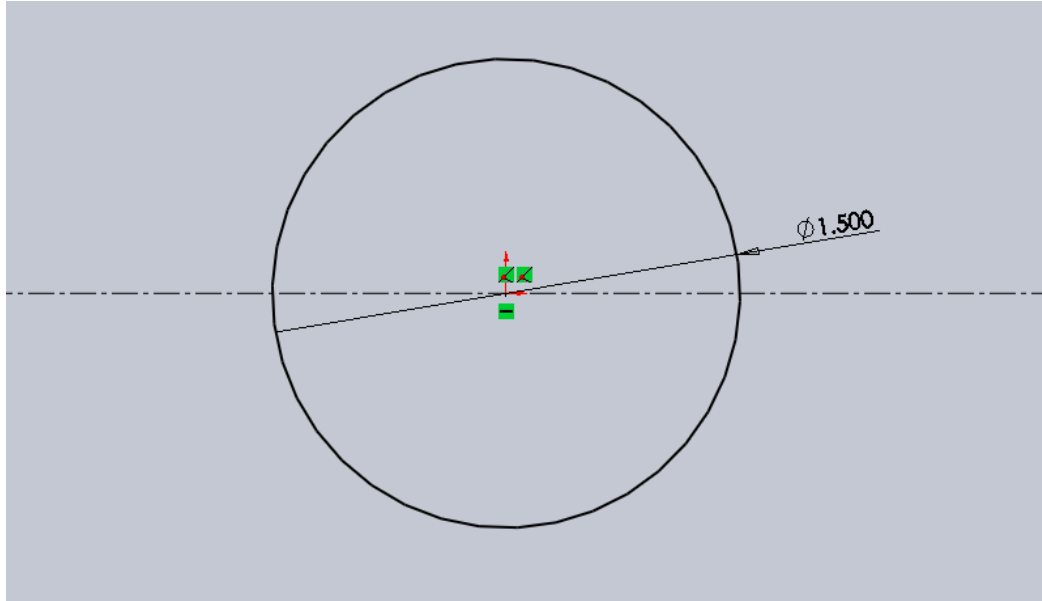


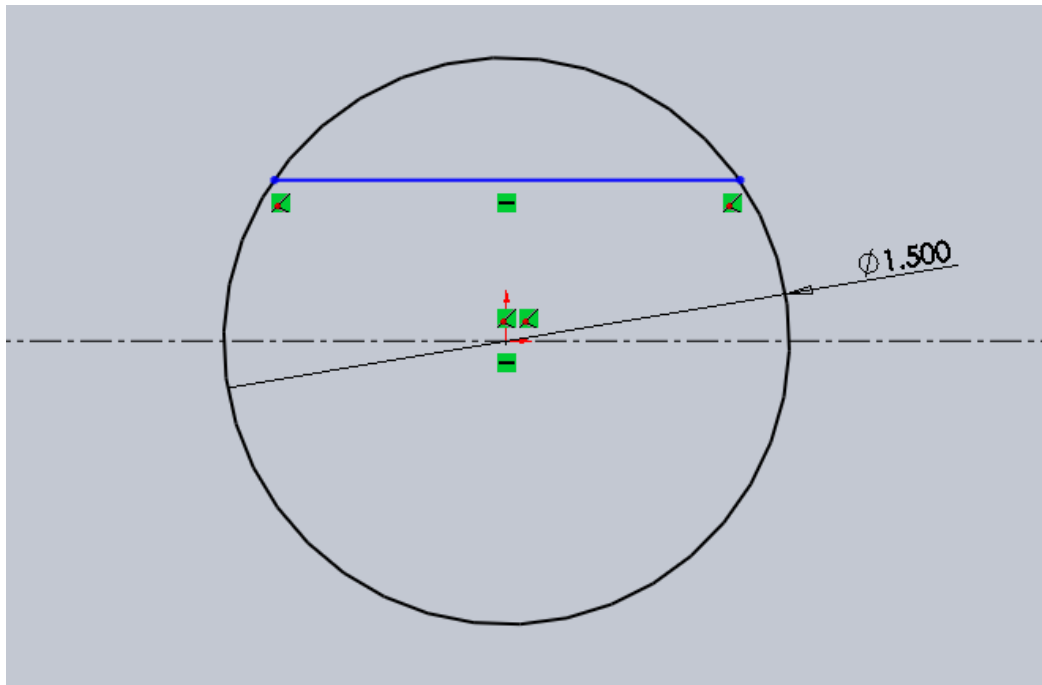
Machine Shop Part

*Save file as “lastname_firstname_hw5.sldprt”

