24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-1729
GD&T TECHNIQUES APPLICATION AND ANALYSIS USING MCOSMOS

Julyana Maria de Medeiros Quirino
João Bosco de Aquino Silva
Universidade Federal da Paraíba
julyanaquirino@yahoo.com.br
jbosco@ct.ufpb.br

José Carlos de Lima Junior
Universidade Federal da Paraíba
limajrcarlos@hotmail.com

Manoel Fideles
Universidade Federal da Paraíba
fidelesjp@hotmail.com

Abstract. *The geometric dimensioning and tolerancing system is an old tool that only nowadays has gained space in the Brazilian metrology scenario. With comprehensive tolerance zone concepts and symbology that facilitates the understanding of technical drawings, it is widely used in the automotive industry, and has significant importance in the aeronautical industry. Its importance has been recognized in both industry and academia, due to the development and improvement of computational tools, making GD&T a viable alternative to the Cartesian system. This article aims to present a comparison between the GD&T and, the traditionally used, Cartesian method highlighting key concepts for the understanding of the geometric method and its importance. A bibliographic survey on GD&T applied to coordinate measuring machines initiated the development of this study, followed by an application of the GD&T techniques in a coordinate measuring machine with the help of its computational system, the MCOSMOS. The experimental results demonstrated the importance and efficiency of GD&T in control and in dimensional and geometric analysis of manufactured parts.*

Keywords: *GD&T, tolerance, coordinate measuring machine.*

1. INTRODUCTION

GD&T, created in the 1940s, is a language that unequivocally communicates the specifications of engineering projects for quality assurance. As its name implies, it conveys the ideal geometry and tolerances for a part. (P.J. Drake Jr, 1999). In contrast to traditional sizing and tolerancing methods, GD&T does not only question the quality of finished parts manufacturing, but also the quality of the engineering drawings that were used to produce the part in question. (QUIRINO, 2017)

Differences in the two systems symbologies are also important. GD&T has a more complex language than the Cartesian system, encompassing not only dimensional tolerance, with variations in dimensions size, but also tolerance of geometric characteristics.

The successful implementation of the geometric dimensioning and tolerancing system in the industry heavily depends on administrative support. Business management must be aware of the financial benefits of GD&T and must commit to ensuring that implementation barriers are overcome and benefits are achieved. There are two main barriers to effective implementation of GD&T. Its highly complex symbolic language, which requires deep understanding to be coded and interpreted in a functional way, and the fact that if applied incorrectly can increase costs and cause rejection of good parts. (Tandler, 2011).

Among the standards that regulate the usage of GD&T are ASME Y14.5 (2009) and the set of specific ISO standards adopted by the great majority of companies that use GD&T techniques. The ASME Y14.5 standard is the mainly used standard not only in the United States but also in several other countries, as it is stable and the design intent is a highlighted aspect. In addition, this standard has a mathematical definition and can be found in several languages. (Strafacci Neto, 2010).

According to de Souza and Wandeck (2011), the most used technology for the geometric control of products in the industry is the measurement by coordinates. Since the interpretation of drawings is the first step of geometric control, a correct understanding of GD&T techniques and an establishment of effective measurement strategies are necessary to avoid measurement errors.

The application of GD&T in coordinate measuring machines allows the analysis of geometric tolerances in an efficient and automatic way, provided that correct measurement strategies are defined, since the current MMCs have truly mechatronic systems with capacity to automate complex operations, such as tolerance analysis. (QUIRINO, 2017)

2. GEOMETRIC TOLERANCES

According to ASME Y14.5 (2009), circularity is a condition in which a surface, that is not a sphere, has all its points intersected by any plane perpendicular to an axis or curve, and these points must be equidistant from this axis or curve. In the case of a sphere, all points on its surface intersected by any plane passing through a common center are equidistant from its center.

The circularity tolerance specifies the tolerance zone limited by two concentric circles within which each circular element of the surface must be, and is applied independently in any plane that intersects the surface. (ASME Y14.5, 2009).

There are three relations of orientation: angularity, parallelism and perpendicularity. Angularity is the condition of a surface, center plane of an element or axis of an element at any specified angle of a reference plane or reference line. Parallelism is the condition of a central surface or plane of an element, equidistant from all points of a reference plane, or of an axis of an element, equidistant along its length to one or more reference planes or line of reference, and perpendicularity is the condition of a surface, the center plane of an element, or axis of an element at a right angle to a reference plane or reference line. (ASME Y14.5, 2009).

