
- Again use the Add option
- Pick the new sketch profile.
- With the right mouse button select the spiral axis, (minus) **–Z**.
- Number of turns = **0.75**
- Distance along axis per turn = **6**.
- Taper angle =**0**.
- Select the lead in/out option as "**in**".
- Specify lead in/out radius =**5**.
- Specify the lead in/out angle =**90**.
- From the option buttons ensure that the Lead in/out option "**End**" is set to **both**.
- Complete the command

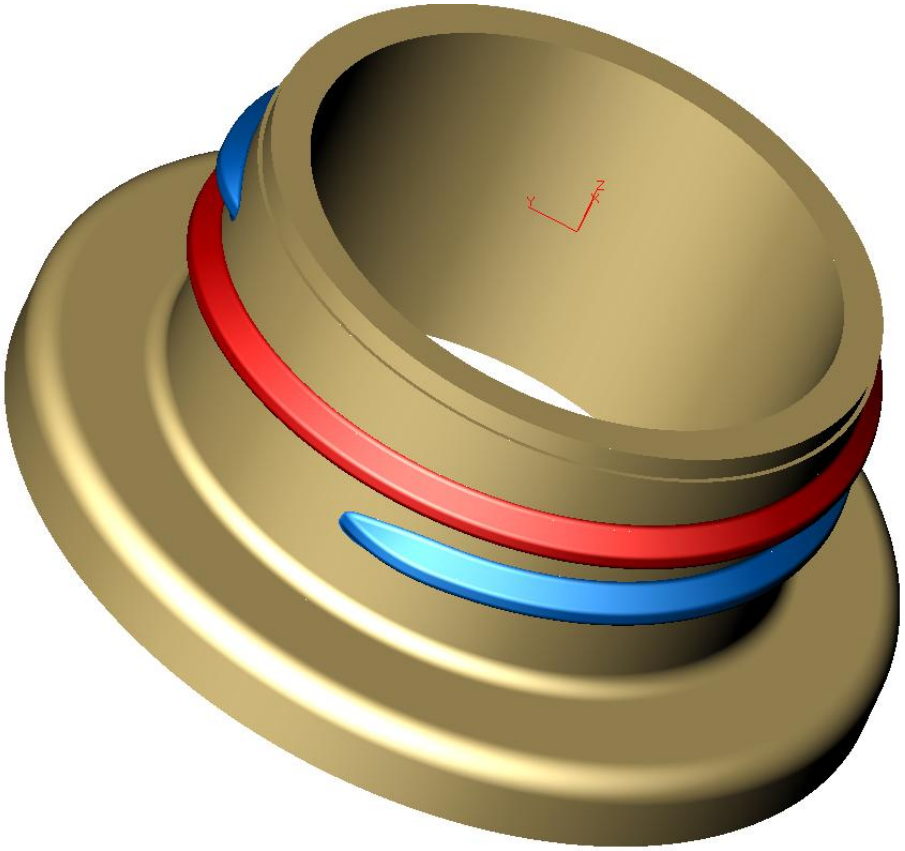


Use chain-select to apply a **0.3mm fillet** at the intersection of the second thread and the neck as shown below in blue.

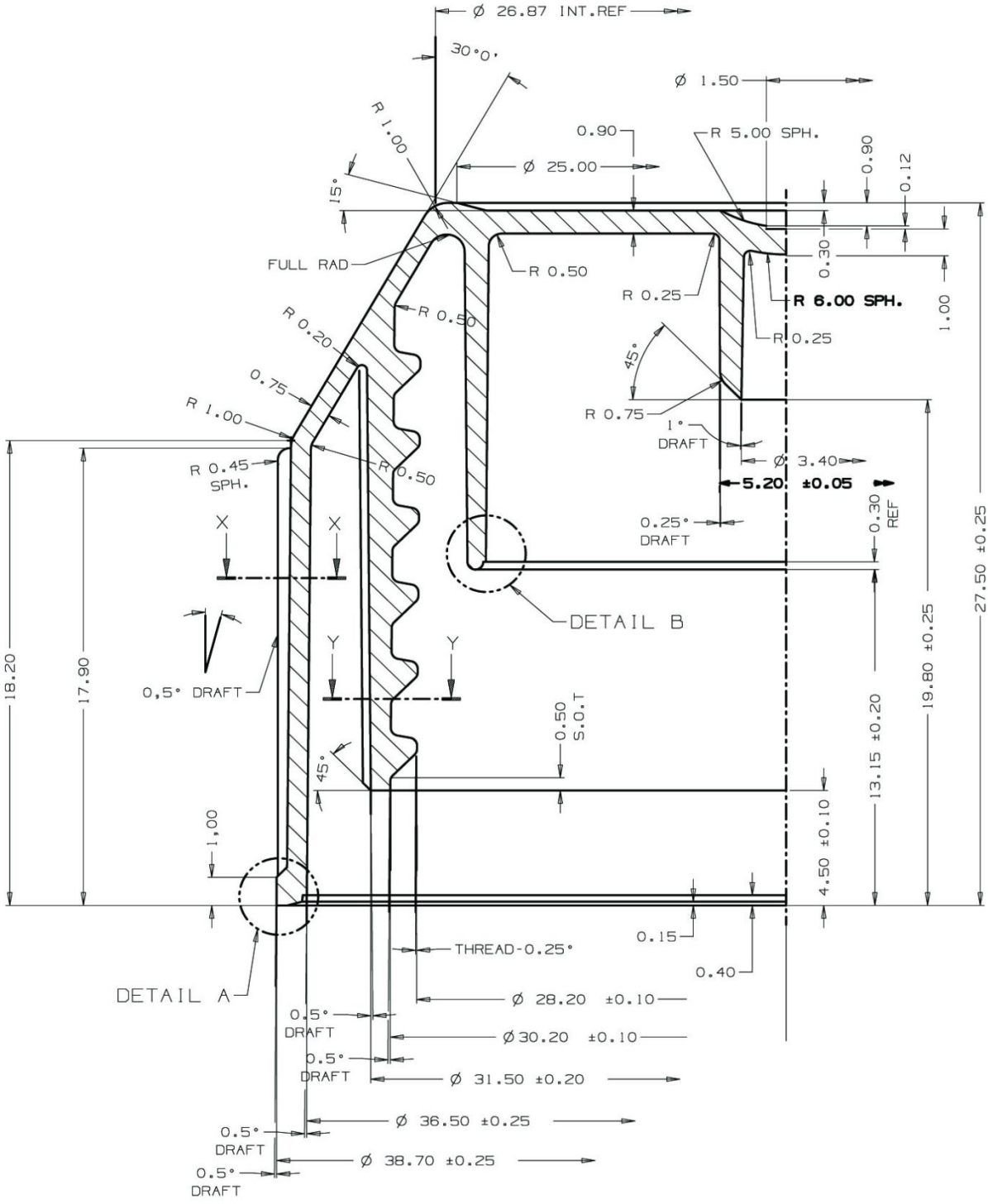


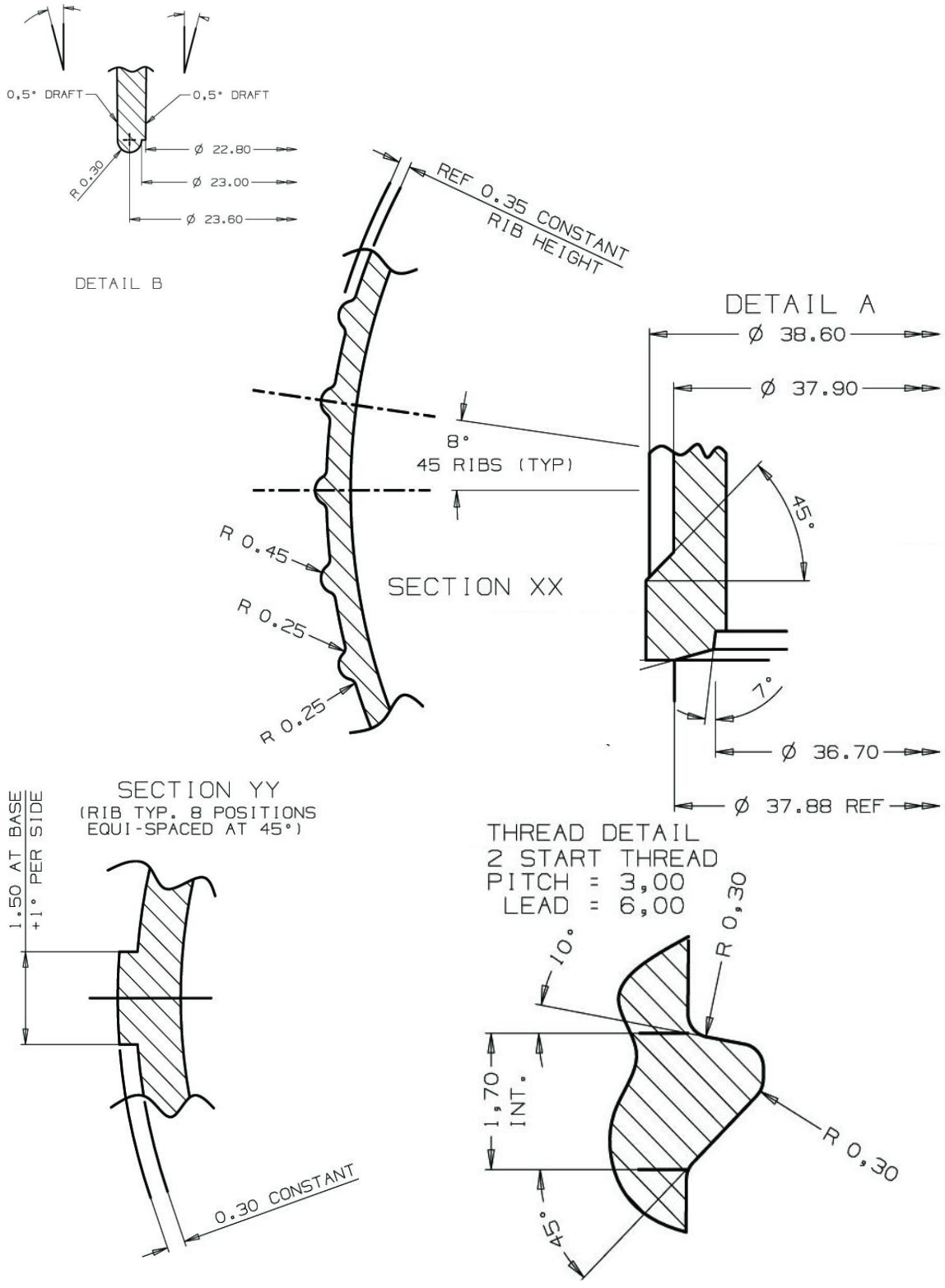
Finally **cut** a **23.8**mm diameter hole through the center of the part.





## Example three, The Cap




0,5° DRAFT
0,5° DRAFT
Ø 22.80
Ø 23.00
Ø 23.60
R 0.30
DETAIL B
REF 0.35 CONSTANT RIB HEIGHT
8°
45 RIBS (TYP)
SECTION XX
R 0.45
R 0.25
R 0.25
DETAIL A
Ø 38.60
Ø 37.90
45°
7°
Ø 36.70
Ø 37.88 REF
SECTION YY
(RIB TYP. 8 POSITIONS EQUI-SPACED AT 45°)
1.50 AT BASE
+1° PER SIDE
0.30 CONSTANT
THREAD DETAIL
2 START THREAD
PITCH = 3,00
LEAD = 6,00
10°
R 0,30
1,70 INT.
R 0,30
45°

## create a new part called **“Cap”**

When using the sketcher you may find it helpful to set the grid spacing to 1mm since this part is relatively small.

The Cap is basically a revolved shape. As in the first example you could create a single sketch that would describe most of the shape of the cap. However to reiterate, it is sometimes easier to split the design into several simpler sketches. There is no correct way, use the method that best suits you. For this example we will again use multiple sketches to create the part.

Start by creating the main shape of the cap which will be a revolved solid.

- Select the **Revolve** command.
- Create the sketch on the **XZ** Plane.
- We are going to create the sketch based on the drawings shown at the start of this exercise. The drawings are detail drawings of the final part. It is sometimes a problem to create a part from final detail drawings. The sketches can become unnecessarily complicated if the user tries to completely follow the detail drawings too religiously. Remember that the final detail dimensions and constraints etc. are not always those preferred when designing the part. The example here for instance will often show conditions where the outside profile of the part is dimensioned from some feature from the inside profile of the part and vice versa. Or maybe the user wants to base his design on assembly techniques referencing dimensions and sizes of other parts when constructing the new part. All of this does not matter of course when creating the detail drawings. Normally these drawings are created after the part(s) has been designed. This is not the case in this tutorial, but we have to have something to start from, hence in this case the final sketches will be used as a guide when creating our cap part.

## Cap main body sketch stage one.

Create four lines shown above.

Note the end point at zero is fixed in X and Y.

