Introduction to Solid Modeling

Solid modeling is a process of creating 2-D profiles, extruding 3-D volumes, and applying features to create a part. Having worked through the Basic training tutorials, one should be familiar with many of the solid modeling commands in ThinkDesign. However, solid modeling is more than just capturing the shape and dimensions of a part. It's also important to convey design intent to whomever views the model, even if it's yourself. In this task we shall move away from introducing commands, and instead, apply them to a "real world" application. We shall model a three-hole swivel clevis. Starting with a profile, we shall create a solid to which we will add additional features and details to create a manufacturable part. We suggest you to go through 15 Rules of Solid Modeling article before doing this task.

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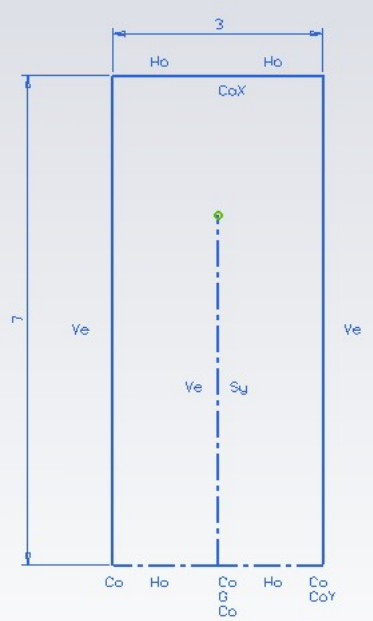
1. Step 1: Create the profile

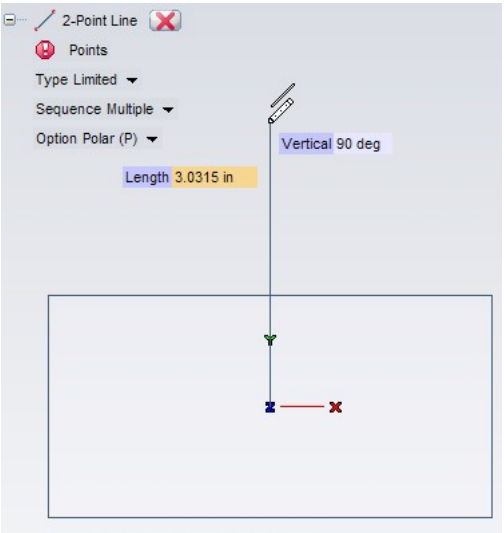
How one models a particular part is just as important as the model itself. The decisions taken while creating the model will affect any future additions and modifications you (or someone else) will make to the model. Therefore one should communicate design intent through the model. With this in mind, our first strategic decision will be to take advantage of the natural symmetry and model only half the part, then mirror it. Often this will save the amount of work one must do. However, the not so obvious side-effect is that the designer defines the parting plane of the part. The second choice is to describe sthe basic hape by modeling the U-shape extrusion. We shall start by creating the first profile.

First sketch a vertical centerline starting from the origin. This will serve as a point of reference to constrain the profile. For now, do not worry about placement or exact values. In a few moments we shall constrain the profile to set the size and lock it in place.



- Start with a **New Model** file and set the units to Inches.
- Click Insert > Profile > 2D.
- Draw a vertical **Two-point Line** from the **Work Plane Origin**.

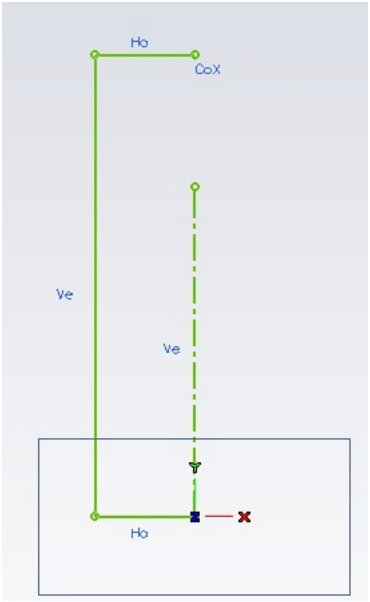
The length is not that important because it's going to be a reference line, but if you would like, make the Length5 in.



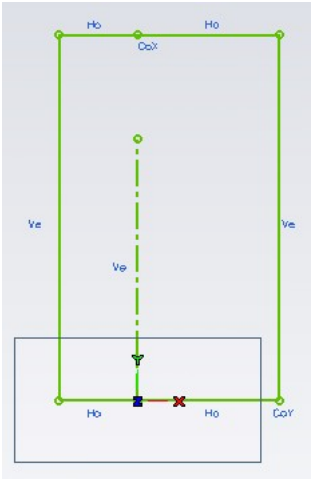
Use Make Reference to convert the line into reference geometry.

We shall create the U-shaped Profile with the open end facing down.

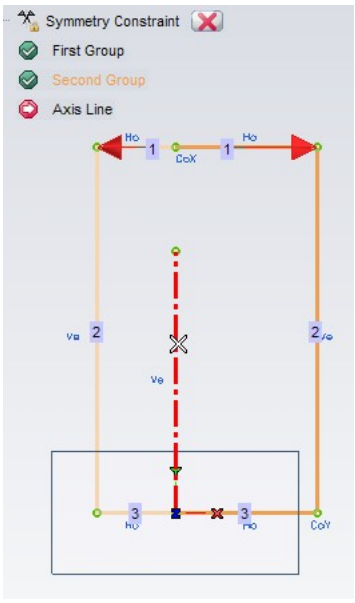
- Sketch a Profile of about 1.5in wide and 7in long as shown below.

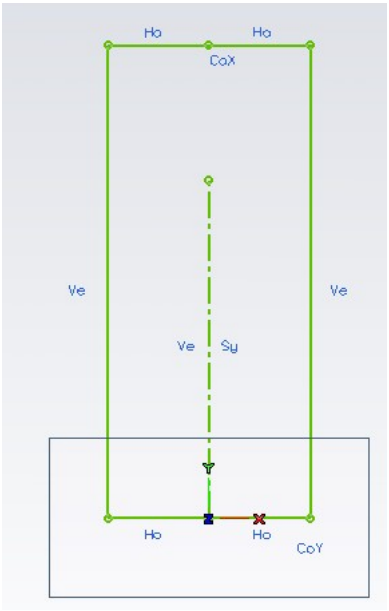


Create a similar Profile on the other side of the Reference line, with some arbitrary values of length and Width. Since the profile is Symmetric, we can apply the Symmetric Constraint to complete the Rectangle.

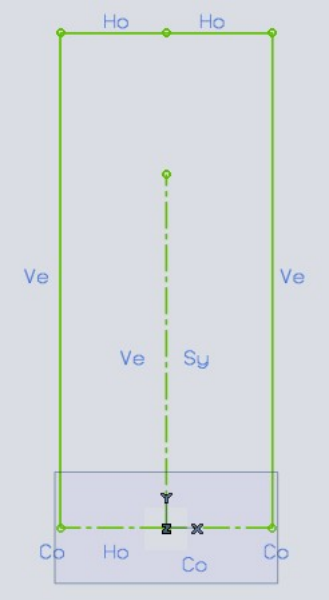


Apply the **Symmetry Constraint** between the Curves on the either side of the Reference Line as shown below. Select the Curves on the left hand side of the Reference line as First Group and Curves on the Right Hand side as Second Group and Refernce line as the Axis.





Use Make Reference to convert the Bottom Horizontal lines as Reference Geometry.

