What is Geometric Dimensioning and Tolerancing?

- A system of dimensioning with emphasis on the function of the part and its relationship to mating parts.
- A system involving the use of symbols to represent the characteristics of the tolerances being specified.

Why use symbolic specifications?

- Symbols have uniform meaning.
- Symbols are compact, quickly drawn, and readily adaptable to computer applications.
- International acceptance.

Why use Geometric Tolerancing?

- Fully defines engineering design intent.
- Lowers manufacturing costs.
- Required for some government contracts.
- Acceptability (being used worldwide)

Intro to GD&T

- Geometric tolerances provide a refinement to size tolerances.
- Size tolerances still apply.
- Geometric tolerances control form, orientation, and function
  - examples are:
    - perpendicularity, flatness, parallelism, etc.

Geometric tolerancing: topics

- The geometric characteristics and their symbols.
- Other related symbols and terms.
- Feature control frame and the datum feature symbol

Geometric tolerancing: topics

- The distinction between:
  - form
  - orientation
  - profile
  - runout
  - and location tolerances
Geometric tolerancing: topics

- Tolerance zones (definition and shape)
- The basic rules for interpretation of geometric tolerances
- Introduction to inspection concepts

Definitions

- Feature:
  - Any portion of an object.
  - May be a point, edge, centerline, or a planar or curved surface.
  - May also be a size, as in the width of a slot or the diameter of a cylinder. This is called a “feature of size”

Definitions

- Element:
  - Any line, real or imaginary, that can be drawn upon a surface.
  - For example, all elements of a plane are straight lines drawn in any direction.
  - Elements of a cylinder are circles having the same radius as the cylinder, or straight lines parallel to the axis.

Plane elements

Cylindrical Elements

Definitions

- Tolerance zone:
  - The area taken up by the total amount of permissible variance in a dimension, a geometric form, or a position.
  - 2D zones:
    - area between two parallel lines, space between two concentric circles.
  - 3D zones:
    - rectangular solid, cylinder, space between two concentric cylinders.
Definitions

• MMC/LMC
  – Maximum material condition occurs when a size feature contains the most permissible material.
  – When MMC is used as a geometric tolerance modifier, the tolerance only applies when the controlled feature or datum is at its MMC (size).
  – Least material condition is the opposite

• Regardless of Feature Size (RFS)
  – Tolerance specified applies no matter how large or how small the manufactured size of the feature.
  – Part must still meet specified size tolerances.

Definitions

• Virtual Condition
  – Maximum material condition size of feature with the associated geometric tolerance value
  – For external feature (cylinder, etc)
    • Virtual condition is MMC plus the geometric tolerance value
  – For internal feature (hole, slot, etc.)
    • Virtual condition is MMC minus the geometric tolerance value
  – Virtual condition size is used for functional gages

Definitions

• Functional Gage
  – Precision machined component meant to simulate mating component
  – Used to inspect controlled feature

Feature Control Frame

• Contains:
  – Kind of control (geometric characteristic)
  – The geometric tolerance
  – Any modifiers (such as MMC, RFS)
  – Datum references (and datum reference modifiers), if required

Datum references

• Datum is identified by letter.
• Precedence of data is signified by placement of the reference letters, e.g. the order of the letters.
• No significance to alphabetic sequence.
Datum references

• Two letters separated by a dash indicates a common datum (axis or a centerplane) is established between datum features.
• No precedence is implied, together they form a single datum reference.

Datum Feature Symbol

• Used to identify the feature(s) on a part from which functional relationships (such as the geometric characteristics) are established.
• Each feature identified as a datum on the part requires a different letter.
• It is preferred to begin at A. The letters I, O, and Q, may not be used.

Datum Feature versus Datum

• A datum feature is an actual feature of a part which is used to establish a datum.
• Since the datum feature refers to an actual part feature, it includes all the inaccuracies and irregularities of the part feature.

Datum Feature versus Datum

• A datum is a theoretically exact point, axis or plane derived from the true geometric counterpart of a specified datum feature.
• The datum is the origin from which the geometric characteristics of a part are established.