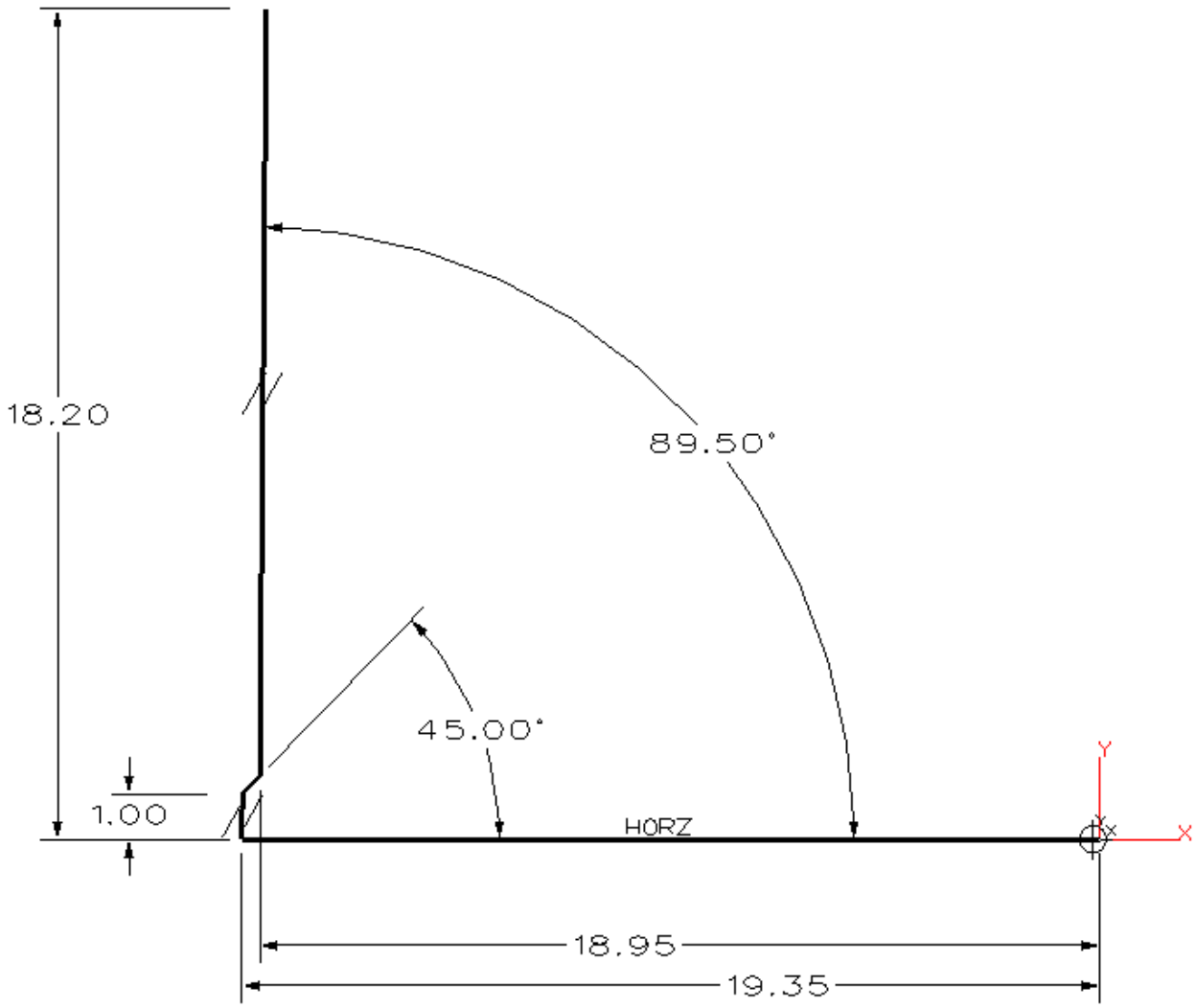
Also note that the small 1mm “vertical” line is parallel to the long 89.5 degree line.

Change the values in your sketch to match those shown above.

Tip, when entering the value 19.35 to the outside, type in 38.7/2 as this is shown as a diameter dimension on the detail drawing. This helps to avoid simple arithmetic errors.

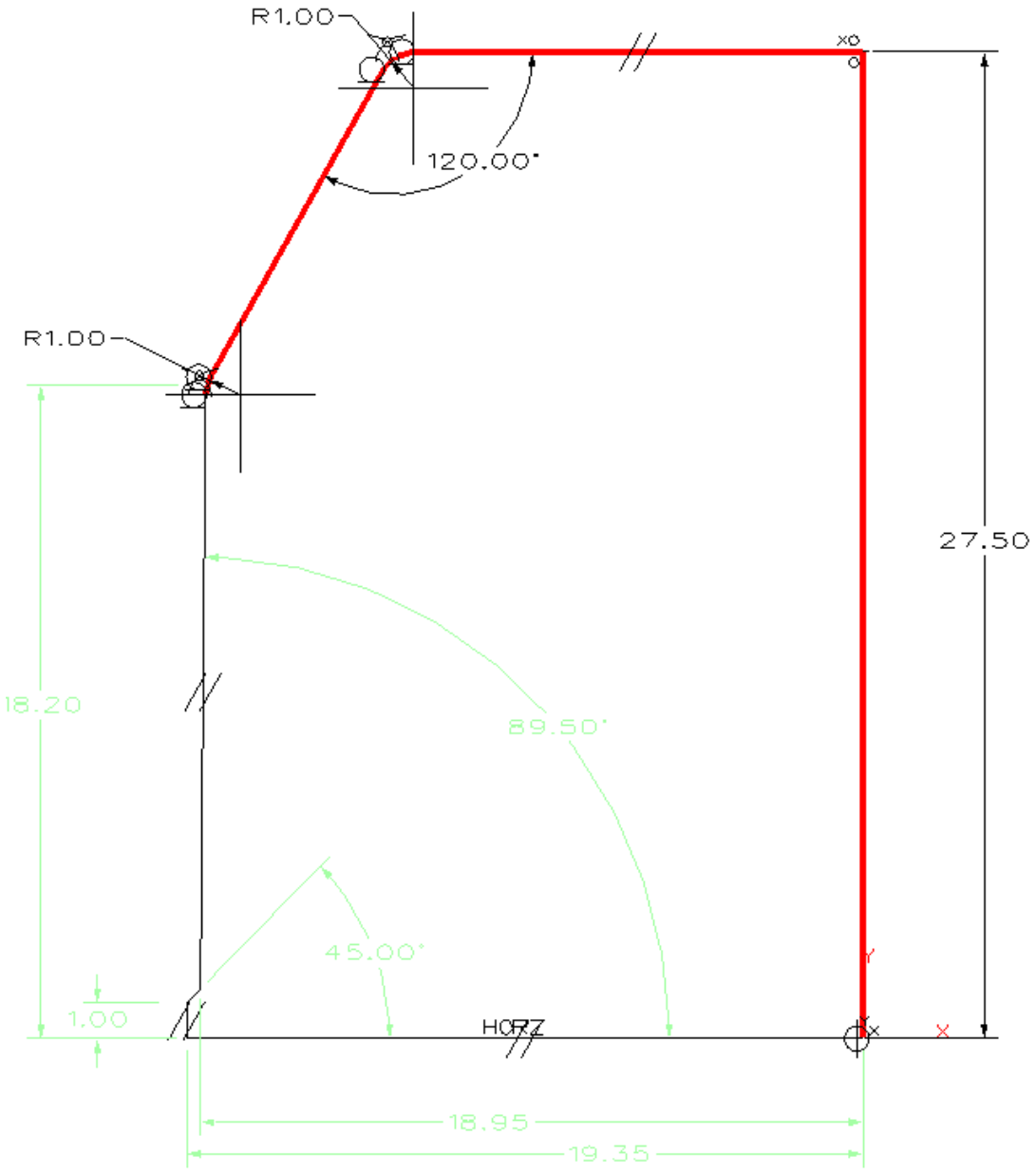
Note: This portion of the sketch should be Fully Constrained.

Continue on the next page.



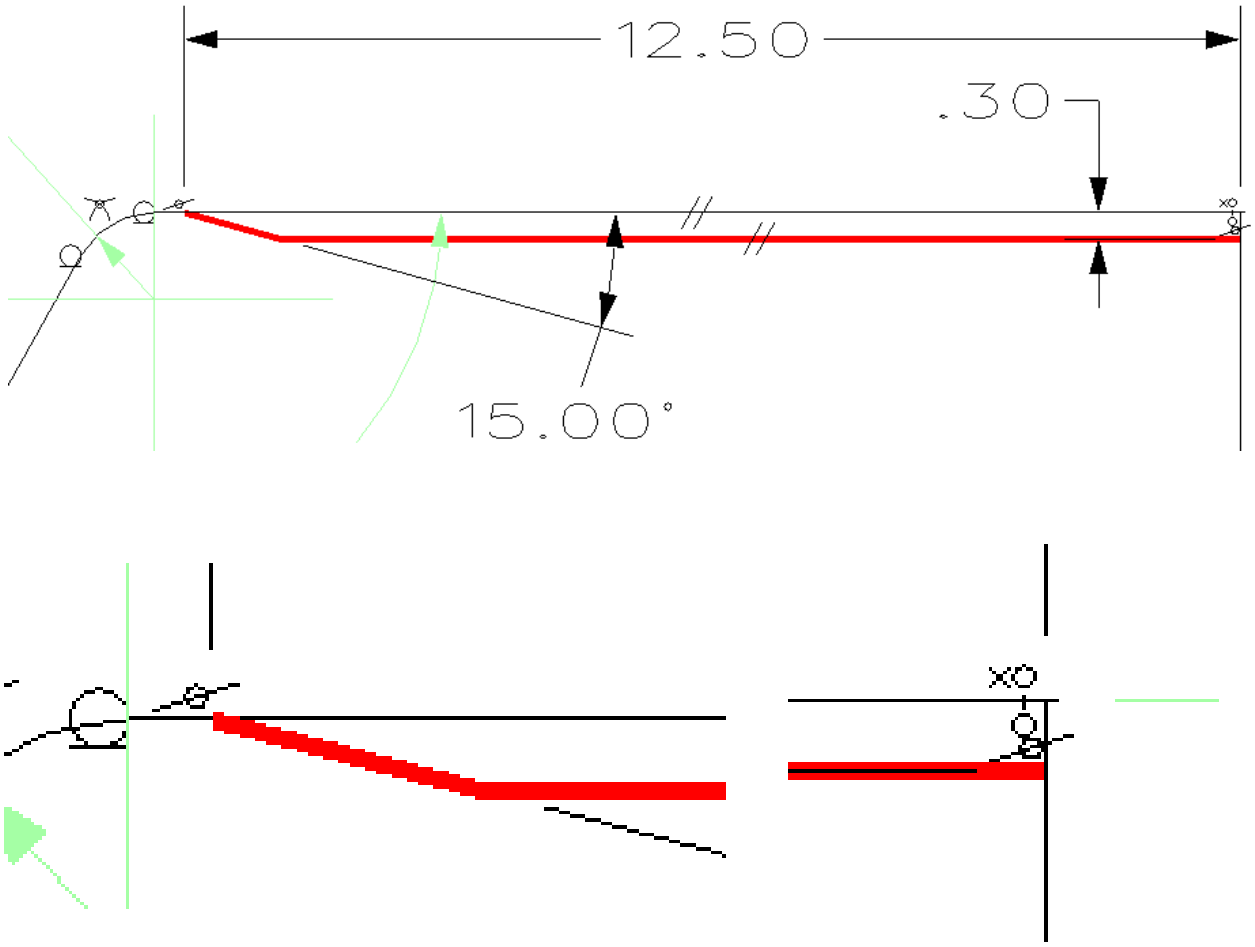
## Cap main body sketch stage two.

- Create the three lines shown above in red. Note the top horizontal line is parallel to the base horizontal line, use right mouse button “focus”.
- Create the two fillets also shown in red.
- Change the values as shown above.



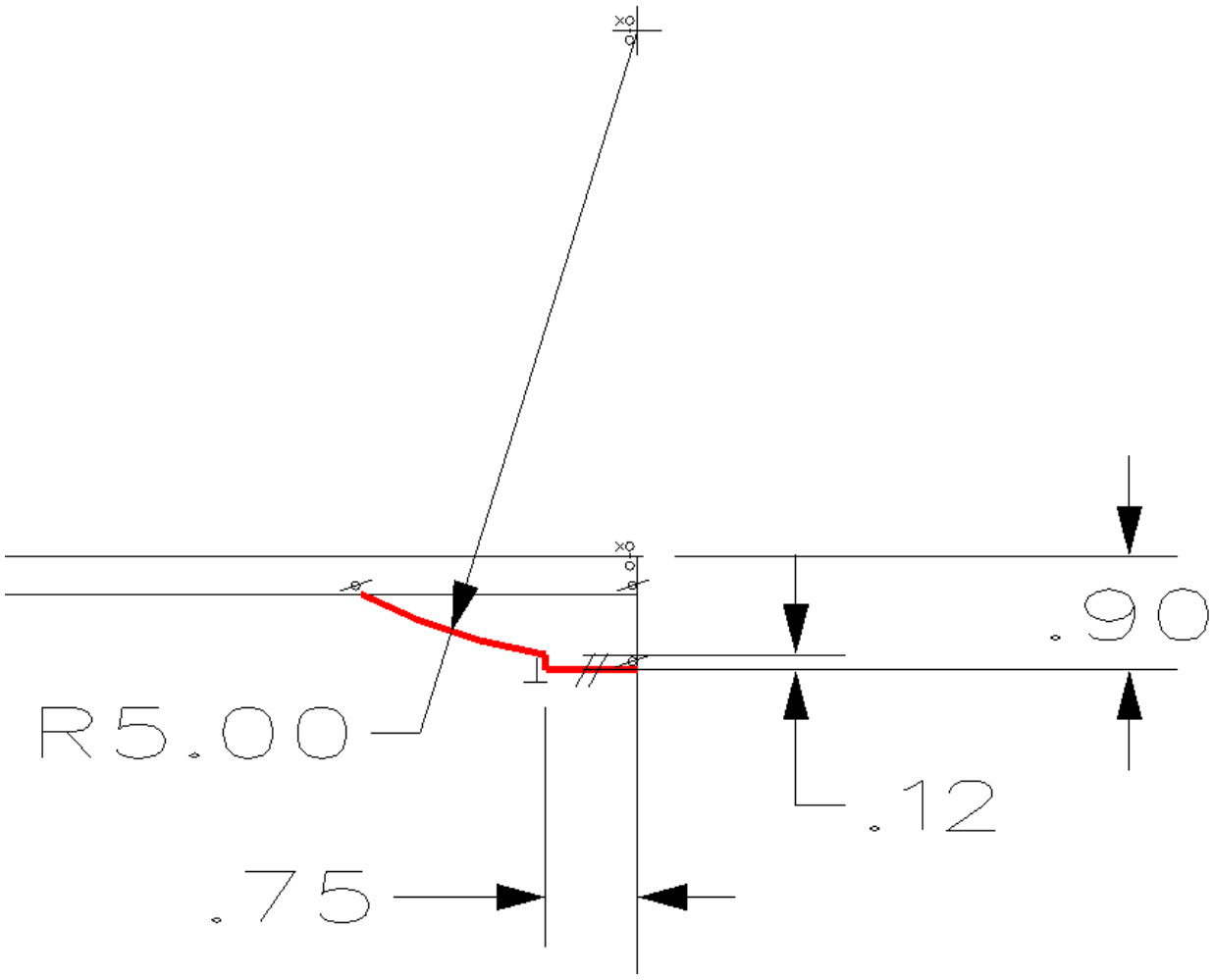
## Cap main body sketch stage three.

- Create the two lines shown in red.
- Change the values to those shown.
- Notice the constrains at both ends.



