Solids and features

This task will familiarize you with few concepts like Linear sweep with offset,New Adaptive measure, Inherit Pattern, Fillet Collision, Chamfer Collision etc.

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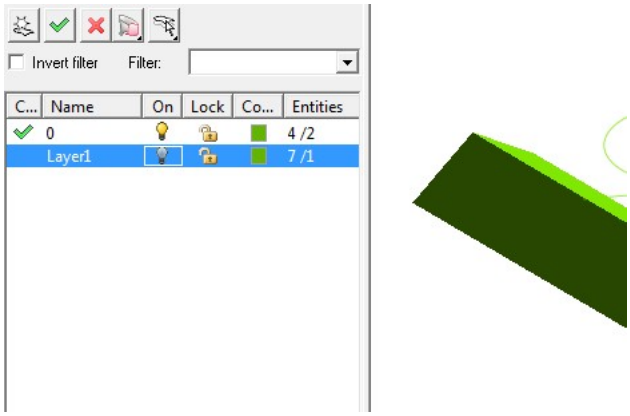
1. Step 1 :Linear Slot

Open Solid_feature.e3 from the downloaded folder if it is not already open. This file will reside in the files folder inside the corresponding task folder wherever it is downloaded.

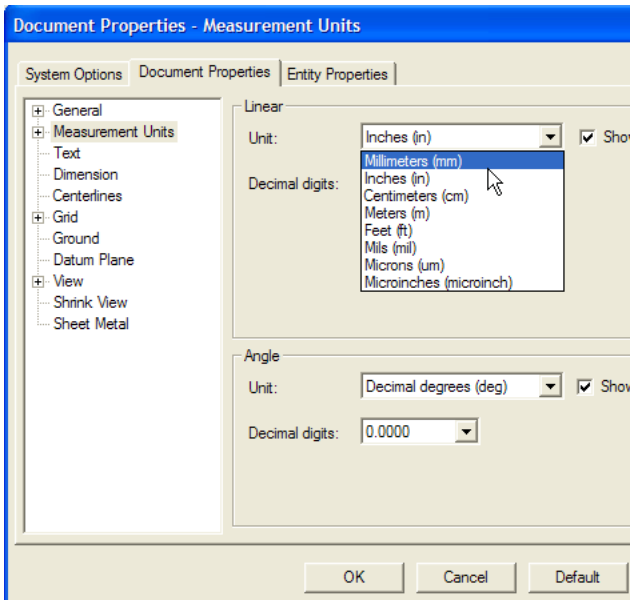
Here, the Arm and Hinge Solids of the Coupling Link are already made. The Hinge solid is hidden. The profiles and the dimensions of the Arm_Solid are put in Layer No. 1 and the Visibility is unchecked. Only the required profiles are shown for the further operations.



Now let us see how we can change the unit of measure.

- Right click on the graphics and say **Options/Properties**.
- Check the unit of the model under Document Properties tab.
- Observe that it is in Inches.

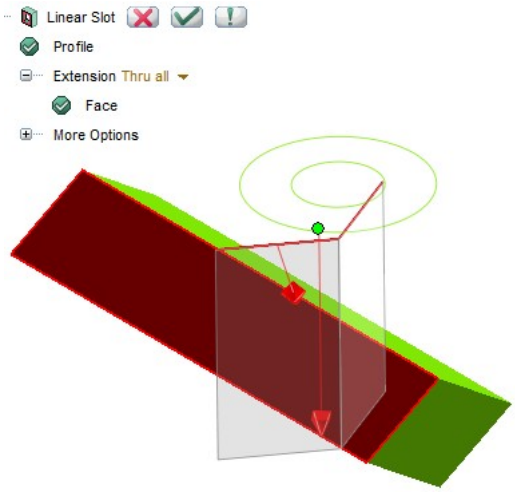
We need the unit of measure to be in millimeters.



- Change it to Millimeters.
- Hit OK.

So now the model file is set to mm and all the future changes will be recorded in millimeters.

- Start the **Linear Slot** command.
- Select the Cut_Profile from the history tree for the Profile selection.
- Select the side Face of the Arm_Solid.
- Make sure the Extension is set to Thru all in the Selection List.
- And also the arrow should be pointing away from the screen. If not, double click on the arrow to reverse the direction



- OK to cut the slot.

