2D to 3D

In this task, you'll use an existing AutoCAD® drawing to create the model we need. First, you'll open the .dwg file and remove overlaps,then insert the .dwg file into an empty model, then use the 2D geometry to create the camma model
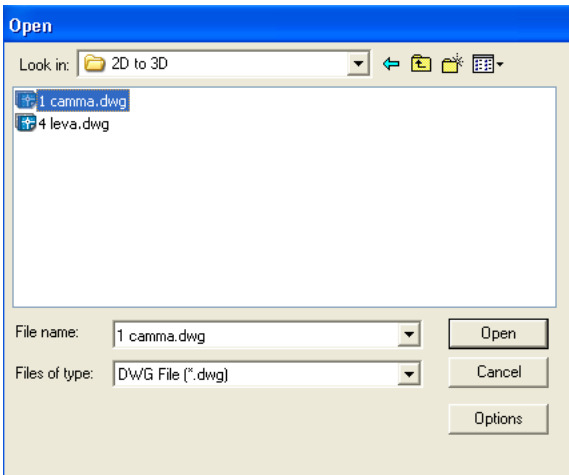
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1. Step 1: Import the file

In this step we'll open the AutoCAD drawing file camma.dwg. We'll remove the overlaps and compress the drawing.
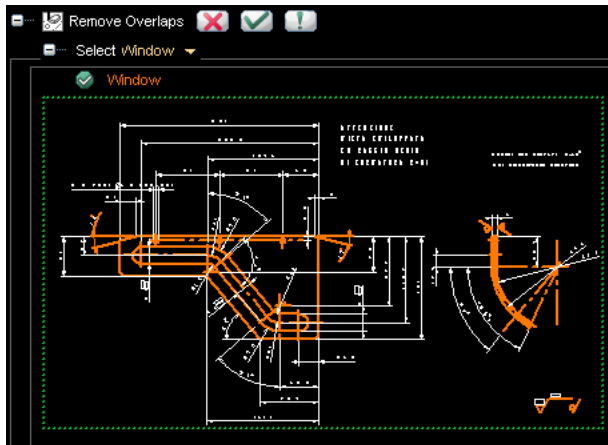
The Remove Overlaps command enables you to merge partially or totally overlapping (or consecutive and aligned) entities (lines and arcs) into one single entity.

Open 1 camma.dwg from the insallation folder.

We'll remove the overlaps if any using Modify drafting remove overlaps.

- Set the drawing view as current

- Activate command Remove Overlaps

- Set as window in the selection tree and select two points identifying the two opposite corners of the drawing view.

ThinkDesign generates a remove overlaps report in a small window with details.
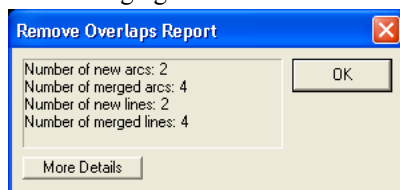

After removing the overlaps we'll compress the drawing

Sidebar title goes here.

The Compress command enables you to compress the memory space occupied by the current model by recovering memory areas occupied by deleted entities.

Activate **Tools**↘**Compress**and click yes to the warning window appears.

Compressing the memory space resets the Undo/Redo buffer. After compression, any previous operations will not be undoable. Click OK to the warning appears
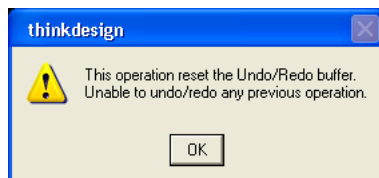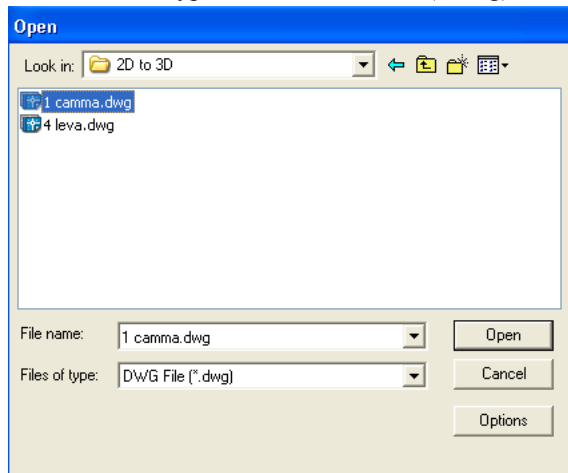

Save and close the drawing file.

Pick **New Model** from the New Document drop down in the Standard toolbar OR pick **File**↘**New...**↘ **Model**from the pull down menu.

Okay...we're ready to import the drawing using **Insert From File**

• Pick **Insert**↘**From File**from the pull down menu

- Set the file type filter to DWG File (*.dwg).

- Locate and Select ConnectingRodEnd.dwg from the task installation directory.

- Hit Open to import the file.

While importing a file thinkdesign gives you some options on what to import and also allows you to modify the position of the items you're importing.

- Set the Transfer option to All. This option imports all the entities in the file.

- Uncheck Work Plane Position in the Modify box. Leaving this option unchecked locates the imported entities by matching the origin of the imported file to the current Work Plane origin in the model.
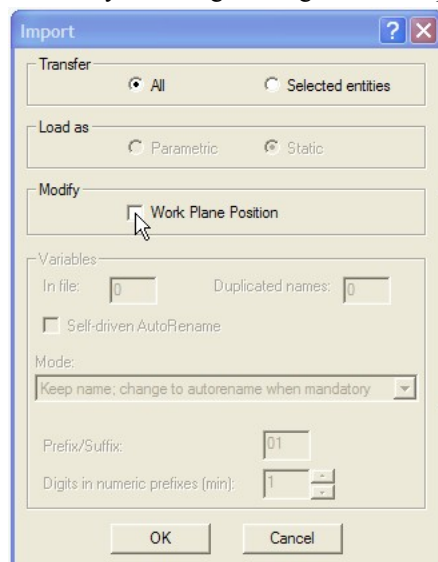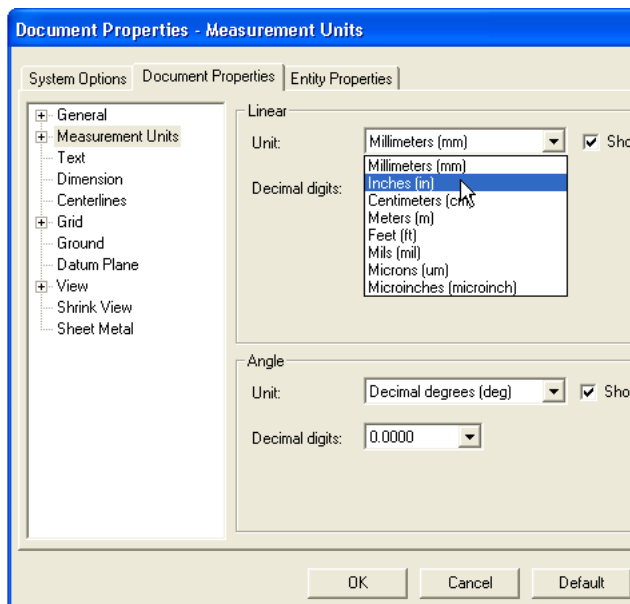

- Hit OK to complete the import.

- Click **Fit View** and take a look.
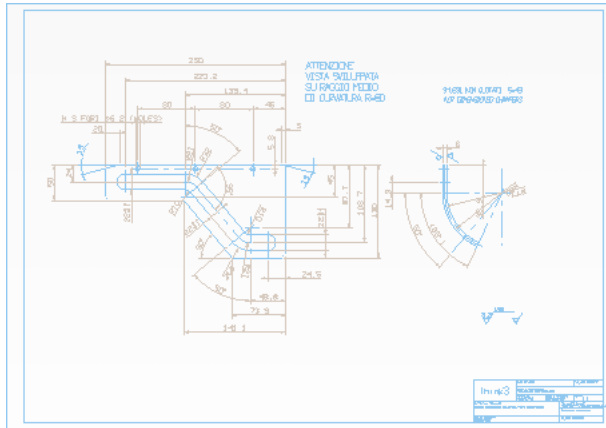
Now let's change the measuring unit to Inches.

- Right click in Graphics area and say **Options/Properties**

- Change the unit to Inches under Measure Units of the Document Properties tab.

- Hit OK.

- Select **Compress**.