## Cap main body sketch stage four.

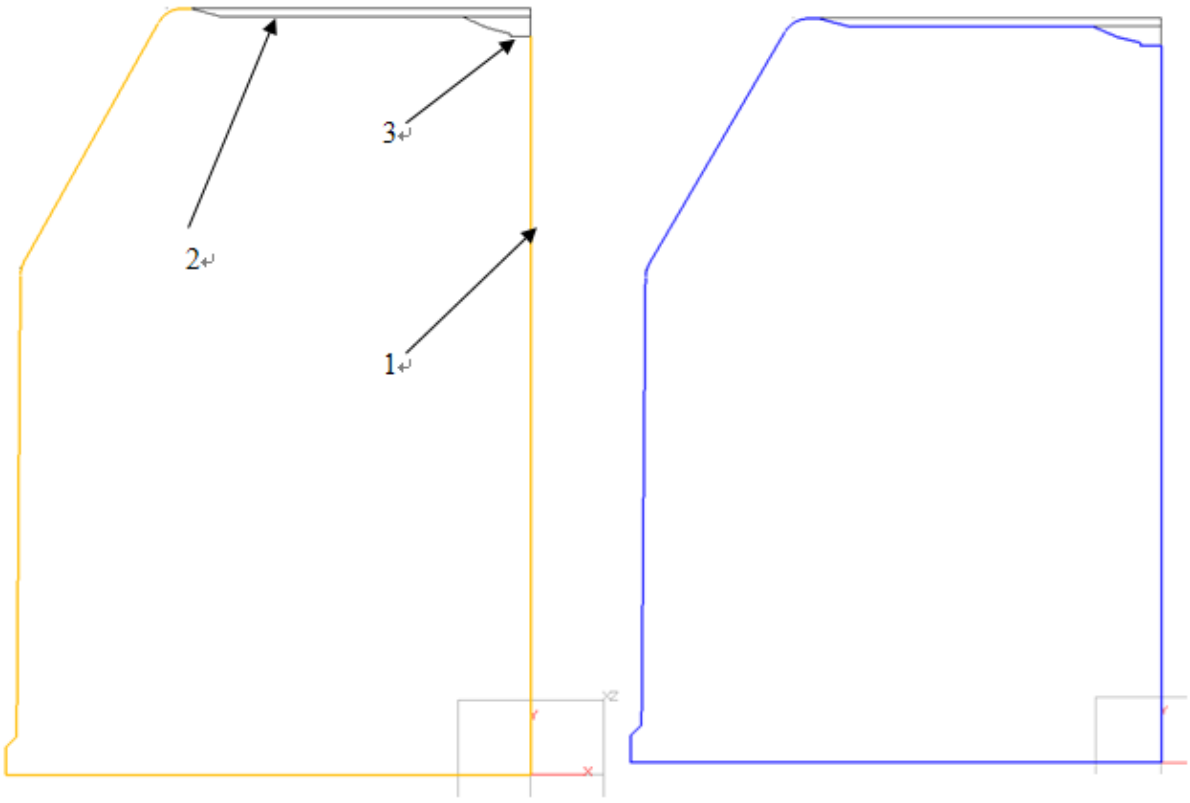
- Create the two lines and the arc shown above.
- When creating the geometry, be careful not to apply unwanted constraints. One of the lines might line up with other geometry and automatically apply an align in Y constraint for example. **How can you avoid this?**
- Note the align constraint in X (shown above in red). This fixes the center point of the 5mm arc to be aligned in X. **Why is this important?**
- Apply the dimensions as necessary and alter the values to those shown above.



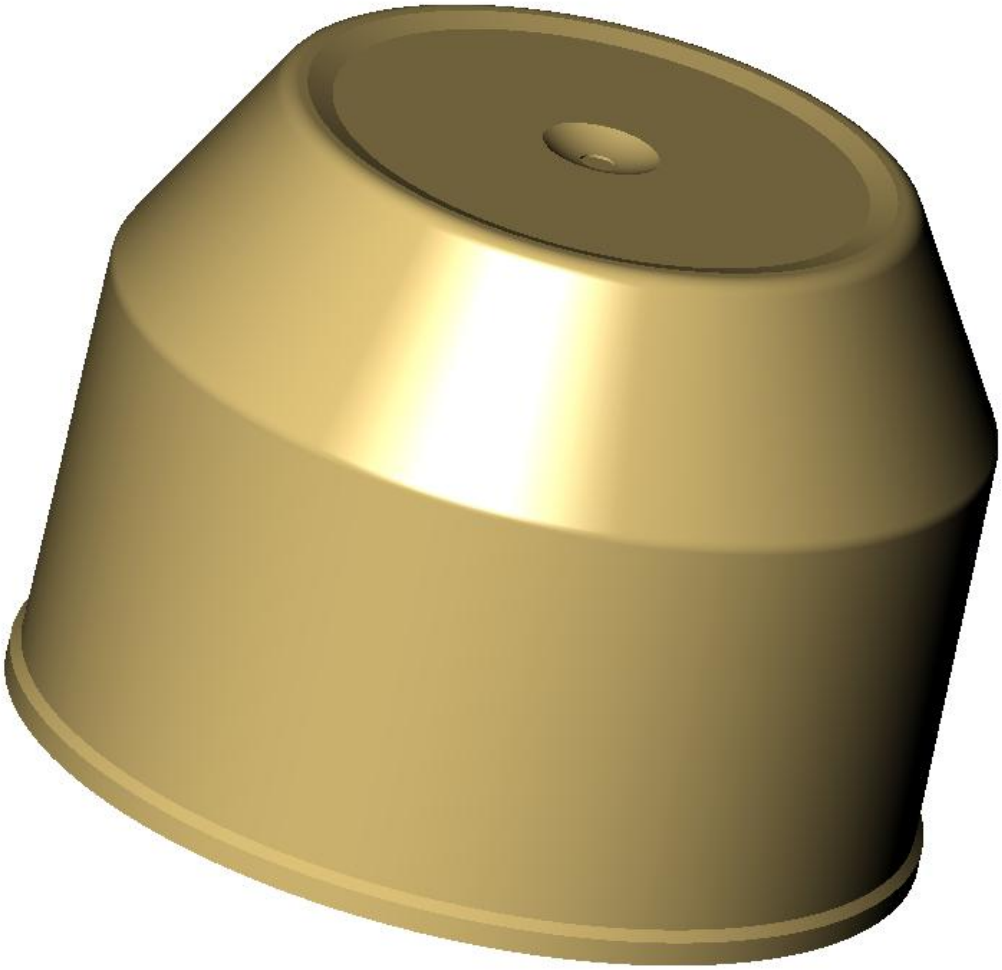
## Cap main body sketch stage five.

At this stage we could continue with the sketch and define the inside features of the cap, but in our example we will define the inside in separate sketches. Notice that the geometry has not been trimmed in certain places. We do not have to trim all sketch geometry in order to make a valid profile. We will make use of the trace profile command. For our example I have started picking the profile by selecting the vertical center line and picking the rest of the entities in a counter-clockwise direction.

Select the **trace profile** command.



- The dimension and constraints in the figures shown have been removed for clarity.
- After the third and final pick the profile will turn blue indicating a closed profile.
- Exit the sketcher.
- Complete the revolved base command. Revolve about the **Z** Axis, start **0**, end **360**, as shown below.

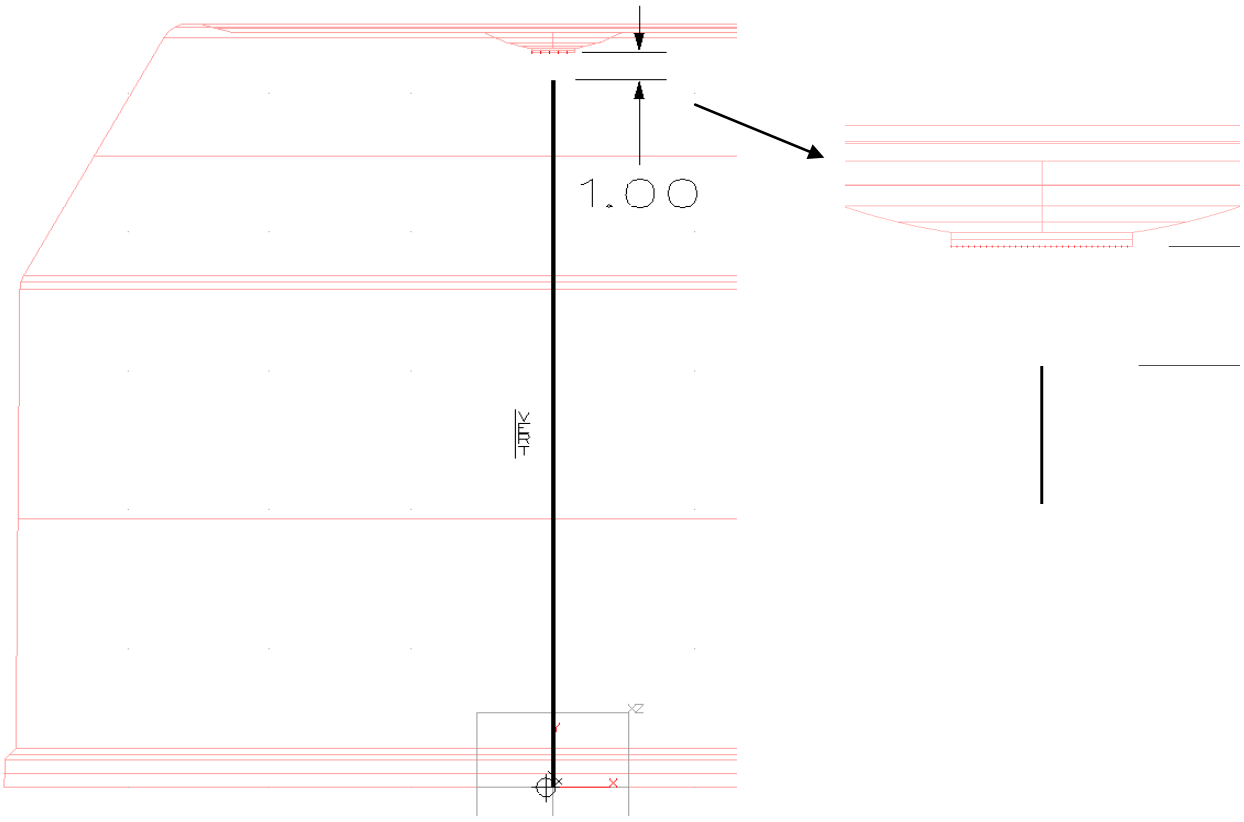


The next step is to create a revolved cut and remove most of the inside features of the cap.

Select the **Revolve- Remove** command and start creating the sketch on the **XZ** Plane.

Note in the following sketch you will often need to create reference edges from the existing solid in order to correctly dimension some of the sketch entities. When creating the following sketch if you are at any time unsure as to which reference edges need to be picked, please see your tutor.

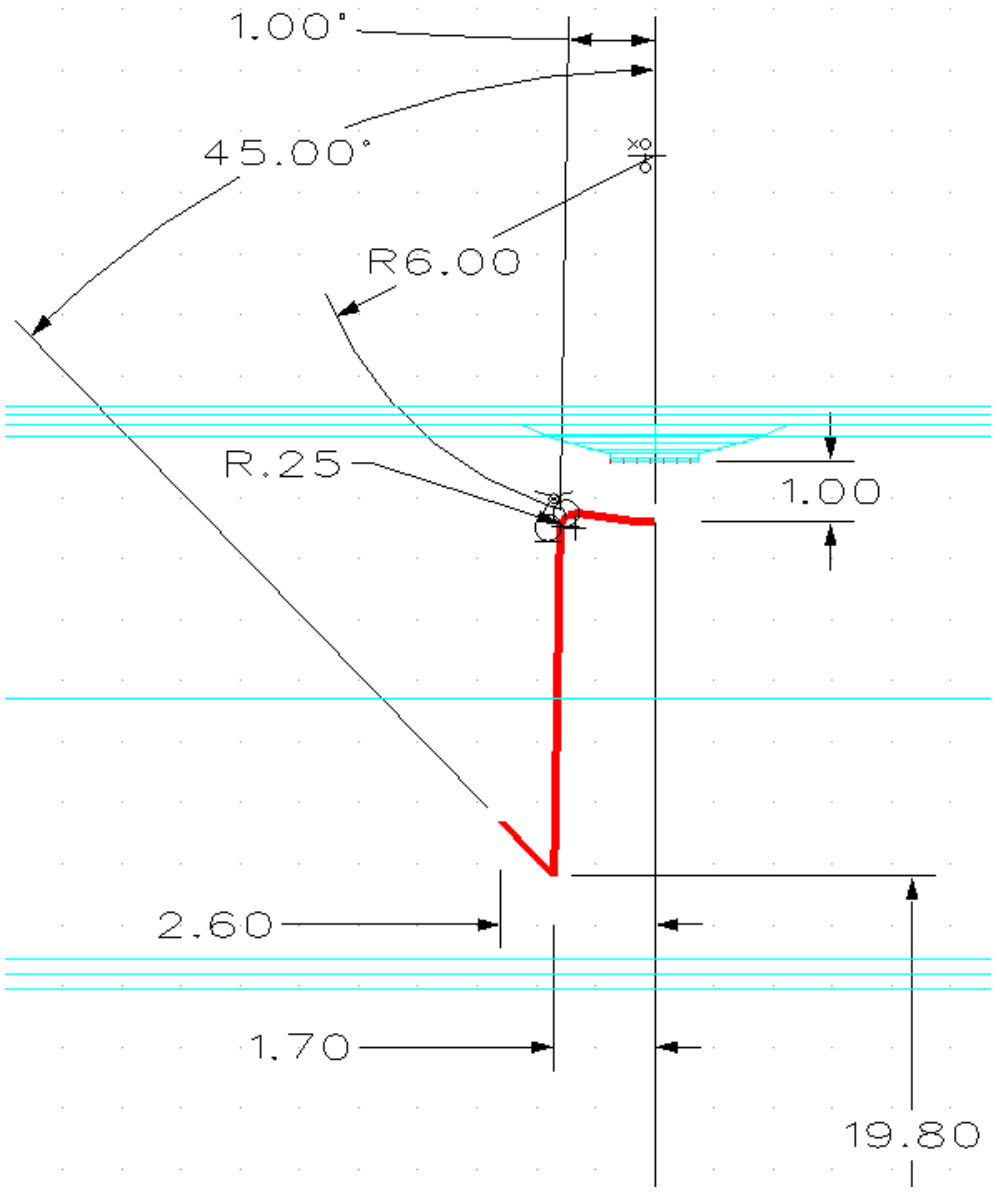
## Inside revolve sketch stage one



- Start by creating a vertical line starting at zero.
- Create a reference curve from one of the edges shown (red dotted). 
- Dimension the part and if necessary change the value as sown.