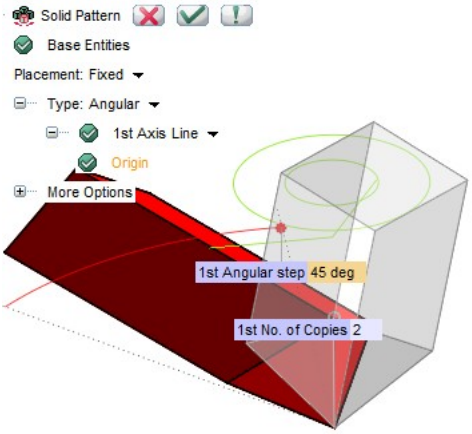
2. Step 2 :Pattern Arm Solid and Fillet collision.

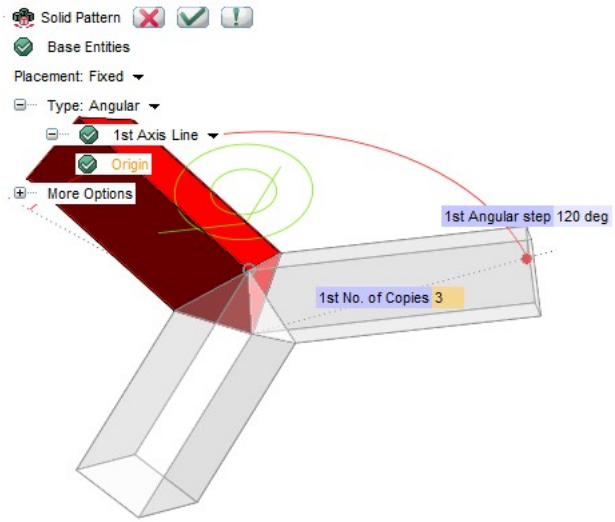
We need to create three Arms using the **Pattern Solid** command.

- Click on **Fit View**.
- Start the **Pattern Solid** command.
- Set Type as **`Angular`**.
- Set Placement as **`Fixed`**.
- Select the Arm_Solid as the Base Entities in the Selection List.

Use the Edge of the solid as the axis of rotation.

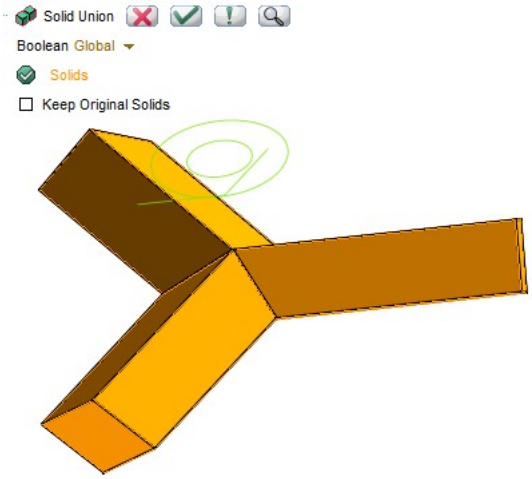
- Set the 1^Axis Line in the Selection List
- Click on the edge of the Arm as indicated below.





- Set the 1st No. copies to 3.
- Set the1st Angular step to 360/3.
- Hit OK.

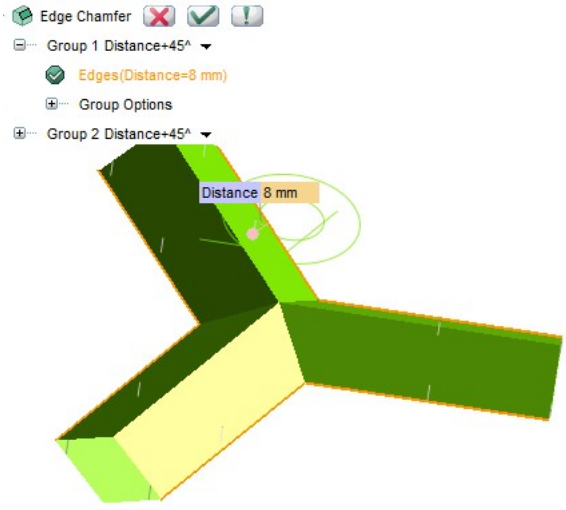
- Now **Solid Union** all the three Arm_Solids.
- Start the **Solid Union** command.
- Click on each of the three solids.