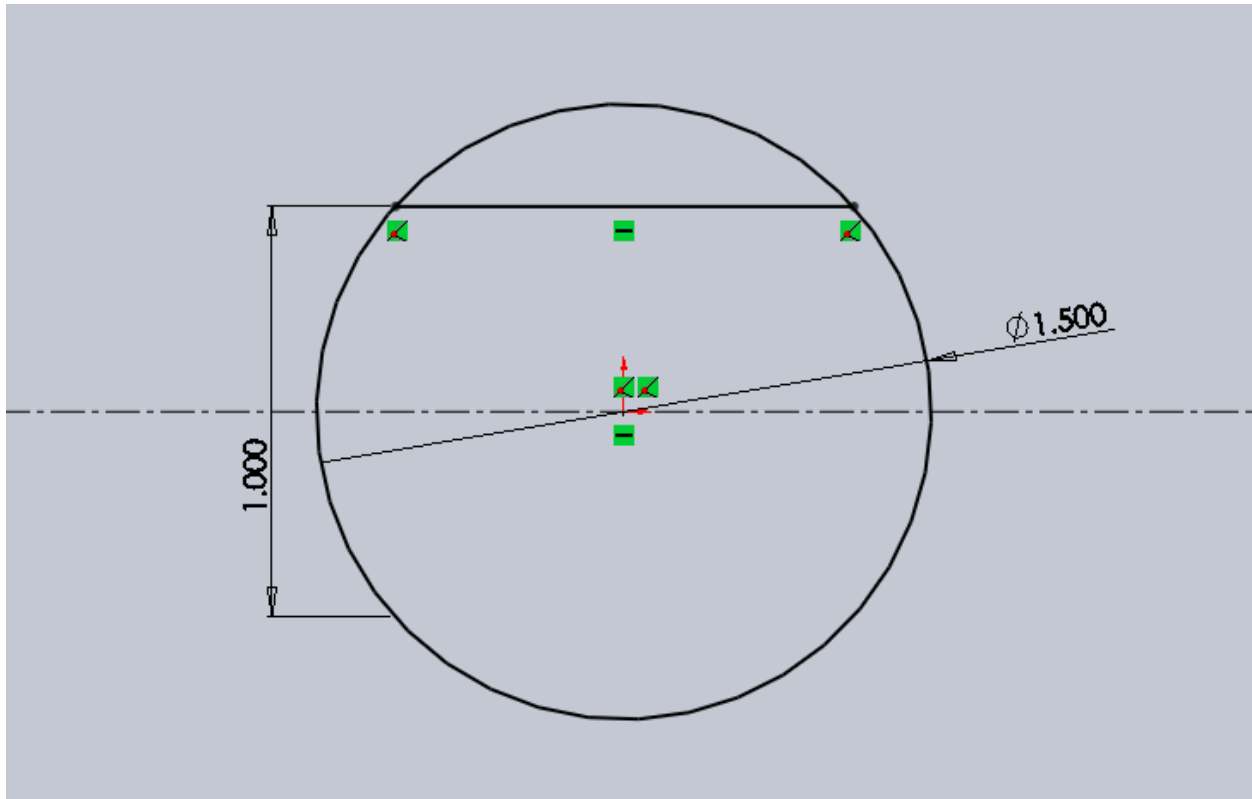
1. Set units to INCHES.
2. Create a sketch on the FRONT plane.
3. Create an infinite centerline as the x-axis. (Make sure this line is BLACK. If not, add the constraint to make the origin and the centerline COINCIDENT).
4. Draw a circle with center at the origin of DIAMETER = 1.5 in.