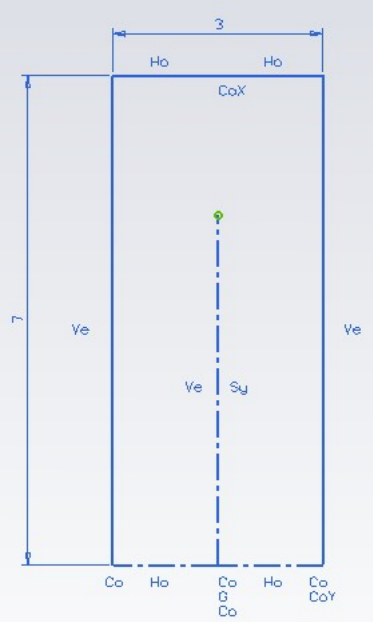


Add geometric constraints to the profile starting with a **Ground Constraint**.

- Add a **Ground Constraint** to the bottom of the vertical centerline.
- Add a **Coincident Constraint** between the bottom end point of the reference line and the any one end point of the Bottom Horizontal Reference line.

Let us add some Dimensions inorder to completely constraint the model.

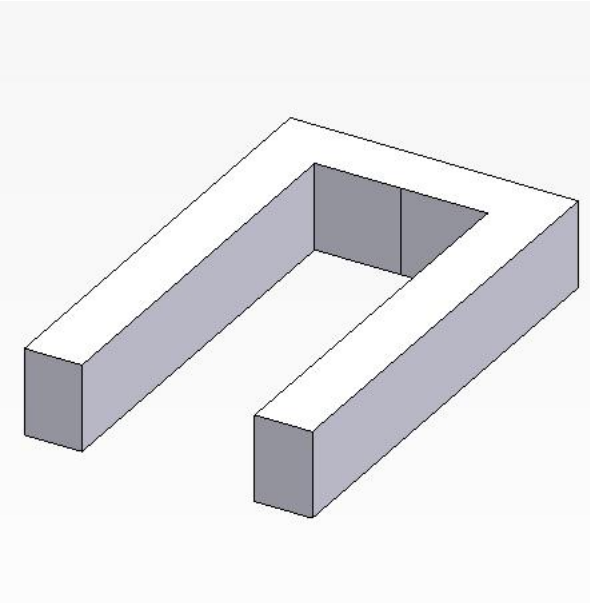
- Start the **Smart Dimension** command.
- Add the height and set it to 7 in Measure7 in.
- Add the width and set it to 3 in Measure3 in.
- And finally add the height to the Centre Reference line.



The profile will not be fully constrained because the centerline has one end point that is not constrained. It is not important that we do so, and it does not matter because we can still create a solid from a partially constrained profile. That's what we shall do in the next step.

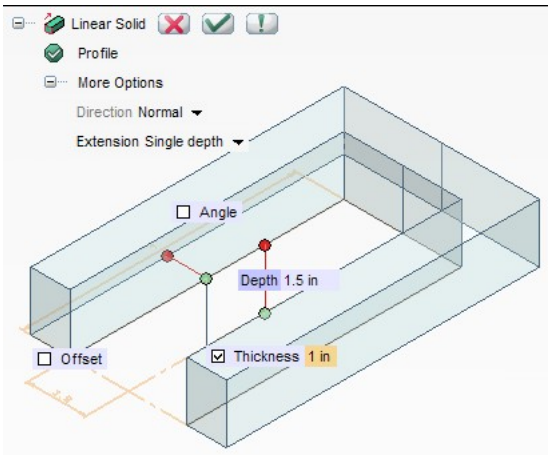
2. Step 2: Today's letter is the letter U

We are now going to take the profile and create a **Linear Solid**. You might be objecting right now because the profile is not closed. (Remember, that reference geometry does not add material to a solid.) Again, we have captured design intent. In this case, the profile describes the inside surface of the U-shaped solid. We want the wall thickness around the profile to be uniform, and so to create the solid we shall simply "thicken" the wall. You will see as we create the profile. (Alternatively we could have created a closed profile and then had the flexibility to individually control the wall thickness.)



Create a **Linear Solid** from the profile.

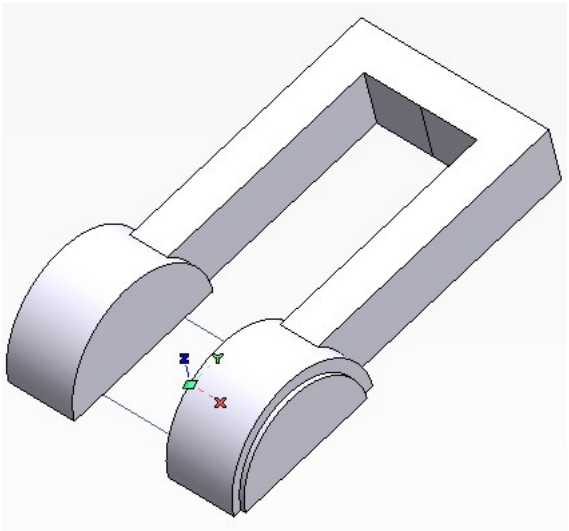
- Start the **Linear Solid** command
- Make the Depth 1.5 in. Depth1.5 in
- Expand More Options.
- Turn on the Thickness and set it to 1 so that the thickness is added to the outside.
- Hit OK



Sometimes what we do not do is as important as what we do do. We had the opportunity to add draft to the part, and it is good practice that we add draft early in the modeling process because, for example, we cannot add draft to a surface that shares an edge with a fillet. However, adding draft now would affect the ability to add the main boss because the surface from which the boss will originate will no longer be at a right angle. We shall worry about that boss later and deal with the secondary bosses in the next step.

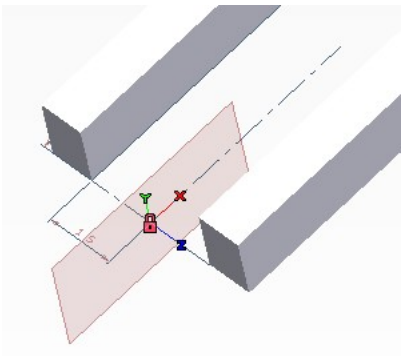
3. Step 3: Who's the boss?

Because we are only building half the part, we shall build only half of the two bosses. We have many ways of doing this. The two natural ways include creating half a cylinder by either sweeping a half-moon cross-section or to rotate a rectangle around an edge. After either operation, we remove the middle section. Even though the rotational sweep method is relatively easier, we shall choose the linear sweep method because we want the radius dimension to appear in a side view drawing. Let's do that first and then we will remove the excess material in the next step.



We shall need to rotate the Work Plane because we want to create a profile so that the sweep direction is along the horizontal reference line.

- Select **Edit**➪**Work Plane**➪**Align to X**and pick the centerline.
- Select **Edit**➪**Work Plane**➪**Align to Z**and pick the horizontal reference line..

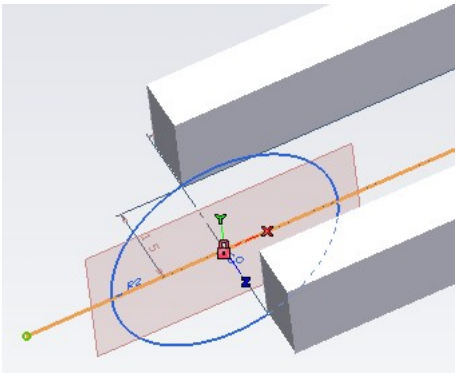


Now create the half-moon profile.

