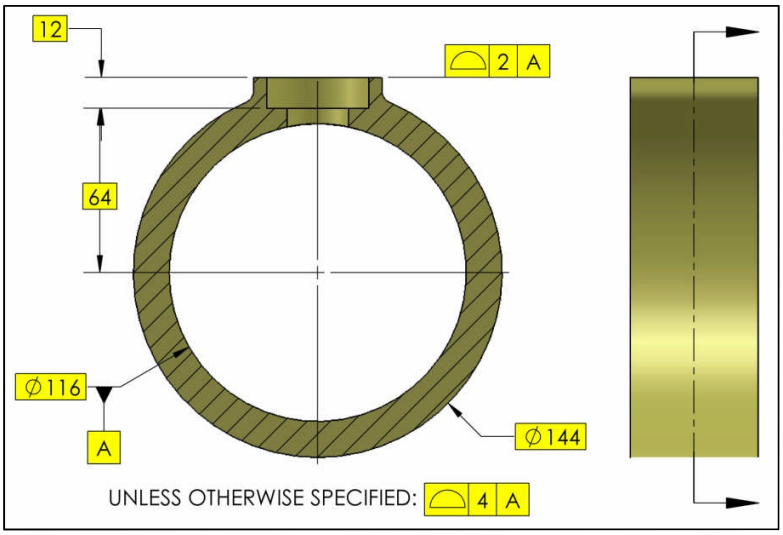
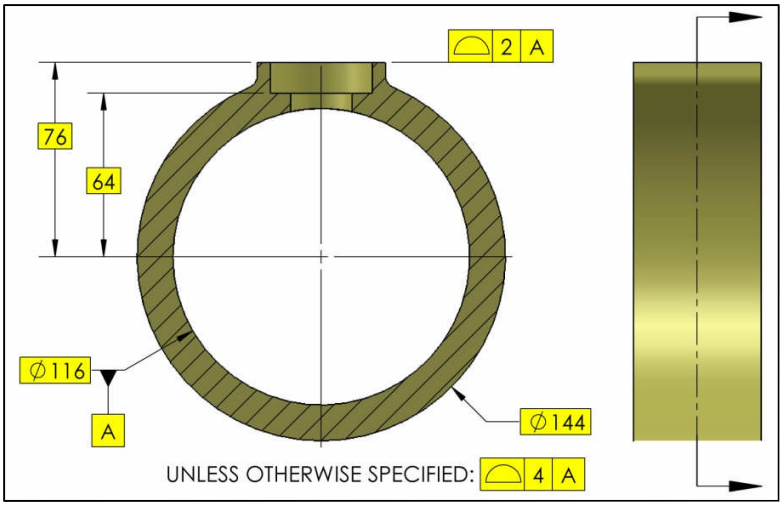
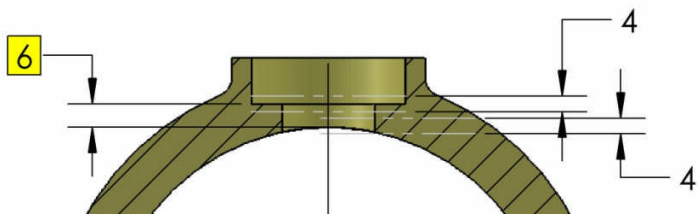
GD&T Tip #287

Basic Dimension Placement Doesn't Matter

Here is a question for you. Look at the drawings. The depth of the counter bore is 12 BASIC. On the actual part, what is the deepest and shallowest the counter bore can be according to the first drawing? If you said from 9 to 15, you would be correct. Now, look at the second drawing. In order to determine the BASIC depth of the counter bore you have to subtract 64 from 76 which once again gives you 12 BASIC. On the actual part, what is the deepest and shallowest the counter bore can be according to the second drawing? The answer does not change. The placement of basic dimensions doesn't change the meaning of the drawing the way it used to with directly toleranced dimensions. Geometric tolerancing takes the tolerances off the dimensions and puts them where they belong, on the features. Basic dimensions are the goal but are not toleranced. The bottom of the counter bore relative to the datum axis can vary 4 mm total because of the general profile tolerance in the note. The surface at the top of the counter bore can vary 2 mm as is stated in the profile of a surface feature control frame. Therefore, the total variation between the two surfaces can be 6 mm (4+2). Half will increase the basic distance and half will decrease the basic distance. So the depth can be thought of as 12 ±3.







For real fun, calculate the minimum wall between the bottom of the counter bore and the inside diameter. It can be as thin as 2 mm. Keep in mind that the profile tolerances are on the surfaces--not the dimensions.

http://www.tec-ease.com/premium/gdt-tips-view.php?q=287 to see Don Day explaining this Tip.

Please email us any suggestions or topics that you would like to see covered in our GD&T Tip Series.

www.tec-ease.com