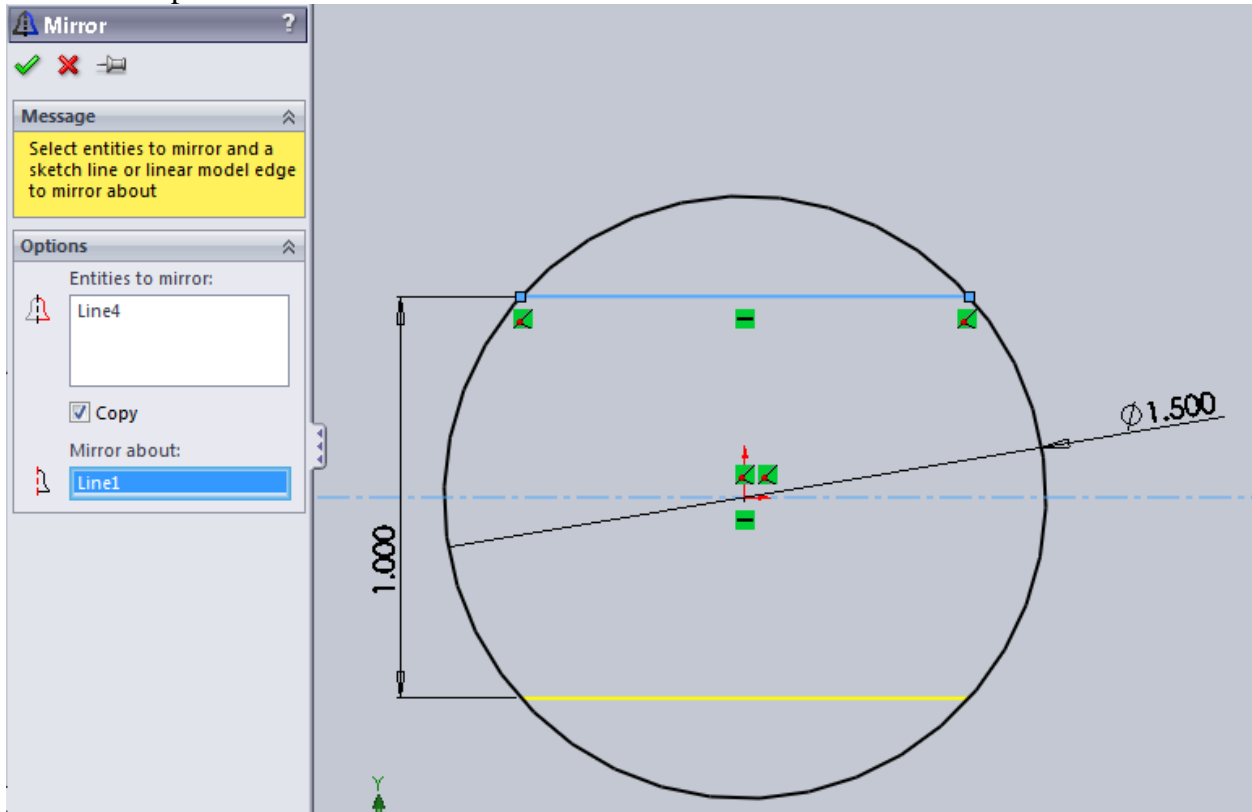
5. Draw a horizontal line across the circle above the x-axis.



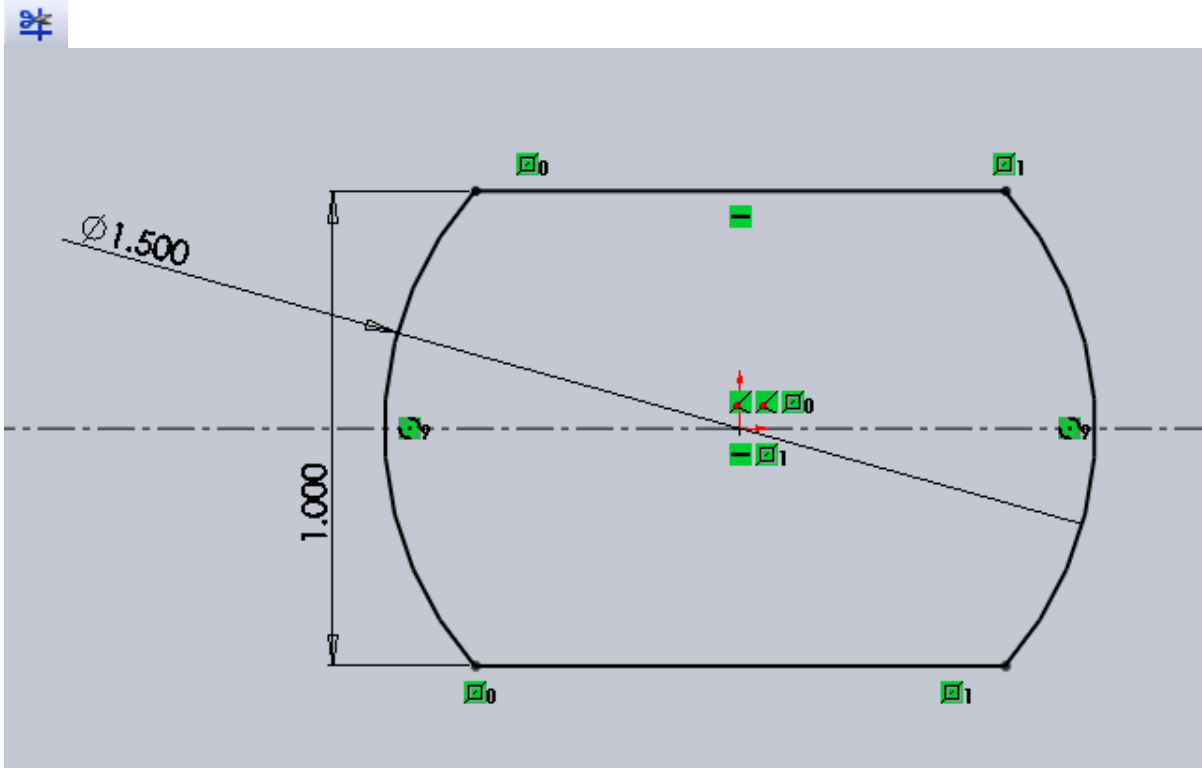
6. Create a mirrored dimension from the horizontal line across the x-axis with a total distance of 1 in.



