

ANSYS 12 - Crank - Step 2

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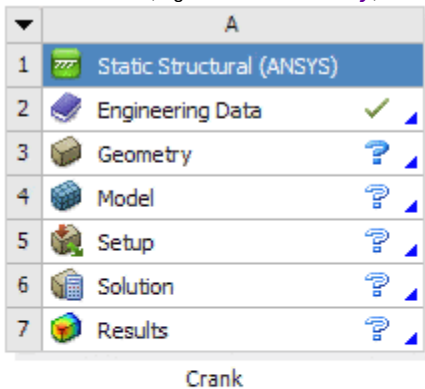
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Step 2: Geometry

Note: You can import geometry from Solidworks. Just make sure that when you create your geometry in Solidworks, you have the Crank body and Handle body.

In the **Crank** cell, right click on **Geometry**, and click **New Geometry...**



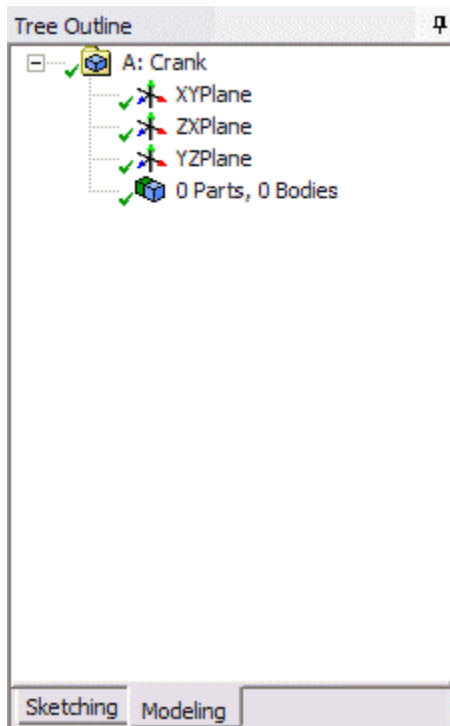
At this point, a new window, ANSYS Design Modeler will be opened. You will be asked to select desired length unit. Select **Inch** unit and click **OK**.

Creating a Sketch

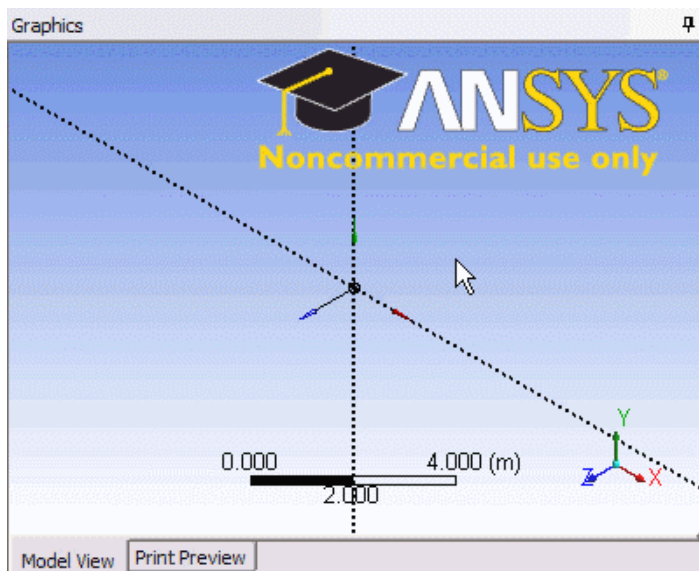
Like any other common CAD modeling practice, we start by creating a sketch.

Start by creating a sketch on the **XYPlane**. Under **Tree Outline**, select **XYPlane**, then click on **Sketching** next to **Modeling** tab. This will bring up the **Sketching Toolboxes**.

