Solid Modeling 1

In this task we will learn to design a part using some advanced ThinkDesign solid modeling features. Although there are different ways of designing using solid modeling, we shall learn a unique way of doing it using ThinkDesign.

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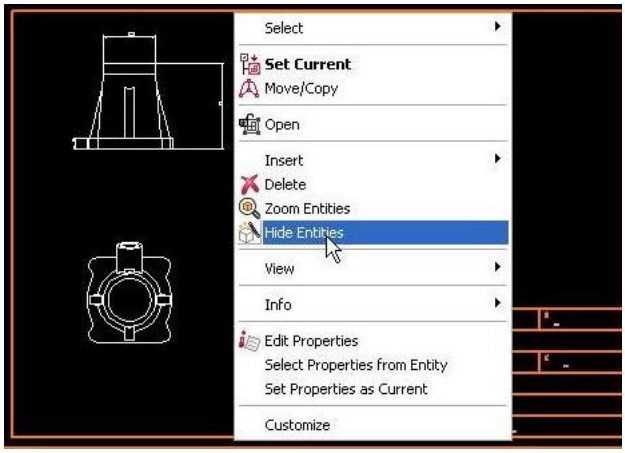
1. 1. Rotational solid and linear protrusion.

- Open the example.dxf file. If it does not open automatically with this task, you can find it in your web training folder.

Open the model file from the files folder inside the task folder wherever it is downloaded

- Open a new model file and name it "new_model.e3". Set the windows to **Tile Vertically**.
- Hide the title block and border of the drawing as shown below.



- With your mouse, drag a selection window around the three views of the drawing so that all three views are highlighted.

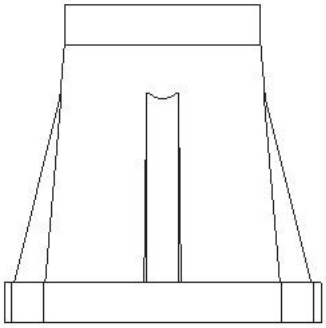
Drag the three views from the drawing and drop them onto the new_model modeling window.

Note:

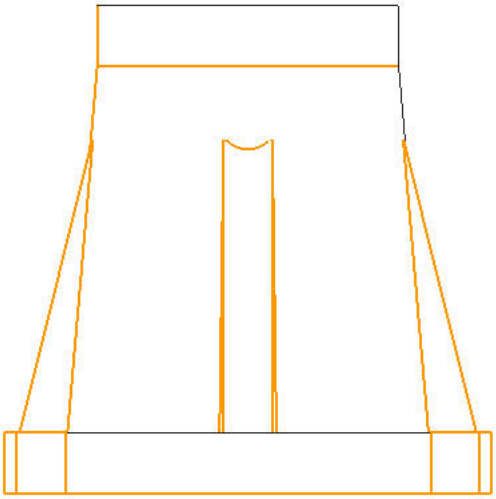
Groups will be exploded because in 3D environment there are no groups available. ThinkDesign uses other tools like Solids and Components for creating unique grouping of objects.

Confirm that the 2D geometry has been imported into the 3D model.

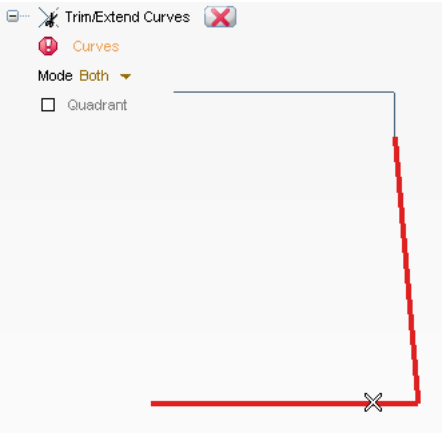
- Close the example.dxf file and start working on new_model.e3 file.
- Hide the work plane.
- Let us work on the Top view of the imported geometry.



