Lecture # 27

Geometric Tolerances

Geometric Tolerances

The competitive manufacturing of mating parts need mass-production and interchangeability.

These parts which have been produced with all their dimensions within their predetermined limits, need not be made in the same shop or even in the same company.

So, the aim is to manufacture the components in the true shape and size at the economical rate with highest quality. but

Because of limitations of men and machines, the parts produced are going to differ in the shape and size from its true geometrical form.

This deviation of part from its true geometrical form is known as *geometrical deviation and this* variations of geometrical form is known as *geometrical tolerances.*

The deviations of size of part from its designed dimensions are termed as *dimensional deviation and this* variations of size- dimensions are known as *dimensional tolerances.*

Geometric Tolerances

In a drawing of machine or machine parts, when it is submitted to the manufacturing shops. The following additional information should be incorporated:

(i) Quantity to be produced

(ii) Raw material size

(iii) Geometrical tolerances (permissible errors in geometrical form of a part)

(iv) Dimensional tolerances (permissible errors in dimensions of a part)

(v) Surface roughness or surface quality

(vi) Manufacturing methods

(vii) Applicable National standards

(viii) Special instructions.

The drawing of orthographic views of any machine parts with above information is known as production drawing or blue-print.

Geometric Tolerances

A shape of machine part is usually of prisms, cylinders. cones and spheres.

The geometrical deviations in these solids are bound to occur during production.

The geometrical tolerances indicate the conditions of

straightness,

flatness,

parallelism,

perpendicularity,

angularity,

symmetry,

concentricity and

roundness.

Geometric Tolerances

Tolerances of straightness:

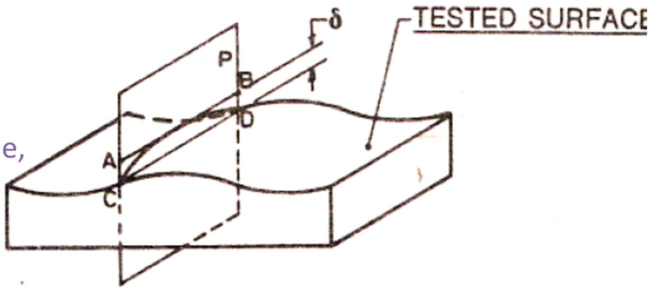
It is determined as the distance between two parallel lines in plane *P* right angle with the tested surface.

A straightness tolerance controls:

1 the straightness of a line on a surface,

2 the straightness of a line in a single plane,

3 the straightness of an axis.



(i) STRAIGHTNESS

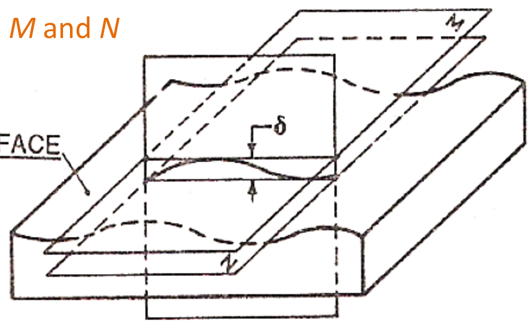
Geometric Tolerances

Tolerances of flatness:

It is expressed as the distance between two planes *M* and *N* which enclose the tested surface between them.

Flatness tolerances control the divergence or departure of a surface from a true plane.

The tolerance of flatness is the specified zone between two parallel planes.



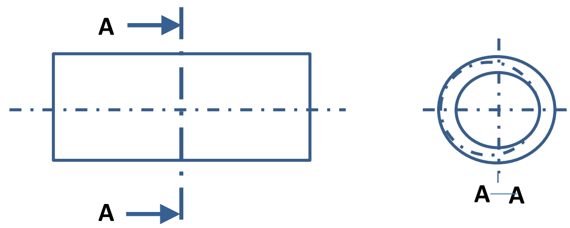
(ii) FLATNESS

It does not control the squareness or parallelism of the surface in relation to other features, and it can be called for independently of any size tolerance.

Geometric Tolerances

Circularity (roundness)

Circularity is a condition where any point of a feature's continuous curved surface is equidistant from its centre, which lies in the same plane.



The tolerance of circularity controls the divergence of the feature, and the annular space between the two coplanar concentric circles defines the tolerance zone, the magnitude being the algebraic difference of the radii of the circles.

Geometric Tolerances

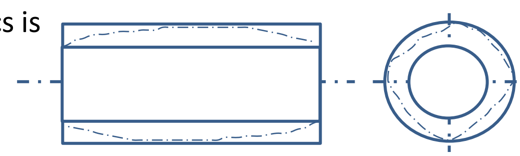
Cylindricity

The combination of parallelism, circularity and straightness defines cylindricity when applied to the surface of a cylinder, and is controlled by a tolerance of cylindricity.

The tolerance zone is the annular space between two coaxial cylinders, the radial difference being the tolerance value to be specified.

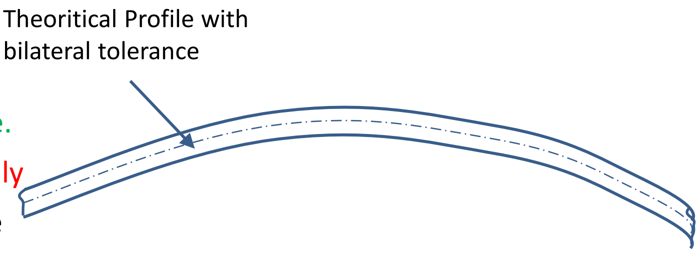
It should be mentioned that, due to difficulties in checking the combined effects of parallelism, circularity and straightness,

it is recommended that each of these characteristics is toleranced and inspected separately.



Geometric Tolerances

Profile tolerance of a line



Profile tolerance of a line is used to control the ideal contour of a feature.

