GUI Concepts

In this task, you'll create this rod cap using 2D sketching tools, then sweep a solid from the 2D geometry and add some fillets.

Table of Contents

1. Step 1: Get Ready to Sketch!

As you go through the steps, keep these things in mind:

- If you get into trouble, you can always click **Undo** to go back to the previous step.
- To exit a command, press the Esc key or hit the Cancel button in the Selection List.

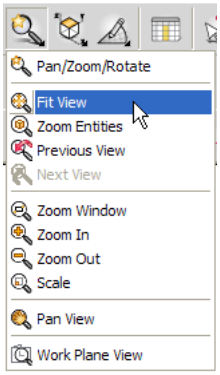
Also, don't forget to set the focus back to the ThinkDesign window after scrolling or clicking in this browser window. You can do so:

- By clicking on the title bar of the ThinkDesign window OR....
- By clicking on a blank part of the ThinkDesign window.

Don't click in the Graphics area, especially when a command is active as you might end up picking a point!

Let's start by closing the History Tree and fitting the view to the window.

- Place the cursor on the vertical bar separating the History Tree from the Graphics Area. You'll see the cursor change to ⇹.
- Drag the bar to the left to close the History Tree.
- Click **Fit View** or hit the F key to fit the model to the current view.



In the next step, we'll start drawing

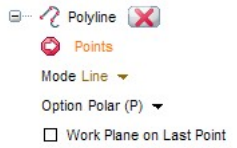
2. Step 2: Create the shape

In this step, we'll use the existing construction lines to sketch the main shape of the rod cap.

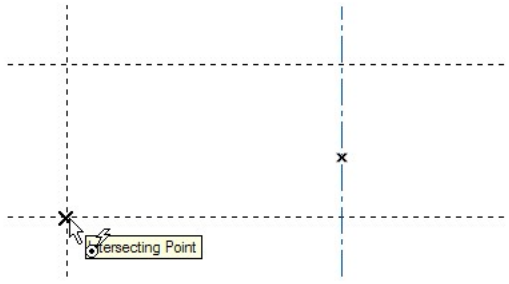
Start the **Polyline** command to draw the profile of the rod cap.

- Click the **Polyline** button on the Drafting toolbar OR..
- Select **Insert**➪**Drafting**➪**Polyline**from the pull down menu.

The Selection List -- located in the top left corner of the Graphics Area -- displays all the options needed to complete the desired command. The default options for the **Polyline** command is Mode Line and Option Polar (P). These are the options we'll use to start.

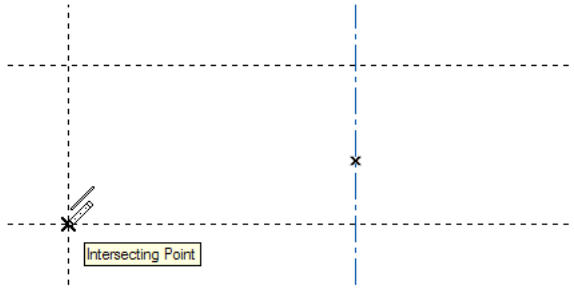


ThinkDesign displays Points, an arrow with a red background icon and orange text in the Selection List when it is requesting input for a particular feature or command. It also displays command prompt in the lower right hand corner of the main application window (see image below). It's important to keep an eye on the prompt as to know what's happening. The **Polyline** command is prompting you to Enter the line start point in the command prompt and showing a Points in the Selection list.

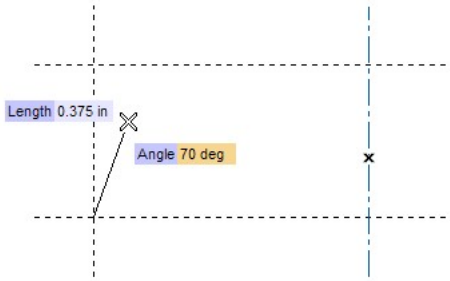


- Snap to the lower left intersection of the two dashed construction lines

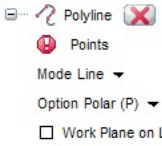
You'll know you have the exact point when you see the X and the Intersecting Point tool tip.

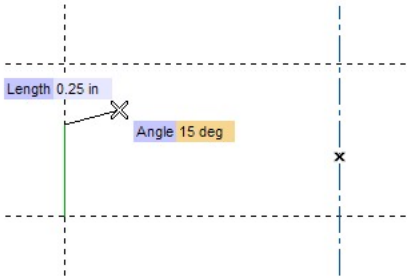


The first line is 0.375 long at an angle of 90° -- we'll use the Mini Dialogs to input those values. Since we have two Mini Dialogs (Length and Angle) we need to determine which one is active to receive keyboard input. In this case, we know it is the Length because the input area is orange.



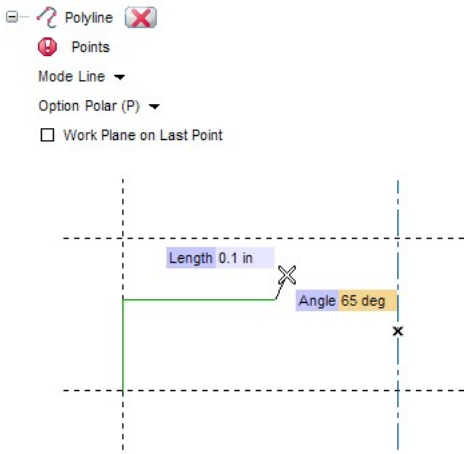
- Type 0.375 and hit the Tab key to set the length. Length0.375 in





- Hit the Tab key again to move the cursor to the Angle Minidialog
- Type 90.
- Hit Tab or to set the angle value. Angle90 deg

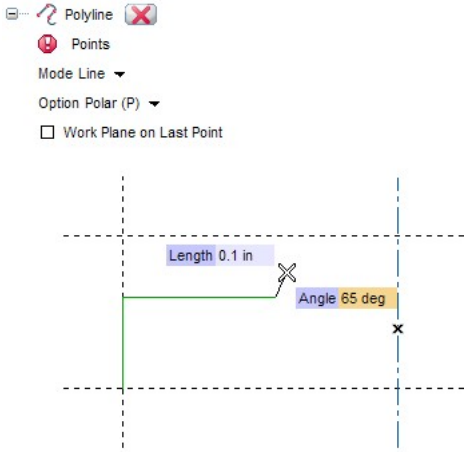
Remember that when you type values into Mini Dialogs that aren't activated (i.e. cursor isn't flashing), use the number keys across the top of the keyboard. If you want to use the number pad instead, make sure Num Lock is on. Otherwise you'll activate the dynamic viewing commands we learnt in the last task.