Hit OK to complete the command.

Now we'll use **Edge Chamfer** to chamfer the edges of the Arms.

- Select the **Edge Chamfer** command.
- Select the six edges for the New Group.
- Make sure **`Group 1`** is set at **`Distance+45^`** in the Selection List
- Set the Distance as 8 mm.

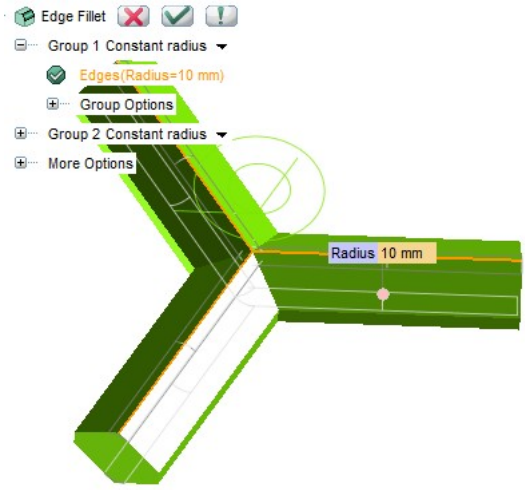


Hit OK.

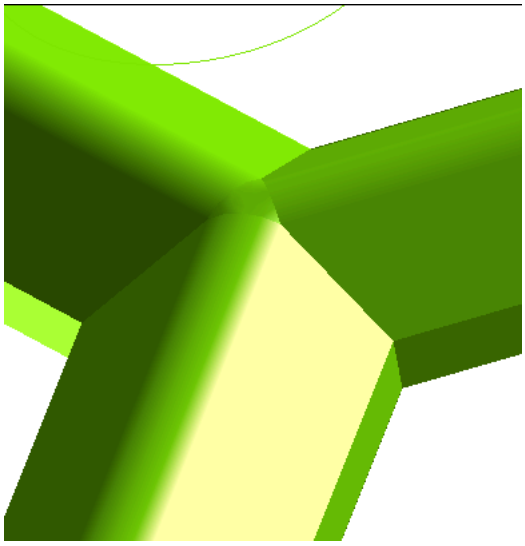
Now let’s add the Fillets to the top and bottom edges of the Arms.

- Start **Edge Fillet**.

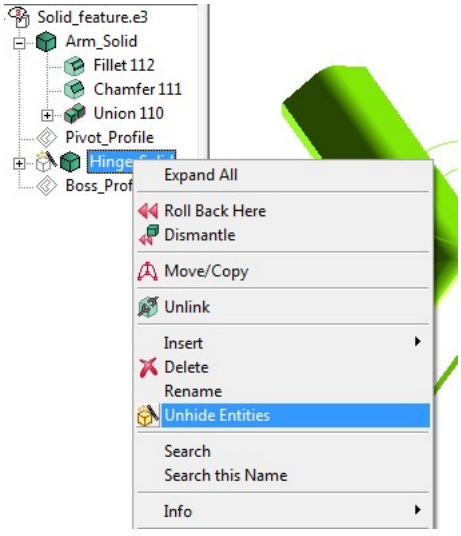
- Select the six edges of the arm feature.



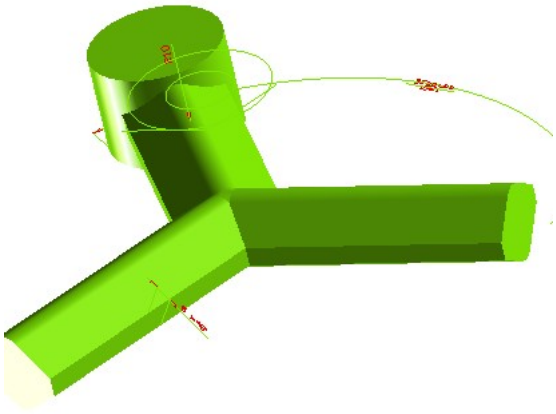
- Set the radius as 10 mm.
- Hit OK.



