Solid Part Modeling 2

In this second part of Solid Modeling we use ThinkDesign to create a complex cast part. The model will become "multishell" in the beginning of the task, but we shall learn how we can still build features on it. So let's get started!

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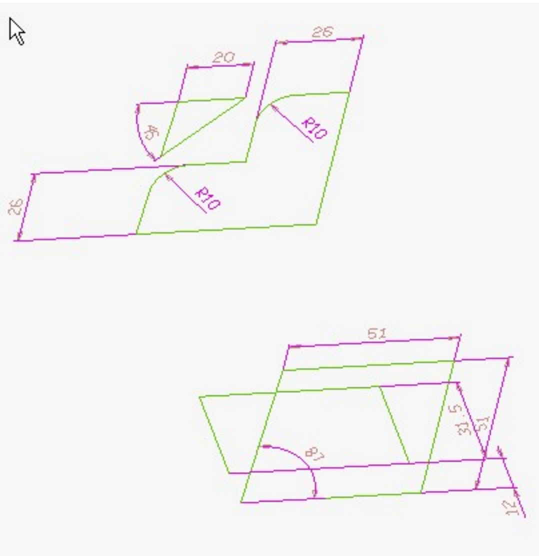
1. Step 1 : Profiles and Features.

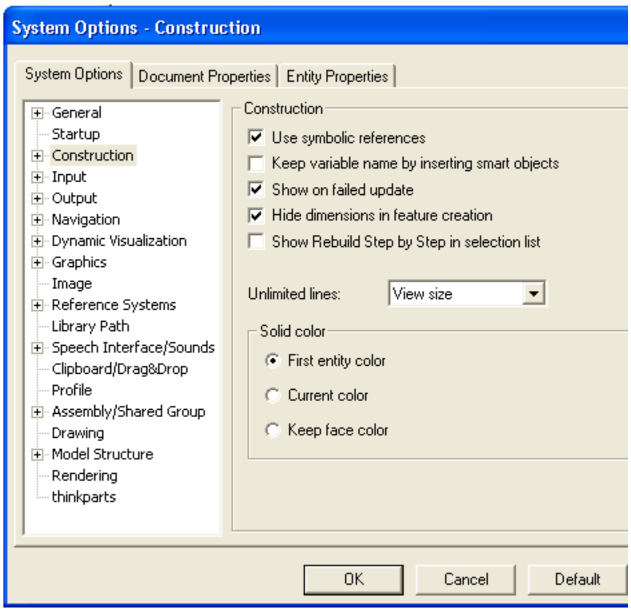
The final model is shown above. Notice that the final part is inherently symmetric and so we can save time by modeling only a quarter of the part. We will then mirror and copy it to complete the model.

The profiles have already been created. We can start with 3D commands on these existing profiles.

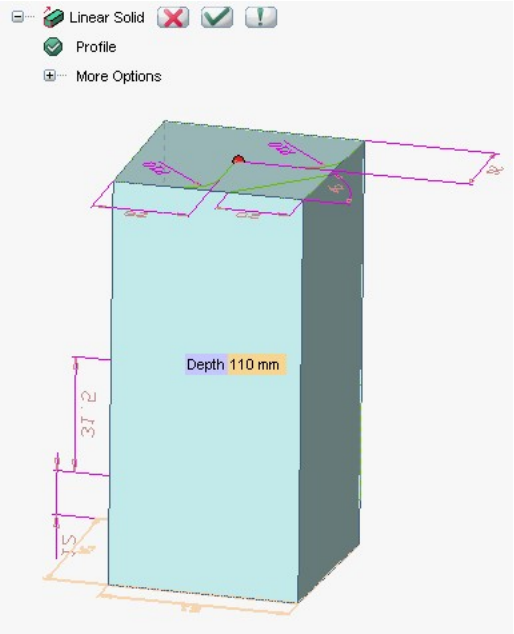
Hide the work plane by hitting the W key - we do not need it to be shown.



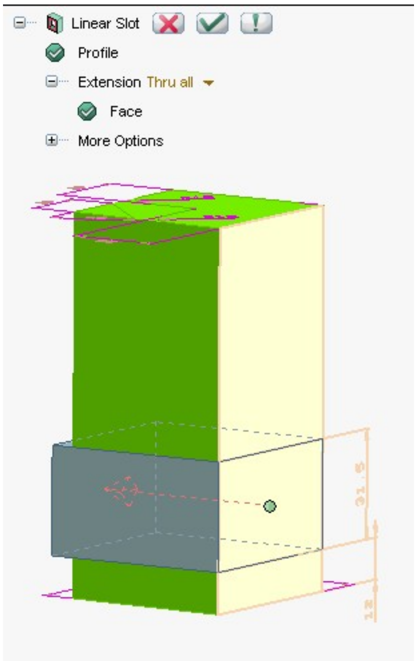
- Select **Tools** → **Options/Properties**.
- Go to the System Options tab.
- Select Construction in the history tree.
- Check the box Show on Failed updates if it is not checked.
- Say OK.



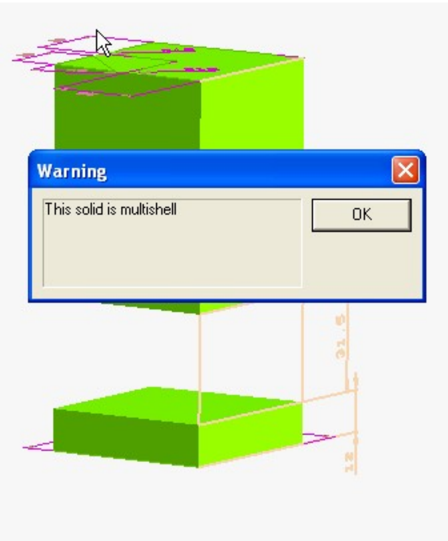
- Now start command **Linear Solid**.
- Select the Base_Profile from the history tree.
- Set the Depth value to 110 mm.
- Click OK.



Using LS_Profile1 and the **Linear Slot** command, cut the solid in two.

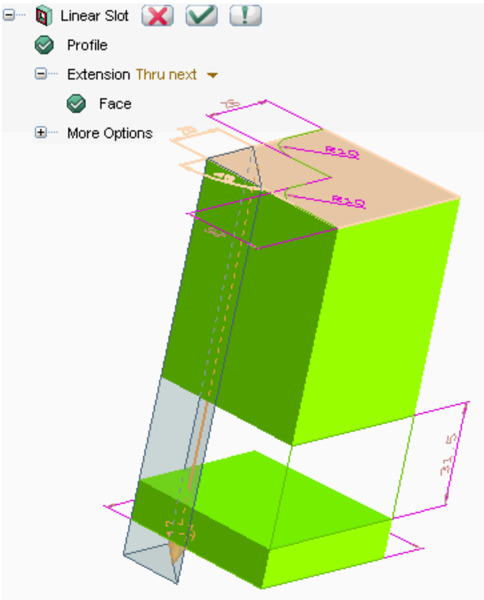


This results is a multishell solid. Click OK and go ahead with the task.



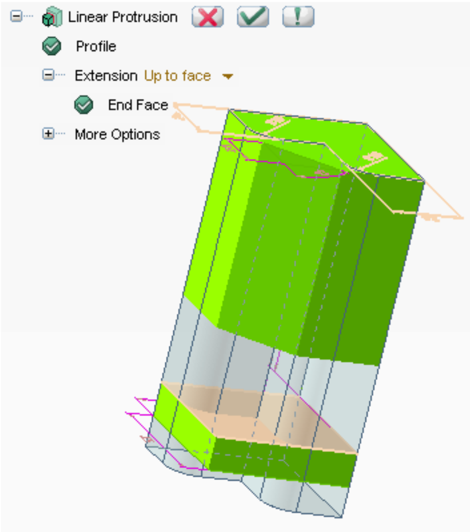
Make another **Linear Slot** - Through Next, using the profile LS_Profile2.

Using this Extension Thru next option, ThinkDesign will end the slot when it finds the first surface.



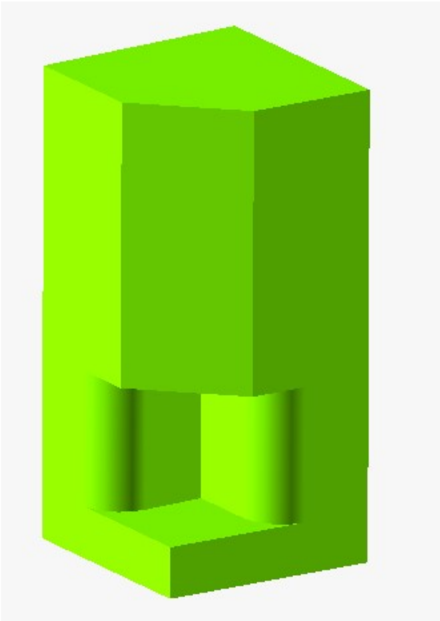
Create a protrusion using the Extension Up to face option. In this way the object will result in manifold solid.