With both values specified, the line is drawn and you're on to the next one, a horizontal line with a length of 0.625.

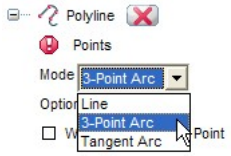
- With the Length box active, type 0.625 to set the distance of this line: Length0.625 in
- Hit the Tab key twice to switch to the other Mini Dialog -- the Angle box
- Type 0 to set the angle of this line: Horizontal0 deg

- Hit Enter to create the line.



For the next segment we'll use a 3-Point Arc.

- Choose 3-Point Arc option from the Mode drop down list.



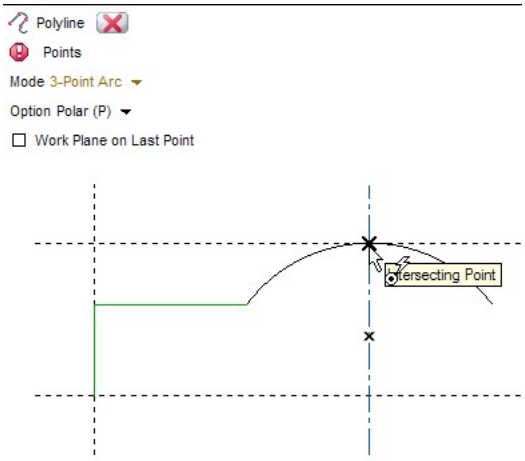
The first point we'll enter is the endpoint.

- Set the length to 1. Length1

- Set the angle to 0. Horizontal0 deg

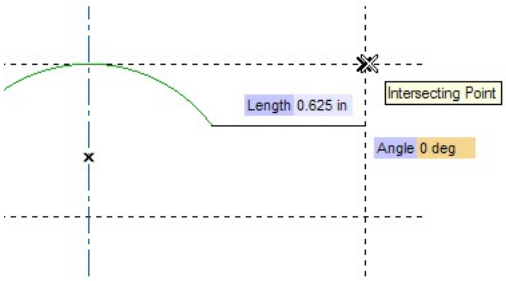
The second point we need is the middle point.

- Snap to the intersection of the centerline and the upper dashed construction line. Remember to look for the 'X' for the Intersecting Point.



The next segment extends horizontally to the right vertical construction line. This time, we'll combine Mini Dialog input with a snap to get the point we need.

- Set the angle to 0 (horizontal) to lock the line orientation. Angle0 deg
- Snap to the point where the upper horizontal construction line and the right vertical construction line intersect.

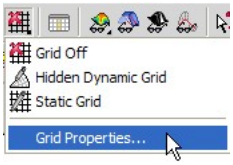


Continue down through the vertical line.

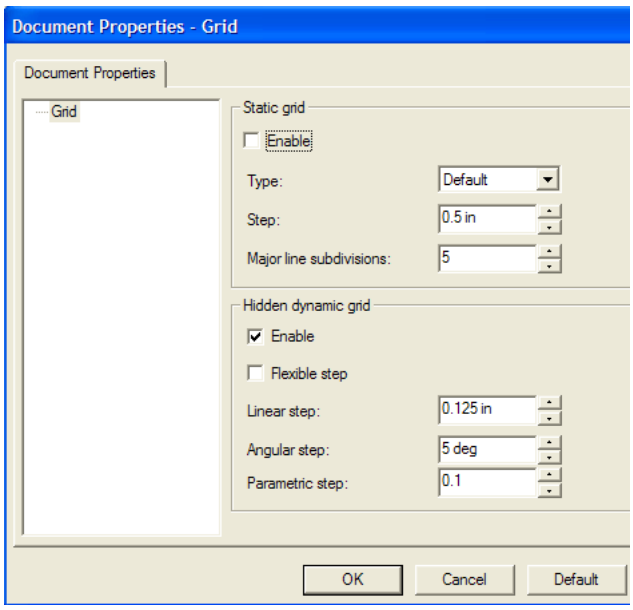
- Snap to the intersection of the lower horizontal construction line and the right vertical construction line.

We've typed in a lot of values up to this point; now let's try another input method: Set up the Dynamic Grid so

the cursor will move in preset increments. The Grid Properties can be set from the Standard toolbar - Grid pull down menu as in the image below or from the menu **View➪Grid➪Properties**

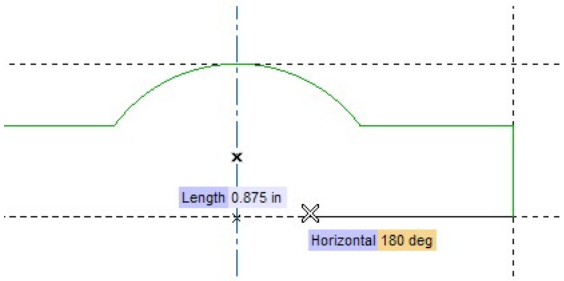


- Under Hidden Dynamic Grid.
- Make sure ☑ Enable is checked.
- Uncheck Enable flexible step for now, until you get used to the dynamic grid.
- Set the Linear step: to 0.125.
- Close with OK.



Continue to the left with a horizontal line, angle 180 and length 0.875.

- Drag the cursor to the left until the length of the line is 0.875. Length0.875
- Drag the cursor up or down until the angle is 180. Horizontal180
- Pick the point where both values are correct.

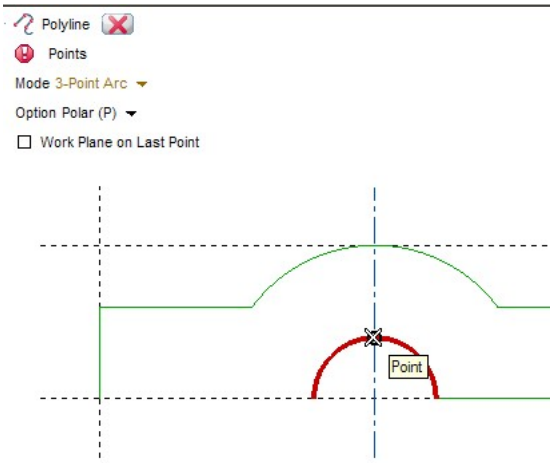


Time for another 3-Point Arc. Remember that we need the endpoint first, then the midpoint. For the endpoint,

you can either type or drag the Mini Dialog values.

