MECH 313 Engineering
Drawing & Design

Lecture 5

Geometric Dimensioning and Tolerancing
Outline

- Projected tolerance zone
- Datum targets
- Circularity and cyclindricity
- Profile tolerancing
- Correlative tolerances
- Positional tolerancing for non-cylindrical features
- Positional tolerancing for multiple pattern feature
- Formulas for positional tolerancing
- Summary of rules for geometric tolerancing
The application of the projected tolerance zone concept is recommended when the variation in perpendicularity of threaded or press-fit holes could cause fasteners such as screws, studs, or pins to interfere with mating parts (Fig. 16-10-1). Interference can occur even if both the parts are made within tolerance limits.

![Diagram](image)

Fig. 16-10-1 Illustrating how a fastener can interfere with a mating part.
Projected tolerance zone

- Figure 16-10-2 shows how the projected tolerance zone concept is used in such cases.
- The inclination of the threaded hole is significant in the assembly as the angle of the fastener depends.
- Another important thing to consider is the thickness of the mating part.
- So the minimum projected tolerance zone is always the thickness of the mating part.

Fig. 16-10-2 Basis for projected tolerance zone.
Projected tolerance zone

- Specifying a projected tolerance zone will not interfere fasteners with mating parts.

The projected tolerance zone is shown by symbol $P$ and the value of the projection given after the symbol within the feature control frame.

- In some cases, the projected tolerance zone can be equal to the maximum height of the pin after installation instead of the thickness of mating parts.
Datum targets

Rationale

- When datums are mentioned, it is generally the full surface that is indicated.
- But it is not always practical mainly for 3 reasons
  - The feature surface may be too large that a gage designed to make contact fully maybe expensive or difficult to use
  - Functional requirements of the part may necessitate use of portion of a datum surface (contact point of mating parts in an assembly)
  - Surface selected as datum may not be sufficiently true (eg. Casting, sheet metal and formed parts)
- To avoid these problems a datum target method is used
Datum targets

Datum Target Symbol

- Points, lines, and areas on datum features are designated on the drawing by means of a datum target symbol.
- The symbol is placed outside the part with a radial line directed to the target point.

Fig. 16-11-1 Datum target symbol.

Fig. 16-11-2 Identification of datum targets.
Datum targets

**Datum Target Points**
- Shown on the desired location of the datum with ‘X’

**Datum Target Lines**
- Shown with a phantom line or direct view and/or ‘X’ on the edge view where the line appears as a point

**Datum Target Areas**
- When it is determined that an area or areas of flat contact are necessary to ensure establishment of datum, a target area of desired shape and size is specified. It is indicated by section lines enclosed by phantom line.

Fig. 16-11-3 Symbol for a datum target point.
Circularity and cylindricity

Circularity

- Circularity refers to a condition of a circular line or the surface of a circular feature in which all points on the line or on the circumference of a plane cross section of the feature are the same distance from a common axis or center point.
- Common errors are Ovality, Lobing and Irregularities

Fig. 16-12-1  Common types of circularity errors.
Circularity and cylindricity

**Circularity tolerance**

- It is similar to straightness, but wrapped in a circular cross section.
- The geometric symbol for circularity is a circle with dia 1.5 times that of lettering height.
- Circularity tolerance specifies the width between 2 circular rings within which the circumference of the feature in that plane should lie.

**Fig. 16-12-2**

16-12-3  Circularity tolerance applied to a cylindrical part.
Circularity and cylindricity

Circularity tolerance

- Each circular element of the surface must be within the limits of size
- Circularity is a surface form tolerance and it cannot be modified on LMC or MMC
- Circularity tolerance must be less than the size tolerance

Fig. 16-12-4  Circularity tolerance applied to a sphere.
Circularity and cylindricity

Circularity tolerance for non cylindrical parts

- Since the end view of these parts contain many circles, always the circularity tolerance is shown in the longitudinal surface.