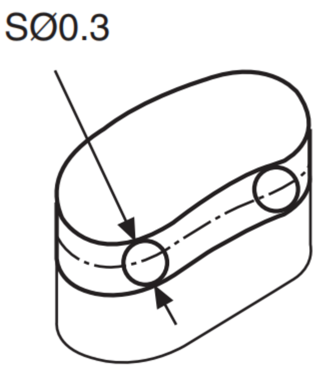
The contour is defined by theoretically exact boxed dimensions and must be accompanied by a relative tolerance zone.

This tolerance zone, unless otherwise stated, is taken to be equally disposed about the true form, and the tolerance value is equal to the diameter of circles whose centres lie on the true form.

If it is required to call for the tolerance zone to be positioned on one side of the true form (i.e. unilaterally), the circumferences of the tolerance-zone circles must touch the theoretical contour of the profile.

Geometric Tolerances

Profile tolerance of a surface



Profile tolerance of a surface is used to control the ideal form of a surface, which is defined by theoretically exact boxed dimensions and must be accompanied by a relative tolerance zone.

The profile-tolerance zone, unless otherwise stated, is taken to be bilateral and equally disposed about its true-form surface.

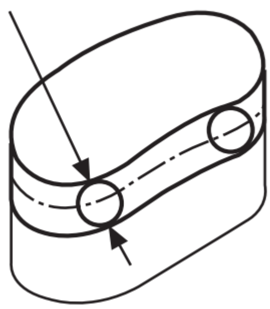
Geometric Tolerances

Profile tolerance of a surface

The tolerance value is equal to the diameter of spheres whose centre lines lie on the true form of the surface.

The zone is formed by surfaces which touch the circumferences of the spheres on either side of the ideal form.

If it is required to call for a unilateral tolerance zone, then the circumferences of the tolerance-zone spheres must touch the theoretical contour of the surface.



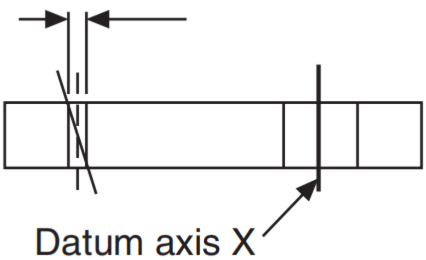
Geometric Tolerances

Parallelism

Two parallel lines or surfaces are always separated by a uniform distance.

Lines or surfaces may be required to be parallel with datum planes or axes.

Tolerance zones may be the space between two parallel lines or surfaces, or the space contained within a cylinder positioned parallel to its datum.



The magnitude of the tolerance value is the distance between the parallel lines or surfaces, or the cylinder diameter.

Geometric Tolerances

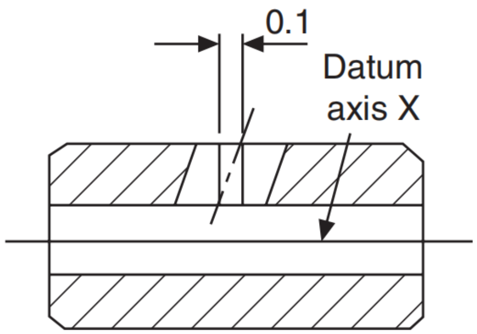
Perpendicularity (Squareness)

perpendicularity is the condition when a line, plane or surface is at right angles to a datum feature.

The tolerance zone is the space between two parallel lines or surfaces;

it can also be the space contained within a cylinder.

All tolerance zones are perpendicular to the datum feature.



The magnitude of the tolerance value is the specified distance between these parallel lines or surfaces, or the diameter of the cylinder.

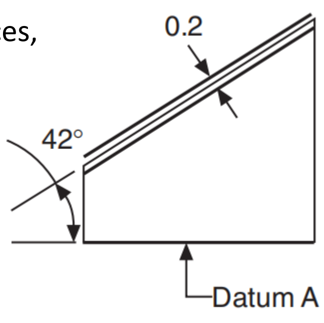
Geometric Tolerances

Angularity

Angularity defines a condition between two related planes, surfaces, or lines which are not perpendicular or parallel to one another.

Angularity tolerances control this relationship.

The specified angle is a basic dimension, and is defined by a theoretically exact boxed dimension and must be accompanied by a tolerance zone.



This zone is the area between two parallel lines inclined at the specified angle to the datum line, plane, or axis.

The tolerance zone may also be the space within a cylinder, the tolerance value being equal to the cylinder diameter: In this case, symbol Ø precedes tolerance value in the tolerance frame.

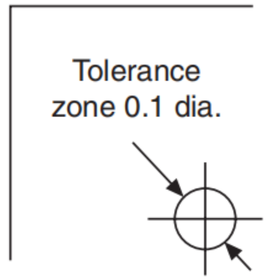
Geometric Tolerances

Position

A positional tolerance controls the location of one feature from another feature or datum.

The tolerance zone can be the space between two parallel lines or planes, a circle, or a cylinder.

The zone defines the permissible deviation of a specified feature from a theoretically exact position.



The tolerance value is the distance between the parallel lines or planes, or the diameters of the circle or cylinder.

The theoretically exact position also incorporates squareness and parallelism of the tolerance zones with the plane of the drawing.

15

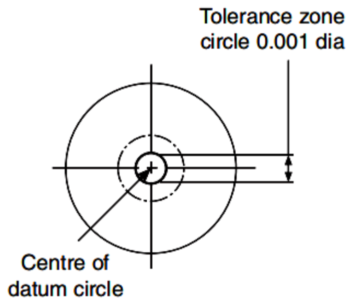
Geometric Tolerances

Concentricity and coaxiality

Two circles are said to be concentric when their centres are coincident.

Two cylinders are said to be coaxial when their axes are coincident.

The deviation from the true centre or datum axis is controlled by the magnitude of the tolerance zone.



16

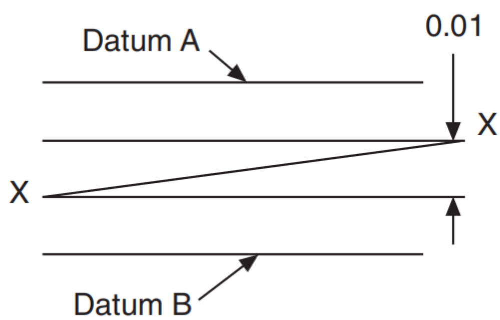
Geometric Tolerances

Symmetry

Symmetry involves the division of spacing of features so that they are positioned equally in relation to a datum which may be a line or plane.