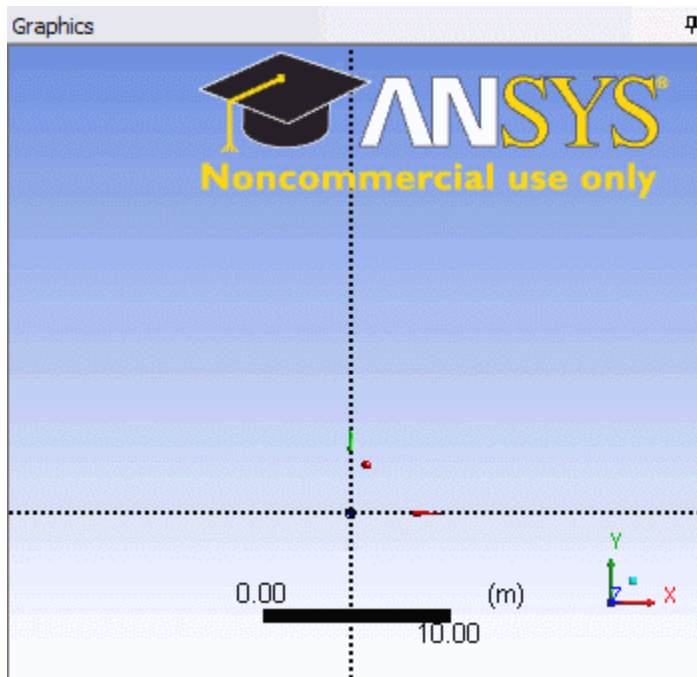
Note: In sketching mode, there is **Undo** features that you can use if you make any mistake.



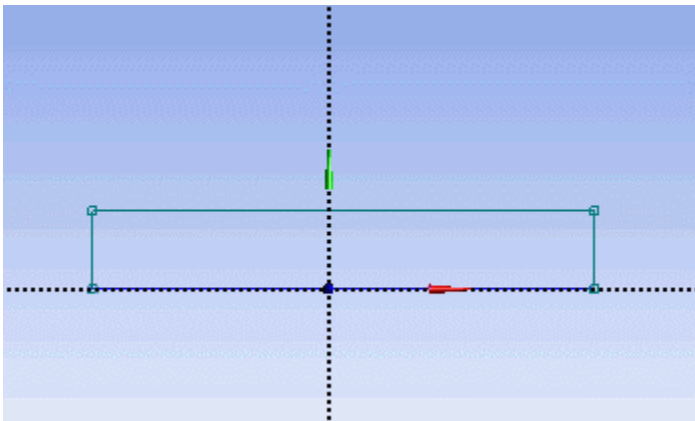
On the right, there is a **Graphic** window. At the lower right hand corner of the Graphic window, click on the **+Z** axis to have a normal look of the **XY Plane**.



Under **Sketching Toolboxes**, click on **Draw** tab. We will first create the outline of the crank. Select **Rectangle** and draw in the **Graphics** window as shown below. Make sure that the rectangle is coincident with the x-axis (Letter C will appear).



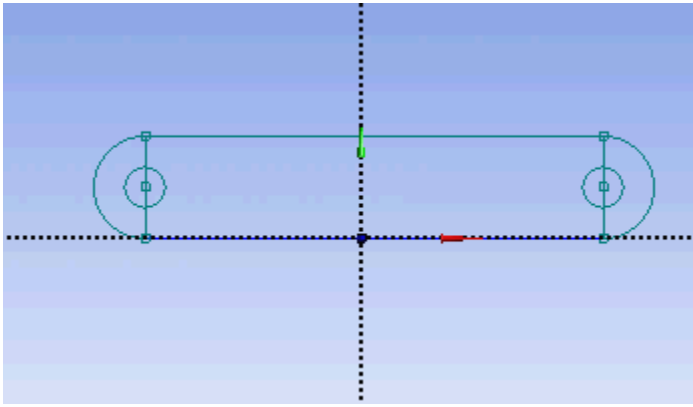
Note: You do not have to worry about dimensions for now, we can dimension them properly in the later step. Next draw the semi-circular arc at both end of rectangle. Under **Draw** Tab, select **Arc by Center** and execute the drawing as shown.



Draw the crank circular hole next. Under **Draw** tab, click on **Circle** and execute the drawing as shown.



Next, we will dimension the current drawing. Under **Sketching Toolboxes**, select **Dimensions** tab. You will see a list of dimensioning options. We will use the default **General** option. Execute the geometry dimensioning as shown below.



Input value for D2, H3, L1 respectively under **Details View** tab.