Observe that the dimensions are now shown in Inches.

Now we have the entire drawing imported, border and all. In the next step we'll create the base solid from the front view geometry.

2. Step 2: Create the Base Solid

In this step we'll use the front view geometry to sweep the base solid symmetrically with the **Linear Solid** command.

We've imported the whole drawing, which gives us some extra stuff we really don't need to see. Let's take a look at the Layer manager tab in the history tree.

As you can see, there are several layers in this file. Let's try turning off all layers except layer0 and 1 by clicking on the bulb icon in the layer manager.
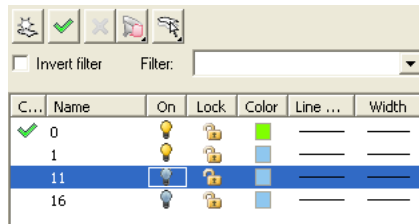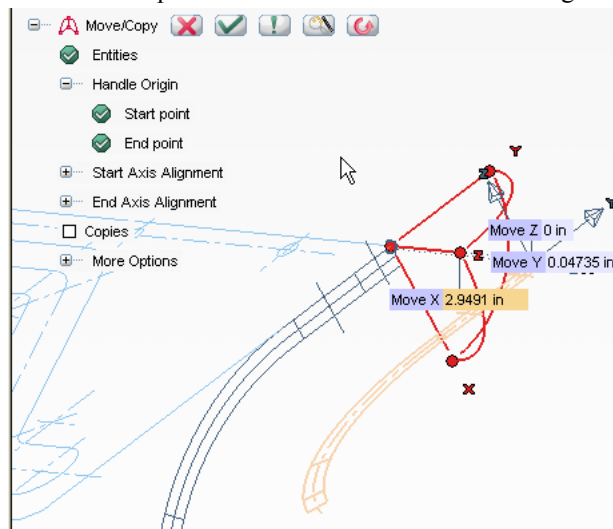

Now let's select the side view of the drawing and orient it for 90 degrees in x direction using move/copy entities command.
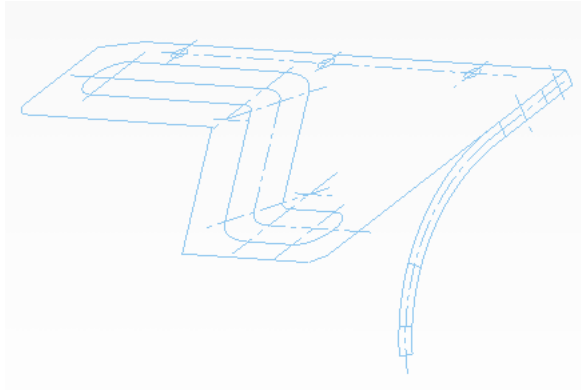
- Select the side view of the drawing

- Reset the start point as shown in the image below

- click and drag the handle xz or enter 90 deg in the mini dialog box appears.
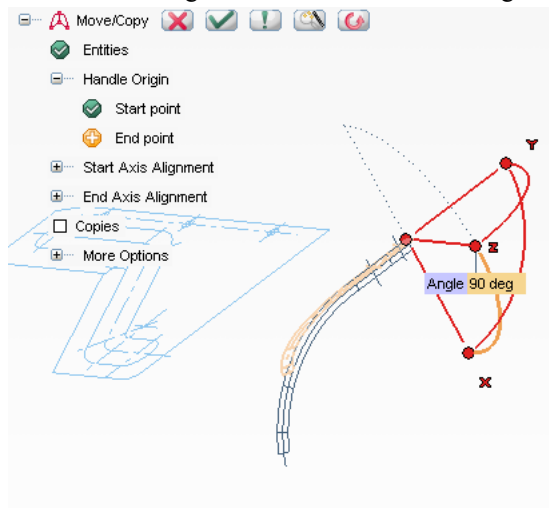
Select the endpoint as shown in the front view to align both the views.


The aligned view looks like this.

Right click on the workplane and select move in the context menu and place it on any point of the side view and then orient the view using ALT and up/down or left/right arrow keys and place the workplane clicking on view in the right click context menu of workplane so as to get the view oriented as shown.
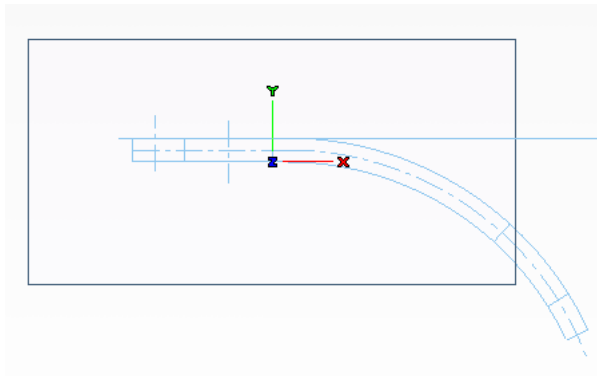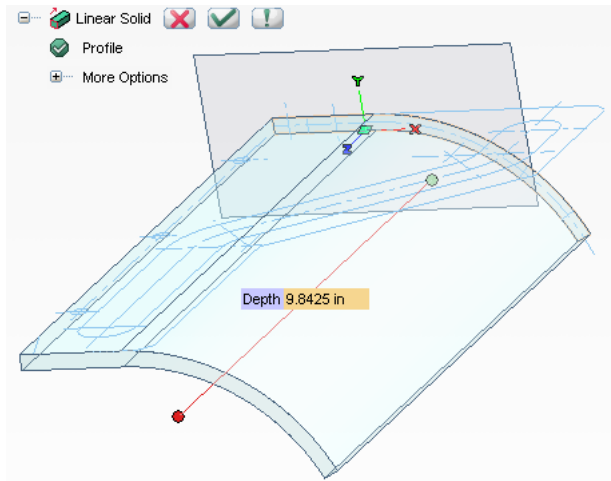

Select the chain of curves as shown and insert 2D profile using right click context menu or Insert profile 2D
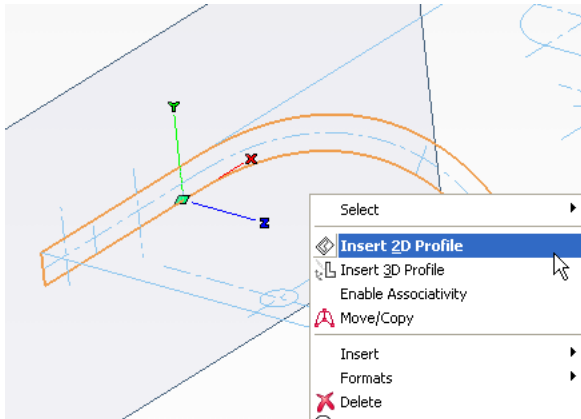
New paragraph. Replace this text.

Sweep a solid from the selected geometry with **Linear Solid**.

• Start **Linear Solid**.

• Click and drag the red dot dynamically and place it to the end point of the other end of the front view.

- Hit OK to create the solid

- Rotate the view to take a look.

3. Step 3: Sweep a Slot

In this step, we'll use some sheetmetal commands also to get the slots done on the solid using curves from front view of the drawing. we'll use the view geometry to cut some slots in the base solid.
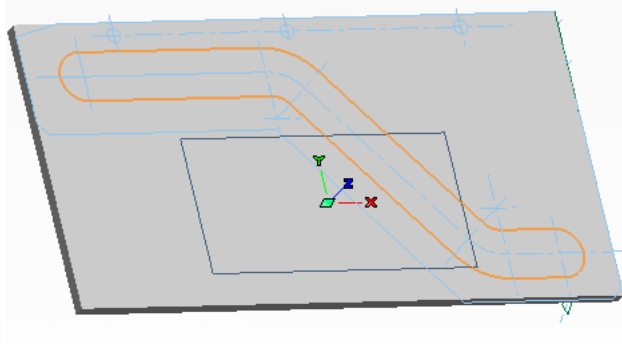
- Activate the command **Insert⇨Sheet Metal⇨Unbend**