The tolerance zone is the space between two parallel lines or planes, parallel to, and positioned symmetrically with the datum.

The tolerance magnitude is the distance between these two parallel lines or planes.



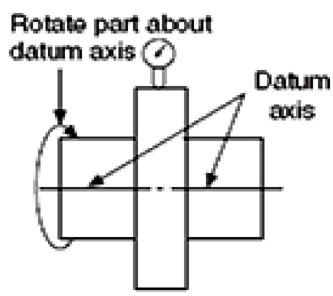
Symmetry also implies perpendicularity with the plane of the drawing where depth is involved.

Geometric Tolerances

Circular run-out & Total run-out

Circular run-out is a unique geometrical tolerance.

It can be a composite form control relating two or more characteristics, and it requires a practical test where the part is rotated through 360° about its own axis.



The results of this test may include errors of other characteristics such as circularity, concentricity, perpendicularity, or flatness, but the test cannot discriminate between them.

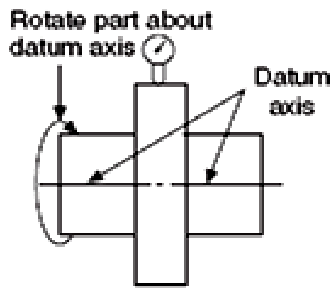
It should therefore not be called for where the design function of the part necessitates that the other characteristics are to be controlled separately.

Geometric Tolerances

Circular run-out & Total run-out

The sum of any of these errors will be contained within the specified circular run-out tolerance value.

The tolerance value is equal to the full indicator movement of a fixed point measured during one revolution of the component about its axis, without axial movement.



Circular run-out is measured in the direction specified by the arrow at the end of the leader line which points to the toleranced feature.

It must always be measured regardless of feature size, and it is convenient for practical purposes to establish the datum as the diameter(s) from which measurement is to be made, if true centres have not been utilized in manufacturing the part.

Geometric Tolerances

Symbols for geometrical characteristics

Type of Tolerance	Characteristics to be toleranced	Symbol	Datum needed	Applications
Form	Straightness	—	No	A Straight line. The edge or axis of a feature
	Flatness	⏥	No	A plane surface
	Roundness	○	No	The periphery of a circle.Cross-section of a bore, cylinder, cone or sphere
	Cylindricity	⌭	No	The combination of circularity, straightness and parallelism of cylindrical surfaces. Mating bores and plungers
	Profile of a line	⌒	No	The profile of a straight or irregular line
	Profile of a Surface	⌓	No	The profile of a straight or irregular surface

Geometric Tolerances

Symbols for geometrical characteristics

Type of Tolerance	Characteristics to be toleranced	Symbol	Datum needed	Applications
Orientation	Parallelism	//	Yes	Parallelism of a feature related to a datum. Can control flatness when related to a datum
	Perpendicularity	⊥	Yes	Surfaces, axes, or lines positioned at right angles to each other
	Angularity	∠	Yes	The angular displacement of surfaces, axes, or lines from a datum
	Profile of a line	⌒	Yes	The profile of a straight or irregular line positioned by theoretical exact dimensions with respect to datum plane(s)
	Profile of a Surface	⌓	Yes	The profile of a straight or irregular surface positioned by theoretical exact dimensions with respect to datum plane(s)

Geometric Tolerances

Symbols for geometrical characteristics

Type of Tolerance	Characteristics to be toleranced	Symbol	Datum needed	Applications
Location	Position	⌖	*	The deviation of a feature from a true position
	Concentricity and Coaxiality	◎	Yes	The relationship between two circles having a common centre or two cylinders having a common axis
	Symmetry	⌯	Yes	The symmetrical position of a feature related to a datum
	Profile of a line	⌒	Yes	The profile of a straight or irregular line positioned by theoretical exact dimensions with respect to datum plane(s)
	Profile of a Surface	⌓	Yes	The profile of a straight or irregular surface positioned by theoretical exact dimensions with respect to datum plane(s)

* If two or more groups of features are shown on the same axis, they shall be considered to be a single pattern when not related to a datum

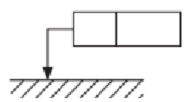
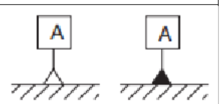
Geometric Tolerances

Symbols for geometrical characteristics

Type of Tolerance	Characteristics to be toleranced	Symbol	Datum needed	Applications
Runout	Circular runout	↗	Yes	The position of a point fixed on a surface of a part which is rotated 360° about its datum axis
	Total runout	⌰	Yes	The relative position of a point when traversed along a surface rotating about its datum axis

Geometric Tolerances

Additional Symbols for geometrical characteristics

Description	Symbols
Toleranced feature indication	
Datum feature indication	
Datum target indication	R4 / A1
Theoretically exact dimension	$\boxed{50}$
Projected tolerance zone	P

Geometric Tolerances

Additional Symbols for geometrical characteristics

Description	Symbols
Maximum material requirement	M
Least material requirement	L
Free state condition (non-rigid parts)	F
All around profile	
Envelope requirement	E
Common zone	**CZ**

Geometric Tolerances

Method of indicating geometrical tolerances on drawings

Geometrical tolerances are indicated by stating the following details in compartments in a rectangular frame.

