Please create the given geometry using SolidWorks. The *revolve* function should be very helpful for completing this lab assignment.

SECTION VIEWS:

Creating section views is fairly simple in SolidWorks. Once you have finished creating the part, insert the LEFT SIDE view only on the UD title block - as shown above. To create a section view, select the view you just inserted and then either use Layout or go to *Insert>Drawing View>Section*. SolidWorks will then ask you where to make the cut in the part to project the section view. Draw a line through the center of the LEFT SIDE view you inserted previously (see the line marked A-A above for reference). This will create a plane, normal to the view that it was defined on, slicing the geometry into two different sections. The arrows define which side will be shown when the projected section is placed. Once you have defined the line, drop the projected section view as shown, fill in the title block with all the pertinent information, and raise your hand to indicate you are finished.

NOTE: Remember, as always, the pictures are guidelines for you; it is your responsibility to bring over your dimension, eliminate any tangency edges, and make sure hidden lines and centerlines are shown.