Also according to ASME Y14.5 (2009), the position is the location of one or more dimensional elements relative to one another or to one or more reference elements. A position tolerance defines the zone in which each center, line or central plane of a dimensional element is allowed to vary from a theoretically exact true position. When specified an MMC(maximum material condition) or LMC(lower material condition) concept, the position tolerance defines a boundary, defined as the virtual condition, located in the theoretically exact true position, which cannot be violated by the surface or surfaces of the dimensional element in question.

3. EXPERIMENTAL PROCEDURE

For the accomplishment of this paper, measurements were taken in two aluminum parts with identical nominal dimensions, using a coordinate measuring machine available in the Precision Engineering Laboratory at Universidade Federal da Paraíba, Brasil.

MCOSMOS (Mitutoyo Controlled Open Systems for Modular Operation Support) is a package of Mitutoyo metrology programs designed to operate integrated with coordinate measuring machines. In order to carry out GD&T measurements and verifications, the GEO PAK module of the MCOSMOS program package was used, which is a module used for the measurement and analysis of geometric elements.

Comparisons of position tolerance, circularity, parallelism, angularity and perpendicularity were performed in both parts using the GEO PAK module of the MCOSMOS software package. Figure 1 shows the tolerated elements and presents the nomenclature used in this paper.

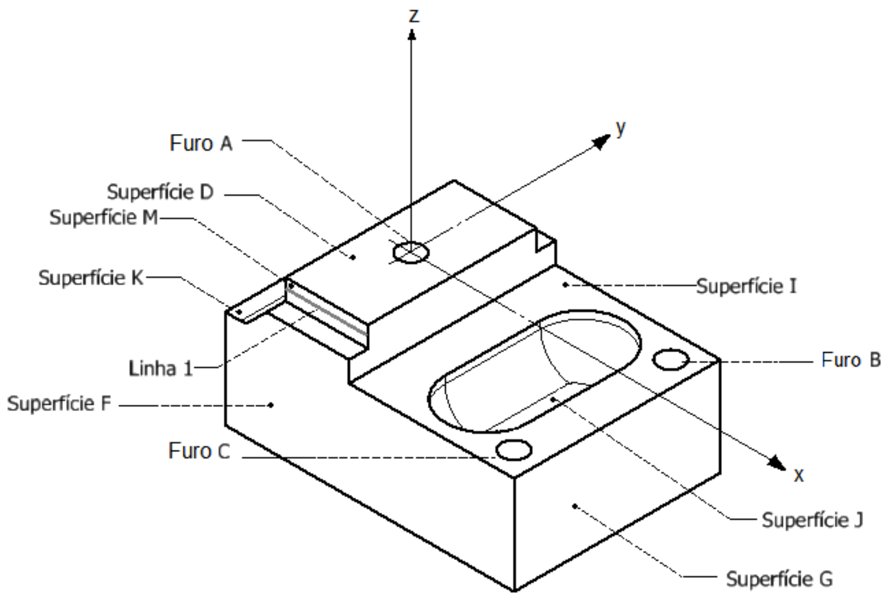

Figure 1. Perspective view with the tolerated elements

4. RESULTS AND DISCUSSION

From the data of the nominal coordinates and measurements of the holes, B and C in the part 1, obtained in the measurements, were found their position errors. In part 2, it was used the same measurement and tolerance procedure used in part 1, and the results are shown in Table 1.

Table 1. Position tolerance and errors

Hole	Tolerance (mm)	Error Part 1 (mm)	Error Part 2 (mm)
B	0,0800	0,0322	0,0641
C	0,0800	0,0394	0,0490

The position deviations found are compatible with the parts manufacturing process, performed in a CNC (Computer Numerically Controlled) machining center. This type of manufacturing presents optimum accuracy and repeatability, resulting in parts with consistent elements that meet design specifications.

The circularity of the holes A, B and C was also analyzed, and Table 2 shows the determined tolerance and the circularity errors of each hole of parts 1 and 2. All holes of the two pieces presented significantly smaller circularity errors than the tolerance determined in the design. It is important to note that there might be a refinement of these circularity error values when considering more analysis points. In this study, for each hole, four points in the same depth were considered.