(a) the characteristic symbol, for single or related features;

(b) the tolerance value

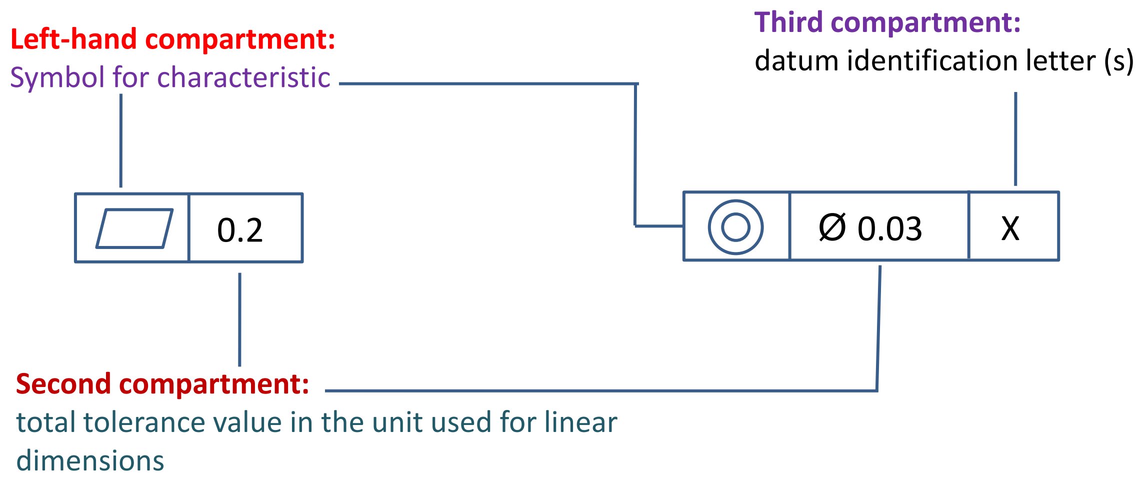
(i) preceded by Ø if the zone is circular or cylindrical,

(ii) preceded by SØ if the zone is spherical;

(c) Letter or letters identifying the datum or datum systems.

Geometric Tolerances

Applications of Geometrical Tolerances



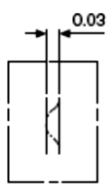
Geometric Tolerances

Straightness

Case 1

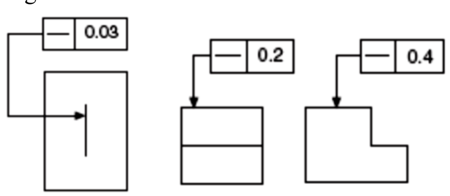
Product requirement

The specified line shown on the surface must lie between two parallel straight lines 0.03 apart.



Drawing instruction

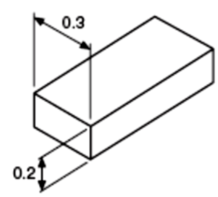
A typical application for this type of tolerance could be a graduation line on an engraved scale.



Case 2

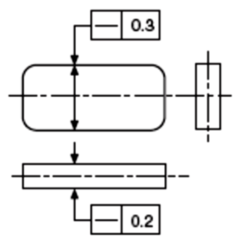
Product requirement

The axis of the whole part must lie in a boxed zone of 0.3 · 0.2 over its length.



Drawing instruction

As indicated, the straightness of the axis is controlled by the dimensions of the box, and could be applied to a long rectangular key.



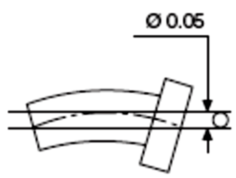
Geometric Tolerances

Straightness

Case 3

Product requirement

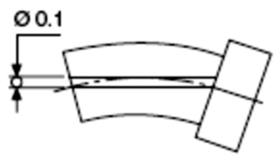
The axis of the whole feature must lie within the cylindrical tolerance zone of 0.05.



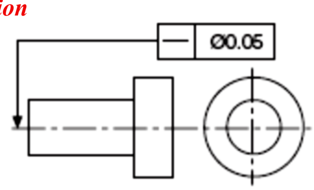
Case 4

Product requirement

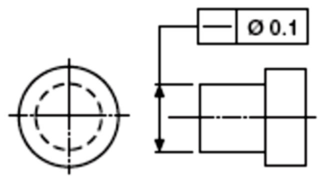
The geometrical tolerance may be required to control only part of the component. In this example the axis of the dimensioned portion of the feature must lie within the cylindrical tolerance zone of 0.1 diameter.



Drawing instruction



Drawing instruction



29

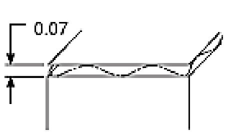
Geometric Tolerances

Flatness

Case 1

Product requirement

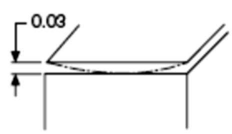
The surface must be contained between two parallel planes 0.07 apart.



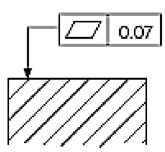
Case 2

Product requirement

The surface must be contained between two parallel planes 0.03 apart, but must not be convex.

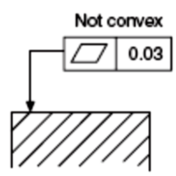


Drawing instruction



Drawing instruction

Note that these instructions could be arranged to avoid a concave condition.



30

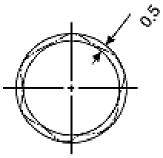
Geometric Tolerances

Circularity (roundness)

Case 1

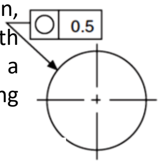
Product requirement

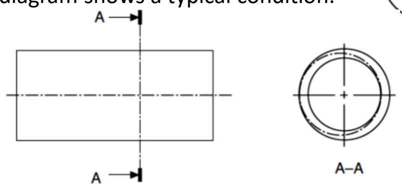
The circumference of the bar must lie between two co-planar concentric circles 0.5 apart.



Drawing instruction

Note that, at any particular section, a circle may not be concentric with its axis but may still satisfy a circularity tolerance. The following diagram shows a typical condition.

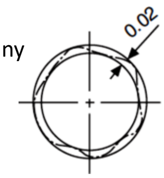




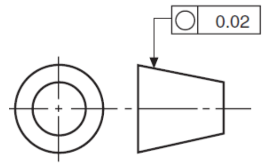
Case 2

Product requirement

The circumference at any cross section must lie between two coplanar concentric circles 0.02 apart.