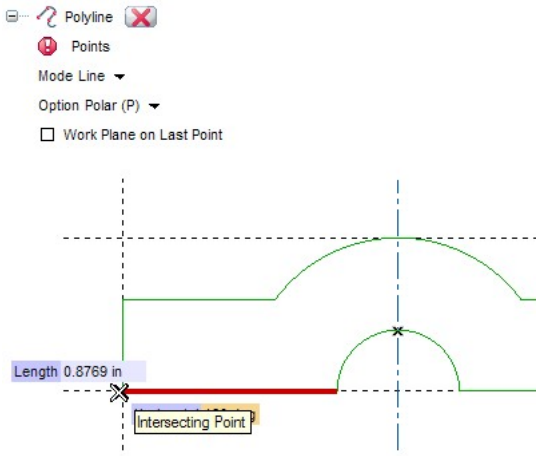
- Select the 3-Point Arc option from the Mode drop-down list.
- Set the length to .5. Length0.5
- Set the angle to 180. Horizontal180
- Snap to the point (the little x) on the centerline.

You'll know you have the right point when a small circle appears around it, and the point tool tip is visible.

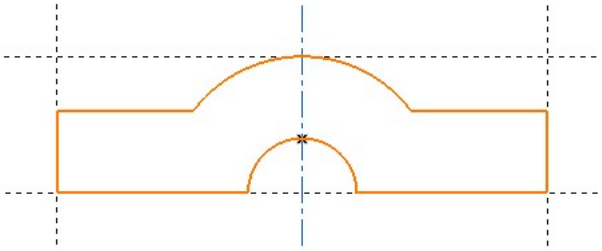


Now let's close the shape.

- Move the cursor near the starting point.
- Pick the point at which the preview line turns red and the starting point is highlighted.



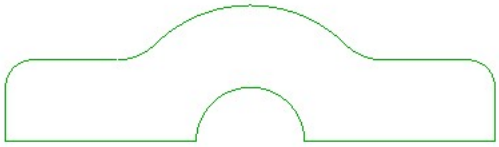
Closing the shape automatically ends the **Polyline** command, selecting all of the segments.



We've covered lot of ground here. Let's move on to the next step and add some 2D fillets to the shape we've just created.

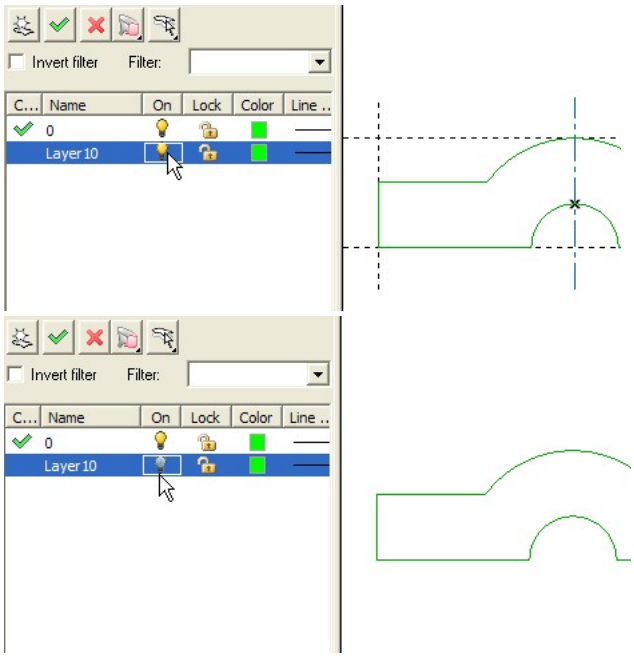
3. Step 3: Refine the Shape

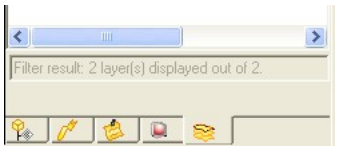
In this step, we'll add some 2D fillets to refine the shape of the rod cap as shown below.



We don't need the construction geometry any more, so let's turn off the **Output Layers**.

- Click **Format**→**Layers**from the pull down menu.
- In the Show list, select Relevant Layers.
- Uncheck the Vis box for Layer 10 and add a Description if you want.
- Click OK.



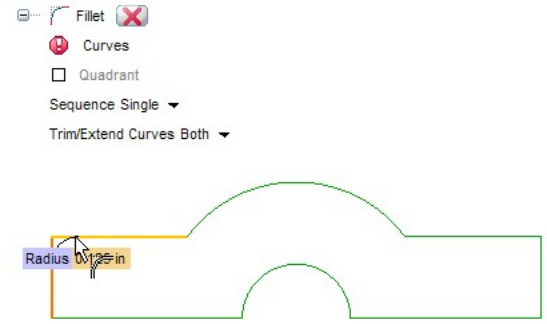


With the construction lines out of the way, let's fillet some corners.

- Click **Insert Fillet** on the Drafting toolbar OR pick **Insert**➪**Drafting**➪**Fillet**from the pull down menu.
- Type .125 and hit Tab or Enter to set the fillet radius. Radius0.125

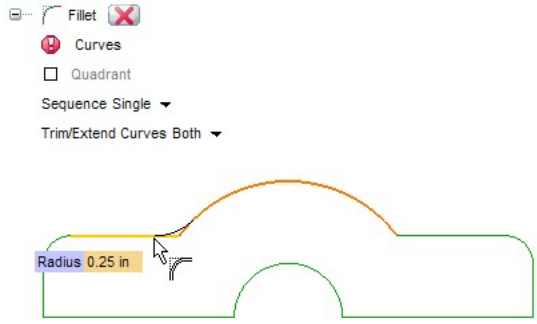
The Mini Dialog in the **Insert Fillet** command works just like Mini Dialogs in the **Polyline** command.

- Select the left vertical line near the top.
- Select the upper left horizontal line near the left side.



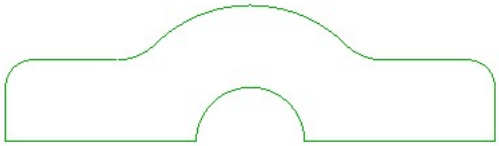
Repeat the procedure for the upper right corner.

- Pick the right vertical line, near the top.
- Pick the upper right horizontal line, near the right end.



Now we'll add 0.25 fillets between the upper horizontal lines and the upper arc.

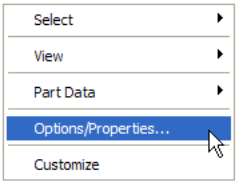
- Change the radius value to 0.25. Radius0.25
- Pick the upper arc, near the left end.
- Pick the upper left horizontal line, near the right end.
- Repeat for the right side of the arc.



