1

CHAPTER ONE – Part 2

Geometrical Tolerances

Main references:
http://www.gdandtbasics.com/gdt-symbols/
http://www.engineeringessentials.com/gdt/

CONTENTS

Dimension

A **dimension** is "a numerical value expressed in appropriate units of measure and indicated on a drawing and in other documents along with lines, symbols, and notes to define the size or geometric characteristic, or both, of a part or part feature"

Dimensions on part drawings represent nominal or basic sizes of the part and its features.

The dimension indicates the part size desired by the designer, if the part could be made with no errors or variations in the manufacturing process

Tolerances

A ***tolerance*** is "the total amount by which a specific dimension is permitted to vary. The tolerance is the difference between the maximum and minimum limits"

Variations occur in any manufacturing process, which are manifested as variations in part size

Tolerances are used to define the limits of the allowed variation

Types of Tolerances

A ***Dimensional tolerance*** is the total amount a specific dimension is permitted to vary, which is the difference between maximum and minimum permitted limits of size.

A **Geometric tolerance** is the maximum or minimum variation from true geometric form or position that may be permitted in manufacture.

Geometric tolerance should be employed only for those requirements of a part critical to its functioning or interchangeability. These are much more difficult to measure/verify as compared to dimensional tolerances.

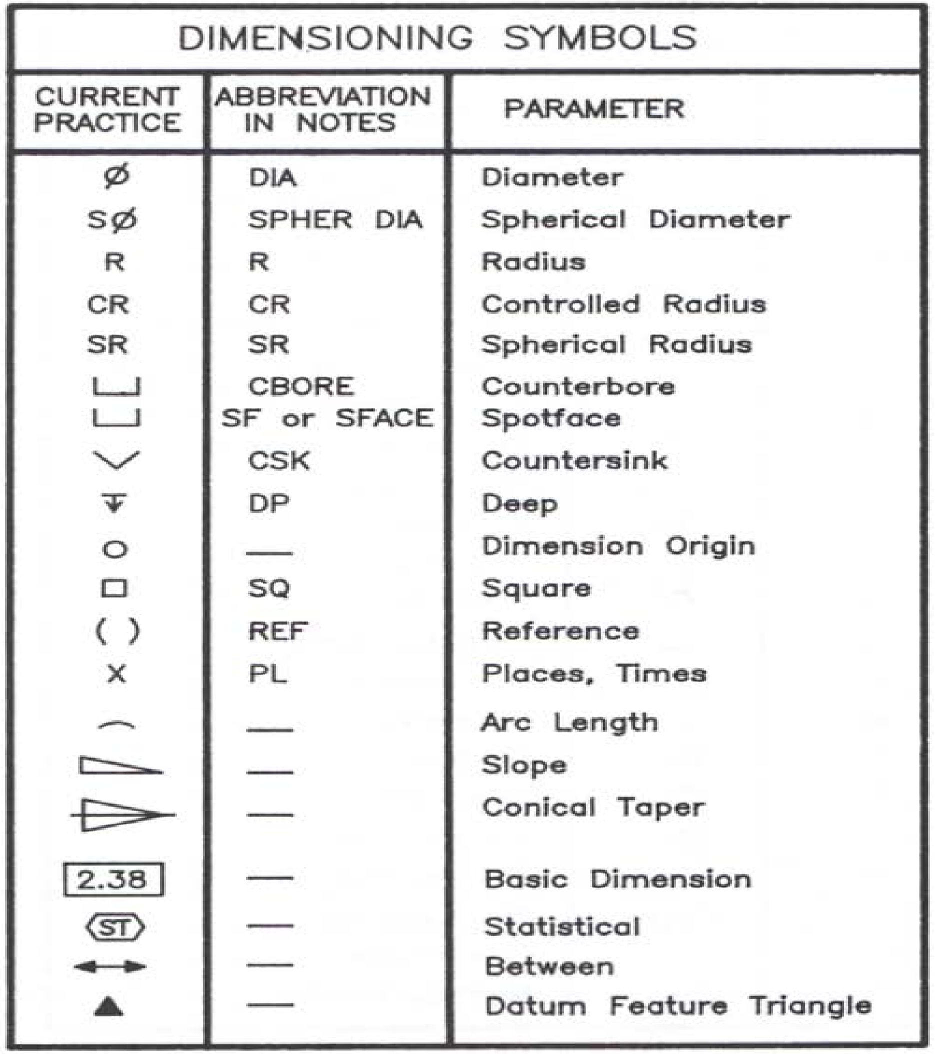
Overview of Geometric Tolerances

Geometric tolerances define the shape of a feature as opposed to its size.