Table 2. Circularity tolerance and errors

Hole	Tolerance (mm)	Error Part 1 (mm)	Error Part 2 (mm)
A	0,03	0,0009	0,0003
B	0,03	0,0009	0,0019
C	0,03	0,0003	0,0008

After the geometric characterization of the holes of the two parts, the perpendicularity of four of its surfaces with the surface D as a reference, shown in Figure 1, was calculated with the maintenance of the perpendicularity until the reference length, equivalent to the entire extension of the reference surface D. The results are presented in Table 3.

Table 3. Perpendicularity tolerance and errors

Surface	Tolerance (degrees)	Error Part 1 (degrees)	Error Part 2 (degrees)
E	0,0700	0,0276	0,0245
F	0,0700	0,0395	0,0440
G	0,0700	0,2413	0,0330
H	0,0700	0,0216	0,0102

In part 1 the values of the angles of surfaces E, F and H with respect to surface D are within the tolerance determined in the design, but the angle between surfaces D and G exceeded by 0.1713 ° the design tolerance value.

The determined tolerances and parallelism errors for surfaces I, J and K, also using surface D as reference, of parts 1 and 2 are shown in Table 4. The high deviation value between surfaces D and K, for both parts 1 and 2, can be justified by the rounding between surfaces K and L.

Table 4. Parallelism tolerance and errors

Surface	Tolerance (degrees)	Error Part 1 (degrees)	Error Part 2 (degrees)
I	0,0700	0,3680	0,0206
J	0,0700	0,0378	0,0845
K	0,0700	1,0474	1,4070

In part 1, the only angle that meets the design specifications is the angle between the surfaces J, located at the base of the elliptical feature of part 1, and D. As there is a rounding between the surfaces K and L the measurements on the surface K may have been because the extension of the surface is small and there is a high probability that the sensor tip touches a region of the rounding, increasing the angle relative to the surface D.

In part 2, the only angle analyzed that meets design specifications is the angle between surfaces I and D. The high deviation value between surfaces D and K, as in part 1, can be justified by the rounding between surfaces K and L.

The last geometric characteristic analyzed was the angularity between the surface L and the reference surface K considering the maintenance of the angle to the reference length equivalent to the entire extension of the reference surface K. The results of this analysis, presented in Table 5, demonstrate that these were the larger errors found in both parts.

Table 5. Angularity tolerance and errors

Surface	Tolerance (degrees)	Error Part 1 (degrees)	Error Part 2 (degrees)
I	0,0700	1,9829	2,9191

The angular errors found can be justified by the manufacturing process of the parts which, although it was executed in a numerically controlled by computer machining center, for the surface L was done by staggering due to a limitation of the angular range of the available tools.

5. CONCLUSIONS

Experimental procedures were performed with the purpose of evaluating the GD&T tools available in embedded software of a coordinate measuring machine, in particular the tools that perform analysis of position, circularity, perpendicularity, parallelism and angularity. The GEOPAK application of the MCOSMOS software package was effective in calculating and presenting geometric tolerances. An additional advantage of using a computer numerically controlled (CNC) machine was the possibility of creating an automatic inspection program, which allowed a significant reduction of the parts inspection time.

6. REFERENCES

ASME Y14.5, 2009. "Dimensioning and Tolerancing - Engineering Drawing and Related Documentation Practices". New York, USA.

de Souza, A. R. and Wandeck, M., 2011. "Desafios Operacionais e Metrológicos da Medição por Coordenadas no Ambiente de Manufatura Industrial". In *Proccedings of the 2º International Congress of Mechanic Metrology*. Natal, Brazil.

Drake Jr., P.J., 1999. *Dimensioning and Tolerancing Handbook*. Mcgraw Hill, New York.

Quirino, J.M.M., 2017. *Estudo das técnicas de GD&T e sua aplicação em máquinas de medição por coordenadas*. Master's thesis, Universidade Federal da Paraíba, Brazil.

Strafacci Neto, G., 2010. "Por que Geometric Dimensioning and Tolerancing (GD&T)?". *Revista Metrologia e Instrumentação*, p. 88 – 90.

Tandler, B., 2011. "GD&T:Realize its promisses: GD&T is the only means we have to say precisely what we want see that others can do precisely what we say". *Quality*, p. 34 – 36.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.