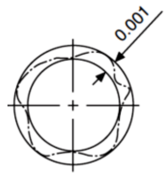
Drawing instruction



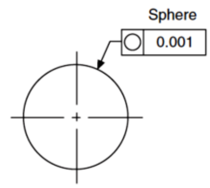
Case 3

Product requirement

the periphery of any section of maximum diameter of the sphere must lie between concentric circles a radial distance 0.001 apart in the plane of the section.



Drawing instruction



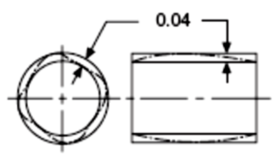
Geometric Tolerances

Cylindricity

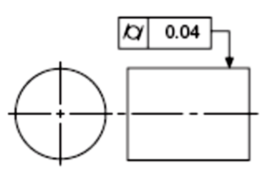
Case 1

Product requirement

The whole curved surface of the feature must lie between an annular tolerance zone 0.04 wide formed by two cylindrical surfaces coaxial with each other.



Drawing instruction



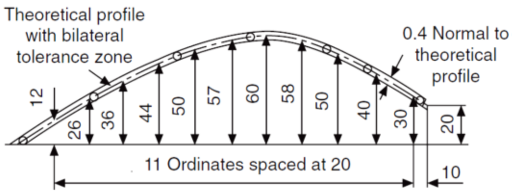
Geometric Tolerances

Profile tolerance of a line

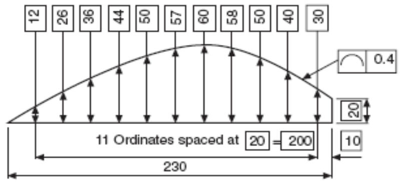
Case 1

Product requirement

The profile is required to be contained within the bilateral tolerance zone.



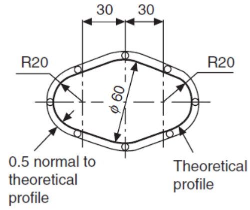
Drawing instruction



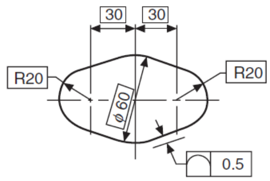
Case 2

Product requirement

The profile is required to be contained within the unilateral tolerance zone.



Drawing instruction



Geometric Tolerances

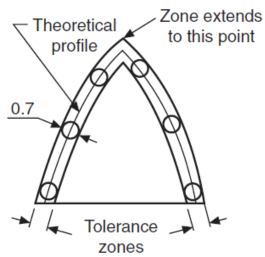
Profile tolerance of a line

Case of tolerance profile of a feature having a sharp corner.

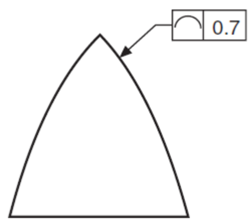
As shown in figure, the inner tolerance zone is considered to end at the intersection of the inner boundary lines, and the outer tolerance zone is considered to extend to the outer boundary-line intersections. Sharp corners such as these could allow considerable rounding; if this is desirable, then the design requirement must be clearly defined on the drawing by specifying a radius or adding a general note such as 'ALL CORNERS 0.5 MAX'. It should be noted that such radii apply regardless of the profile tolerance.

Product requirement

In the example given, the product is required to have a sharp corner.



Drawing instruction

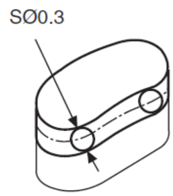


Geometric Tolerances

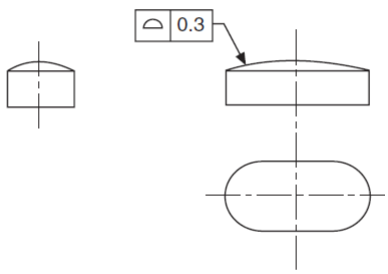
Profile tolerance of a surface

Product requirement

The tolerance zone is to be contained by upper and lower surfaces which touch the circumference of spheres 0.3 diameter whose centres lie on the theoretical form of the surface.



Drawing instruction



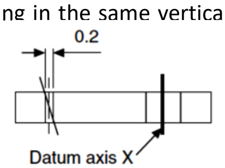
Geometric Tolerances

Parallelism

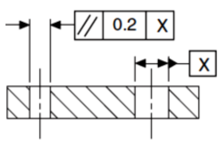
Case 1

Product requirement

The axis of the hole on the left-hand side must be contained between two straight lines 0.2 apart, parallel to the datum axis X and lying in the same vertical plane.



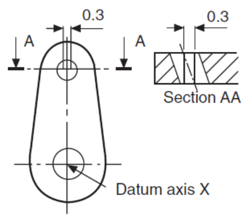
Drawing instruction



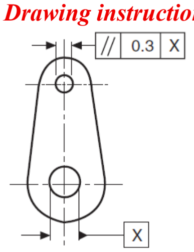
Case 2

Product requirement

The axis of the upper hole must be contained between two straight lines 0.3 apart which are parallel to and symmetrically disposed about the datum axis X and lie in the same horizontal plane.



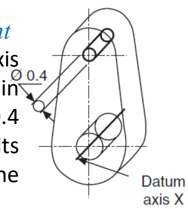
Drawing instruction



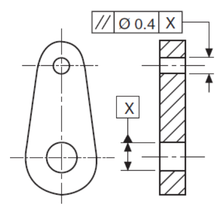
Case 3

Product requirement

The upper hole axis must be contained in a cylindrical zone 0.4 diameter, with its axis parallel to the datum axis X.



Drawing instruction



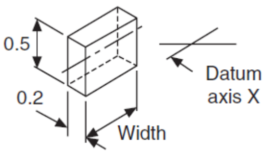
Geometric Tolerances

Parallelism

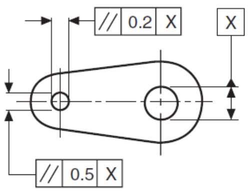
Case 4

Product requirement

The axis of the hole on the left-hand side must be contained in a tolerance box 0.5 × 0.2 × width, as shown, with its sides parallel to the datum axis X and in the same horizontal plane.