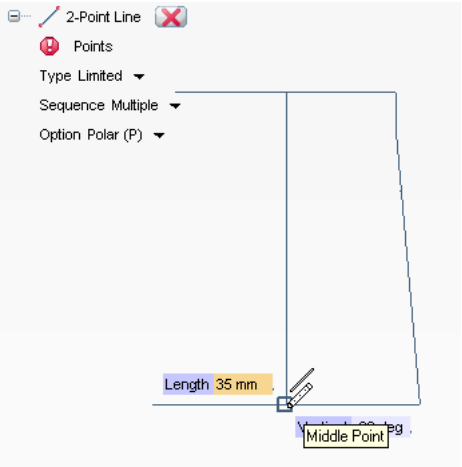
- **Delete Selection** all the lines as shown in the image file.



- Using **Trim/Extend Curves** command, join the two curves.

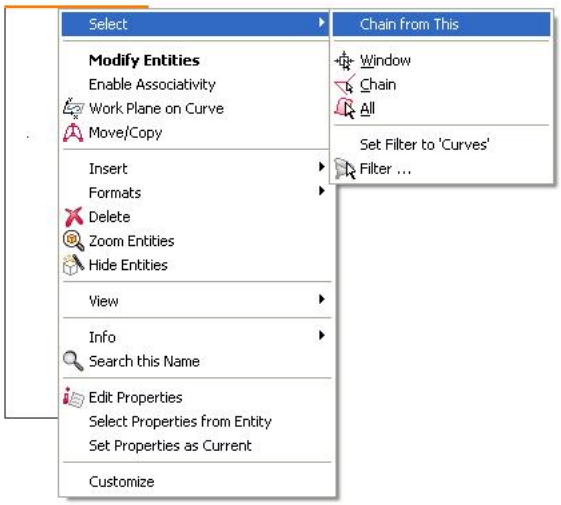


- Activate **Two-point Line**.
- Select the mid point of the top horizontal line using **Mid Point Snap**.
- and select the mid point of the bottom horizontal line.

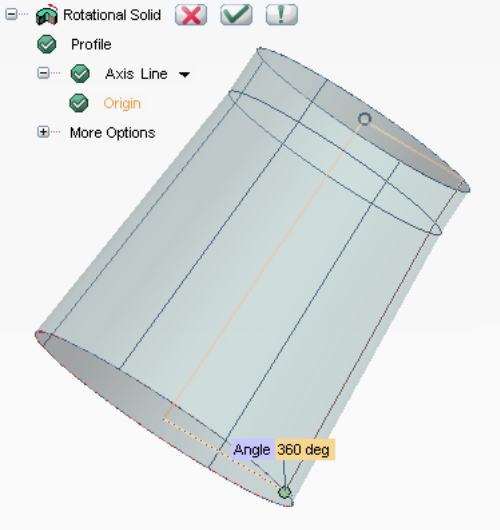


Delete the left side extended lines using **Smart Delete** and also delete the small lines OR dots if any,since it is imported from 2D file.

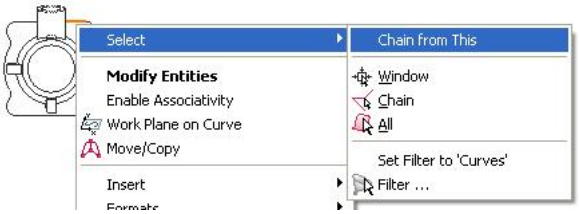
Right click on one of the lines and say Chain from this.



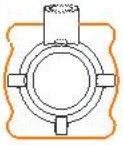
- By selecting the extreme left vertical curve as a Axis line, create **Rotational Solid**.



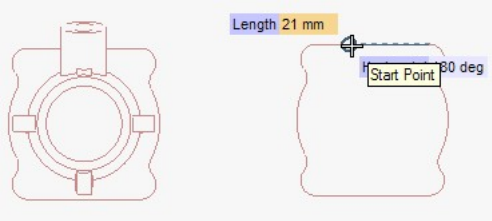
Now we need to create a base. Let's select the top horizontal line of the front view and say Chain from this, as shown below.



