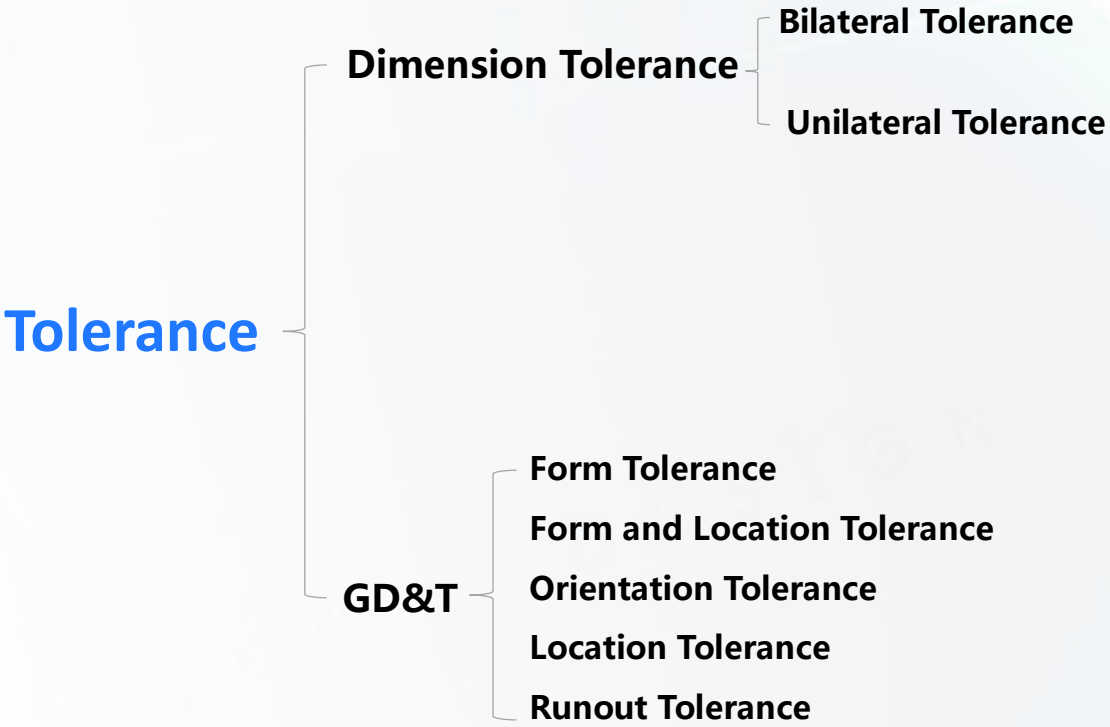
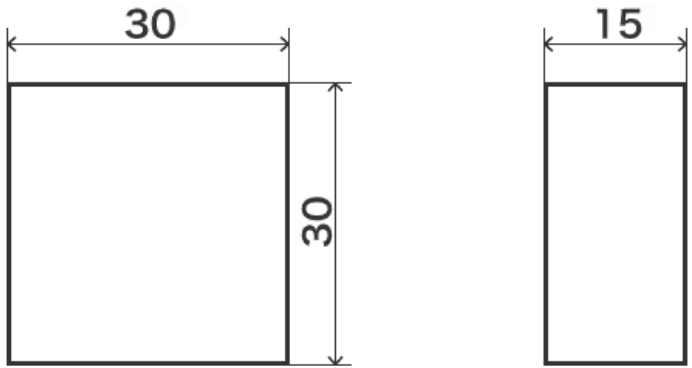
Types of Tolerance

www.sogaworks.com



Dimensional Tolerance

Dimensional tolerance is the tolerance applied to the dimensions marked in the drawing, dimensional objects such as length, distance, position, angle, size, aperture, fillet and chamfer, etc. It is used to indicate tolerances different from general tolerances. Unlike general tolerances, dimensional tolerances have no clear standards and can be arbitrarily specified according to the designer's intention, but the range of achievable tolerances is limited depending on the processing method, etc. Dimensional tolerances include 2 types, bilateral tolerances and unilateral tolerances.