**Fig. 16-12-5** Circularity tolerance applied to noncylindrical features.
Cylindricity is a condition of a surface in which all points of the surface are the same distance from a common axis. This is a composite control of form tolerance that includes circularity, parallelism and straightness.
Cylindricity tolerance

- Cylindricity tolerance specifies a zone bounded by 2 concentric cylinders within which the surface should lie.
- Here the tolerance applies to both longitudinal and circular elements of the feature.
- The feature control frame can be directed to either view.
- Cylindricity tolerance must be less than the size tolerance.
Circularity and cylindricity

Typical errors

- Cylindricity tolerance is similar to flat tolerance and controls only the surface. So MMC / LMC do not apply.
- In circular and cylindrical tolerance, the zone must lie in a space equal to half the size tolerance.

NOTE: CYLINDRICITY TOLERANCE MUST LIE WITHIN LIMITS OF SIZE.

16-12-8 Permissible form errors for part shown in Fig. 16-12-7.
Profile tolerancing

Profiles

- A profile is the outline form or shape of a line or surface. A line profile may be the outline of a part or feature as depicted in a view on a drawing. It may represent the edge of a part, or it may refer to line elements of a surface in a single direction, such as the outline of cross sections through the part. In contrast, a surface profile outlines the form or shape of a complete surface in three dimensions.

- A line profile may contain lines, arc or curved lines.

- Surface profile may contain flat, spherical or cylindrical surfaces, or surfaces composed of various lines in 2 or more directions.

H = HEIGHT OF LETTERS

Profile symbols.
Profile tolerancing

Profile of a line tolerance

- Is directed to a surface of any shape
- Used when it is not desirable to control the entire surface as a single entity
- If the tolerated line refers to a surface, it applies to all line elements parallel to one that is tolerated
- If nothing is specified, it is bilateral, the zone equally dispersed about the basic profile
- If unilateral variation is needed, they are specified by using phantom lines on the required side

Fig. 16-13-2 Simple profile with a bilateral profile tolerance zone.

Fig. 16-13-3 Unilateral tolerance zones.
Profile tolerancing

All round profile tolerance

- If the profile tolerance need to control all round the part, then the symbol for all round control is used

Dimensioning method

- The true profile is established with basic dimensions enclosed in a rectangular box to indicate the location of profile tolerancing
Profile tolerancing

**Dimensioning method**

- The height of the profile is \( .9 \pm .01 \). This is separately measured.
- The radius 1.5 is a basic dimension and becomes part of the profile.
- As per this drawing the profile tolerance zone has a radii of 1.497 to 1.503 and it can float within 0.89 and 0.91. **Position and form as separate requirements.**
Profile tolerancing

Profile of a surface tolerancing

- Applies over the whole surface (like flatness) instead of controlling lines and line elements (like straightness)
- This can be directed to any view, but mostly to the view that shows the shape of the profile
- Indicates a zone having form of the basic surface of uniform width equal to the tolerance value within which the entire surface must lie
Profile tolerancing

Profile of a surface tolerancing

- Profile tolerance is contained within the size limits
- All rules are similar to profile of a line tolerancing, except here a reference to datum is required to provide proper orientation of the profile
- All round symbol can be used here too to mention profile of surface tolerancing applies around the feature

Fig. 16-13-14  Profile-of-a-surface tolerance referenced to two datums.
Correlative tolerances

- **Correlative tolerancing** is to control 2 or more features intended to be correlated in position or attitude (coplanarity, concentricity, coaxiality, etc).

**Coplanarity**

- refers to the relative position of two or more flats that are intended to lie in the same geometric plane.
- Profile of surface tolerance symbol is used to treat 2 or more surfaces as single interrupted or noncontinuous surface.