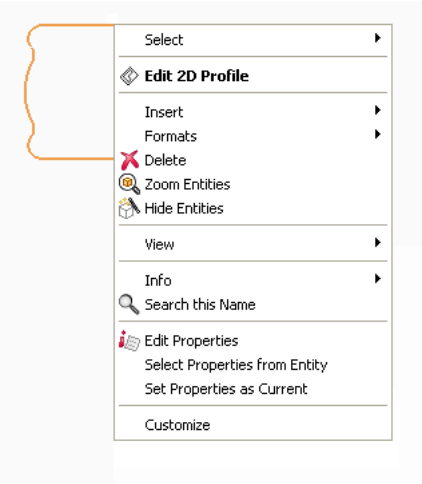
- Hold down the ctrl key and Drag and Drop these curves over to the right in order to copy them.



- Double click on the start point of the open curve and join the end point by dragging that curve horizontally to the one opposite to it as shown below.

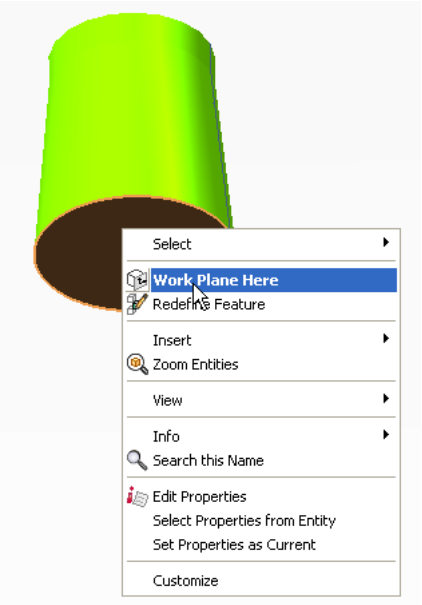


To covert these curves into a Profile select the Edit Profile option in the context menu.



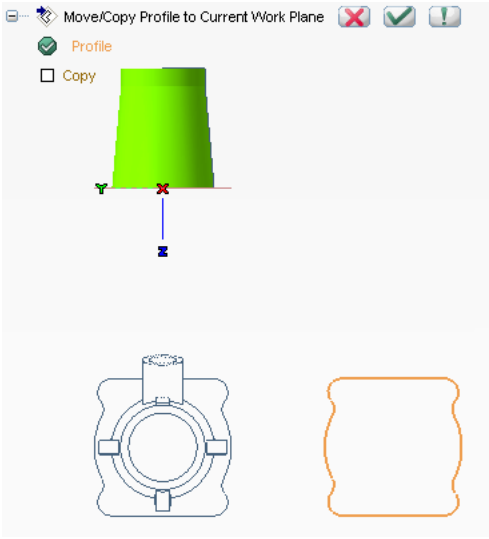
Get back to the Model mode by double clicking.

Now move the Work plane on the face of the Rotational solid.



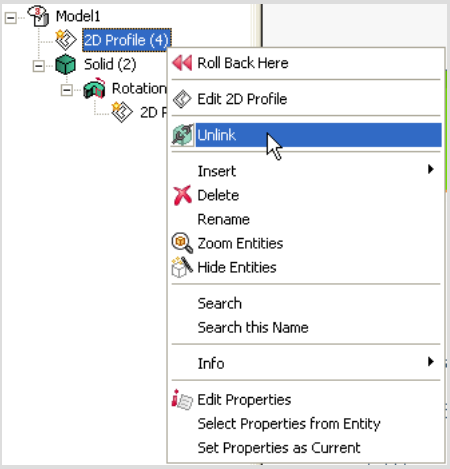
Note that, the profile is now perpendicular to the work plane. And we need to move this profile on to the work plane.

- Using **Move/Copy to Current Work Plane**, move the profile to the center of the base.



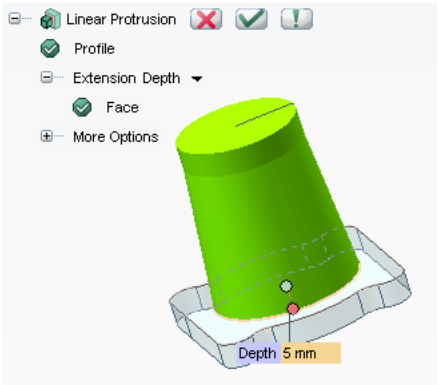
Note:

If we can not move the profile on the flat base of the Rotational Solid, then right click on the profile in History tree and say Unlink.