Now **Unhide Entities** the Hinge solid and take a look.



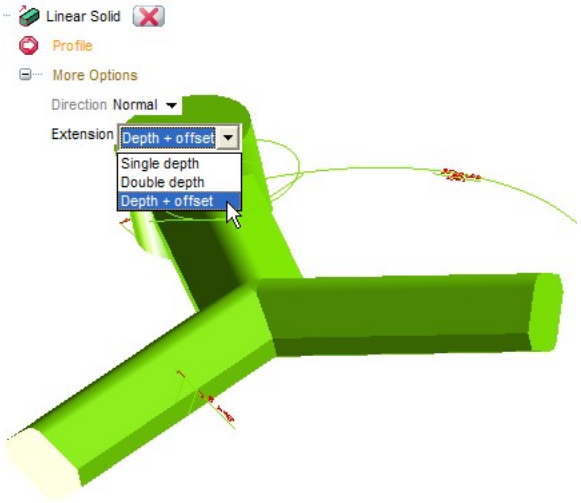
- Right click on **`Hinge_Solid`** in the History Tree.
- Hit **Unhide Entities**



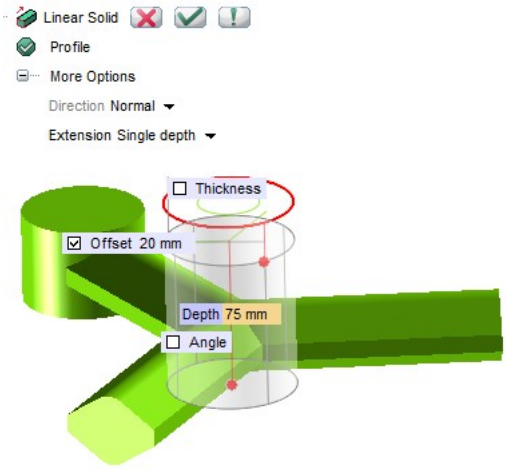
3. Step 3 :Linear sweep with offset and Adaptive measure.

Let us create Pivot solid using Depth+Offset option in Linear solid.

- Activate **Linear Solid**.
- Change the Extension to Depth+offset.
- Select the Pivot_Profile.



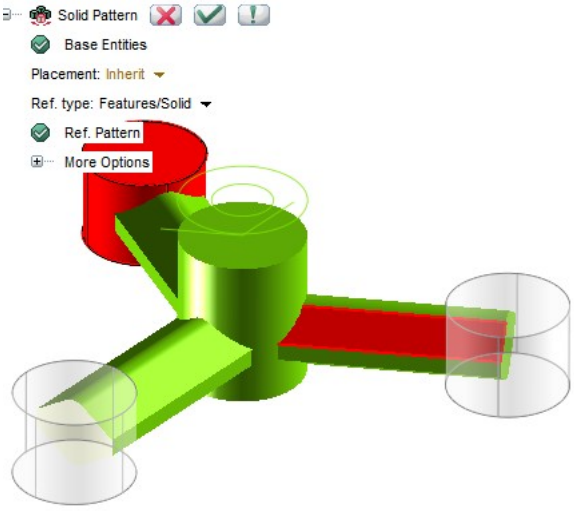
- Set the Depth to 75 mm.
- And Offset distance to 20 mm.



- Say OK.

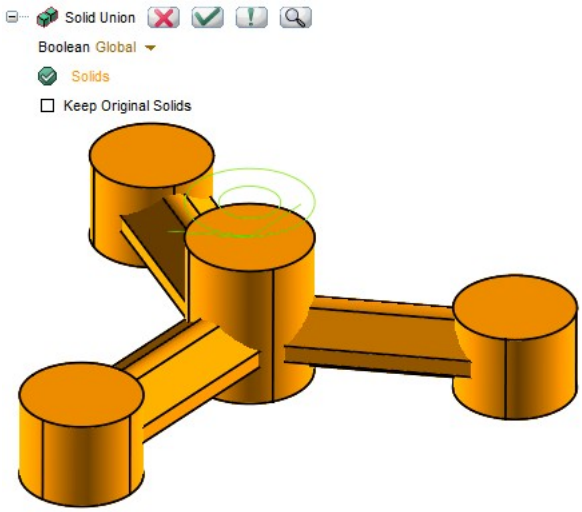
Now let us create a pattern. Here the easy method of patterning the Hinge is using the previous pattern feature by the way of **`Inherit`** option in **`Pattern`**.

