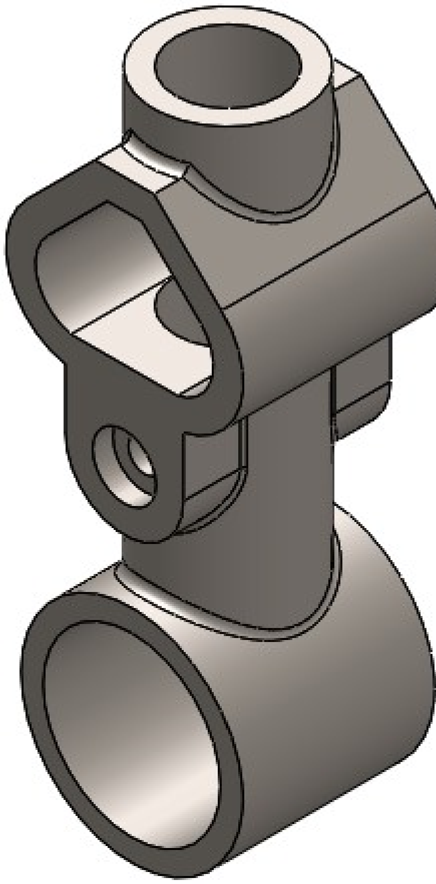
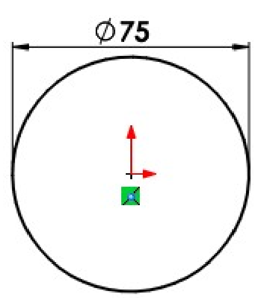
Homework No. 22 (Bonus)
SolidWorks Modeling

In this exercise[1], you will create a new model using a combination of features.



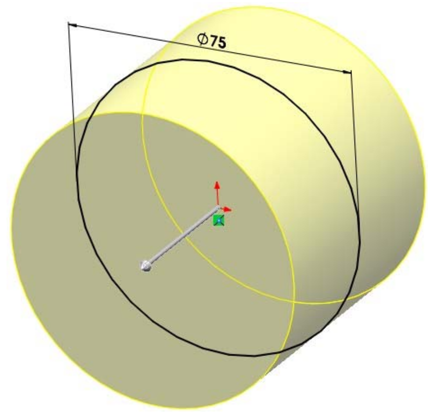
1. Begin a new part file and set the units to **MMGS**, with two-decimal precision. Change the part material to Plain Carbon Steel. *(Hint: To change material, right-click on Material in the FeatureTree and select "Edit Material").*Start by opening a new file.



2. Start a sketch on the Front Plane.

3. Sketch the geometry shown to the right, using dimensions and geometric relations to make the sketch fully defined. (Hint: Note the location of the origin).

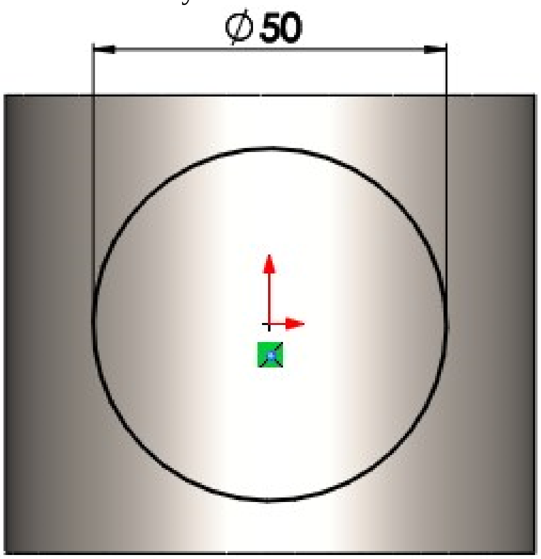
4. With the sketch fully defined, create a **Boss Extrude** feature with a depth of **65mm**.
   *(Hint: Use a **Mid-Plane** End Condition to make sure the Front Plane*



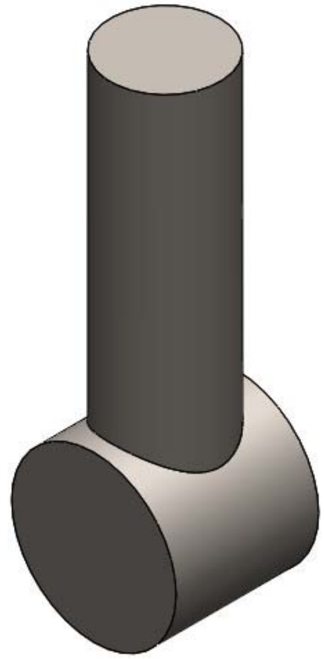
[1] This exercise is partly based on a Solid Professor lesson. http://www.solidprofessor.com/

passes through the center of the part).

