



Metalworking

## Mastering Machining Tolerances: The Keys to Interchangeable Parts

Kip Hanson | Jan 06, 2025

Historians might argue over who is most directly responsible for the notion that manufactured parts should be interchangeable, an idea that led to the development of dimensional tolerancing standards and what is now known as metrology.

The fact remains, however, that it was a group effort.

Industry notables include inventor of the screw-cutting lathe Henry Maudslay; cotton gin and musket manufacturer Eli Whitney; Carl Edvard Johansson, without whom we'd have no "Jo" blocks; the so-called "father of scientific management," Fredrick Taylor; and many others, all of whom contributed to the concept of interchangeability.

So have well-known organizations such as the International Organization for Standardization (ISO), American National Standards Institute (ANSI), and the American Society of Mechanical Engineers (ASME), bodies that helped to formalize the measurement standards now applied on a global basis.

### Types of Tolerances

One of the most fundamental aspects of any *discussion on metrology*, part interchangeability and adherence to measuring standards is—you guessed it—tolerancing. According to ASME standard Y14.5: Geometric Dimensioning and Tolerancing (GD&T), tolerance is defined as "the total amount a dimension or feature is permitted to vary. The tolerance is the difference between the maximum and minimum limits."

But what precisely does that mean? The answer depends on several factors, chief among them the type of feature being measured. The Y14.5 standard is more than 300 pages long, so there's not room to dive

too deeply into it here, but broadly speaking, the dimensions of machined and fabricated part features are controlled by the *allowable deviation* from their intended length, width, diameter and so on—their tolerance, in other words.

For instance, a 0.500-inch diameter bored or drilled hole could have a bilateral tolerance of +0.010/-0.002; in this case, the feature's upper limit is determined by adding the first value to its basic size (i.e.,  $0.500 + 0.010 = 0.510$ ), while the lower limit comes from subtracting the second value ( $0.500 - 0.002 = 0.498$ ). Similarly, a 2.00-inch-long sawed blank might be dimensioned as +0.05/-0.00. Here, the part meets tolerance if it measures between 2.00 inches and 2.05 inches long.

Further, there are the five types of tolerances applied to part surfaces and their relationships to one another. These include:

- **Tolerances of Form:** As its name suggests, straightness might be used to define how much the centerline of the shaft described earlier can deviate from a perfectly straight line. Flatness is often used to control surfaces that have been ground or milled flat; circularity could be applied to a stamped hole or spherical form, and cylindricity would determine the allowable amount of barrel or taper within a deep hole or turned diameter.
- **Tolerances of Orientation:** Parallelism controls the permissible deviation between two planes or axes intended to be perfectly parallel. Similarly, angularity defines the amount by which an angled surface or line can deviate from a reference surface (datum), just as perpendicularity is used for surfaces, lines, and axes that have been machined or fabricated at right angles to one another.
- **Tolerances of Position:** Used to control the center-to-center distance between features such as holes, slots, bosses, etc., or their center-to-datum values—if a drilled hole must be located 1.5 inches from a part edge (the datum), the callout might read  $1.500 +0.015/-0.005$ . Positional tolerances might also be applied to coaxial features—two adjacent turned diameters, for example—or a pair of symmetrical features such as slots or milled bosses.

Less common are tolerances of profile for controlling the size, form, orientation and location of more complex features such as cam surfaces and airfoil shapes relative to their intended profile, as well as tolerances of runout, used to define the allowable deviation of one cylindrical feature to another.

**“You need to verify that your measurement approach is appropriate for the part geometry and its tolerance, and then double-check everything if there’s any doubt.”**

Patrick Sullivan  
Mitutoyo

With these are modifiers such as LMC (Least Material Condition), MMC (Maximum Material Condition) and RFS (Regardless of Feature Size), each of which can have a profound impact on a part's manufacturability and, therefore, cost. As stated before, ASME Y14.5 is a lengthy read, and anyone wishing to master metrological principles should have a good handle on this and other publications.

## High-Quality Measuring Equipment

If you'd like to learn more, Mitutoyo America Corp.'s *Metrology Handbook: The Science of Measurement* is a great place to start. That said, it represents but one small piece of the metrology equipment provider's extensive library of *on-demand learning* materials. The company also offers *YouTube* videos, as does its distributor, *MSC Industrial Supply*.

Mitutoyo strategic distribution sales specialist Patrick Sullivan has plenty to say about part measurement and tolerancing. Education on the topic is a little like eating Jell-O with dinner, he

says—there’s always room for more—and he strongly encourages shops of all experience levels to stay current on new ***technology, standards and equipment***.

“One of the biggest considerations is the need for high-quality measuring equipment,” he says. “A lot of times, manufacturers will attempt to save a few bucks by going to lower-priced tools only to find they are less repeatable. It can be a painful learning experience and end up costing them more in the long run.”

## **Choosing the Right Measurement Approach**

Similarly, shops sometimes use the wrong measuring tool for the application and its tolerance requirements. For example, some vision systems are accurate to within a few ten-thousandths of an inch, but that doesn’t make them suitable for round shafts where triangulation and other out-of-roundness conditions might be present.

“It’s easy to get caught up on a system’s ease of use or non-contact capabilities, not realizing it’s the wrong tool for that particular part feature,” Sullivan says. Put another way, simpler is sometimes better.

In the shaft example just given, a Mitutoyo V-Anvil micrometer would easily pick up any triangulation, as would a V-block and dial indicator, while a ring gauge would provide a functional assessment of the shaft diameter. In situations like these, multiple measurement methods are often needed to ensure compliance.

“You need to verify that your measurement approach is appropriate for the part geometry and its tolerance, and then double-check everything if there’s any doubt,” he adds. “That’s part of the First Article Inspection (FAI) process, which includes ensuring Gage Repeatability and Reproducibility (Gage R&R). Both require the manufacturer to use the right tools for the measurement job. Basically, don’t put all your eggs in one basket.”

**What are your biggest machining tolerance challenges? Tell us in the comments below.**

[www.mscdirect.com/betterMRO](http://www.mscdirect.com/betterMRO)

Copyright ©2024 MSC Industrial Supply Co.