- Start **Pattern Solid** command.
- Set the **`Placement`** to **`Inherit`** in the Selection List.
- Set Ref.type to **`Features/Solid`**.
- Select the Hinge as the Base Entity.
- Select the Arm as the reference for pattern.



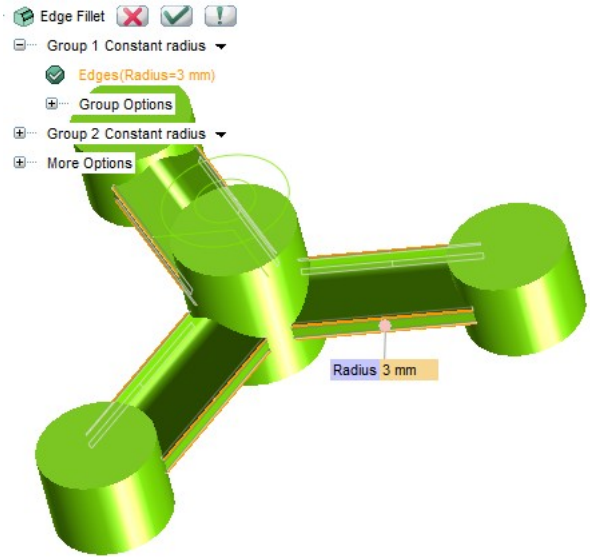
- Hit OK to complete the command

Solid Union all the Solids as explained earlier.



Now, we should add fillet to all the chamfer edges as well as the other edges of the three Arms.

- Start **Edge Fillet** command.
- Select 12 chamfer edges of the Arm_Solid

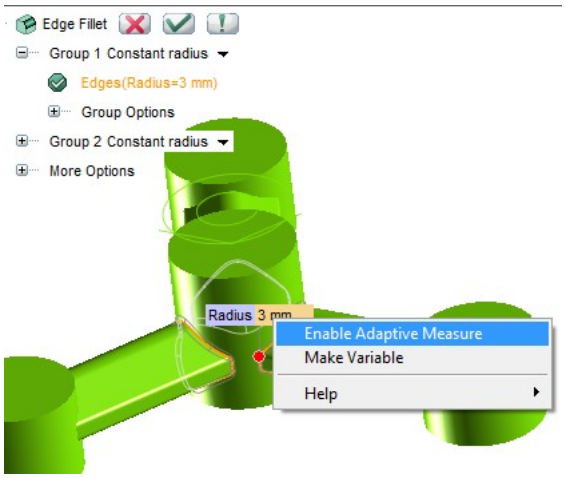


- Set the radius to 3 mm.
- Hit OK

Let us see the new feature Adaptive measure now. A new Adaptive Measure mechanism has been developed, that enables you to link the values of dimensional parameters of solids and features to those of existing entities in the model.

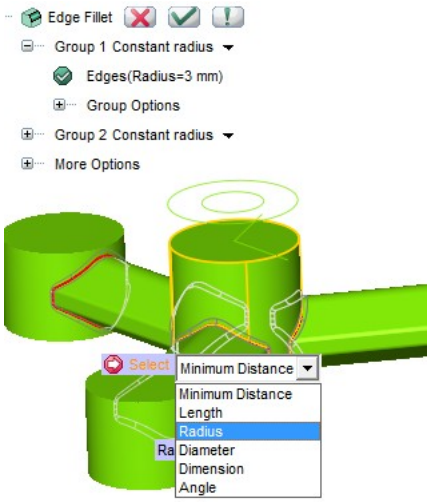
Again activate the **Edge Fillet** command and select the edges as shown below.

- Right click on the MiniDialog box and say "Enable Adaptive Measure ".