Datum Feature versus Datum

• The definition implies that a datum is “perfect”.
• Since perfect parts cannot be produced, a datum on a physically produced part is assumed to exist in the contact between the datum feature surface and the precision manufacturing and inspection equipment such as machine tables, surface plates, gage pins, etc.

Datum Feature versus Datum

• It is from this processing equipment that measurements are taken and dimensions of features verified.
• This contact of the actual surface with the precise equipment is assumed to simulate contact with a mating part surface.
Datum types:
- Points (apex of a cone)
- Lines (line on a surface)
- Planes (flat surfaces)
- Cylinders (holes)
- Widths (slots and keyways)
- Axes (center of two features)

Datum Establishment
- Establish data on functional features.
  - Features which have a function for assembly, inspection, etc.
  - Non-functional features may be used for ease of manufacture and inspection.
- Mating parts should have the same datum specifications.
- Choose features that are accessible and of sufficient length.

3 Datum Plane Reference
- Application of three mutually orthogonal planes to the part as data (primary, secondary, tertiary).
- Established on a non-cylindrical part by:
  - 3 point contact on the primary plane
  - 2 point contact on the secondary
  - 1 point on the tertiary

Tolerances of Form
- Flatness
- Straightness
- Circularity
- Cylindricity

3 Datum Plane Reference
- Typically, the surface most influential to part orientation in assembly is designated as primary.
- Most often used with Positional Tolerances (hole patterns)

Tolerances of Form
- Provides methods by which to control part geometry where size and location dimensions along do not adequately do so.
- Form tolerances state how far a face or feature is permitted to vary from the desired geometry as indicated by the drawing.
Tolerances of Form

- Form Tolerances refer to a single feature and do not include a datum reference.
- The tolerances relate the feature to a perfect geometric counterpart of itself.

Flatness

- The condition of having all points and elements within its boundary lie in a single plane.
- Flatness tolerance specifies a tolerance zone defined by two parallel planes within which the specified surface must lie.
- Flatness tolerance should be equal to or less than, one-half of the size tolerance affecting feature.

Straightness

- Condition whereby an element of a surface or an axis is a straight line.
- Tolerance specifies a tolerance zone within which the element or axis must lie.
- Applied in the drawing view where the element or axis appears as a line.

Circularity

- For a cylinder or cone:
  - all points of the surface intersected by any plane perpendicular to an axis are equidistant from that axis.
- For a sphere:
  - All points of the surface intersected by any plane passing through a common center are equidistant from that center.

Cylindricity

- Specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie.
- May be thought of as a circularity tolerance extended to control the entire surface (both circular and longitudinal elements)
- A difficult tolerance to inspect, condition better controlled by runout tolerance
Tolerances of Orientation

- Types:
  - perpendicularity
  - parallelism
  - angularity
- Always require a datum reference (related control)

Tolerances of Orientation

- When applied to a plane surface:
  - provides a design refinement not controlled by size dimensions
  - controls “flatness” as well as orientation

Tolerances of Orientation

- When applied to a feature of size (pin, hole):
  - is a refinement of orientation within the stated locational tolerances
  - therefore, it must be less than the governing locational tolerances for the features involved

Perpendicularity

- The condition of a surface, median plane or axis being at a right angle to a datum plane or axis.
- Can be a refinement of another control, often position.

Tolerance zone:

- two parallel planes perpendicular to datum plane or axis within which:
  - the surface of feature must lie
  - the center plane of the feature must lie
  - the axis of the considered feature must lie

Tolerance zone:

- a cylindrical tolerance zone perpendicular to the datum plane/axis within which the axis of the considered feature must lie
- zone defined by two parallel lines perpendicular to datum plane/axis within which a line element of the surface must lie (radial)
Surface perpendicularity

- Sometimes referred to as "squareness"
- Considered feature (surface) must be within any specified size limits.
  - Tolerance applied in direction to maintain size limits
- Should be applied in the view where relationship is depicted.

Perpendicularity: Cylindrical Feature

- Specifies a cylindrical tolerance zone, diameter symbol is included preceding the perpendicularity value in control frame.
  - Axis of considered feature must lie within zone.
- Default condition (no symbol) is RFS
  - Tolerance stated is maximum, regardless of size of feature.