We will focus on three basic types of geometric tolerances:

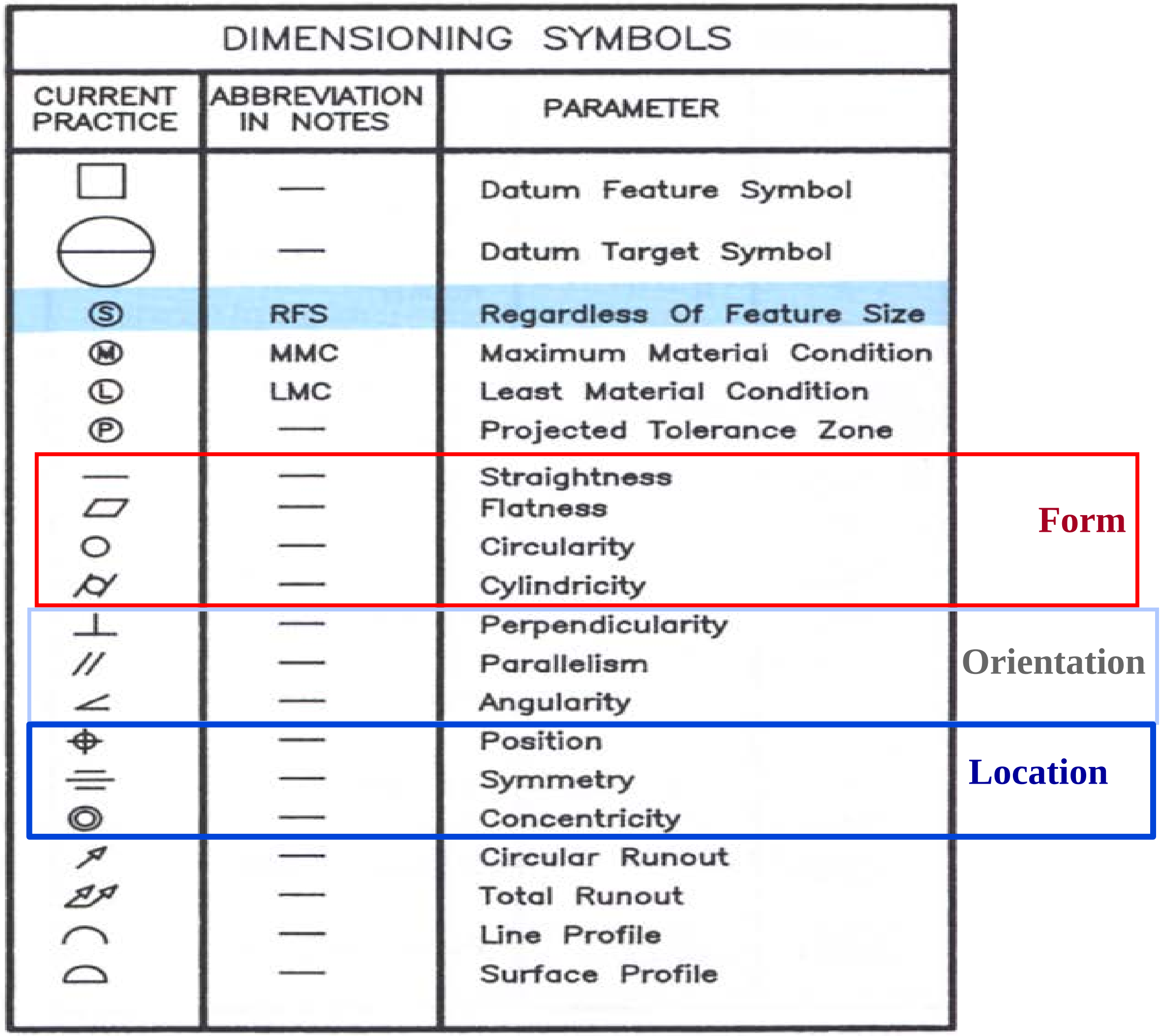
1. **Form tolerances:** straightness, circularity, flatness, cylindricity;
2. **Orientation tolerances;** perpendicularity, parallelism, angularity;
3. **Location tolerances:** position, symmetry, concentricity.

