Geometric tolerances
A workshop guide

Supplementary to Standard 51-3-5
**INTRODUCTION**

This guide is intended for use in Production and Quality areas. It illustrates and interprets the Geometric Tolerancing Symbols used on drawings.

More detailed instructions for designers are given in 51-3-5 'Geometric Tolerancing'.

**RELATED STANDARDS**

- 51-3-3 Glossary of Drawing Practice Terms
- 51-3-4 Engineering Drawing Practice
- 51-3-5 Geometric Tolerancing
- 51-3-6 Datums

**GEOMETRIC TOLERANCING SYMBOLS**

<table>
<thead>
<tr>
<th>TYPE OF TOLERANCE</th>
<th>CHARACTERISTIC</th>
<th>SYMBOL</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRAIGHTNESS</td>
<td></td>
<td></td>
<td>2</td>
</tr>
<tr>
<td>FLATNESS</td>
<td></td>
<td></td>
<td>3</td>
</tr>
<tr>
<td>ROUNDNESS</td>
<td></td>
<td></td>
<td>4</td>
</tr>
<tr>
<td>CYLINDRICITY</td>
<td></td>
<td></td>
<td>5</td>
</tr>
<tr>
<td>PROFILE OF A LINE</td>
<td></td>
<td></td>
<td>5</td>
</tr>
<tr>
<td>PROFILE OF A SURFACE</td>
<td></td>
<td></td>
<td>7</td>
</tr>
<tr>
<td>PARALLELISM</td>
<td></td>
<td></td>
<td>8</td>
</tr>
<tr>
<td>PERPENDICULARITY</td>
<td></td>
<td></td>
<td>9</td>
</tr>
<tr>
<td>ANGULARITY</td>
<td></td>
<td></td>
<td>10</td>
</tr>
<tr>
<td>POSITION</td>
<td></td>
<td></td>
<td>11</td>
</tr>
<tr>
<td>COAXIALITY/CONCENTRICITY</td>
<td></td>
<td></td>
<td>12</td>
</tr>
<tr>
<td>SYMMETRY</td>
<td></td>
<td></td>
<td>13</td>
</tr>
<tr>
<td>RUNOUT</td>
<td></td>
<td></td>
<td>14</td>
</tr>
<tr>
<td>TOTAL RUNOUT</td>
<td></td>
<td></td>
<td>15</td>
</tr>
<tr>
<td>DATUM</td>
<td></td>
<td></td>
<td>16</td>
</tr>
<tr>
<td>DATUM TARGET</td>
<td></td>
<td></td>
<td>17</td>
</tr>
<tr>
<td>BOX DIMENSION</td>
<td></td>
<td></td>
<td>18</td>
</tr>
<tr>
<td>TRUE POSITION</td>
<td></td>
<td></td>
<td>19</td>
</tr>
<tr>
<td>PROJECTED TOLERANCE ZONE</td>
<td></td>
<td></td>
<td>19</td>
</tr>
<tr>
<td>REGARDLESS OF FEATURE SIZE</td>
<td></td>
<td></td>
<td>20</td>
</tr>
<tr>
<td>MAXIMUM MATERIAL CONDITION</td>
<td></td>
<td></td>
<td>21</td>
</tr>
</tbody>
</table>
**STRAIGHTNESS**

**EXAMPLE**

- \[ -\Delta 0.03 \]

**INTERPRETATION**

The tolerated surface must lie wholly within a tolerance zone formed by parallel lines 0.03 apart.

**NOTE**

During actual measurement the component must NOT be rotated.

**FLATNESS**

**EXAMPLE**

- \[ -\phi 0.03 \]

**INTERPRETATION**

The axis of the component must lie wholly within a cylindrical tolerance zone 0.03 in diameter.

**SPECIAL CASE**

During actual measurement the component must NOT be rotated.

**CONCAVE SURFACE**

**CONVEX SURFACE**
ROUNDNESS

EXAMPLE

INTERPRETATION

The entire periphery at the considered cross-
section, must lie within a tolerance zone 0,03-
wide, the annular zone being formed by two
concentric circles whose radii differ by 0,03.

Note
Roundness can only be truly measured on a
roundness measuring instrument.
The use of vees or centres in conjunction with an
indicating device always brings in factors which
detract from the truth of the measurement.

CYLINDRICITY

EXAMPLE

INTERPRETATION

The considered surface must be contained
between two coaxial cylinders, the radii of which
differ by 0,04.

Note
Cylindricity is difficult to measure in practice,
but can be verified by measuring Roundness,
Straightness and Parallelism separately.
PROFILE OF A LINE

EXAMPLE

INTERPRETATION

TRUE PROFILE

TOL ZONE 0.15

The line must lie wholly within a tolerance zone 0.15 wide equally disposed about the perfect profile. The tolerance is always bilateral unless otherwise shown.