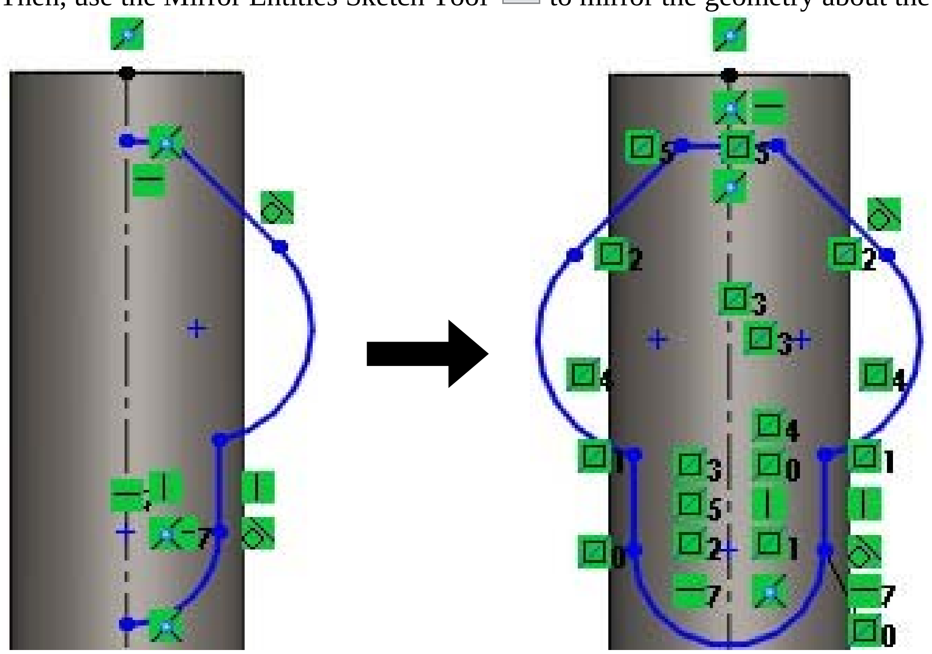
5. Next, create the vertical cylinder. Begin a sketch on the **Top Plane**.
6. Sketch the geometry shown below, using dimensions and geometric relations to make the sketch fully defined.



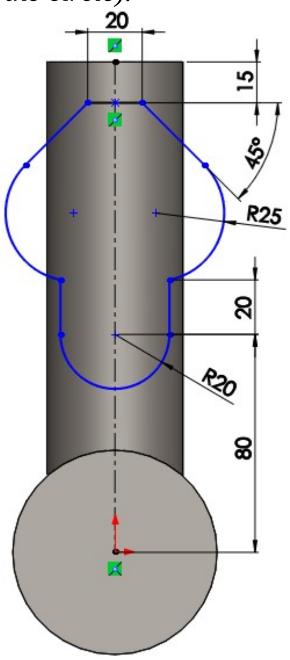
7. Create a **Boss Extrude**, and enter a **Blind Depth** of **180mm**.



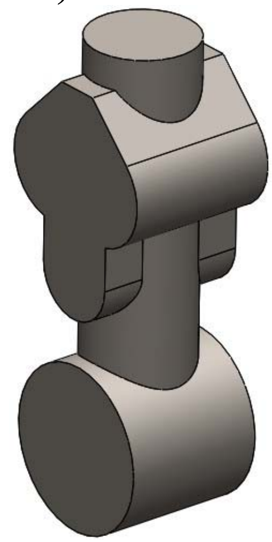
8. Next, create another extrude feature. Begin a sketch on the **Front Plane**, and sketch the following geometry. Then, use the Mirror Entities Sketch Tool to mirror the geometry about the centerline.