Symbols for Geometric Tolerances



DIMENSIONING SYMBOLS

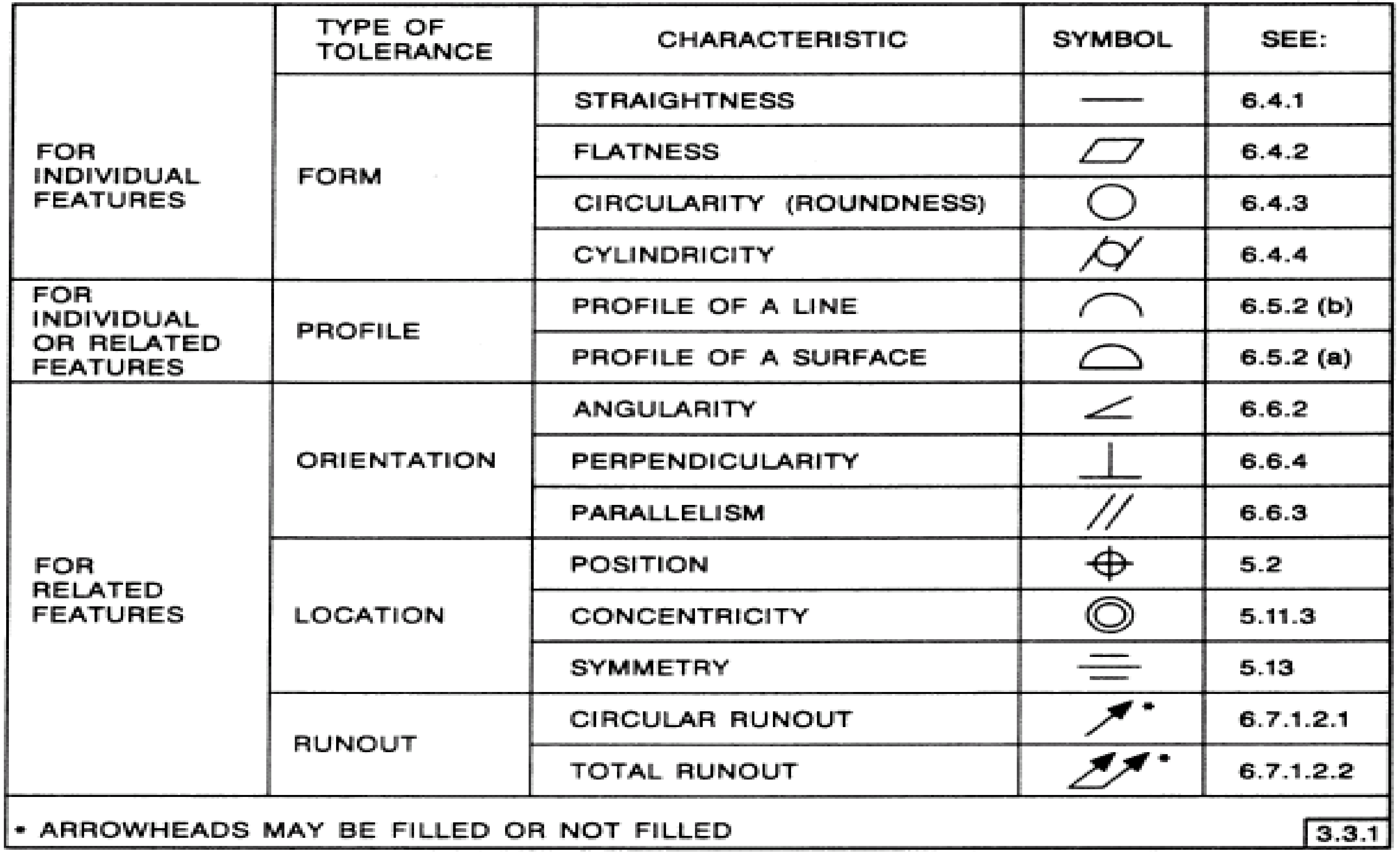
CURRENT PRACTICE	ABBREVIATION IN NOTES	PARAMETER
Ø	DIA	Diameter
SØ	SPHER DIA	Spherical Diameter
R	R	Radius
CR	CR	Controlled Radius
SR	SR	Spherical Radius
⌴	CBORE	Counterbore
⌴	SF or SFACE	Spotface
⌵	CSK	Countersink
↧	DP	Deep
○	—	Dimension Origin
□	SQ	Square
( )	REF	Reference
X	PL	Places, Times
⌒	—	Arc Length
◺	—	Slope
⌲	—	Conical Taper
2.38	—	Basic Dimension
⟨ST⟩	—	Statistical
←→	—	Between
▲	—	Datum Feature Triangle



DIMENSIONING SYMBOLS

CURRENT PRACTICE	ABBREVIATION IN NOTES	PARAMETER	
□	—	Datum Feature Symbol	
⊖	—	Datum Target Symbol	
Ⓢ	RFS	Regardless Of Feature Size	
Ⓜ	MMC	Maximum Material Condition	
Ⓛ	LMC	Least Material Condition	
Ⓟ	—	Projected Tolerance Zone	
—	—	Straightness	Form
⏥	—	Flatness	
○	—	Circularity	
⌭	—	Cylindricity	
⊥	—	Perpendicularity	Orientation
//	—	Parallelism	
∠	—	Angularity	
⌖	—	Position	Location
⌯	—	Symmetry	
◎	—	Concentricity	
↗	—	Circular Runout	
⌰	—	Total Runout	
⌒	—	Line Profile	
⌓	—	Surface Profile	

Most Common Symbols



	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	SEE:
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	⏤	6.4.1
		FLATNESS	⏥	6.4.2
		CIRCULARITY (ROUNDNESS)	○	6.4.3
		CYLINDRICITY	⌭	6.4.4
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒	6.5.2 (b)
		PROFILE OF A SURFACE	⌓	6.5.2 (a)
FOR RELATED FEATURES	ORIENTATION	ANGULARITY	∠	6.6.2
		PERPENDICULARITY	⊥	6.6.4
		PARALLELISM	//	6.6.3
	LOCATION	POSITION	⌖	5.2
		CONCENTRICITY	◎	5.11.3
		SYMMETRY	⌯	5.13
	RUNOUT	CIRCULAR RUNOUT	↗*	6.7.1.2.1
		TOTAL RUNOUT	⌰*	6.7.1.2.2

* ARROWHEADS MAY BE FILLED OR NOT FILLED

3.3.1

Feature Control Frame

A geometric tolerance is prescribed using a feature control frame.

It has three components:

1. the tolerance symbol,
2. the tolerance value,
3. the datum labels for the reference frame.

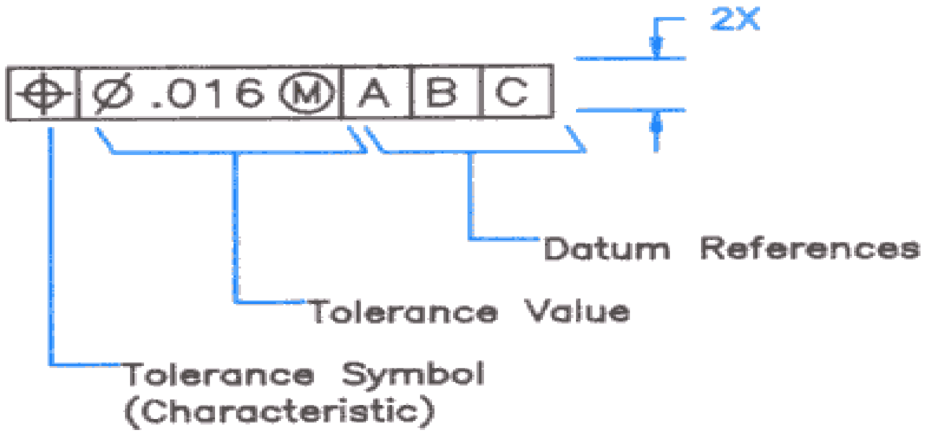


Figure 2-18. Feature control frames are always read from left to right.