- Click on OK to complete the command.

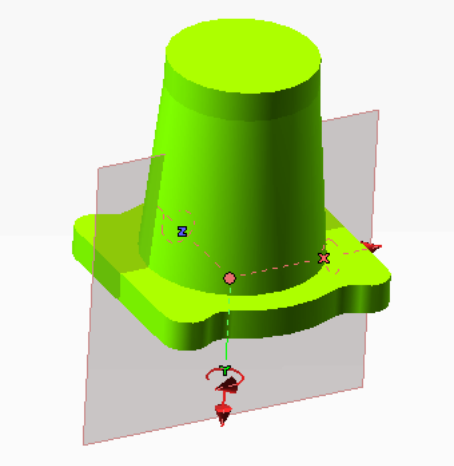
- Protrude it downwards to the Depth5mm.



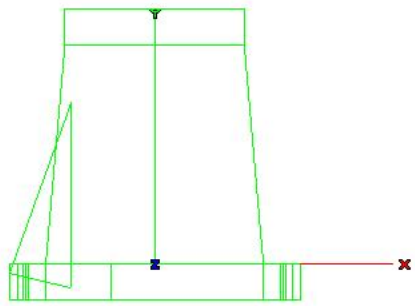
- Hide all 2D geometry.

2. 2. Creating Ribs, Fillets and its pattern.

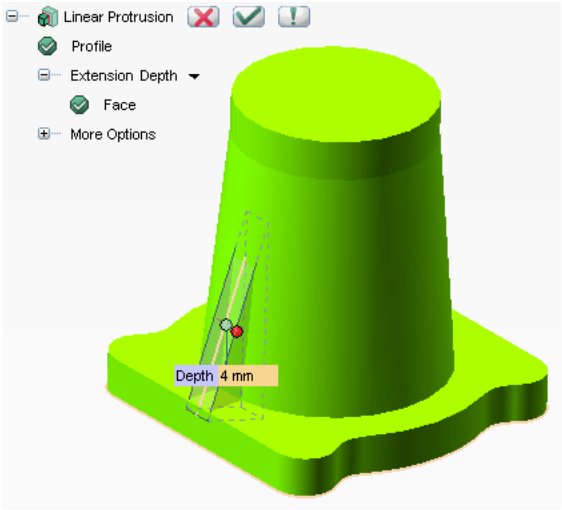
- Set the view to work plane by pressing the F8 key.
- Change the work plane orientation by rotating it -90 degrees along X axis.



- Start the **Polyline** command.
- Go to the Profile mode.
- Draw a line with Length25 and make a triangle as shown below.

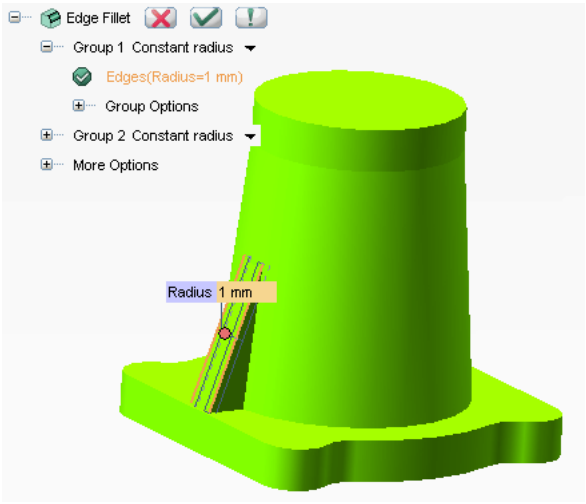


- Next, select the **Linear Protrusion** command and set the Extension: Depth
- For Face choose the tapered cylindrical surface.
- Set the Depth4mm.
- Right click on Depth and choose Symmetric so that the profile is centered in the protrusion.



- Click on OK to complete the command.

- Apply the **Fillet Edges** command to create a Radius 1 fillet at both edges of the protrusion we just created.