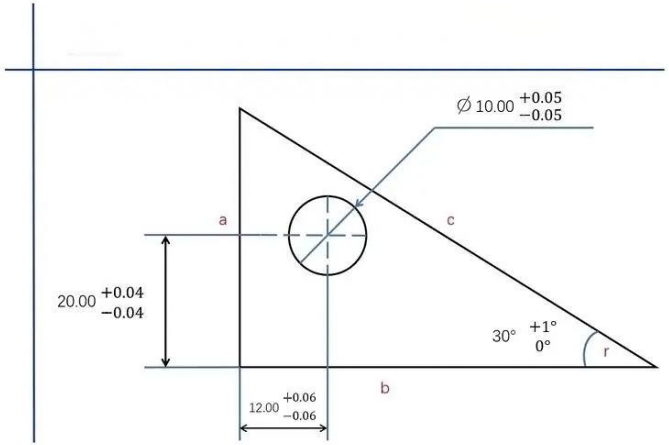


Bilateral Tolerance

Bilateral tolerance refers to the allowable variation of a dimension that exists within a specified range on either side of the reference dimension. In other words, the dimension may vary in both the upper and lower directions relative to the reference dimension.

Example

If the basic size of the hole is 10mm and the bilateral tolerance is ±0.05mm, then the actual dimension range of the shaft is 9.95mm to 10.05mm.

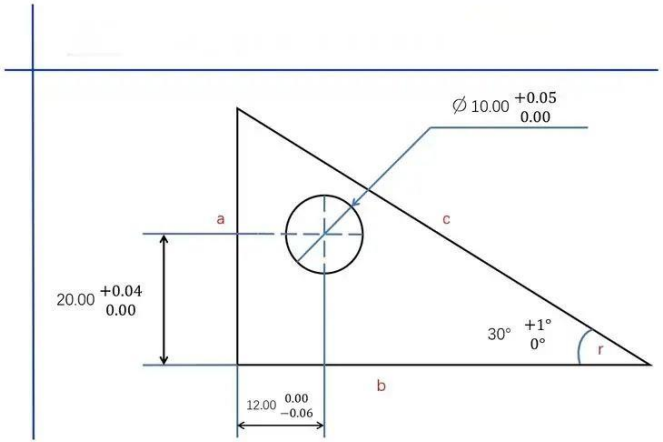


Unilateral Tolerance

Unilateral tolerance, however, refers to the allowed variation of a dimension to be on only one side of the basic dimension; that is, the acceptable tolerance range is limited to one direction.

Example

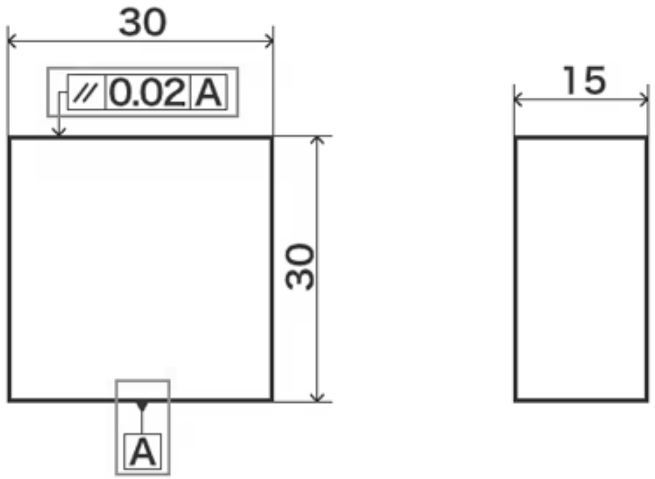
If the basic size of a hole is 10 mm and the unilateral tolerance is +0.05mm, then the actual size range of the hole is from 10.00 mm to 10.05mm.



GD&T

GD&T, short for Geometric Dimensioning and Tolerancing, is a system for defining and communicating design intent and engineering tolerances that helps engineers and manufacturers optimally control variations in manufacturing processes.

Geometric Dimensioning and Tolerancing can be divided into four categories: form tolerance, orientation tolerance, location tolerance, and runout tolerance, which in total consists of 13 types.