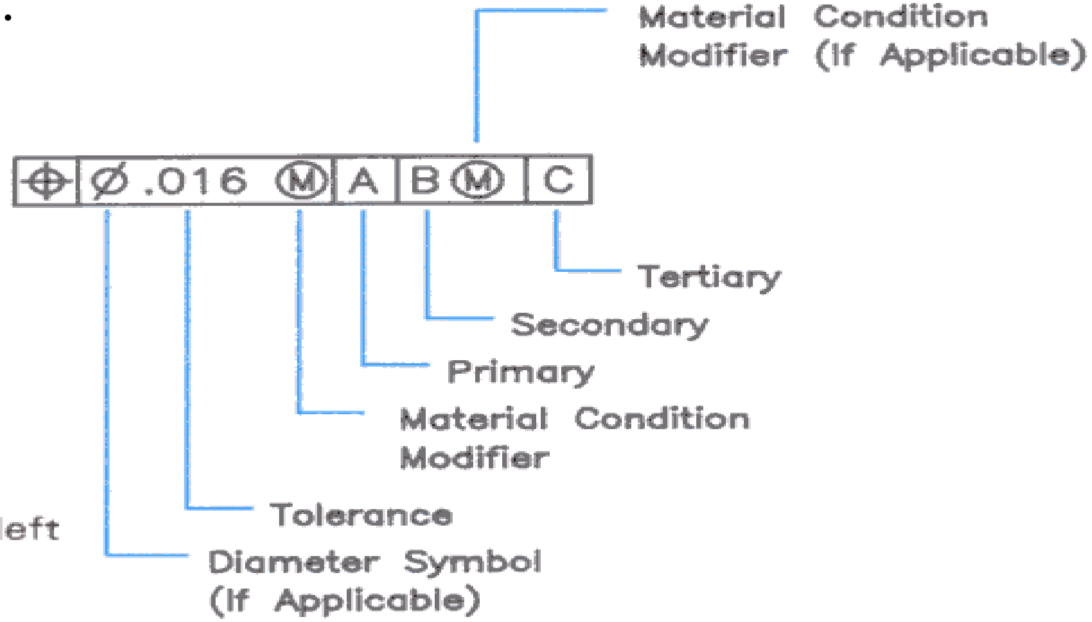
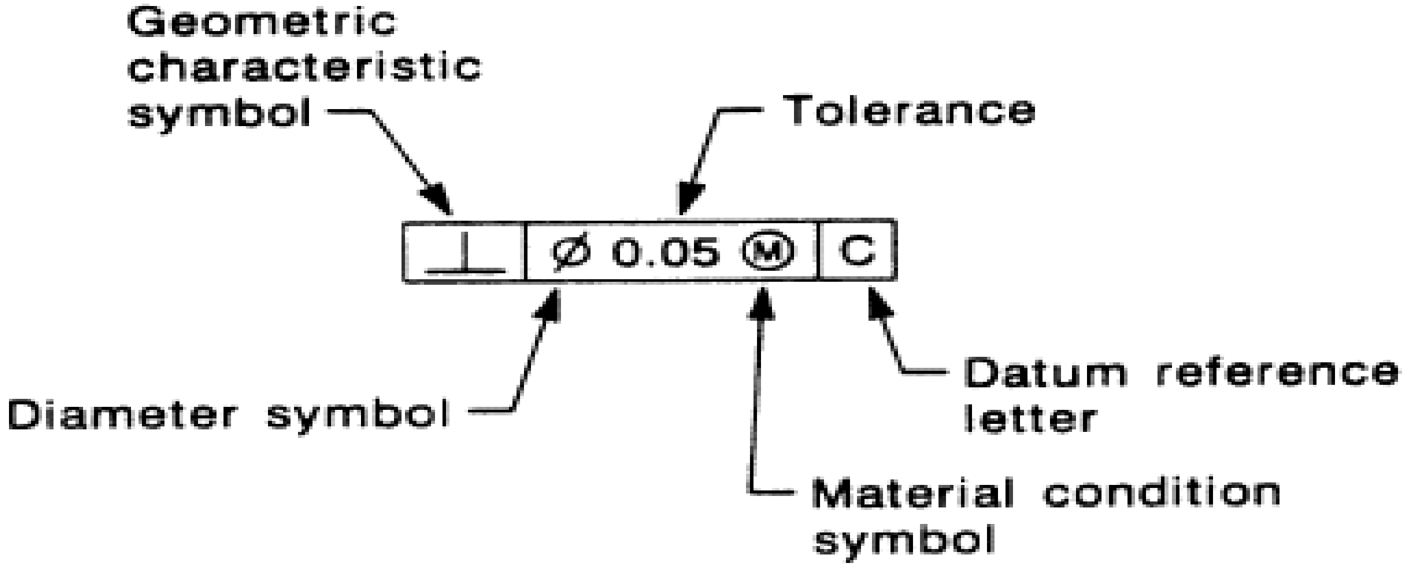


Figure 2-19. Whether a diameter symbol and material condition modifier are used, or omitted, depends on the desired tolerance specification and the type of feature being controlled.

