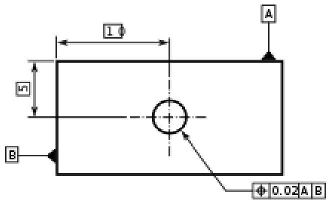
WIKIPEDIA

Geometric dimensioning and tolerancing



Example of geometric dimensioning and tolerancing

Geometric Dimensioning and Tolerancing (GD&T) is a system for defining and communicating engineering tolerances. It uses a symbolic language on engineering drawings and computer-generated three-dimensional solid models that explicitly describes nominal geometry and its allowable variation. It tells the manufacturing staff and machines what degree of accuracy and precision is needed on each controlled feature of the part. GD&T is used to define the nominal (theoretically perfect) geometry of parts and assemblies, to define the allowable variation in form and possible size of individual features, and to define the allowable variation between features.

- Dimensioning specifications define the nominal, as-modeled or as-intended geometry. One example is a basic dimension.
- Tolerancing specifications define the allowable variation for the form and possibly the size of individual features, and the allowable variation in orientation and location between features. Two examples are linear dimensions and feature control frames using a datum reference (both shown above).

There are several standards available worldwide that describe the symbols and define the rules used in GD&T. One such standard is American Society of Mechanical Engineers (ASME) Y14.5-2009. This article is based on that standard, but other standards, such as those from the International Organization for Standardization (ISO), may vary slightly. The Y14.5 standard has the advantage of providing a fairly complete set of standards for GD&T in one document. The ISO standards, in comparison, typically only address a single topic at a time. There are separate standards that provide the details for each of the major symbols and topics below (e.g. position, flatness, profile, etc.).

Contents

History

The origin of GD&T has been credited to a man named Stanley Parker, who developed the concept of "true position" in 1938.[1] While very little is known about the life of Stanley Parker, it is recorded that he worked at the Royal Torpedo Factory in Alexandria, Scotland. Parker's work was used to increase production of naval weapons by new contractors.[2]

Dimensioning and tolerancing philosophy

According to the ASME Y14.5-2009[3] standard, the purpose of geometric dimensioning and tolerancing (GD&T) is to describe the engineering intent of parts and assemblies. The datum reference frame can describe how the part fits or functions. GD&T can more accurately define the dimensional requirements for a part, allowing over 50% more tolerance zone than coordinate (or linear) dimensioning in some cases. Proper application of GD&T will ensure that the part defined on the drawing has the desired form, fit (within limits) and function with the largest possible tolerances. GD&T can add quality and reduce cost at the same time through producibility

There are some fundamental rules that need to be applied (these can be found on page 7 of the 2009 edition of the standard):

- All dimensions must have a tolerance. Every feature on every manufactured part is subject to variation, therefore, the limits of allowable variation must be specified. Plus and minus tolerances may be applied directly to dimensions or applied from a general tolerance block or general note. For basic dimensions, geometric tolerances are indirectly applied in a related Feature Control Frame. The only exceptions are for dimensions marked as minimum, maximum, stock c reference.
- Dimensions define the nominal geometry and allowable variation. Measurement and scaling of the drawing is not allowed except in certain cases.
- Engineering drawings define the requirements of finished (complete) parts. Every dimension and tolerance required to define the finished part shall be shown c the drawing. If additional dimensions would be helpful, but are not required, they may be marked as reference.
- Dimensions should be applied to features and arranged in such a way as to represent the function of the features. Additionally, dimensions should not be subject to more than one interpretation.
- Descriptions of manufacturing methods should be avoided. The geometry should be described without explicitly defining the method of manufacture.
- If certain sizes are required during manufacturing but are not required in the final geometry (due to shrinkage or other causes) they should be marked as non-mandatory.
- All dimensioning and tolerancing should be arranged for maximum readability and should be applied to visible lines in true profiles.
- When geometry is normally controlled by gage sizes or by code (e.g. stock materials), the dimension(s) shall be included with the gage or code number in parentheses following or below the dimension.
- Angles of 90° are assumed when lines (including center lines) are shown at right angles, but no angular dimension is explicitly shown. (This also applies to oth orthogonal angles of 0°, 180°, 270°, etc.)

- Dimensions and tolerances are valid at 20 °C / 101.3 kPa unless stated otherwise.
- Unless explicitly stated, all dimensions and tolerances are only valid when the item is in a free state.
- Dimensions and tolerances apply to the length, width, and depth of a feature including form variation.
- Dimensions and tolerances only apply at the level of the drawing where they are specified. It is not mandatory that they apply at other drawing levels, unless th specifications are repeated on the higher level drawing(s).

