Geometric Dimensioning and Tolerancing

The Common Thread of a Multifunctional Design Team

By Donald E. Day, Chairman, Mechanical & Quality Technologies
Monroe Community College, Rochester, NY

ABSTRACT

Once the design team has performed their DFA/M analysis, it is essential to document the solutions. Old methods of dimensioning and tolerancing leave the engineering document open to interpretation. The national standard on Geometric Dimensioningand Tolerancing (ANSI Y14.5M-1982) provides a method of defining parts with one clear meaning. A review of the dimensioning and tolerancing of parts by the multifunctional team answers the questions related to reliability, tooling, inspection, function, etc. prior to release of the drawing. Numerous examples will be presented that illustrate how this approach has reduced costs and design time while improving quality.

The engineering drawing or file is a legal document. It is a contract between departments or customers and vendors. Just like any other legal document, it should represent the drafter's intent and be free of any **loopholes**. It is estimated that over 80% of the engineering documents generated in the United States are flawed in some way. Often the ideal geometry is well defined. It is the amount of acceptable variation or tolerance from the ideal or nominal geometry that is usually unclear. In most companies the responsibility of assigning tolerances is left up to the drafter or CAD operator without significant input from the areas that must live with the specifications.

Problems resulting from overly tight tolerances, unclear geometry and non-producible designs include:

Scrap

Rework

"Use as is" decisions

Requests for engineering changes

In addition to creating a Robust Design that is insensitive to noise (variation) in the factory and field through DFA/M analysis, the multi-functional team should concern itself with:

1. Defining the ideal geometry on the drawing or CAD file
2. Applying critical and producible tolerances where needed
3. Maximizing production tolerances without sacrificing quality and reliability

Defining the ideal geometry on the drawing or CAD file

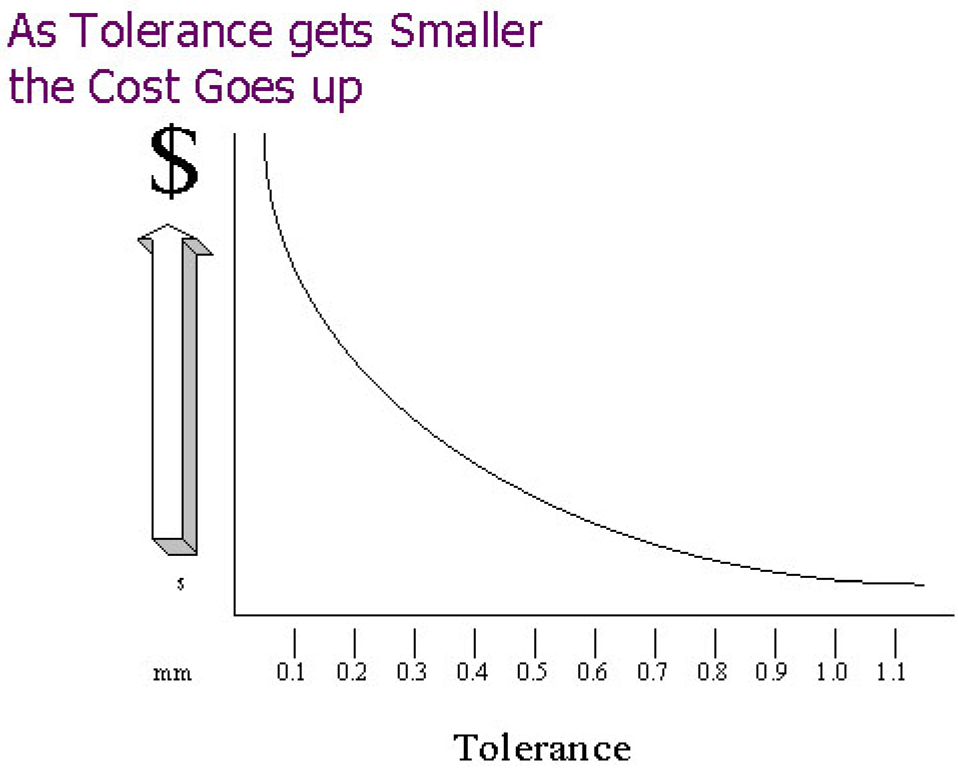
Designers and engineers do a pretty good job of defining the Basic or nominal geometry. This geometry is transferred to the model shop via a drawing or downloadable CAD file to produce prototype parts. Great care is usually taken to hold dimensions to nominal or Basic dimensions. A functioning prototype is taken as a signal that the geometry is correct and it is safe to "sign off" on the design. Unfortunately, manufacturing has to contend with flash, burrs, draft, tool wear, shrink, sink, etc. that cause variations in the parts geometry that are often not present in prototype models. Process capability studies and statistical process control help reduce these variations but cannot eliminate them. The design, therefore, must tolerate some degree of variation.