- select the flat face for fixed face/edge as shown

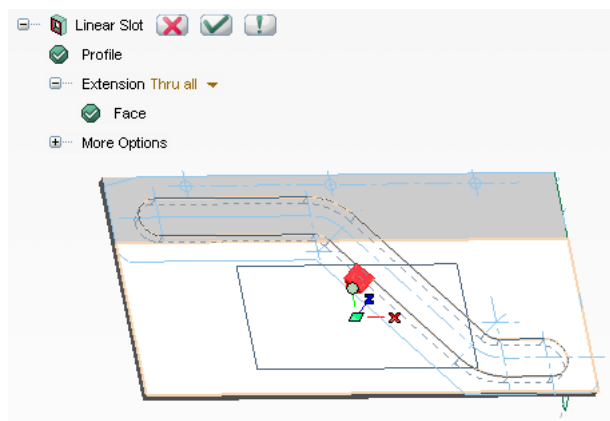- See the preview and hit OK to the command.
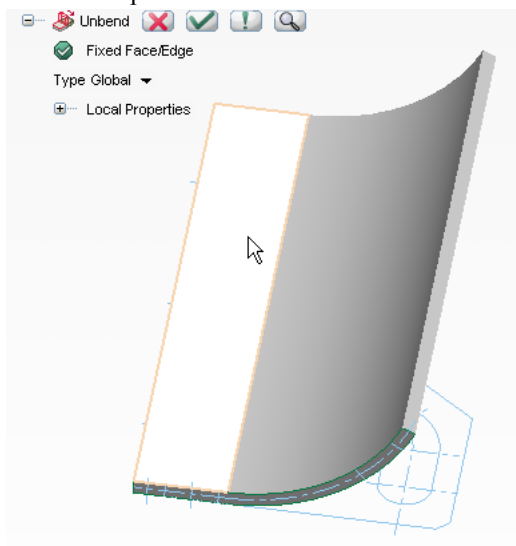
Place the workplane on the face as shown using right click and work plane here in context menu.
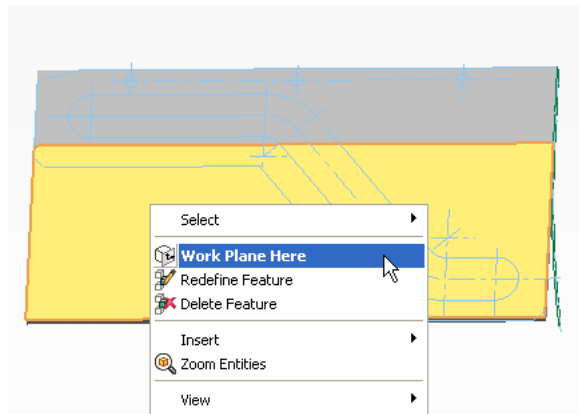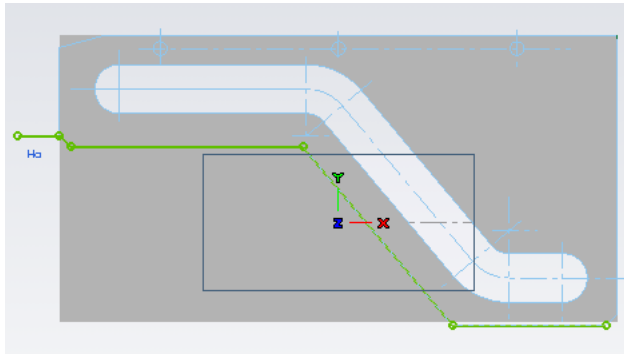
Select the closed loop of curves from the front view of the drawing to make a slot.
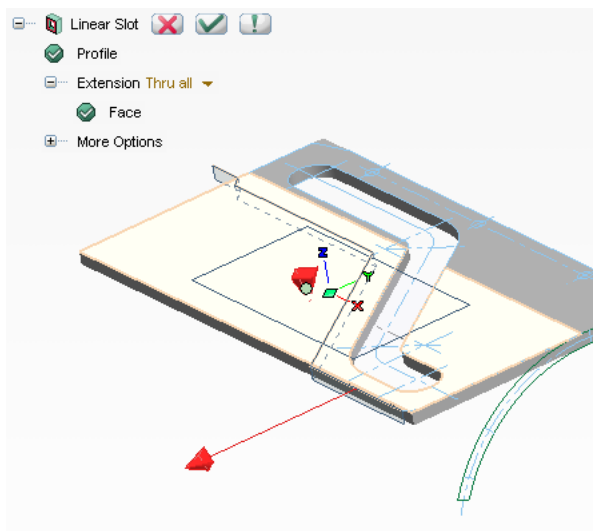

- Start the **Linear Slot** command

- Make sure the Extension is set at Thru All in the Selection List

- Select the face of the base solid as shown

Next,Select the set of curves and make it as an open profile to make one more linear slot.
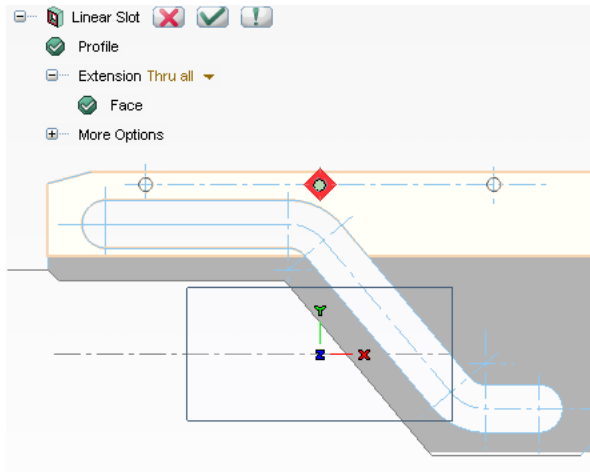
- Start the **Linear Slot** command

- Set the extension thru all in the selection tree

- Select the face of base solid as shown and make sure the arrow pointing towards outside
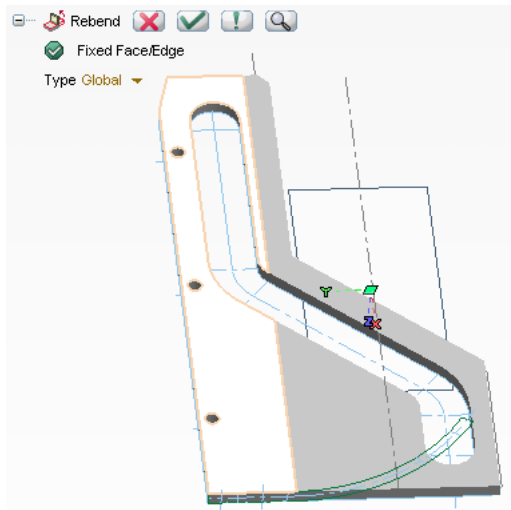

Repeat the same linear slot command to make a slot at the edge selecting the edge curve as profile as shown.
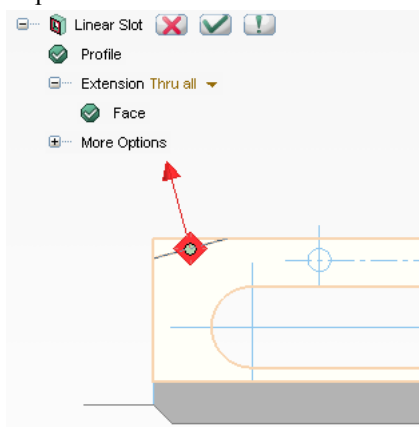
Select the three small circles as profile and make the linear slot with thru all option.
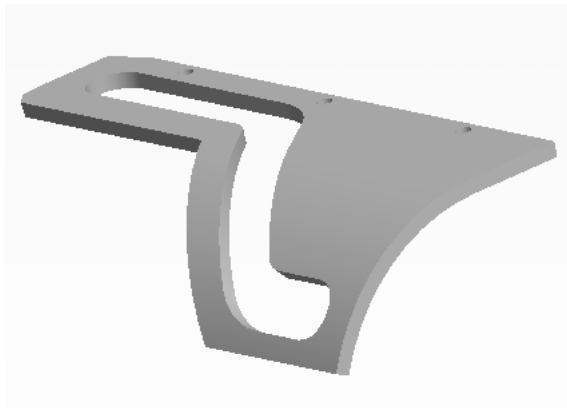
Now we are done with all the slots. We apply **Insert**↳**Sheet Metal**↳**Rebend**and select the same flat face selected for unbend to get the required shape as per the drawing.

Hide all the curves and profiles using **Hide Entities** command.