16-14-1 Specifying profile-of-a-surface tolerance for coplanar surfaces.
Correlative tolerances

**Coplanarity**

- Sometimes, it may be desirable to mention which surface needs to be used to establish the tolerance zone.
- In that case, datum features are added and the identifiers marked in the feature control frame.
- Both unilateral and bilateral tolerance zone shown in this...
Correlative tolerances

Coplanarity

- Coplanar systems required to be perpendicular to the axis of the hole

(A) DRAWING CALLOUT

(B) TOLERANCE ZONE .004 WIDE PARALLEL TO DATUM B

Fig. 16-14-3 Surface referenced to a datum system.
Correlative tolerances

Concentricity

- Concentricity is a condition in which two or more features, such as circles, spheres, cylinders, or cones, have a common axis or center.

Concentricity Tolerance

- Concentricity tolerance controls variation in position, or eccentricity of the center line in relation to the datum feature.
- Specifies a cylindrical tolerance zone having a diameter equal to the specified tolerance whose axis coincides with a datum axis.
Concentricity
Concentricity

Parallelism

Angularity

Large axis shifting accuracy to concentricity tolerances
Correlative tolerances

Concentricity Tolerance

- The frame is attached to the leader or located below the dimension pertaining to the feature
- Concentricity tolerance and the datum reference, because of their unique characteristics, are always used on an RFS basis
- Figure 16-14-6 shows the extreme variations of eccentricity and parallelism that a concentricity tolerance will permit

16-14-6  Concentricity of cylindrical features.
Correlative tolerances

Concentricity Tolerance

- Concentricity tolerance may be referred to a datum system instead of a single datum.

Here the tolerance zone is perpendicular to datum A and concentric with the axis of datum B in the plane of datum A.

\[16-14-7\] Concentricity referenced to a datum system.
Correlative tolerances

Coaxiality

- Coaxiality is very similar to concentricity: with coaxiality, two or more circular or similar features are arranged with their axes in the same straight line.
- Eg. Counterbored hole, shaft having parts turned to different dia. along its length
- 4 methods of controlling coaxial features
  - Positional tolerancing
  - Runout tolerancing
  - Concentricity tolerancing
  - Profile tolerancing
Correlative tolerances

* Positional Tolerance control
  - When surfaces are cylindrical and control of axes in MMC, this is recommended
  - This permits use of simple gage for inspection
  - To control coaxiality within limits of size 0 tolerance is specified at MMC
  - Variations in coaxiality are permitted only when features depart from MMC
Coaxiality – Positional Tolerancing

(A) DRAWING CALLOUT
Coaxiality – Positional Tolerancing

Φ.010 TOLERANCE ZONE AT MMC

DATUM AXIS A

SEE DATUM FEATURE IN THE TABLE IN THE CHART BELOW.

MAXIMUM ALLOWABLE DISTANCE BETWEEN AXIS OF DATUM FEATURE AND AXIS OF CONSIDERED FEATURE (EQUAL TO ONE-HALF THE POSITIONAL TOLERANCE).
Coaxiality – Positional Tolerancing

<table>
<thead>
<tr>
<th>CONSIDERED FEATURE SIZES</th>
<th>DATUM FEATURE SIZES</th>
<th>.500</th>
<th>.498</th>
<th>.496</th>
<th>.494</th>
<th>.492</th>
<th>.490</th>
</tr>
</thead>
<tbody>
<tr>
<td>.990</td>
<td>.005</td>
<td>.006</td>
<td>.007</td>
<td>.008</td>
<td>.009</td>
<td>.010</td>
<td></td>
</tr>
<tr>
<td>.988</td>
<td>.006</td>
<td>.007</td>
<td>.008</td>
<td>.009</td>
<td>.010</td>
<td>.011</td>
<td></td>
</tr>
<tr>
<td>.986</td>
<td>.007</td>
<td>.008</td>
<td>.009</td>
<td>.010</td>
<td>.011</td>
<td>.012</td>
<td></td>
</tr>
<tr>
<td>.984</td>
<td>.008</td>
<td>.009</td>
<td>.010</td>
<td>.011</td>
<td>.012</td>
<td>.013</td>
<td></td>
</tr>
<tr>
<td>.982</td>
<td>.009</td>
<td>.010</td>
<td>.011</td>
<td>.012</td>
<td>.013</td>
<td>.014</td>
<td></td>
</tr>
<tr>
<td>.980</td>
<td>.010</td>
<td>.011</td>
<td>.012</td>
<td>.013</td>
<td>.014</td>
<td>.015</td>
<td></td>
</tr>
</tbody>
</table>