- Activate **Linear Protrusion**.
- Select LP_Profile as Profile.
- And Up to face from Extension list.
- Select the top face of the second solid as the End Face. See image below for details.



- Click OK from the Selection List.

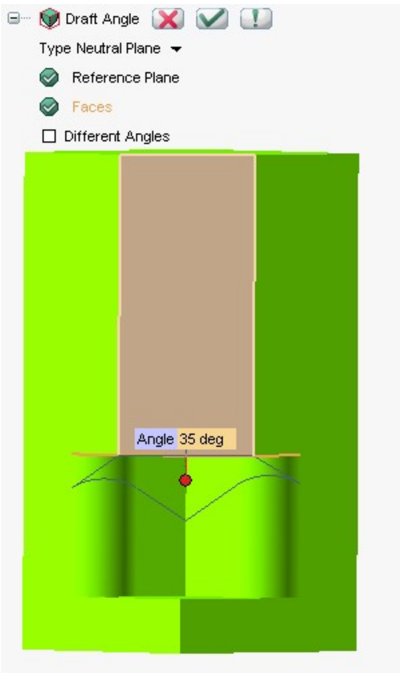
Hide all the profiles and dimensions using **Hide Entities**.



Note that now the model is a manifold solid.

Let's add material to the solid.

- Start the **Draft Angle** command.
- Select the top face as the face to be drafted.
- Set the angle to 35. Angle35.
- Click OK.

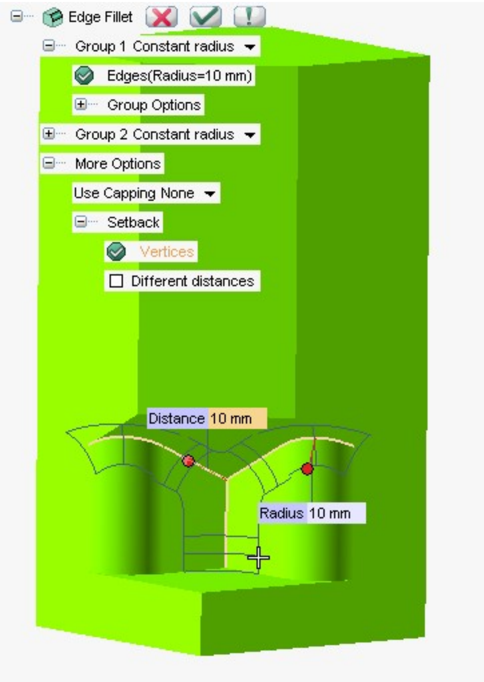


Let's add some fillets in the next step.

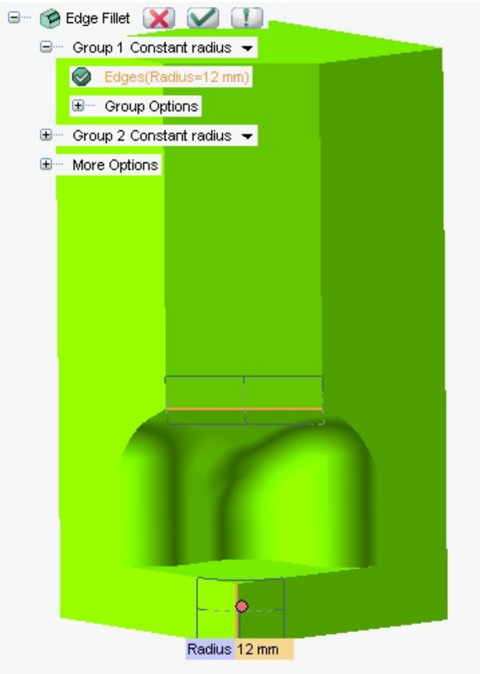
2. Step 2 : Fillets and Shell features.

Starting from 2007.1 version, It is possible to add Setback to fillets created on three or more merging edges. Setback is the distance from the vertex up to which a capping face of a fillet will extend on a filleted edge. The Setback option has been provided in Edge Fillet selection list.

- Start **Fillet Edge**
- Select the Edges as shown below. Set Radius10 mm
- Expand the More Options and Setback option. For the Vertices select the Point at which all these surfaces meet. Set Setback distance Distance10 mm.
- Hit OK.

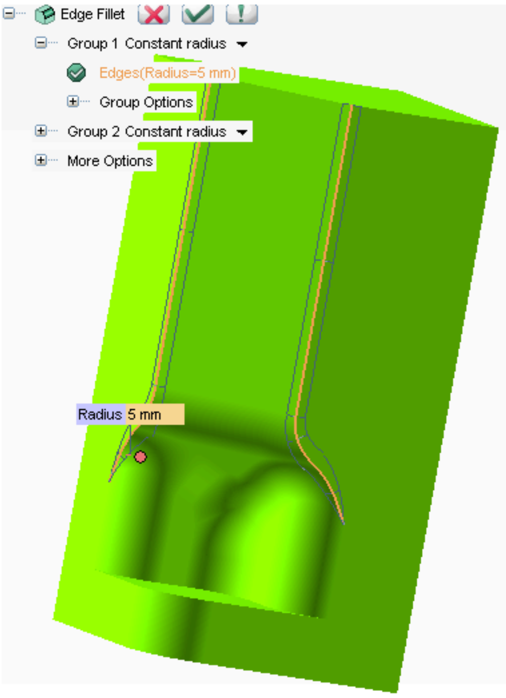


Insert another fillet of R12 on the edge as shown below.



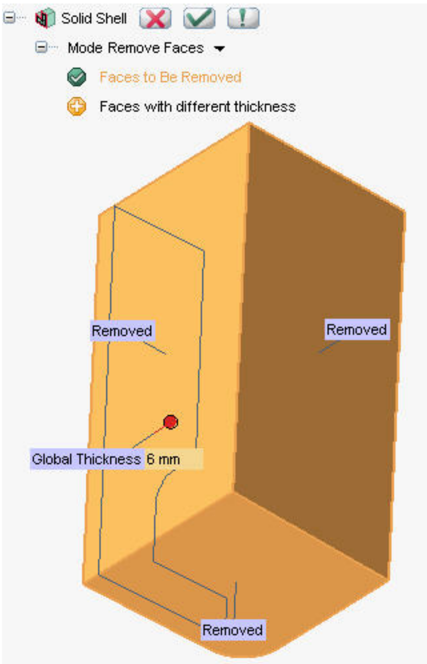
Click either Apply or OK from the Selection List.

And two fillets of R5 on the long vertical edges.

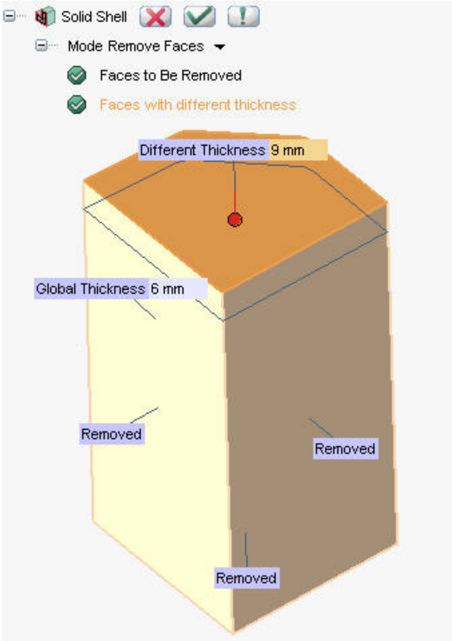


Now let's make shell out of this solid with different thicknesses.

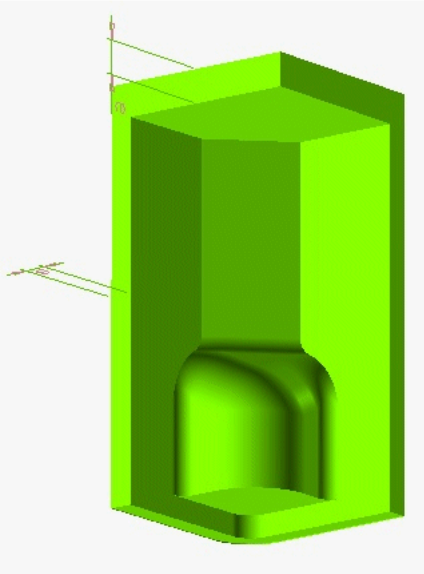
- Start the **Solid Shell** command.
- Set the ModeRemove Faces.
- Select the two side faces and the bottom face of the solid and set the Global Thickness6.
- We can either right click on the Graphics area and click Continue or click Faces with different thickness to add a different thickness to the part.