(Note: The rules above are not the exact rules stated in the ASME Y14.5-2009 standard.)

Symbols

Tolerances: Type of tolerances used with symbols in feature control frames can be 1) equal bilateral 2) unequal bilateral 3) unilateral 4) no particular distribution (a "floating" zone)

Tolerances for the profile symbols are equal bilateral unless otherwise specified, and for the position symbol tolerances are always equal bilateral. For example, the position of a hole has a tolerance of .020 inches. This means the hole can move +/- .010 inches, which is an equal bilateral tolerance. It does not mean the hole can move +.015/-.005 inches, which is an unequal bilateral tolerance. Unequal bilateral and unilateral tolerances for profile are specified by adding further information to clearly show this is what is required.

Geometric tolerancing reference chart
Per ASME Y14.5 M-1982

Type of control	Geometric characteristics	Symbol	Character (Unicode)	Can be applied to a surface?	Can be applied to a feature of size?	Can affect virtual condition?	Datum reference used?	Can use Ⓜ modifier?	Can use Ⓢ modifier?	Can be affected by a bonus tolerance?	Can be affected by a shift tolerance?
Form	**Straightness**	⏤	⏤ U+23E4	Yes	Yes	Yes (note 1)	No	Yes (note 1)	No (note 5)	Yes (note 4)	No
Form	**Flatness**	⏥	⏥ U+23E5	Yes	No	No	No	No	No (note 5)	No	No
Form	**Circularity**	○	○ U+25CB	Yes	No	No	No	No	No (note 5)	No	No
Form	**Cylindricity**	⌭	⌭ U+232D	Yes	No	No	No	No	No (note 5)	No	No
Profile	**Profile of a line**	⌒	⌒ U+2312	Yes	No	No	Yes (note 2)	No	No (note 5)	No	Yes (note 3)
Profile	**Profile of a surface**	⌓	⌓ U+2313	Yes	No	No	Yes (note 2)	No	No (note 5)	No	Yes (note 3)
Orientation	**Perpendicularity**	⊥	⟂ U+27C2	Yes	Yes	Yes (note 1)	Yes	Yes (note 1)	No (note 5)	Yes (note 4)	Yes (note 3)
Orientation	**Angularity**	∠	∠ U+2220	Yes	Yes	Yes (note 1)	Yes	Yes (note 1)	No (note 5)	Yes (note 4)	Yes (note 3)
Orientation	**Parallelism**	∥	∥ U+2225	Yes	Yes	Yes (note 1)	Yes	Yes (note 1)	No (note 5)	Yes (note 4)	Yes (note 3)
Location	**Symmetry**	⌯	⌯ U+232F	No (note 6)	Yes (note 6)	Yes (note 6)	Yes (note 6)	No (note 6)	No (note 6)	No (note 6)	No (note 6)
Location	**Position**	⌖	⌖ U+2316	No	Yes	Yes	Yes	Yes	Yes	Yes (note 4)	Yes (note 3)
Location	**Concentricity**	◎	◎ U+25CE	No	Yes	Yes	Yes	No	No (note 5)	No	No
Run-out	**Circular run-out**	↗	↗ U+2197	Yes	Yes	Yes (note 1)	Yes	No	No (note 5)	No	No
Run-out	**Total run-out**	⌰	⌰ U+2330	Yes	Yes	Yes (note 1)	Yes	No	No (note 5)	No	No

Notes:

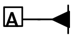
1. When applied to a feature-of-size.
2. Can also be used as a form control without a datum reference.
3. When a datum feature-of-size is referenced with the MMC modifier
4. When an MMC modifier is used.
5. Automatic per rule #3.
6. The symmetry symbol's characteristics were not included in the version of the chart that this chart is derived from. The symmetry symbol was dropped from the Y14.5M standard around 1982 and re-added around 1994.

Symbols used in a "feature control frame" to specify a feature's description, tolerance, modifier and datum references

Symbol	Modifier	Notes
Ⓕ	Free state	Applies only when part is otherwise restrained
Ⓛ	Least material condition (LMC)	Useful to maintain minimum wall thickness
Ⓜ	Maximum material condition (MMC)	Provides bonus tolerance only for a feature of size
Ⓟ	Projected tolerance zone	Useful on threaded holes for long studs
Ⓢ	Regardless of feature size (RFS)	Not part of the 1994 version. See para. A5, bullet 3. Also para. D3. Also, Figure 3-8.
Ⓣ	Tangent plane	Useful for interfaces where form is not required
Ⓤ	Unequal Bilateral	Appears in the 2009 version of the standard, and refers to unequal profile distribution.