Applying critical and producible tolerances where needed

Often tolerances are brought forward from old designs or looked up in references that were written back when 10% scrap was the norm (Acceptable Quality Level — AQL). Today manufacturing is being told to produce "zero defects". Designers are led to believe, for example, that a reasonable size tolerance on a drilled hole is +.004, -.001. A shop making this hole and being told they must demonstrate a process capability index of 1.33 or 2.00 is forced to ream or bore the hole after drilling to meet the customer's requirements. Often, especially on clearance holes, the tolerance can be several times the recommended practice and not in any way affect the performance of the assembly.

Generally the geometric characteristics of 10-20% of the parts features are critical to its fit and function. If not controlled adequately, poor product performance and reliability will result. There is usually a tradeoff between tightening the controls or tolerances and part cost. See Figure 1.

Tolerances determine the process.



In general:

The tighter the tolerance--

The more expensive the process

For decades the battle cry of design engineering has been "function determines the tolerances". The best design in the world is worthless if it cannot be produced. Granted, tolerances must be limited to assure that functional requirements are met. On the other hand, if the tolerances specified are tighter than the capabilities of the available processes, the design is not 100% producible. In other words, manufacturing is destined to produce scrap. Involving design, manufacturing, quality, vendors, purchasing agents and others in the selection of critical tolerances assures a producible design that will meet the needs of the customer at the lowest possible cost. If the members of the team are involved with the assigning of tolerances that are practical, they will appreciate the reasons for meeting the specifications of the critical characteristics.

Geometric tolerancing provides a means of documenting the tolerancing conclusions arrived at by the design team. By utilizing the concept of material condition modifiers, it is possible to maximize production tolerances without sacrificing quality and reliability. Assigning proper datums to parts assures repeatability and reproducibility of measurements for process control and quality assurance. In order to properly apply geometric dimensioning and tolerancing, several questions must be answered. These include:

- which features are most critical
- will the effect on the part's performance be altered as a feature's size varies
- which feature interrelationships are critical
- which features are not critical
- what inspection methods will be used to control the process and are they adequate
- are several processes used to manufacture the part
- are the tolerances reasonable
- is it producible
- what process or processes will be used
- is the entire feature or only a portion critical
- is the main concern:
    - interchangeability
    - alignment of multiple parts
    - maintaining a minimum wall thickness
    - providing minimum material for a subsequent machining or forming operation
    - dynamic balance
    - feature location, orientation, form or size
    - control of a pattern of features

No one person can possibly have the answers to these questions. It requires the input of the multifunctional team to properly define a part's requirements. All too often these questions are not asked until the tooling produces a bad part, the fixture or gage isn't working or the customer complains. When the questions are asked early in the design cycle, the answers can be documented at the cost of adding a symbol, line, number or note to a drawing or CAD file. If the questions are left unanswered until late in the product cycle, the cost of the answer is usually an engineering change costing hundreds if not thousands of dollars.

Involving the team in tolerancing:

- **increases their awareness of the product's needs**
- **causes the right questions to be asked**
- **is an educational experience for all those involved**
- **provides two-way communication between departments**
- **makes those involved feel like a part of the solution — not the problem**
- **gives everyone a vested interest in making the product work**

Maximizing production tolerances without sacrificing quality and reliability

Once the critical characteristics of the part and its features have been identified and controlled, the tolerances of the remaining part features should be established at maximum values where possible. Maximizing the production tolerances of noncritical features assures that needless expense is not incurred to produce those features. This activity often does not require the involvement of the entire team.