7. Mirror the top line about the centerline.



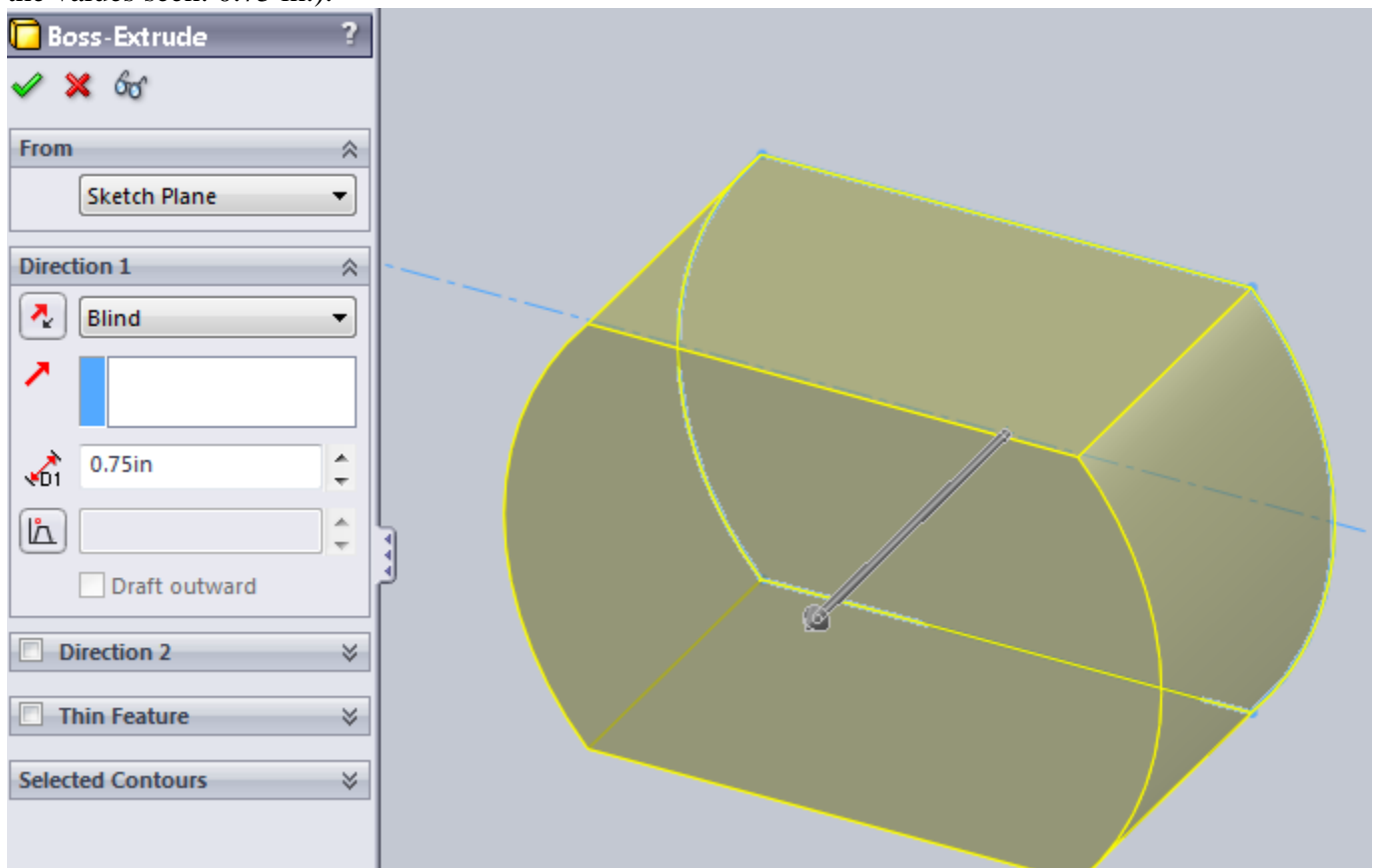
8. Trim the top and bottom portions of the circle with the POWER TRIM tool under TRIM ENTITIES.