Datums and datum references

A **datum** is a virtual ideal plane, line, point, or axis. A **datum feature** is a physical feature of a part identified by a **datum feature symbol** and corresponding **datum feature triangle**, e.g.,



These are then referred to by one or more 'datum references' which indicate measurements that should be made with respect to the corresponding datum feature .

Data exchange

Exchange of geometric dimensioning and tolerancing (GD&T) information between CAD systems is available on different levels of fidelity for different purposes:

- In the early days of CAD, exchange-only lines, texts and symbols were written into the exchange file. A receiving system could display them on the screen or print them out, but only a human could interpret them.
- *GD&T presentation*: On a next higher level the presentation information is enhanced by grouping them together into *callouts* for a particular purpose, e.g. a *datum feature callout* and a *datum reference frame*. And there is also the information which of the curves in the file are leader, projection or dimension curves and which are used to form the shape of a product.
- *GD&T representation*: Unlike GD&T presentation, the GD&T representation does not deal with how the information is presented to the user but only deals with which element of a shape of a product has which GD&T characteristic. A system supporting GD&T representation may display GD&T information in some tree and other dialogs and allow the user to directly select and highlight the corresponding feature on the shape of the product, 2D and 3D.
- Ideally both GD&T presentation and representation are available in the exchange file and are associated with each other. Then a receiving system can allow a user to select a GD&T callout and get the corresponding feature highlighted on the shape of the product.
- An enhancement of GD&T representation is defining a formal language for GD&T (similar to a programming language) which also has built-in rules and restrictions for the proper GD&T usage. This is still a research area (see below reference to McCaleb and ISO 10303-1666).
- *GD&T validation*: Based on GD&T representation data (but not on GD&T presentation) and the shape of a product in some useful format (e.g. a boundary representation), it is possible to validate the completeness and consistency of the GD&T information. The software tool FBTI from the Kansas City Plant is probably the first one in this area.
- GD&T representation information can also be used for the software assisted manufacturing planning and cost calculation of parts. See ISO 10303-224 and 238 below.

Documents and standards

ISO TC 10 *Technical product documentation*

- ISO 128 *Technical drawings – Indication of dimensions and tolerances*
- ISO 7083 *Symbols for geometrical tolerancing – Proportions and dimensions*
- ISO 13715 *Technical drawings – Edges of undefined shape – Vocabulary and indications*
- ISO 15786 *Simplified representation and dimensioning of holes*
- ISO 16792:2015 *Technical product documentation—Digital product definition data practices* (Note: ISO 16792:2006 was derived from ASME Y14.41-2003 by permission of ASME)

ISO/TC 213 *Dimensional and geometrical product specifications and verification*

In ISO/TR 14638 *GPS – Masterplan* the distinction between fundamental, global, general and complementary GPS standards is made.

- Fundamental GPS standards
  - ISO 8015 *Concepts, principles and rules*
- Global GPS standards
  - ISO 14660-1 *Geometrical features*
  - ISO/TS 17, orientation and location
  - ISO 1101 *Geometrical tolerancing – Tolerances of form, orientation, location and run-out*

- Amendment 1 *Representation of specifications in the form of a 3D model*
- ISO 1119 *Series of conical tapers and taper angles*
- ISO 2692 *Geometrical tolerancing – Maximum material requirement (MMR), least material requirement (LMR) and reciprocity requirement (RPR)*
- ISO 3040 *Dimensioning and tolerancing – Cones*
- ISO 5458 *Geometrical tolerancing – Positional tolerancing*
- ISO 5459 *Geometrical tolerancing – Datums and datum systems*
- ISO 10578 *Tolerancing of orientation and location – Projected tolerance zone*
- ISO 10579 *Dimensioning and tolerancing – Non-rigid parts*
- ISO 14406 *Extraction*
- ISO 22432 *Features used in specification and verification*
- General GPS standards: **Areal and profile surface texture**
  - ISO 1302 *Indication of surface texture in technical product documentation*
  - ISO 3274 *Surface texture: Profile method – Nominal characteristics of contact (stylus) instruments*
  - ISO 4287 *Surface texture: Profile method – Terms, definitions and surface texture parameters*
  - ISO 4288 *Surface texture: Profile method – Rules and procedures for the assessment of surface texture*
  - ISO 8785 *Surface imperfections – Terms, definitions and parameters*
  - Form of a surface independent of a datum or datum system. Each of them has a part 1 for the *Vocabulary and parameters* and a part 2 for the *Specification operators*:
    - ISO 12180 *Cylindricity*
    - ISO 12181 *Roundness*
    - ISO 12780 *Straightness*
    - ISO 12781 *Flatness*
  - ISO 25178 *Surface texture: Areal*
- General GPS standards: **Extraction and filtration techniques**
  - ISO/TS 1661 *Filtration*
  - ISO 11562 *Surface texture: Profile method – Metrological characteristics of phase correct filters*
  - ISO 12085 *Surface texture: Profile method – Motif parameters*
  - ISO 13565 *Profile method; Surfaces having stratified functional properties*