## Inside revolve sketch stage two

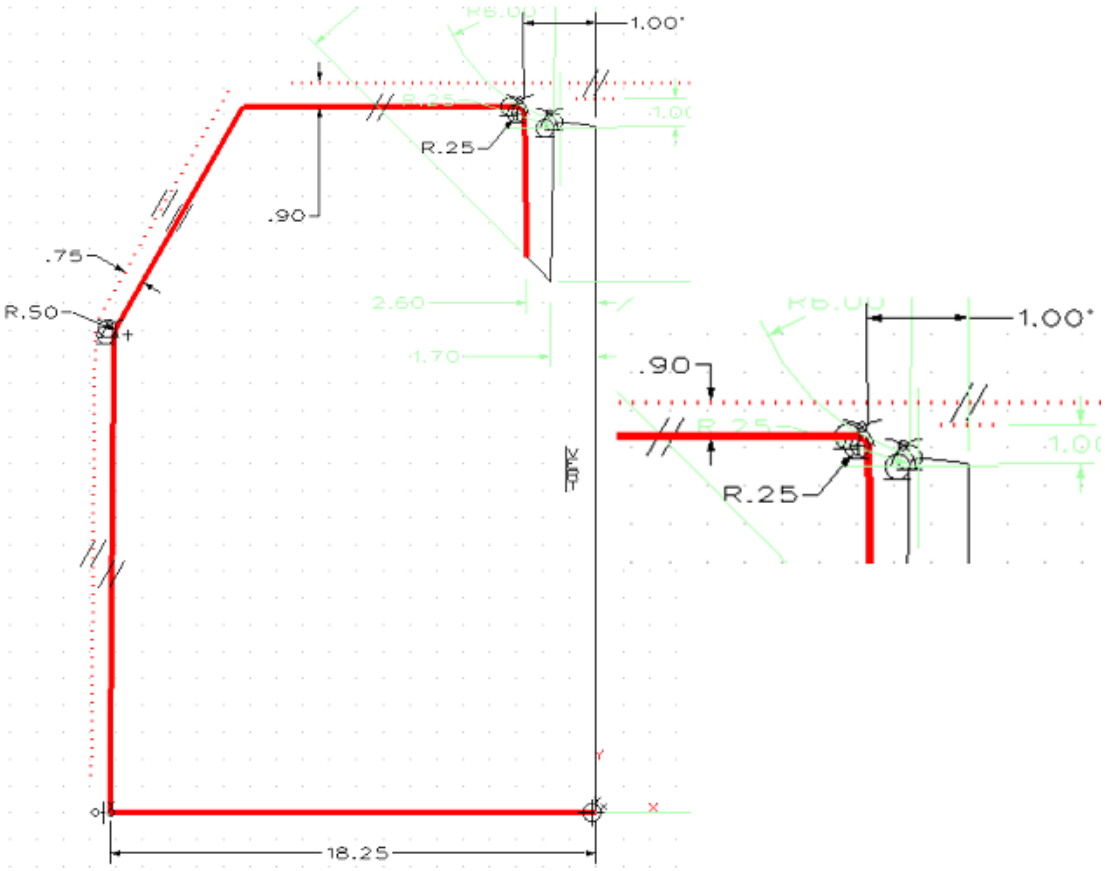
- Create the 6mm arc. Create the two lines. Create the .25 fillet.
- Attach an aligned in X constraint to align the center of the 6mm arc to the end point of the first vertical line. This will ensure a tangent condition at the 6mm arc when the cut is finally revolved.
- Apply and edit the dimensions to the values show above. Note the 19.8 dimension is to the zero point.




## Inside revolve sketch stage three

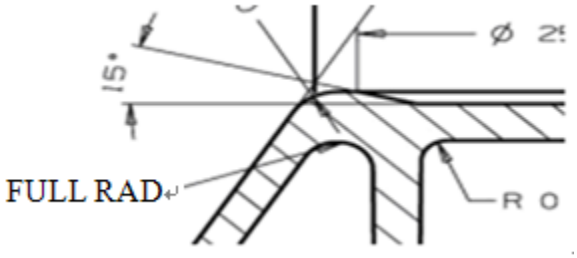
- Create three reference curves from outside edges of the original revolved solid; ask your tutor if the edges are not shown clear enough in the figures above.
- Create five lines then two fillets as shown.
- Use the right mouse button (focus) when creating the three lines that have to be parallel to the outside edges.

- Also when creating the geometry make sure unwanted constraints are not automatically applied, if necessary use the right mouse button and toggle smart pick off, or zoom in closely when creating the entities. Always be aware of what constraints are being applied when creating geometry or the results may not be what you expected.
- Add the necessary dimensions as shown in black and change the values as required.

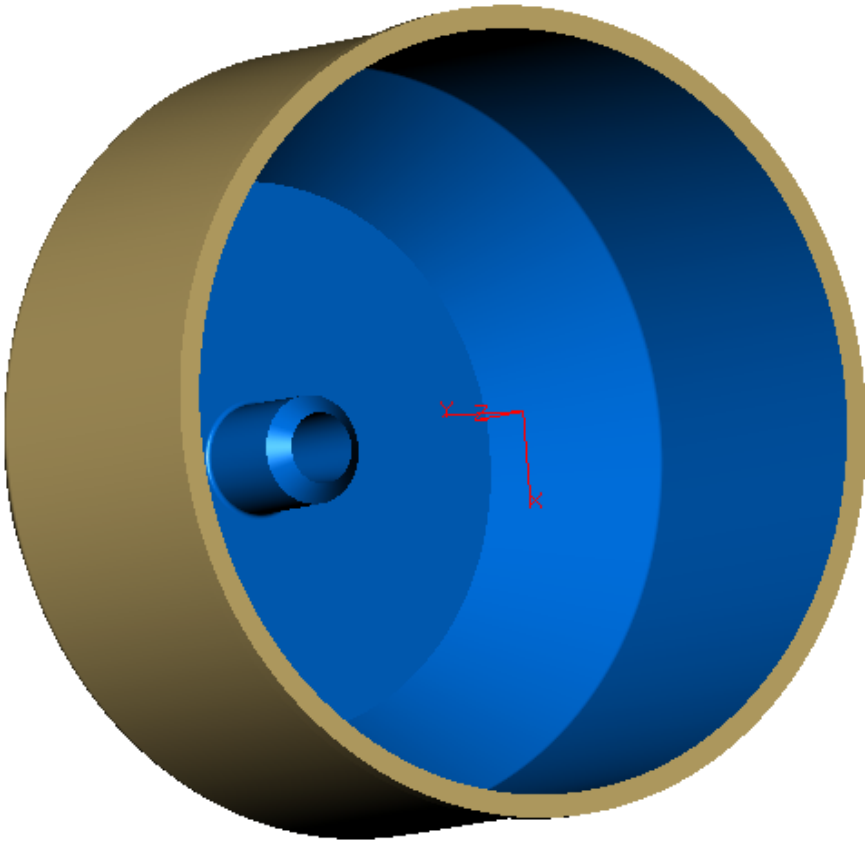



At this stage we could continue defining sketch geometry including for example the rib shown in detail B on the original drawing. Notice where this rib attaches to the cap there is a radius (shown as “Full Rad” on the detail drawing). This rib could be created in the current sketch, but let’s exit the sketch and create this revolved rib with a separate

sketch. This then would require a surface to be created which would have to be tangent to three faces. How we accomplish this will be described shortly.




- Exit the sketcher
- Complete the revolved cut command as shown opposite.

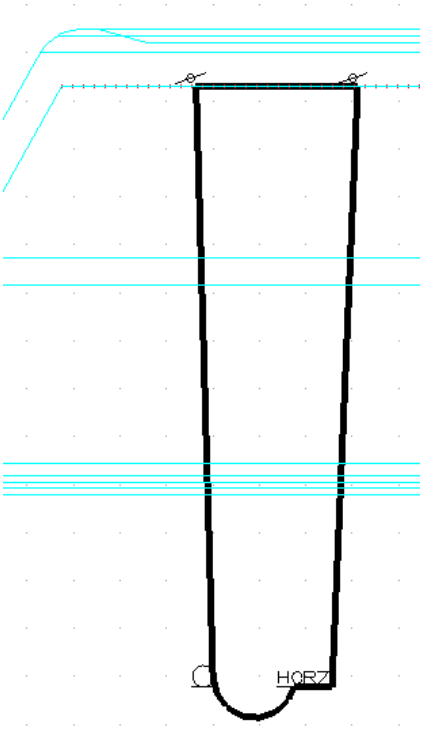


## Create the revolved rib shown in detail B.

- Select the **Revolve - Add** command.
- Create a new sketch on the **XZ** Plane.

## Detail B rib sketch stage one

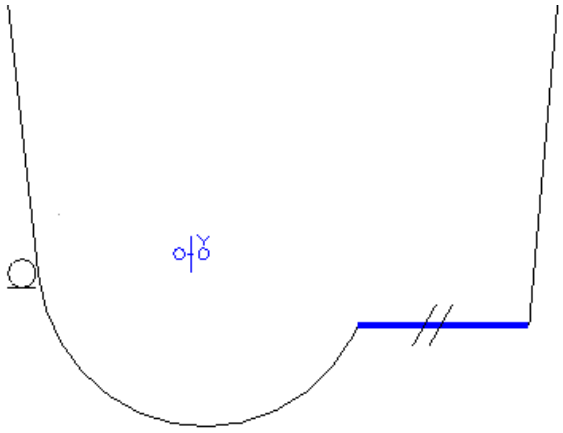
- Create a reference curve from the correct inside edge of the existing revolved solid.
- Create the four lines and one arc as shown opposite.
- Note 1. Use the right mouse button and select "**Default**" in order to fix the necessary lines to the reference curve.
- Note 2. Make sure at this stage there is no align in Y constraint attached to the short horizontal line and the arc.
- Note 3. The sketch is clearly not the correct size at this time.



- Attach an **aligned in Y** constraint as shown. 
- The base point is the left hand end point of the blue horizontal line.
- The point to align is the Center point of the arc (not the left end point)
- The sketch will solve automatically.

- Because the “vertical” lines are not truly vertical the left hand end point of the arc will have a slightly different Y value than the center.

**Detail B rib sketch stage two.**

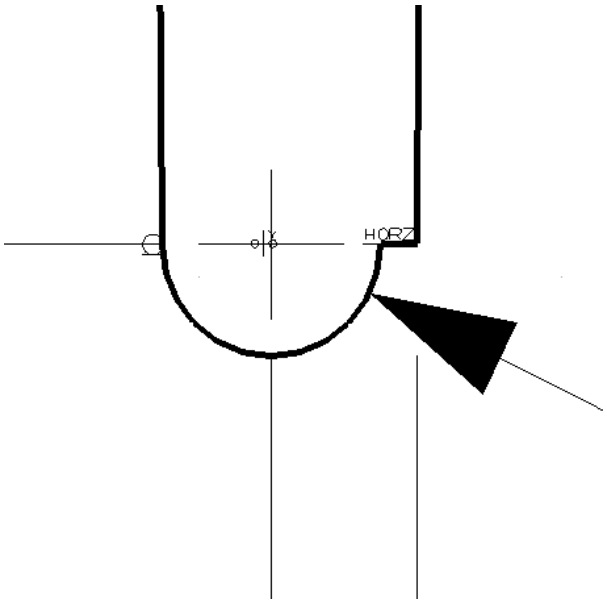


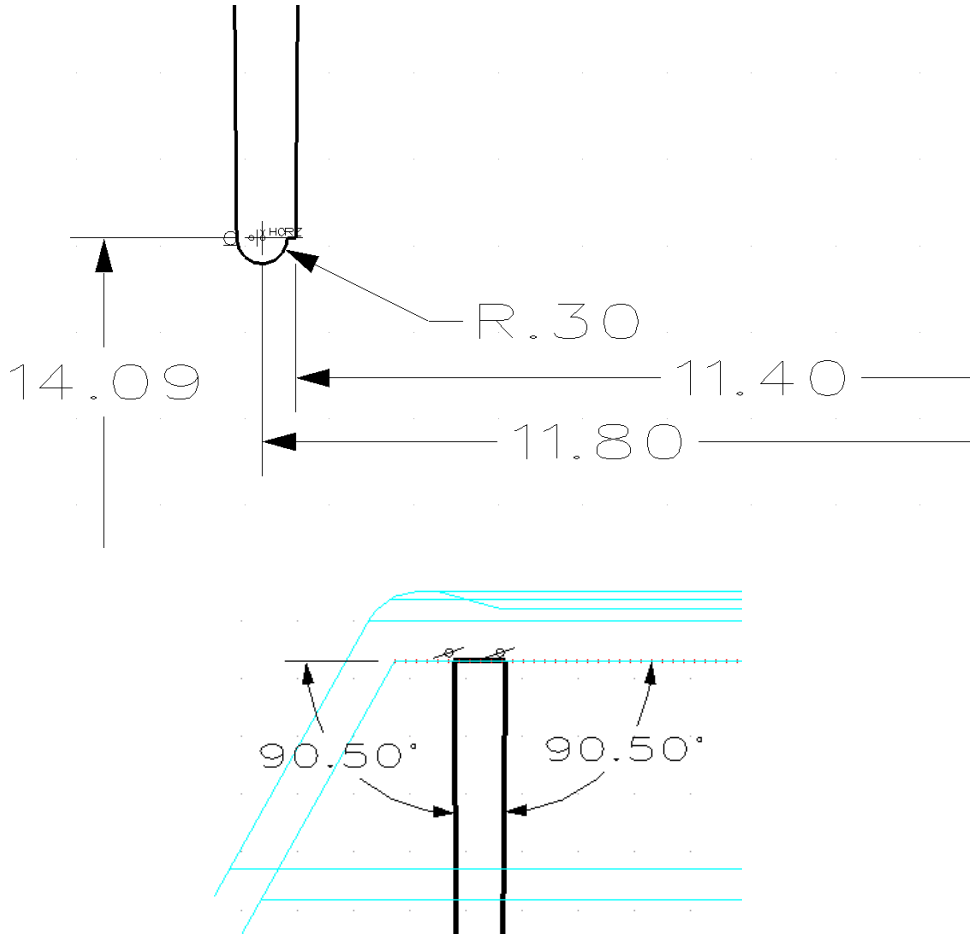
**Detail B rib sketch stage three.**

- Attach the dimensions then modify the values as shown in the figures opposite and below.
- Tip: If when editing any of the dimension values, the arc flips to the wrong side. Undo the edit; change the step size of the increment/decrement menu to a small value (0.1mm). Then use the increment/decrement to gradually bring the

dimension to approximately the correct size. Then type in the correct value as normal.

