Geometric Dimensioning and Tolerancing

Engr1170

Chap 12
GDT

• GDT
  – Is a method for defining parts based on how they function using ASME/ANSI symbols, it places tolerances on the form itself (Form tolerance)

• It is a powerful tool in reducing the cost of precision parts

• Additionally, widespread acceptance of worldwide standards (ISO 9000 series) has resulted in a greater need for GDT
GDT

• Standards are needed to specify not only that some geometry is controlled but how it is controlled

• For example, a drawing may say a feature is round but how round?

• How is that roundness inspected?
SPC

• Statistical Process Control (SPC)
  – Monitors the design process through the use of control charts

• Data is collected from samples at various points within the process,

• Variations in the process that may affect the quality of the end product or service can be detected and corrected.
SPC

• This reduces waste as well as the likelihood that problems will be passed on to the customer.

• Often the only way to tolerance features such that SPC can be used effectively is through the use of GDT.
Size Tolerance

• Why can’t size tolerance alone be sufficient?

• Is the top of the table in Figure 12.1 flat?
Size Tolerance

- The tolerance on the 30 inch height is 30 ±1 inch
- The table is therefore flat within 2 inches
- Suppose we wanted it be flat within ± 0.25 inch?
- This might make the height tolerance too restrictive!
Size Tolerance

• Could use a NOTE, for example
  1. TABLE TOP TO BE FLAT WITHIN ½ INCH TOTAL. or ..........

• A Flatness tolerance
Questions to Ask

• Why kind of part would be rejected with these tolerances?

• Will the rejected parts be unusable?

• Will we reject all the parts we cannot use?

• What are answers to the above questions for our table example?
Maximum Material Condition

• Recall: **Maximum Material Condition (MMC)** is the condition in which a part will contain the most material (i.e. weigh the most)

• Other related Terms of Importance
  • LMC
  • RFS
  • Departure from MMC
  • Perfect Form at MMC
Maximum Material Condition

- LMC  - Least Material Condition: Part has its least amount of material

- RFS  - Regardless of feature size: condition of material *not* be considered

- Departure from MMC
  - Gives the value by which parts deviate from their largest size (external) or smallest size (internal)

- Perfect Form at MMC
  - the manufactured part must never violate MMC boundaries
Flatness

- A flatness tolerance specifies a tolerance zone defined by two parallel lines within which the surface must lie.
Datum

• Datum surfaces and datum features are used as references to control other features on a part.

• Datum features can be actual features on the part, such as a point, line, plane, cylinder, or other geometric form assumed to be exact.
Datum

• Datum surfaces and datum features are used as references to control other features on a part.

• Datum features can be actual features on the part, such as a point, line, plane, cylinder, or other geometric form assumed to be exact.
Straightness

- Specifies a tolerance zone within which an axis or all points of the indicated element must lie.

- Straightness indicates that the element of a surface or an axis is a straight line.
Straightness

• Specifies a tolerance zone within which an axis or all points of the indicated element must lie.

• Straightness indicates that the element of a surface or an axis is a straight line.
Straightness

• Any element of the shaft must be straight within 0.002 units

• Imagine two separate straight lines set apart by the tolerance amount
Straightness

- An acceptably straight element fits within the tolerance zone
Circularity

- A circularity (roundness) tolerance specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie.
Cylindricity

• A cylindricity tolerance specifies a tolerance zone bounded by concentric cylinders within which the surface must lie.
Parallelism

- A *parallelism tolerance* specifies a tolerance zone defined by two parallel planes or by two lines parallel to a datum plane or axis within which the surface or axis of the feature must lie.

Parallelism for an axis

Parallelism for a plane
Perpendicularity

- A *perpendicularity tolerance* indicates that a surface, median plane or axis is at 90 degrees to a datum plane or axis.
Concentricity

• A *concentricity tolerance* indicates that a cylinder, cone, hex, square or surface of revolution shares a common axis with a datum feature. It controls the location for the axis of the indicated feature within a cylindrical tolerance zone whose axis coincides with the datum axis.
Concentricity

This on the drawing...

Extreme locational variation

0.1 diameter tolerance zone

Extreme attitude variation

Median line of this surface

Median points derived from this surface must lie within the 0.1 diameter tolerance zone

Axis of datum feature A

...means this
Other Form Tolerances to look up!

- Profile tolerance
- Positional tolerance
- Angularity
- Read chap 12.1-12.6

- Read 12.7 and 12.8 for an appreciation of GDT applications
GDT Rule 1

• Where only a tolerance of size is specified, the limits of size of an individual feature prescribed the extent to which variations in its geometric form, as well as size, are allowed (ASME Y14.5-1994)

• Ex., shaft with 0.5 inch diameter with a tolerance of ±0.005 only implies this also controls the circularity of the shaft