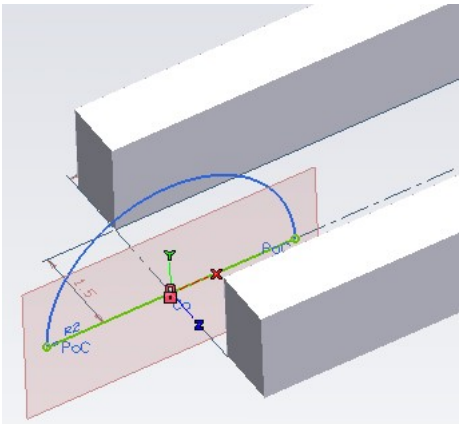
- Go to Insert > Profile > 2D.

You probably want to switch to an isometric view to see the snap point more clearly.

- Create a **Center Circle** with Radius2 in and make sure to snap to the end point -- **Tools➪Snap➪End Point**-- where the centerline intersects the horizontal reference line.
- Create an **Line Given Angle** of Angle0 deg and use **Snap to Arc Center** to snap to the center of the circle.



- Use **Smart Delete** and delete the extra curves so that you are left with the semi circular shape shown below.

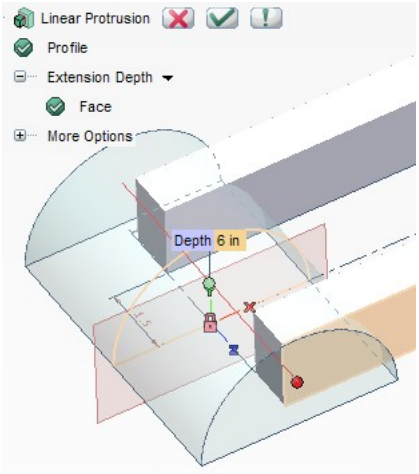


Constrain the profile.

- Notice that the **Point On Curve Constraint** between the horizontal line and the arc center is already in place.

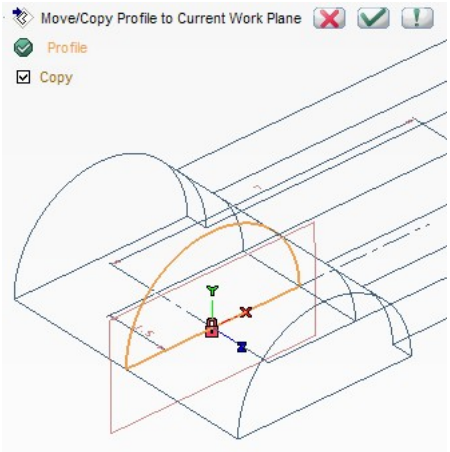
Now create the **Linear Protrusion**

- Start **Linear Protrusion**.
- Pick a side face on the U-shape.
- Uncheck Thickness and then collapse More Options if it's expanded
- Right click Depth and select to Symmetric.
- Then set it to 6 in Depth6 in and click OK.



Reuse the boss profile to create an extension.

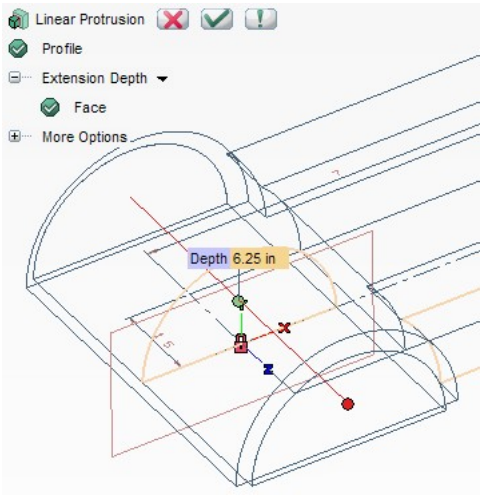
- Switch to a **Wireframe View** display mode.
- Select **Move/Copy to Current Work Plane**



- Select the half-moon profile and check the Copy option to duplicate the profile.
- Use the History Tree, and right click the original half-moon profile under the Linear Protrusion and select **Hide Entities**

Adjust the size of this new Profile and then create the extension.

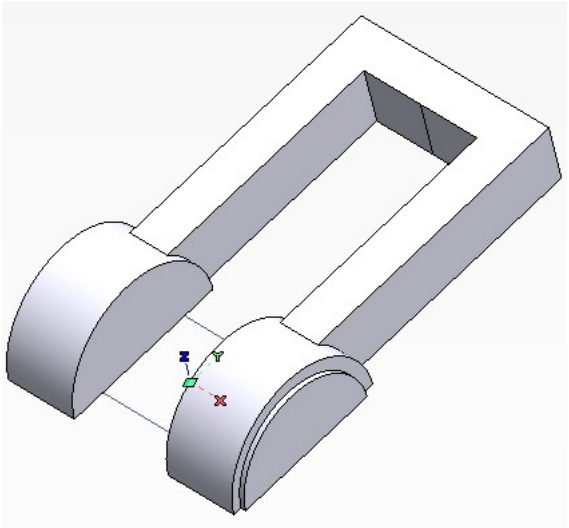
- Double click the new half-moon profile to switch to Profile Mode.
- Change the radius dimension to 1.75.
- Create a **Linear Protrusion** from this profile similar to the previous protrusion that is 6.25. You may be prompted to **Rebuild Model**.



On to the removal step.

4. Step 4: Sorry, but you're cut from the team

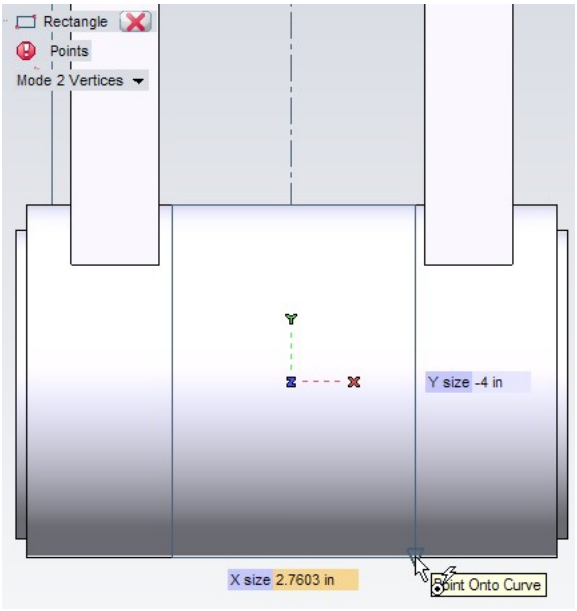
In this step we shall cut-out the material in the middle to reform the U-shape. Now why did not we just create one boss and then mirror it to the other side and skip the cut operation? With any CAD system, there are limitations. In this case, we already decided before we started modeling that we would use the **Mirror Solid** command. What you did not know is that the limitation here is that you cannot mirror a mirrored feature. Remember what was said earlier. Design and modeling decisions will affect available options later in the modeling process.



Start off by creating the rectangular profile for the cut.

