Dimensioning and tolerancing

What is GD&T? Is it Greek? Hieroglyphics? With all the little symbols and numbers within, it must be some kind of code. Well, it is a code, sort of. GD&T (or GDAT) is the Geometric Dimensioning and Tolerancing language used on engineering drawings for communication between design, production and quality control. GD&T certainly is not new since it has been around, in some form or another, longer than most of us realize.

GD&T is a national and international standard. It is based on the dimensioning and tolerancing standard ASMEY 14.5M-1994. ASME is an acronym for the American Society of Mechanical Engineers and GD&T is primarily a mechanical standard. According to ASME, GD&T helps "to harmonize the United States' practices and methodology with the universal standards trends toward more efficient worldwide technical communication." Use of this language or tool "can provide economic and technical advantages" stated the ASME.

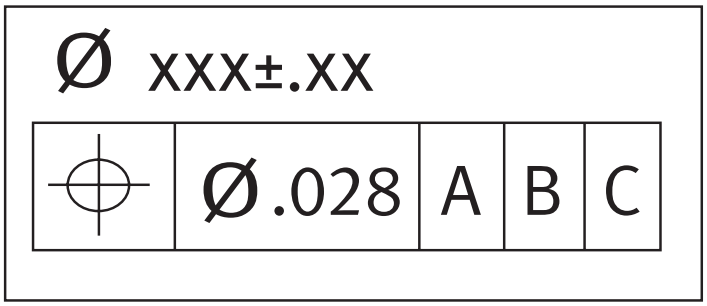

Figure 1: Codes look puzzling? This is an example of Geometric Dimensioning and Tolerancing.

Mechanically inclined

A PCB is a mechanical piece that is relatively simple. Typically, a PCB is a rectangular, flat piece with no notches or cutouts. Please do not start calling and e-mailing me about how complicated your board outline is. I realize there will always be exceptions. But the fact remains that mechanically speaking, PCBs are relatively simple compared to sheet metal stampings, conical gear teeth and worm gears.

Because GD&T takes so much mechanical engineering information into account, including plenty that is of no use to the PCB designer, many designers are scared away from this standard. Designers who read a book or take a college course on GD&T can come away overwhelmed by it.

But board designers do not have to understand every aspect of GD&T. If you select the parts of the standard that apply to rectangular, flat pieces, GD&T will become understandable and useful to you. The IPC-2615 "Printed Board Dimensions and Tolerances" describes the parts of the ASME Y14.5M-1994 that apply to PCBs. And, as with any language, the more you use it, the easier it becomes.

More tolerance is good

Now that we have gotten past those issues, let us move on to the really big reason for using GD&T on your PCB fabrication drawings. When used properly, GD&T will give your board manufacturer increased tolerance, creating a more manufacturable product at lower cost, while not adversely affecting the final fit of your finished product. It will also allow you to convey to your fabricator which holes, slots or edges are particularly important to your final product.

Now, before you say, "I do not want to give my board fabricator increased tolerances. I want my board to fit into its final package," let us look at how you dimension boards now.

Typically, the location of any hole, slot or feature is dimensioned as some location plus or minus a set amount. In our example, we will use location x dimension and location y dimension plus or minus 0.254mm. What you have effectively said to your board fabricator is that, as long as the center of this hole is anywhere within this 0.508mm square zone, the part is acceptable. Conversely, if the center of this hole is anywhere outside of this 0.508mm square zone, the part is not acceptable. Pretty simple, right?

Well, let us look at this a little closer. If the center of this hole is x +0.254mm and y +0.254mm, it is still within the square and therefore acceptable. When you measure this hole location, it is a total of 0.356mm away from your target location. If this hole is x +0.0mm and y +0.279mm, it is outside the square and therefore not an acceptable part. When you measure this hole location, it is a total of 0.279mm away from your target location.

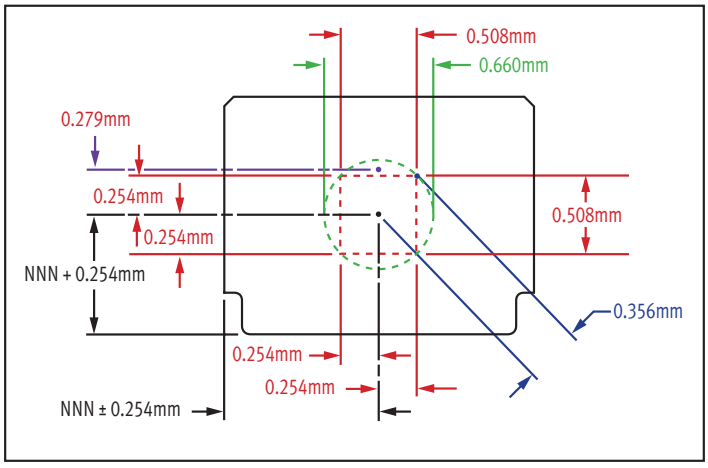

Figure 2: The tolerance zones indicate that if a hole is outside the square, it is not an acceptable part.

What you have effectively done is accepted a part with a hole located further away from your target than the part that you rejected. If, in fact, the first scenario provides an acceptable part, then the second situation is, in reality, a good part also. Conventional dimensioning techniques make the second part a reject.

By applying GD&T to this part, we can define a circular tolerance zone as opposed to a square zone. This will assure that you do not reject a part that is actually closer to your target values than an accepted part that is further away from your target values.

This may be an over-simplification but you get the point. By applying GD&T and defining a circular tolerance zone, you have increased the tolerance by 57 percent without affecting the fit of your final product. Before everyone who already understands GD&T decides to call or e-mail me, let me clarify this. The tolerance zone is actually a 3D cylinder. The top and bottom of this cylindrical tolerance zone are the top and bottom surfaces of the PCB.

Be sure not to go overboard when applying this standard. If you are not careful, you can use GD&T to "over-dimension" a part, which makes drawings confusing and can make the board too costly to manufacture.

It is difficult to summarize the ASME Y14.5M-1994 and its processes in a few paragraphs. There are many other advantages to GD&T, including bonus tolerances, the use of material modifiers and profile tolerancing, and the definition of your datum planes. I hope I have shed a little light on a dimensioning and tolerancing standard that, if used properly, will produce a more manufacturable board and possibly save your company money, too.

[Printed Circuit Design]

By John Sabo
Senior PCB Technical Analyst
Rockwell Automation
E-mail: cjkelly@ra.rockwell.com