- Select the top face and assign the thickness value 9 Different Thickness9



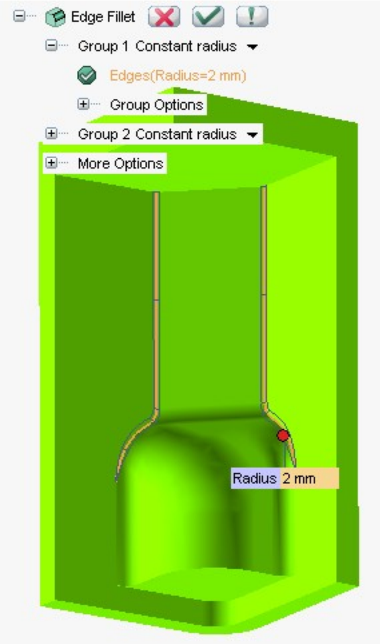
- Click OK.



NOTE: if we observe the inner part of the model, the last two fillets added were removed during the shell operation. Try to change the radius of the last two fillets from 5 to 8. Rebuild the model. What do you see?

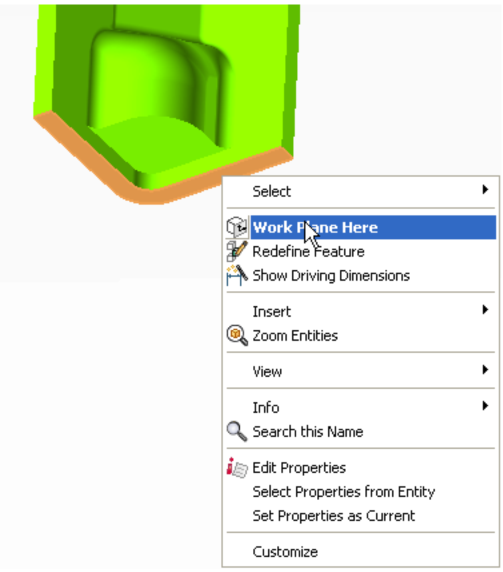
Undo the fillet modification to restore the fillet to R5.

- Insert a fillet of R2 to remove the sharp inner edges.



3. Step 3 : Base Part of the Model.

Right click on the lower surface of the model and click Work Plane Here.

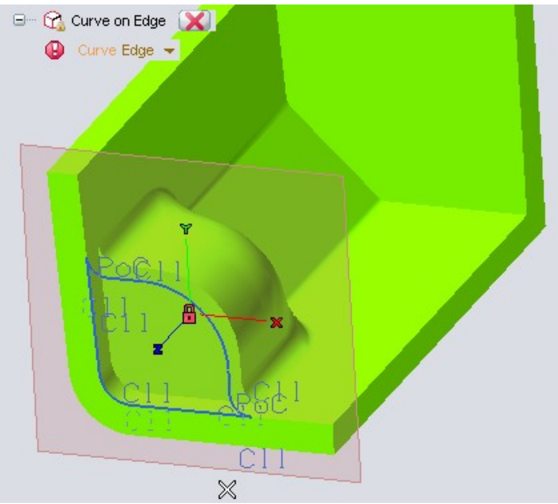


Go to 2D Profile mode.

Create a Profile using the **Curve on Edge** command to copy the shape.

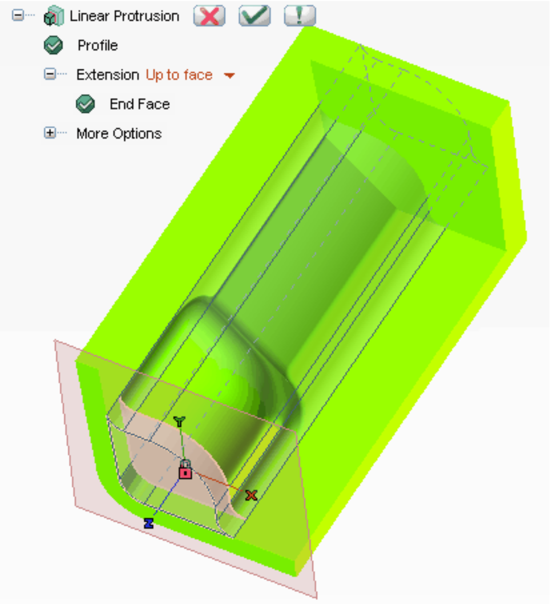
Note :

To avoid an over-defined profile using the **Curve on Edge** command, select the arcs first and then the lines.

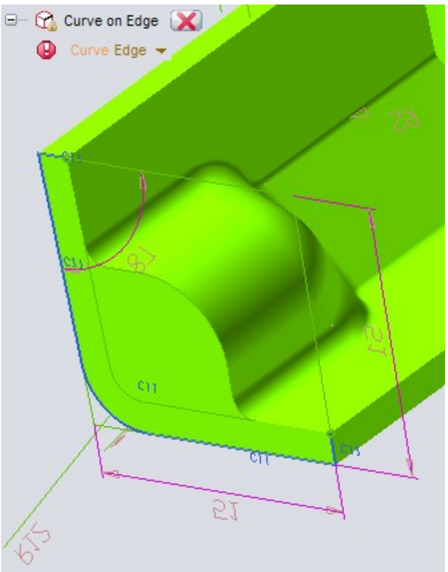


- Create a **Linear Protrusion** using the Up to face option.
- Click More Options.
- Check for Extended Face.

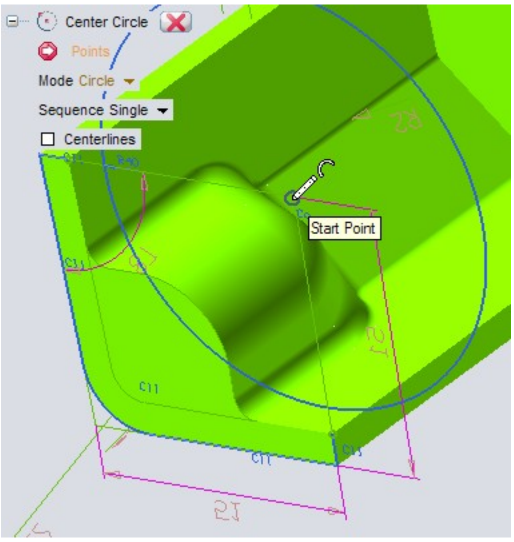
For more details on Extended Face refer to Extension in Up to face Mode.



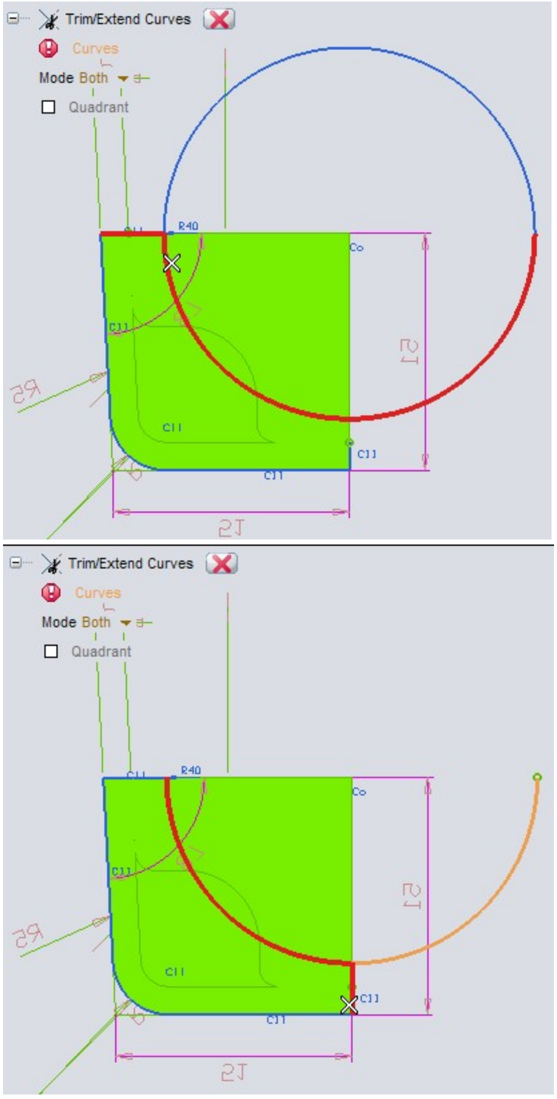
- Use **Unhide Entities** to unhide the Base_Profile.
- Go to 2D Profile mode.
- Create a profile of the outer edges of the bottom surface using the **Curve on Edge** command.