Overview

Type	Characteristics	Symbol
Form	Linearity	
	Flatness	
	Roundness	
	Cylindricity	
Form and Location	Profile of Line	
	Profile of Plane	
Orientation	Parallelism	
	Perpendicularity	
	Angularity	
Location	Position	
	Coaxiality	
	Symmetry	
Runout	Circular Runout	
	Total Runout	

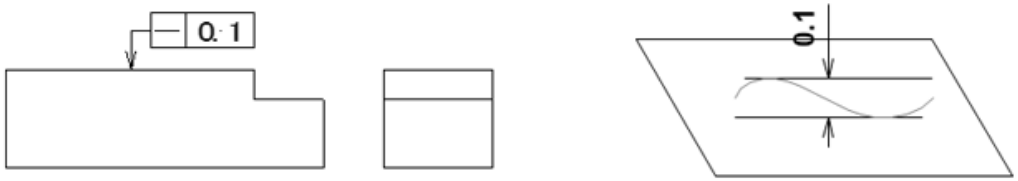
Form Tolerance

Linearity

Linearity is the allowable deviation from a straight line over a specified length or surface. It is used to define how much a feature of a part can vary from being perfectly linear.

Example

In a given plane, the line segments to be inspected shall lie between two parallel lines at a distance of 0.1mm.



Flatness

Flatness is a geometrical condition that defines the deviation of a surface from an ideal plane. It provides a metric of how much the surface deviates from ideal flatness, and thus it represents the homogeneity of a surface over its whole area.

Example

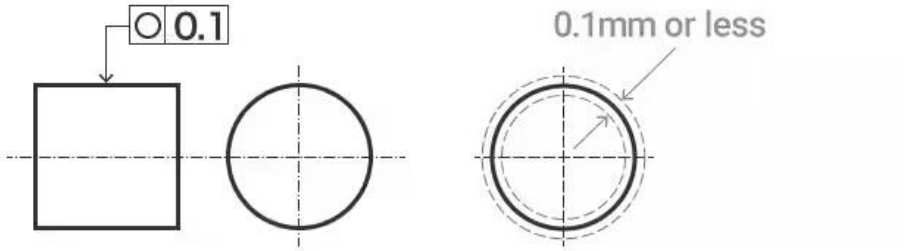
This surface shall be between two parallel planes separated by only 0.3 mm.

Roundness

Roundness, also commonly called circularity, is the geometric condition that defines the extent to which the form of a feature, such as a cylinder, hole, or sphere, departs from a perfect circle in any given cross-section.

Example

The outer circumference of any cross-section of a shaft cut perpendicularly shall fall between two concentric circles just 0.1mm apart on the same plane.

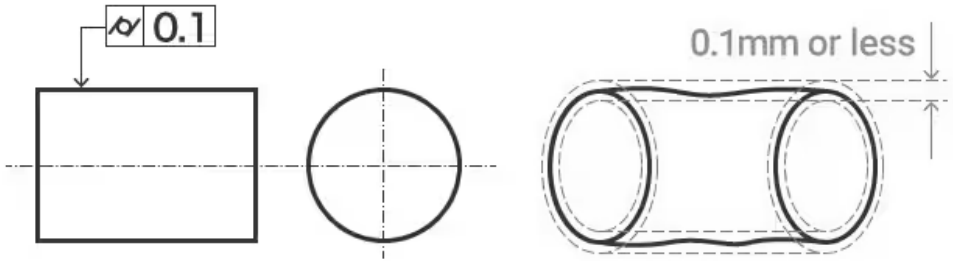


Cylindricity

Cylindricity is a geometrical condition that measures the extent to which the form of a cylindrical feature conforms to that of an ideal cylinder. It measures the uniformity of the surface both along the length and around the circumference of the cylinder.

Example

The target plane has to be in between two coaxial cylinders only 0.1 mm apart.



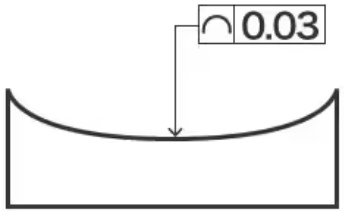
Form and Location Tolerance

Profile of Line

The condition required for retaining the perfect form of a curve of any shape on a prescribed plane of a part. Profile tolerance of a line The allowable deviation of the actual contour line of a non-circular curve.

Example

The projected profile on any cross-section parallel to the projection plane shall lie between the two envelopes created by a circle of diameter 0.03 mm, centered on the line that has a theoretically exact profile.