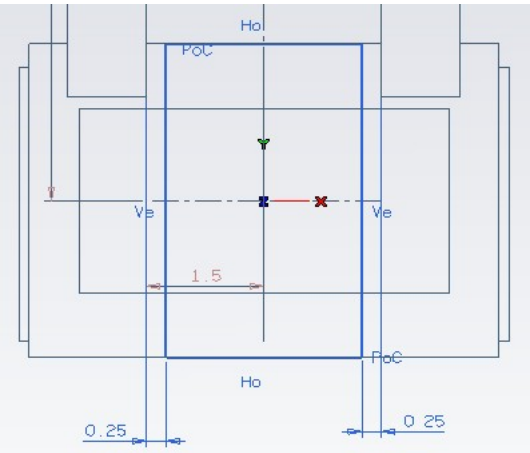
- Right click the Work Plane and Reset to World.
- Switch to a **Top View** and use **Hide Entities** to hide the dimensions and profiles.
- Click the Profile tab.

- Start the **Rectangle** command and set the Mode to 2 Vertices -- Mode 2 Vertices -- if it is not already.
- Use the **Point on Curve Snap**, to place each corner of the rectangle so that is lies on the edges of the boss.



Add dimensions to the profile.

- Start **Smart Dimension** command.
- Add a 0.25 in dimension between the left edge of the rectangle and the closest inside bottom edge of the U-shape solid.
- Do the same for the right edge of the rectangle.

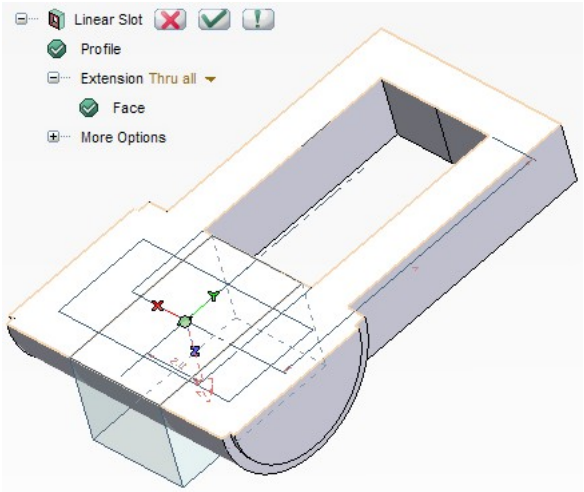


Again we captured design intent. We could have just as easily defined the width of the cut and made it symmetric about the centerline using a width/half-width pair. By dimensioning the extensions instead we guarantee that no matter how wide the clevis is, the extensions will be 0.25 inches from the inside surface.

Also notice how this particular profile extensively references geometry off the solid. The dimensions reference edges on the solid. The Point on Curve constraints also reference edges on the solid. In fact, all the profiles so far, except for the first, reference the solid in some manner. The first profile is the only grounded and independent profile. It is a good practice to ground only the first profile, and then reference existing geometry for all subsequent features and profiles.

That is enough of a pause. Let's now use the new profile to create the cut.

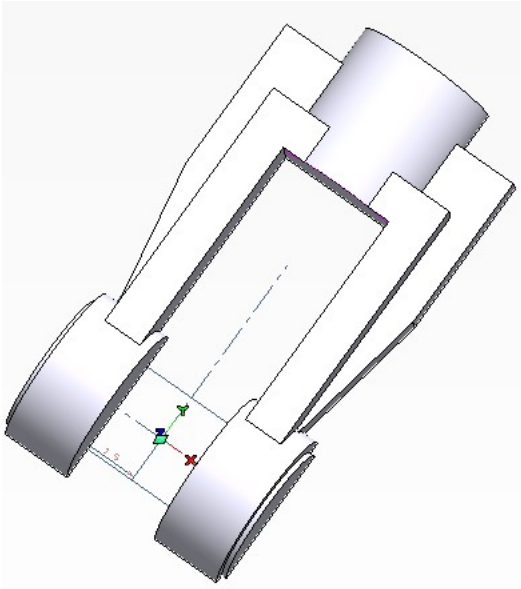
- Start **Linear Slot** command.
- Pick the bottom face. It may help to change the display mode to **Shaded View and Boundaries**
- Turn off the Symmetric option.
- In the Selection List, set Extension Thru all.
- Hit OK.



Now that we finished the secondary bosses, we shall next create the main boss.

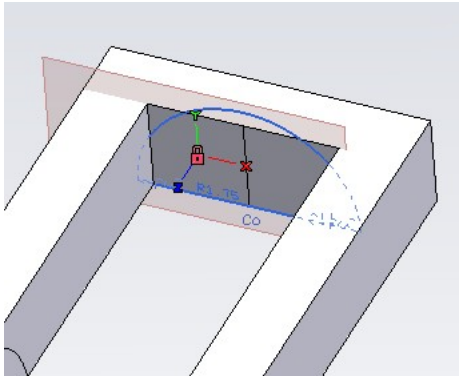
5. Step 5: The big boss

Adding the top boss will be similar to adding the secondary bosses. We shall duplicate the half-moon profile from Step 3. It is more efficient to reuse existing geometry when possible. However, before we copy the profile, we should move the work plane to the top of the inside of the U-shape solid.



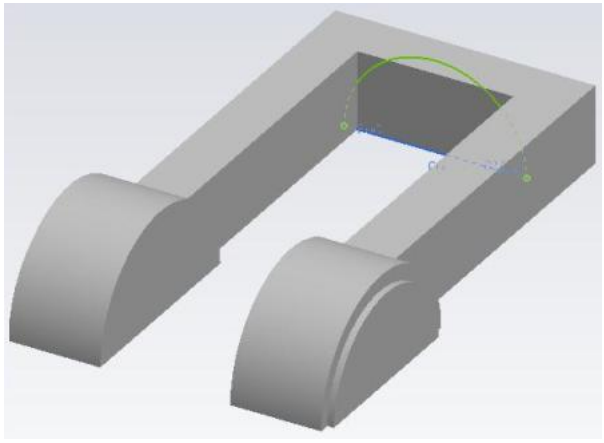
Unhide the necessary profiles, position the work plane, and then duplicate the half-moon profile.

- Use **Hide Entities** to hide the profile used for the slot, and use **Unhide Entities** to redisplay the first half-moon profile.
- Double click the inside top surface to move the Work Plane.
- Highlight theWork Plane, right for context menu, and select Move, and snap to the middle of the bottom edge. Use **Tools**➪**Snap**➪**Mid Point**if necessary.
- If necessary, use **Edit Work Plane** to rotate the Work Plane so that Y points upwards.
- Use **Move/Copy to Current Work Plane** to copy the half-moon profile like we did in Step 3.



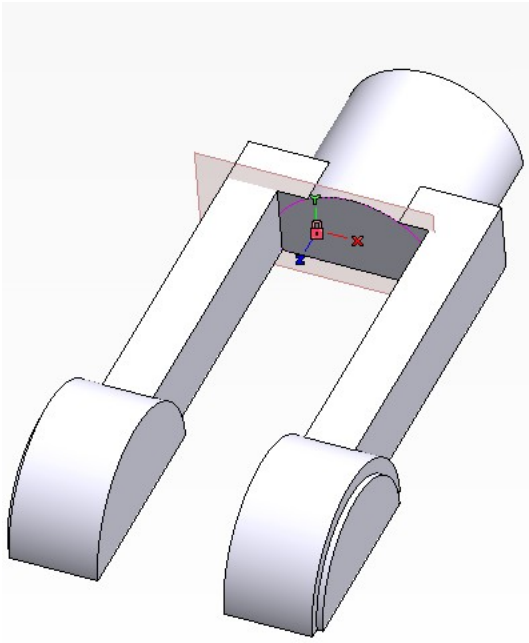
Reconstrain the new profile.