Case Studies:

Case 1 -- Castings

A manufacturer of gear cutting equipment estimated that their machining center was idle 20% of the time while the operator shimmed the set up of castings. This activity seemed necessary to assure that once the machine started it would clean up the desired cast surfaces. If the casting did not clean up, they would blame the foundry for not providing sufficient material as a machining allowance. The foundry in turn would blame the machinist for not aligning the casting properly on his machine tool. By applying datum targets to the print that were representative of the machining center's fixturing, the company was able to communicate to the foundry how the casting would be staged and where they expected material for machining allowance. In addition to eliminating idle machining center time, both the company and the foundry can now determine where the process needs correction when their respective processes begin to drift out of control. Identifying the appropriate datum targets to stage the part required input from the foundry, the machining area, quality assurance and product design.

Case 2 -- Sleeve Bearing

A recent design review involved a part similar to the one illustrated in Figure 2. The part is a sleeve bearing used in a DC motor. Two of these bearings are pressed into the moter end caps to hold the armature shaft. The alignment and position of the holes in the bearings is critical to the fit andfunction of this assembly. When it was suggested that geometric tolerancing be added to the hole in order to control its location and attitude relative to theoutside surface which was to be pressed into the end caps of the motor, the design engineer insisted the drawing not be touched. The argument given was that the vendor had submitted sample bearings which had passed a life test with flying colors. As far as the design engineer was concerned, this drawing must be okay since the sample parts, made to the drawing, passed the test. They intend to make three million motors with two bearings per motor. Imagine being a vendor who is asked to submit sample bearings with the possibility of receiving a six million part contract. The vendor submitted the best bearings he could find — as near perfect as possible and not representative of production bearings.

Several months later during the first production run a problem of shafts binding in the bearings appeared. The problem was due to the hole in the bearing being too eccentric to the outside diameter. As a result, they changed the drawing (Figure 3) to include a geometric tolerance controlling the hole relative to the outside diameter (datum -A-). The change would have cost a few minutes at a CAD terminal prior to release. At this point in the program, the cost is not only for the drawing change but also for scrapped parts and a new tool.

Case 3 -- Throttle Pulley Sector

The drawing labeled Figure 4 is of a throttle pulley sector used to guide throttle cables around a 90o corner. This part is used on a popular two-engine passenger jet flown by several domestic airlines. The two counterbores shown in the sectional view receive bearings which determine location and orientation of the part in the next assembly. The bearing bores and the 7.062 radius are machined in different machines and fixtures. There is nothing on the drawing relating the radius to the axis of the counterbores. When the eccentricity of this radius to the bearing bores is too great, the throttle cable can walk off the sector past the guide pins. To date on at least four separate occasions this has occured causing one of the two engines to, without warning, shut down. On every occasion the aircraft was carrying passengers but fortunately was able to safely land powered by the remaining engine. Figure 5 illustrates the use of geometric dimensioning and tolerancing used to control the radius relative to the bearing diameters. If the machining area had been made aware of the critical relationship of the radius to the bores, their approach to the processing of this part would have taken into account this functional relationship.