Feature Control Frame

- How do you read this feature control frame?

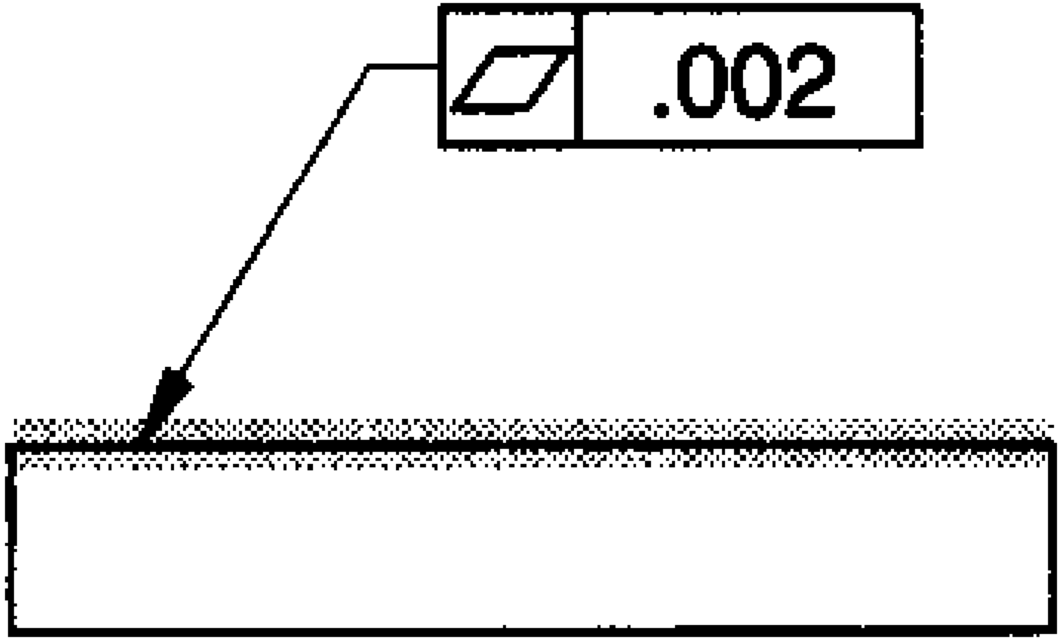
“The specified feature must lie *perpendicular* within a tolerance zone of *0.05 diameter* at the *maximum material condition*, with respect to *datum axis C*.”



In other words, this places a limit on the amount of variation in perpendicularity between the feature axis and the datum axis. In a drawing, this feature control frame would accompany dimensional tolerances that control the feature size and position.

Geometrical Tolerances (Form)