Congratulations!!. We finished the 3D model as per the 2D data 1 camma.dwg. Final model looks like this.
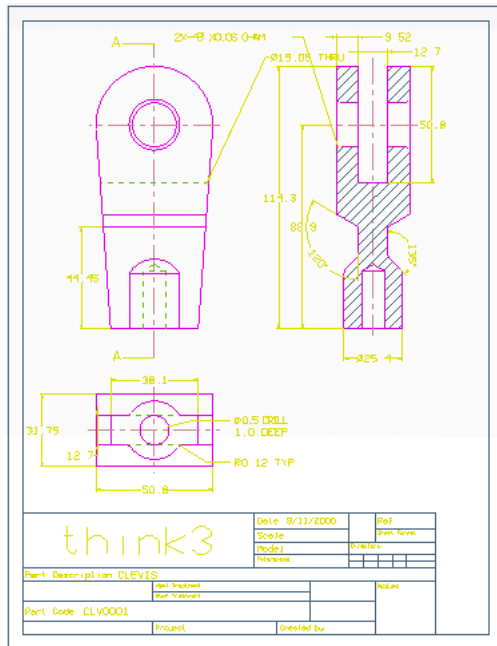

4. Step 4: Another Approach.

In this step we will try to make a 3D model using ConnectingRodEnd.dwg with a little different approach than the 1 camma model.

Repeat Step 1 to remove overlaps and compress the drawing and get the ConnectingRodEnd.dwg into 3D model

environment using **Insert From File**

As you can see, there are several layers in this file. Let's try turning off a few of them.

- Pick any part of the border. Notice that which layer it belongs to. Switch off the layer 6 by clicking the bulb icon in layer manager tab in history tree.

- Switch off layer 4 and click for Input layer 0

Now let's use the front view to create the base solid by selecting the geometry we need. We'll start by using Select Chain to get the boundary, then we'll add the hole and the center line.

- Select the boundary of the front view by Select Chain.

- Hold the Shift key and pick the inner circle (not the chamfered edge) to add it to the selection set.

- Keep holding the Shift key and select the vertical center line. We'll use the center line as a reference line later.
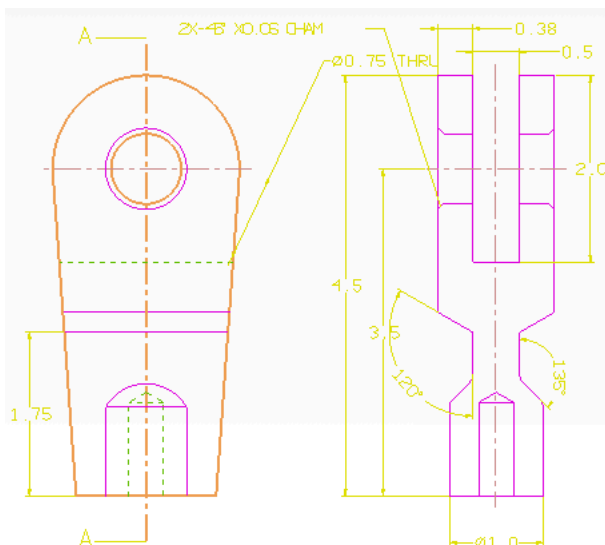
Sweep a solid from the selected geometry with**Linear Solid**

- Start **Linear Solid**

- Set the depth as Depth1.25 in

- Right click on the depth Mini Dialog and hit Symmetric in the context menu. The Symmetric option sweeps the depth symmetrically about the Work Plane, half the depth above the plane and half the depth below
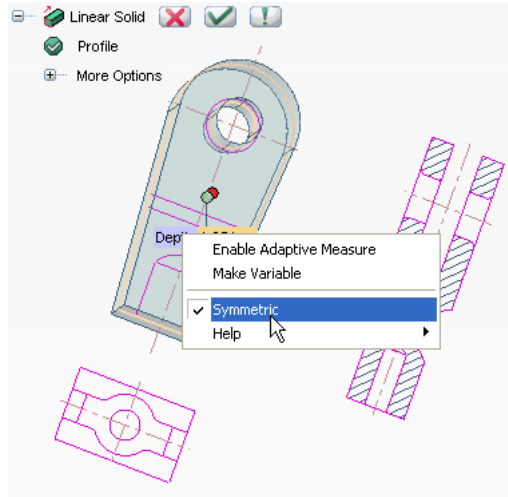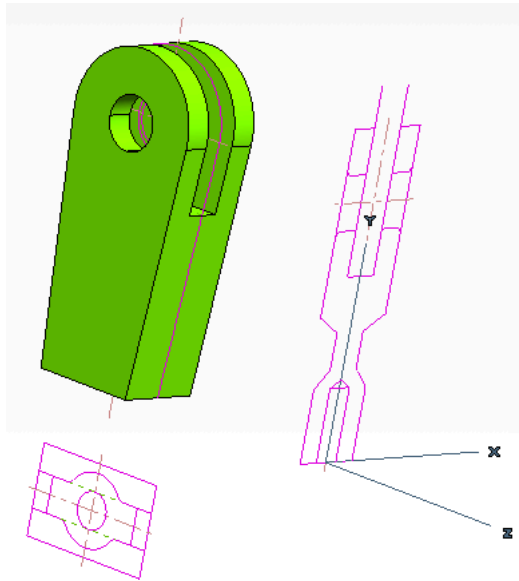

- Hit OK to create the solid

- Rotate the view and take a look.
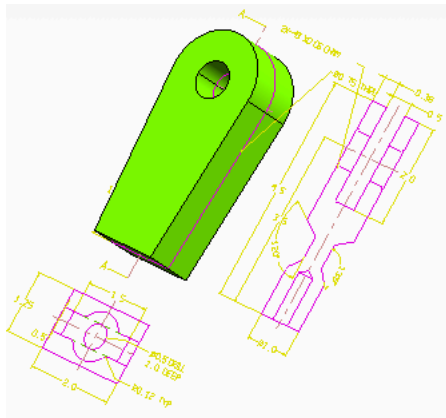
Next, we'll project the slots from the side view onto the new solid

we'll use the section view geometry to cut some slots in the base solid.

Before we can project the section view geometry onto the base solid, we need to rotate the entire view about the centerline. This is a job for **Move Copy Entities**.

- Modify the model view to **View on Work Plane**.

- Pick **Edit↘Move/Copy** from the pull down menu OR Select the **Move Copy Entities** button from the Edit toolbar. .

- Select the section view geometry with a crossing window select.

NOTE:

Any time a selection calls for multiple objects, you can use the selection window. Just click and hold the left mouse button then drag the window across the objects you want to select. Dragging from right to left selects everything inside the window and everything the window crosses, hence the name crossing window. Dragging from left to right selects only those objects completely inside the crossing window.
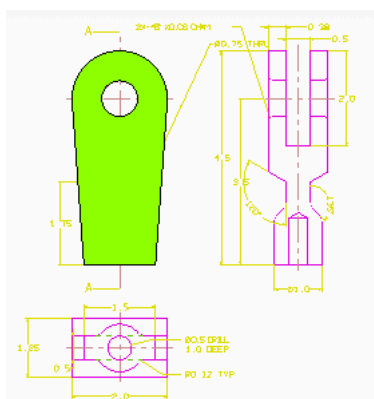
Odds are you selected some of the dimensions from the front view with that crossing window.

- Hold the ⎡Ctrl⎤and pick any objects from the front view that were selected by the crossing window.

- Make sure the green depth dimension on the solid is not selected.

Switch to the **Right Front View**. Now set the Rotate options:

- Expand the Handle Origin option in the Selection List.

- Hit the Start Point option.

- Pick the intersection of the vertical center line and lower horizontal line. .
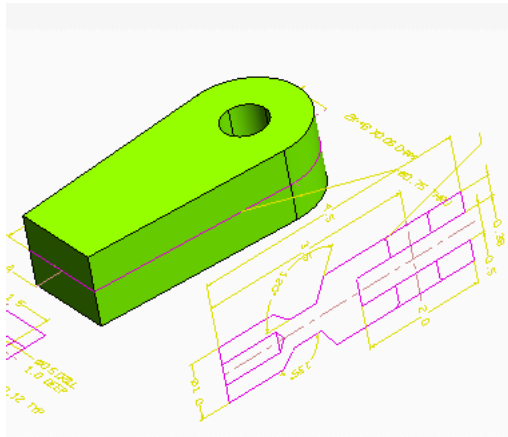
NOTE:

In this case, the midpoint of the base line is the same point as the intersection. Either point will work.
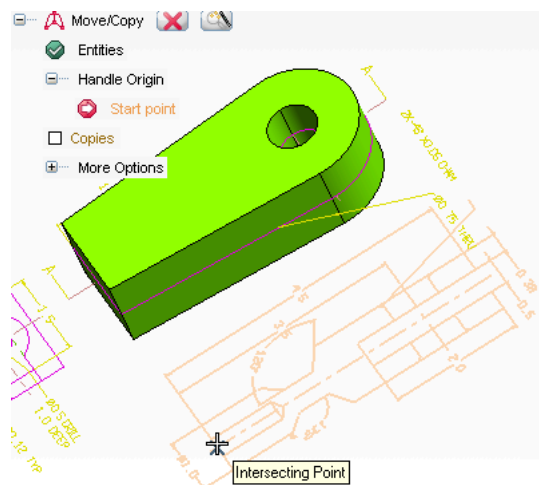
- Set the Angle90 deg.

