Drawing quality is an important issue for manufacturing. Flawed drawings can mean reworking parts to meet blueprint specs, creating excess scrap, and even scrapping parts that were functionally acceptable. Poor drawing quality can easily double manufacturing ramp-up costs and delivery time for that first product. Most manufacturing and quality engineers agree that half the part drawings they see are flawed and require five to 10 engineering changes before they define a manufactureable part.

Common drawing problems include undefined reference frames, inadequate or overly restrictive tolerances, and bilateral rather than positional tolerancing of bore locations. Frequently, manufacturing must make assumptions about a poorly drawn part, hoping to process it like a similar part. The result is often non-functional parts that cause production and delivery delays and expense for negotiations and rework.

The Price of Ambiguity

Creating unambiguous part drawings is a demanding thought process. The function of the part—and the processes to manufacture and inspect it—must be thoroughly understood and communicated. If a designer intends for a part to be inspected with side A presented to the gage first, with side B as a secondary point of contact, but the inspector puts side B to the gage first, the result can be good parts scrapped and nonfunctional parts approved. It's clear that creating drawings that define functional, manufactureable, and inspectable machine parts should be a high priority.

Which is more economical: getting drawings done quickly and working out the bugs later, or spending more time on drawings to eliminate production delays and rework? Assume two engineers produce a drawing for the same part. The first uses coordinate dimensioning, the second, GD&T (actually American National Standard Y14.5M). The first engineer spends 20 hrs and produces a flawed drawing. The second spends 30 hrs and produces a drawing that includes a thorough tolerance analysis as well as checks with manufacturing and quality engineering. If drawing cost is $50/hr, the GD&T drawing costs $300 more.

Coordinate dimensioned drawings are faster to draw, but too often flawed and expensive. Geometric dimensioning and tolerancing takes more thought, but the payoff for manufacturing is big.

Alex Krulikowski
President, Effective Training Inc.
Westland, MI

Will the GD&T drawing's extra cost be justified by production savings? The paperwork costs for drawing revisions range from $500 to $2000 per revision, not including costs for scrapped parts and tooling or gaging changes. If the quicker, 20-hr drawing requires five to 10 changes to make it interpretable, these costs must be added to com-
1. Geometric tolerancing example.

2. Coordinate dimensioning of same part.

The real costs of each drawing. Assuming six revisions at $1000 each, the total cost for the 20-hour drawing would then be $1000 + $6000 = $7000. Thus the 30-45 hr drawing at $1500 and no changes saves $5500, with additional savings in ramp-up time and meetings by highly paid personnel to determine what the designer really intended. Given these considerations, the GD&T drawing is a real bargain.

GD&T’s increasing popularity with the real costs of each drawing. Assuming six revisions at $1000 each, the total cost for the 20-hour drawing would then be $1000 + $6000 = $7000. Thus the 30-45 hr drawing at $1500 and no changes saves $5500, with additional savings in ramp-up time and meetings by highly paid personnel to determine what the designer really intended. Given these considerations, the GD&T drawing is a real bargain.

GD&T’s increasing popularity bears this out. "ANSI Y14.53 is our most popular standard," says Cal Gomez of the American Society of Mechanical Engineers (ASME). "GD&T is now used by most medium to large-sized manufacturing firms in the US."

Yet there is little comparative data to show the benefits of quality drawings, and current accounting methods fail to reveal the expense of poor-quality drawings. Without such information, management may be reluctant to fund the training programs needed to make GD&T work.

Training is crucial in GD&T, and the burden of training falls on the manufacturing sector. Engineering students often have several course offerings in traditional coordinate dimensioning, but only a single course in GD&T and little focus on its application.

Fluency in GD&T for the design engineer requires a minimum of 80 hrs of interactive classroom sessions, testing, and six months of on-the-job training plus occasional refresher courses. Others, such as machine programmers, shop supervisors, and quality engineers, require about 60 hrs of instruction over a six-month period to deal effectively with ANSI Y14.5 drawings.

Lack of training causes conflicts between design and manufacturing over GD&T interpretation. Designers with only a brief exposure to GD&T applications create drawings that consume hundreds of thousands of dollars in company resources. It’s not surprising that they simply copy dimensions and tolerances when they haven’t the training to determine them otherwise.

What’s a Quality Drawing?

Let’s compare drawings done in geometric tolerancing and coordinate dimensioning. Are both complete, precise, and unambiguous?

Figure 1 shows a part dimensioned in GD&T, and Figure 2 the same part in coordinate dimensioning. Do they both say the same thing?

- Part straightness. Both drawings specify a part thickness of 0.250 ± 0.010. But how straight must the part be? Figure 1 includes a geometric tolerance (straightness symbol) applied to the thickness (area A). This symbol means that the part must be straight enough to pass between two parallel plates spaced 0.250" apart.

The coordinate version of this drawing says nothing about straightness. A part could be within the thickness tolerances, yet significantly bowed and unacceptable. To ensure straightness, the coordinate drawing must include a note stating: "The part must pass through two parallel plates spaced 0.250" apart."