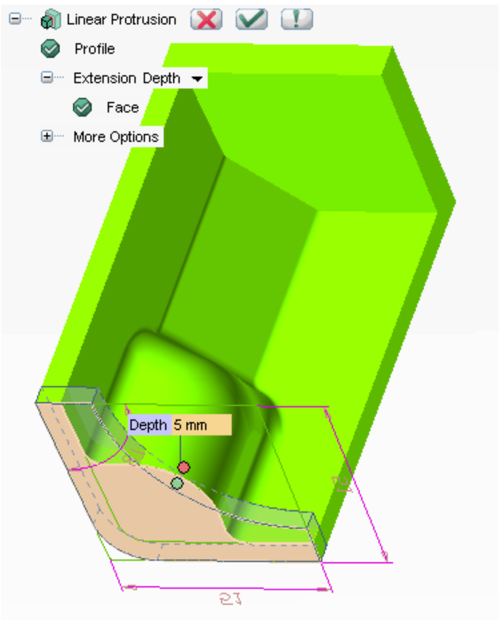
- Draw a **Center Circle** keeping the vertex of the Base_Profile as the center of the circle as shown.
- Set the radius value to R40.
- Hit Esc.



Use **Trim/Extend Curves** to trim the unnecessary arcs.

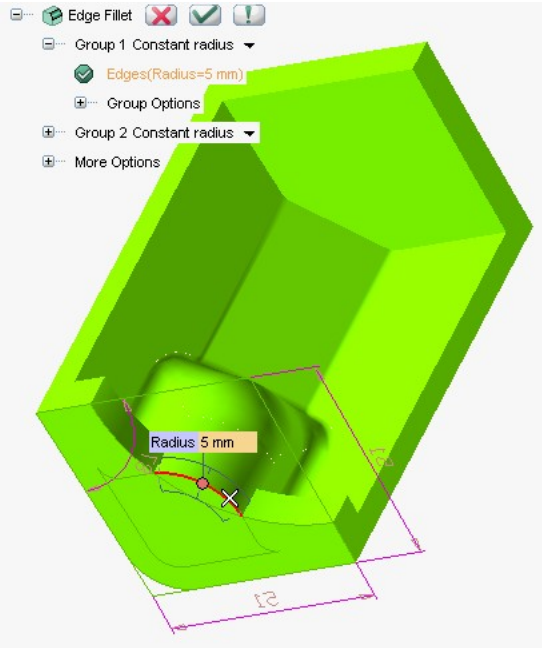


Next, extrude the profile into the part to a depth of 9mm, using **Linear Protrusion**.

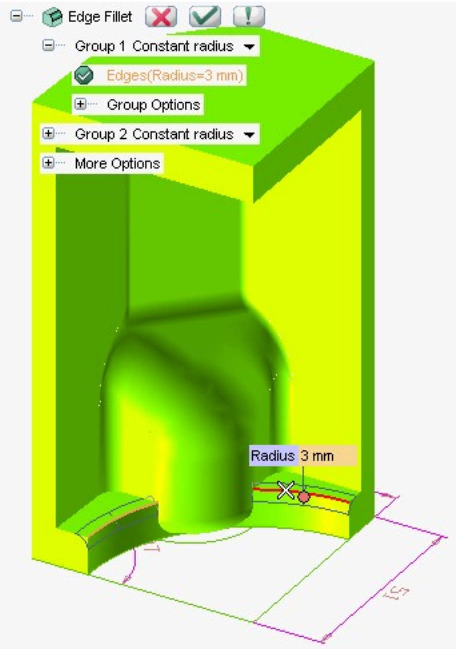


Start the **Fillet Edges** command.

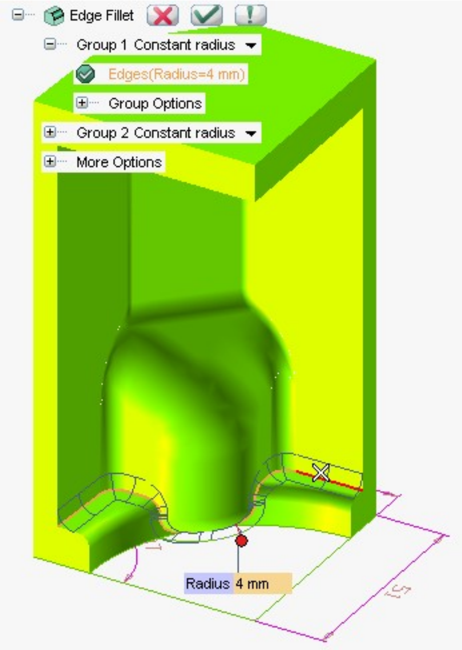
Let's start filleting the edges. Insert a fillet of radius R5 for the edge shown below.



Create another fillet of radius R3.

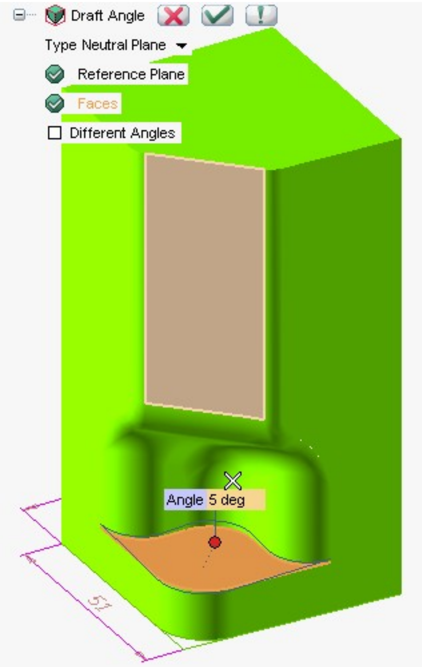


... and another fillet of R4.



4. Step 4 : Mirroring and Union of solids.

Add more material to the part using the **Draft Angle** command, choosing the Reference Plane and Face shown below, and a draft angle of Angle5.

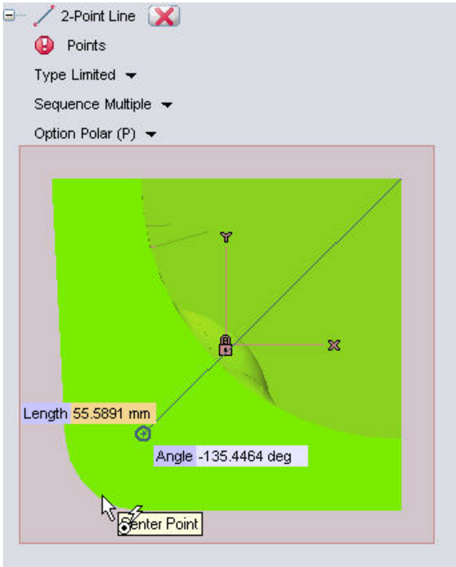


Let's make a slot.

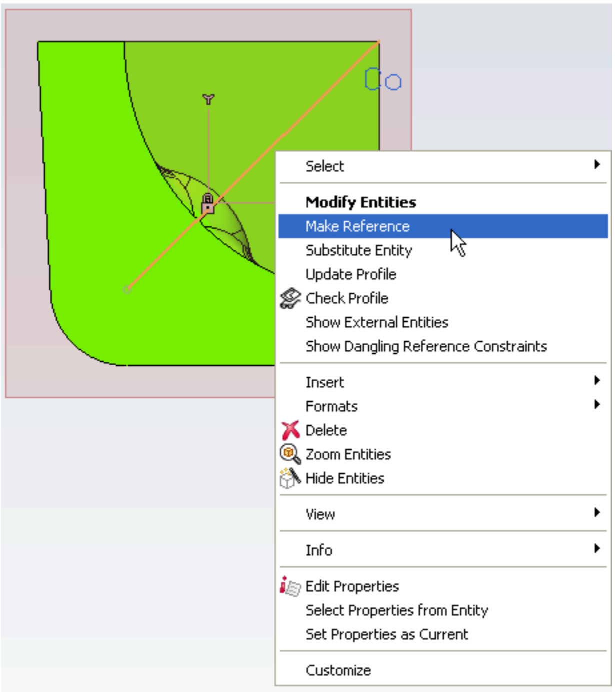
Press W to show the work plane and, if it is not, move it to the bottom surface.

