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:

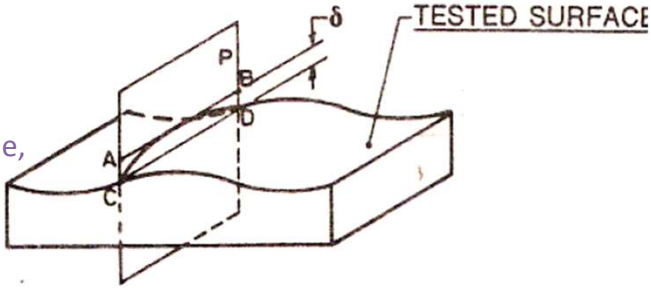
It is determined as the distance between two parallel lines in plane *P* right angle with the tested surface.

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

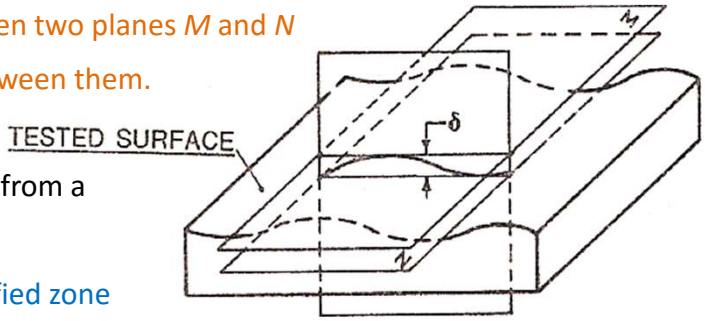
Geometric Tolerances

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.



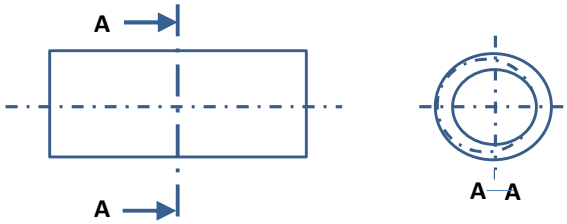
(ii) FLATNESS

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

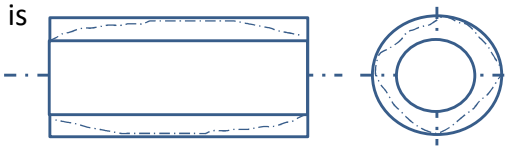
Cylindricity

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,

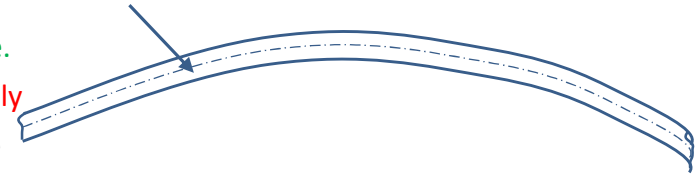
it is recommended that each of these characteristics is toleranced and inspected separately.



Geometric Tolerances

Profile tolerance of a line

Theoritical Profile with bilateral tolerance



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

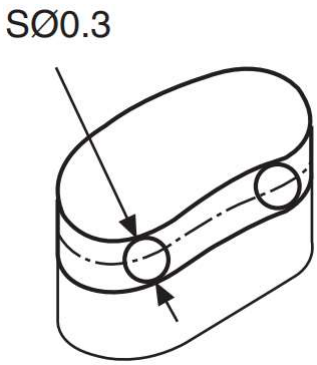
The contour is defined by theoretically exact boxed dimensions and must be accompanied by a relative tolerance zone.

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

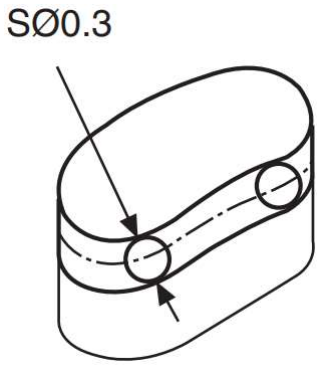


Profile tolerance of a surface is used to control the ideal form of a surface, which is defined by theoretically exact boxed dimensions and must be accompanied by a relative tolerance zone.

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



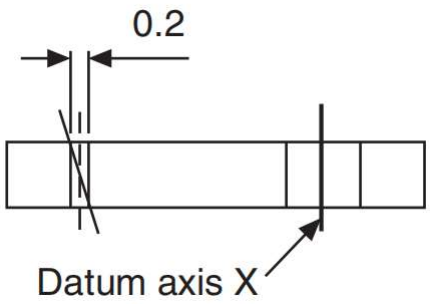
The tolerance value is equal to the diameter of spheres whose centre lines lie on the true form of the surface.