Cylindrical feature at MMC

- Typically requires inspection by functional gauge
- Tolerance zone size will increase as the part size deviates from MMC.

Parallelism

- The condition of a surface or center plane being equidistant at all points from a datum plane, or
- An axis equidistant along its length from a datum axis.

Tolerance zone:

- Two parallel planes, parallel to a datum plane within which:
  - The surface or center plane must lie
  - The axis of the considered feature must lie

Tolerance zone:

- A cylindrical tolerance zone parallel to one (or more) datum plane(s) or a datum axis within which the axis of the feature must lie.
- Two parallel lines, parallel to a datum plane or axis, within which the line element of the surface must lie.
Parallelism

- Parallelism tolerance must be less than the associated size dimension.
- Parallelism tolerance must be less than one-half the size tolerance.
- Same material conditions apply as perpendicularity
  - RFS is default, MMC permits variance in zone size with part size deviation

Angularity

- The condition of a surface or axis being at a specified angle (less than 90 degrees) from a datum plane or axis.

Tolerance zone

- Two parallel planes at the specified basic angle from a datum plane or axis, within which
  - the surface or plane must lie.
  - the axis of the considered feature must lie

Tolerance zone:

- A cylindrical tolerance zone whose axis is at the basic angle from the datum within which the axis of the considered feature must lie.
- Two parallel lines at a basic angle from the datum within which a line element of the surface must lie.

Angularity

- Desired angle must be included as a basic size.
- Part must be within size tolerance limits.
- Note that tolerance zone is never “wedge-shaped”.

Tolerance of Profile

- A profile is the planar shape or outline of a feature.
- Profiles established by projecting the object onto a plane or indicating cross-sectional cuts.
Tolerance of Profile

- Profile tolerance specifies a uniform boundary along the desired ideal profile within which the feature elements must lie.
- Tolerance may be applied to and/or composed from arcs, radii, curves, lines and flat surfaces.

Profile tolerance is specified as follows:

- An appropriate section or view is required which shows the desired profile.
- The profile is defined by basic dimensions, angles, radii, etc.
- The profile tolerance may be applied either bilaterally (both sides) or unilaterally (one side) to the true profile.
- Unilateral application is indicated by including a phantom line profile and a

Profile of a line:

- Tolerance zone is 2-dimensional extending along the length or width of the considered feature.
- Tolerance applies perpendicular to the line profile at all points along its length.

Profile of a surface:

- The tolerance zone is 3-dimensional extending along the length and width or circumference of the considered feature(s).
- Usually requires datum specification to ensure proper relation of the profile to mating surfaces.
- Where a mounting surface is established as the datum, the profile tolerance zone is established as perpendicular to the mounting surface.

Profile of a surface:

- Profile of a surface may be applied “all around” the shape of a part.
- In such cases, the profile is controlled in size and form simultaneously.
- The words or symbol for “all around” may be added to the control frame.

Runout tolerances:

- Applied to features that have a centerline
- Requires minimum datum reference of part axis
- Specified as either
  - Circular runout (less restrictive)
  - Total runout (more inclusive)
Circular runout

- Provides control of circular elements of a surface.
- May be applied to cylinders, cones, and surfaces of revolution (anything circular in cross-section).
- Less stringent than total runout since only circular elements controlled.

Circular runout

- A type of “spot-check” for rotational surface accuracy relative to the part axis.
- Is a composite control in that cumulative variations of circularity and cylindricity are controlled.

Circular runout

- Runout is applied independently at each circular measuring position as the part is rotated 360 degrees about the datum axis.
- Each circular element must be within the full indicator movement (FIM) as indicated by the stated tolerance.
- Circular runout requires a datum reference and is only applied on a RFS basis.

Total runout

- May be applied to surfaces of revolution about a datum axis and to surfaces constructed at right angles to the axis.
- Total runout applies to the composite control of all surface elements together and the respective feature measuring positions as the part is rotated through 360 degrees.

Total runout

- All concerned features must be within the FIM across the total surface controlled with one setting.
- Variations such as circularity, straightness, taper, concentricity, and profile are controlled and reflected on the datum axis surface requirement.