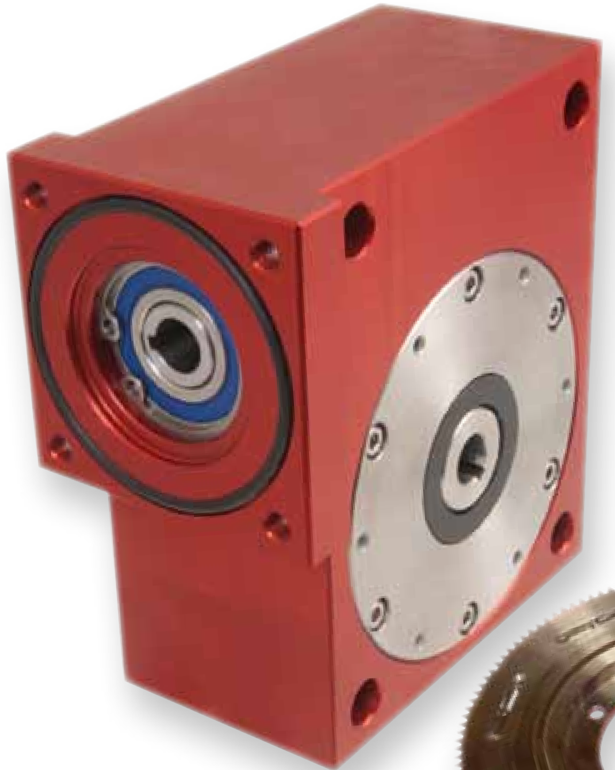
Working with DIMENSIONAL TOLERANCES



Understanding the nuances of tolerances makes life easier for engineers.

Applying dimensional tolerances to mechanical parts helps ensure proper fit and function.

Authored by:
Stefan Menin
Mechanical Engineer
Ondrives.US Corp.
Freeport, N. Y.
Edited by **Kenneth J. Korane**
ken.korane@penton.com

Key points:
- Always use dimensional tolerances.
- By using suitable tolerances, engineers pass the responsibility for making the part correctly to the manufacturer.
- Don't expect machine shops to verify untoleranced dimensions.

Resources:
Ondrives.US, *www.ondrives.us*

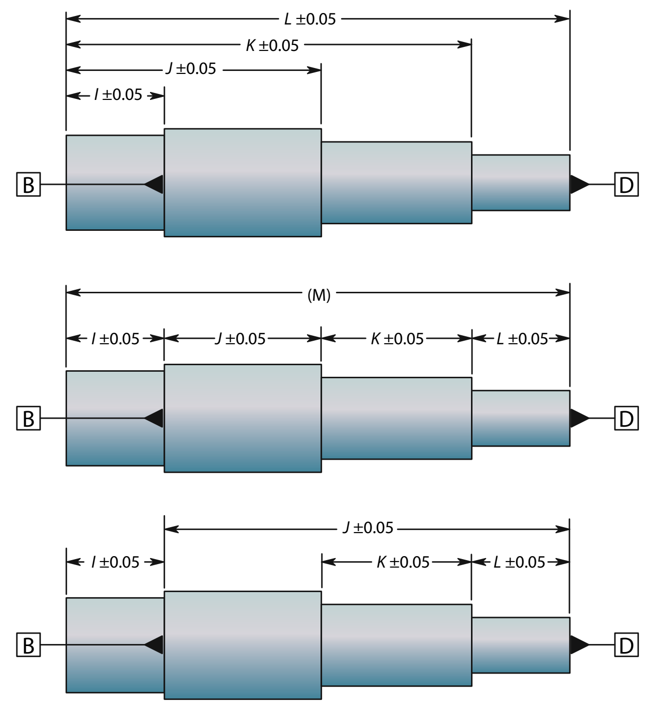
RS# 622

Engineers who discount the importance of dimensional tolerance do so at their own risk. It's like ignoring the odds in Vegas, and often with the same disappointing results. For instance, design a 0.875-in.-diameter shaft to mate with a precision bearing, send the drawings to a reputable and experienced manufacturer and, without specifying tolerances, the bearing may or may not fit on the finished part.