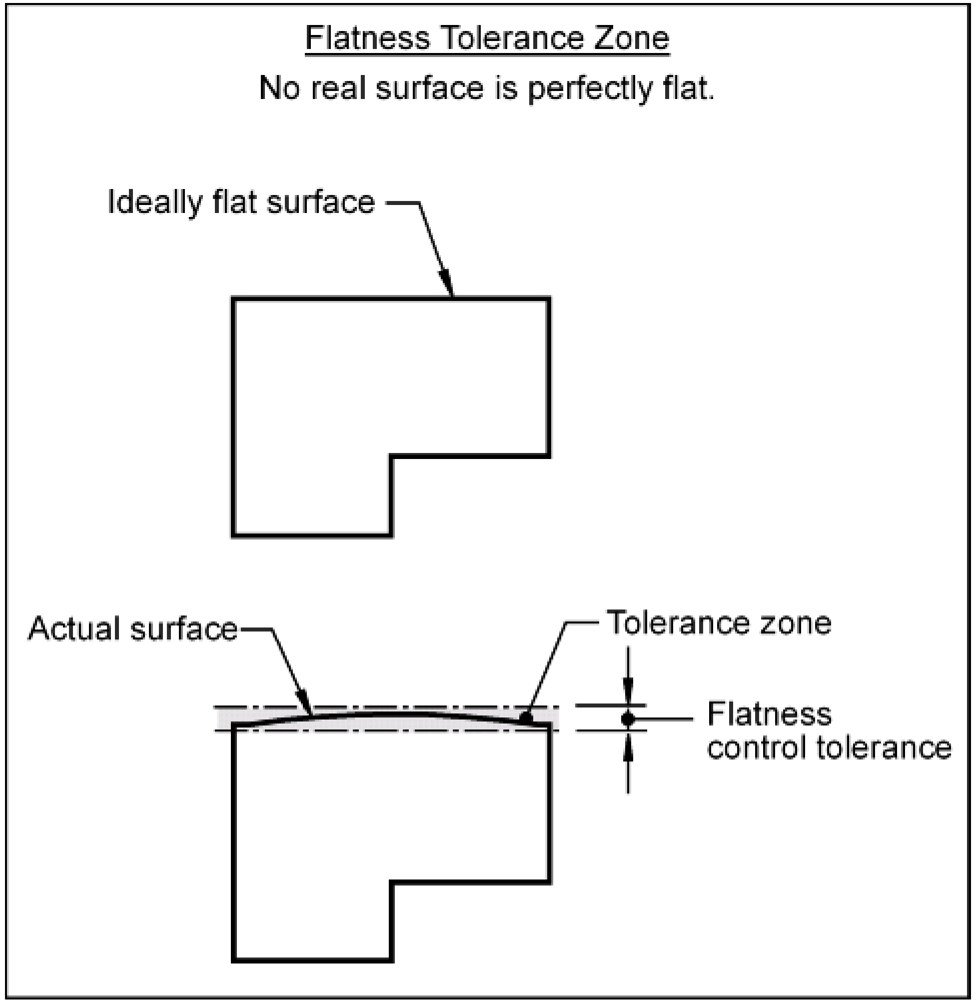
⏥ Flatness



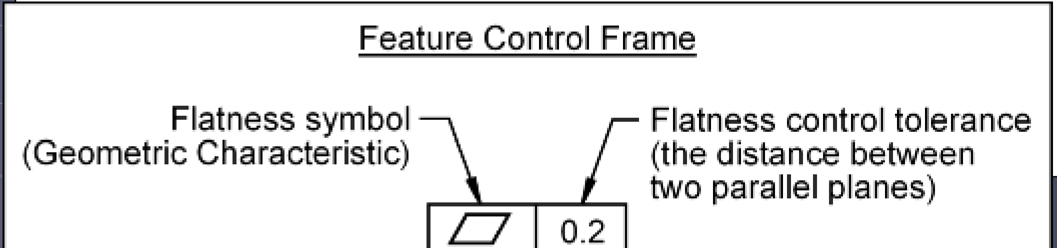
All points on the indicated surface must lie in a single plane, within the specified tolerance zone.

The flatness tolerance defines a distance between parallel planes that must contain the highest and lowest points on a face.

Flatness


Flatness Tolerance Zone

No real surface is perfectly flat.


Feature Control Frame

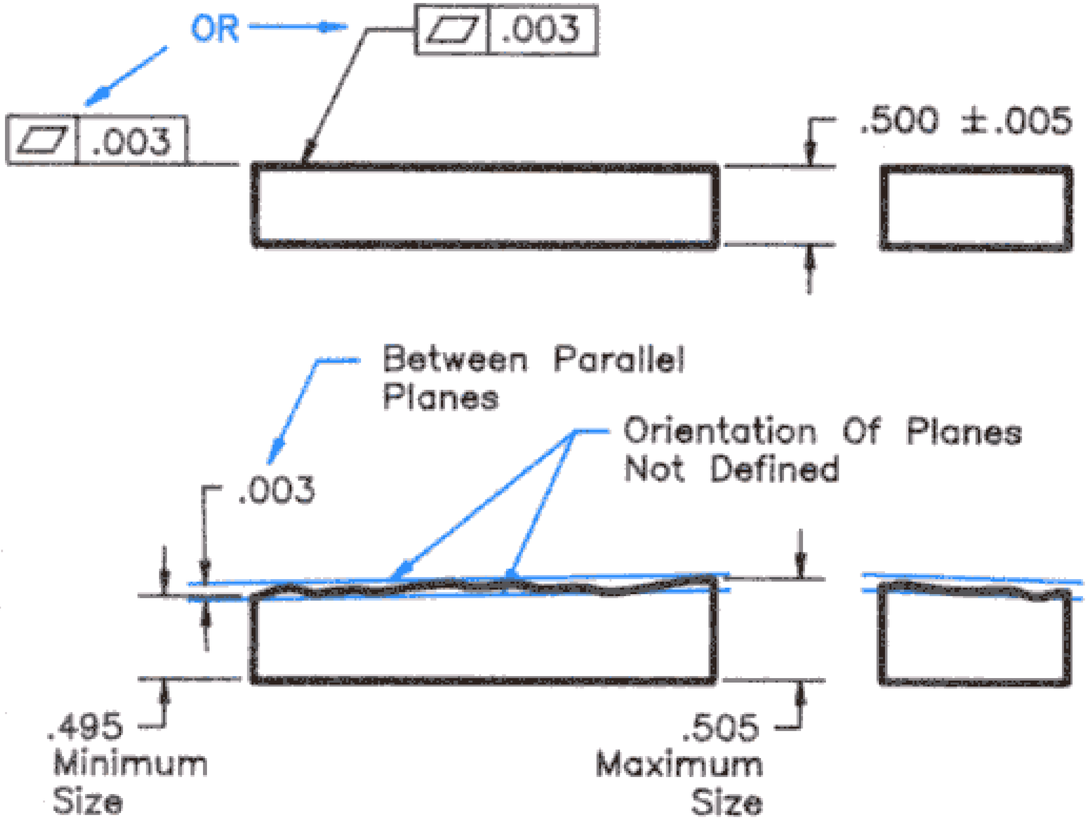
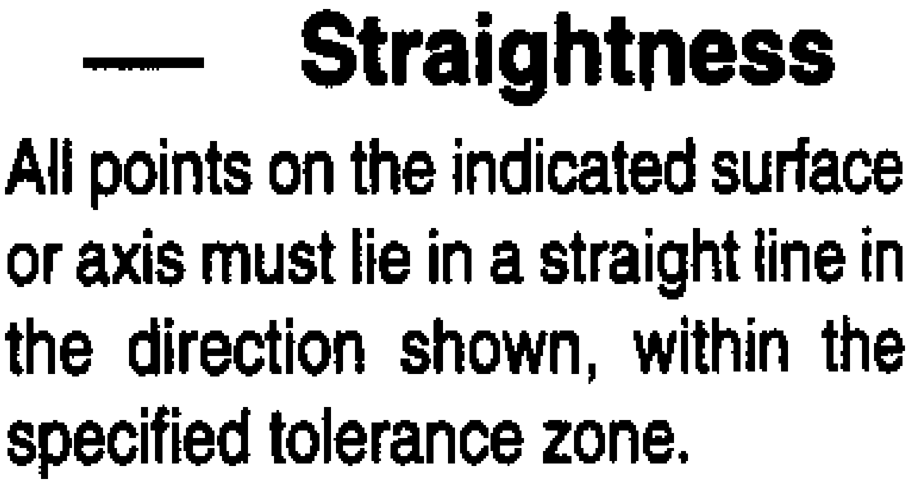