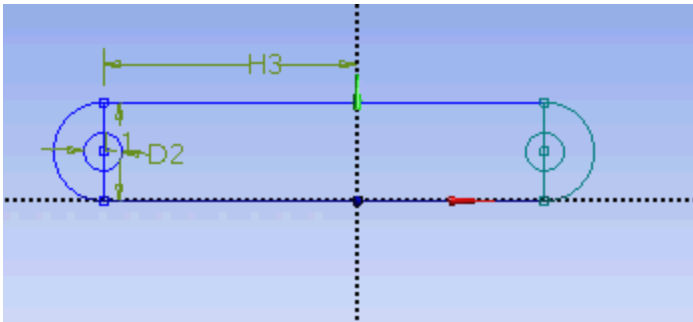
D2 = 0.5 in

H3 = 3.3465 in

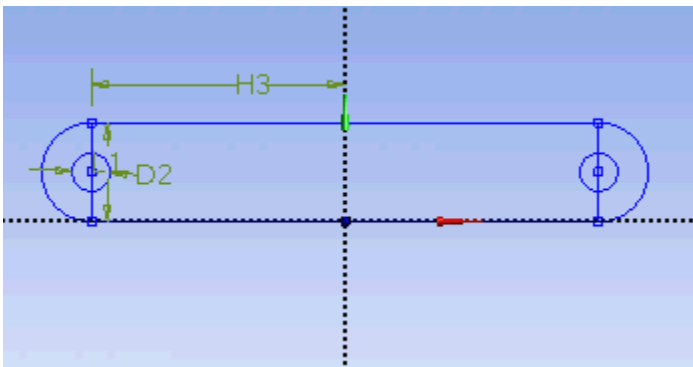
L1 = 1.299 in

Note: Your dimension naming might not be the same as the one shown here. It is fine. Just make sure your value correspond to correct dimensions.

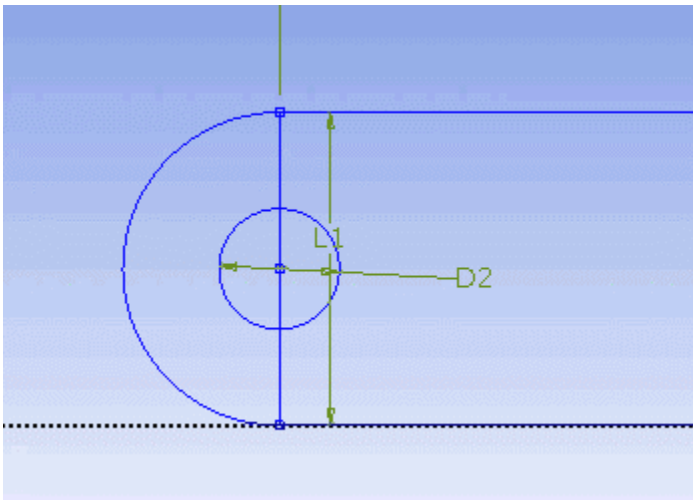
Next, we need to make sure that the sketch is symmetric on both side. Under **Sketching Toolboxes**, select **Constraints** tab. Select **Midpoint** and constrain the geometry in the **Graphics** window as shown below.



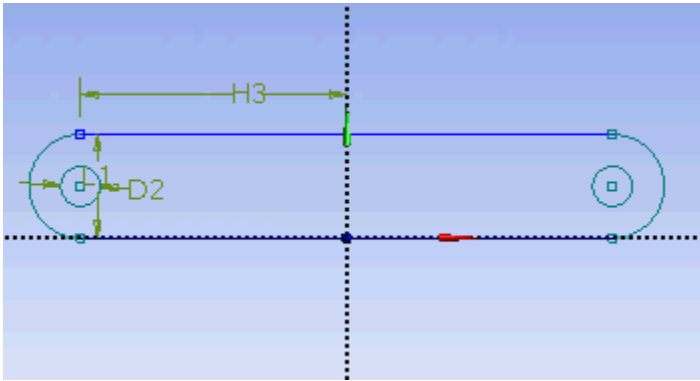
Next, we will make sure that the both circular hole is of same dimension. Under **Constraints** tab, select **Equal Radius**. Execute constraint in the **Graphics** window as shown below.