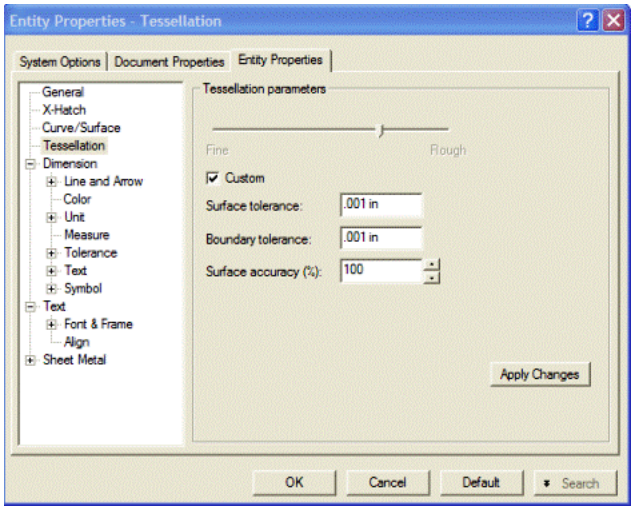
- Hit Esc to end the command.

Adjusting the graphics. Now we want to make our model look a little better. If you've noticed, the curves in our sketch don't look very smooth. We can adjust the graphics to smooth out our curves.

- Make sure that nothing is selected and right click in the background (Graphics Area) and in the context menu select **Options/Properties**



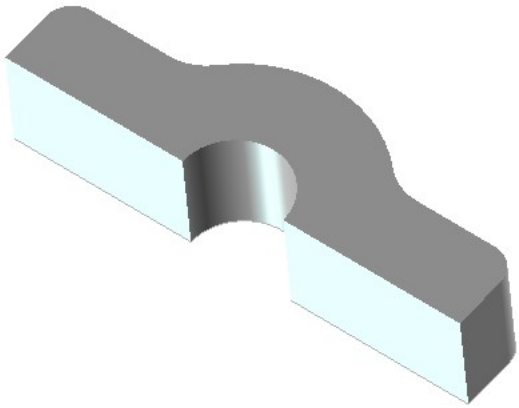
In this dialog window, select Document Properties tab and select Tessellation branch. Make the Boundary tolerance and Surface tolerance equal to .001 then click OK.



Looking good! In the next step, we'll use these curves to create a solid.

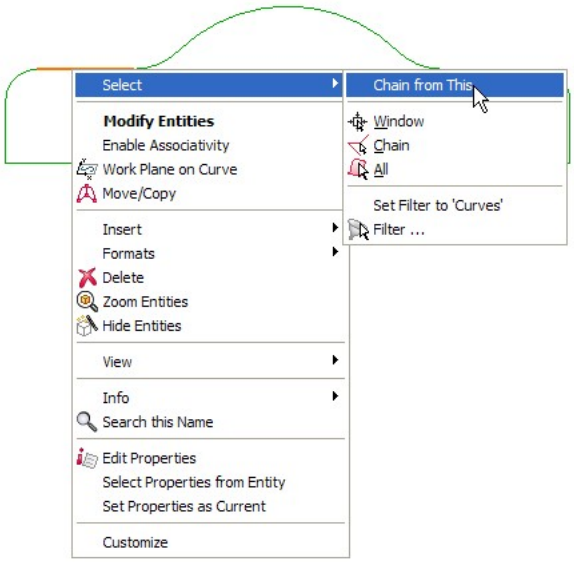
4. Step 4: Sweep the Solid

It's time to make a solid from our new shape.



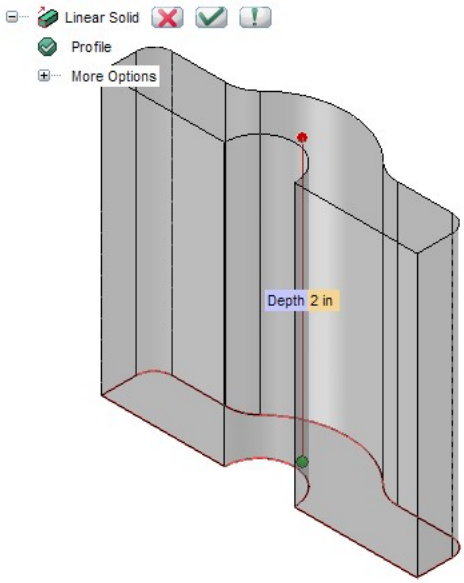
First, we need to reselect all the curves that make up the profile of the rod cap. We'll use the **Select Chain** selection tool.

- Click **Edit**➪**Select**➪**Chain**from the pull down menu and pick a curve OR right click on one of the curves and hit Select > Chain from This in the context menu.



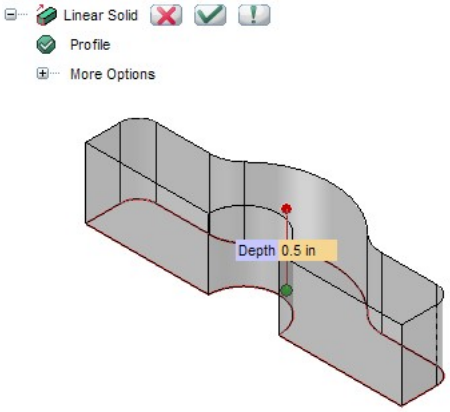
With the curves selected, we'll sweep a solid with **Linear Solid** command.

- Click the **Linear Solid** button in the drop down in the Solids toolbar OR select **Insert**➪**Solid**➪**Sweep**➪**Linear Solid**from the pull down menu.



- Grab the Handle (the red dot) and pull it to dynamically adjust the depth of the solid.

The green handle (green dot) decides whether or not it will be symmetric when it is double clicked.

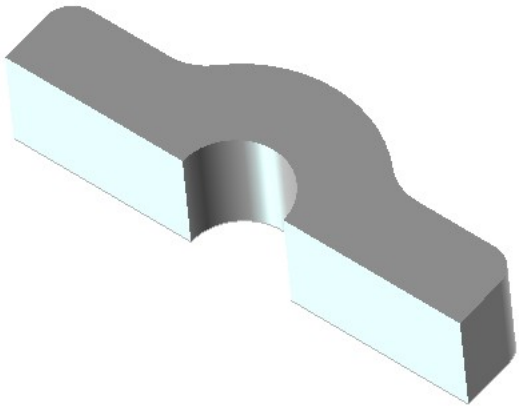


Set the depth to 0.5.

- Click on the input area of the Mini Dialog and type 0.5 OR drag the handle. Depth0.5.
- Click the OK button in the Selection List OR right click and pick OK from the context menu.

Either method confirms the settings and creates the solid.