(B) ALLOWABLE DISTANCES BETWEEN AXES

Fig. 16-165  Positional tolerancing for coaxiality.
Correlative tolerances

Positional Tolerance control

Individual hole

Group of holes

Hole at MMC, GT = Ø .001 (each) and Ø .010 (group).

Hole at LMC, GT = Ø .011 (each) and Ø .020 (group).
Correlative tolerances

Positional Tolerance control

Φ .010 AT MMC, FOUR COAXIAL TOLERANCE ZONES LOCATED AT TRUE POSITION RELATED TO THE SPECIFIED DATUMS WITHIN WHICH THE AXES OF THE HOLES, AS A GROUP, MUST LIE

Φ .001 AT MMC, FOUR COAXIAL TOLERANCE ZONES WITHIN WHICH THE AXES OF THE HOLES MUST LIE RELATIVE TO EACH OTHER

(B) TOLERANCE ZONES

16-14-9 Positional tolerancing for coaxial holes of the same size.
Correlative tolerances

**Runout**

- Runout is a composite tolerance used to control the functional relationship of one or more features of a part to a datum axis.
- Each feature is within the runout tolerance when rotated around the datum axis.
- It is the full indicator movement (FIM) in inspection terminology.
Correlative tolerances

Circular Runout

- Provides control of any circular element along a surface (NOT ENTIRE SURFACE, and it does not control in any other direction).
- The tolerance is usually applied at any usual measuring position when part is rotated 360°.
- When applied to surfaces around the axis it controls circularity and coaxiality (HOW).
- When done at surfaces 90° to axis it controls wobble at all positions.
- When runout tolerance applies to specific portion, it is indicted by chain line and basic dimension.
Correlative tolerances

**Total Runout**

- Provides control of entire surface and not only of each circular element
- The indicator is moved to extent of the surface while the part is rotated 360°
- Reading is taken along the surface without resetting the indicator. The difference between the maximum and minimum position of the indicator is the total runout
- The tolerance zone here is the space between the 2 cylinders separated by specified tolerances and coaxial with the datum axis.
- Total runout is more costly to verify than circular runout and hence seldom used
Correlative tolerances

Total Runout

- In the previous example, the datum is established by centers drilled at the ends of the part. This is ideal to mount the part on these centers for measurement.
- When centers are not provided, any cylindrical or conical surface is used to establish datum axis. Here 2 datum diameters act as a single datum axis.
Positional tolerancing for noncylindrical features

Non-cylindrical features - MMC

- It finds great use in controlling position of holes (lecture 4)
- In addition, it also finds use in locating non cylindrical features such as slots, grooves, tabs etc
- For non cylindrical features, the tolerance zone is used to locate the center plane by establishing two parallel surfaces of the feature
- The dia symbol $\Phi$ is removed from the feature control frame
- And it is mostly done at MMC condition

Fig. 16-9-14  Positional tolerancing—MMC.
Positional tolerancing for noncylindrical features

Non-cylindrical features

• If PT is applied to non-cylindrical features (slots) at MMC
  • The slot must be within the limit of size
  • The gauge should be having width equal to VC of the slot (size at MMC minus the positional tolerance value)
• If located at the true position the gauge must be able to enter the slot
Positional tolerancing for noncylindrical features

Non-cylindrical features

Slot position may vary as shown, but no point of either side surface shall be inside of W.