We will next trim off the unwanted edge. Under **Modify** tab, select **Trim**. Execute the trimming in **Graphics** window as shown below.



At this point, we have the outline of the crank done. We need to add the cut out in the middle of the crank. First start with a sketching of the cut-out. Under **Draw** tab, click on **Line** and execute drawing in Graphic window as shown below.



We would want the four corners of the cut out we created to be of following coordinate.

(-0.7972, 0.1642)

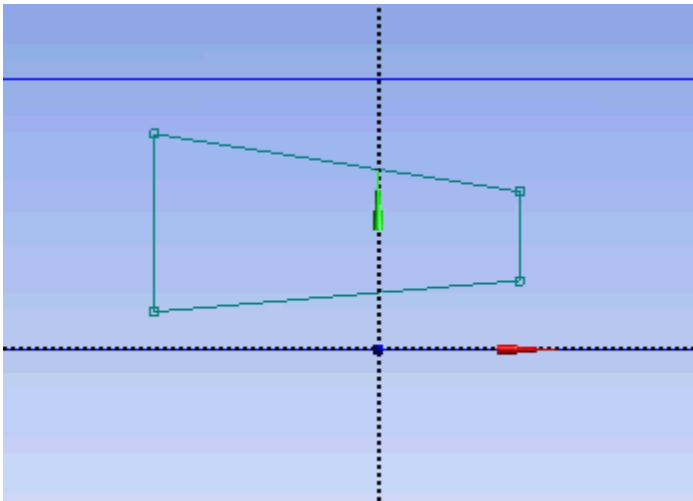
(0.7972, 0.3248)

(0.7972, 0.9744)

(-0.7972, 1.1368)

Unfortunately, there is no easy way in specifying the coordinates of a point. So we will have to dimension them accordingly using the dimensioning tools.

Click on **Dimensions** tab and use **General** tool and execute the dimensioning as shown.



Under Details View, enter the value for the dimensions.

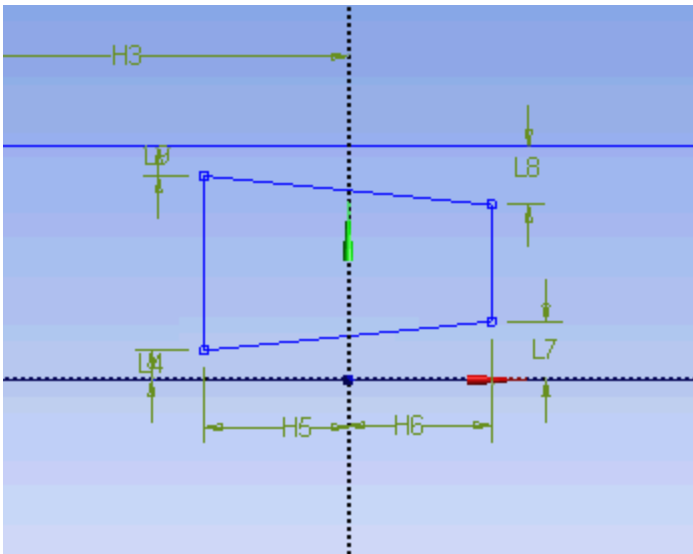
H5 = 0.7972 in

H6 = 0.7972 in

L4 = 0.1642 in

L7 = 0.3248 in
L8 = 0.3248 in
L9 = 0.1642 in

Next we will add fillet to the corner of the cut out. Click on **Modify** tab and select **Fillet**. Enter a **Radius** value of 0.177 in. Execute the command in **Graphic:** window as shown.