- Tip: When editing the 11.4 and 11.8 dimensions for example, if both numbers need to grow, change the big one first(11.80). If they need to shrink, change the smaller one first (11.40). If you change the wrong one first, the rib will flip. UNDO the edit.
- Or you can use the Delay Solve option

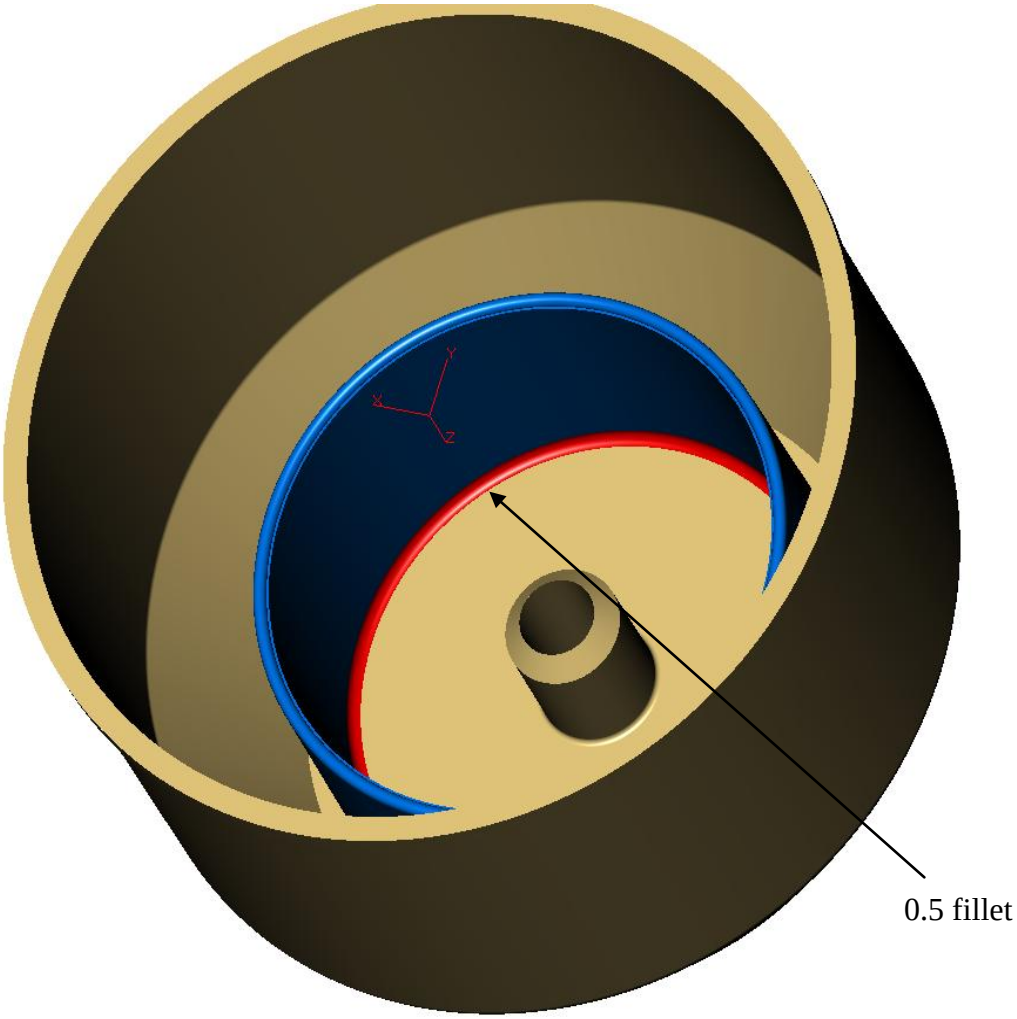





- You should have a closed profile. The Trace Profile is not required.
- You should check to make sure it is closed.
- Use **INQUIRE > Show curve connectivity** . Fix the profile if there are any gaps or overlaps

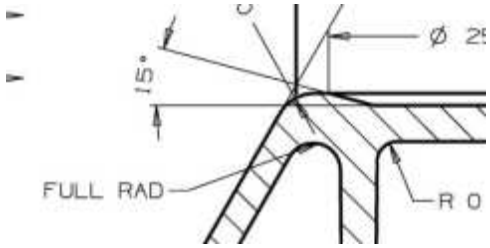
Do this for all sketched except the simplest circle or rectangle sketches.

- Exit the sketcher and complete the revolve command as shown below in blue.




Create a 0.5mm **fillet** as shown above in red.

## Creating the “Full Rad” feature on the detail B rib shown below.




The face required needs to be tangent to three separate faces. The radius of the face is not known and is calculated from the tangent condition of three faces.

As stated previously this feature could have been defined in the revolved cut sketch, but we want to introduce the “**Blend Face**” command (Freeform Tab). 

Use this command to create a quick face by selecting curves, edges, or faces. Spine curve options are available and you can constrain the face to pass through a curve or a face. A **circular** or **conic** face cross-section can be selected using the Arc Type command option. Experimenting with the various input types and options will produce a variety of faces including as in our example a face that is tangent to three other faces.

A visit to the Help Manual on this topic is definitely encouraged.

The faces that need to be selected are in this case, more easily picked if you toggle the display to shade on.

Toggle shade on. 

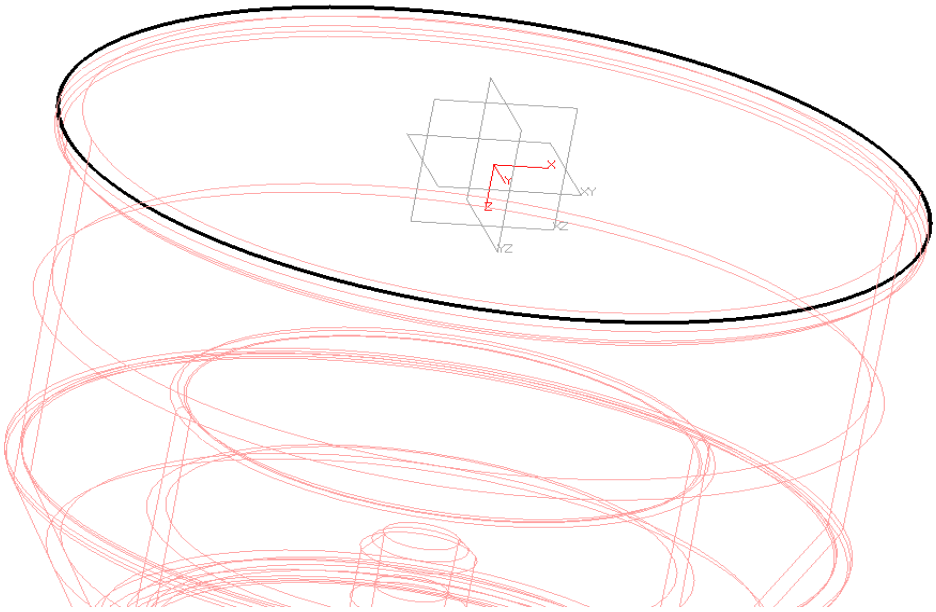
Select the **Blend Face** command.

**Natural** - No drive curve is used. A face similar to a rolling-ball fillet is generated. No additional steps are required.

**Spine** - Select a spine curve as the drive curve.

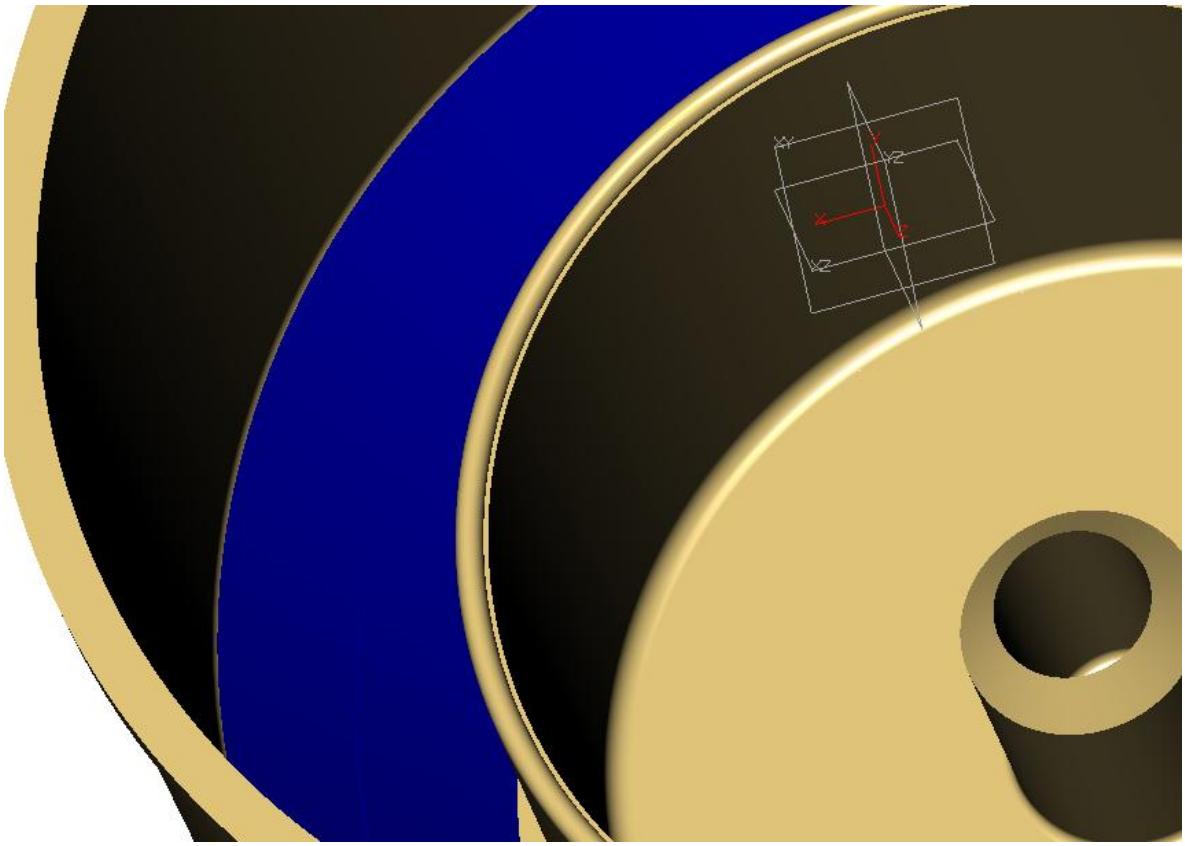
**Parallel** - Select a direction defining the normal of a parallel plane. The face is created by moving an infinite plane containing cross-sections of the quick face parallel to the plane defined by the selected normal.

- Pick **Spine** from the optional inputs and pick the outside edge as shown below in black.
- The face is created by moving an infinite plane normal to, and from the start to the end of the spine curve.
- Use this option or the Parallel option whenever possible, this uses a “sliding disk” when calculating the face instead of a “rolling ball”. Although in this case the results would be the same as the Natural option, the calculations are much simpler.

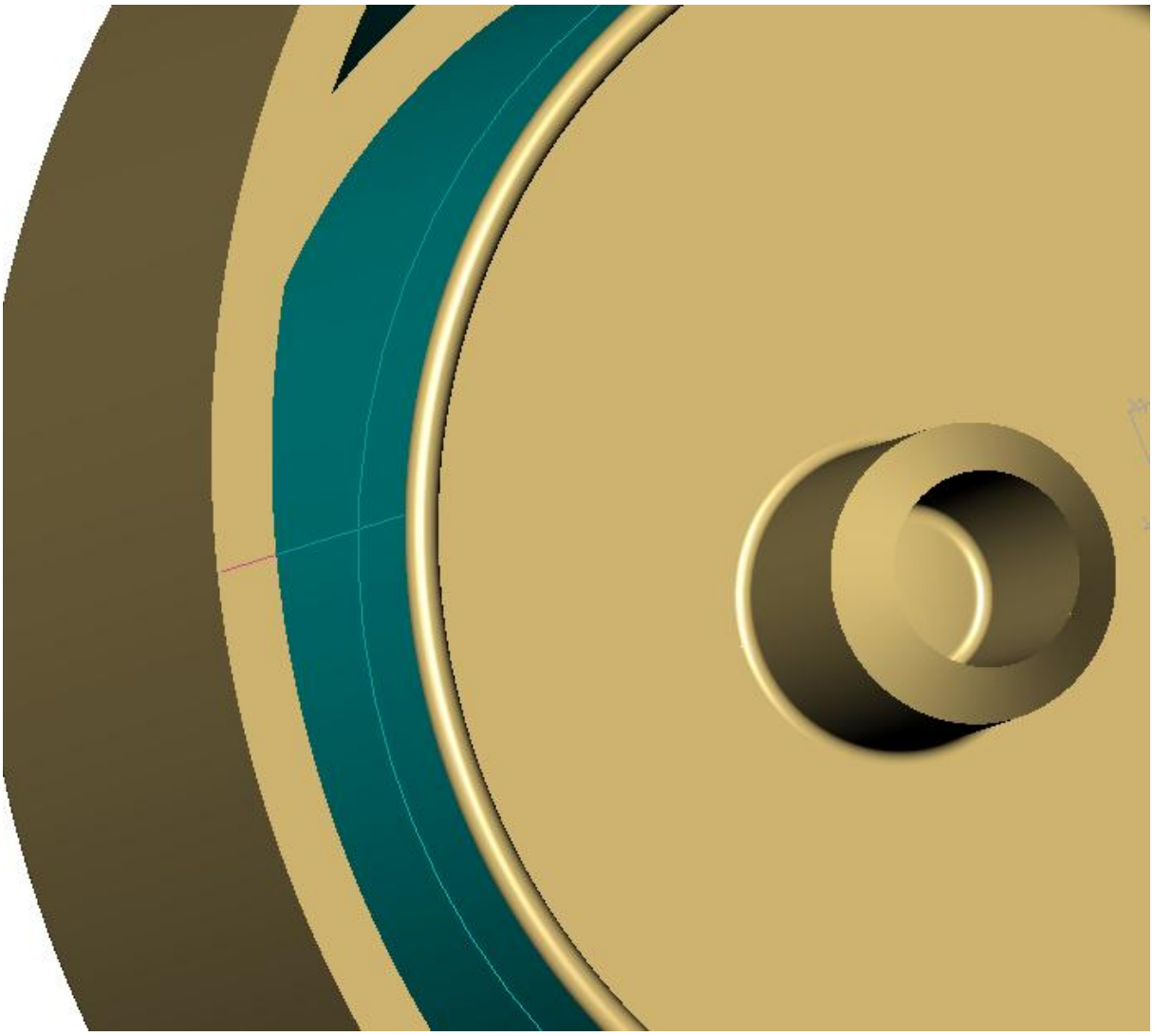


After picking the Spine curve, set the pick filter to “**Face**”.