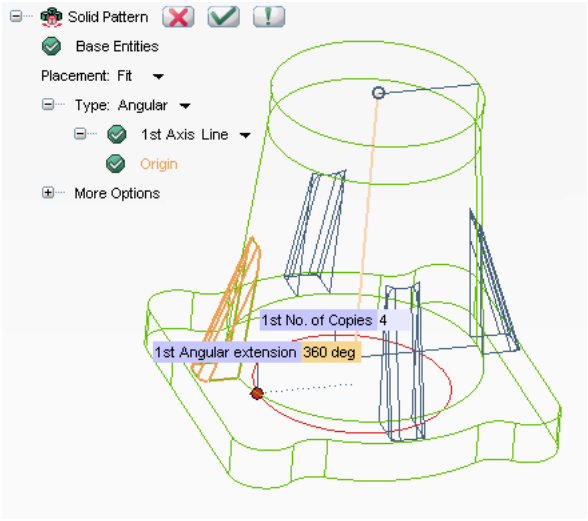


- Click on OK to complete the command.

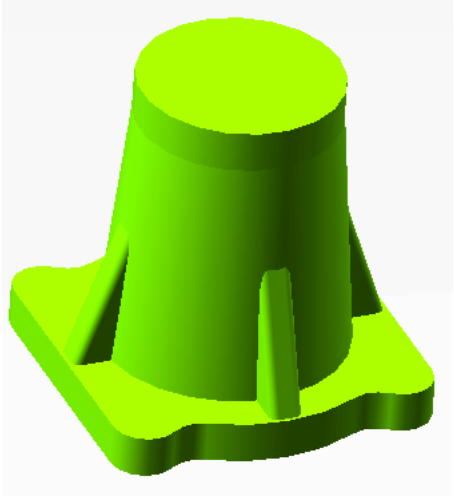
Next, we shall copy the protrusion and the fillet we just created to the three other sides of the model by using the **Pattern Solid** command. Create a Pattern using **Pattern Solid** command by selecting the protrusion and fillet created whole together to make 4 copies by giving 360 degrees.

Note:

As you can see, ThinkDesign may alert you that smart mode will be activate. In this case ThinkDesign copies the logic of the selected features, not just the geometry. Copies are obtained by creating a new feature based on the original data in the new position in order to keep references.



- Click OK.

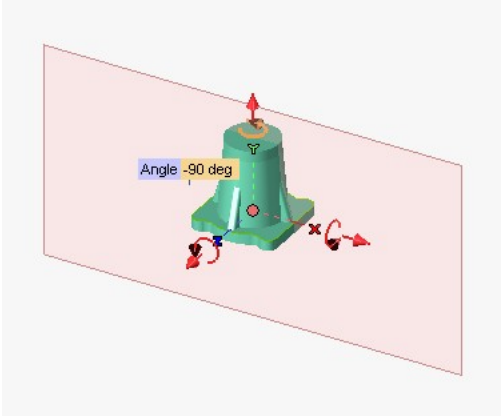


3. 3. Creating Rotational Protrusion and slot.

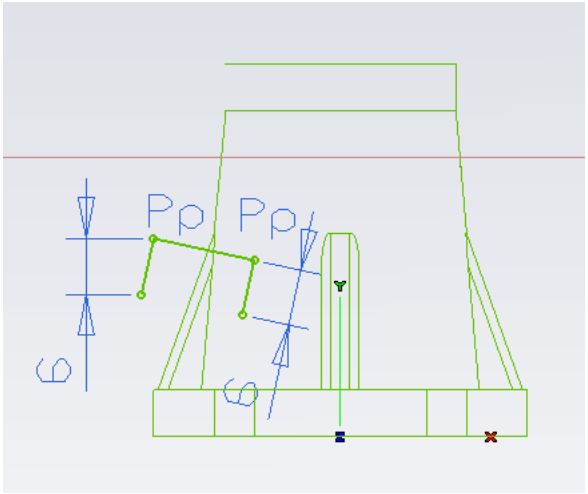
Let us create a Boss on one of the 4 ribs created.

Create a Boss on the rib which is on the flat side face of the Base protrusion.