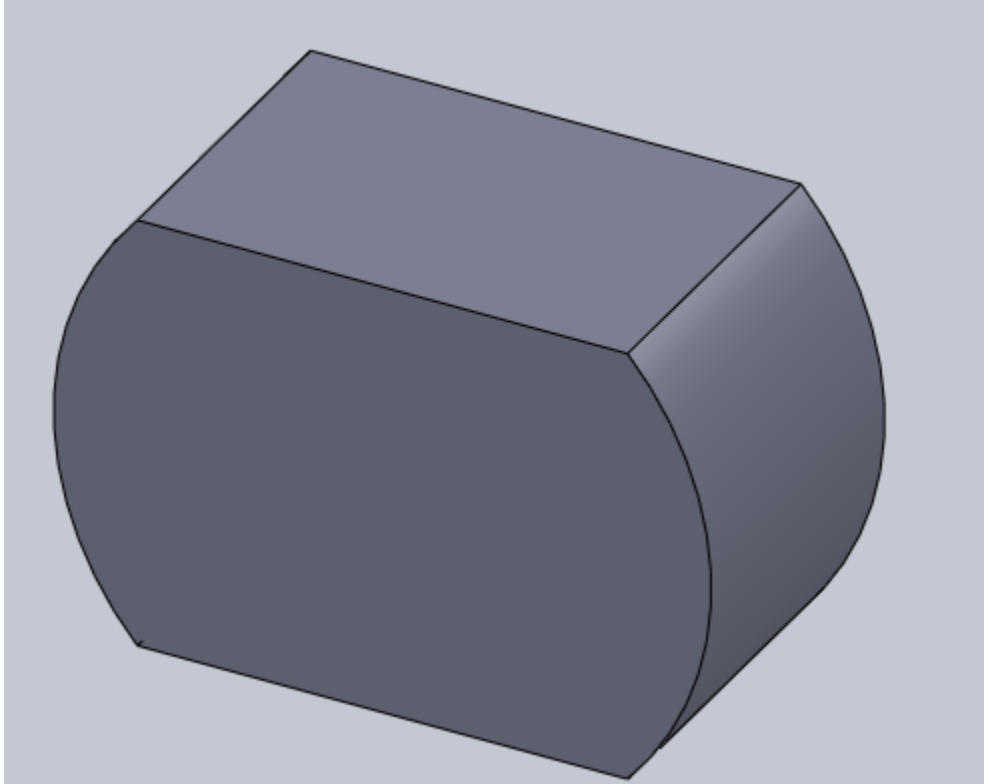
9. Exit the sketch and click on the EXTRUDE BOSS/BASE button under the FEATURES tab.