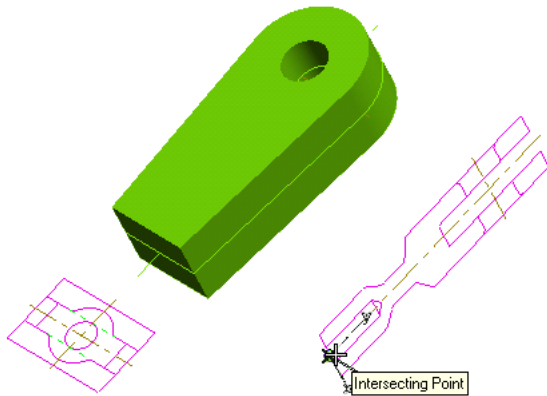- Click OK to rotate the section view geometry.
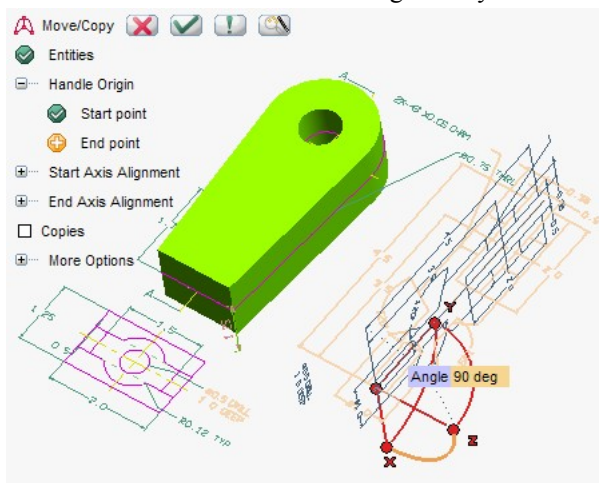
We really don't need to see the dimensions for a while, so let's turn them off.

Pull down the layers in layer toolbar and click on the bulb icon to swich of the layers dimension

Now we need to move the Work Plane to the section view so that we can edit the geometry a bit.


- Click once on the workplane to get the edit mode and you can see the Arrowheads and rotater .Rotate the Work Plane 90° about the Y axis and hit OK.

With the Work Plane in position we can start on the upper slot using the section view geometry. The first step is to extend the lines a bit to make a clean cut.

Double click on the first line, about 1/4 to 1/3 of the way down from the upper end. Zoom in a little to get a better view if needed.

This starts the Edit Entities command. Double clicking on the endpoint of a line edits the length and angle. Double clicking back from the endpoint edits just the length, leaving the angle fixed.
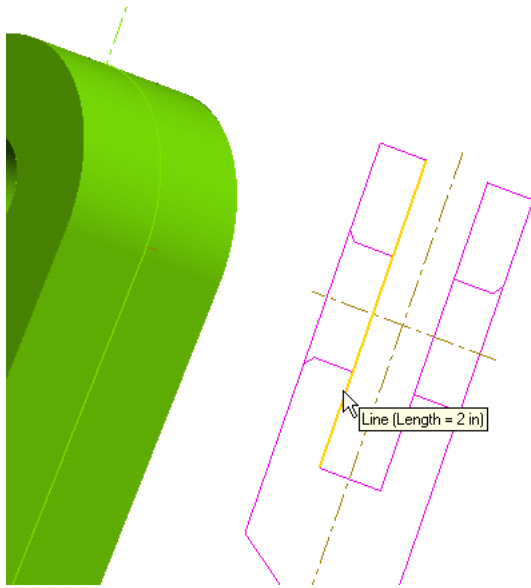
Drag the line endpoint past the corner to a length of 2.5 or so and pick the point. (The actual length isn't critical; we just need a little wiggle room to work with.)
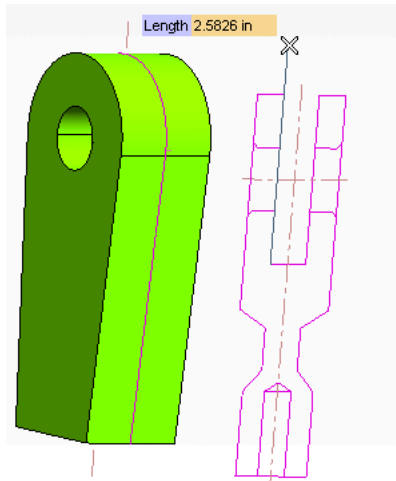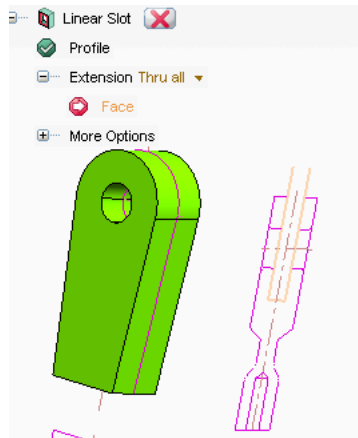

Repeat the procedure for the other line.
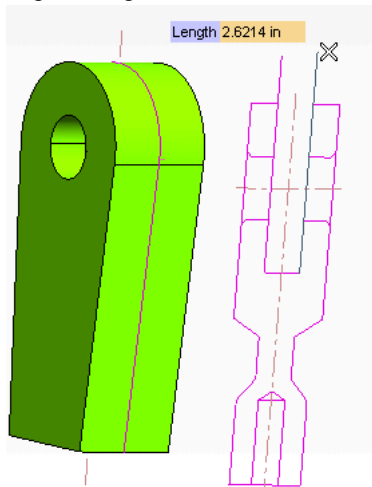
Now we'll reselect the lines and sweep the **Linear Slot**.

- **Select Chain** one of the lines you've just extended.
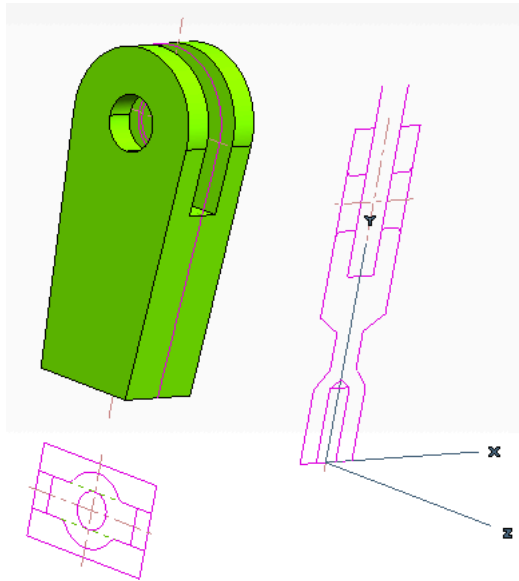
- Start the **Linear Slot** command.

- Make sure the Extension is set at Thru All in the Selection List.

- Select the side Face of the base solid. .