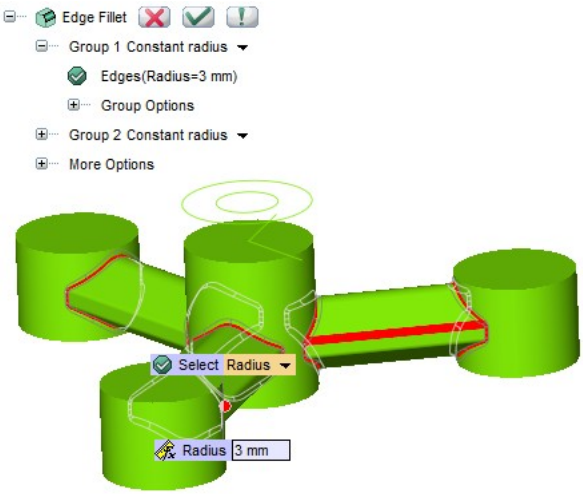


When we select it, a drop-down list with several available choices is displayed

- Select "Radius " from the pull down list.

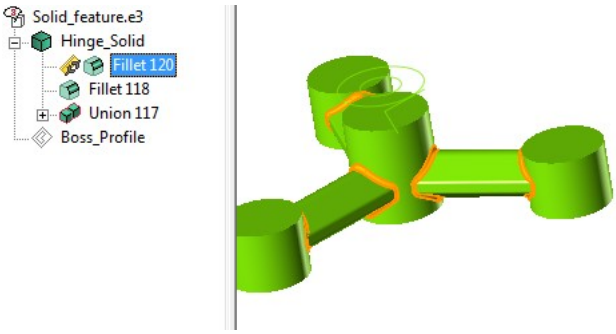


- Select the Fillet feature created in the last step.



- Hit OK.

Please observe the fillet in the History tree.

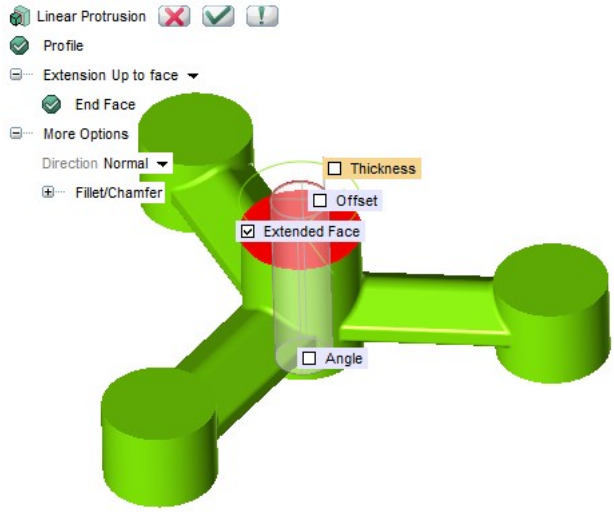


4. Step 4 : Linear protrusion and Simple hole with double thread.

In this step, we shall add a protrusion onto the existing solid.

- Click on the **Linear Protrusion** button on the Solids toolbar.
- Set the Extension Up to face.
- Select the Boss_Profile as Profile .

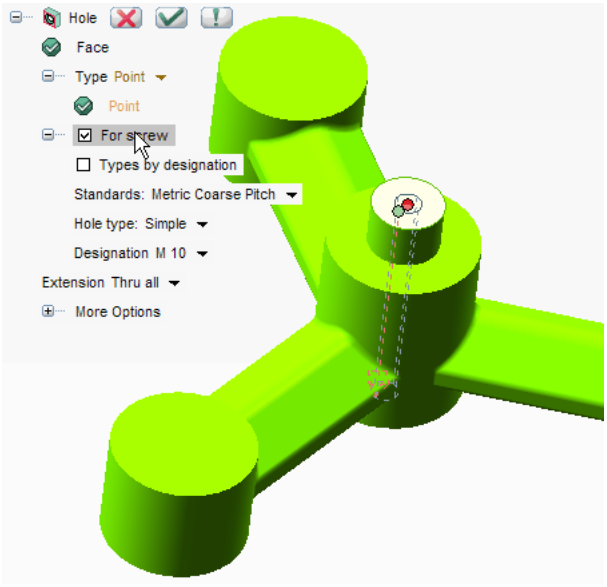
- Select the top face of the Pivot_Solid as End Face.