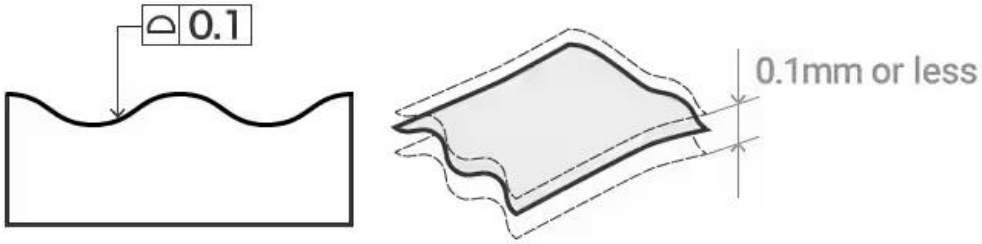


Profile of Plane

Profile of a plane is the condition of maintaining the ideal shape of any curved surface on a particular part. The profile tolerance of a plane is the permissible variation of the actual contour line of a non-circular curved surface from the ideal contour surface.

Example

The destination plane should lie between two envelope planes created by a sphere with a diameter of 0.1 mm, whose center is on the plane having a theoretically perfect profile.



Orientation Tolerance

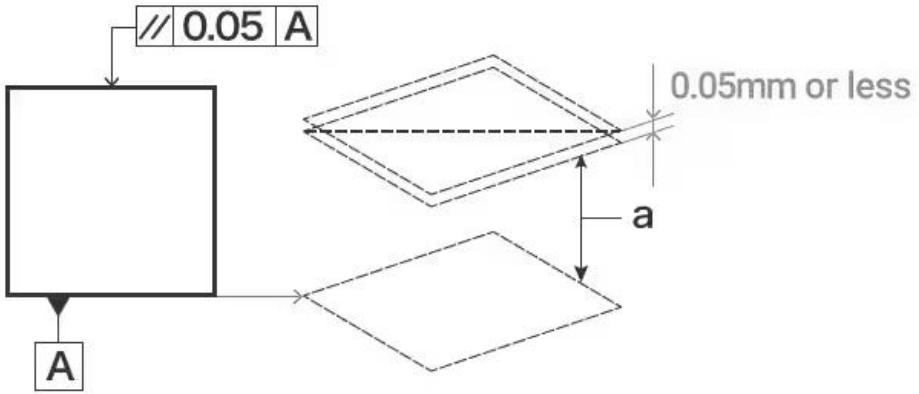
Parallelism

//

Parallelism is the acceptable variation (deviation) of a feature (e.g., surface, axis or line) with regards to being parallel from a designated reference (e.g. a datum plane, axis, or line). Whilst, it looks like flatness has been discussed again, parallelism involves datum(reference plane or line).

Example

The plane identified by the indication arrow must be parallel to datum plane A and lie between two planes that are only 0.05 mm apart in the direction of the indication arrows.



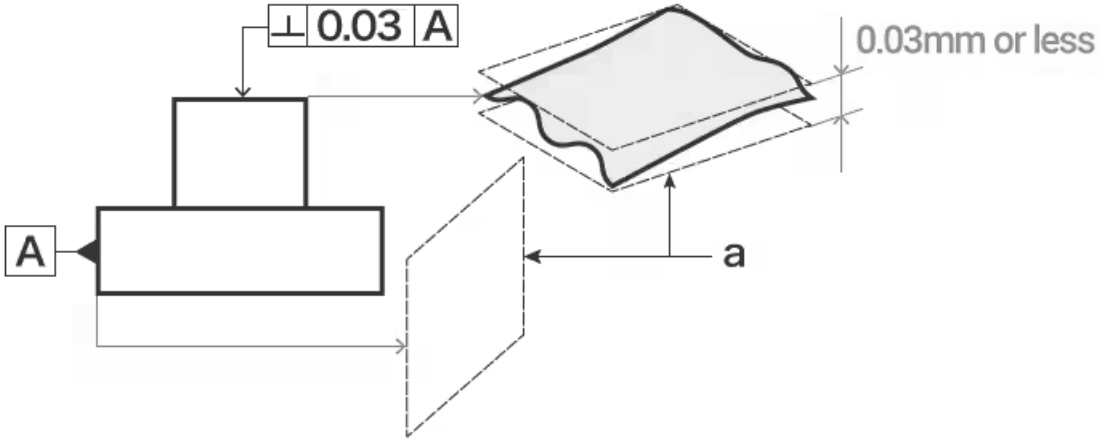
SOGAWORKS
Accelerate Your Innovations

Perpendicularity

Perpendicularity is a geometric condition that assesses the degree to which a feature, such as a surface, axis, or line, aligns at a right angle (90°) to a reference feature, which may include a plane or axis.