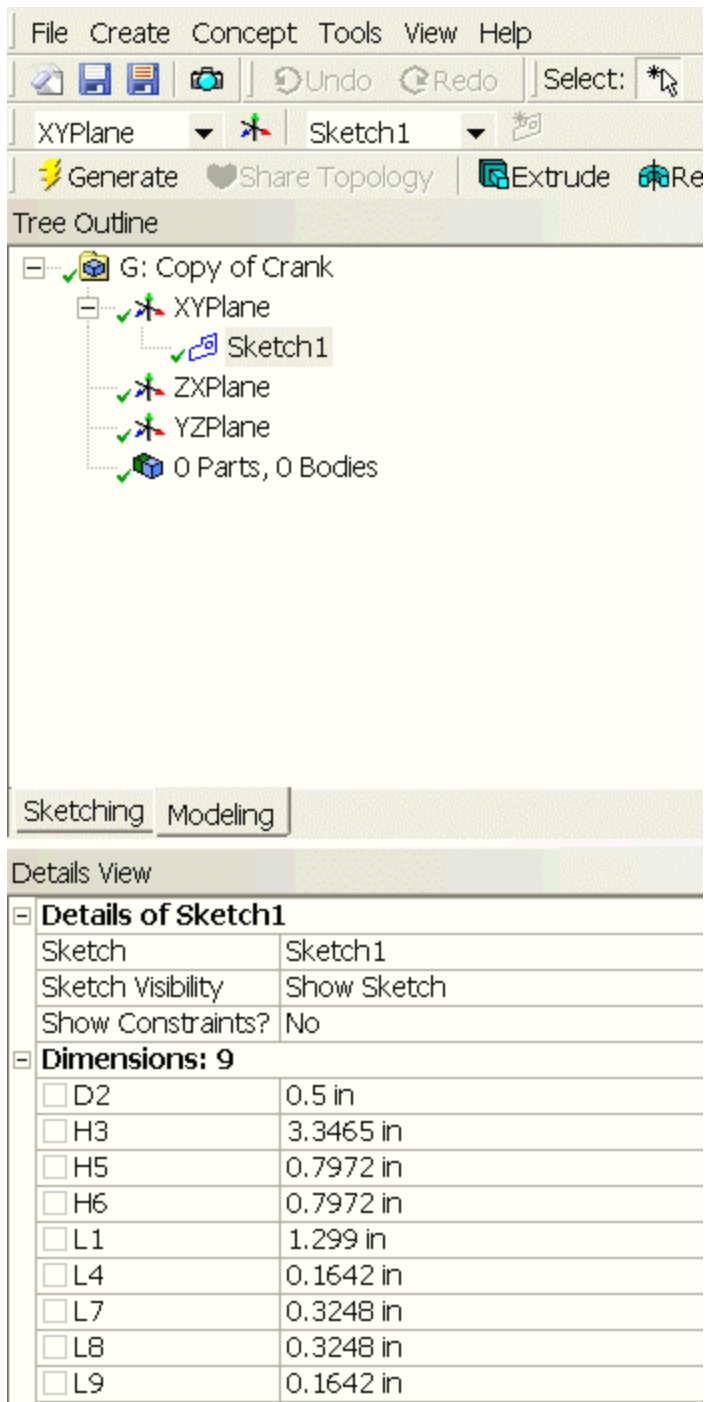


Now we have the sketch of the crank done. Next we will perform extrude feature on this sketch.

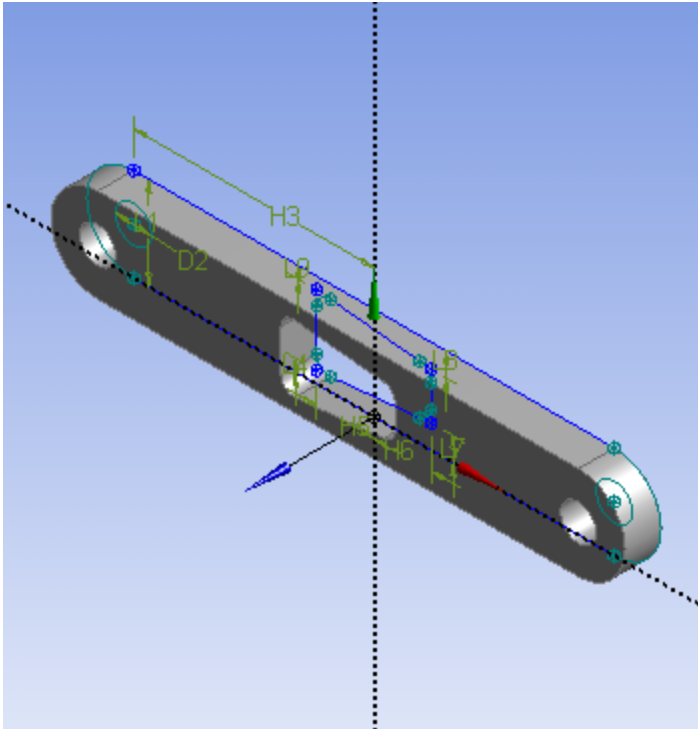
Create > Extrude

Under **Details View**, **Sketched1** is automatically selected as default **Base Object**.