ASME standards

- ASME Y14.41-2009 Digital Product Definition Data Practices
- ASME Y14.5 – Dimensioning and Tolerancing
- ASME Y14.5M-1994 Dimensioning and Tolerancing
- ASME Y14.5.1M-1994 Mathematical Definition of Dimensioning and Tolerancing Principles

ASME is also working on a Spanish translation for the ASME Y14.5 – Dimensioning and Tolerancing Standard.

GD&T standards for data exchange and integration

- ISO 10303 *Industrial automation systems and integration — Product data representation and exchange*
  - ISO 10303-47 *Integrated generic resource: Shape variation tolerances*
  - ISO/TS 10303-1130 *Application module: Derived shape element*
  - ISO/TS 10303-1050 *Application module: Dimension tolerance*
  - ISO/TS 10303-1051 *Application module: Geometric tolerance*
  - ISO/TS 10303-1052 *Application module: Default tolerance*
  - ISO/TS 10303-1666 *Application module: Extended geometric tolerance*
  - ISO 10303-203 *Application protocol: Configuration controlled 3D design of mechanical parts and assemblies*
  - ISO 10303-210 *Application protocol: Electronic assembly, interconnection, and packaging design*
  - ISO 10303-214 *Application protocol: Core data for automotive mechanical design processes*
  - ISO 10303-224 *Application protocol: Mechanical product definition for process planning using machining features*
  - ISO 10303-238 *Application protocol: Application interpreted model for computerized numerical controllers* (STEP-NC)

See also

- Specification of surface finish
- Engineering fit
- Engineering tolerance

References

1. "GD&T | Geometric Dimensioning and Tolerancing | Quality-One" (http://quality-one.com/gdt/). *quality-one.com*. Retrieved 2017-07-28.
2. "Bibliography for Dimensioning and Tolerancing" (https://www.circuitousroot.com/artifice/drafting/drawing-studies/dt/bibliography-for-dt/index.html#parker-1956). *www.circuitousroot.com*. Retrieved 2017-07-28.
3. *Dimensioning and Tolerancing, ASME y14.5-2009*. NY: American Society of Mechanical Engineers. 2009. ISBN 0-7918-3192-2.

Further reading

- McCale, Michael R. (1999). "A Conceptual Data Model of Datum Systems" (PDF). *Journal of Research of the National Institute of Standards and Technology*. **104** (4): 349–400. doi:10.6028/jres.104.024

- Henzold, Georg (2006). *Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection* (2nd ed.). Oxford, UK: Elsevier. ISBN 978-0750667388.
- Srinivasan, Vijay (2008). "Standardizing the specification, verification, and exchange of product geometry: Research, status and trends". *Computer-Aided Design*. **40** (7): 738–49. doi:10.1016/j.cad.2007.06.006
- Drake, Jr., Paul J. (1999). *Dimensioning and Tolerancing Handbook*. New York: McGraw-Hill. ISBN 978-0070181311.
- Neumann, Scott; Neumann, Al (2009). *GeoTol Pro: A Practical Guide to Geometric Tolerancing per ASME Y14.5-2009*. Dearborn, MI: Society of Manufacturing Engineers. ISBN 978-0-87263-865-5.
- Bramble, Kelly L. (2009). *Geometric Boundaries II, Practical Guide to Interpretation and Application ASME Y14.5-2009*. Engineers Edge.
- Wilson, Bruce A. (2005). *Design Dimensioning and Tolerancing*. US: Goodheart-Wilcox. p. 275. ISBN 978-1-59070-328-1.

External links

- General tolerances for linear and angular dimensions according to ISO 2768
- Interactive map of GD&T
- What is GD&T
- The importance of GD&T
- GD&T Glossary of Terms and Definitions
- GDT: Introduction
- ASME Certification
- Changes and Additions to ASME Y14.5M
- NIST MBE PMI Validation and Conformance Testing Project Tests implementations of GD&T in CAD software