- Activate 2D Profile mode.
- Select the **Two-point Line** command.
- The start point should be the upper-right corner vertex and the end point is the center point of the arc as

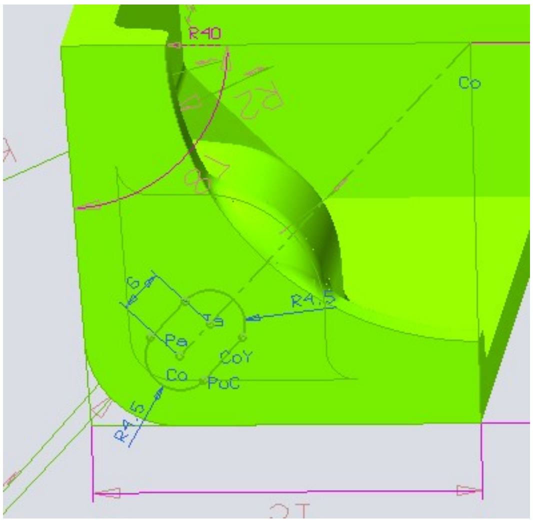
shown below. Select **End Point Snap** and **Snap to Arc Center** for ease of selection.



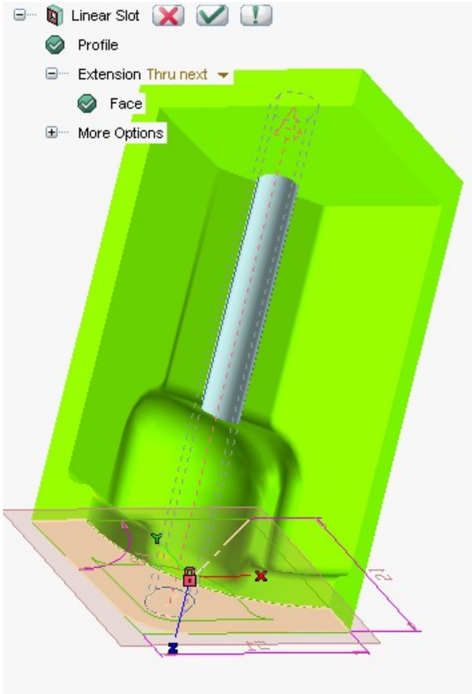
Right click on the line and say Make Reference.



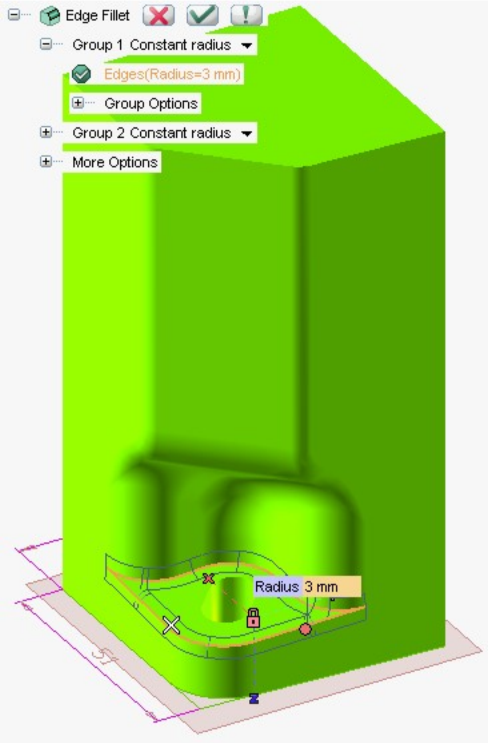
Create an oblong hole of radius R4.5 and center distance of 6, as seen in the image below.



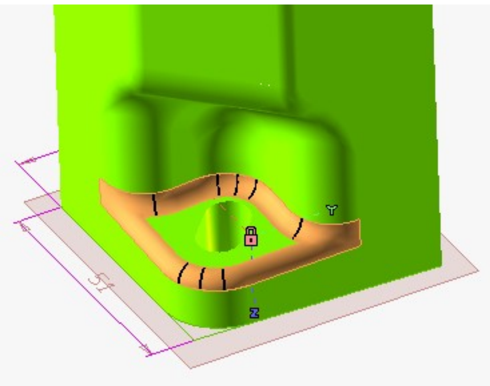
Using this profile make a **Linear Slot** with Extension Thru next.



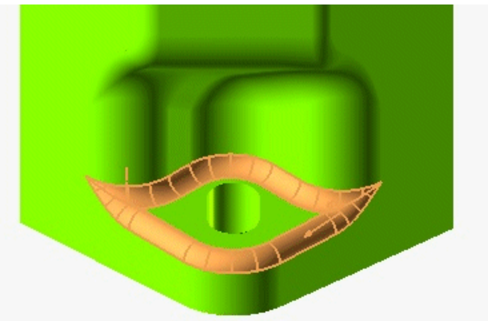
Now fillet two edges in a single **Fillet Edges** command.



The result is shown.



Undo the last operation and make the same fillets with two different commands - and examine the result. Notice that there is a difference in results depending on the order and method in which edges are being selected to fillet.

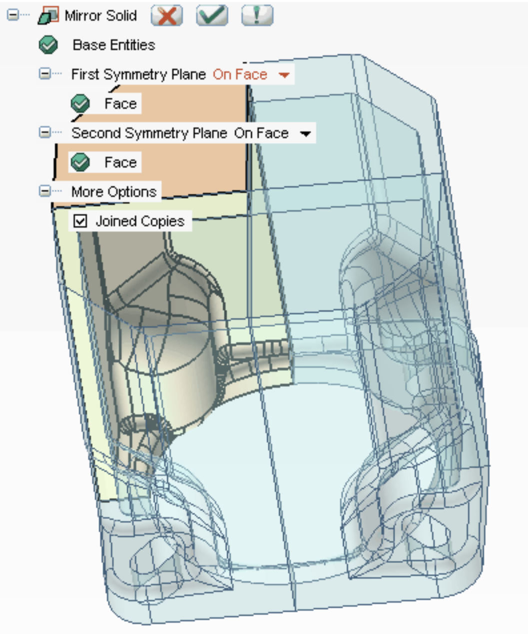


Undo this result and fillet the edges again as in the first step - with one single Fillet command.

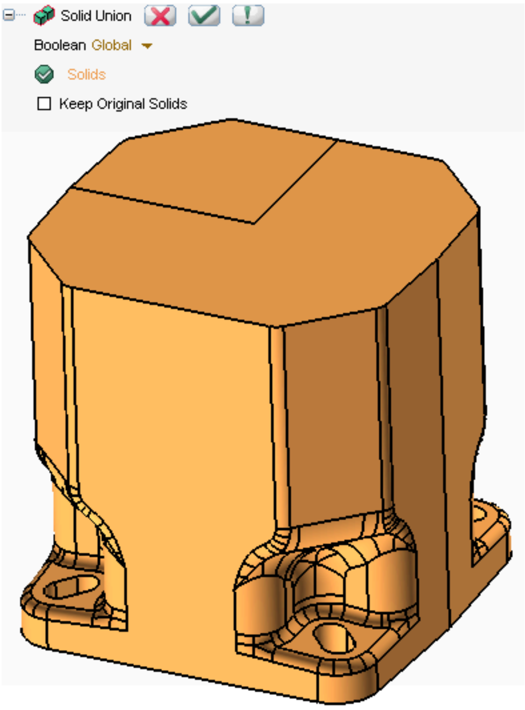
Now let's make a mirror of this solid to create the final part.

- Start the **Mirror Solid** command.

- Select the side face of the solid as First Symmetry Plane Face.
- And select the other face as Second Symmetry Plane Face.
- Check Joined Copies under More Options to create single solid instead of 3.
- Click OK.



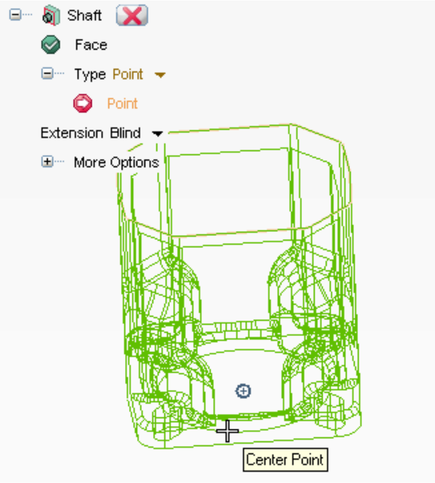
Use the **Solid Union** command to create one solid from the three.



5. Step 5 : Simple Shaft and Hole features.

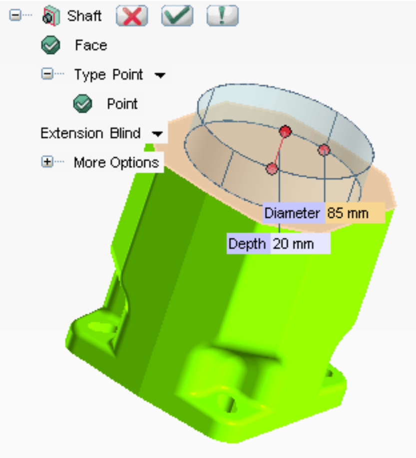
Make a **Shaft** on the top surface of the solid.

