# Week 5 Assessment:

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

*Front, Top, Right, Isometric.*

1. 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.*

1. Name & Describe the following Dimension/Tolerance types:
   1.  *Deviation - This communicates the allowable +/- deviation from the nominal dimension.*
   2.  *Limits - This communicates the allowable limits of the dimension (the nominal is not shown).*
   3.  *Basic - This has no tolerance, and therefore may only be used when GD&T is utilized to define the tolerance range.*
2. 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.*

1. 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.*