- Right click the new profile, and select Edit Profile.
- Change the radius to 1.75.
- Put a **Collinear Constraint** between the line and the bottom edge.
- Next place a **Coincident Constraint** between the arc center and the centerline end point.



Create the boss feature.

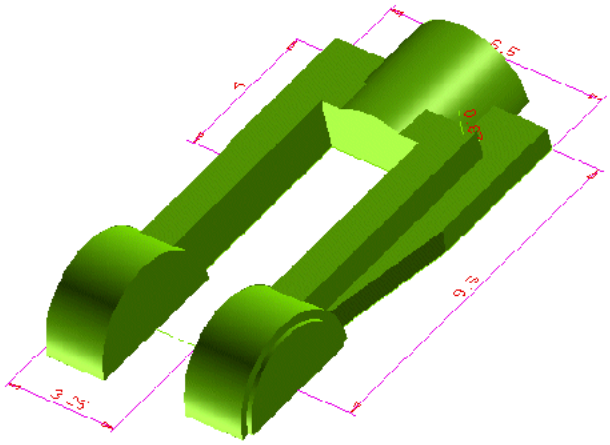
- Start **Linear Protrusion** command.
- We want the length of the protrusion to be -3.5. Depth-3.5
- Hit OK.



Let's add the final major feature, the outer webbing.

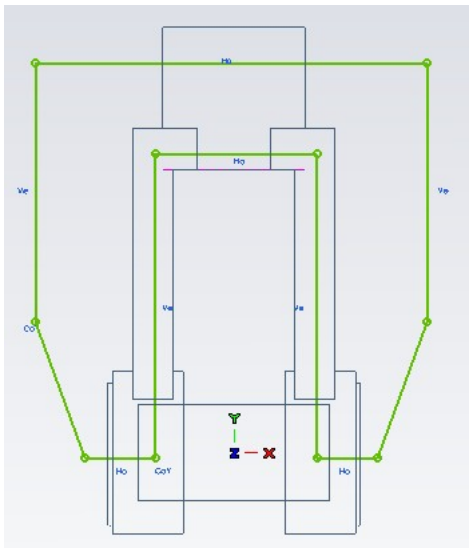
6. Step 6: What a complicated web we weave

The webbing is a feature to reinforce the clevis. It will be a **Linear Protrusion**. This time we shall add draft at creation time. The only complicated part will be constraining the profile.



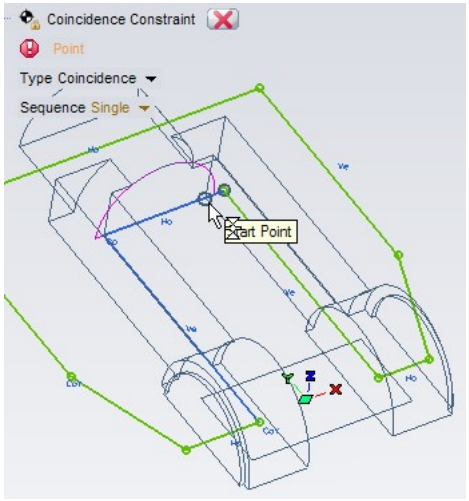
Sketch the profile.

- Change the display mode to **Wireframe View**, right click the Work plane and click Set to World, switch to a **Top View**, and hide all dimensions and profiles except for the first profile.
- Click the Profile tab.
- Use **Polyline** to sketch the shape similar to the one below. Do not snap to anything just yet.



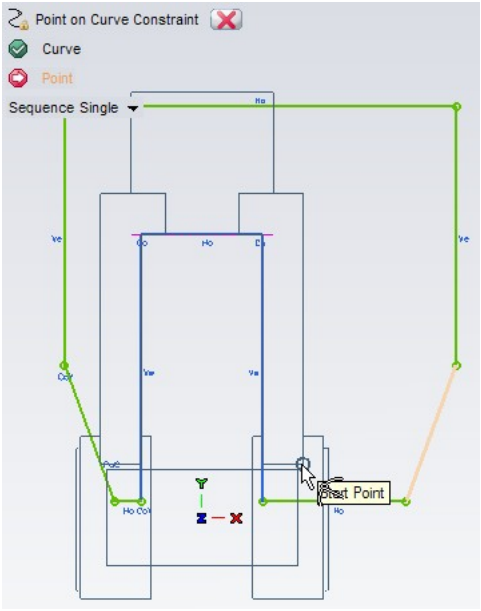
Add a coincident constraint as shown in the image below to lock the profile to the solid.

- Next add **Coincident Constraint** between the top inside corners of the profile and the top inside corners of the solid. Be certain to pick the corner that's part of the bottom surface of the solid so rotate the view as needed.



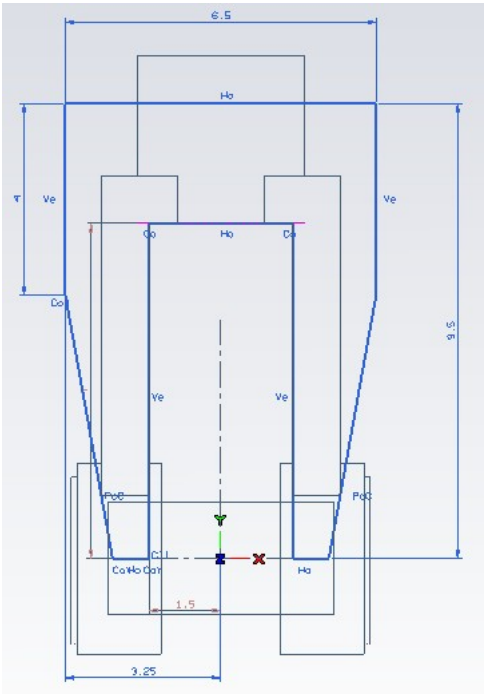
Add the remaining geometric constraints to control the shape.

Finally add **Point On Curve Constraint** between the angled edges and the corners formed by the intersection between the secondary boss and U-shaped solid. NOTE: if the Profile looks skewed or crooked, just drag one of the vertical lines to return it to a better shape.



Add dimensional constrains.

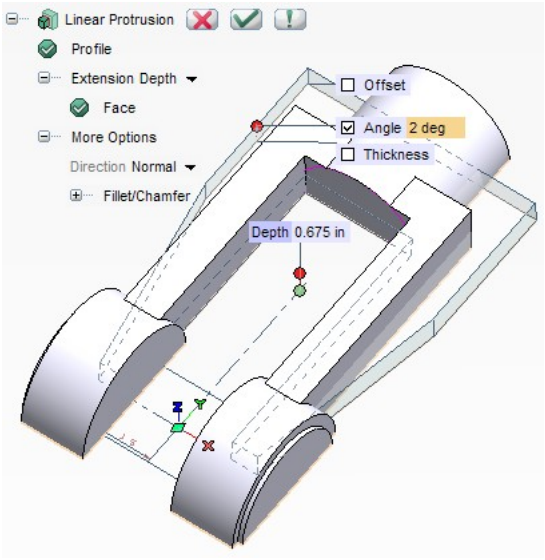
- Use the **Smart Dimension** command to add the over all width of 6.5.
- Follow that with a half dimension of 3.25 in. Remember to use the centerline.
- Define the overall height to be 9.5 in.
- Then all that is left is to define the distance from the top to the middle corner which is 4 in.