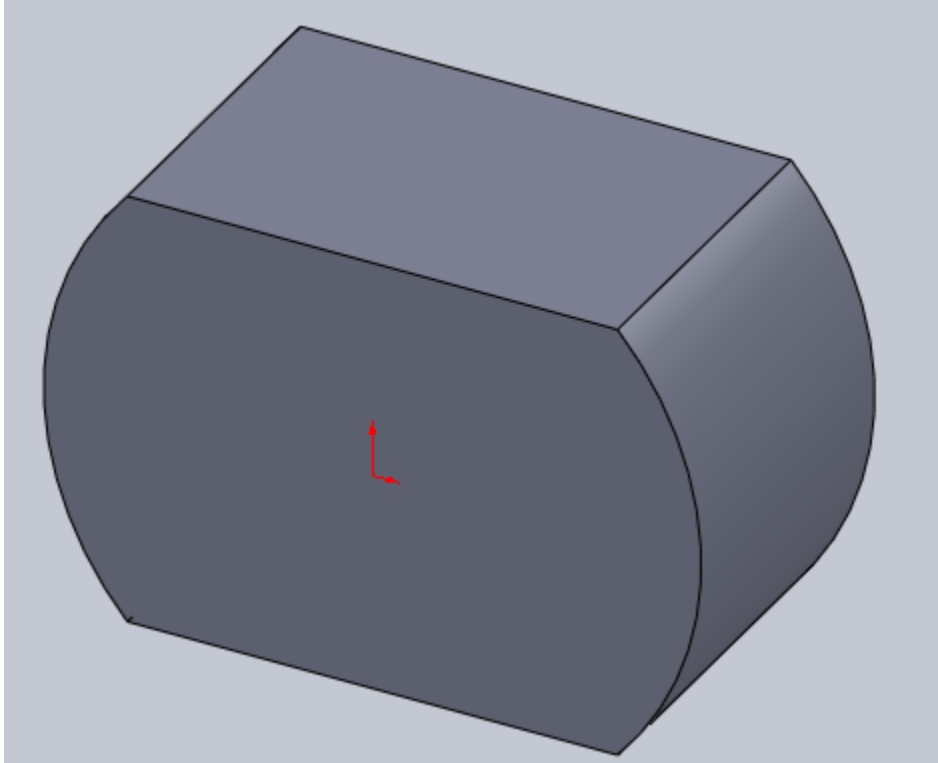
10. Next, click on one of the existing entities of your sketch. The following menu should now appear (input the values seen: 0.75 in.):



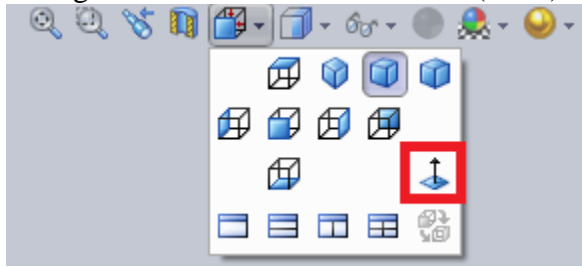
11. Check the green box to complete the extrusion. The sketch should now appear to take the following form:



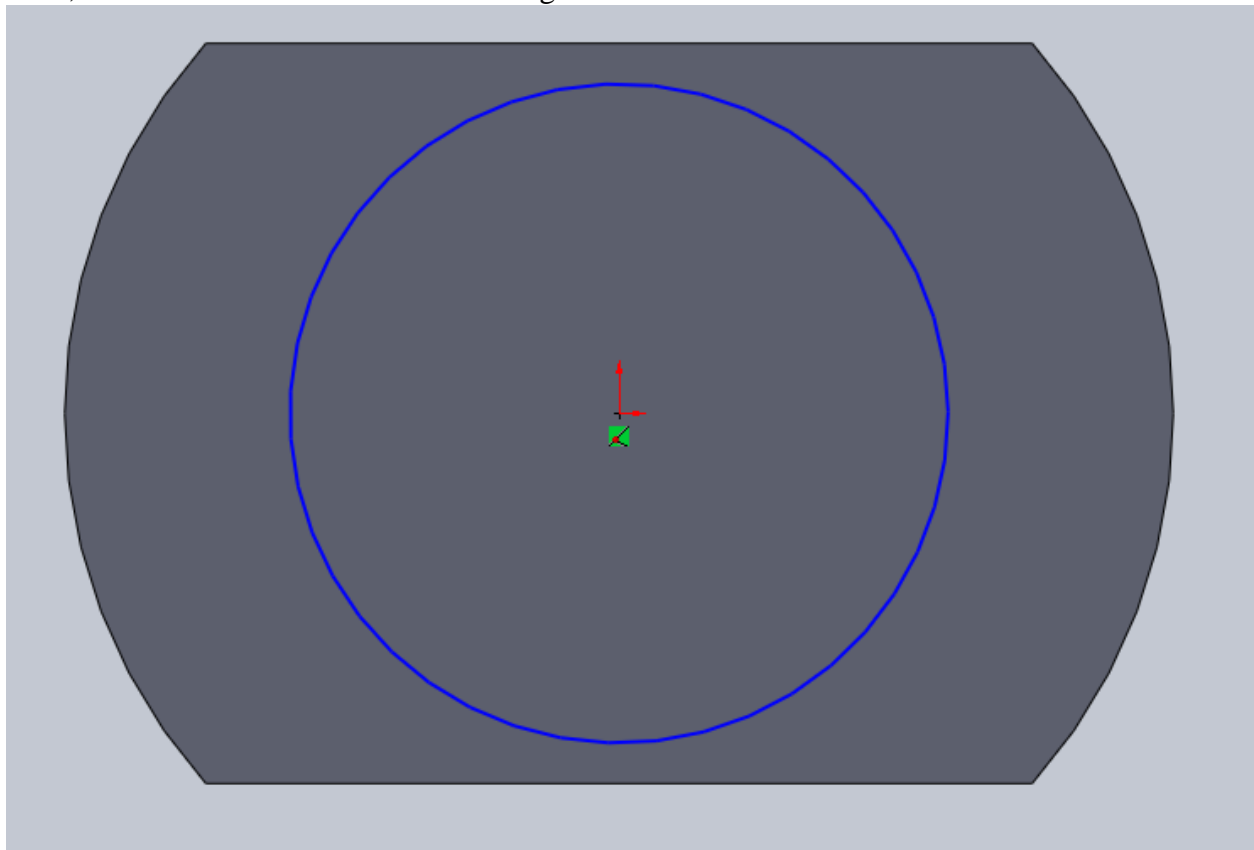
12. Click the sketch button again. However, when choosing which plane to sketch on, select the face that was previously extruded up to 0.75 in. The origin now appears on this face.