(A) Slot shown in extreme right position

Side surfaces of slot may vary in attitude, provided W is not violated and slot width is within limits of size.
Positional tolerancing for noncylindrical features

Non-cylindrical features

Fig. 16-15-3  Positional tolerancing applied to slots at MMC.
Positional tolerancing for multiple patterns

- Positional tolerancing may also be used to locate multiple patterns of features if each pattern is referenced to common datums and referenced in the same order of precedence.
- Actual centres of all the holes must lie within the tolerance zone measured from Datums A, B, C.
- By doing PT for multiple patterns, the whole object as a group can be measured with a single gage.
- Hence most of PT is done on an MMC basis for ease of checking with gauge.
Positional tolerancing for multiple patterns

16-6-1 Multiple pattern of features
Positional tolerancing for multiple patterns

Tolerance zones for hole patterns shown in Fig. 16-183.
Positional tolerancing for multiple patterns

PT for Multiple patterns

Two different GT.

- If such a requirement is not needed to control different group of multiple patterns in one drawing, then a term SEP REQT is added below the feature control frame to differential between the groups that require different tolerance zones.
Formulas for positional tolerancing

**Purpose**

- To provide formulas for determining required PT or the sizes of mating features to ensure proper assembly.
- They are valid for all types of features are patterns, and will give ‘no interference, no clearance’ fit at MMC and features located at the extreme of their Positional Tolerancing zone.

- \( F = \) maximum diameter of fastener (MMC limit)
- \( H = \) minimum diameter of clearance hole (MMC limit)
- \( T = \) positional tolerance diameter

- Subscripts are used when more than one feature of size or tolerance is involved. For example:
  - \( H_1 = \) minimum dia of hole in part 1
  - \( H_2 = \) minimum dia of hole in part 2

*Fig. 16-17-1*  Formula symbols.
Formulas for positional tolerancing

**Floating Fasteners**

- Where two or more parts are assembled with fasteners such as bolts and nuts and all parts have clearance holes for the bolts, the condition is termed a floating-fastener case.
- When the fasteners are of same dia and same clearance and positional tolerance for the parts to be assembled are desired.

\[ T = H - F \]

**Example 1**

- If the fasteners shown in 16-17-2 are .312 in. in maximum dia and the clearance holes are .339 in. in minimum dia (both at MMC) the required positional tolerance is

\[ T = 0.339 - 0.312 \]
\[ = 0.027 \text{ in. for each part} \]
Formulas for positional tolerancing

- Any number of parts with different hole sizes and PTs can be mated if the formula is applied to each part individually.

**Example 2**

- Figure 16-17-3 shows two parts with positional tolerances of 0.030 in. on an RFS basis.
- In this example the minimum hole diameter for both parts is 
  \[ H = F + T = 0.500 + 0.03 = \Ø.530 \text{ in.} \]
Formulas for positional tolerancing

- Figure 16-17-4 shows these parts (16-7-3) with holes in an extreme position.
- If MMC is specified, the calculations are same as that for RFS but.
- In RFS, if hole sizes approach MMC, more clearance is provided all round.
- In MMC the clearance increases as the part moves away from MMC to LMC.

![Diagram showing extreme positions of holes in MMC and RFS basis](image1)

![Diagram showing holes in extreme position for fig 16-7-3](image2)

Between .985” and 1.015”.
Formulas for positional tolerancing

Fixed Fasteners
When one of the parts to be assembled has restrained fasteners, such as screws or studs in tapped holes, the condition is termed a fixed-fastener case.

Unequal Tolerances and Hole Sizes
It is sometimes desirable to have different tolerances for location or different hole sizes in each of the assembled parts. One reason may be because one part already exists and the other must be designed to mate with it. In such cases the hole sizes and the positional tolerances must be separated, and the previous formula, \( H = F + T \), becomes

\[
H_1 + H_2 = 2F + T_1 + T_2 \quad \text{or} \quad T_1 + T_2 = H_1 + H_2 - 2F
\]

For example, if it is desirable that the part with the tapped holes in example 4 have a larger positional tolerance than the part with clearance holes, \( T \) can be separated into \( T_1 \) and \( T_2 \) in any appropriate manner such that

\[
T = \frac{[T_1 + T_2]}{2}
\]
Formulas for positional tolerancing

**Fixed Fasteners**

- When one of the parts to be assembled has restrained fasteners, such as screws or studs in tapped holes, the condition is termed a fixed-fastener case.
- When the fasteners are of same dia and same clearance and positional tolerance for the parts to be assembled are desired.
- Allowable positional tolerance for fixed fasteners is 1 half of that of floating fasteners (subject to ‘⊥’ errors covered by projected tolerance zone discussed).