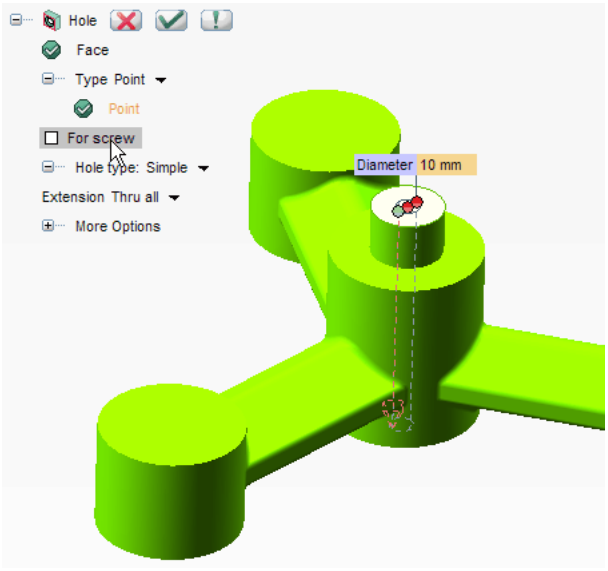
Click OK to end the command.

- Now start **Hole** command.
- Set the Extension to Thru all.
- Select the top face of the Boss as the Face .
- Snap to the center point of the Boss as the Center point for the hole.
- Check the screw option
- Under the screw dropdown add Standards: Metric Coarse Pitch, Hole Type: Simple and lastly Designation:

M10



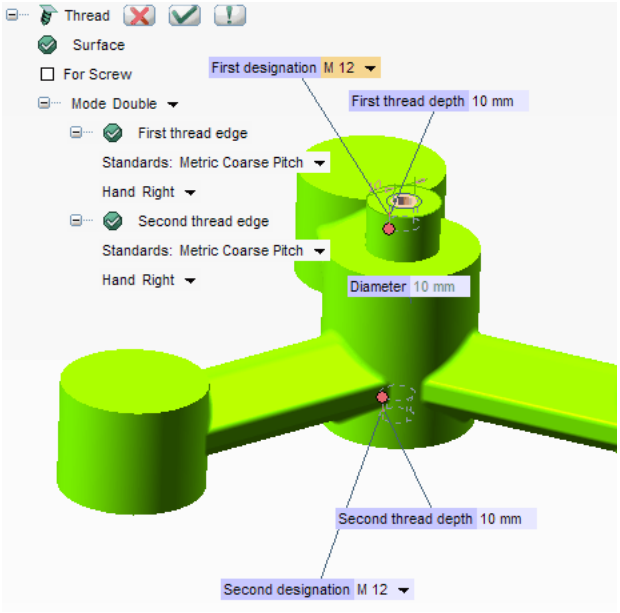
Uncheck the screw option now



Click OK to add a hole of dia 10 to the feature.

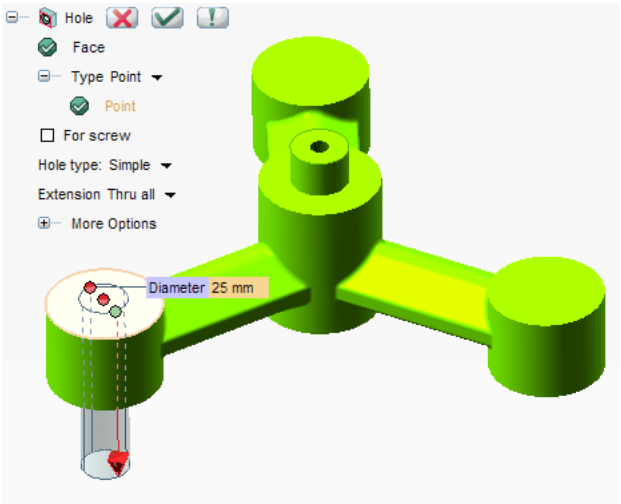
Now start **Insert**➪**Solid**➪**Hole/Shaft**➪**Thread**command.

