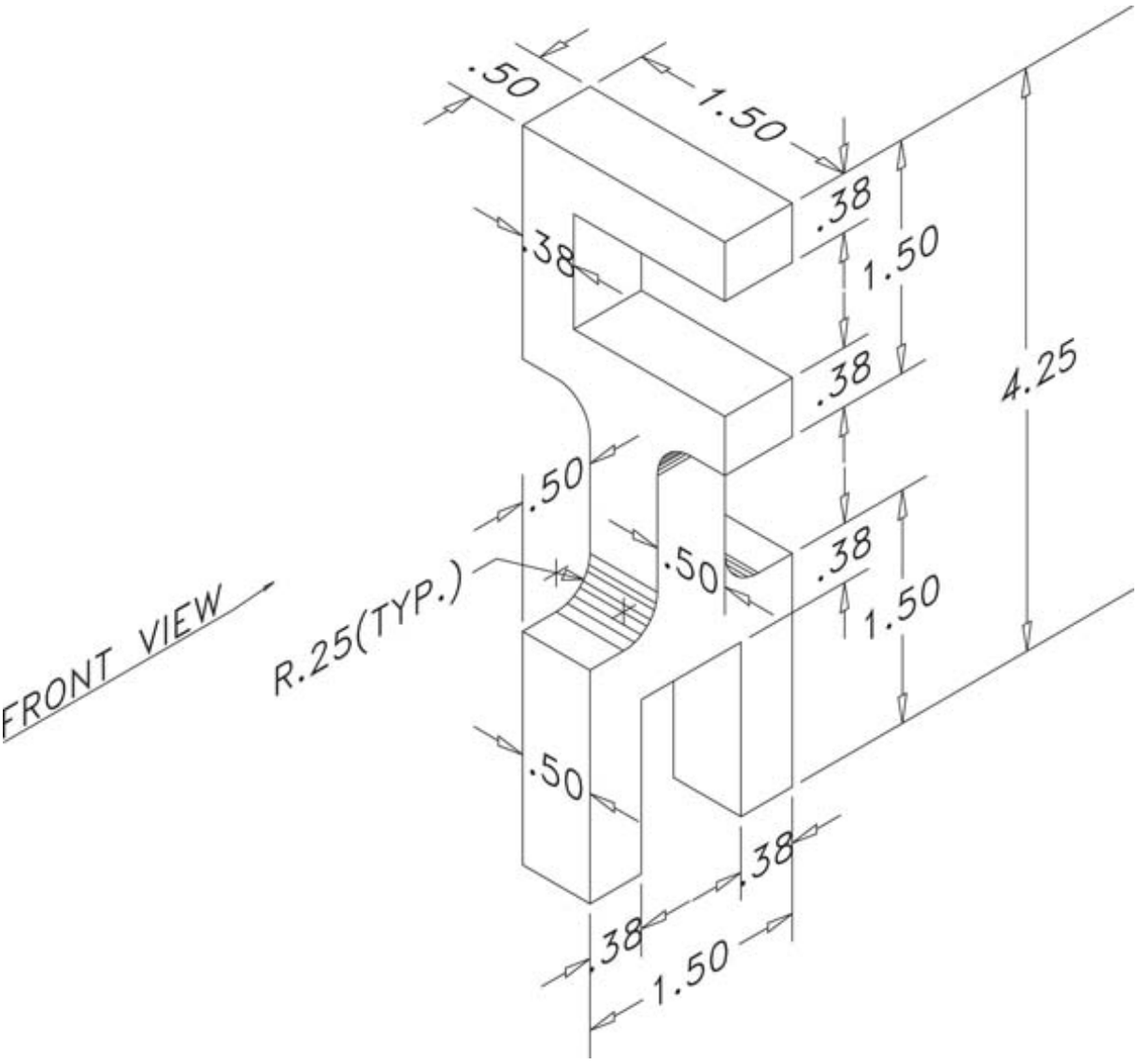
**Homework 2: Solid Modeling of Parts in SolidWorks**
Due *electronically via CANVAS* by 11:59 PM the night **before** your class meeting. Refer to submission instructions for information regarding how to properly submit your files. Refer to the following tutorials for guidance: Introduction to Solidworks, Parts, Fillets, Pattern Features. Segments of the class lecture videos concerning the above topics as well as reference geometry may be of assistance.