Create the web feature.

- Start **Linear Protrusion** command.
- Pick the bottom face

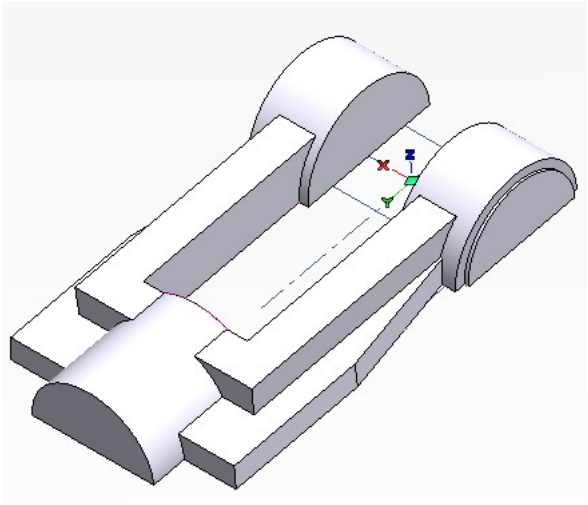
- Set Depth-0.675 in.
- Expand More Options, then set Angle2 deg. Invert if necessary.
- Hit OK.



Now we are ready to add draft to the rest of the part.

7. Step 7: You've been drafted

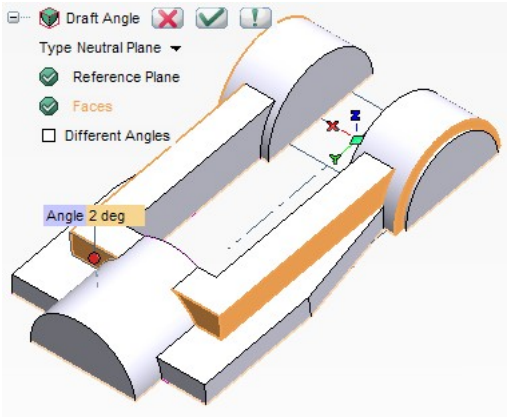
Draft will allow the tool to eject the part from the mold more easily. However, we will not add draft to all the side surfaces. We want to keep some surfaces at a right angle, namely the boss surfaces. This is because those surfaces will mate to another part. The plan is to have those surfaces machined in post-processing.



Add draft to the non-mating surfaces that are perpendicular to the parting plane.

- Start the **Draft Angle** command.
- Select the bottom face as the Reference Plane
- For the Face selection, pick the side surfaces around the part that are orthogonal to the Reference Plane. Ignore both faces of the secondary bosses and the top face of the main boss.

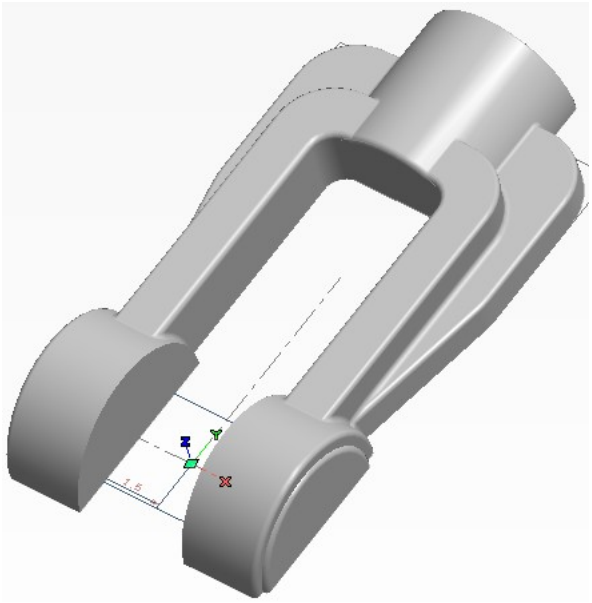
- Set the angle to -2 deg. Angle-2 deg The negative sign points the angle inward.
- Hit OK.



The angle is slight but if we were to zoom in, we would notice the difference. Had we applied draft before we created the main boss, the boss would be pointed at a 2 degree angle below the horizontal. We would not want that. Next we will add fillets.

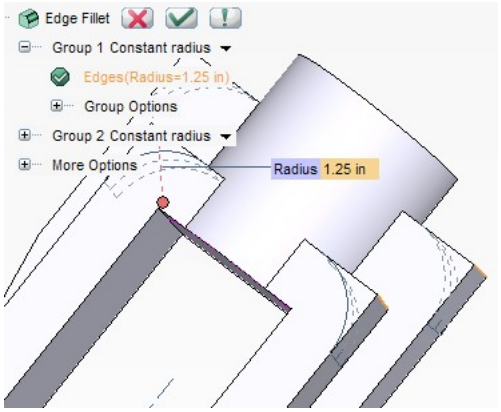
8. Step 8: As he rounds the corners...

Now that we have drafted all the surfaces we want to draft, we can add fillets to the part. The order in which we apply these fillets is very important. Many of these edges are joined at sharp corners. We will apply the fillets in a order such that we create a series of tangent edges.

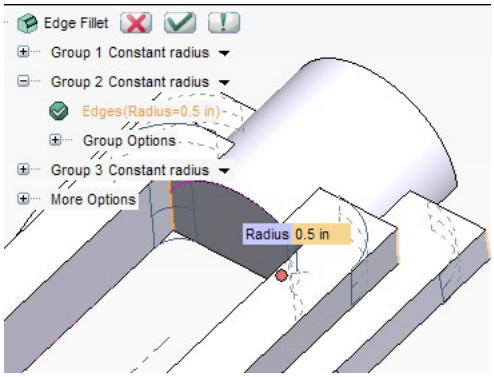


Fillet the major corners.

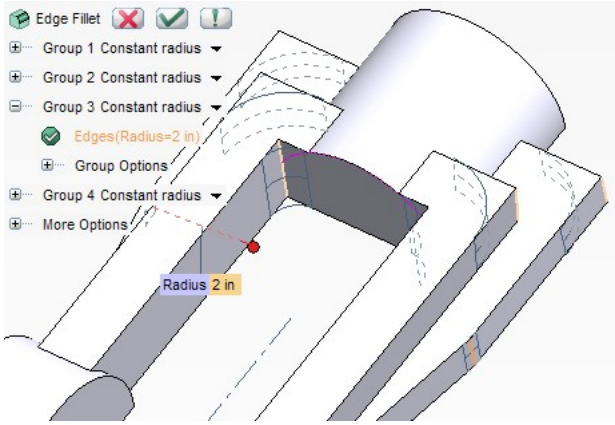
- Start **Edge Fillet** command.
- For the first group, create a 1.25 fillet at each top corner of the webbing.



- Click Group 2 from the Selection List, and create a 0.5 fillet at each top inside corner of the U-shape feature.



- And for the last group, click Group 3 and create a 2 fillet at each outside middle corner of the webbing.

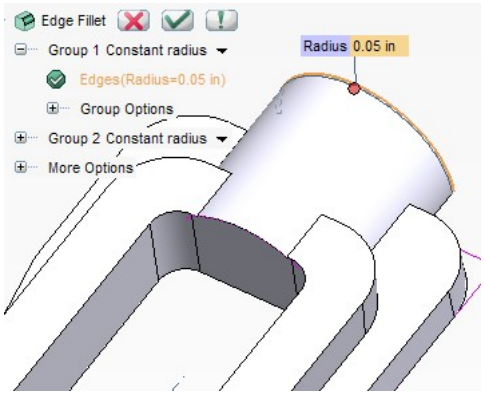


- Hit Apply.

