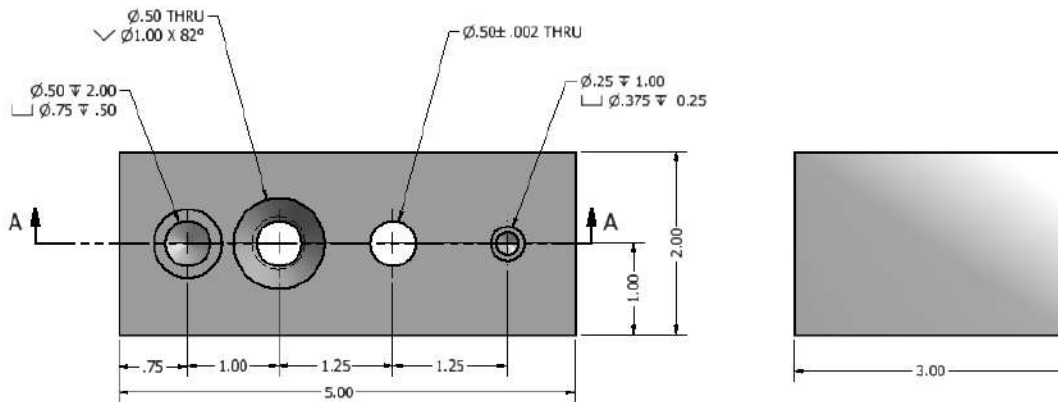


Drilled Part/ Holes

Name: _____

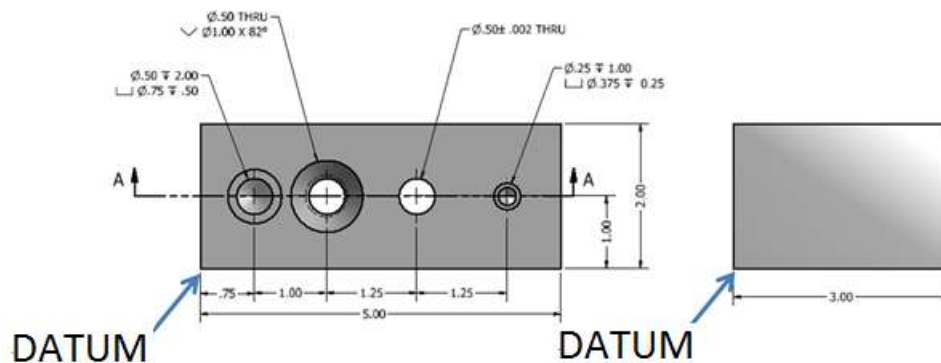
1. Cut and paste the front and right side views of the drill block below into your engineering notebook. Sketch an isometric view of the object in your notebook. Then create a **full scale** section view as indicated by the cutting plane line, but do not dimension the section view. Use appropriate object and section lines. Note that the section view will appear larger than the views below (which are not shown to scale).



2. Create a solid model of drill block in number 4 above. Then create a computer-generated fully dimensioned multi-view part drawing showing the front view and full section view (as indicated by the cutting plane line A-A). Save the files and document the file name and location below.

Part file name:		Part file location:	
Drawing file name:		Drawing file location:	

1. Open the drill block drawing file that you created in Activity 7.2 Sectional Views. The drill block drawing is shown below.



Edit the dimensions to show specific tolerances to the following dimensions. Note that instructions for including tolerance specifications on an Inventor drawing are given below.

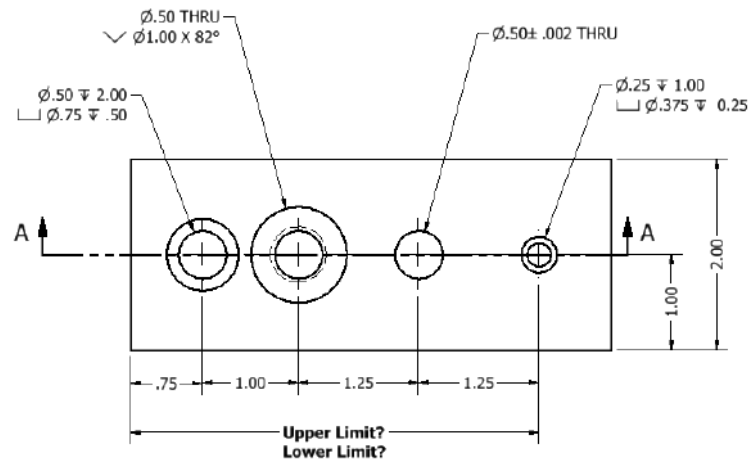
- a. A bilateral (symmetric) tolerance of 0.10 inches for the overall length, width, and depth.
- b. Stack limit dimensions to locate the holes along the depth dimension (2 in.) such that the dimension can vary between the dimension given above and 0.03 inches larger.
- c. To better control the hole locations, use datum dimensioning for the dimensions along the long dimension (5 in.). Note that the datum location is shown on the drawing above.
- d. A unilateral (deviation) tolerance of +.003 inches for the counter bore diameter on the 0.25 inch diameter hole. Change the precision of the counter bore diameter to show three digits to the right of the decimal place.
- e. A bilateral (symmetric) tolerance of 0.002 on the 0.50 inch diameter counter sunk hole.

Add the following General Tolerances notes to the drawing.

Linear Dimensions	Angles = $\pm .5^\circ$
X.X = $\pm .020$	
X.XX = $\pm .010$	
X.XXX = $\pm .005$	

2. Consider the drill block from the question above.

- a. If the detail drawing used chain dimensioning to locate the holes along the 5 in. dimension as shown in the image below (and therefore the tolerances are additive), what would be the upper and lower limit of the dimension from the left edge to the center of the 0.25 inch diameter hole on the right (see below) assuming general tolerances apply?



- b. What is the upper and lower limit of the dimension from the left edge to the center of the 0.25 inch diameter hole in the drawing you created (using datum dimensioning) assuming general tolerances apply?