# Dimensioning voice

## Page2

**Page2\_2**

When dimensioning a part, you need to keep in mind that the type and placement of the dimensions are controlled by the ASMEY14.5 standard. This standard lists several rules and best practices.

## Page4

**Page4b**

If you have a feature that occurs several times, you don’t have to dimension each individual feature. You can use the repeated feature symbol which looks like an X. You indicate the number of times that the feature is repeated followed by the X and then a space followed by the feature size. This applies to equally spaced features as well. Looking at the figure you can see that the holes are all spaced 15 mm apart. The dimension reads that there are 10 spaces and each space is 15 mm. The 150 mm dimension is a reference dimension stating the total spacing. The next figure shows equal angles.

## Page5

**Page5b**

If you are looking at a drawing and a feature looks like it makes a 90 degree angle, you may assume that it does. For example, a corner or the angle between holes.

**Page5c\_1**

There are several rules that influence manufacturing. In addition to these rules, the dimension text and placement of the dimensions influence how a part is made. When you are dimensioning a feature that is made with a specific manufacturing process such as drill, you do not specify this manufacturing process. The process used is left up to the manufacturing department. Some of these standard processes use gage or code numbers it identify things such as drill size. These code numbers may be placed on the drawing, but the decimal value of the drill needs to be included.

Some more obscure rules state that all dimension apply at 20 degrees. This is because some materials expand significantly at elevated temperatures. Also, a nonmandatory dimension may be given specifying the size of the part prior to manufacturing.

**Page5d\_1**

One very important rule is that dimensions imply function. You want to choose your dimensions based on the function of the part. When a person reads your print, he or she should get some idea of the part function and which surfaces are important. Let’s take a look at the two figures shown. In the figure, try to locate the surfaces that most of the dimension originate from. These surfaces are known as datum features. They are important surfaces. Notice that there are two dimensions that don’t originate from the datum features. This is the .50 dimension and the 1.00 dimension. The 1.00 dimension placement is telling us that the distance between the holes is important. Perhaps these hole will mate up with pins on another part. The .50 dimension is telling us that the height of the cylinder is important.

In the second figure we see an alignment gage where a hub is aligned on a base. Note the datum features. They are indicated by datum feature symbols. These are boxes with letter inside. Note that datum feature A is a surface that mates up with the base. Mating surface make good datum features. The hub has two holes that mate up with pins. We would expect that these distances would not originate from the datum feature.

**Page5e\_1**

Every dimension is toleranced and these dimension apply to the un-deformed state of the part. You’ve probably never heard of a tolerance as applied to a dimension. We need them because nothing can be manufactured to an exact value. We need to specify a range of values that the feature can be and still function correctly. That’s a tolerance. A range of values. A maximum and minimum value.

If you look at the figure, I’ll point out the different toleranced dimensions. In the upper left corner of the drawing there are two toleranced dimensions. The top one gives a maximum and minimum size that the hole can be. The bottom one gives a nominal size with values for how much the hole size may vary. Look at the .12 dimension on the right side. This dimension doesn’t look like its toleranced, but it is. It is covered by the block tolerance, which says that all dimension having 2 decimal places can vary by plus or minus .05. The dimension in the middle. The one with the box around it is called a basic dimension. This is a non-toleranced and theoretically exact dimension. These dimensions are usually used in conjunction with GD&T (which stands for geometric dimensioning and tolerancing). We will not be covering GD&T but it is a very important topic.

This should not happen, but if a dimension is not explicitly toleranced and it is not covered by a block toleranced, then the tolerance is based on the number of decimal places the dimension has. For example, if a dimension reads .50, then any feature size is acceptable if it rounds to .50.