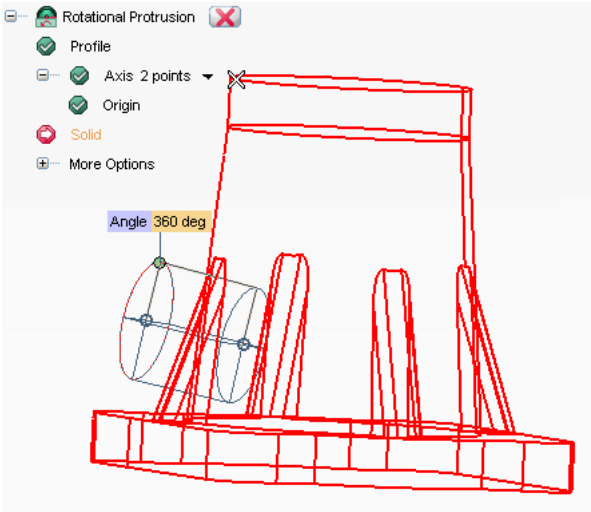
So rotate the work plane about -90 degrees along Y axis, as shown.



Set the view on work plane (F8) and go to Profile mode. Draw a profile as shown, using the **Polyline** command.

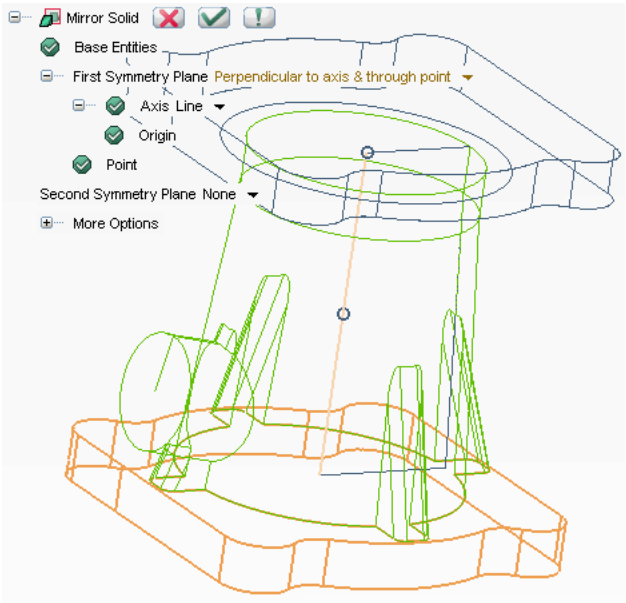


Create a **Rotational Protrusion** by selecting 2 end points of the profile as a axis line.

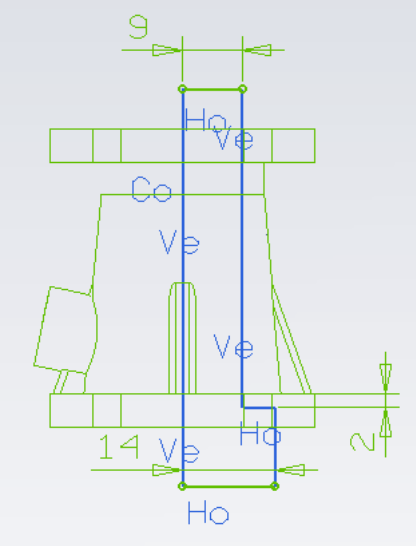


- Click on OK to complete the command.

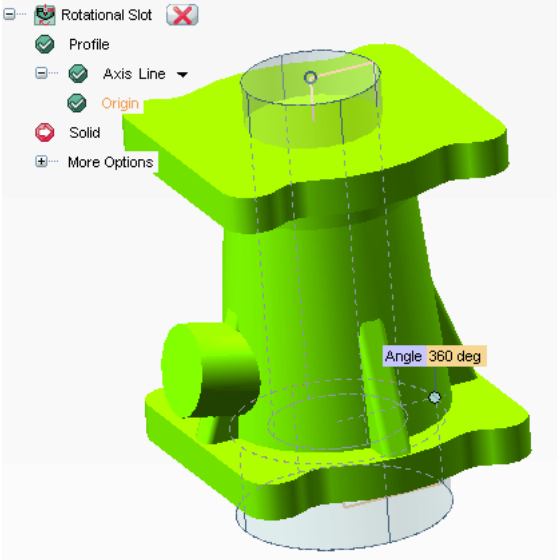
Mirror the base solid on the other side of the rotational solid.



