Geometric Tolerancing (topics)

The geometric characteristics and their symbols

Other related symbols and terms

The feature control frame and the datum feature symbol

Maximum material condition and Regardless of feature size

The distinction between:

- form
- orientation
- profile
- runout
- and location tolerances

Tolerance zones

The Basic Rules

Why use geometric tolerancing:

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

Why use symbolic specifications:

- Symbol has uniform meaning
- Are compact, quick drawn, readily adaptable to computer applications
- International acceptance (no language restriction)

- Definitions

Feature: Any portion of an object. May be a point, edge, centerline, or a plane or curved surface. Can also be a size, as in the width of a slot or diameter of a cylinder (aka size feature).

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 diameter as the cylinder, or straight lines parallel to the axis.

Regardless of Feature size: Tolerance specified applies no matter how large or small the manufactured size of the feature.

MMC/LMC: Maximum material occurs when a size feature contains the most permissible material. When used as a modifier in a feature control frame, the tolerance specified applies only when the controlled feature or datum is at its MMC (size).

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

- **Feature Control Frame**

  Contains:

  Kind of control (geometric characteristic)
  The geometric tolerance
  Any modifiers (such as MMC, RFS, etc.)
  Datum References: and datum reference modifiers (see above)

- **Datum Indication**

  Datum precedence is signified by the placement of the datum reference letters. e.g. order of letters indicates order of preference. No significance to alphabetic sequence.

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

- **Datum feature symbol**

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

- **Tolerances of form**

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

  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 a surface having all points within its boundary lie in a single plane. The flatness tolerance specifies a tolerance zone defined by two parallel planes within which the specified surface must lie.

  **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 appears as a line.
**Circularity:** The condition where:
1) for a cylinder or cone, all points off the surface intersected by any plane perpendicular to an axis are equidistant from their axis.
2) for a sphere, all points of the surface are intersected by any plane passing through a common center are equidistant from that center.

**Cylindricity:** Condition of a surface of revolution in which all points of the surface are equidistant from a common axis. Tolerance specifies a zone bounded by two concentric cylinders within which the surface must lie.

Tolerance zone is bounded by two concentric circles within which each circular element of the surface must lie.

---

**Datum Definition and Specification**

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

**Datum:** A datum is theoretically exact point axis or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location of geometric characteristics of a part are established.

Data indicated on drawings as points, axes, or planes are theoretical and do not exist on the actual parts. Data are assumed to exist in and be simulated by the processing equipment, such as machine table and surface plates. It is from this processing equipment that measurements are taken and the dimensions of features are verified.

**Datum Reference Plane:** Application of three mutually orthogonal planes to the part. It is established on a non-cylindrical part by 3 point contact (minimum) on the primary plane, 2 point on the secondary and one on the tertiary. Typically the most influential surface to part orientation is designated as primary.

**General Rules:**
Establish data on functional features
Non-functional features may be used for ease of manufacture and inspection
Mating parts should have the same data specifications
Choose data that are accessible and of sufficient length

Data are: Points (apex of cone) Lines (line on a surface) Planes (flat surface) Cylinders (holes) Widths (key way) Axis (center of two features)

---

**Tolerances of Orientation:** (Perpendicularly, Parallelism, Angularity)
Applied when: Relationship of surface of size features are required but which do not include positional control or a refinement of orientation within a position control is required.
**Perpendicularity:** The condition of a surface, median plane or axis at a right angle to a datum plane or axis. Can be a refinement of another control, often position.

Tolerance zone is:
- defined by two parallel planes perpendicular to a datum plane or axis (surface or median plane of considered feature must lie within)
- defined by two parallel planes perpendicular to a datum axis (axis of considered feature must lie within)
- defined by two parallel lines perpendicular to a datum plane or axis (element of considered surface must lie within)
- a cylindrical tolerance zone perpendicular to a datum plane within which the axis of the considered feature must lie

**Surface Perpendicularity**

Sometimes referred to as "squareness"
Always requires a datum reference
Perpendicularity applied to a surface also controls flatness of the surface to the extent of the specified tolerance
Considered feature must be within any specified limits of size
Should be applied in view where relationship is depicted

**Cylindrical feature at MMC**

Requires a datum reference
Defines a cylindrical tolerance zone perpendicular to a datum plane within which the axis of the considered feature must lie
Diameter symbol is included preceding the perpendicularity value in the control frame

**Non-cylindrical feature, RFS/MMC**

Requires a datum reference
Tolerance zone is the distance between two parallel planes separated by the stated tolerance value and perpendicular to the datum plane.
If RFS, tolerance stated is the maximum, regardless of feature size
Controlled feature must lie within any specified limits of size and position.

**Parallelism:**
Parallelism is the condition of a surface equidistant at all points from a datum plane; or an axis equidistant along its length from a datum axis.
Parallelism always requires a datum reference.
Parallelism must be less than the associated size dimension and less than one-half the size tolerance.
Parallelism applied to a surface includes flatness to the extent of the stated parallel tolerance.

Tolerance zone is defined by:
Two parallel planes or lines parallel to datum plane or axis.
A cylindrical zone whose axis is parallel to a datum axis within which the axis of the considered feature must lie.
Angularity:

The condition of a surface or axis at a specified angle (less than 90 degrees) from a datum plane or axis. Tolerance zone is defined by two parallel planes at the specified base angle, within which the surface or the axis of the considered feature must lie. Requires a datum reference. Desired angle must be included as a basic angle. Angularity applied to a surface includes flatness to the extent of the stated angularity tolerance.

- Tolerances of Profile

A profile is the shape or outline of a feature within a given plane. Profiles are established by projecting the object onto a plane or by indicating cross-sectional cuts through the object.

Profile tolerance specifies a uniform boundary along the desired true profile within which the feature elements (surface or line) must lie.

Profile tolerance may be applied to and/or composed from, arcs, radii, curves, lines or flat surfaces.

Profile tolerance is specified as follows:
An appropriate section or view is required which shows the basic desired profile.
The profile is defined by basic dimensions, angles, radii, etc.
The profile tolerance may be applied, as desired, either bilaterally (both sides) or unilaterally (one side) to the true profile.

Unilateral application is indicated by including a phantom line profile and a second arrowhead.

Profile of a surface:

The tolerance is three 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 mounting surfaces. Where a mounting surface is specified as the datum, the profile tolerance zone is established as perpendicular to the mounting surface.

Profile of a line:

The tolerance zone is two 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 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

Circular Runout:

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

- Provides composite control of all surface elements. 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 thru 360 degrees.
- 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 in the datum axis surface requirement.