Select the center point of the bottom circle using **Snap to Arc Center** for the shaft center to place the shaft on top surface.

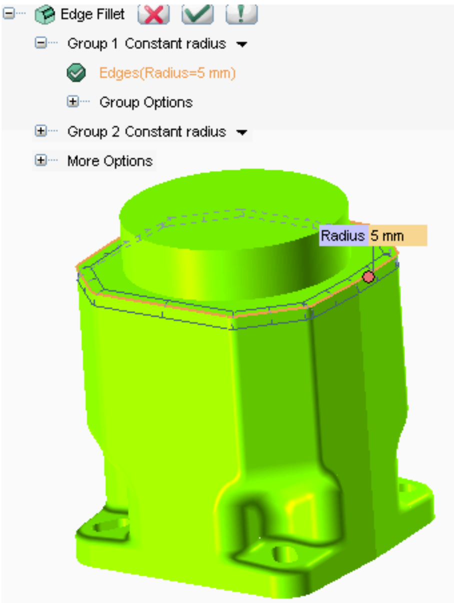


Set the Diameter85 and Depth20.

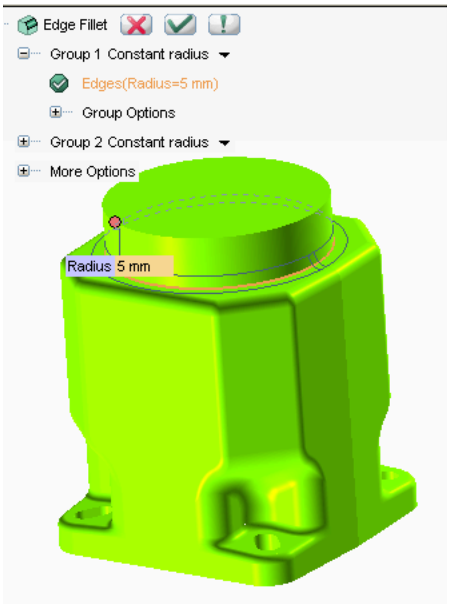
Click OK.



Insert a fillet of R5 on the polygonal edge...

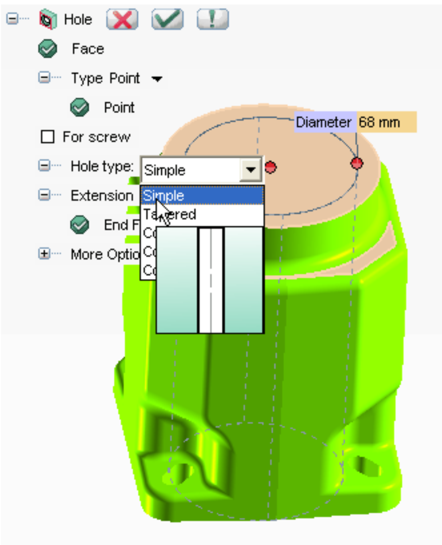


... and R5 for shaft lower edge.

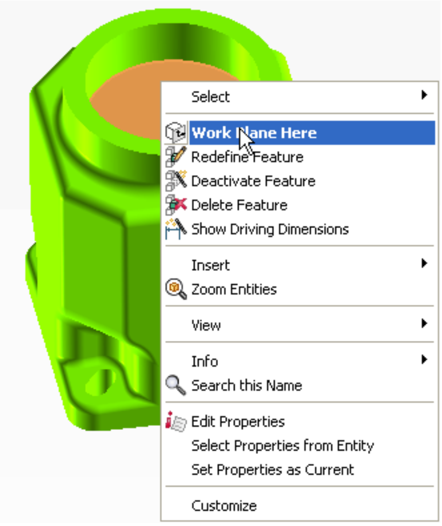


Add a **Hole** to the shaft.

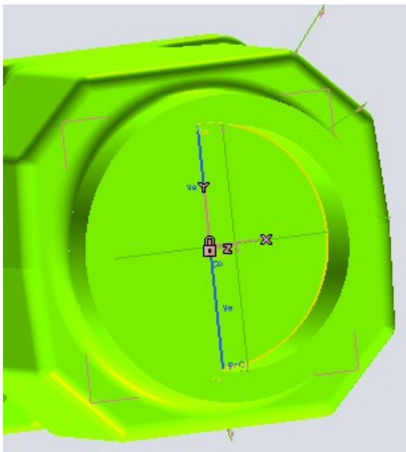
- Select the Extension to Up to face.
- Select the face on which the shaft was created as End Face. (See highlighted face below.)
- Click OK.



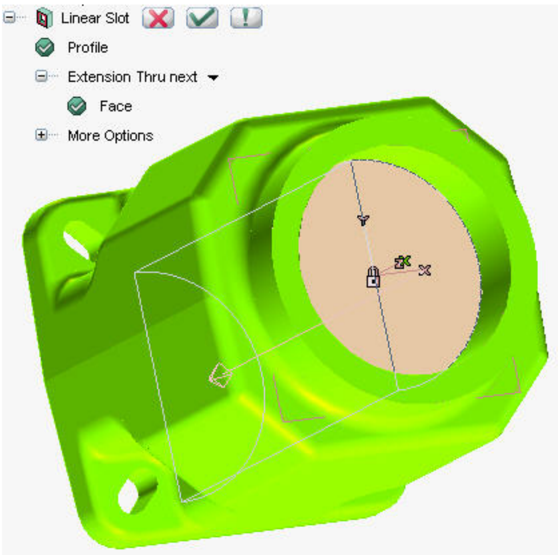
Right click on the bottom face of the shaft and click Work Plane Here.



Using **Curve on Edge** and **Two-point Line** create a half circle Profile in 2D Profile mode.



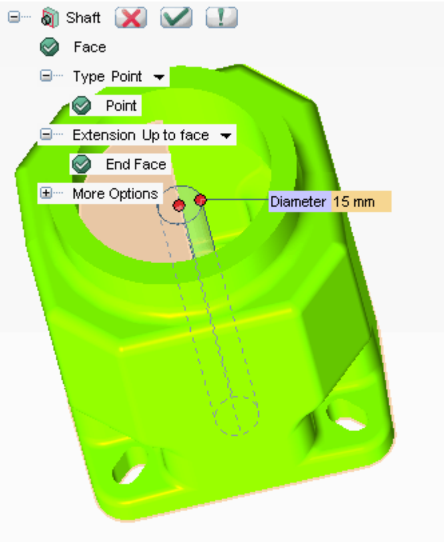
Cut the solid using **Linear Slot** with the setting Extension Thru next.



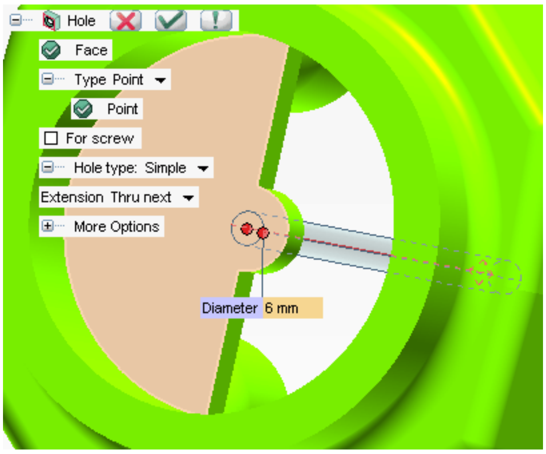
6. Step 6 : Inner part of the Model.

Make another **Shaft** of Diameter15

- Select the bottom face of the shaft.
- Set the Extension to Up to face.
- For Point, select the center point of the arc using **Snap to Arc Center**.
- For End Face, select the other side of the half circle (see image below for clarity).
- Click OK.