We need to make a hole using **Rotational Slot**. Create a profile as shown.



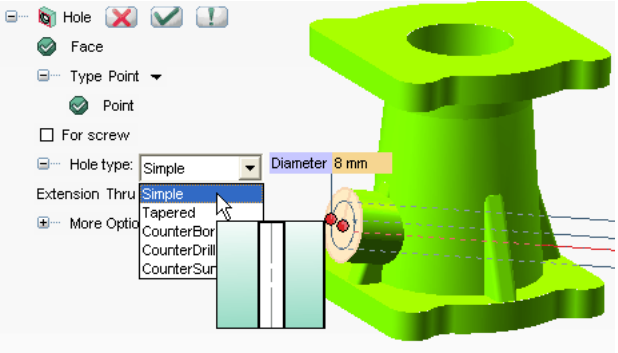
Create a slot using **Rotational Slot**.



- Say OK.

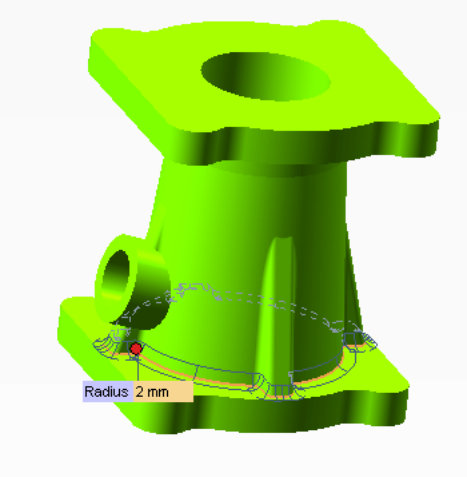
4. 4. Creating Holes and pattern.

Now, Insert a **Hole** of Diameter8 using thro next option.

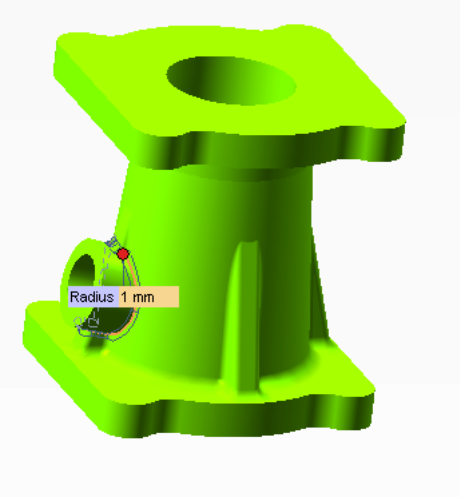


- Click on OK to complete the command.

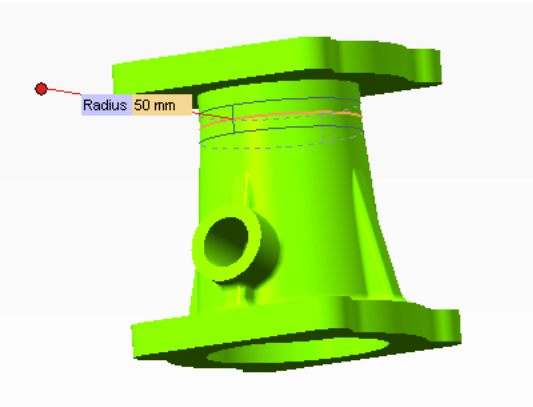
Now its time to apply fillets to the model.



Apply a fillet of Radius1mm to the bottom edge of the boss.



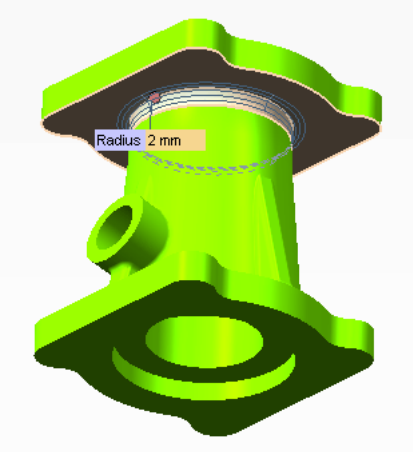
...and Radius50 mm as shown below.