Make sure the direction arrow is pointing into the slot. If it's not, double click on it to invert the direction.
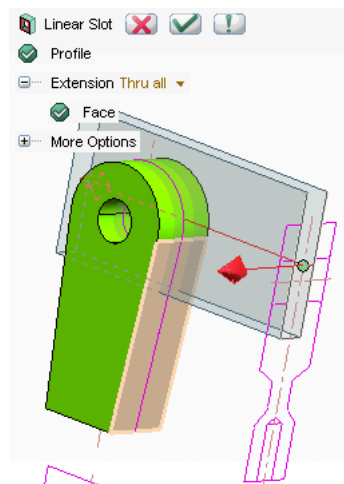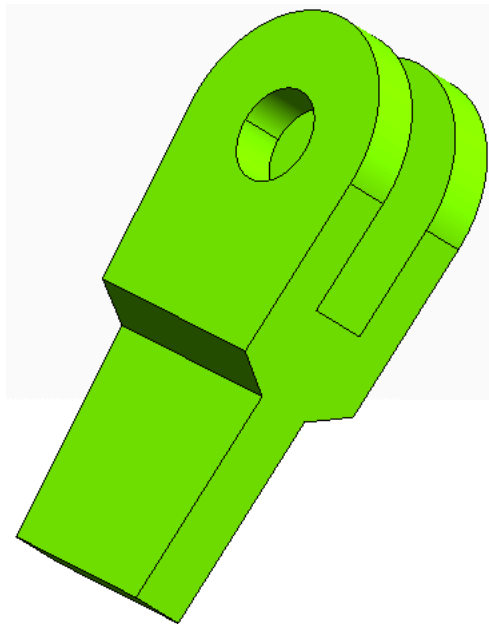

Hit OK to cut the slot.

Nice work! In the next step, we'll add the lower cut.

In this step, you'll sweep a second **Linear Slot** and use **Mirror Solid** for the feature.
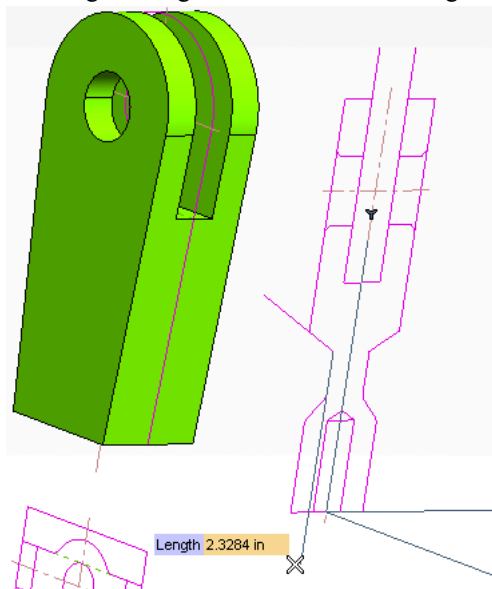

For starters, let's extend the lines that make up the next slot.

- Double click on the angled line, about 1/3 of the way from the endpoint.

- Drag the line out to a length of about .5 Length0.5 in

If you have invoked both a Length and a Angle minidialog, keep the Angle set at 150.
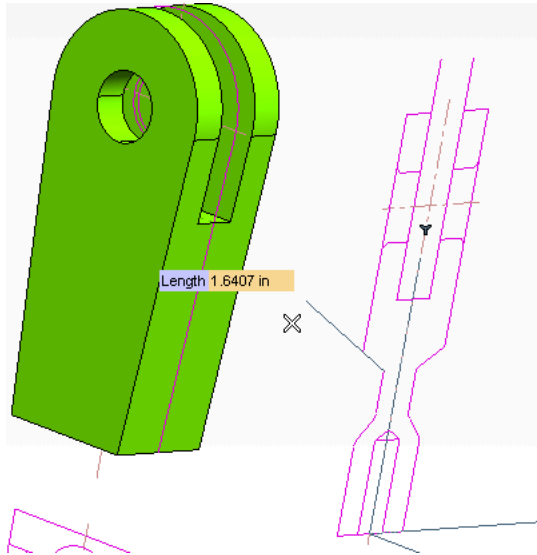
- Double click on the lower third of the vertical line.

- Drag the length down to about 2. Length2 in

Now select the two lines we've just extended and sweep another **Linear Slot**.

- **Select Chain** one of the lines you've just extended.

- Start the **Linear Slot** command.

- Select the side Face of the base solid.

- If the Extension is not already set to Thru All in the Selection List, set it now.

- Make sure the direction arrow is pointing out of the slot. (If it's not, double click on it to invert the direction.)

- Hit OK to cut the slot.
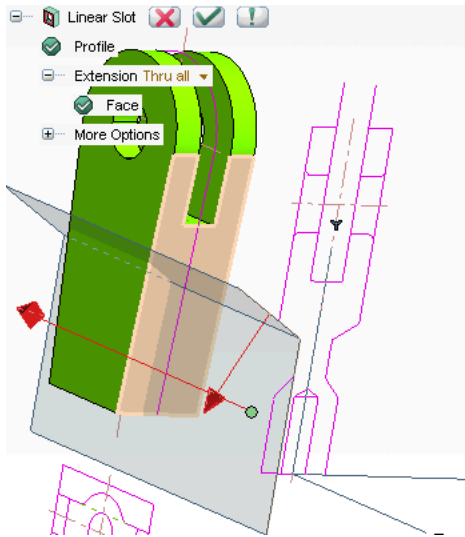
The slot on the back of the part is the same as the one we just cut. Instead of cutting a second slot, we'll use **Mirror Solid** for this one. That way, if we ever change the first slot, the mirrored copy will change, too.

- Start **Mirror Solid** from the toolbar or menu

- Select the face from the slot we just swept as the Base Entities to mirror.

- Change First Symmetry Plane Through axis & point in the Selection List.
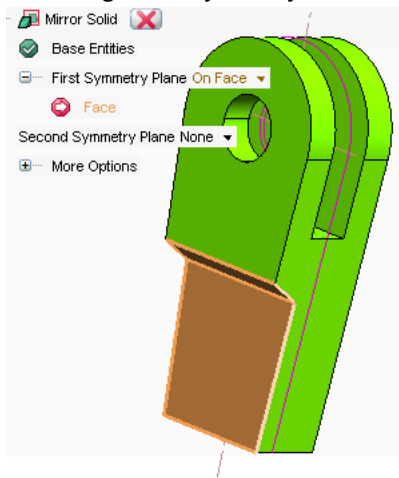
To mirror in 3D space, you need to define a plane about which to mirror (as opposed to just a line in 2D mirror). The Reference Plane options list offers several methods for defining this plane. We don't have a face we can use, but we do have an axis line and a point. We'll use them to define the plane.
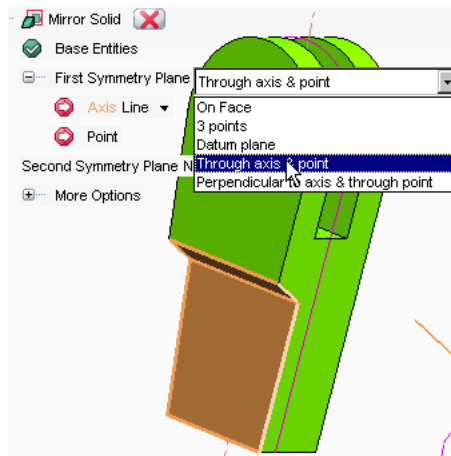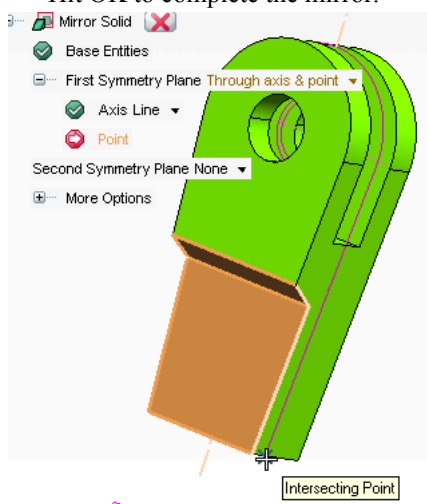
- Select the centerline you included in the original profile as the Axis line.
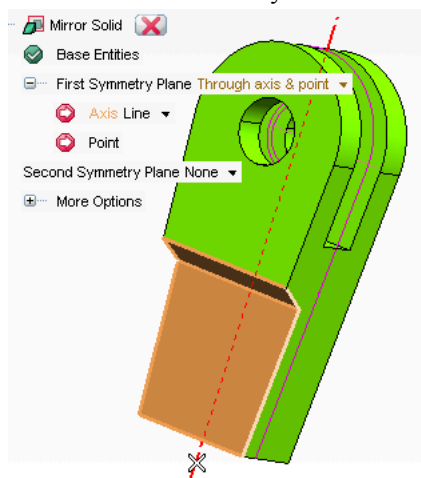
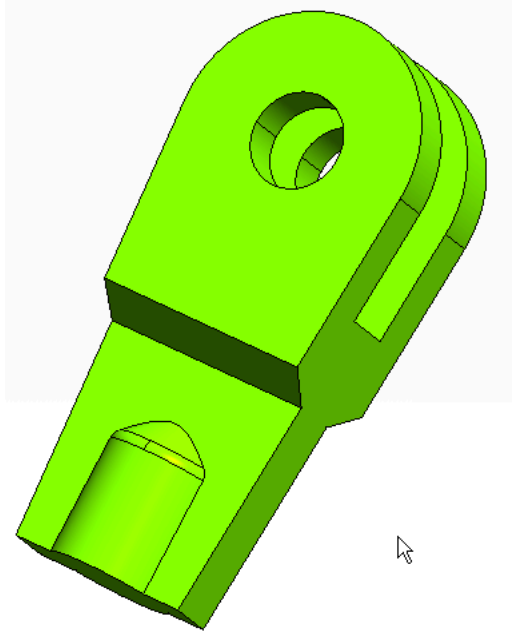- Pick the corner point of the original profile as the Point.