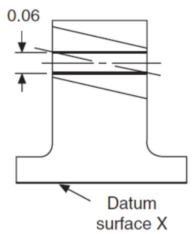
Drawing instruction



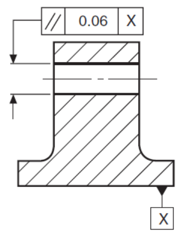
Case 5

Product requirement

The axis of the hole must be contained between two planes 0.06 apart parallel to the datum surface X.



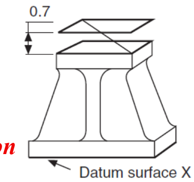
Drawing instruction



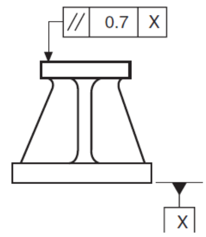
Case 6

Product requirement

The top surface of the component must be contained between two planes 0.7 apart and parallel to the datum surface X.



Drawing instruction



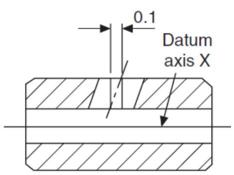
Geometric Tolerances

Perpendicularity (Squareness)

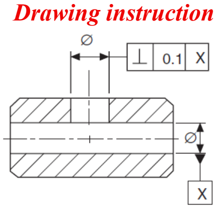
Case 1

Product requirement

The axis of the vertical hole must be contained between two planes 0.1 apart which are perpendicular to the datum axis.



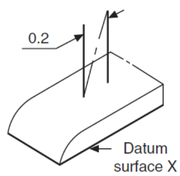
Drawing instruction



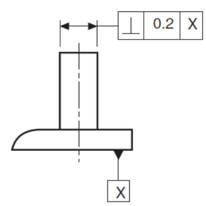
Case 2

Product requirement

The axis of the upright must be contained between two straight lines 0.2 apart which are perpendicular to the datum. Squareness is controlled here in one plane only.



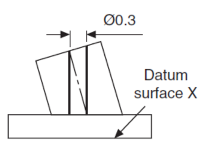
Drawing instruction



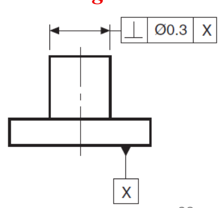
Case 3

Product requirement

The axis of the column must be contained in a cylindrical tolerance zone 0.3 diameter, the axis of which is perpendicular to the datum surface X. Squareness is controlled in more than one plane by this method.



Drawing instruction



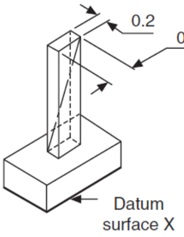
Geometric Tolerances

Perpendicularity (Squareness)

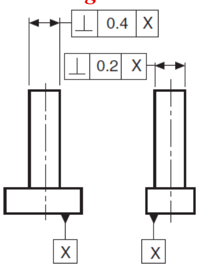
Case 4

Product requirement

The axis of the column must be contained in a tolerance- zone box 0.2 × 0.4 which is perpendicular to the datum surface X.



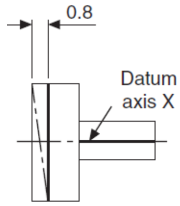
Drawing instruction



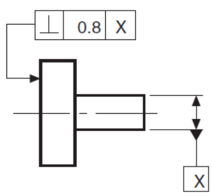
Case 5

Product requirement

The left-hand end face of the part must be contained between two parallel planes 0.8 apart and perpendicular to the datum axis X.



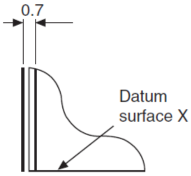
Drawing instruction



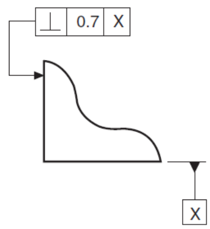
Case 6

Product requirement

The left-hand surface must be contained between two parallel planes 0.7 apart and perpendicular to the datum surface X.



Drawing instruction



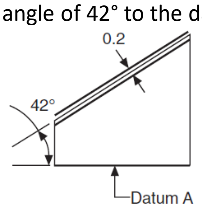
Geometric Tolerances

Angularity

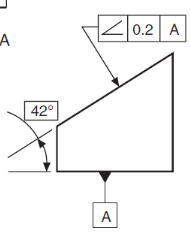
Case 1

Product requirement

The inclined surface must be contained within two parallel planes 0.2 apart which are at an angle of 42° to the datum surface.



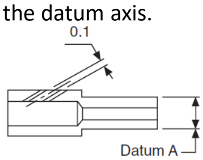
Drawing instruction



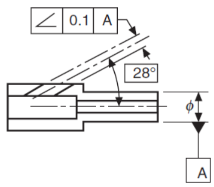
Case 2

Product requirement

The axis of the hole must be contained within two parallel straight lines 0.1 apart inclined at 28° to the datum axis.



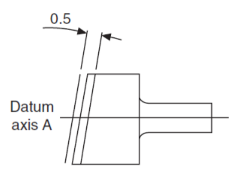
Drawing instruction



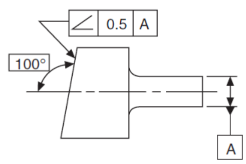
Case 3

Product requirement

The inclined surface must be contained within two parallel planes 0.5 apart which are inclined at 100° to the datum axis.



Drawing instruction



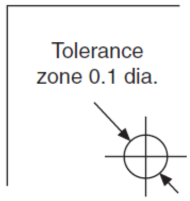
Geometric Tolerances

Position

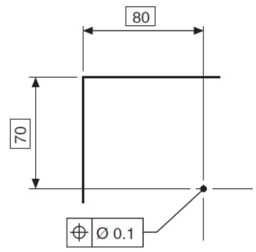
Case 1

Product requirement

The point must be contained within a circle of 0.1 diameter in the plane of the surface. The circle has its centre at the intersection of the two theoretically exact dimensions. If the point were to be located by three dimensions, the tolerance zone would be a sphere.