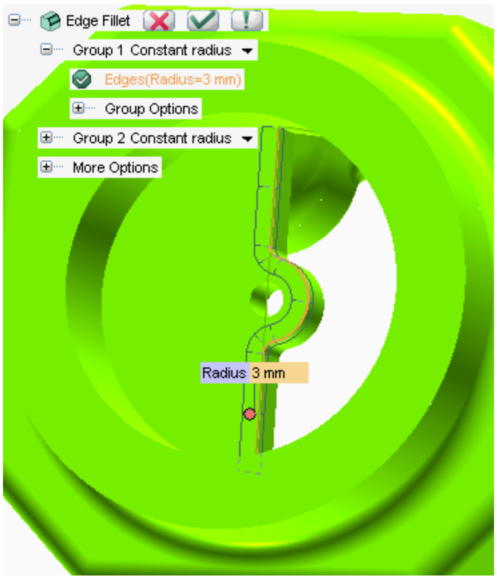


Using center point of the shaft, create a Thru Next **Hole** of Diameter6

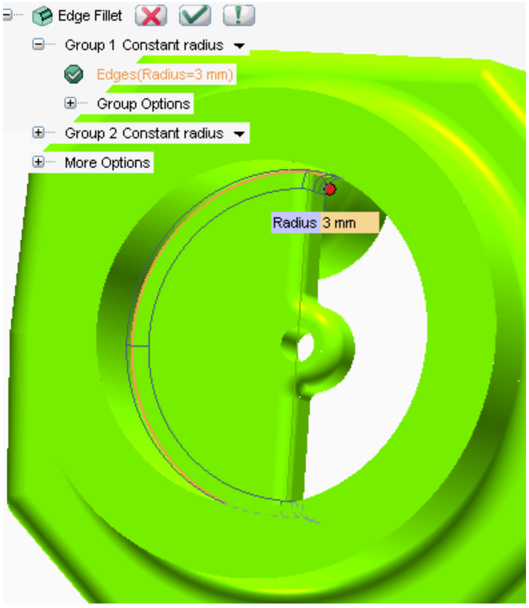


Insert fillets of R2 at the sharp edges of the shaft using the **Fillet Edges** command.

Insert another fillet of R3 on the top edge, as shown below.

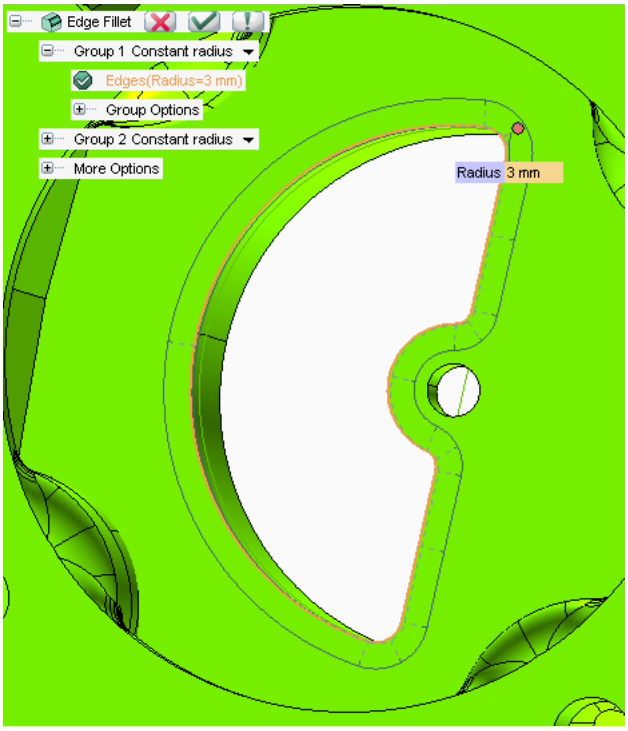


Insert one more fillet of R3 for the edge of the half circle.



Now fillet the inside continuous edge at a value of R3. We need to rotate the model using **Pan Zoom Rotate** for the selection.

NOTE: filleting is an art. :-) The order in which fillets are created on a part will have a direct effect on the part's appearance. The fillets just created could have been inserted on the part in many different ways. If you like, try few more methods and examine the results.

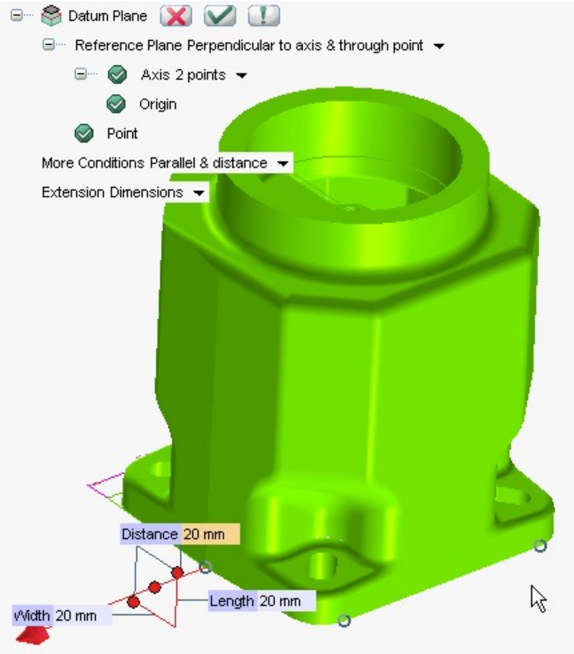


7. Step 7 : Datum Plane and Insert Feature Mode.

Create a **Datum Plane** to extrude a circle on the slanted face.

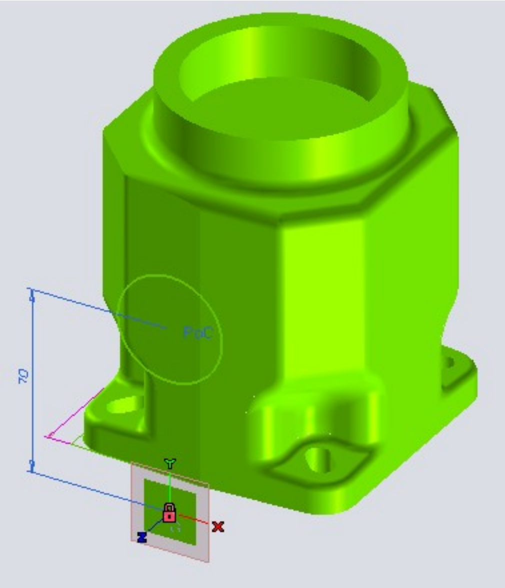
- Select **Insert**⇨**Datum**⇨**Plane**.
- Set the option Reference Plane to Perpendicular to axis & through point.

- Change the Axis to 2 Points.
- Select two points as shown in the image below. If needed, use the **End Point Snap** tool to assist you. You may need to change the display to **Wireframe View**.
- For the Point, again select the same point (in the red square below).
- Change the Distance to 20 so that the plane is away from the model.
- Click OK.

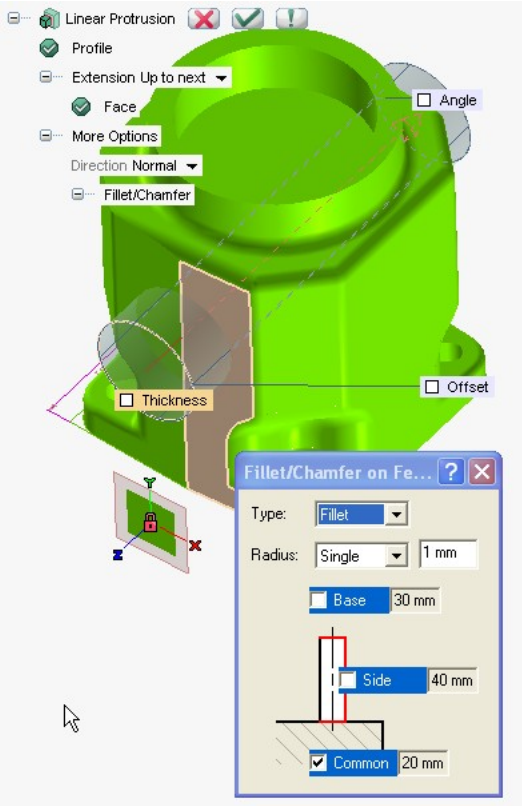


Move Work Plane onto this new Datum plane.

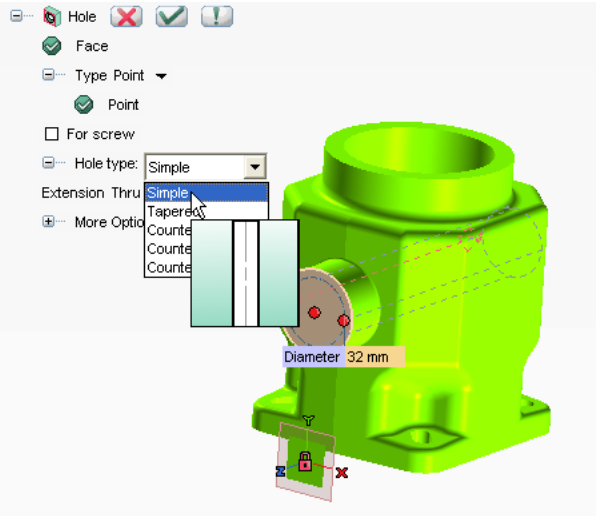
- Go to 2D Profile mode.
- Draw a **Center Circle**, dimensioned 70mm from the bottom edge of the model. Set the radius of the circle 20.
- The center point of the circle should be on the face where 2 slanted surfaces meet.



