Geometric Dimensioning and Tolerancing of Castings

There is nothing really new about Geometric Dimensioning and Tolerancing so I was surprised by a question an engineer asked recently concerning GD&T. I had just finished speaking to a group of engineers about investment casting design and the first question asked was to express my opinion of “GD&T casting drawings”. Admittedly confused I lamely answered with something akin to “I’m good with it” and felt fortunate that there was no follow-up question on the same subject.

Truthfully, I am so accustomed to working with Geometric Dimensioning & Tolerancing (ASME Y14.5) that I likely haven’t given it much thought for a couple of decades. Now having revisited the subject I believe my initial response the GD&T questions wasn’t too bad after all because I am “good with it” but didn’t provide much insight. So, in my opinion GD&T is the better method for an engineer to define part requirements than was the old Plus / Minus system. It is, however, a more difficult system to learn and interpret and as a consequence there seem to be greater numbers of errors made with GD&T.

I also believe that GD&T paved the way for dimensionless drawings that have created a much larger quandary for designers of cast products. (This, perhaps, with a little clarification, might have been the true subject of the question being asked of me?) However, for the purposes of this essay I’ll confine my attention to addressing Geometric Dimensioning and Tolerancing of a fully dimensioned casting drawing.

If you’ve worked with castings you likely realize that dimensional variation accumulates in the longer dimensions. The Investment Casting Institute has published a Linear Tolerance Table* that translates roughly to ±.005 inch of tolerance per inch of length. Therefore, a dimension of 2-inches would require a ±.010 tolerance and a 4-inch dimension ±.020. It is important to remember that a linear dimension is the distance between two points such as can be measured with a pair of Vernier calipers.

Let’s compare the two methods as to how one would dimension and tolerance three coplanar holes on a surface. One point of clarification: The following example considers a casting in an ideal state and so is presumed to be perfectly flat and square. As already discussed a
linear dimension is the distance between two points and not a point to a (datum) plane. In the "real world" tolerance consideration is 3-dimensional (-X-, -Y-, -Z-) and must also include the size and shape of the datum plane.

In a Plus / Minus drawing the dimensions to three coplanar cast holes might be tolerance as follows:

Hole A: Ø.50 ± .005 5.00 ± .025 from -X- 5.00 ± .025 from -Y-
Hole B: Ø.50 ± .005 2.00 ± .010 from -X- 2.00 ± .010 from -Y-
Hole C: Ø.50 ± .00510.00 ± .050 from -X- 10.00 ± .050 from -Y-

In GD&T drawing the locational dimensions become Basic and the feature (hole) is designated with a True Position (feature location) tolerance linked to a Maximum Material Condition:

Hole A: Ø.50 ± .005 5.00 Basic from -X- 5.00 Basic from -Y- 070 True Position MMC
Hole B: Ø.50 ± .005 2.00 Basic from -X- 2.00 Basic from -Y- 028 True Position MMC
Hole C: Ø.50 ± .00510.00 Basic from -X- 10.00 Basic from -Y- 140 True Position MMC

GD&T True Positions have circular tolerance zones where the tolerance zones in a Plus / Minus system are rectangular, so the GD&T system does provide a "smidge" of additional tolerance. Otherwise these two systems are pretty much equivalent.

Note, however, that a ± .010 locational tolerance is roughly equivalent to a true position of .028. Rookie engineers sometimes make the mistake of presuming that a .020 inch (±.010) of a Plus / Minus tolerance zone is the same size as .020 of True Position geometric tolerance where in actuality there quite a bit of difference. Geometric dimensions originate from the nexus or 0,0 point of the -X- and -Y- datums (it's Pythagorean’s value of c in the a² + b² = c²) where Plus / Minus systems dimensions are measured along an axis perpendicular to the datum plane (a & b).

It is readily apparent from the above tables that the dimensions furthest from the datums require significantly more tolerance in either system. This emphasizes the point that datum selection is an important consideration for the Tolerancing of a part regardless of the system being used. I can suggest that one way to help mitigate the stack up of tolerance is to centralize the datum structure.

Let’s revisit the previous scenario but this time with the nexus of -X- and -Y- datums better centralized and being coincident with the centerlines of Hole A

In the Plus / Minus system dimensions and tolerances for the three holes become:

Hole A: Ø.50 ± .0050.00 ± .000 from -X-0.00 ± .000 from -Y-
Hole B:Ø.50 ±.0053.00 ±.015 from -X-3.00 ±.015 from -Y-
Hole C:Ø.50 ±.0055.00 ±.025 from -X-5.00 ±.025 from -Y-

In the GD&T system the dimensions and tolerances become:

Hole A:Ø.50 ±.0050.00 Basic from -X-0.00 Basic from -Y-..000 True Position MMC
Hole B:Ø.50 ±.0053.00 Basic from -X-3.00 Basic from -Y-.042 True Position MMC
Hole C:Ø.50 ±.0055.00 Basic from -X-5.00 Basic from -Y-.070 True Position MMC

As you can see, the effect of centralizing the two datums is to average the cumulative deviation bilaterally, in two directions, instead of unilaterally and shifting all of the tolerance requirements to the “far end”. The centralized –X- and –Y- datums should create a more acceptable tolerance condition for the designer.

The ±.005 inch/inch linear tolerance was established by the Investment Casting Institute as a guideline to assist companies in the design manufacturable castings for its member foundries. O’Fallon Casting and many other Institute members manufacture castings to tolerances tighter than those being recommended by the ICI. We suggest that in instances where a designer is being driven to work inside of the ICI recommended tolerances, that they confer with a foundry to confirm their capability.

The use of Geometric Dimensioning and Tolerancing on a fully dimensioned drawing should not present an issue to a foundry. In any dimensioning system the size of the part and the location of the datum planes will have a direct impact on the amount of tolerance necessary for a manufacturable and affordable casting. Centralized datums will prevent the stack-up of part variation from manifesting itself in one side of the casting by averaging the deviations in two directions.

If you have any questions regarding Geometric Dimensioning and Tolerancing of castings, please contact your O’Fallon Casting Sales Engineer.