Example

The plane represented by the indicating arrow shall be located between two parallel planes that are perpendicular to datum plane A, with a diameter of 0.03 mm.

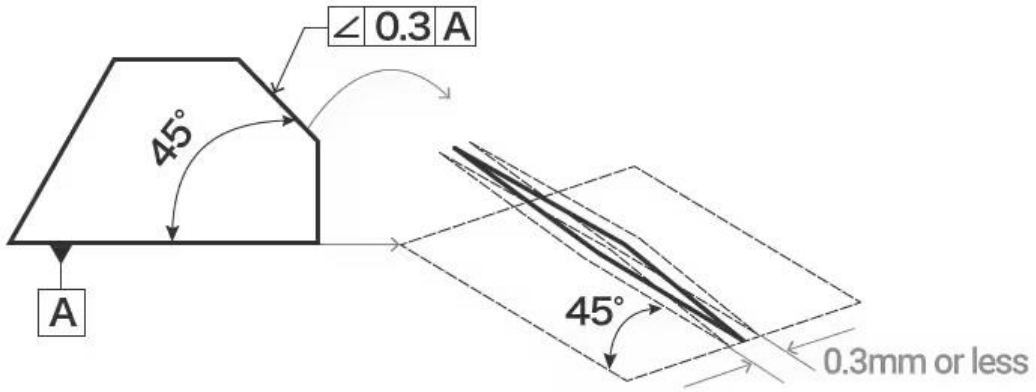


Angularity

Angularity measures the amount that a feature, such as a surface, line, or axis, is oriented at an assigned angle, other than 90° (perpendicularity) or 0° (parallelism), with respect to a reference datum.

Example

The plane indicated by the indication arrow shall be theoretically exactly angled by 45 degrees to the datum plane A and between two parallel planes only 0.3 mm apart in the direction of the indication arrows.



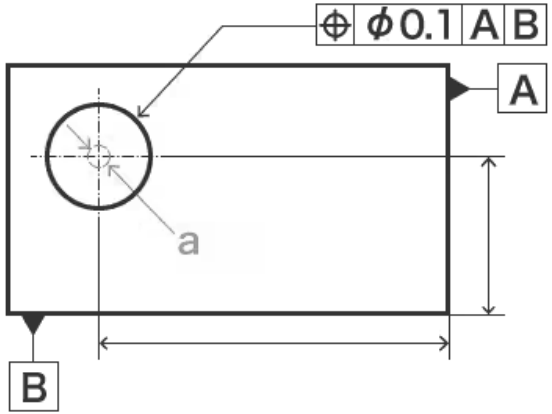
Location Tolerance

Position

Position is used to find the exact location of a component's point, line, and surface relative to a reference.

Example

The center of the circle shown by the indication arrow shall be within a circle having a diameter of 0.1 mm.

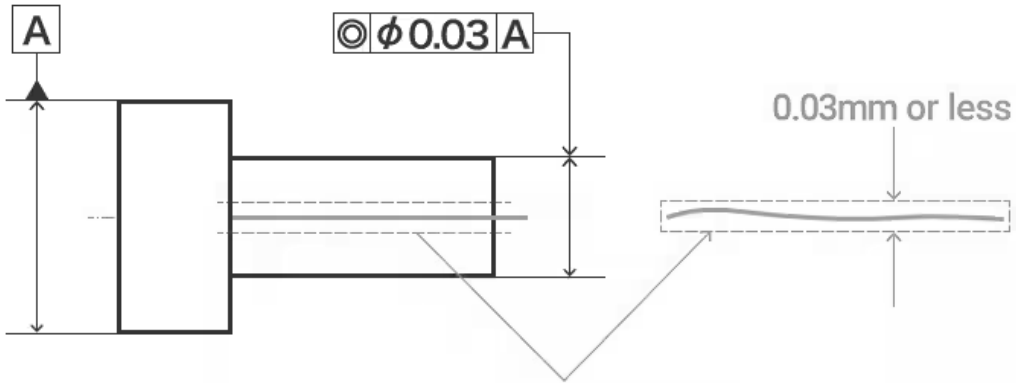


Coaxiality

Coaxiality ensures that the axis of a cylindrical feature, such as a shaft, hole, or tube, coincides exactly with the axis of a reference datum.

Example

The axis of the given cylinder shall lie within a cylinder that uses datum axis line A as its axis and has a diameter of 0.03 mm.