- Hit OK to complete the mirror.

In the next step, we will revolve the boss.

5. Step 5: Create the Boss

In this step we'll finish up with the section view geometry by adding the boss using the **Rotational Protrusion** command..


We need to extend some lines for the boss. Rather than just extending them in an unconstrained fashion, we'll use the **Edit✏Trim/Extend Curves**command. This command doesn't just trim, it extends, too!

- Toggle off **Show Work Plane**

- Start **Trim/Extend Curves**.

- Set the Mode to Both.

- Pick the upper end of the angled line of the boss.

- Pick the lower end of the centerline.

Notice that the centerline is trimmed and the angled line is extended.
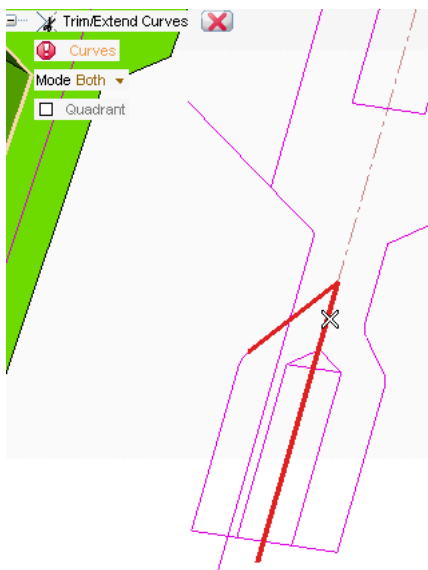
- Pick the left side of the base line.

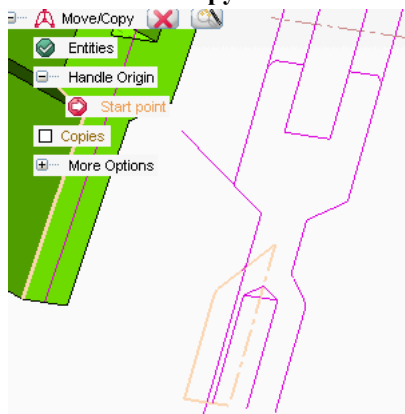- Pick the upper portion of the centerline.
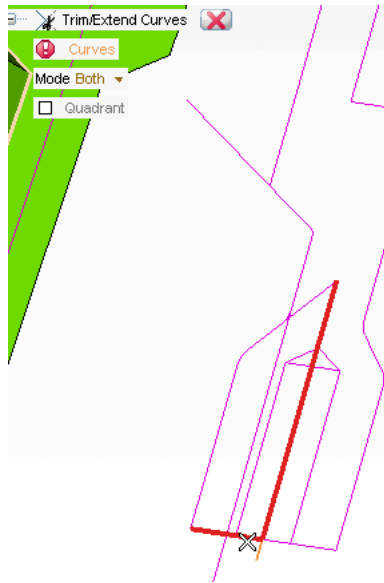

This time, both lines are trimmed.

End the command with `Esc` or **Select Entity**

Since this feature is a revolved sweep, we need to move the geometry into a appropriate position before we create the boss. **Move Copy Entities** will do the trick.

- Select the geometry by **Select Chain**.
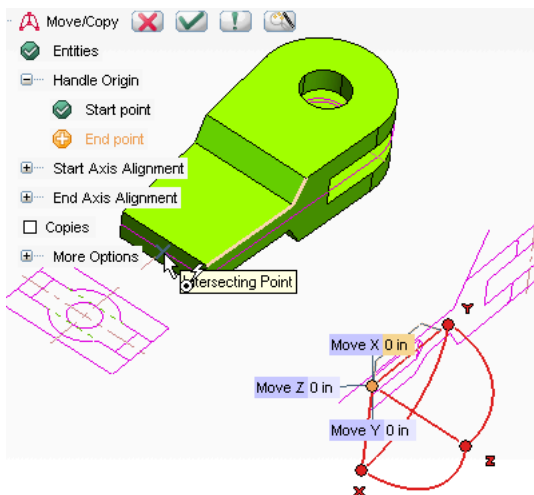
- Start **Move Copy Entities**.

Pick the lower end point of the centerline as the Start Point for the move.
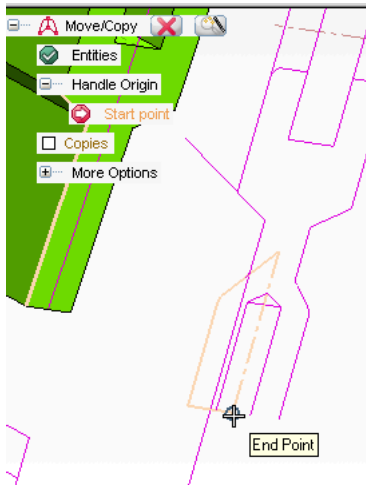
- Pick the intersection of the centerline and the baseline of the original profile to move the geometry to the End Point.

The minidialogs for translation along an axis appear, but we'll just click the End point instead of entering a value.
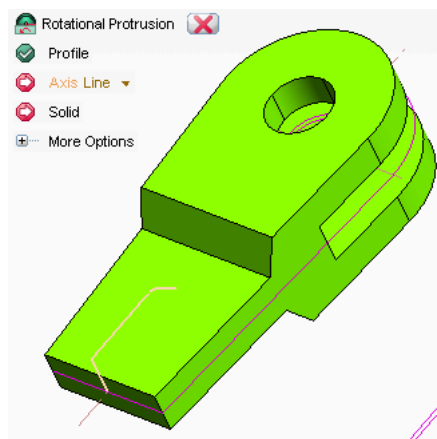

- Hit OK to complete the command.

Before we can revolve the boss, we need to move the Work Plane to match the boss profile..

- Toggle the work plane back on using **Show Work Plane**.

- Right click on the Work Plane and hit Move.

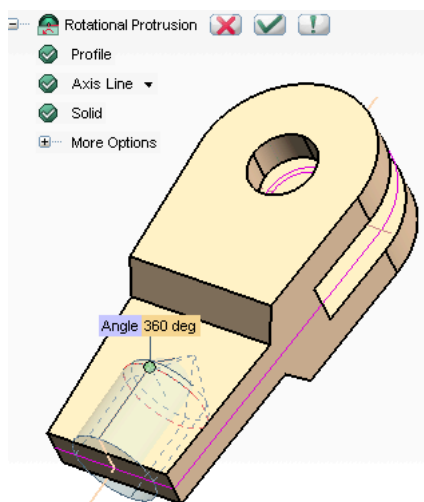- Pick the same point we used to move the boss profile.
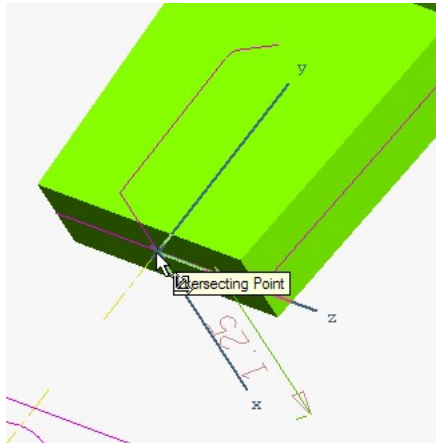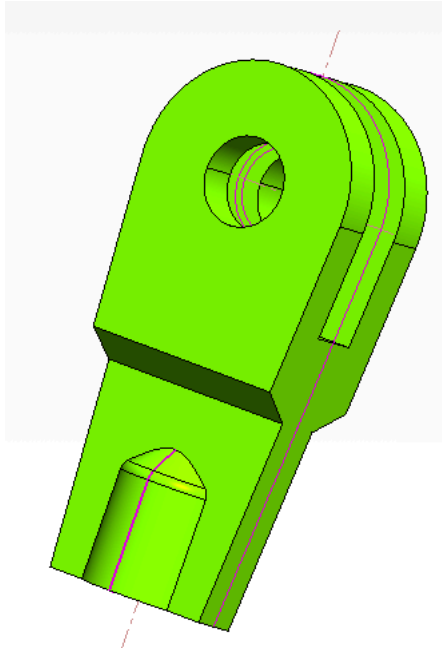
Sweep the boss with **Rotational Protrusion**.