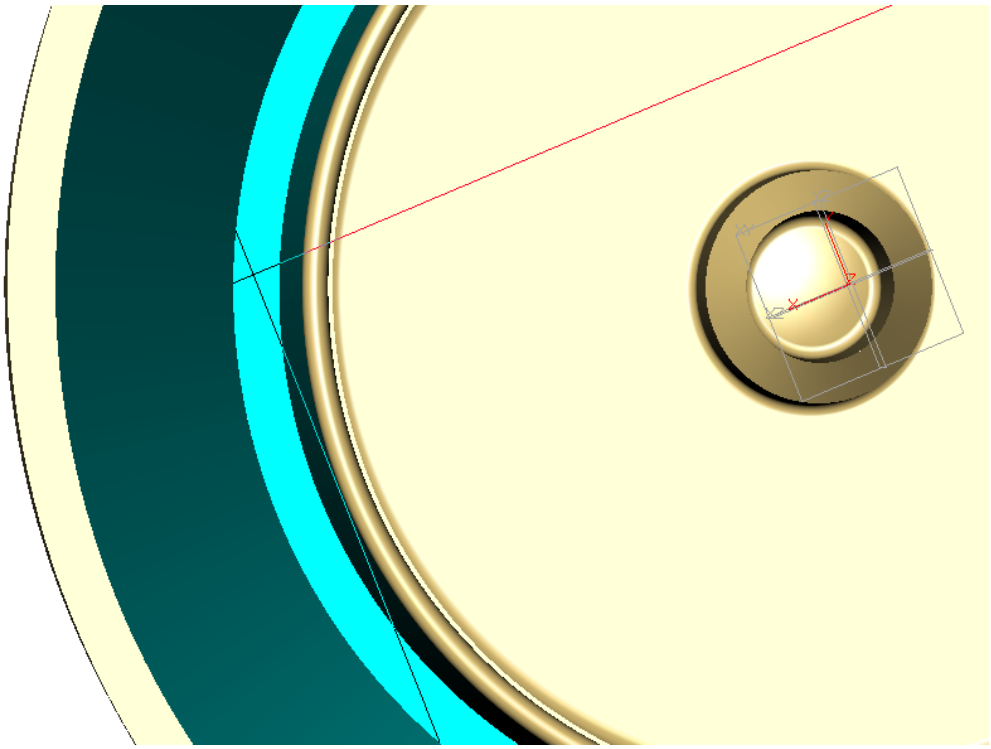
Pick the first face, which is the inside sloping face shown below in blue.



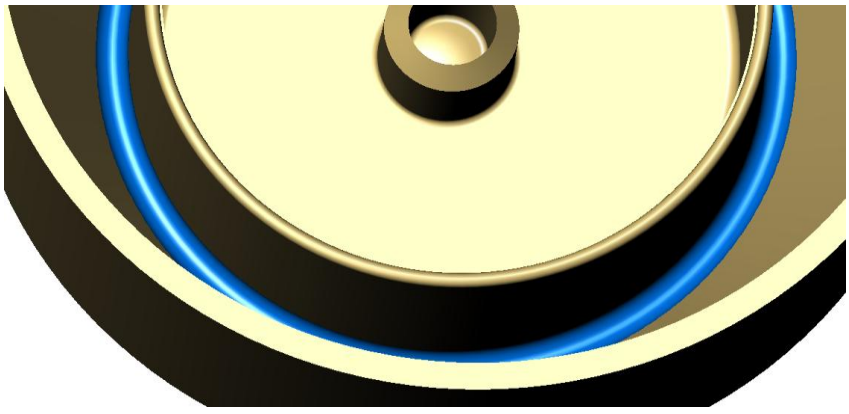
Pick the second face as shown below in cyan.



Pick the third and final face, which is the small flat face on the inside of the part as shown below in cyan.

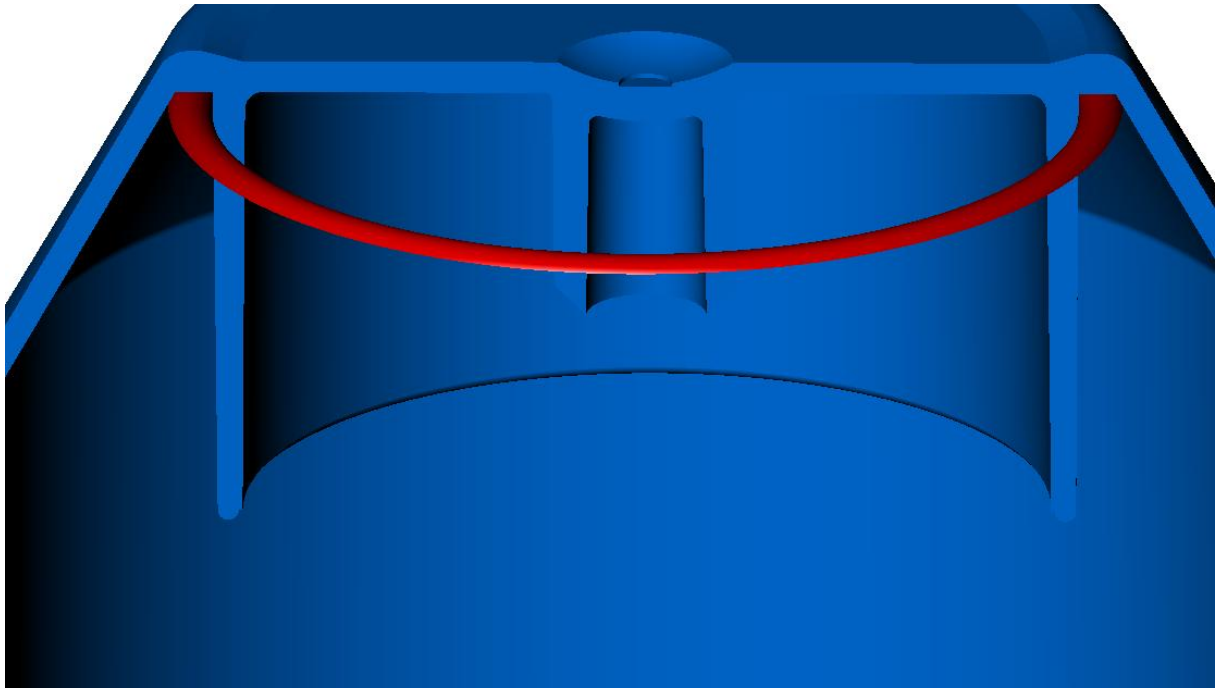


Complete the command and the face created is shown below in blue.



This is a powerful way of creating a face tangent to three other faces. The radius of the "fillet" face is in this case constant, but consider the case where the inside rib is not circular as in our example, but some other shape. An Ellipse shape perhaps, or even a closed tangent curve of some shape. In these examples the fillet face has a constantly varying value, the quick face is an ideal solution for these cases.

However, in our model we have the original revolved solid and a face as shown below.



- The figures show the face and the revolved solid.
- The solid has been cut in half for clarity.
- To prove this to yourself you can set the **pick filter to shape**, then **draw a window around the part**. The message window should reflect "**2 picked**"
- There is an option on the **Blend Face** command to join this face on creation.
- Right click **Redefine Last**
- Pick "**Sew**" from the sewing option pulldown.
- Pick **OK**
- The hybrid modeling command has now acted like a solid fillet.

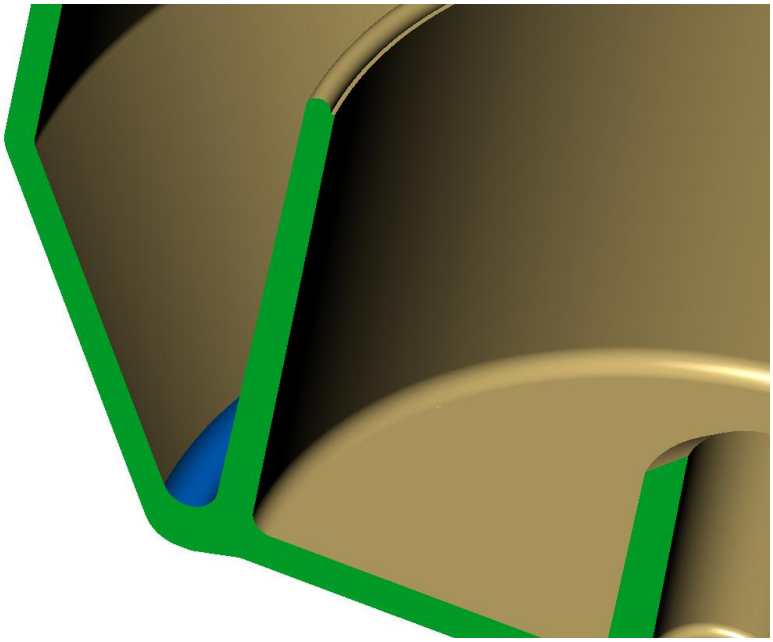
- There are a couple of other methods to join this face. **Redefine last** again and set **sew** to **no action**.

How can we merge the face and the revolved solid together without having to resort to using several trimming operations etc.?
ZW3D is a hybrid modeler. We can use the “Combine shape add” command and simply join the two shapes together.

Question: What is the “Normal” of a face.
Question: We need to check if the face normal is pointing in the correct direction, why?
Question: What would happen to our model if the face normal points inwards instead of outwards?
Question: If the face normal is pointing the wrong way how do we reverse it?

- Select the **Combine Shape > Add** command and pick the revolved solid as the base shape and the face as the shape to be added.

The final joined solid is shown below in cross section for clarity.

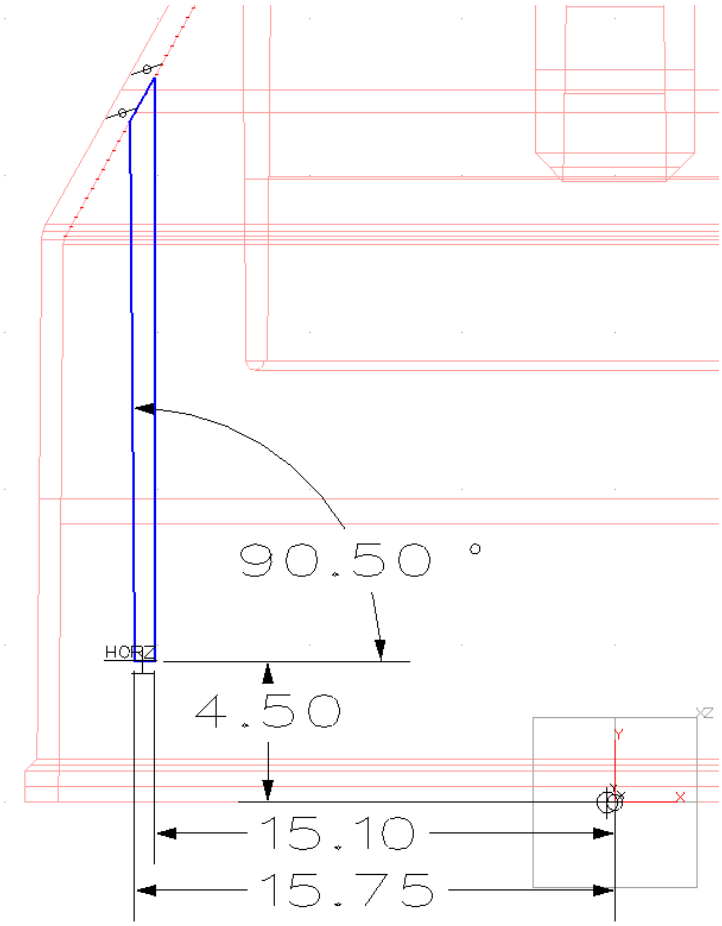


## Create the circular rib for the thread.

- Select the **Revolve - Add** command.
- Create the sketch shown below on the **XZ** Plane.
- Note the use of a reference edge and also the trace profile command.

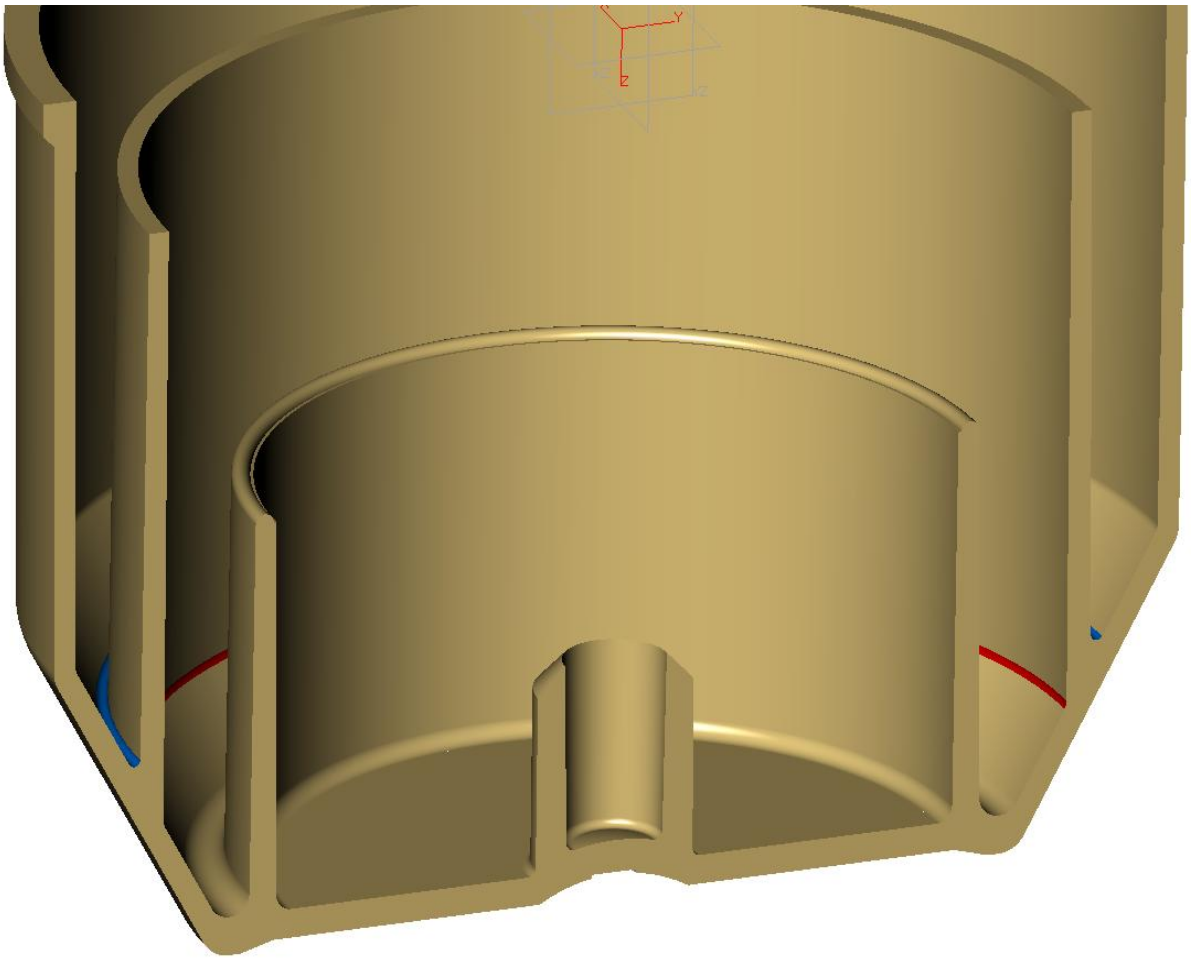
You would not need the Trace Profile if you close the gap at the top yourself.