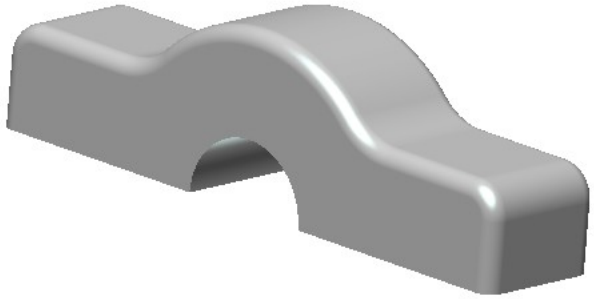
- Set the shading to **Shaded View and Boundaries** and have a look.



We're almost done. In the last step, we'll add some 3D fillets to finish the part.

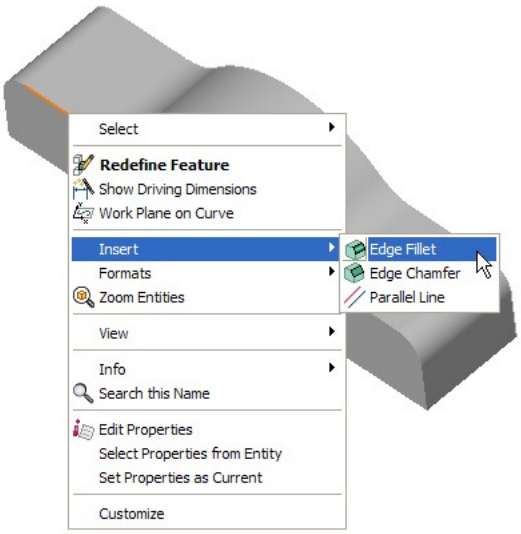
5. Step 5: Adding 3D Fillets

In this step, we'll finish the rod cap by adding fillets to two edges with **Fillet Edges** tool.



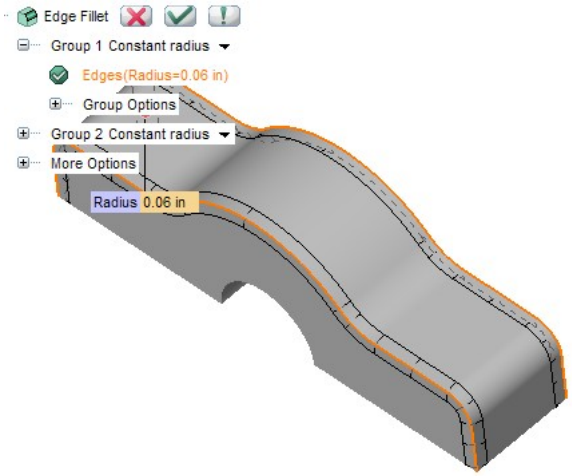
Let's add Fillet!

- Click and hold the right button of the mouse to rotate the model so that you can see its far face.
- Right click on the top front edge and select Insert Fillet>Edges from the context menu.

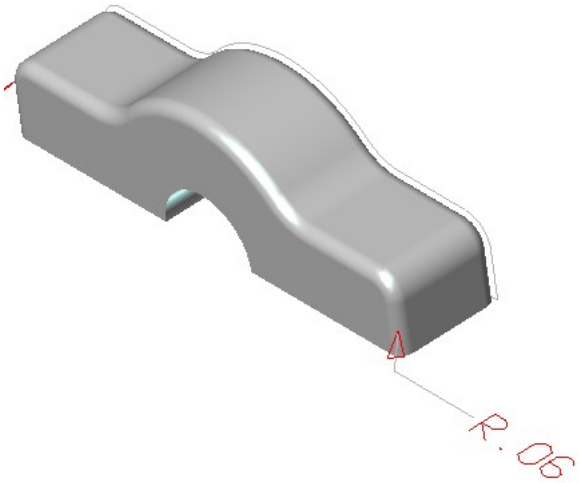


- Type 0.06 and hit Tab to set the fillet radius Radius0.06.

- Pick the edge on the backside of the model to fillet that edge as well.

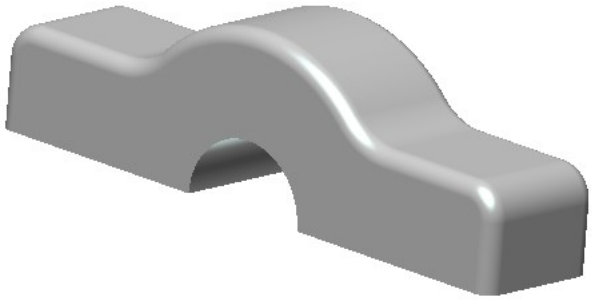


- Right click and pick OK from the context menu OR click the OK button in the Selection List to complete the fillets.



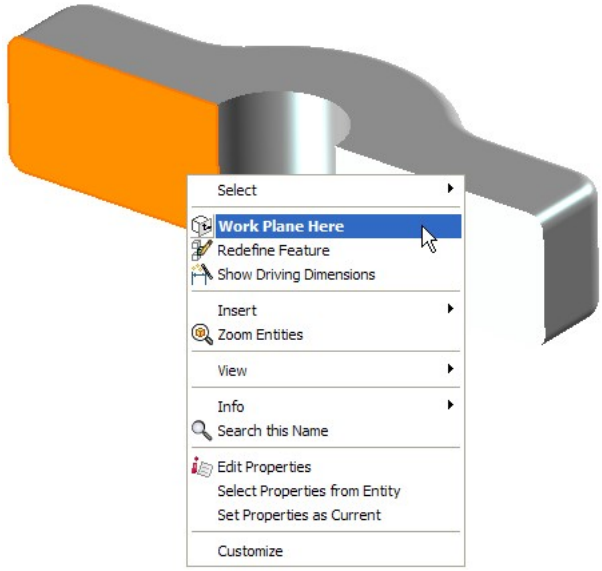
To finish the job, click on the **Hide Entities** button in the Attributes toolbar to hide all the entities other than the solid.

- OR choose **View**➪**Hide Entities**from the pull down menu and click on the entities other than the solid.

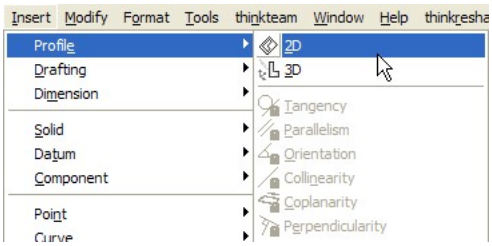


6. Step 6: Creation of Base solids.

Set the work plane on lower face of the solid by right clicking on the face and click on Work Plane Here.

