

Creating a Circumference dimension in SolidWorks

Often times we find the need to dimension arcs based on the circumference of the arc instead of the diameter of the arc. Or, it may be that we want to display a circumference dimension on a SolidWorks drawing. Circumference dimensions can be added using the SolidWorks Smart Dimension command. As a side note I like to setup up the "D" key as a keyboard short cut for the SmartDimension command in SolidWorks. Learn how to add custom keyboard shortcuts in SolidWorks. In order to create a circumference dimension, simply activate the smart dimension command and then select; the start point of the arc, the arc and the end point of the arc. The order in which you select the entities is not critical, but you will have to be careful to make sure you don't drop the dimension before you finish selecting all the entities.