- Note. The circular rib is drafted **0.5** degrees on one side only. If you wish you would be able to add draft on the other face (for a distance of 0.5mm) after the thread has been created.




- Exit the sketcher and complete the revolve command.

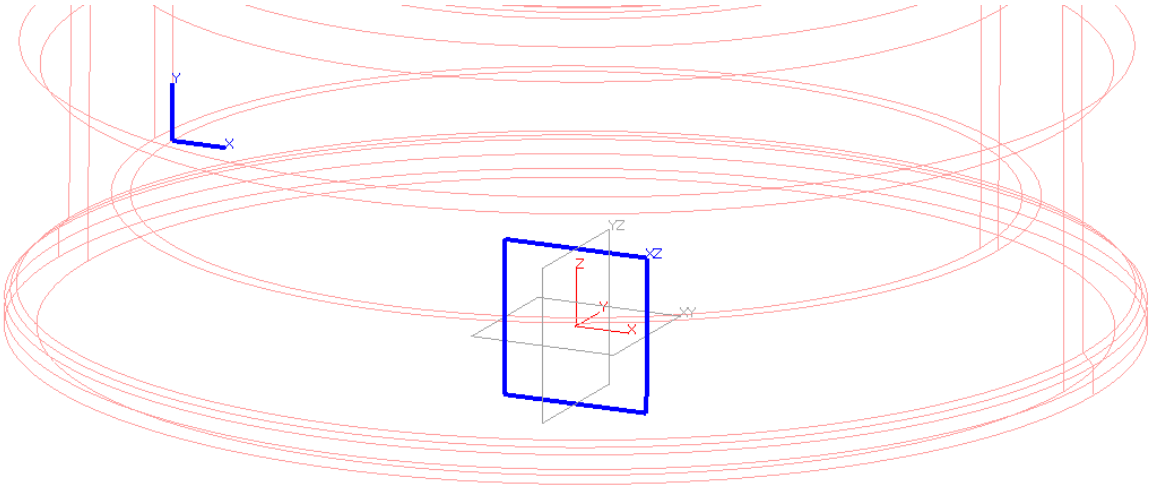
- Create two **fillets** where the rib meets the main body. The fillet shown in blue has a radius of **0.2**mm and the fillet shown below in red has a radius of **0.5**mm.
- The part is shown below in cross section for clarity only.



## Creating the two-start thread.

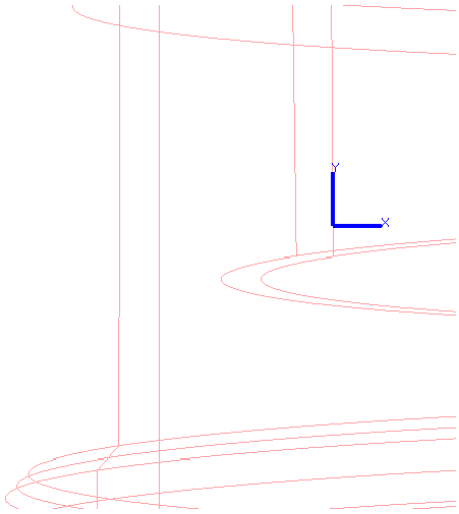
In order to locate the thread sketch profile, we will first have to place a datum plane in the correct position.

- Select the command "**Insert Datum** ". 
- Locate the new plane parallel to the **XZ** Plane at the point shown below.



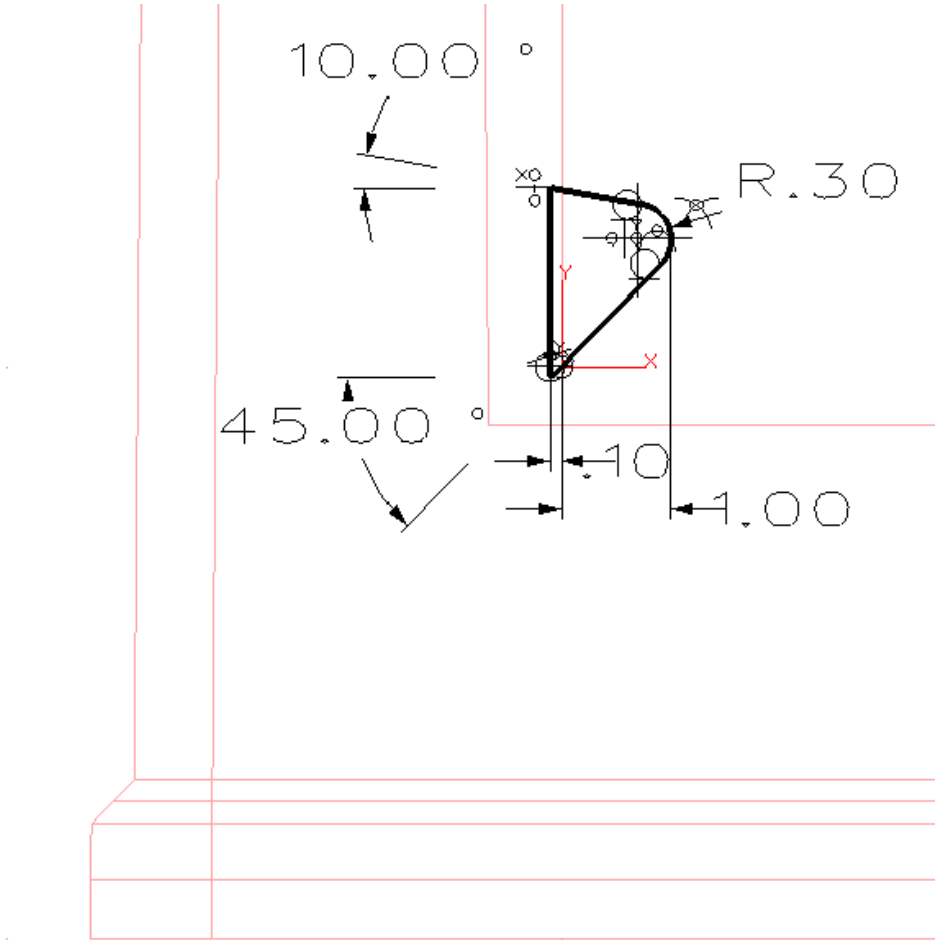

- Select the **move entities along a direction** command within the **Move** command under the **Shape Tab**.
- Pick the previously created plane and move it up **0.5** mm along the **Z axis** as shown opposite.

You could have combined this move with the Insert Datum command by using the Right-click Offset location option when picking the Origin.



In our previous example we created the thread form sketch in the part. This time we will insert a previously defined global sketch.

- Select **Edit** pull-down menu**/Copy/External Sketch**.
- Pick the global sketch “Laser cap thread form” supplied with this tutorial.
- Locate the sketch on the previously created datum plane as shown below.
- Note the 0.1mm dimension; this saves us having to offset the thread form face to avoid the unnecessary coincident face situation. This sketch has the 0.1mm value built in.




Select the **Spiral sweep** > **Base** command.

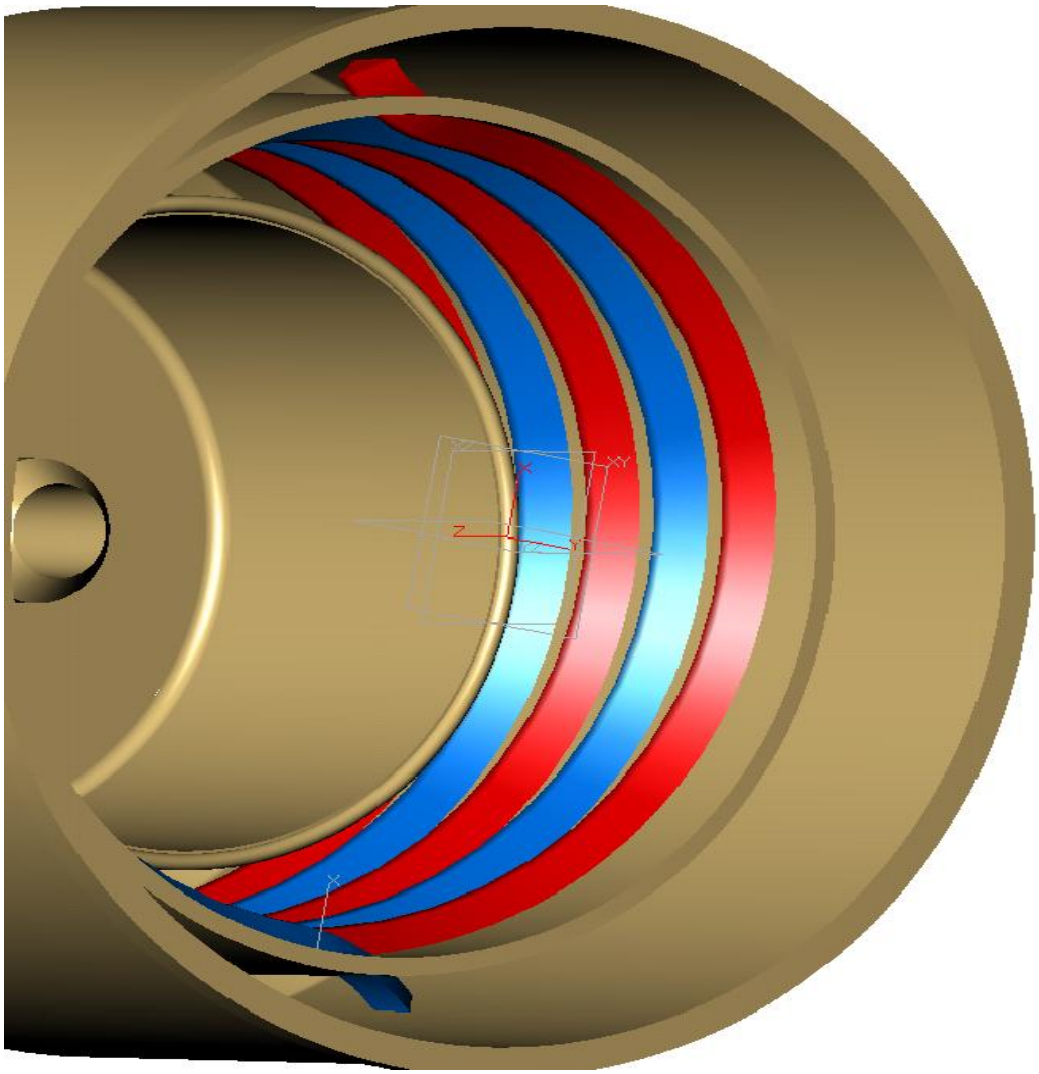
- Specify the lead in/out to be "**Both**".
- Direction is positive **Z** axis.
- Number of turns is **2.5**
- Distance per turn is the thread lead of **6**mm.
- Taper angle =**0**
- Select the lead in/out option to be "**Out**".
- Lead out radius = **5**
- Lead out angle =**45**

The first thread is shown opposite in blue.

Notice the thread run out goes right through the circular rib. We will deal with this shortly.

The thread is two-start (two full threads).

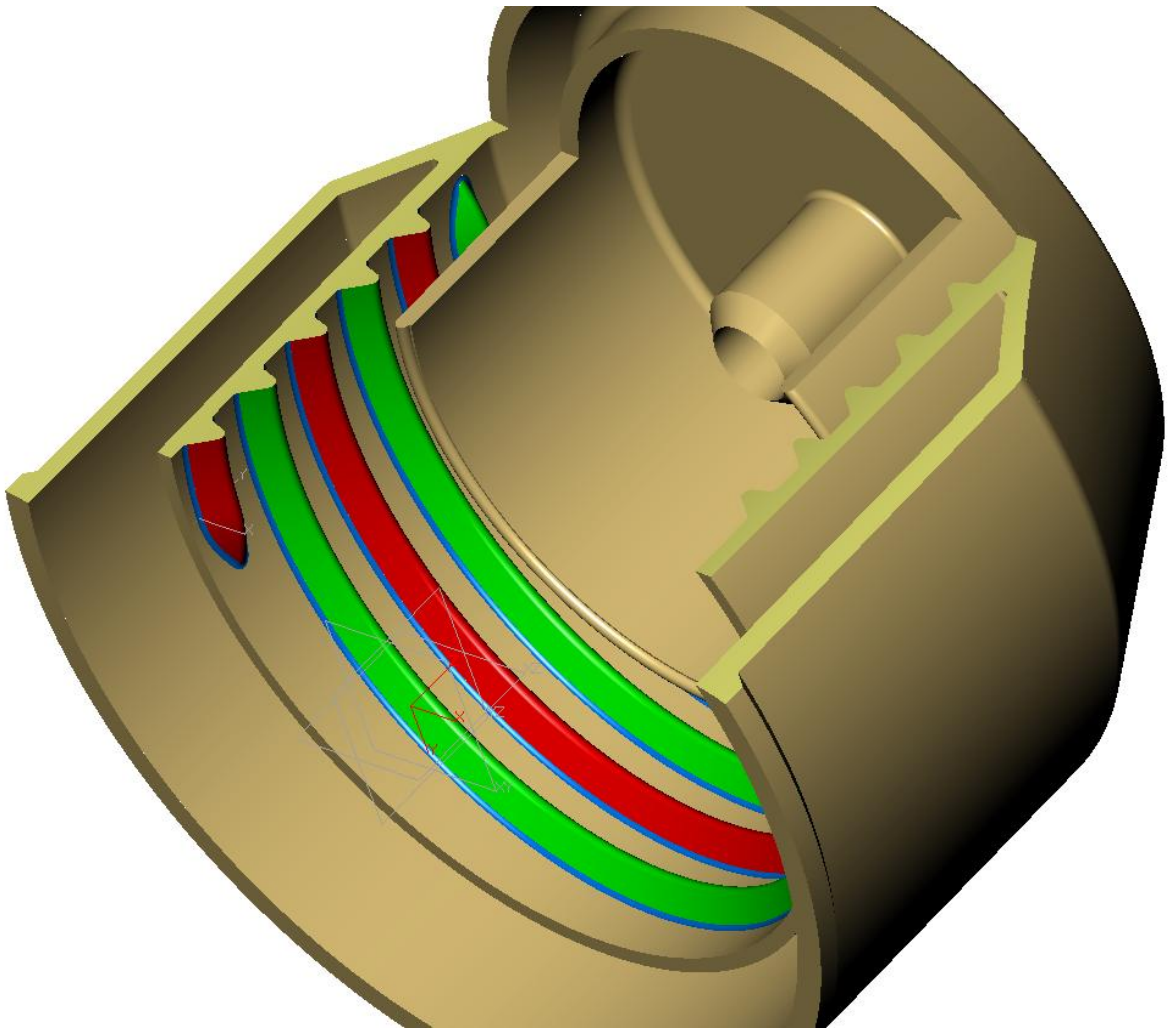
- **Copy** the existing thread form around the **Z** Axis.
- Make **one** copy and the rotation value is **180** degrees.
- As shown below in Red and Blue.