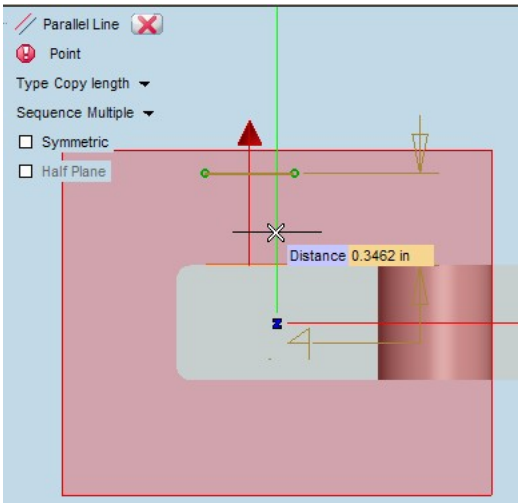


Go to Profile mode.

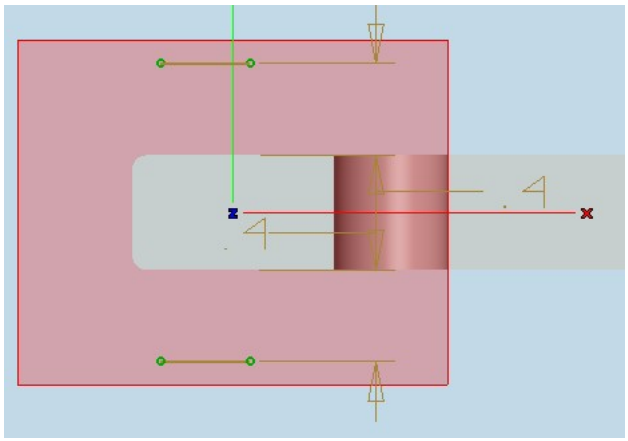


Select the **Parallel Line** command.

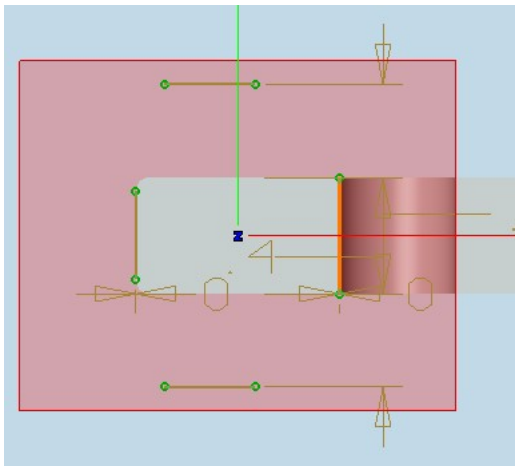
- Select the edge as shown.
- Keep the option Type to Copy length.
- Set the offset distance to 0.4.



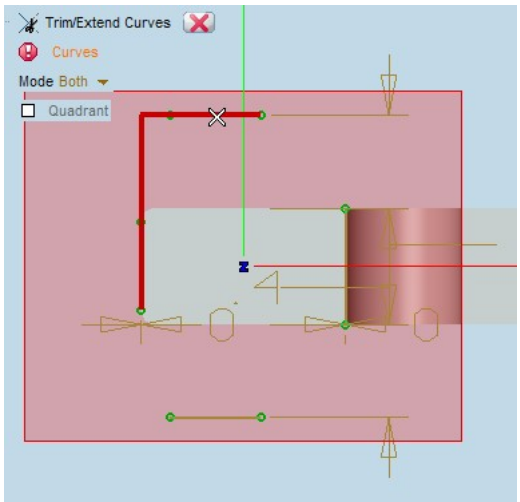
Similarly draw another parallel line by selecting the bottom edge of same offset distance 0.4.



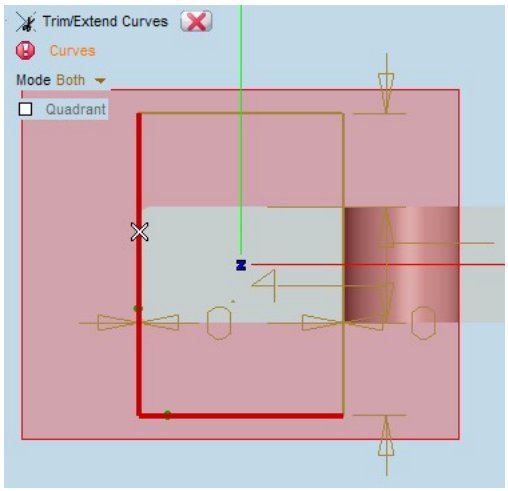
Using the same command to create other 2 parallel lines selecting the vertical edges with zero offset value.



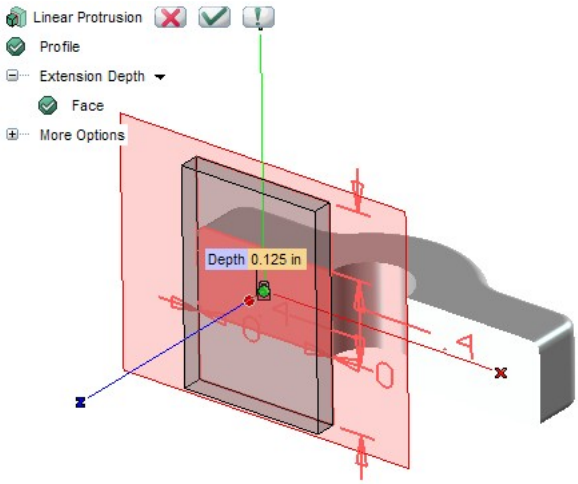
- Activate command **Trim/Extend Curves**.
- Select the left vertical and upper horizontal lines to extend.



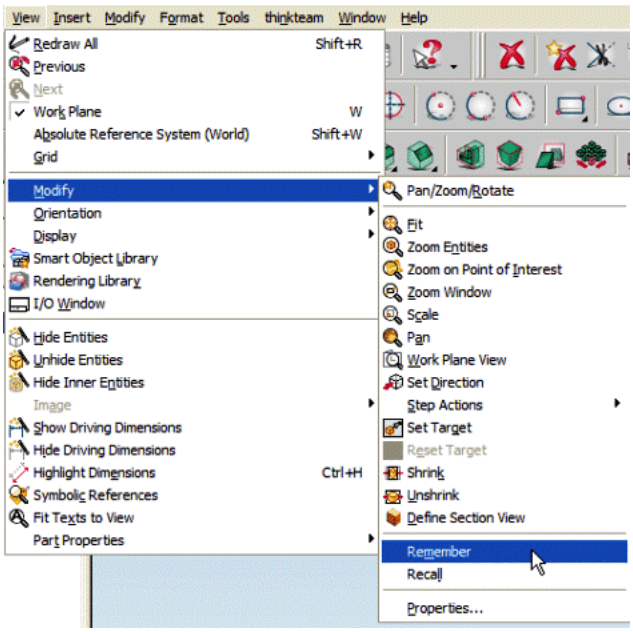
- Similarly extend four sides and make a rectangle as shown.