- Select the hole surface
- Select mode: Double
- Select upper hole as 1st thread edge & lower hole as 2nd thread edge, and lastly add the values as shown for 1st & 2nd designation & thread depth respectively.



Similarly put a hole of diameter 25 mm to the Hinge.

- Start **Hole** command.
- Set the Extension to Thru all.
- Do NOT check for the screw box.
- Select the top face of the Hinge as the Face to be selected.
- Snap to the center point of the hinge as the Point of reference.
- Enter the diameter of the hole as 25 mm.

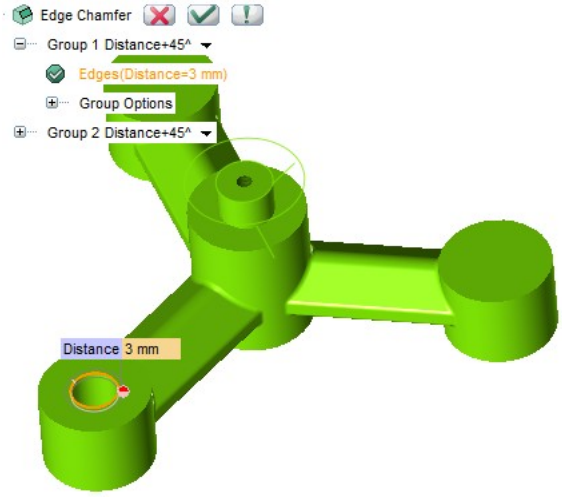


- Hit OK to insert the hole.

Now add chamfer to it as explained earlier with **Edge Chamfer** command.

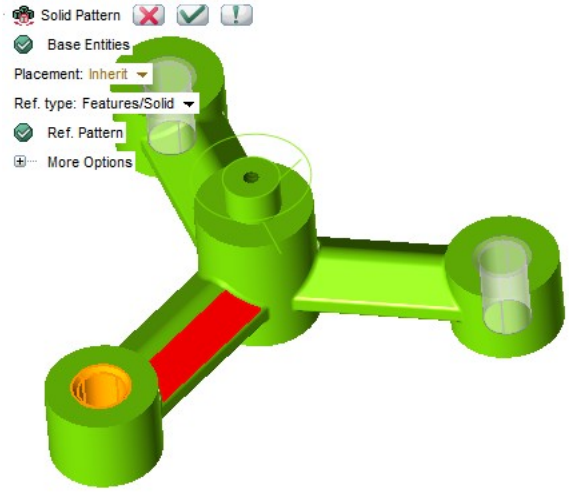
- Start **Edge Chamfer** command.

- Select the edge of the hole for the New Group.
- Make sure Group 1 is set at Distance+45^ in the Selection List.
- Set the Distance3 mm.
- Hit OK.

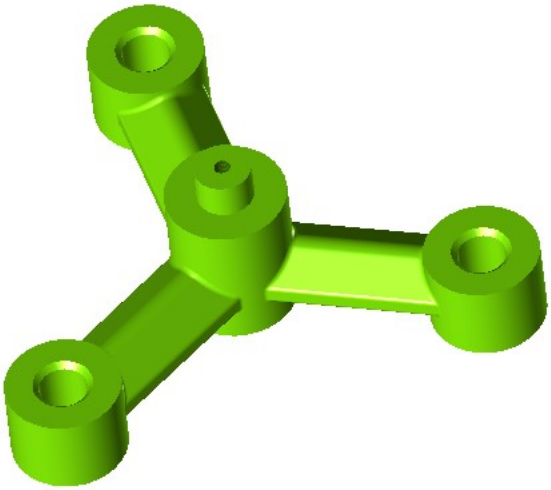


Finally, **Pattern Solid** these two features (the Hole and Chamfer) using Inherit pattern by selecting either the Arm feature or the Pivot feature for the pattern.

- Start **Pattern Solid** command .
- Pick the Hole and Chamfer for Base entities.
- Set Placement:Inherit.
- Set Ref. type:Features/Solid and pick the Hinge as reference.



Click OK to finish.



Nicely done!! Now get rid of all the dimensions and profiles by using the **Hide Entities** command.