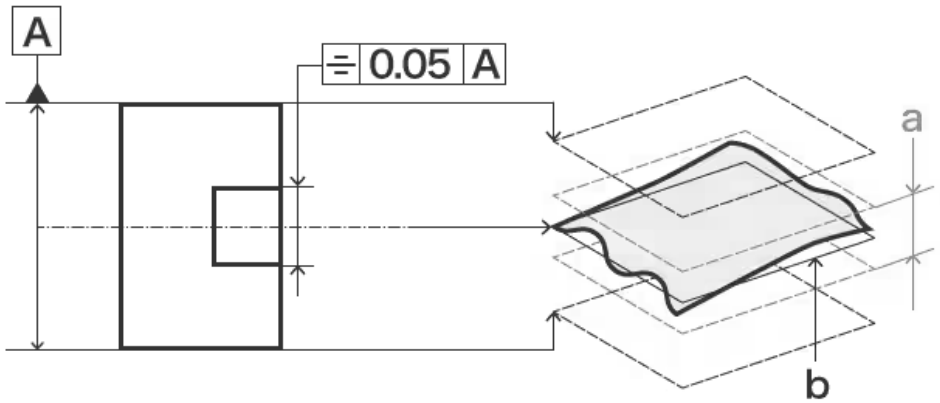


Symmetry

Symmetry measures the evenness with which a feature, or set of features, is distributed about a central reference axis, plane, or point.

Example

The center plane marked shall be between two parallel planes symmetric to datum center plane A and separated from each other by 0.05 mm.



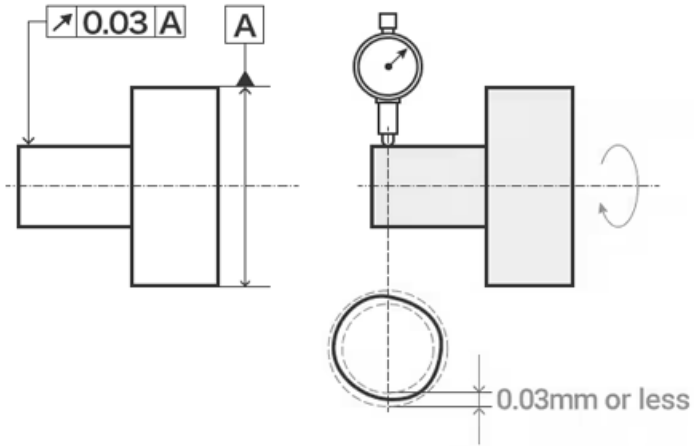
Runout Tolerance

Circular Runout

Circular runout specifies the runout of any part of a circumference when a part is rotated. To meet the circular run-out requirement, the run-out of the measured value when the part is rotated must be within the specified range.

Example

The radial runout of the cylindrical surface, as indicated by the arrow, must not exceed 0.03 mm on any measurement plane perpendicular to the datum axis when the part is rotated once around the datum axis.

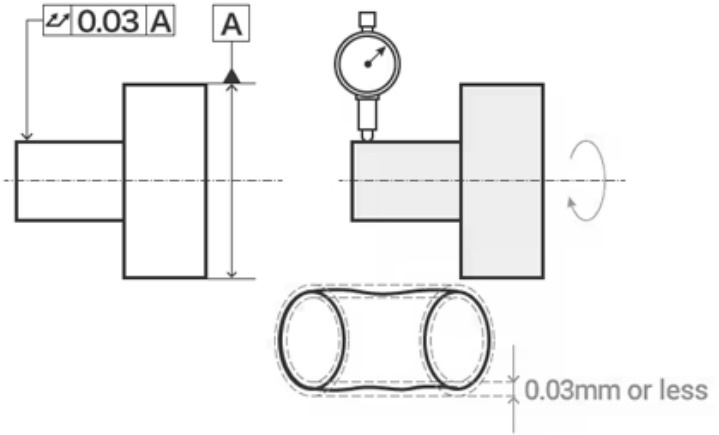


Total Runout

Total runout specifies the runout of the entire surface of a part when it is rotated.
To meet the total run-out requirement, the run-out of the measured value of the entire cylinder surface must be within the specified range.

Example

As indicated by the arrow, the total radial runout of the cylindrical surface must not exceed 0.03 mm at any point along the surface when the cylinder is rotated around the datum axis.



THANKS