How can this happen? A machine shop that sees an untoleranced diameter, without knowing the design intent, may apply a standard tolerance for three-decimal-place untoleranced dimensions, ±0.005 in. Unfortunately, this may result in interference, where the hole is smaller than the shaft diameter, which prevents the parts from sliding together. They might have to be pressed together, but if too large of an interference exists, it will degrade performance, especially in bearings. The manufacturer will gladly offer to rework the pieces to fit — for an additional charge — which will extend delivery times and possibly lead to missed deadlines and budgets.

And make no mistake, this happens all the time. While many machine shops take due diligence to verify nontoleranced dimensions, many don't. Engineers who know the importance of tolerances, and how to use them, can easily avoid this problem.

Tolerance accumulation



Be aware of tolerance accumulation. As shown here, although every length has the same tolerance, overall length can vary, depending on how tolerances are applied.

Understanding basics

If parts did not need to interact with each other, or if the world were perfect, there would be no need for tolerances. But parts are made either from larger pieces of material or built up from a powder or liquid, so there is no guarantee they will be exactly the size one requires.

It's important to understand what tolerance is. ASME Y14.5M defines it as "the total amount a specific dimension is permitted to vary. The tolerance is the difference between the maximum and minimum limits." This can be shown as upper and lower limits $\binom{0.2500}{0.2498}$ or an allowable amount above and below a nominal dimension ($0.2500^{+0.0000}_{-0.0002}$, 0.2499 ±0.0001). Both of these methods define the same range of allowable dimensions. In this example, a finished part is acceptable when its dimension is anywhere between 0.2498 and 0.2500 in.; outside of this range, it is rejected. This range of allowable dimensions is the tolerance band. The larger the difference between the upper and lower limits, the larger the tolerance band, referred to as a "looser" tolerance. Conversely, a smaller tolerance band is considered a "tighter" tolerance.

Always use tolerances. *Always*. Ambiguity is not the friend of the mechanical designer. If you leave a dimension without a tolerance, no one else will know the importance, or unimportance, of the dimension.

Benefits

When tolerances are used correctly, engineers have much to gain. They ensure that parts will fit properly and work as designed within an assembly, be it a sliding or press fit. They can also reduce costs. Unnecessarily tight tolerances make parts more expensive to produce. There is no reason to apply a ±0.0002 tolerance when ±0.002 will do. Also, although some manufacturers apply their own standard tolerances to nontoleranced dimensions, many will not even begin making parts until all features are defined, which consumes valuable time and possibly pushes out delivery time.

Expecting parts to be made to the machinist's best effort is not acceptable. The machinist does not know how parts interact, nor is that his or her responsibility. Furthermore, one machinist's "best effort" may be maintaining features to within a few ten-thousandths of the dimension indicated, whereas another may make the feature 0.015-in. larger or smaller than indicated.

Tolerances should not be used with hesitation. Just because a larger tolerance band is used, it doesn't mean parts will be sloppily made. In fact, depending on the manufacturer's standards, shipped parts might have even tighter tolerances than originally specified. One good example is the bore of a gear. The specification might be $ø0.250^{+0.000}_{-0.002}$ but the manufacturer may machine the bore to a tighter tolerance of $ø0.2500^{+0.0000}_{-0.0005}$ simply because it is the particular manufacturer's standard and this tighter tolerance is critical to the gear-cutting process.

Another significant benefit is that by using suitable tolerances, engineers pass the responsibility of making the part correctly to the manufacturer. But remember, it's not the manufacturer's job to figure out design intent. If the part is within tolerance but doesn't fit or function properly, the manufacturer cannot be held accountable. Dimensions without tolerances leave the acceptable limits open, and it's not the manufacturer's responsibility to determine what is acceptable.