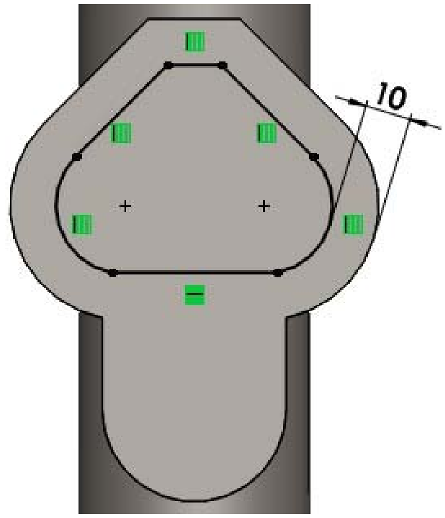
9. To fully define the sketch, dimension the sketch as shown below. *(Hint: To add the 10mm dimension, select the top of the cylinder, and while holding down the Shift Key, select the circle).*

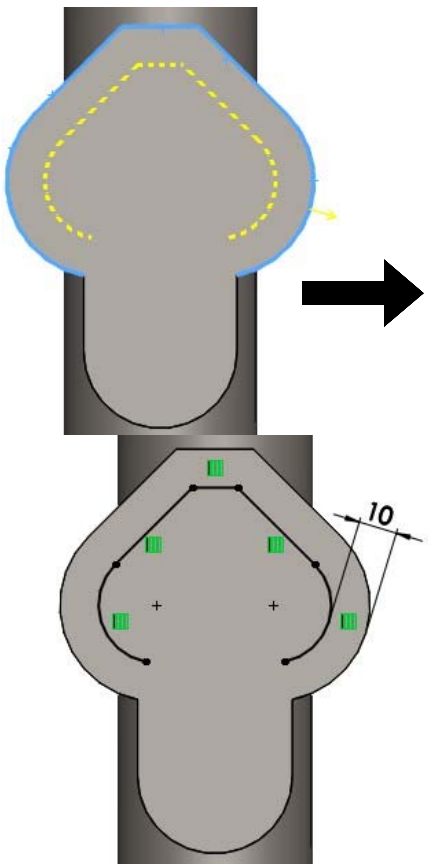


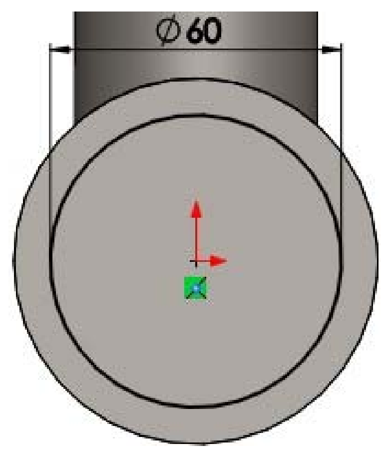
10. With the sketch fully defined, create a **Boss Extrude** feature of **70mm**. *(Hint: Notice in the image below that the feature is extruded in two directions).*



11. With the main geometry created, let’s now create the holes.
    a. To create the cut-out in the bottom cylindrical extrude, sketch the geometry shown below, using dimensions and geometric conditions to make the sketch fully defined. *(Hint: Create a single sketch on the Front Plane to create the sketch geometry needed for the two holes).*
    b. To create the cut-out in the last extrude feature created, Use the **Offset Entities**  Sketch Tool to create the sketch geometry shown below. Launch **Offset Entities** from the Sketch tab of the CommandManager, and when the PropertyManager appears, select the geometry shown below. Type in an offset of **10mm.**