Create three (3) solid models based on the following isometric views. Assume all length dimensions are in inches and angular dimensions in degrees.

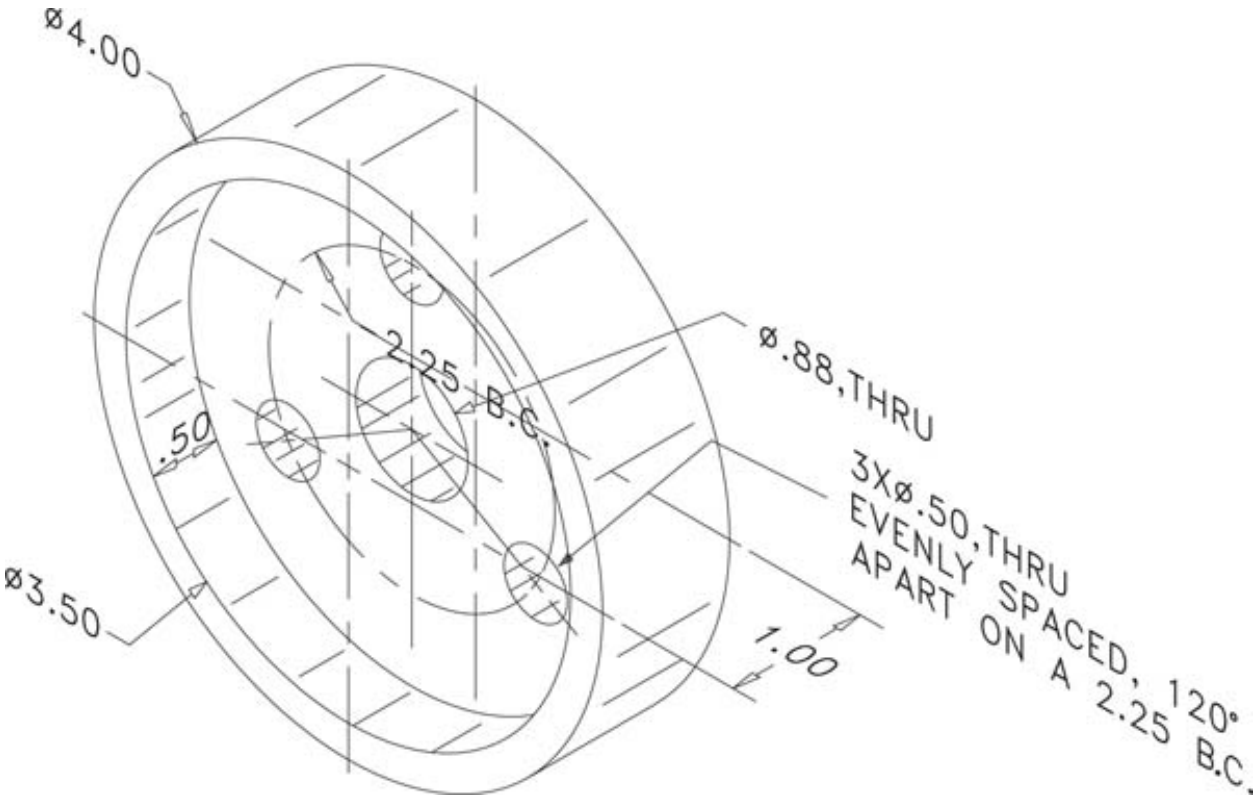



(Typ. stands for typical. In other words, the indicated fillet radius is the same for any other fillets shown.)

Create two (2) separate solid models based on the following isometric diagram:

1. Using **only** extruded features; and,
2. Using **only** revolved features.

You may additionally use a circular pattern feature in each model.




(B.C. stands for bounding circle. In other words, the centers of the holes lie at the same radial distance from the center of the bounding circle, which is also the center of the part.)