- Activate **Linear Protrusion**.
- Change Extension option to Up to next.
- Select one of the slanted surface as Face.
- Click More Options and Fillet/Chamfer.
- Change the Type to Fillet, Radius to Single and assign the value for the radius to 1 mm.
- Click OK.



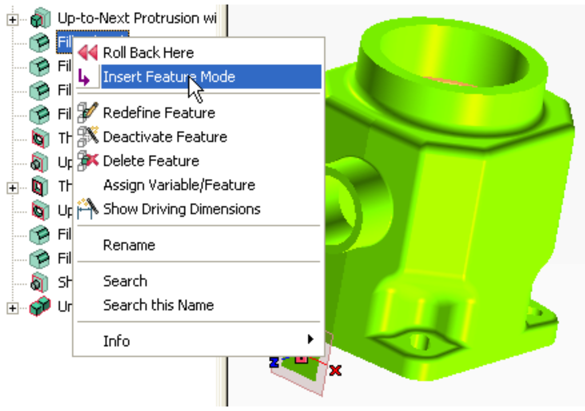
Add a **Hole** to the shaft using Thru Next option, and the Diameter32.



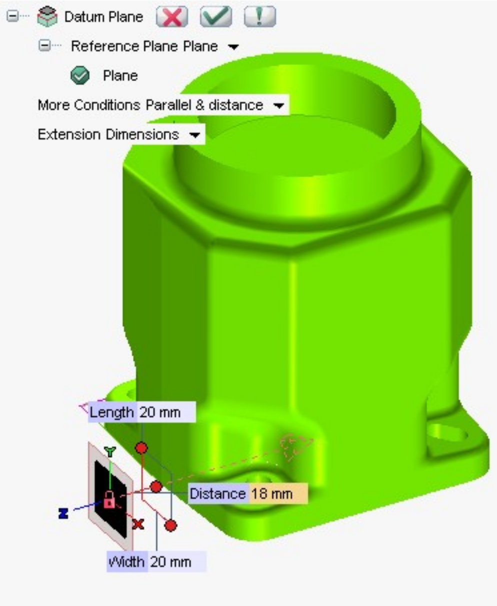
Before the protrusion and the hole features, we need to insert another rectangular protrusion on the same face. We need to insert additional features in the middle of the history tree of the model.

We can use the Insert Feature Mode to add the rectangular protrusion before these two features.

- Right click on the the last fillet and click Insert Feature Mode. (Notice the appearance of the model change. We are going back in time on this model!)

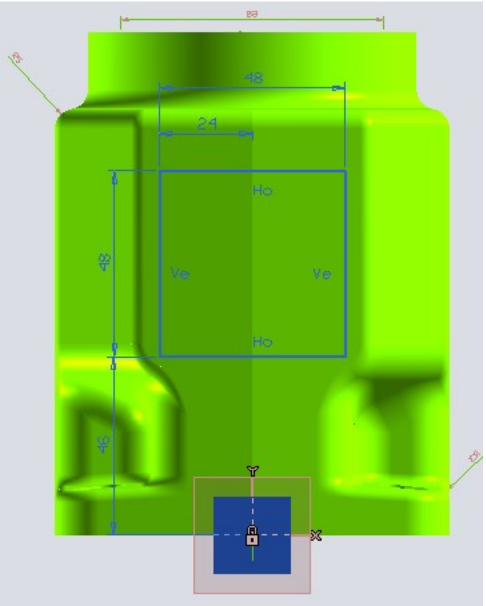


Create another **Datum Plane** which is parallel to the first one at a distance of 18mm.

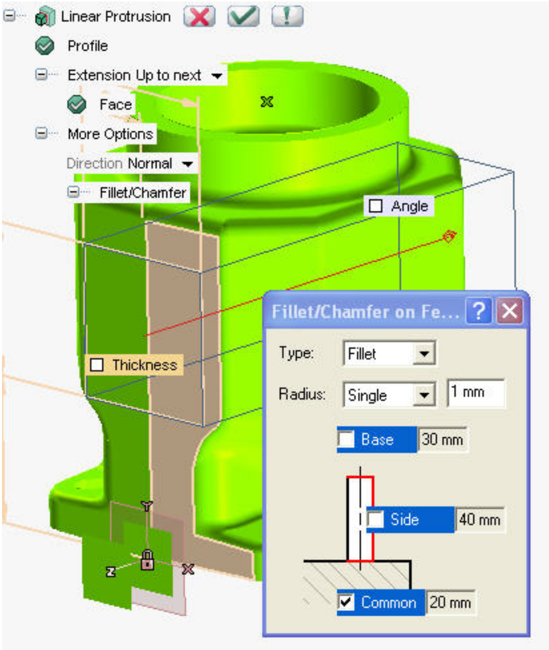


Right click on this new datum plane and click Work Plane Here.

In 2D Profile mode, draw a **Rectangle** of 48X48 as shown in the image below.



Using the Up to next option, make a **Linear Protrusion** on the part, and use More Options to place a fillet R1 at the base.



Note :

The operations may be slower during Insert Feature Mode.

Move the Insert Feature Mode to the top of the History Tree. Now the protrusion and hole will be activated.

Let's create the Protrusion and the Hole on the other side of the part using the **Mirror Solid** command.

- Start **Mirror Solid** command.
- Select the Protrusion feature. Click OK at the Smart Mode message.
- Now select Hole feature too.
- Select 3 points to Mirror.
- Click OK.

Note that the fillet is inserted on the right edge.

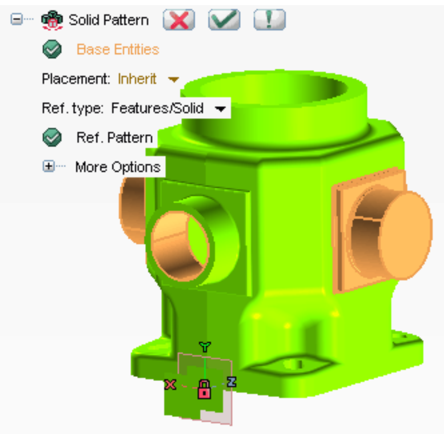
Smart Mode :

In Smart mode we can select multiple features. For example, we can mirror a set of features composed of:

- holes/slots and fillet/chamfers performed on the holes/slots, and
- protrusions/shafts and fillet/chamfers performed on the protrusions/shafts.

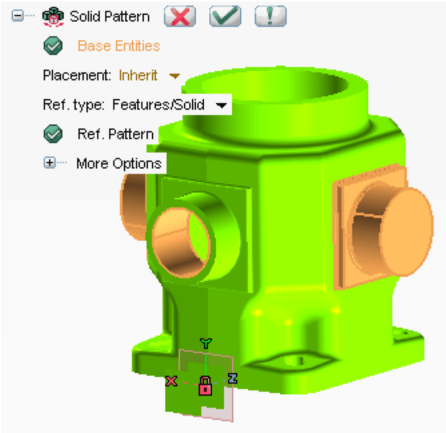
We need the same features to be on adjacent sides also.

- **Undo** the **Mirror Solid** feature.
- Start the **Pattern Solid** command.
- Select the Protrusion as the Base Entities
- Change the Type: to Angular and 1st Axis to 2 Points.
- Select top and bottom center points of the circle using **Snap to Arc Center**.
- Set 1st No. copies to 4 and 1st Angular extension to 360.
- Click OK.

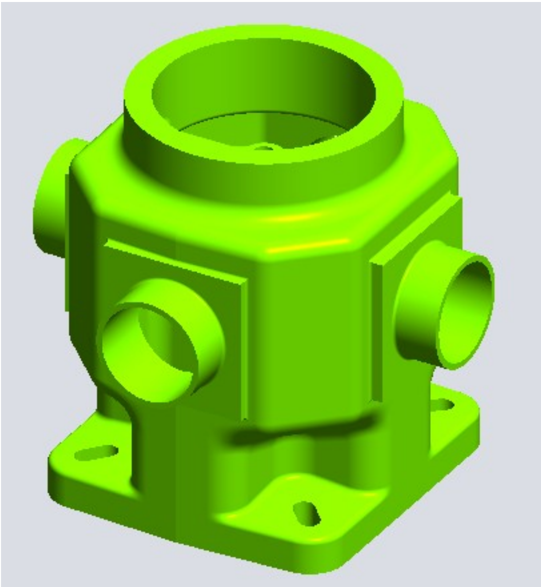


Now let's make pattern of the hole feature using the Placement: Inherit type of Pattern.

- Again start **Pattern Solid** or press Enter to activate the last used command.
- From the selection list, change the Placement: to Inherit.
- Select the one of the patterned solids as Ref. Pattern.
- Select the through hole as the Base Entities.



- Click OK.



Finished! Great job!