Tolerances

02/21/11

Tolerances

Tolerance is defined as the total amount a part is allowed to vary.

Example:

– A specification of +/- .004 results in a tolerance of .008.
– A dimension of 1.004 has a tolerance of .004 1.000

Selection of Tolerances

Tolerances should be as generous as possible.
The tighter a tolerance the more expensive the part will be to manufacture.
Charts are used to determine the tolerances that can be achieved with various machining processes.

International Tolerance Grades

A set of tolerances that vary according to basic size and provide a uniform level of accuracy within the grade.

Example:

– A Dimension of 50H8 is for a basic size 50mm, with H noting that the tolerance is on the hole.
– The 8 refers to the IT tolerance grade.

International Tolerance Grade
International Tolerance Grades

Key Terms:
- International Tolerance Grade
- Fundamental Deviation
- Basic Size

Geometric Characteristic Symbols

- Form
- Profile
- Orientation
- Location
- Runout

- Designated by a series of symbols
The Geometric Characteristic symbols, also called feature control symbols. They are placed in a feature control frame, (box), along with a tolerance value, they may also be accompanied by a Datum reference, and modifiers. Modifiers are typically M for Maximum Material Condition or L for Least material condition.

- **MMC**: Maximum Material Condition: occurs when the shaft is largest and the hole is smallest.
- **LMC**: Least Material Condition: occurs when the shaft is smallest and the hole is largest.
Geometric Tolerancing

02/24/11

- Geometric tolerances state the maximum allowable variations of form or position from the perfect geometry implied on the drawing.

- Saves money by providing for maximum producibility of the part through maximum production tolerances.

- It also provides “Bonus” tolerances in many cases.

- Ensures that design, dimensional and tolerance requirements relate to actual function.

- Ensures interchangeability of mating parts at assembly.

- Provides uniformity and convenience in drawing and interpretation.

- Reduces guesswork and controversy.

When is Geometric Tolerancing Used?

- When part features are critical to function and interchangeability.

- When functional gagging techniques are required.

- When a datum references are required to help establish consistency between manufacturing and gagging operations.
Feature Control Frames

Limits of Size

• Unless otherwise stated the limits of size of a feature prescribe the extent within which variations of geometric forms as well as size are allowed.

“Bonus” Tolerances

The “Bonus” tolerance comes into play when the shaft is smaller than the MMC designated in the second example.

Rule #1!

• Variations of form:
  - The form of an individual feature is controlled by its limits of size to the extent that follows:
  - The surface or surfaces of a feature shall not extend beyond a boundary envelope of perfect form at MMC.
  - No variation of form is permitted if the feature is produced at MMC form.
  - The actual size of an individual feature at any cross section will be within the specified tolerance of size.

Regardless of Feature Size

• When MMC or LMC are not specified, the tolerance may be assumed to be Regardless of Feature Size, or RFS.
  - In this instance a “Bonus” tolerance would not be an option.
Rule #1

• Rule #1 does not apply to relationships between features, (size or form features)

• Therefore limits of size do not control orientation or location relationships between individual features.

Cylindricity

The tolerance zone is composed of two parallel planes.

Flatness

The surface must be within the specified tolerance of size and must lie between two parallel planes .025 apart.

GDT

February 24

Use of Datums

• A Datum is the origin from which the location or geometric characteristics of features of a part are established.

• The datum plane will be the simulated in the inspection gauge.

Profile

• All points must fall between two boundary surfaces placed .6 mm apart.
Angularity

If two surfaces are shown parallel on a drawing,
the size dimensions of the surfaces controls the
gap width.
Poor inspection due to lack of datums and
gap width has same dimensions as size.
Not a good way to design.
Perpendicularity Applied to a Cylindrical FOS

Perpendicularity with “Bonus” Tolerance

Inspecting for Perpendicularity

Summary

Concentricity

Datum References and Inspection

• A datum is a starting point for a dimension.

• Datums are theoretically ideal locations in space such as a plane, centerline, or point. A datum may be represented either directly or indirectly by an inspection device.
Datum References and Inspection

• The surface of the object which is placed on the inspection device representing the datum is called the **datum feature**.

• Datum features are clearly marked in the drawings to indicate which are the reference surfaces to make measurements from.

• Once a datum is established, the measurements can be taken from it rather than features on the object.

• The object feature representing the datum is aligned/placed on the inspection device and the measurement is taken. The datum establishes the method of locating other features on the object relative to each other.

Datum References and Inspection

• Datums are not only used internal to a part but, more importantly, in relation to mating parts in an assembly.

• Datum features should be selected carefully based on their size, stability, accessibility, functionality, etc.

• Datums are the locators and the **datum reference frame** is the six degrees of freedom from the datum. The six degrees of freedom are the plus and minus directions along the three Cartesian coordinate axes. Another way of looking at the frame is as three orthogonal planes.

• At a minimum, a part will have a **primary datum**. This datum will be chosen based on a number of criteria:

  • **Stability**. This is the most important feature. Often dictating that the largest, flattest surface is chosen.

  • **Functional relationship**. How does the feature mate/interact with other features?

  • **Accessibility**. Can the part be mounted and measured on the inspection device via this feature?

  • **Repeatability**. Variations in the datum feature due to manufacturing should be predictable so they can be accounted for.

• Secondary and tertiary datums, if needed, should be located mutually perpendicular to each other and to the primary datum.

• The primary, secondary, and tertiary datums are identified with the letters A, B, and C, respectively.
**Gaging**

- A **gaging tolerance** establishes when a part is ‘perfect’. For example, an inspection instrument is considered perfect if it is ten times more accurate than the part being measured.
- The **virtual condition** is the combined effect of the largest allowable size (MMC) and the largest amount of geometric distortion. It is, in effect, the **worst** allowable condition of the part.

**Part Inspection**

- There are a number of ways parts are inspected:
  - **Functional gaging.** They are hand built inspection devices made to simplify the inspection of a large number of parts.
  - **Open gaging.** The inspection of parts without any dedicated fixtures. Examples would be inspection with calipers or surface plates.
  - **Feature of size.** A feature that is directly measurable.

**Applying Tolerances**

- There is a five-step process for applying principles to the design process. They are:
  - Isolate and define the functions of the features/part.
  - Prioritize the functions.
  - Identify the datum reference frame based on functional priorities.
  - Select the proper control(s).