

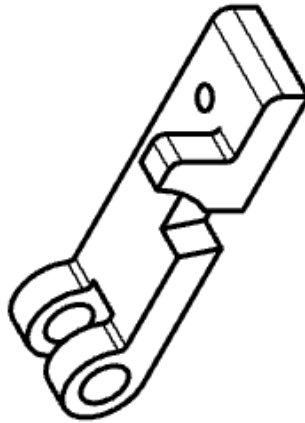
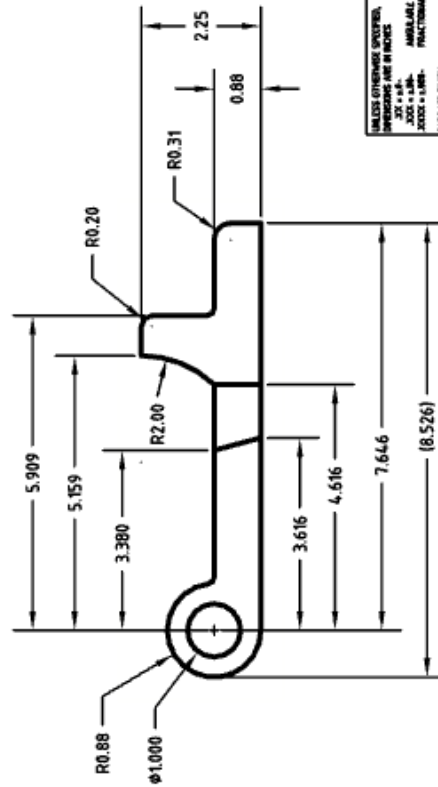
# Onshape College Lesson 5 Homework:



## Exercise #1:

Create the drawing for Part 4 from Lesson 2 as shown on the next page. Pay close attention to the number of decimals for each dimension, and try to think about why some features have tighter tolerances than others! PDF versions of all drawings are included in the folder called ["Drawings for Homework"](#) under the Lesson 5 folder:


NOTES

1. REMOVE ALL BURRS AND SHARP EDGES.
2. UNDIMENSIONED RADII TO BE .13.




UNLESS OTHERWISE SPECIFIED, DIMENSIONS ARE IN INCHES		NAME		DATE	
DIMENSIONAL TOLERANCES PER ANSI Y14.5 DECIMALS = .0001 FRACTIONAL = 1/32		DRAWN CHECKED APPROVED		06/02/2018	
THIRD ANGLE PROJECTION 		SURFACE FINISH 		TITLE Part 4 Dwg	
DO NOT SCALE DRAWING BREAK ALL SHARP EDGES AND REMOVE DIMENSION LINES		INTERNAL FINISH		TEST B PART NO.	
				SHEET 1 of 1	

*Pro Tip: Several new, but commonly used, techniques have been utilized for this drawing.*

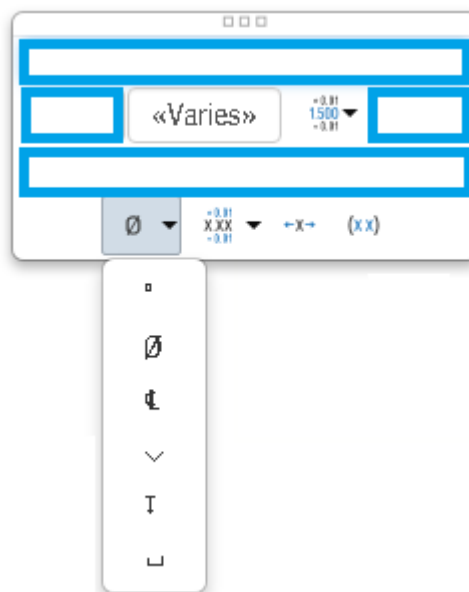
(1) A note has been added in the upper left corner, using the note tool , to remove burrs and sharp edges, which is a standard note to have on a part that is (CNC or manually) machined. This is done for safety, believe it or not, a perfect sharp 90° corner can cut you!

(2) Another note has been added to callout the size of undimensioned radii. This is common when there are numerous, small radii on the part and dimensioning them would add unnecessary complexity to the drawing.

(3) A reference dimension (shown in parenthesis) has been added to the bottom view, by

selecting the parenthesis  icon in the Dimension Panel. In this case, the overall length is already defined by the 0.88 radius and the 7.646 dimensions, but sometimes it is convenient to have an overall dimension on the print. Just adding the dimension would be redundant (and the tolerances would conflict with each other!), so we add parenthesis to the dimension. This means the dimension has no tolerance, and cannot be used as means for rejection (i.e. the part being out of spec).

(4) A “THRU” note has been added to the hole. This is common in machined parts, and should be used where necessary, and where the depth of the hole is not obvious. The Dimension Panel gives us four locations in which to type notes as needed (Above, Before, After, and Below highlighted in Blue) and it also provides some commonly used symbols as well in the drop down (Degree, Diameter, Centerline, Countersink, Depth, and Counterbore) shown here:





## Exercise #3:

Create the following Assembly Drawing using the Clamp Assembly (Pay close attention to which numbered balloons are which!):