Figure 5-25. Flatness tolerances create a tolerance zone bounded by two parallel planes.

Flatness tolerance is **typically used for sliding surfaces**, sealing surfaces, and appearance, etc.

Geometrical Tolerances (Form)



— Straightness

All points on the indicated surface or axis must lie in a straight line in the direction shown, within the specified tolerance zone.

Straightness tolerancing is **typically applied** to a cylindrical features (pins, shafts, bars,etc.) **to control bowing and other distortions**.

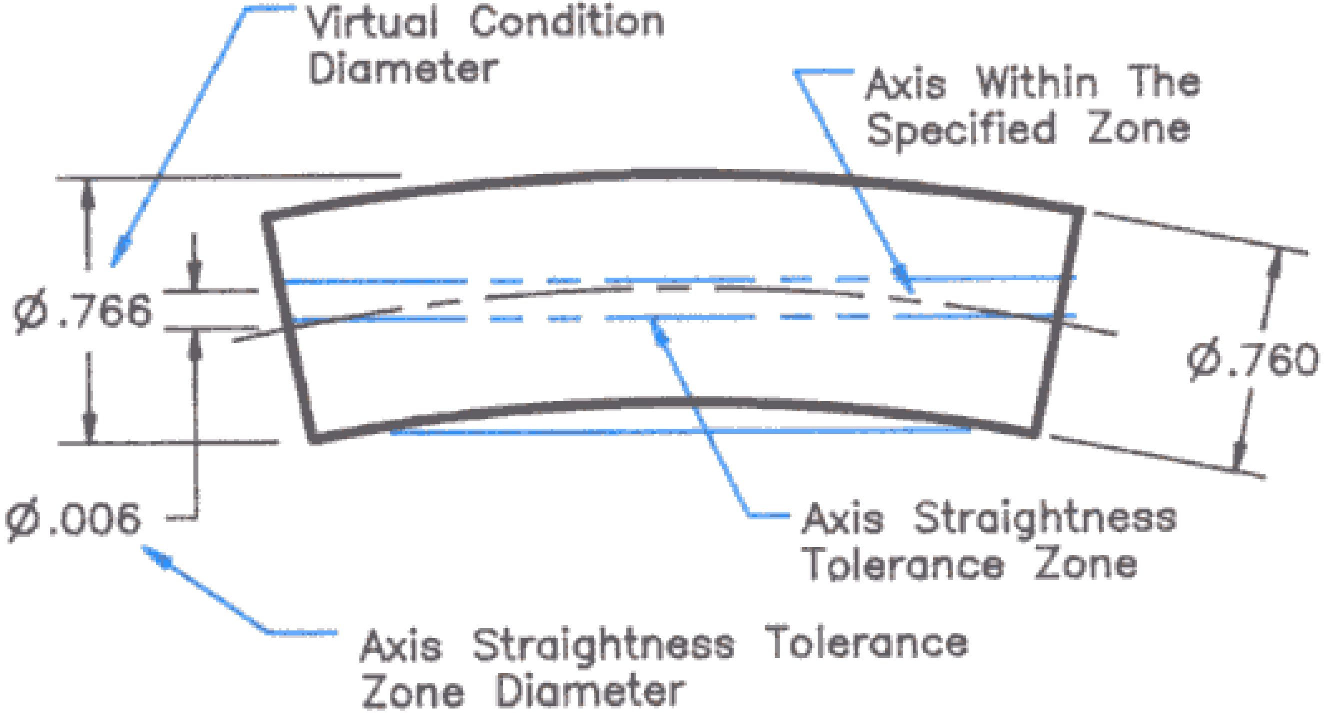
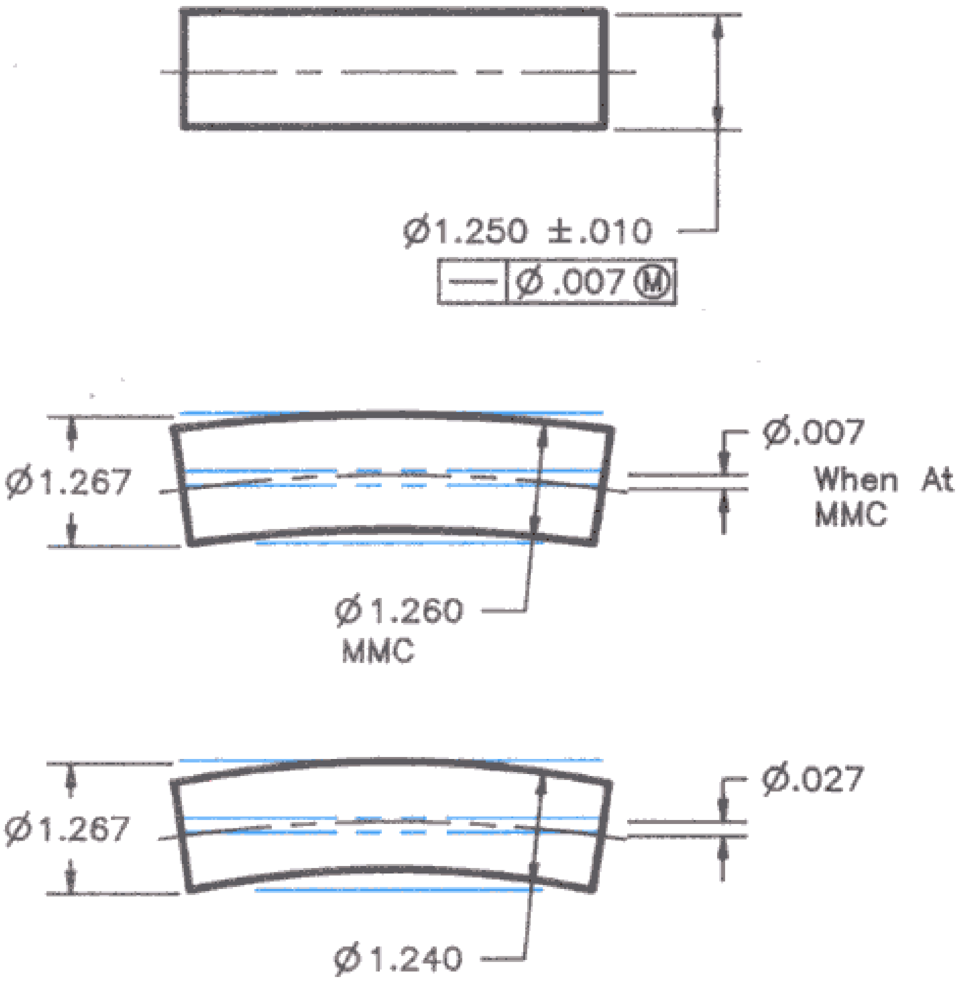