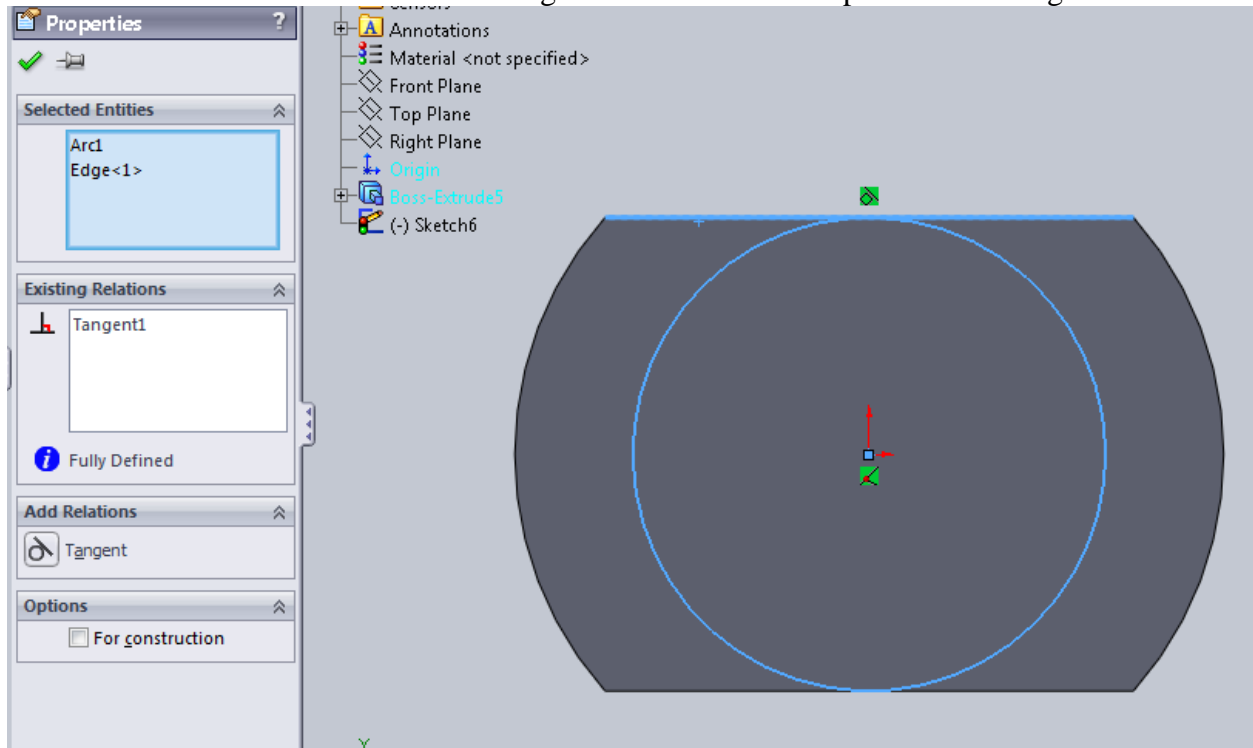
13. Change the VIEW ORIENTATION () to be NORMAL TO the selected plane.