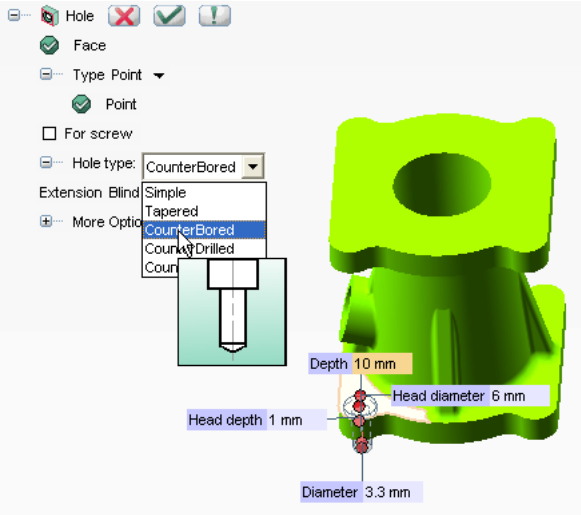
From 2007.1 version, a new feature called the **Fillet Face-to-Face** has been introduced. By using this command it is possible to apply a constant radius fillet by selecting two faces. Let's use this command and create a fillet now.

- Start **Fillet Face-to-Face**.

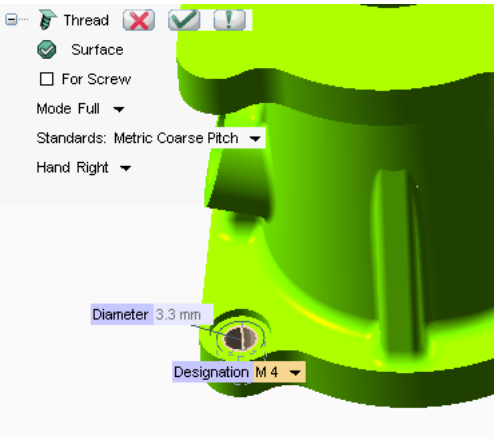
- Select the 2 faces as shown below.
- Set Radius2 mm.



Start the **Hole**command. Select the type as Counter Bored with the dimension entered as given for hole as shown.



Start the **Thread** and set the values as shown.



The **Hole** and **Thread** is created at one corner of the base solid.

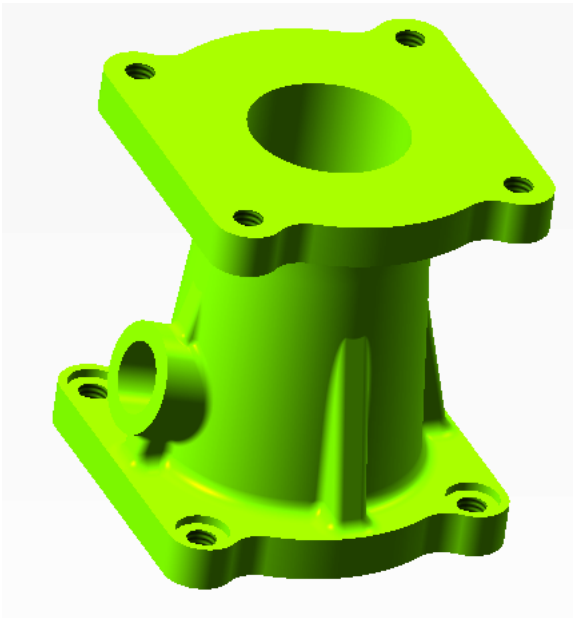
Start **Pattern Solid** command using linear-linear type with 1st Extension30 and 2nd Extension32.

The **Pattern Solid** has been created. Rebuild the model.

The last 3 operations, Hole, Thread and Pattern Solid need to be repeated for the upper part of the mirrored solid.

- Go to **Tools** **Modeling** **Rebuild All** and say Rebuild all.

Our solid model is now complete and ready.



Congratulations!!! Awesome .