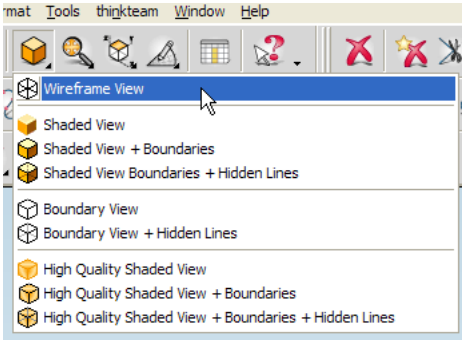
- Activate **Linear Protrusion** command.
- Select extension option as Depth.
- Set the Depth value to 0.125.
- Click OK.



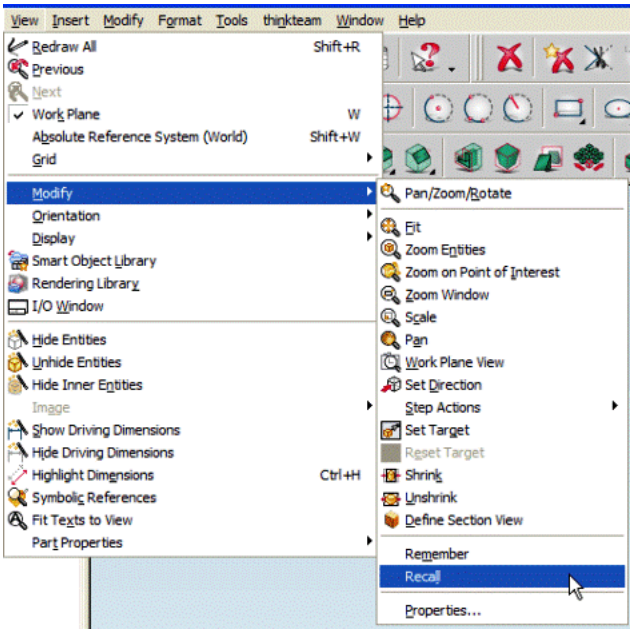
Let us store this view orientation in RAM memory so that you can recall it at a later stage. For this choose the Remember command from the view -> modify menu bar item.



To check whether we could store the View using the new option, rotate the solid and switch to **Wireframe View**.



Now select Recall to get the view orientation that was saved using the Remember command.



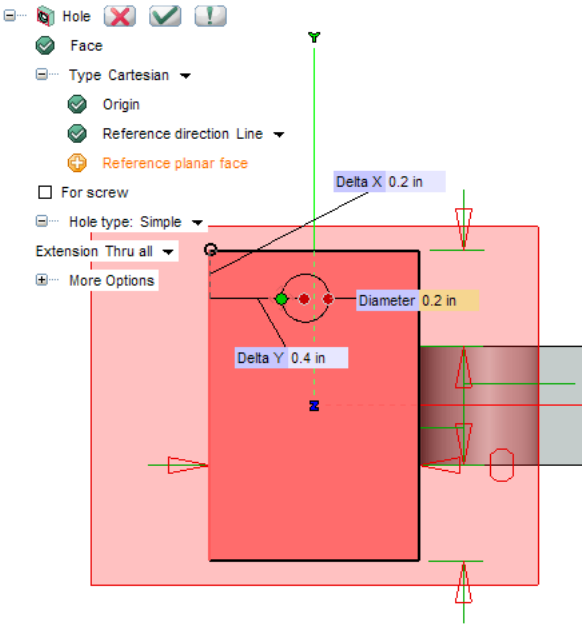
After having successfully used the new commands, let's create mirror solid and pattern of the simple holes in the last step to finish this task.

7. Step 7 : Simple holes and pattern feature.

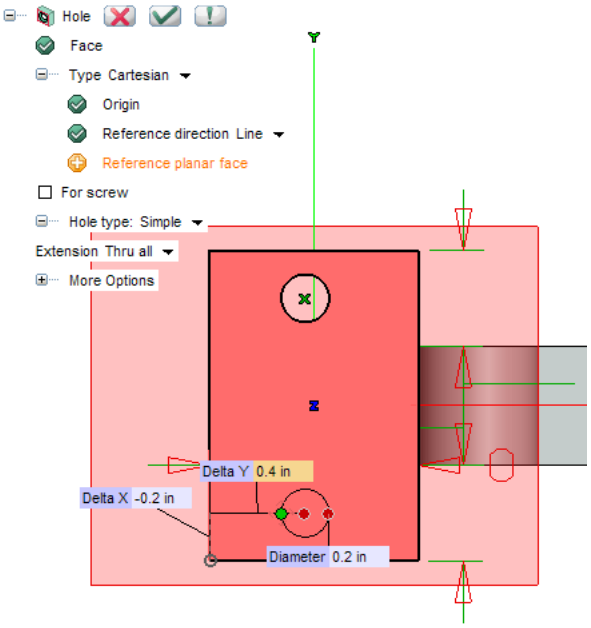
Let's add simple holes to this base solid.

- Activate **Hole**.

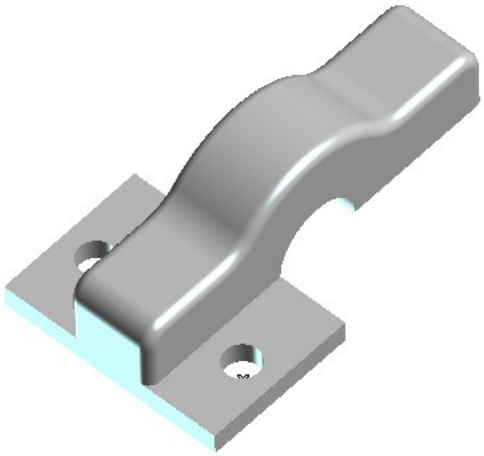
- Select the type to Cartesian.
- Set the values as shown.
- Click Apply.



- Now click Origin in selection list.
- Using **End Point Snap** select the bottom corner point for the Origin.
- Say OK.