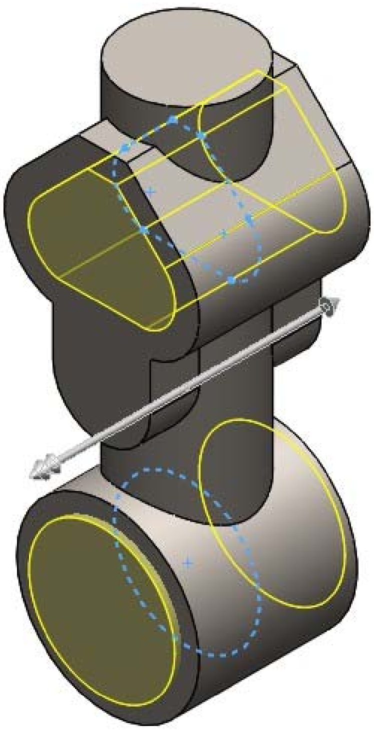




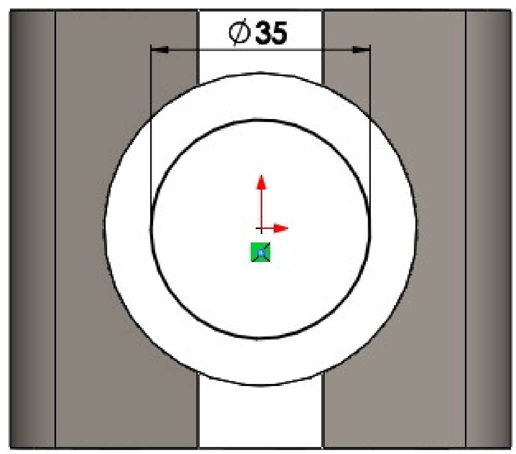


c. To finish the sketch, connect the two end-points with a line as shown to the right.

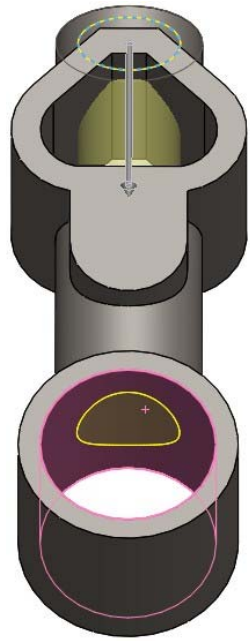
12. With the sketch fully defined, launch the **Extruded Cut**  feature. Set the **End Condition** to **Through All – Both**.



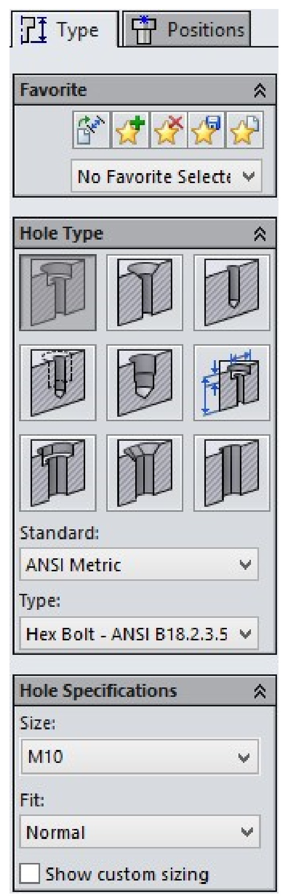
13. Next, create the vertical hole. Start a new sketch on the **top face** of the vertical cylinder. *(Hint: make sure to create the new sketch on the top surface, NOT the top plane).*
14. Sketch the geometry shown below, using dimensions and geometric relations to make sure the sketch is fully defined.



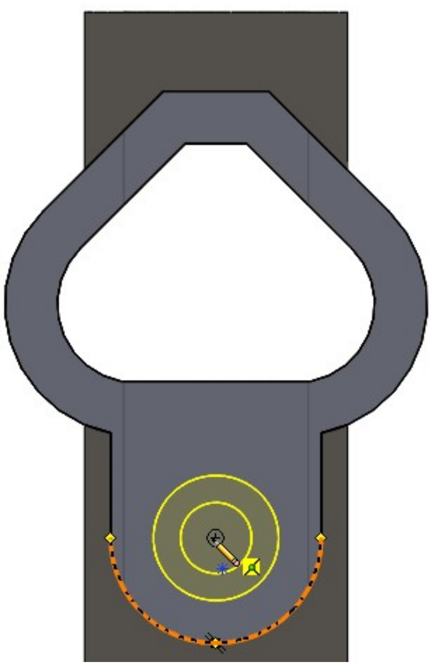
15. To create the cut-out, use an **Up to Surface** End Condition. In the Face/Plane Selection Window, select the face shown below.



16. Next, add the hole using the **Hole Wizard** . Set the Hole Type to **Counterbore.**
    a. Use an **ANSI Metric** Standard and **Hex Bolt** type.
    b. Set the Hole Size to **M10** and Fit to **Normal**.
    c. Set the End Condition to **Through All**.



17. Switch over to the Positions tab and position the hole as shown below. *(Hint: Hover the mouse over the arc to wake up the arc's center point).*



18. To finish the part, add **Fillets** with a **3mm radius** to the edges shown to the right.

19. Save your drawing as HW18.SLDPRT