Profile Services
Oakbank, MB
Canada
(204) 619-3636
jdsykes@profileservices.ca
www.profileservices.ca

8 December 2015

GD&T

Dumbing-Down Your Tolerances

Originally Released on Tec-Ease.ca
6 June 2010

© 2010,2015 Profile Services

© Copyright 2010, 2015 Profile Services
All Rights Reserved

Proprietary and Confidential

The information contained in this document is the property of Profile Services. Any reproduction in part or as a whole without the written permission of Profile Services is prohibited.

PROFILE SERVICES

Notice:

Inspection protocols proposed within this document are individual approaches that offer low risk of accepting out-of-specification parts; other acceptable approaches may exist which are not discussed herein.

It remains the responsibility of the client(s) to verify any conversions, calculations, datums or controls that may be provided by Tec-Ease, Inc. and its representatives.

Unless Otherwise Stated:

- All dimensions are in millimeters [mm].
- Simplified geometries have been used to illustrate key concepts.
- Drawings and graphics may have been left incomplete by intent to illustrate key concepts.
- Geometric Dimensioning & Tolerancing in accordance with ASME Y14.5M-1994.

Revision Level	Release Date	Change Details	By
0	8 December 2015	Release to ProfileServices.ca	J.Sykes

I received an inquiry from a former colleague and trainee; here is the core of the message:

"I have been given an interesting task. It is about the practical side of GD&T, and I hope that you can offer me some advice. We have some existing components with poor measured GD&T, but still function well. The manufacturer requests to modify the drawings to reflect the measured GD&T, but this means deviating from Corporate Standards or standards typically found in Machinery's Handbook. I have to come up with a standardized approach which satisfies design and manufacturing.

I presume this is a common situation between design and manufacturing. What is typically a good approach?

I believe we should relax the GD&T to the point where functional and performance requirements are still met (which should be the objective in the first place). However, determining that "point" is difficult. It seems to me that our Corporate Standards or the Machinery's Handbook standards are much tighter than necessary for most of our business (please correct me if I'm wrong). We can determine the "point" by consulting individuals who have the experience in design and in manufacturing, or better yet follow published "looser" standards.

Is there a "looser" version of the Machinery's Handbook suitable for manufacturing in developing countries where part quality is reduced to the minimal for cost savings?"

The inquiry was a flashback moment for me. I had a long term relationship with that company, and was rather familiar with much of its product lines, practices and standards.

These issues often arise after a GD&T implementation. Typically, there is some level of training for engineering and in-house manufacturing; sometimes even for process planning and inspection. Rarely do companies provide for or require their suppliers to be trained in GD&T. This is a snowballing problem in light of the growing trend to offshore manufacturing to countries which historically do not have basic manufacturing expertise, much less knowledge of Geometric Dimensioning and Tolerancing. The economic reality is that cheap labor, absence of environmental protection, and poor but accepted treatment of workers makes outsourcing a competitive tool. I believe that gives you the context for my advice, as follows:

***First**, don't consider individual components in isolation; consider them as part of a system. The assumption that you can open the tolerances on one component in isolation is fundamentally unsound and potentially hugely expensive. The mating component(s) may be produced anywhere within specification, but today are commonly within a ±6-sigma normal distribution centered on their dimensional goal, with a reasonable process capability. If you open tolerances on an individual component without checking the tolerance stack, you risk making the cheap part cheaper while scrapping or reworking expensive parts which approach tolerance limits. This happened repeatedly as manufacturing directed engineering to open tolerances; rather poor and costly decisions.*

***Second**, ONLY make a decision based on inspection methods and results you have verified and which comply ASME B89, Y14.5.1, Y14.5.*

***Third**, verify that the GD&T on the drawings provide correct & complete component specifications.*

***Fourth**, NEVER dumb-down a design or process to accommodate an incapable supplier. What happens when you move to an even cheaper (and less capable) supplier? Will you revisit the specs again? This is a huge step down a very slippery slope, with a very bad landing at the bottom. Either find a supplier who is already capable, or develop the supplier's skillsets and capabilities.*

***Fifth**, Part 1:*
General tolerancing standards do not address Form, Orientation or Position tolerances, only size (Rule #1 doesn't apply in ISO GD&T). There are a variety of standards available for use. In training, I often add these comments about externally-referenced general tolerance specifications;

- ISO 2768 was released back around 1989. Assuming a research, compilation, filtering and committee processing period of 20 years, the results were based on machines used in average shops in 1970, with data collected at a ±3 sigma level.
- Machines in use in the '70s were likely based on 1930s-1940s technologies.... no NC controls, linear encoders, glass scales or automated tool wear compensation.
- That means that today's shops are expected to be no more capable in average production than they were 70 years ago.

Fifth*, Part 2:*
General tolerancing tables provide a different tolerance for each size range; larger dimensions mean larger tolerances. With currently available manufacturing technology such as glass scales, automated tool wear compensation and in-process measurement systems, the same tolerances are attainable regardless of the workpiece size.

Fifth*, Part 3:*
General tolerancing tables conflict with a general surface profile applied to the drawing; which takes precedence? One of the business units used some very complex rules for determining what tolerance the print user was to maintain based on assorted scenarios. It is better to tolerance the size dimension directly, however additional controls for non-size characteristics would also have to be added (form, location, orientation).