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

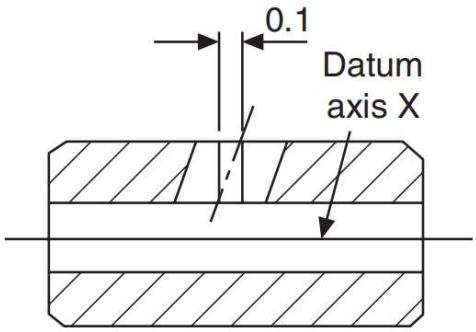
Perpendicularity (Squareness)

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

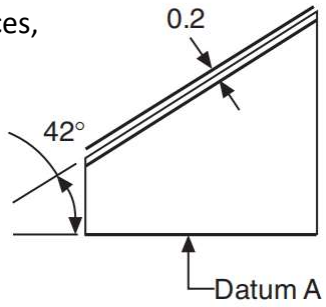
Angularity

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.

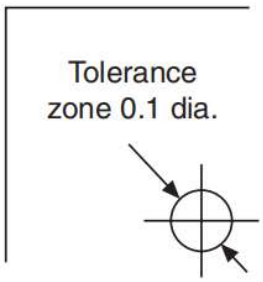
Geometric Tolerances

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.

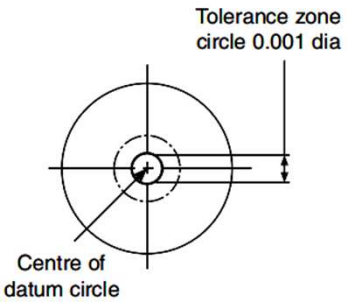
Geometric Tolerances

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.



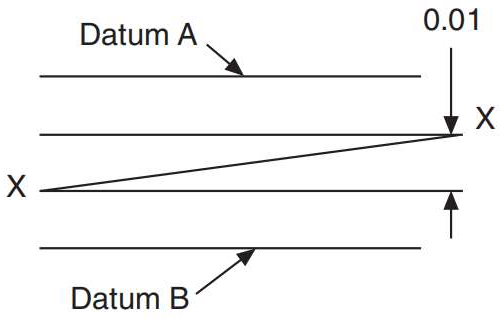
Geometric Tolerances

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.

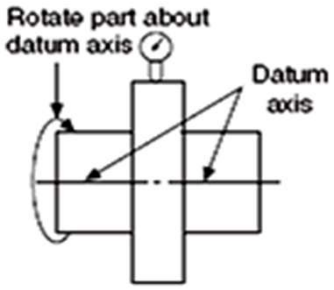
Geometric Tolerances

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.

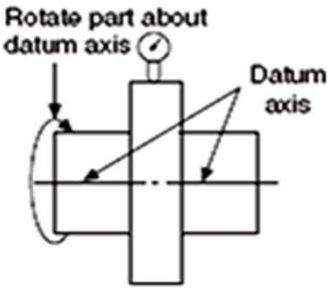
The results of this test may include errors of other characteristics such as circularity, concentricity, perpendicularity, or flatness, but the test cannot discriminate between them.



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				

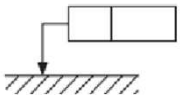
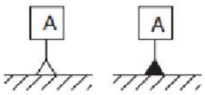
Geometric Tolerances

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	
Theoretically exact dimension	50
Projected tolerance zone	

Geometric Tolerances

Additional Symbols for geometrical characteristics

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

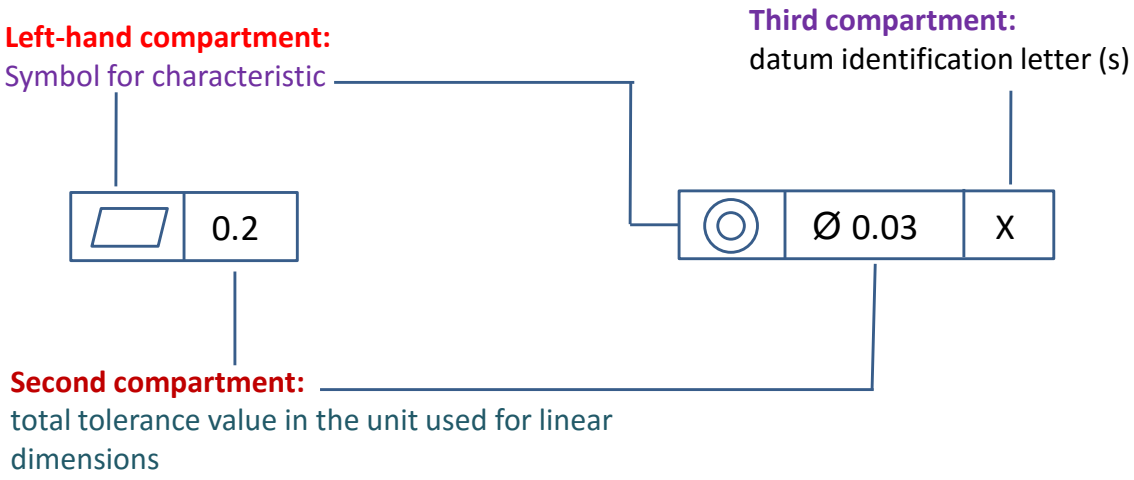
(i) preceded by Ø if the zone is circular or cylindrical,

(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



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

N. D. Bhatt and V. M. Panchal, Machine Drawing, 26th Edition, Charotar Publishing House, Anand, 1995.