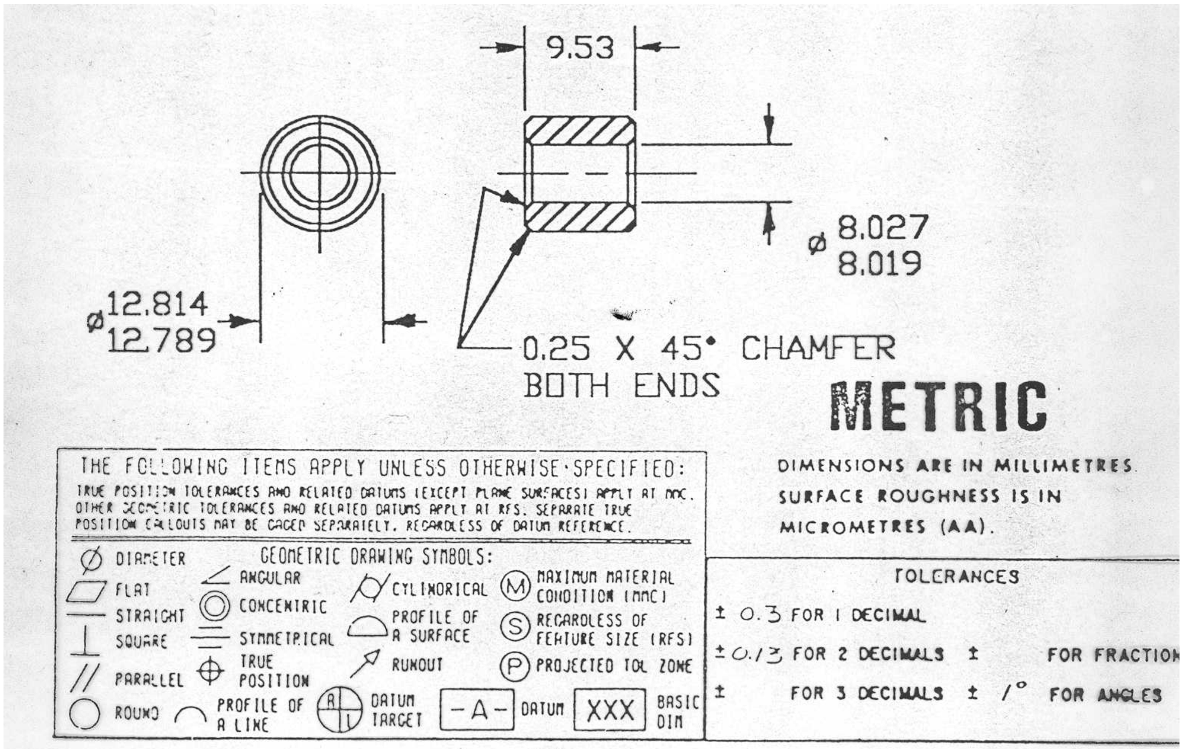
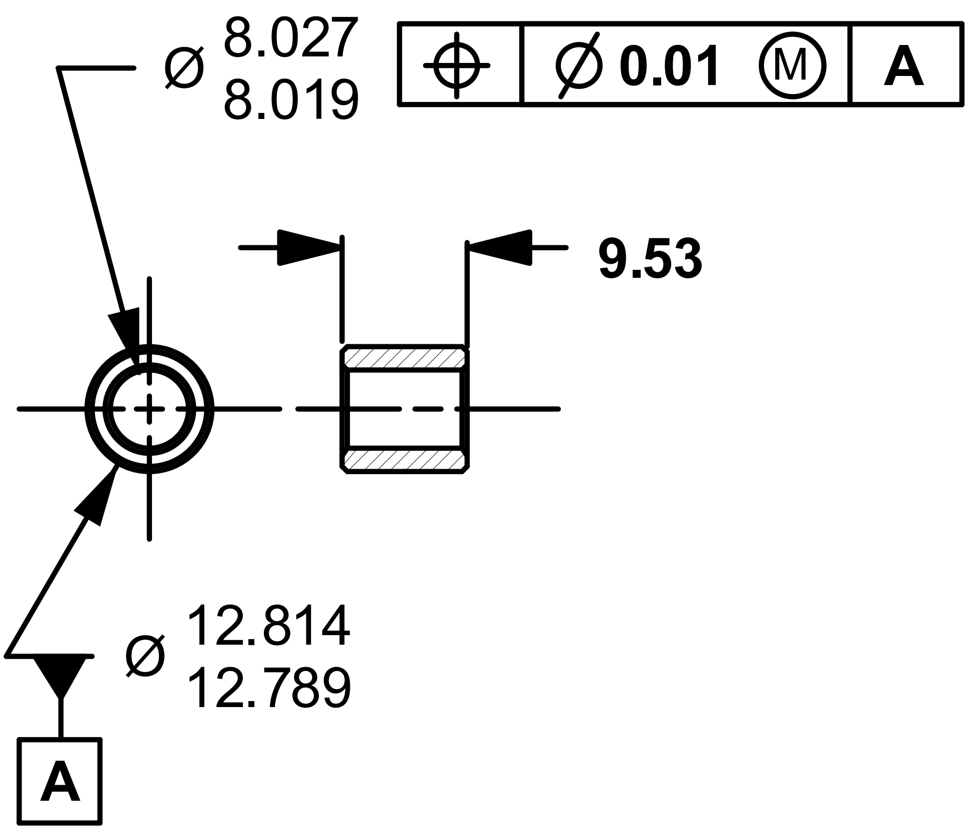