Sixth*, to make tolerance change decisions, you must have intimate knowledge of the individual parts, the system, and the functionality of both. "Hidden" functionalities or performance characteristics have a habit of popping up after a "simple" decision has been made and implemented. We walked thru some functional reviews of our components and systems, clearing our misconceptions of how our parts actually worked based on fundamental engineering principles. Not a trivial exercise, particularly when companies trim their senior knowledge base.*

I hope that my colleague takes my guidance to heart and takes a broad view of the situation before moving forward. He has a difficult task ahead of him.

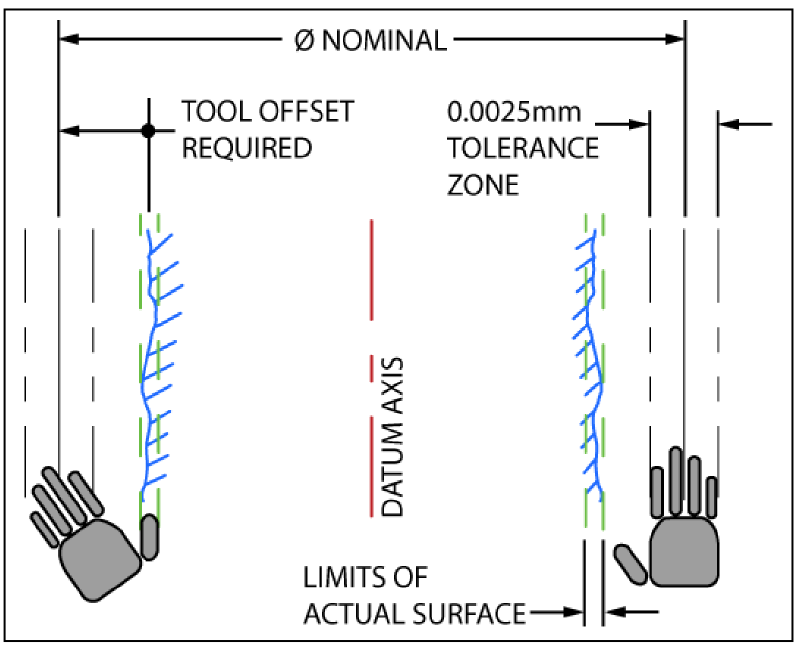
If you will indulge me for a few more minutes, I would like to pass along this relevant anecdote;

At one time, I was responsible for a GD&T implementation involving two business units on three global sites. We selected Y14.5M-1994 as our GD&T standard, then lots of training and coaching for me and my senior team members so that we could start to understand the language of GD&T as we simultaneously started applying GD&T to our drawings. Initially, we worked solely on tolerance conversions with the eventual goal to do tolerance stack-ups and manufacturing capability studies. On the original

© 2010,2015

drawings, we only had plus-minus tolerances, plus a few generic in-house symbols which really translated to "make this as perfect as you can", so we had to guesstimate what values to use where orientation and form were important.
There came the inevitable day when our European facility said that they couldn't hold required tolerances on a particularly important feature on a workpiece. The specification was for a surface profile tolerance of 0.0025mm on a cylindrical feature, centered on the datum axis.
My European colleagues had the GD&T inspection done by a prestigious inspection firm in Germany. They wanted a change in the specification to 0.01mm surface profile. My shop was fine with the existing specification so opening the spec wouldn't affect them. For engineering it was a significant issue; it would lead to a redesign of mating components to ensure correct alignment and orientation in the assembly. We had already tightened and loosened the tolerance over time as new machining centers were used, so there had to be proof of inability before we could make such changes.

So, we reviewed the results, including overlaid traces of the actual geometries and the tolerance zones (see below for the general idea) projected on the wall. I'm not an inspector or data analyst, but I have spent some time working with inspectors and metrology system suppliers. The results clearly showed that we were not within specification, but something about the graphics bothered me, and ¾ hour later I realized what it was. Using my hands as sets of calipers, I showed that the blue squiggly lines representing the



actual surfaces varied by about a thumb-width (green lines), and that they appeared mirrored about the datum axis. Lines representing the

nominal diameter and 0.0025mm surface profile tolerance zones were also shown. On the projection, the profile tolerance zone was roughly four times my thumb width.
It confirmed that while we were producing the part undersized, we were actually holding the cylindrical surface variance to about ¼ of the tolerance. In other words, the process appeared to be capable of holding the tolerance but we had to increase the size.
We had the operator run a compensation on the program, and there ended that debate. If we had blindly done as requested and validated by a supplier, we would have unnecessarily incurred significant redesign costs and may have even corrupted the entire design.

Geometric Dimensioning and Tolerancing (GD&T) is a tool that can help to improve quality and competitiveness when fully and properly implemented across all suppliers of engineering, manufacturing and inspection. Take the right steps to move your company forward.

Author Biography:
Jim Sykes P.Eng., GDTP-S, had twelve years of engineering design experience in the mold and packaging industry before attaining his Geometric Dimensioning and Tolerancing Professional – Senior Level certification and starting Profile Services, an Engineering Support Services provider. Two years later, Jim started teaching and coaching for Tec-Ease Inc. Jim has helped companies in defense, aerospace, automation, medical devices and other industries throughout North America, Eastern and Western Europe, and into Africa. Jim has worked with standards-development organizations for most of the last twenty years, and holds several patents.

Contact Details:
Jim Sykes, P.Eng, GDTP-S jdsykes@profileservices.ca

001-204-619-3636 (Canada)