- Unwanted tolerance accumulation. Figure 1 includes several basic dimensions, a vital element of geometric tolerancing. They identify exact locations from which tolerances are specified. Coordinate dimensioning has no corresponding term—tolerances can accumulate, which is highly undesirable. Because of this, another note must added to the coordinate dimensioning version to keep tolerances from accumulating: "Dimensions marked with * are gage dimensions. For part tolerances see the notes relating to part features."

- Part inspection. As noted previouly, a can yield coordinators established to inspect the this process. Contact datums, tolerances defined surfaces, which to the first touch, third last.
3. Effect on dimension X of how part is placed in gage.

4. Coordinate square tolerance zone versus the bonus tolerance with GD&T's round zone.

5. The closer the hole, the greater the GD&T bonus location tolerance.

metric tolerancing version. "For part inspection, mount the part in a set of three mutually perpendicular planes. Surface A contacts one plane first; surface B contacts the planes second; and surface C, third."

- Hole locations. The coordinate drawing shows two ways in which that dimensioning is inadequate for defining hole locations. First, coordinate dimensioning uses a square tolerance zones, Figure 4, which allow more tolerance for hole centers diagonally than vertically or horizontally. A more logical and functional approach is to allow the same tolerance for the hole centers in all directions, creating a round tolerance zone.

Moreover, coordinate dimensioning requires the tolerance zone for hole location to remain fixed in size regardless of hole size. When part function is considered more logically, however, it is clear that the hole location tolerance zone is most critical when the hole size is smallest. When the hole is larger, its location tolerance can be correspondingly larger without affecting part function.

Square tolerance zones and fixed-size location tolerance zones cause thousands of functional parts to be needlessly scrapped, whereas geometric tolerancing allows them to be used.

The geometric tolerancing drawing (D) contains a symbol (a circle with a slanted line through it) that denotes the shape of the tolerance zone as a diameter. Note the size of the diameter zone is equivalent to the cross-corner distance of the square tolerance zone in Figure 4. This round zone allows 55% more tolerance than the square zone.

The GD&T location tolerance includes a circle-M symbol (maximum material condition) that permits location tolerance to increase as hole size increases. This is extra or "bonus" tolerance (Figure 5) that prevents the scraping of functional parts.

Since coordinate dimensioning doesn't provide for round location zones or extra tolerances, these concepts must be simulated by another note: "Locate the part per hole 1."

When the holes are the smallest diameter, the center of each hole must be located within a 0.025 diameter cylindrical tolerance zone. When a hole is larger than the minimum diameter, subtract the minimum diameter from the actual diameter and add this amount to the permissible tolerance zone. The hole center must be within the larger tolerance zone diameter."

Counterbore location. The geometric tolerancing version specifies counterbore diameter, depth, and location. The coordinate version only specifies bore diameter and depth, allowing the user to choose between two conflicting location interpretations. Follow the location of the small diameter or locate by the tolerated dimension for both diameters. Both are logical interpretations, but only one is intended by the designer.

The geometric tolerancing drawing eliminates this assumption by including a position symbol under the counterbore (C) specifying that the location is relative to the small diameter. To make the same point in the
6. Notes required to make coordinate drawing equivalent to GD&T drawing.

coordinate version, a note must be added. "Verify the location of each counterbore with a stepped gage, the small diameter gage of 0.271 and the large diameter of 0.413 must be in the part diameters freely."

Which Is Better?

Which is simpler, better, and faster? Figures 6 illustrates the notes it would take to make the coordinate dimensioning version similar to the GD&T version. At best, this is only an approximation—the geometric tolerancing version communicates even more information, and this is a very simple demonstration part. Imagine how complicated a drawing for a complex part would be.

Yes, coordinate dimensioned drawings can be very simple, based on assumptions, imperfections, unnecessary tight tolerances, and implementation by lengthy notes, hard to understand. Some designers complain that geometric tolerancing is difficult to use, in fact, it's the job of fully defining the part that's difficult. As the comparison of Figures 2 and 6 shows, GD&T makes the job of completely defining the part much easier.

Geometric tolerancing takes time to master. Quality drawings take more time to create, but the savings in manufacturing time, costs, and usable parts are substantial. Geometric tolerancing deals more effectively with the complexity and stringency of tolerance requirements, which are often not specified in the drawings. As a result, the work of tolerancing designers is more important than ever.

Getting Antsy for ANSI Y14.5M?

A major revision to the geometric dimensioning and tolerancing standard, ANSI Y14.5M, is currently being finalized. This will help support the new technologies of computer graphics (CAD/CAM), electronic packaging, and statistical process controls. The ANSI standard is updated by a world committee at ISO (International Standards Organization). A number of differences in evolution and the ISO policy of "one subject, one standard," the "counterpart" of ANSI Y14-1982 is a number of ISO standards (ISO 1101, geometric tolerancing; ISO 5458, positional tolerancing; ISO 5459, datum; and others).

The three highest proposed changes in ANSI Y14.5M are modifying the datum identification to make it more in line with ISO principles, eliminating the RFS (regardless of feature size) modifier, and expanding composite tolerancing beyond the ISO standard.

ANSI Y14.5M hasn't been updated since 1982. The Y14.5 subcommittee has gathered several issues impacting the standard: computer technology, mathematics, and new measurement techniques (both the orientation and location of holes patterns).