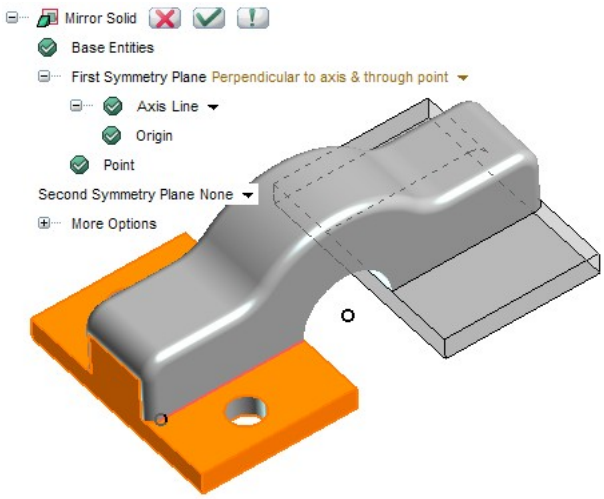


Model looks like this.

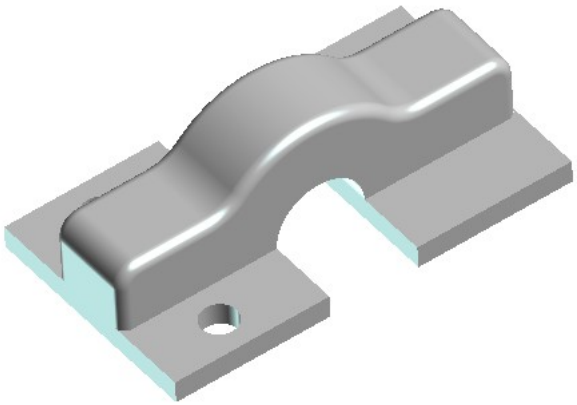


Now let's make a mirror of this solid to create the final part.

- Start the **Mirror Solid** command.
- Select the base solid for the Base entity.
- Select Perpendicular to axis and through point for First Symmetry Plane option.
- Select one of the edges as the axis line.
- Select the centre point of the half circle for the Point selection.

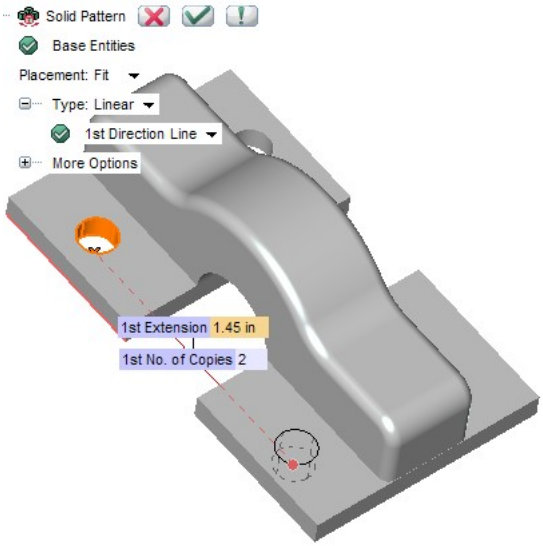


- Click OK.

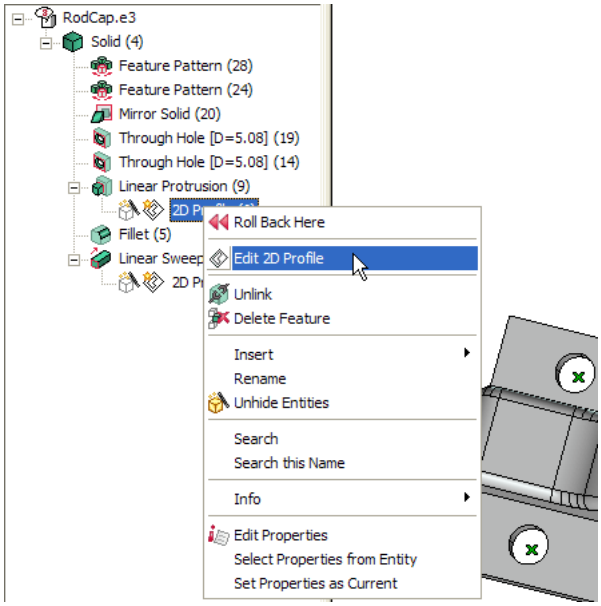


We need to create holes on the other side using the **Pattern Solid** command.

- Start **Pattern Solid** command.
- Set Type Linear.
- Select the Simple holes created as the Base Entities in the Selection List.
- Click on any of the horizontal edges of the base solid as indicated below.
- Set the 1st No. copies 2.
- Set the 1st Extension 1.45in.
- Hit OK.

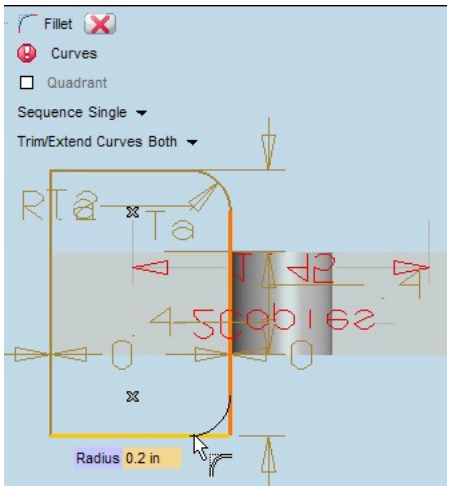


Now select the Linear protrusion profile from the history tree and choose Edit Profile from the context menu to modify the base solid.

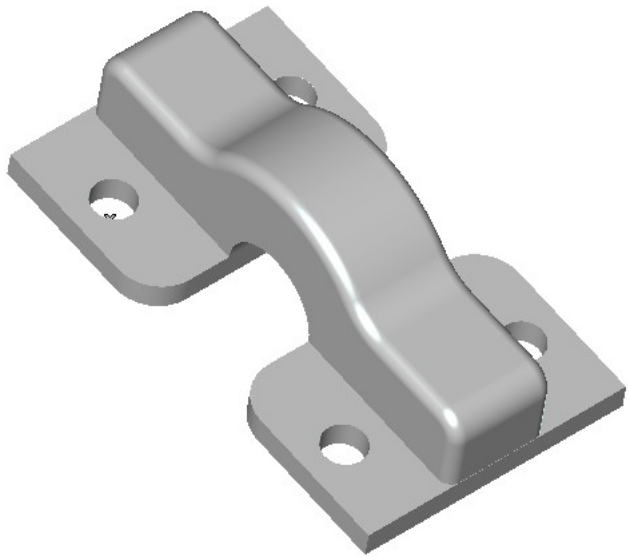


let's **Insert Fillet** to the two inner corners of the base solids.

- Click on **Insert Fillet** on the Drafting toolbar .
- Type 0.2 as the Fillet Radius.
- Select the edges as shown below.
- Double click on the screen to update the model.



Hide dimension lines and profiles to give the model a better look.



Looking good! Congratulations.