- Change the linetype of the centre line from linetype4 to linetype1.

- Select the boss by **Select Chain**.

- Start the **Rotational Protrusion** command.

- Select the centerline as the axis Line.

- Select the Solid.
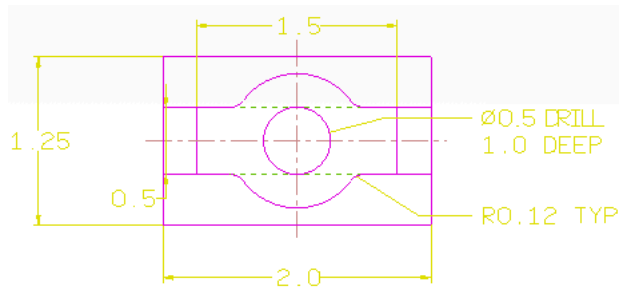
- Hit OK to complete this feature.

We're almost done; just a hole, some fillets and a chamfer or two to finish.

6. Step 6: Add a Hole

Now we'll add a simple hole with **Hole** command.

In case you don't remember the size of the hole, you can turn the dimension layer back on to check.

Click the bulb Icon of layer 3 in layer pull down menu to switch it ON.

Click the bulb Icon of layer 3 in layer pull down menu again to switch it OFF.

The **Hole** command offers a number of options including tapered, countersunk and counter bored holes. In our case, we need a simple hole.

- Start the **Hole** command.

All the hole shape options are located in the Hole Type drop down list, and the default shape is Simple. This dialog box contains the settings that determine the size and shape of the hole.

- Check for the hole type: Simple.

- Select the extension as Blind.

- Select the bottom of the part as the Face on which to place the hole.
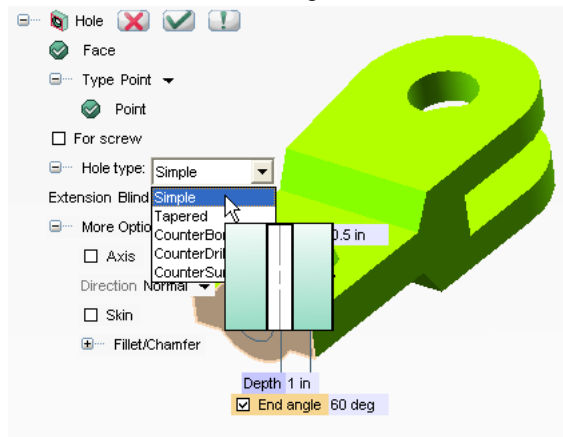
As you've probably noticed, the Point is automatically selected when you pick the face. There are a couple ways to clear it.

- Right click on Point in the Selection List and pick Reset from the context menu.

- The Point is cleared and we're ready to select a new one. Snap to the center of the boss.
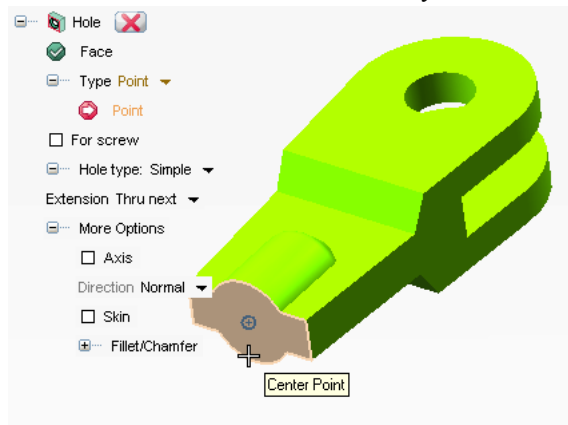
- Set the Diameter to 0.5.

- Set the Depth to 1.

- Click on more options and check the end angle.

- Leave the Angle at the default of 60 deg.

- Hit OK to insert the hole. .
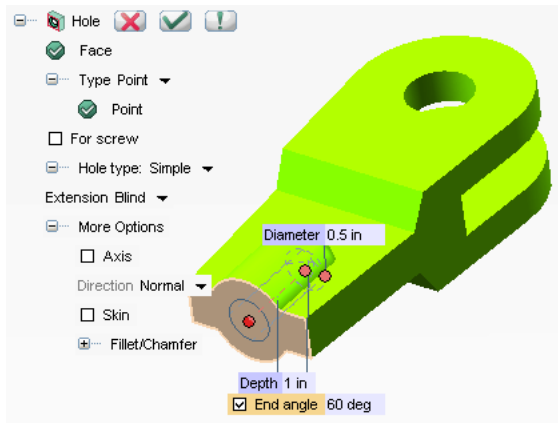
Almost there! In the last step, we'll add the fillets and chamfers.

7. Step 7: Chamfer and Fillet

In this last step, we'll finish the connecting rod end by filleting the boss and chamfering the hole.

Let's start with **Fillet Edges** around the boss. The size of the fillet is 0.12, but you can turn on the dimensions again to check if you'd like..

- Right click on one of the edges of the boss and pick Insert **Fillet Edges**.

- Set the fillet as Radius0.12 in

- Rotate the model and select the same edge on the opposite side of the part.

- Hit OK.
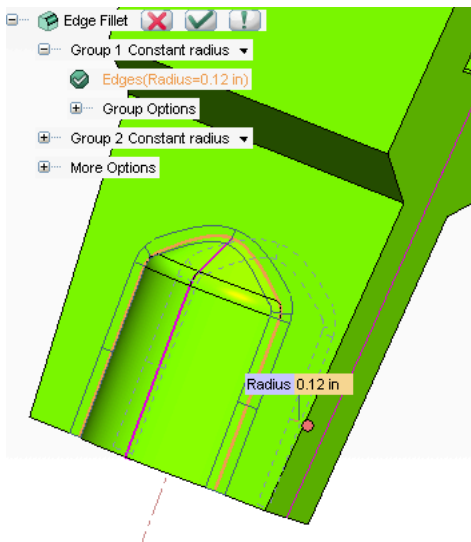
Now we'll use **Chamfer Edge** to chamfer both sides of the through-hole. **Chamfer Edge** works a lot like **Fillet Edges**, so it shouldn't be too hard to figure out. The original chamfer was .06 x 45°; we'll use those same values.

- Right click on the circle representing the edge of the through-hole.

- Pick Insert **Chamfer Edge** in the context menu. .

- Set the Distance0.06 in
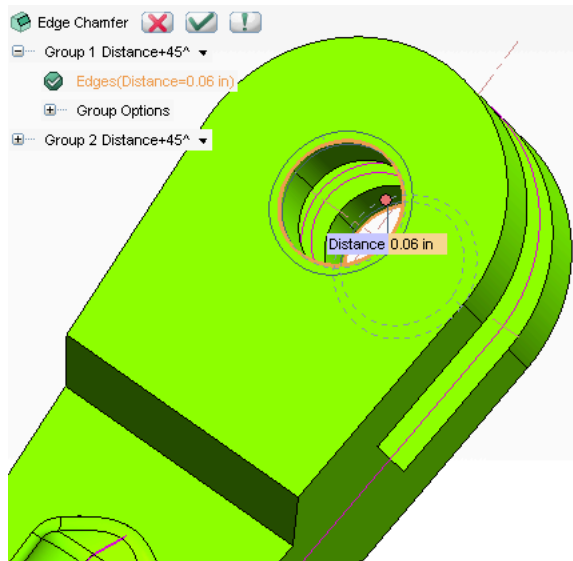
- Rotate the view and pick the circle on the opposite side.

- Hit OK.

Nicely done!! Use the **Hide Entities** option to clean up the profiles and dimensions.