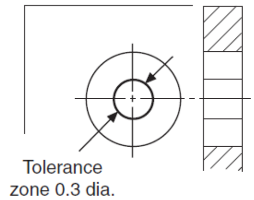
Drawing instruction



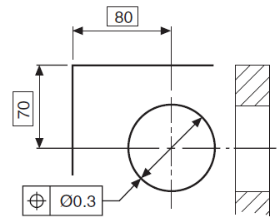
Case 2

Product requirement

The axis of the hole must be contained in a cylindrical tolerance zone of 0.3 diameter with its axis coincident with the theoretically exact position of the hole axis.



Drawing instruction



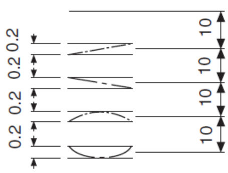
Geometric Tolerances

Position

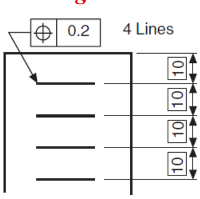
Case 3

Product requirement

Each line must be contained between two parallel straight lines on the surface, 0.2 apart, which are symmetrical with the theoretically exact positions of the required lines.



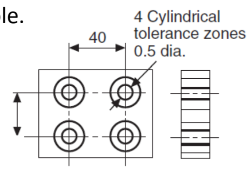
Drawing instruction



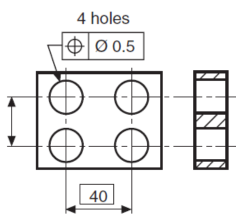
Case 4

Product requirement

The axes of each of the four holes must be contained in a cylindrical tolerance zone of 0.5 diameter, with its own axis coincident with the theoretically exact position of each hole.



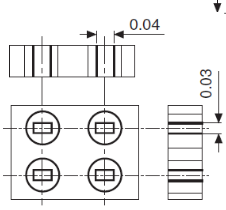
Drawing instruction



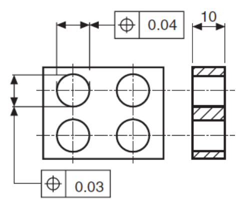
Case 5

Product requirement

The axes of each of the four holes must be contained in a boxed zone of 0.04 × 0.03 × 10, symmetrically disposed about the theoretically exact position of each hole axis.



Drawing instruction



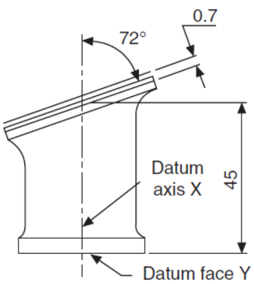
Geometric Tolerances

Position

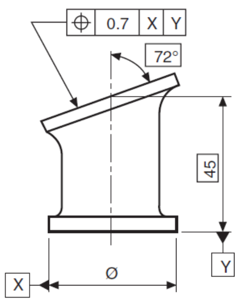
Case 6

Product requirement

The angled surface must be contained between two parallel planes 0.7 apart, which are symmetrically disposed about the theoretically exact position of the surface relative to datum axis X and datum face Y.



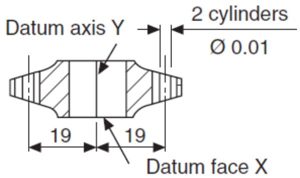
Drawing instruction



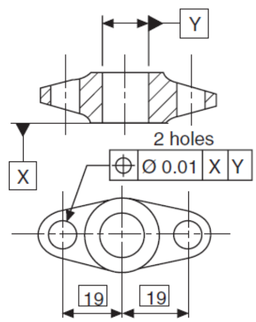
Case 7

Product requirement

The axes of the two holes must be contained in cylindrical tolerance zones of 0.01 diameter, with their own axes coincident with the theoretically exact hole positions related to datum face X and the datum centre-line axis Y.



Drawing instruction



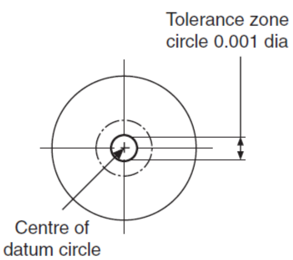
Geometric Tolerances

Concentricity and Coaxiality

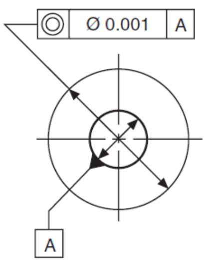
Case 1 (Concentricity)

Product requirement

To contain the centre of the large circle within a circular tolerance zone of 0.001 diameter which has its centre coincident with the datum circle centre.



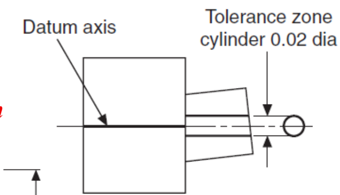
Drawing instruction



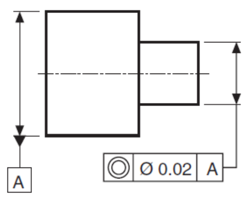
Case 2 (Coaxiality)

Product requirement

To contain the axis of the right-hand cylinder within a cylindrical tolerance zone which is coaxial with the axis of the datum cylinder.



Drawing instruction



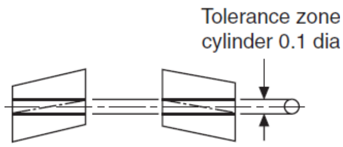
Geometric Tolerances

Concentricity and Coaxiality

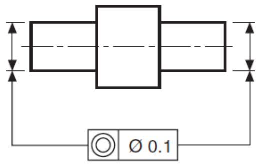
Case 3 (Coaxiality)

Product requirement

To contain the axes of both the left- and right-hand cylinders within a cylindrical tolerance zone.



