Week 5 Assessment:

1. Which four views are included in the "Four View Layout" in Onshape?

   Front, Top, Right, Isometric.

2. What is the difference between a Dimension and a tolerance?

   The dimension is the nominal ("perfect") measurement of our geometry, whereas the tolerance is the acceptable variation of our geometry allowed during manufacturing.

3. Name & Describe the following Dimension/Tolerance types:

   a. $0.380^{+0.000}_{-0.005}$ *Deviation - This communicates the allowable +/- deviation from the nominal dimension.*

   b. $\begin{matrix} 0.385 \\ 0.375 \end{matrix}$ *Limits - This communicates the allowable limits of the dimension (the nominal is not shown).*

   c. $\boxed{0.380}$ *Basic - This has no tolerance, and therefore may only be used when GD&T is utilized to define the tolerance range.*

4. What is the real-world implication of changing a standard dimension on an engineering print from "1.750" to "1.75"?

   It means that we are changing the tolerance to a "looser" one (wider allowable range) as defined by the tolerance block in the drawing format.

5. Do Onshape drawing views update automatically? Why or why not?

   No they do not, as this gives the designer control whether they want the drawing to reflect the latest design or not. It is a form of revision control, since Onshape does not have (or need) a formal PDM system.