Next add fillets to the bosses.

- First select the top edges of the secondary bosses.
- Next pick the edges where the extensions meet the secondary bosses.
- Set Radius0.12 in
- Create a new group in the Selection List, and add a 0.05 fillet to the edge at the end of the main boss.

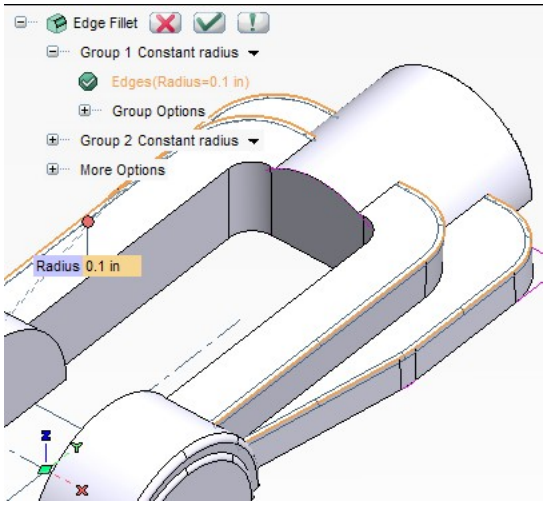
- Hit Apply.



Move on to the next set of fillets.

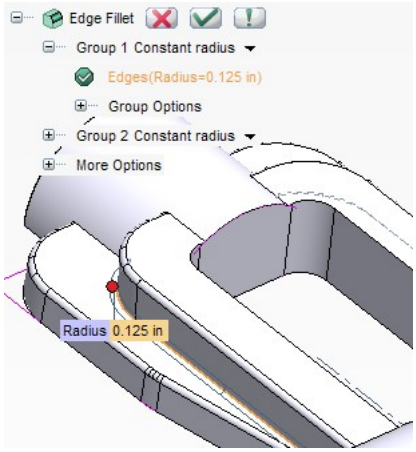
> You do not have to hit Apply all the time when making these fillets. If you want to stop and admire your work, click OK to exit the fillet command. You can always start again!

- Fillet the edges shown on both sides. Notice how the edges chain. This is a direct result of the fillets we added at the beginning. NOTE: If the edges do not chain, expand More Options and check Tangency Chain.
- Set Radius0.1 in
- Hit Apply



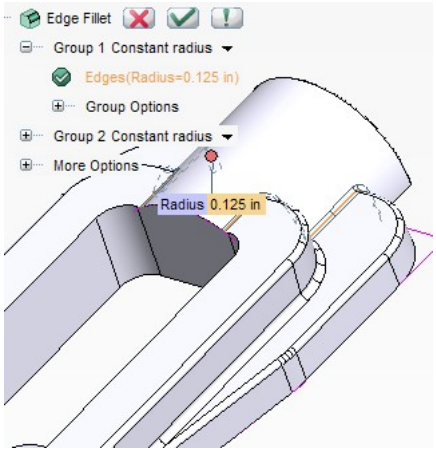
Now fillet the pair of edges in between the previous edges. We could not fillet these edges until we filleted the ones before hand. Otherwise the blend where the webbing meets the secondary boss would have been bad.

- Fillet the edges shown on both sides.
- Hit Apply since the radius is still 0.125.



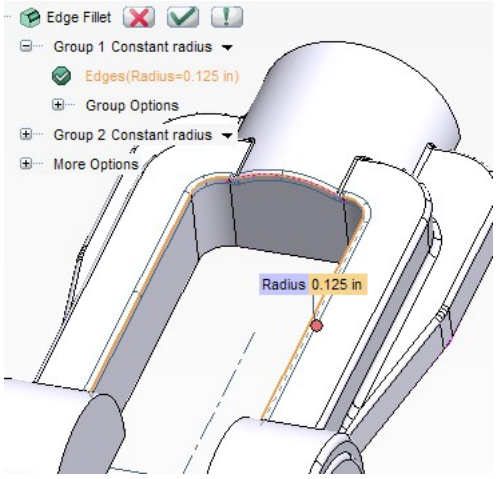
Now fillet the edges where the main boss intersects the U-shape feature.

- Fillet the edges shown on both sides. Again notice how the edges chain as a result of the previous set of fillets.
- Hit Apply.



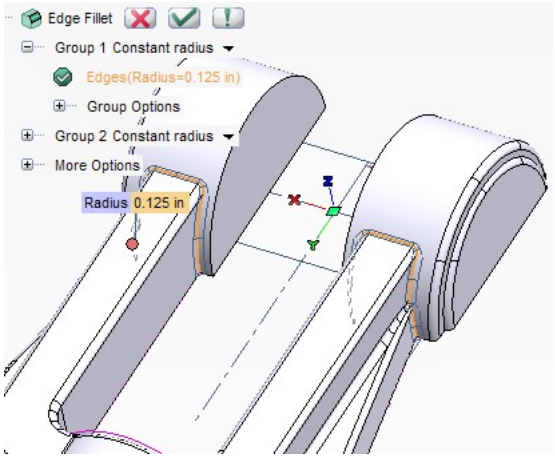
Fillet the inner top edge.

- Fillet the edges shown. See how it chains over the main boss now because of the previous fillet operation.
- Hit Apply



Fillet the edges where the secondary bosses intersect the U-shape feature.

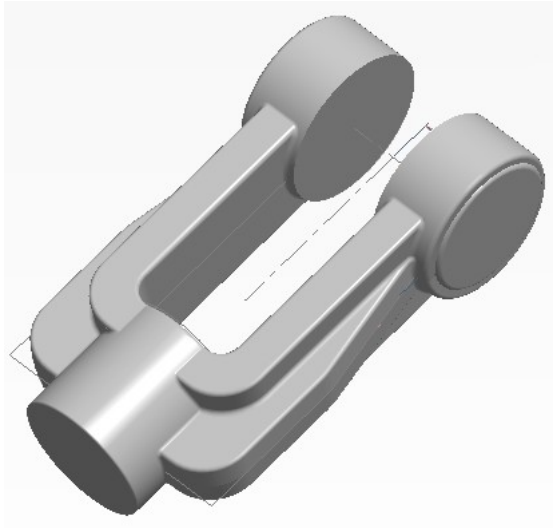
- Fillet the edges shown. Again it chains properly.
- Hit OK.



We finished half the part. Remember that the order that we add features determines the result. It's time to mirror it and then join with the copy to create one solid.

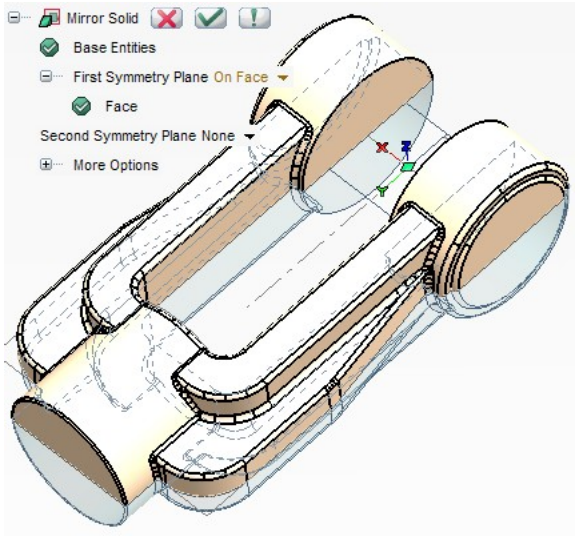
9. Step 9: Mirror, mirror on the wall

In this step we shall mirror the completed half-clevis. The parting edge will be very apparent. However since the **Mirror Solid** command creates a new separate solid, we have to use the Union command to get one single solid again.