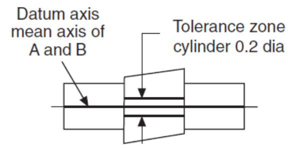
Drawing instruction



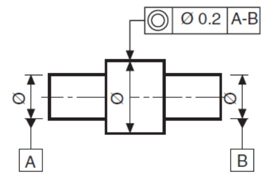
Case 4 (Coaxiality)

Product requirement

To contain the axis of the central cylinder within a cylindrical tolerance zone which is coaxial with the mean axes of the left- and right-hand cylinders.



Drawing instruction



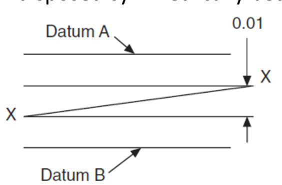
Geometric Tolerances

Symmetry

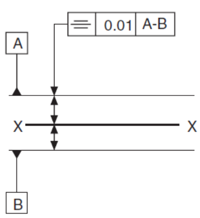
Case 1

Product requirement

The specified line XX must lie in a tolerance zone formed by two parallel straight lines 0.01 apart and disposed symmetrically between datums A and B.



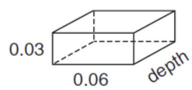
Drawing instruction



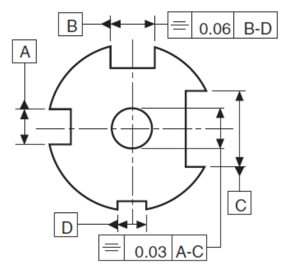
Case 2

Product requirement

The axis of a hole in a plate must lie in a rectangular-box tolerance zone 0.03 × 0.06 × depth of the plate, parallel with and symmetrically disposed about the common median planes formed by slots AC and BD.



Drawing instruction



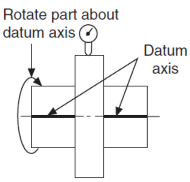
Geometric Tolerances

Circular run-out

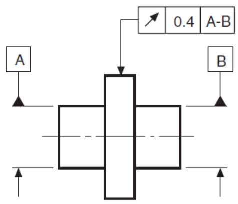
Case 1

Product requirement

The circular radial run-out must not exceed 0.4 at any point along the cylinder, measured perpendicular to the datum axis without axial movement.



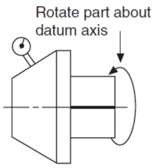
Drawing instruction



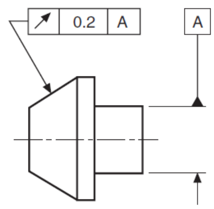
Case 2

Product requirement

Circular run-out must not exceed 0.2 measured at any point normal to the surface, without axial movement.



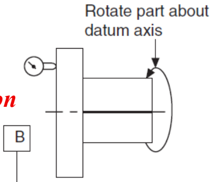
Drawing instruction



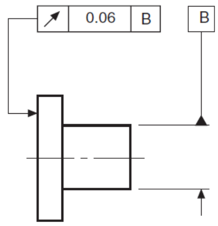
Case 3

Product requirement

At any radius, the circular run-out must not exceed 0.06 measured parallel to the datum axis.



Drawing instruction



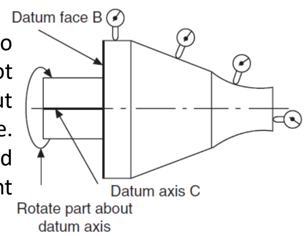
Geometric Tolerances

Circular run-out

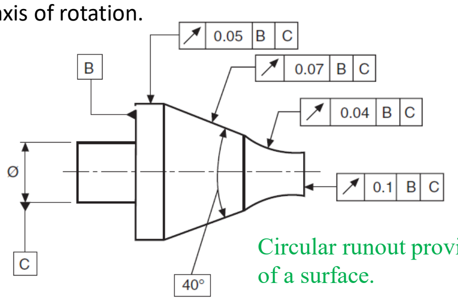
Case 4

Product requirement

The component is required to be rotated about datum axis C, with datum face B set to ensure no axial movement. The circular radial runout on the cylindrical portion must not exceed 0.05 at any point measured perpendicular to the datum axis. The circular runout on the tapered portion must not exceed 0.07 at any point measured normal to its surface. The circular runout on the curved portion must not exceed 0.04 at any point measured normal to its surface. The axial runout of the end face must not exceed 0.1 at any point measured parallel to the datum axis of rotation.



Drawing instruction



Circular runout provides composite control of circular elements of a surface.

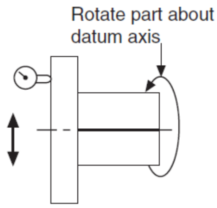
Geometric Tolerances

Total run-out

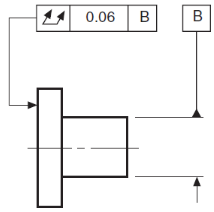
Case 1

Product requirement

The circular radial run-out must not exceed 0.4 at any point along the cylinder, measured perpendicular to the datum axis without axial movement.



Drawing instruction



Note. The symbol means that the measuring instrument is guided across a theoretically exact surface true to the datum axis.

Total runout provides composite control of all surface elements. The complete surface is measured, and not single points, as in circular runout.

Total runout controls cumulative variations of perpendicularity which can detect wobble, also flatness which can detect concavity and convexity.

References

ABDULLA SHARIF, Design of Machine Elements, Dhanpat Rai Publications (P) Ltd, New Delhi, 1995.

V. B. Bhandari, Design of Machine Elements, Third Ed., The McGraw-Hills Companies, New Delhi

R. S. KHURMI and J.K.GUPTA, A Text Book of Machine Design, S.Chand and company ltd., New Delhi, 2000.

http://www.nptel.iitm.ac.in

Colin H Simmons and Dennis E Maguire, Manual of Engineering Drawing, Second edition, Elsevier Newnes. Linacre House, Jordan Hill, Oxford OX2 8DP 200 Wheeler Road, Burlington MA 01803