INDICATION OF UNILATERAL TOLERANCE

TOL ZONE 0.15

The line must lie wholly within a 0.15 tolerance zone completely disposed on the indicated side of the perfect profile.

PROFILE OF A SURFACE

EXAMPLE

INTERPRETATION

TOL ZONE 0.03

The surface must lie wholly within the 0.03 tolerance zone normal to the perfect profile. Tolerance is always bilateral unless otherwise shown.

INDICATION OF UNILATERAL TOLERANCE

TOL ZONE 0.03

Unilateral tolerance may be indicated in the same way as for Profile of a Line, see page 6 facing.
**PARALLELISM**

**EXAMPLE**

![Diagram of parallelism example]

**INTERPRETATION**

The surface must lie wholly between two planes 0.03 apart parallel with the datum surface.

**PERPENDICULARITY (SQUARENESS)**

**EXAMPLE**

![Diagram of perpendicularity example]

**INTERPRETATION**

The surface must lie wholly within two parallel planes 0.03 apart perpendicular to the Datum Surface.

**EXAMPLE**

![Diagram of squareness example]

**INTERPRETATION**

The axis of the feature must lie wholly within a cylinder 0.03 dia perpendicular to the Datum Axis.
ANGULARITY

EXAMPLE

INTERPRETATION

The surface must lie wholly between two parallel planes 0.01 apart at the specified angle to the Datum Surface.

POSITION

EXAMPLE

INTERPRETATION

The centre of the tolerated feature must fall within a zone 0.03 dia with its centre in the exact position indicated by the Boxed Dimensions.
**COAXIALITY/CONCENTRICITY**

**EXAMPLE**

The axis of the tolerated feature must fall within a cylindrical zone 0.03 dia coaxial with the datum axis.

**INTERPRETATION**

Cylinder 0.03 dia.

Datum axis

**SYMMETRY**

**EXAMPLE**

The median plane of the slot must fall between two parallel planes 0.1 apart, equidistant on either side of the datum axis.

**INTERPRETATION**

Datum Diameoter

Datum Axis

TOL ZONE 0.1
RUNOUT

EXAMPLE

INTERPRETATION

The tolerance given is the maximum permissible radial or axial variation of a position during one revolution about the Datum Axis, without axial movement.

TOTAL RUNOUT

EXAMPLE

INTERPRETATION

The tolerance given is the maximum permissible full indicator movement (FIM) during a number of revolutions about the Datum Axis, during which the indicator is traversed along the specified contour.
DATUM TARGET

Points, lines, or limited areas on the workpiece to be used for contact with the manufacturing and inspection equipment to define the required datum or to satisfy the functional requirements.

Refer also to 51-J-6 'Datums'

Datum Points A1, A2 & A3 establish Datum A

Refer also to 51-J-6 'Datums'
**BOXED DIMENSION (TRUE POSITION)**

Boxed dimensions show the true position of a hole, slot, boss, profile or other feature and are theoretically exact location. Boxed dimensions are never individually tolerated but are always accompanied by a positional or zone tolerance of the feature to which they refer.

---

**PROJECTED TOLERANCE**

A Projected Tolerance Zone is one which is situated at some distance from the considered feature so that the projection of a fastener (or other feature) located by the considered feature (thread, press fit, hole etc.) upon installation will have adequate perpendicularity control over its length beyond engagement.

The axis of each hole shall fall within a cylinder 0.02 diameter, the axis of which is in the exact specified position and perpendicular to the datum surface; extending 40mm therefrom.
### Maximum Material Condition

Maximum material condition (M) is the condition wherein the tolerance of form or position of a feature must be met irrespective of where the feature lies within its own tolerance.

**Note:** All geometric tolerances are regardless of feature size unless modified by M (Maximum Material Condition).

**Symbol:**
There is no BSI or ISO standard symbol for Regardless of Feature Size.

---

### Regardless of Feature Size

Regardless of Feature Size is the condition wherein the tolerance of form or position of a feature must be met irrespective of where the feature lies within its own tolerance.

**Note:** All geometric tolerances are regardless of feature size unless modified by M (Maximum Material Condition).

**Symbol:**
There is no BSI or ISO standard symbol for Regardless of Feature Size.

---

**Maximum Material Condition**

Maximum material condition (M) is that condition of a part or feature wherein it contains the maximum amount of material.

For example: Minimum Size Hole  
Maximum Size Shaft

The use of the M principle allows an increase in the specified feature tolerance by indicating that the tolerance applies to the feature only at its maximum material condition.

The M condition concept may be applied to:

- Straightness
- Parallelism
- Perpendicularity
- Angularity
- Position
- Coaxiality/Concentricity
- Symmetry

When applied to a feature (Hole Slot etc) but not to a plane surface or line on a surface.

The M condition may be applied to a datum feature.

The M condition may not be applied to:

- Flatness
- Roundness
- Cylindricity
- Runout
- Profile of a Line
- Total Runout
- Profile of a Surface