Next to **FD1, Depth (>0)**, enter value 0.5 in. Click **Generate**.



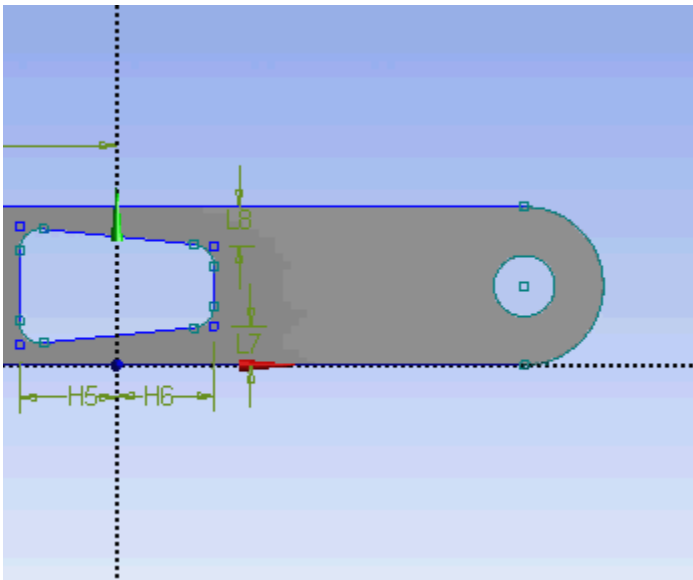
Note that after you click **Generate**, you see a new part created under the **Tree Outline**. This is what you should have in the **Graphics** window at this point.



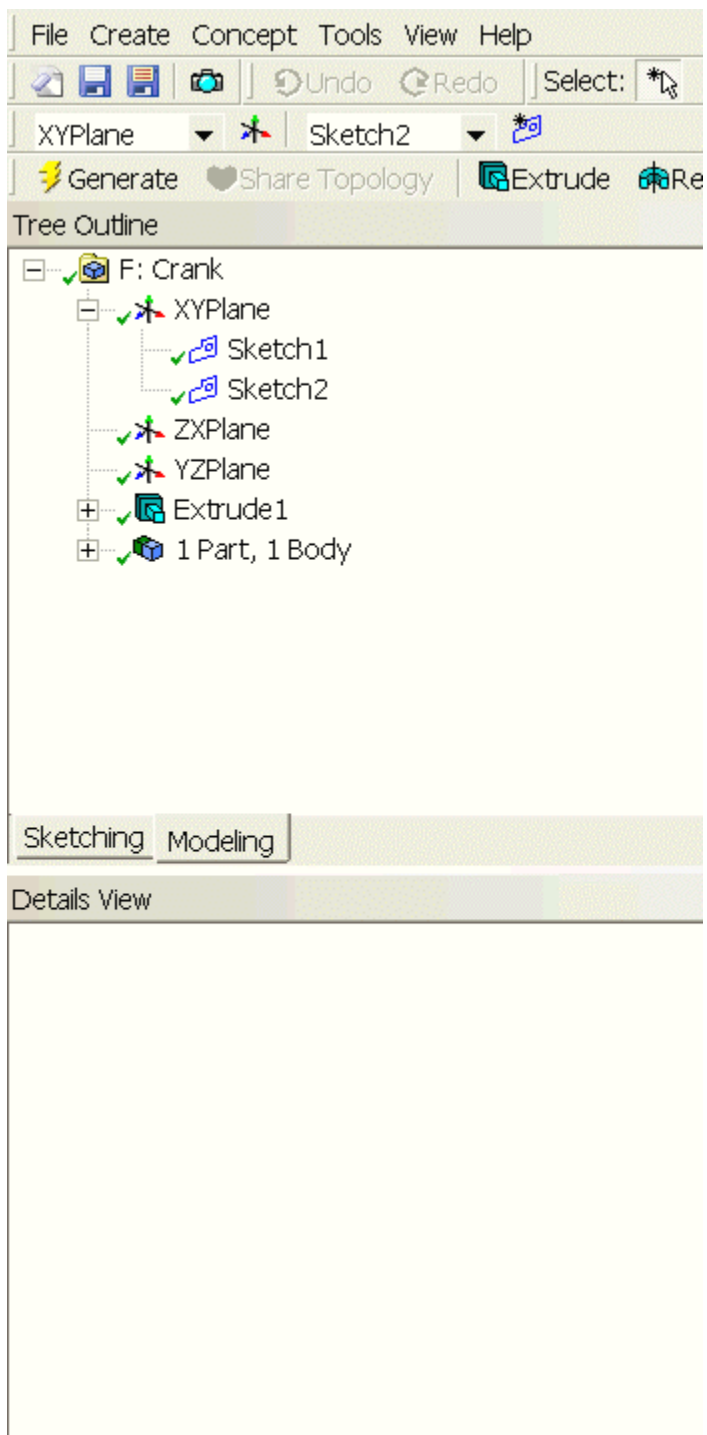
Now let's keep moving by adding a handle to this crank. Under **Tree Outline**, click on **XY Plane** and click on **New Sketch** . This will create a new sketch under **XY Plane**. You will see **Sketch2** created under **XYPlane**. Click on the **Sketching** tab to start sketching the handle. The base sketch for the handle is just a circle. Under **Draw** tab, click on **Circle**.