Figure 3



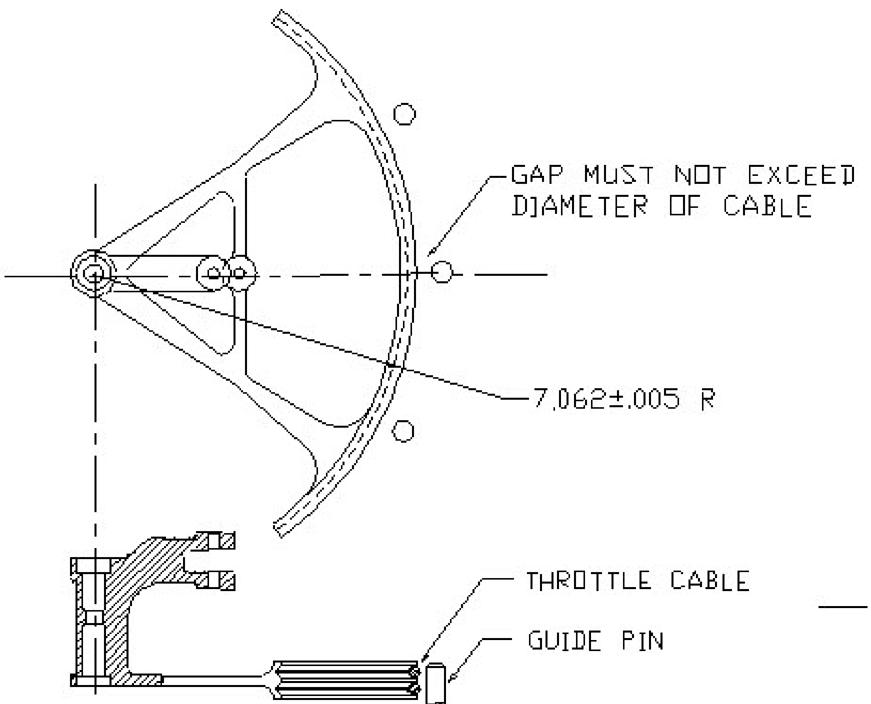
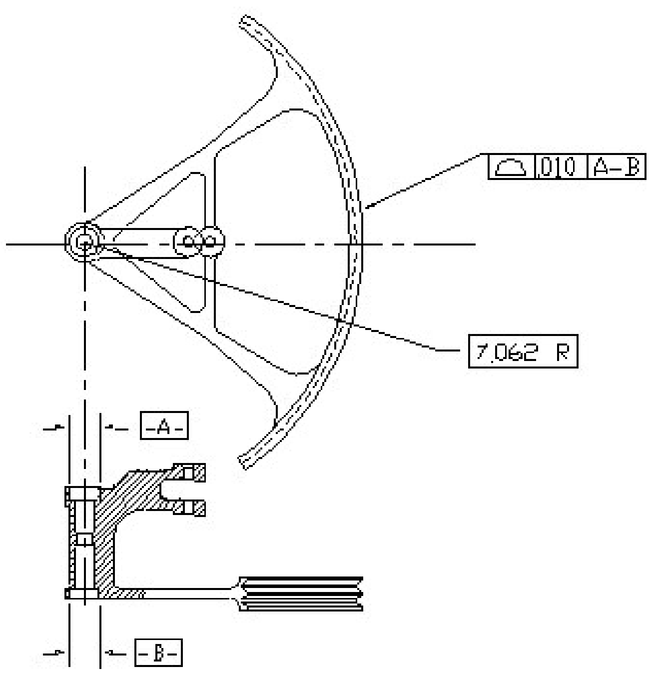

GAP MUST NOT EXCEED
DIAMETER OF CABLE
7.062±.005 R
THROTTLE CABLE
GUIDE PIN

Figure 4


.010 A-B
7.062 R
-A-
-B-