**Example 4**

- If the fasteners shown in 16-17-7 are 1in. in maximum dia and the clearance holes are 1.06 in. in minimum dia (both at MMC) the required positional tolerance is

\[ T = \frac{H - F}{2} \text{ or } H = F + 2T \]

\[ T = \frac{1.06 - 1.00}{2} = 0.03\text{in} \]

for each part.
Formulas for positional tolerancing

- If the fasteners shown in 16-17-7 are 0.5in. in maximum dia and the clearance holes are .560 in. in minimum dia (both at MMC).
- Two different positional tolerances are required, the larger tolerance for the clearance holes.

\[ T = \frac{H-F}{2} \text{ or } H = F + 2T \]

- \( T \) is \{(0.560-0.5)/2\} = 0.030in each Total = 0.060in.
- Here \( T_1 \) can be .024in and \( T_2 \) would be .036in (larger \( \Phi \) for clearance hole).
- The general formula for fixed fasteners where two mating parts have different PT is that \( (T_1 + T_2 = H-F) \)
Formulas for positional tolerancing

Coaxial Features

- Formula for floating fasteners also applies where mating parts have two coaxial features, and one feature is the datum for the other.
- If the available tolerance is to be unequally divided over the parts then
  \[ T_1 + T_2 = (H_1 + H_2) - (F_1 + F_2) \]

**Example 6**
- This is valid only for simple 2 feature parts as shown in 16-17-8.
- By applying the above formula (at MMC)
  \[ T_1 + T_2 = (H_1 + H_2) - (F_1 + F_2) \]
  \[ = (1.002 + .501) - (1.000 + .500) \]
  \[ = .003 \text{ is the total available tolerance} \]
- If T1 is .002in., then T2 is .001in.
Rules for geometric tolerancing

When to Use Geometric Tolerancing

- It is not necessary to use geometric tolerances for every feature on a part drawing.
- In most cases it is to be expected that if each feature meets all dimensional tolerances, form variations will be adequately controlled by the accuracy of the manufacturing process and equipment used.
- This is supplemented by the partial degree of control exercised by the measuring and gaging procedure used.
- If there is any doubt about adequacy of control by dimension tolerancing alone, then form, position or orientation tolerance should be used as discussed.
- It is necessary if shape or form must be held within limits which is not ordinarily possible by normal production methods.
- Also if the parts are subcontracted out for manufacture where widely varying expertise and equipment are available it is better to give complete dimensions and tolerances (including geometric).
Rules for geometric tolerancing

Basic Rules

- Geometric tolerances can be categorized into three basic groups
  - Form, to control the form and shape of features
  - Angularity, to control orientation of features
  - Position, to control location of features

- Any of these tolerances can be applied to lines and surfaces of any size or shape.

- Since straight lines and circular lines, as well as flat and cylindrical surfaces, occur so frequently in practice, special names and symbols have been established for their control. These special designations should be used for such lines and surfaces instead of the categorized names given above
  - Form of a line, straightness and circularity
  - Form of a surface, flatness and cylindricity
  - Orientation of a line, surface, or feature, angularity, parallelism, and perpendicularity
  - Location of features, (true) position, concentricity, and symmetry
Rules for geometric tolerancing

Positional Tolerancing

- The locational tolerance of position may be applied to an axis or a center plane. All positional tolerances, when applied to a feature of size that incorporates a dimension, such as a diameter or thickness, may be modified by RFS, MMC, or LMC

Tolerance of Position

- The RFS, MMC, or LMC must be specified for tolerances of true position on the drawing with respect to the individual tolerance, datum reference, or both, as applicable.
Rules for geometric tolerancing

Limits of Size

1. The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at MMC.
2. Where it is desired to permit a surface or surfaces of a feature to exceed the boundary of perfect form at MMC, the feature control frame must be associated with the size dimension.
3. Tolerances The limits of size do not control the orientation or location relationship between individual features.