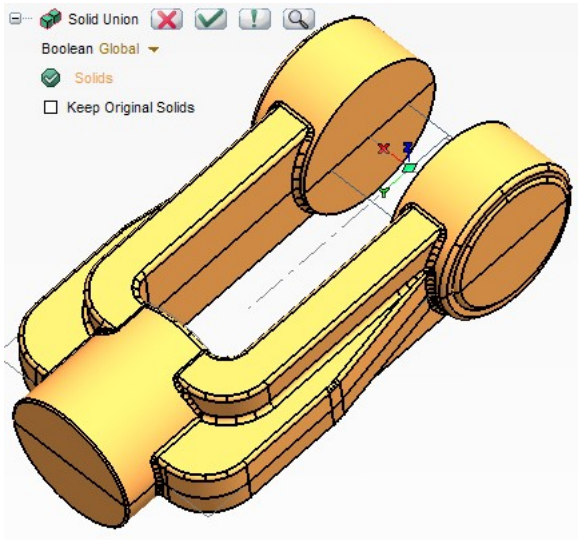
First mirror the solid.

- Reselect the solid (window around it or select it from the History Tree).
- Start **Mirror Solid** command.
- Click the bottom face of the part for the Face prompt.
- Hit OK.



Now join the two solids together.

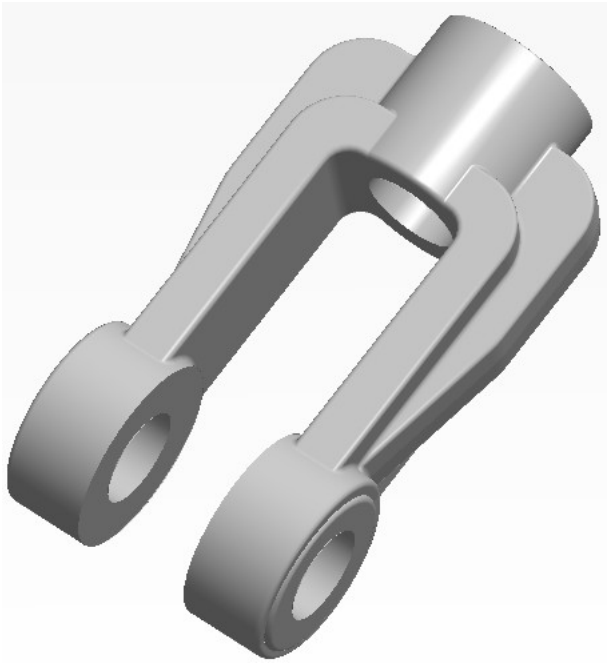
- Start **Solid Union** command.
- Select both solids.
- Hit OK



We are on the homestretch now. We only need to add the holes.

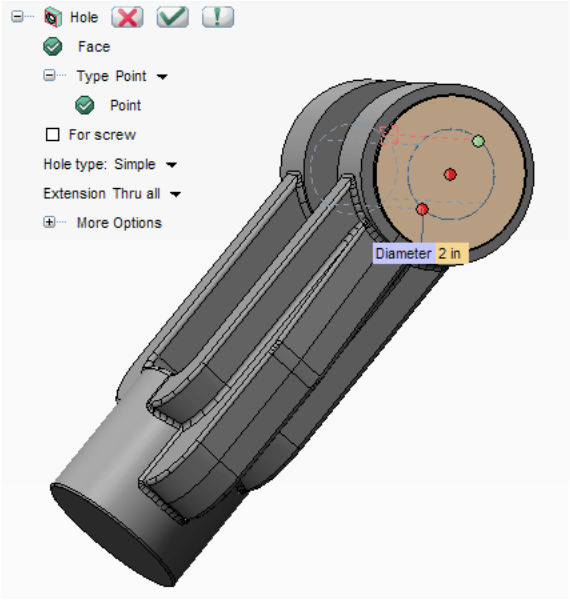
10. Step 10: Adding top boss and holes

For this final step we shall add the holes to the bosses and then add a single chamfer to the main boss hole. You might be wondering why we waited until after the **Mirror Solid** feature to add these holes. The reason is because where we placed the parting plane makes the holes an undercut feature. The holes are going to be drilled as a post-process, anyway. By making the holes last, all the toolmaker needs to do (assuming he/she uses ThinkDesign, too) is to simply delete or deactivate those features, and then there is no undercut. That's the benefit of carefully planning how to model a part.



Add a hole to the secondary bosses.

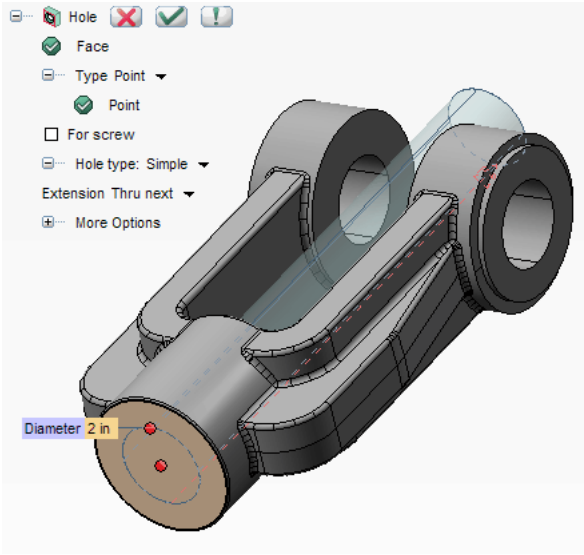
- Start **Hole** command.
- Click the outside face of a secondary boss.
- Drag the center point to the circumference of the face to snap to the arc center.
- Set Diameter2 in
- Hit Apply.



Add a hole to the main boss.

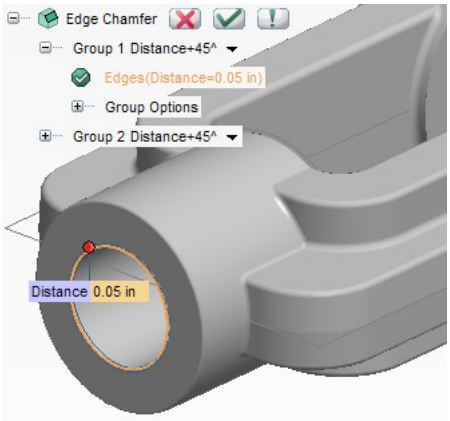
- Click the top face of the main boss.

- Drag the center point to the circumference of the face to snap to the arc center.
- Set the Extension to Thru Next since we only want the hole to go up to the next face.
- Hit OK.



Finally add a chamfer to the top of the main boss hole.

- Start **Edge Chamfer** command.
- Pick the hole edge at the top.
- Set Distance0.05 in.
- Hit OK.



Clean-up the Graphics Area and that's the end of this task. You should now understand the importance of planning your modeling strategy, how decisions early in the process impact decisions later on, and why you must endeavor to communicate your design intent by the decisions you make during that process. Now that you know that solid modeling is more than just capturing the shape of your part, you are on your way to becoming a better designer. :-)