Join the two thread forms to the base solid using Combine shape command.

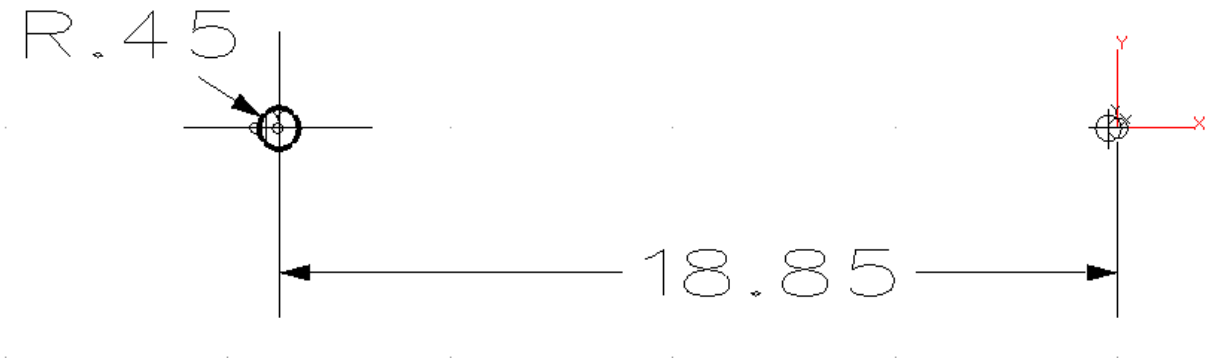
- Pick the revolved solid as the base shape.

- Pick the two thread forms as the shapes to be added.
- Make sure you use the **boundary face** option to add the thread forms to the inside of the circular rib as shown below.
- Create R**0.3**mm **fillets** using chain pick, where the thread form joins the circular rib (two places) as shown below in the cut-away figure.



## Creating the 45 outside ribs.

- Select the **Extrude** command.
- Create the new sketch shown below, but in this case create the sketch on the **XY** Plane.

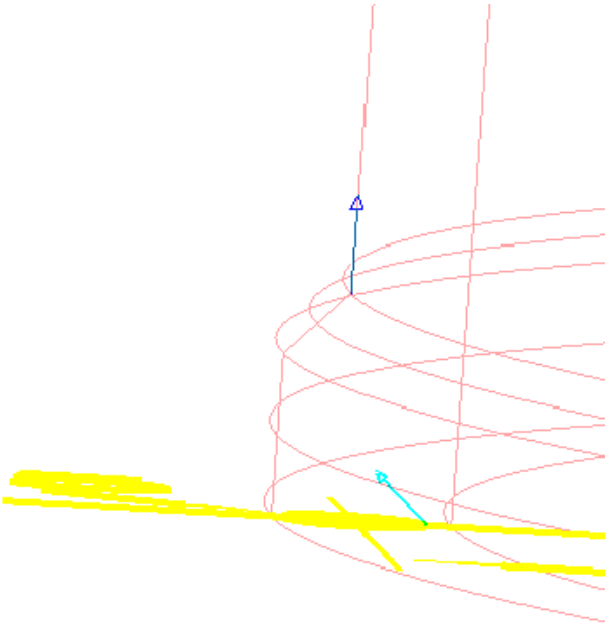



- Exit the sketcher.

The extruded solid needs to be kept parallel to the outside of the cap.

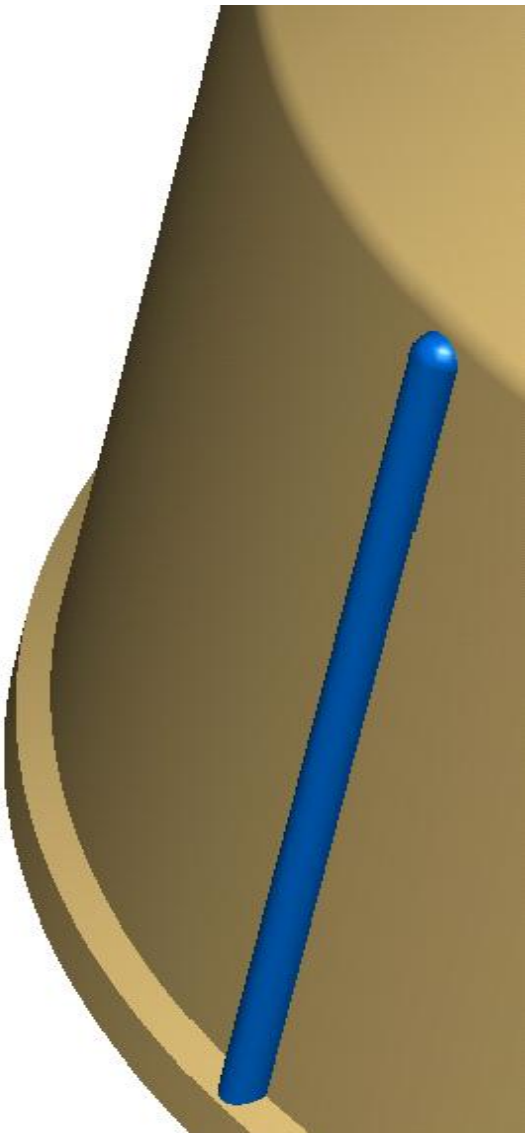
- From the option buttons, select **Direction**.
- Pick the outer edge shown opposite and the solid will extrude in this direction.
- Start =**0**
- End = **17.4**

Complete the rest of the command.



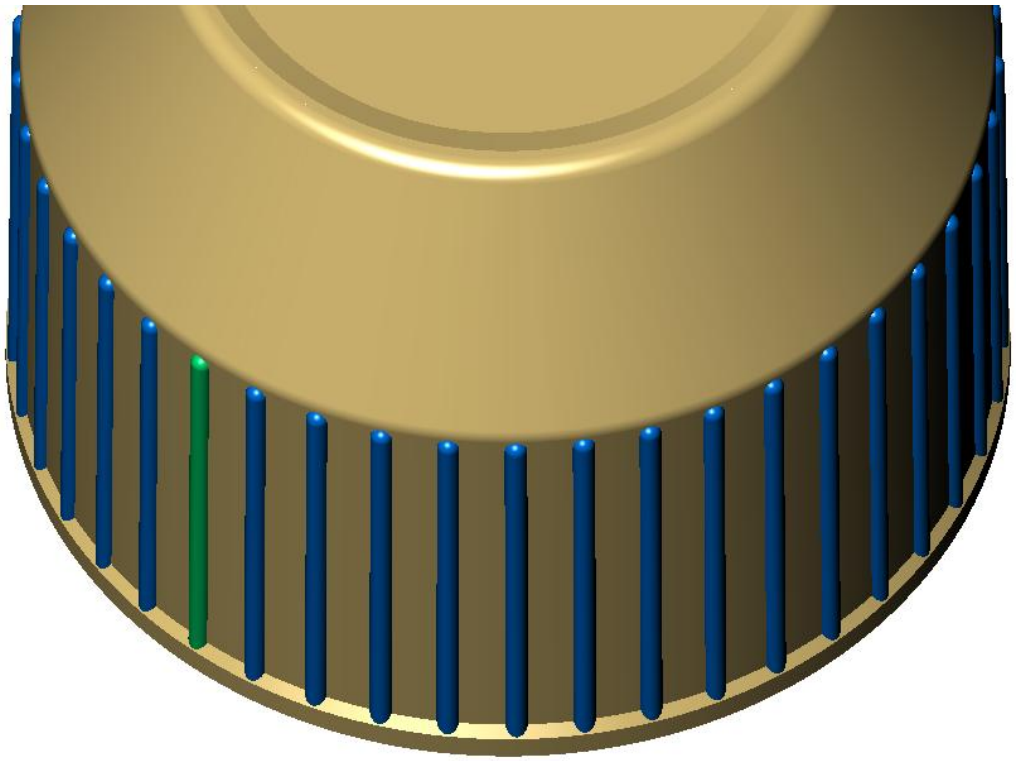
Temporarily blank the Revolve.

- Create a **0.4** mm **fillet** on the top of the cylindrical extruded solid as shown opposite.
- The radius is slightly smaller than the radius of the cylinder but is well within tolerance, the tiny flat on the top of the cylinder will disappear when the shape is joined to the main body.
- If however a full sphere were absolutely required, how would we accomplish this?



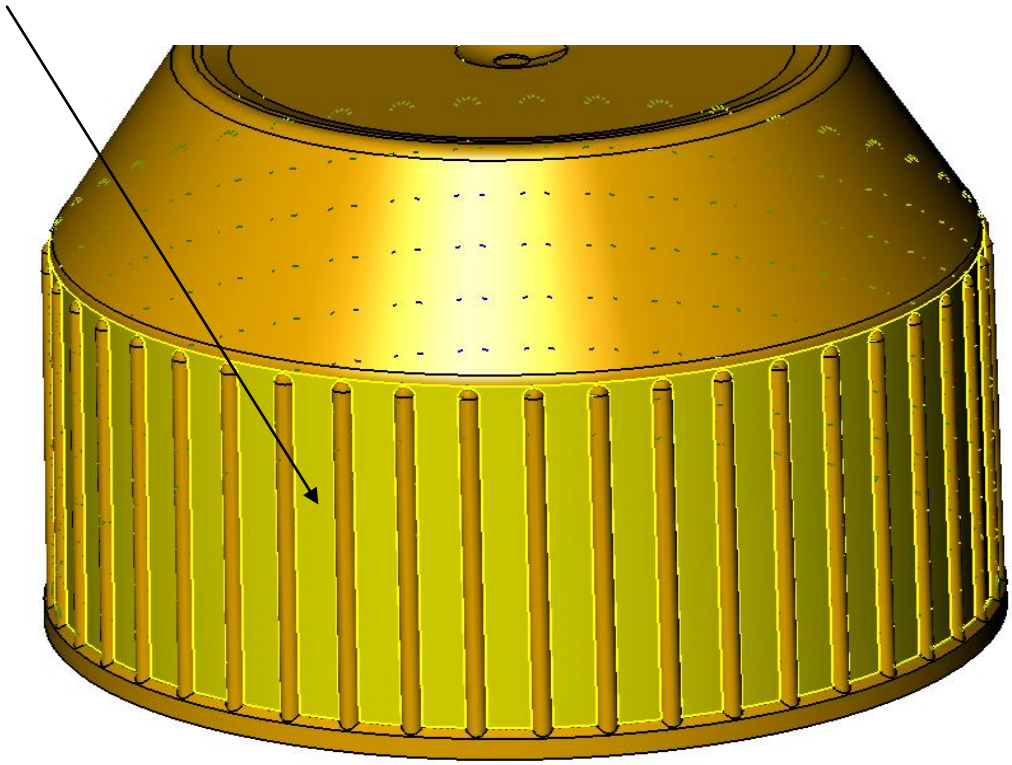
**Pattern** the extruded solid around the **Z** axis **45** total at **8** degrees intervals, as shown below. 

**Add** the 45 ribs to the cap.

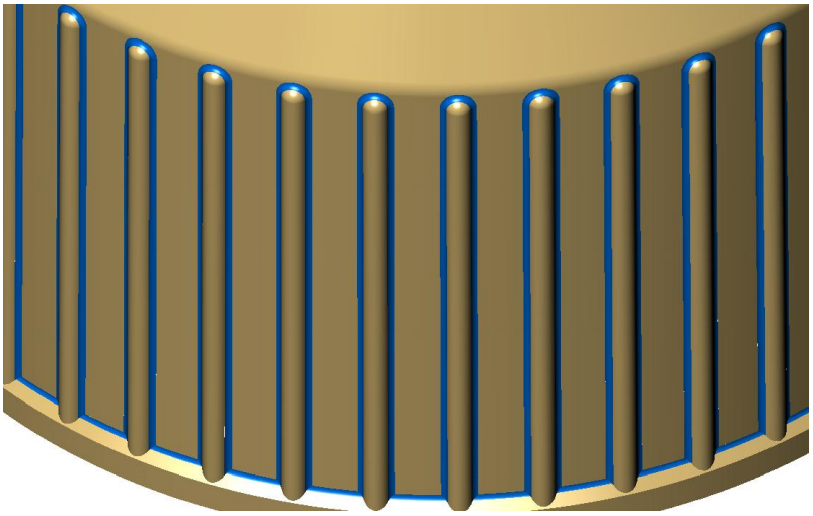


Create a “**Loop fillet**” **0.25**mm radius.

- Pick the face shown below.



- Complete the loop fillet command as shown opposite.
- This command saves you from having to pick 135 edges.



## Complete the details if time permits.

You should now be able to create the revolved cut shown on detail A on the original detail drawing, and also the 8 ribs shown in section YY on the original drawing. You may try these features yourself if time permits. The missing features are included in the example part that is supplied with this tutorial. So if you have any problems step through the history of the example to see how the features were created.

The finished part is shown below.