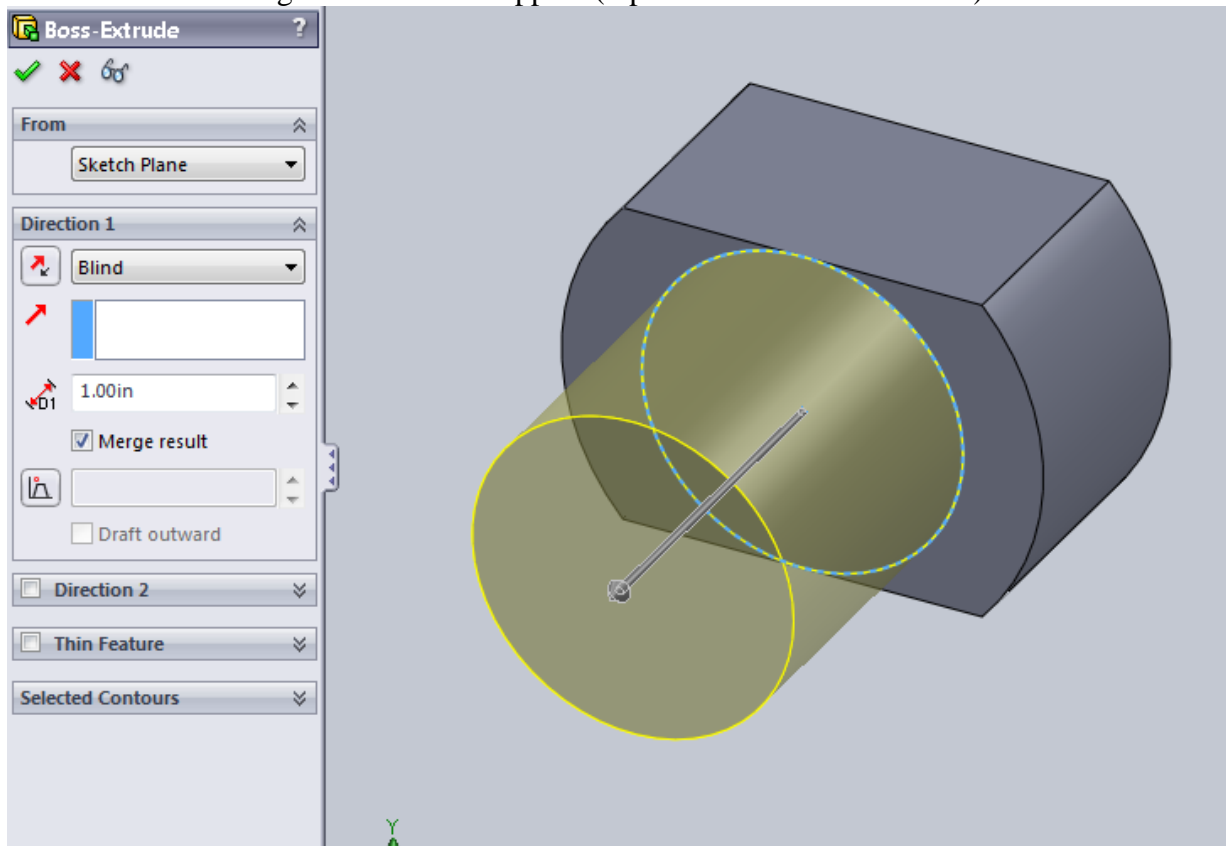
14. Now, sketch a circle with center at the origin.



15. Add a constraint to make the circle tangent with either the top or bottom edge of EXTRUSION 1.



16. Exit the sketch, and again click the EXTRUDE BOSS/BASE button. Select the circle that was just drawn. The following menu will now appear (input the values shown: 1 in.):



The final sketch (in isometric view) should appear like the following:

