Lab 9

Work through the Adding Model Details to Drawings and Adding Notes to Drawings modules using the PLM software titled Detailing using Creo Parametric 2.0

When you have finished, generate a simple part model using the dimensions in the detail drawing shown below. Remember, how you model a part and how you detail it might be different.

What was not covered directly:

Showing Axis: Refer to the Show Model Annotations dialog box as shown. Note that you need to pick on individual features or the entire view to activate the possible display of the axis.

The orthographic views are set to a scale of .5, the isometric has a scale of .25 and that “scale” is moved off the drawing so it will not print.

Place your name in the Title block and print this on A size paper with the Penn State Erie B size title block. Make sure your dimensions are located as shown.
Additional information:

To show the dimensions in the top view such that they only display two places, select the dimensions, then RMB and select properties. Change the Decimal Places to 2.