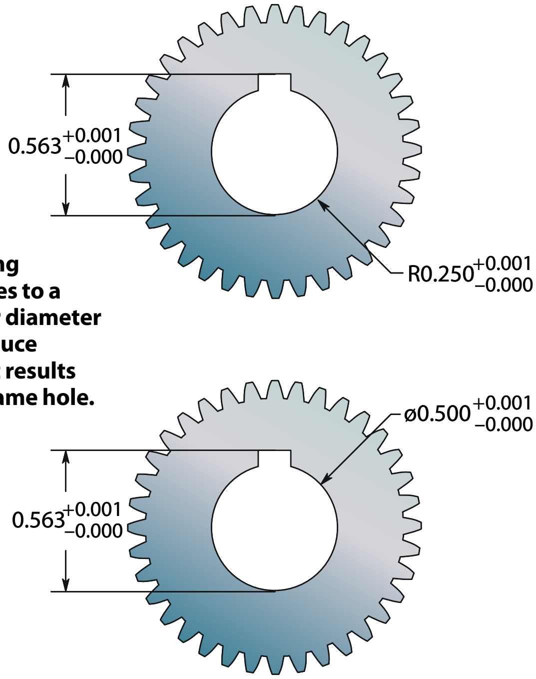
Important considerations

One of the most critical considerations when applying tolerances is to take into account fits. This refers to how shafts will fit into bearings or bushings, motors into pilot holes, and so on. Depending on the application, the part may require a clearance fit to allow for thermal expansion, a sliding fit for better positioning, or an interference (press) fit for holding capability. Information on limits and fits (among a plethora of other information) can be found in the 29th edition of "Machinery's Handbook" for both U. S. customary units and standard ISO fits.

Another consideration is how "tolerance accumulation" (also called "tolerance stack") affects a part. For example, take a shaft with four sections, each of a different diameter (as shown in the "Tolerance accumulation" graphic.) Although every length dimension has the same tolerance, the tolerance between surfaces B and D can be as large as ±0.15 (top figure) or as low as ±0.05 (bottom), depending on where dimensions are placed. It is up to the designer to decide which lengths are critical to the part's function.

Be careful when applying tolerances to a radius or diameter. A tolerance on a radius will double when measured as a diameter. A tolerance on a radius might be looser than intended, while one on a diameter might be tighter than intended. The effect of this is illustrated in the two gear drawings. If the part is manufactured to the dimensions in the top drawing, a hole diameter of 0.502 is acceptable. In the lower drawing, it's not.

Hole tolerances



Specifying tolerances to a radius or diameter can produce different results for the same hole.

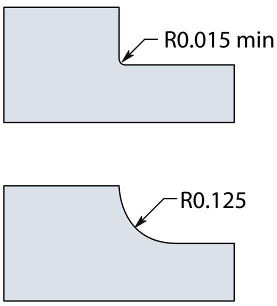
Also, one need not assume measurements will be rounded when determining if a part conforms to the specified tolerance. If a part is measured to be 0.2502 using a dial micrometer or other device, and the part's dimension is supposed to be $0.250^{+0.000}_{-0.002}$ the dimension is not rounded down to three decimal places; it is considered nonconforming. ASME Y14.5M states dimensions "are used as if they were continued with zeros," even if not shown.

Also take into account any plating or finishing processes the part requires. A note should indicate if dimensions apply before or after such processes.

When using either MIN or MAX limits, ensure that if the dimension approaches infinity (in the case of MIN) or is zero (in the case of MAX), it does not hinder the design of the part. The "Corner radius" illustration uses a MIN tolerance, possibly to ensure a radius will reduce stress concentration. However, the lower figure shows a dimension that is within tolerance, but it may hinder the part's functionality. Other features should clearly define unstated limits.

Both the location and size of alignment holes (such as for dowel pins) should not carry the same tolerances as clearance holes (such as for screws to pass through).

Corner radius



When using minimum or maximum tolerances, make sure limiting dimensions do not affect the part's intended function.

A certain deviation from nominal in the location of a dowel pin hole may result in parts that are impossible to assemble, while the same deviation in the location of clearance holes will likely cause no effect to the entire assembly, except for perhaps a near-imperceptible aesthetic oddity.

While thoroughly dimensioning parts is important, one must avoid redundancies. They may cause conflicts in inspection because certain features will be defined more than once in more than one way. If a dimension that overdefines the part is desired, use a reference dimension between parentheses and usually without tolerances, like dimension (M) in the middle shaft drawing. This dimension is derived from others or is repeated, usually in a different view.

Dimensional tolerances are key in making parts right. Using them appropriately will save time spent coordinating with the manufacturer, circumvent design issues, and reduce unnecessary costs. However, while dimensional tolerances are important, so are geometric tolerances in fully defining parts. They deal with geometric features and relationships between features such as form, profile, orientation, and runout, and cannot be defined by dimensions alone. This topic will be explored in a future article. **MD**