Form and Orientation

- The Form tolerances control straightness, flatness, circularity, and cylindricity. Orientation tolerances control angularity, parallelism, and perpendicularity. A profile tolerance may control form, orientation, and size, depending on how it is applied.
Rules for geometric tolerancing

Profile Tolerancing

- The profile tolerance specifies a uniform boundary along the true profile within which the elements of the surface must lie. It is used to control form or combinations of size, form, and orientation.

Coaxiality Control

- Coaxiality is the condition in which the axes of two or more surfaces of revolution are coincident.
<table>
<thead>
<tr>
<th>Geometric Characteristic</th>
<th>Symbol</th>
<th>Applicable to Feature Being Controlled</th>
<th>Applicable to Datum Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straightness</td>
<td></td>
<td>Not applicable for a plane surface or a line on a surface. MMC or RFS applicable if tolerance applies to the median line or median plane of a feature of size, e.g., a hole, shaft, or slot.</td>
<td>No datum reference.</td>
</tr>
<tr>
<td>Flatness</td>
<td>□</td>
<td>Not applicable.</td>
<td>No datum reference.</td>
</tr>
<tr>
<td>Circularity</td>
<td>○</td>
<td>Not applicable.</td>
<td>No datum reference.</td>
</tr>
<tr>
<td>Cylindricity</td>
<td>□</td>
<td>Not applicable.</td>
<td>MMC not applicable.</td>
</tr>
<tr>
<td>Profile of a Line</td>
<td>□</td>
<td>Not applicable.</td>
<td>RFS applicable only to datum features of size having an axis or center plane.</td>
</tr>
<tr>
<td>Profile of a Surface</td>
<td>□</td>
<td>Not applicable.</td>
<td>MMC, LMC, and RFS applicable only to datum features of size having an axis or center plane.</td>
</tr>
<tr>
<td>Perpendicularity</td>
<td>⊥</td>
<td>Not applicable for a plane surface.</td>
<td>Not applicable to a single-plane surface.</td>
</tr>
<tr>
<td>Parallelism</td>
<td>//</td>
<td>MMC, LMC, and RFS applicable if tolerance applies to an axis, or center plane of a feature of size.</td>
<td>MMC, LMC, and RFS applicable only to datum features of size having an axis or center plane.</td>
</tr>
<tr>
<td>Angularity</td>
<td>&lt;</td>
<td>MMC, LMC, and RFS applicable if tolerance applies to an axis, or center plane of a feature of size.</td>
<td>MMC, LMC, and RFS applicable only to datum features of size having an axis or center plane.</td>
</tr>
<tr>
<td>Position</td>
<td>⊙</td>
<td>MMC, LMC, and RFS applicable if tolerance applies to an axis, or center plane of a feature of size.</td>
<td>MMC, LMC, and RFS applicable only to datum features of size having an axis or center plane.</td>
</tr>
<tr>
<td>Concentricity*</td>
<td>ø</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Symmetry</td>
<td>—</td>
<td>Applicable only to RFS.</td>
<td></td>
</tr>
<tr>
<td>Circular Runout</td>
<td></td>
<td>Applicable only to RFS.</td>
<td></td>
</tr>
<tr>
<td>Total Runout</td>
<td></td>
<td>Applicable only to RFS.</td>
<td></td>
</tr>
</tbody>
</table>

* ISO permits concentricity to be used on an MMC basis.
** Arrows may be filled in.

Fig. 16-18-1 Application of MMC, LMC, and RFS.