Why Do We Need Tolerances?

Imagine how many arguments there would be if you had only a dime-store ruler to measure your work with, while your boss used a micrometer! You could spend hours trying to make something as close as possible to one inch long, but your boss' micrometer would probably show that you didn't even come close. Now imagine that when you finally did satisfy your boss, the part went to the inspection department, where they used an electron microscope.

Every measuring instrument has limits to how accurately it is made and how accurately you can read it. There is no such thing as an exact measurement.

In addition, there is no such thing as an exact dimension. Suppose you had to cut 100 boards 12 inches long, using a hand saw. If you have ever tried something like this, you know that when you are done, you can put the boards on end side by side, and you would have nothing that comes close to a flat surface. For carpentry work, though, it is often close enough.

The machines and instruments used in industry are far more accurate than hand saws and rulers. But even these machines, if you measure finely enough, cannot produce exact dimensions. You cannot set them up to produce perfect dimensions. Even on the best of machines, one part produced will not be identical to the next.

By giving a tolerance range on blueprints, the designer makes sure that everyone agrees on how close to the exact dimension the part needs to be. The tolerances are based on the designer's experience and calculations, which tell the designer a part within the range of sizes given will perform without problems.

In the past, people have used tolerance ranges to define how closely they should pay attention to their work. The theory was that any effort to produce parts more exact than specified on the print was a waste of time.

Recent studies have shown, though, that parts produced with a minimum of variation and as close as possible to the nominal dimension assemble more easily and cause fewer problems than parts with a wide range of dimensions, even if they were within the blueprint specifications. These parts may cost a little more to make, but they cost less when you consider the whole picture, including assembly costs, number of finished products requiring rework, and customer complaints.

Therefore, tolerance ranges should never be interpreted as an excuse for sloppy work. They should be used to allow for the statistical variations in dimensions produced by machinery, rather than for careless setups. Today's industry goals call for continuous quality improvement, with blueprint tolerances being the minimum acceptable standard.

Why Geometric Dimensioning and Tolerancing?

Traditional methods of giving dimensions and tolerances on blueprints have several drawbacks. First, they are not designed to be consistent with the function of the part. GDT gives designers the freedom to specify that parts will be designed the way they are to be assembled and used. For example, screw holes are located according to the angle between them and the diameter of the circle they are to be on. Shafts and other rotating parts have specifications for roundness, runout, or concentricity to assure that they will have the symmetry they need in a rotating application. Traditional methods have no way to specify this other than with excessively stringent tolerances and often-cryptic notes from the designers.

This functional design reduces disputes between production and inspection, and those common cases where a component is out of tolerance, but might still be usable. Because of the functional design, the tolerances specified on the drawing are the maximum that can be allowed. Any part that falls outside of these tolerances must be reworked or scrapped.

Second, they are standardized. As mentioned in the above paragraph, the concepts described in GDT often had to be written out as notes on the prints. Designers have never been noted for their literary skills, and this fact has often led to questions and errors when parts were being made. GDT uses symbols with agreed-upon meanings to eliminate confusion and misinterpretations.

Third, they are accepted worldwide. Most of the notation does not depend on a particular language, so a drawing made in another country can be understood easily by machinists here.

Seventy to eighty percent of all major companies in the U.S. use GDT. It is the recognized standard for military contracts. (Krulikowski, Alex. Fundamentals of Geometric Dimensioning and Tolerancing.)

Understanding Tolerance Zones

Maximum Material Condition (MMC) M

There are times when we need to work with a theoretically exact dimension for the purpose of measuring, but the drawing allows a tolerance range for the part. In GDT, the convention that is used is to use the dimension at one end of the tolerance range. Usually this dimension is the one known as the maximum material condition (MMC). Sometimes the opposite -- least material condition (LMC) -- is specified instead. The symbols used for these are ___ for MMC and ___ for LMC.

When you see the MMC symbol, picture what the dimensions of the part would be if it had the most material allowed by the tolerance range. That is, imagine that you are machining the part from a larger piece of material, and you have cut away the least material you can in order to meet the tolerance on the print.

It is easy to understand MMC for an external dimension. Look at the block on the right. The MMC dimension is the one that is the largest allowed. What is the MMC dimension for the length of the block? What is the MMC dimension for the width of the block? What is the MMC dimension for the thickness of the block?

MMC is a little trickier to understand with an internal dimension. Look at the hole in the block above. What dimension of the hole results in the most material being left on the block. If you are confused, consider what dimension would make the block weigh the most. The smallest hole allowed is the maximum material condition.

Maximum Material Condition (MMC)
vs. Regardless of Feature Size (RFS)

As we have discussed, the size of a part is never exactly what is spelled out on the print. This is why we have tolerances. How does the actual part size affect the way a geometric tolerance is measured?

There are two possible ways that the part size can affect the measurement. The designer decides which is more appropriate when making the drawing, and specifies on the drawing which method is to be used.

One of these methods is to apply the tolerance regardless of feature size (RFS). This is the method that is implied if no specification is made on the print. It simply means that the geometric tolerance is interpreted literally.

Consider perpendicularity as an example. Look at the block on the right, with a pin mounted on one face. The designer has specified that it is to be perpendicular within 0.002 in. In other words, the pin needs to be square with the block face, and its axis cannot vary from this position by more than 0.002 inches. To verify that it meets this requirement, you need to find the center at the top of the pin and at the bottom of the pin, and make sure that those points are no more than 0.002 in. apart horizontally.

Alternatively, the designer might have included a specification that this specification applies at maximum material condition, as is shown on the left. This is actually an easier requirement to meet than an RFS specification. A perpendicularity symbol with an MMC specification means that the designer really meant something other than what is said in the control frame. Squareness is not the important feature when MMC is specified. Instead, the designer is concerned about the locations of boundaries of the feature.

Look at the pin on the left above. The MMC specification means that the perpendicularity condition is to be used assuming that the pin is the largest allowable size. To verify that the part meets the requirement, you need to find the center at the bottom of the pin. Then, the top of the pin cannot be outside of a circle with a radius equal to the maximum radius of the pin plus 0.002 inches.

A tolerance of this type is specified when the part has to mate with another part. In the case of the pin above, it has to go into a hole in another part. If the top of the pin is outside of the tolerance circle, the designer knew that it might not fit into the hole.

The first pin, with its tolerance applied regardless of feature size, will have a roller or gear mounted on it. There is no concern for fit as long as the diameter of the pin is within the specified range, but if the pin is not square, the roller will be tilted and will cause parts to wear quickly.