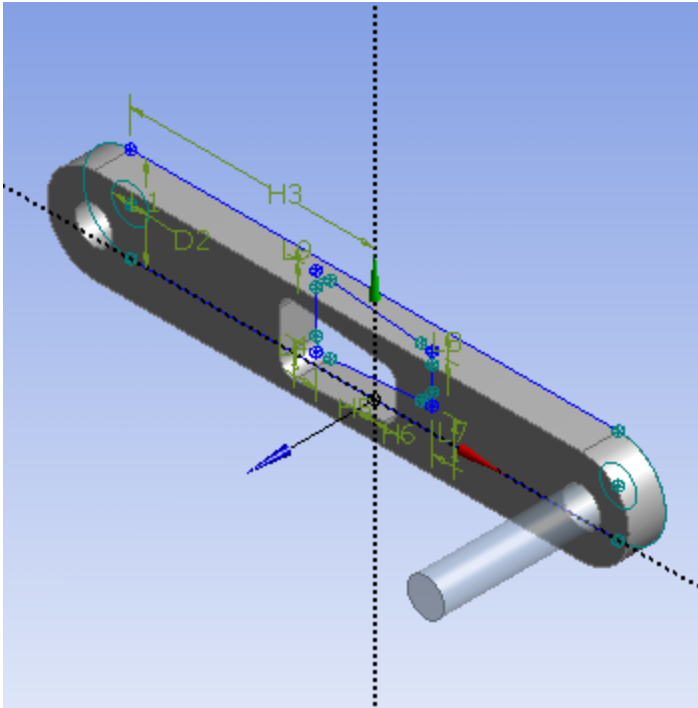
Execute the drawing in the **Graphics** window as shown below.



At this point, we have the crank handle sketch done. We can continue adding **Extrude** feature to the sketch. Click **Extrude**, make sure that **Sketch2** is selected next to **Base Object**. Next to **Operation** change **Add Material** to **Add Frozen**. This is because we want to create the crank as two separate bodies. If we use the default **Add Material**, both the crank and crank handle will be created as a single body. Next to **FD1, Depth (>0)** enter value of 2.645. We can now **Generate** this feature. Under Tree Outline, you will see that we now have 2 Parts and 2 Bodies. Select both parts and perform **Form New Part** operation.



In the **Graphics** window, this is what you should have now.



Congratulations! We are done with the Geometry setup. Let's close the Design Modeler and go back to the Workbench (Your work will be saved. Don't worry).

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