Figure 5-17. A straightness tolerance applied to control axis straightness permits the part to have an axis straightness error when the part is at MMC.

Geometrical Tolerances (Form)

— Straightness



MMC Of Shaft
\+ Tolerance

Virtual Condition

1.260 MMC
\+ .007 Tol

1.267 Vc

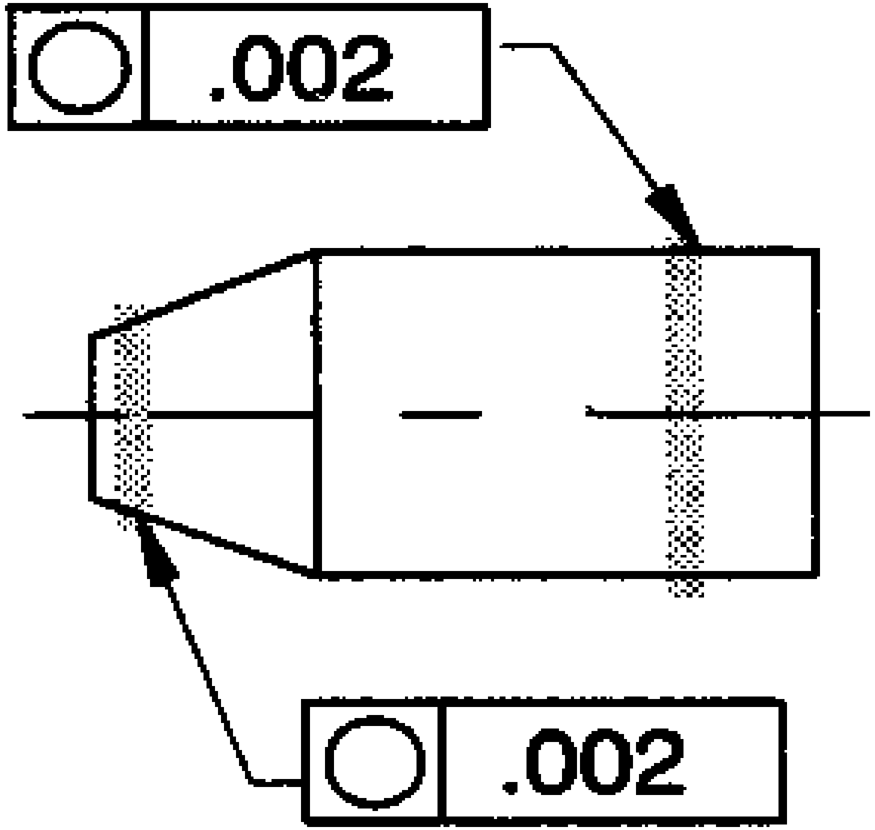
Produced Diameter	Allowable Tolerance
1.260	.007
1.259	.008
1.258	.009
|	|
1.242	.025
1.241	.026
1.240	.027

- Add the straightness tolerance to the maximum shaft size (MMC) to obtain a "virtual condition" Vc, or virtual hole, that the shaft must fit to be acceptable.

Figure 5-18. The permitted axis error applied to a cylinder has an effect on the apparent diameter of the cylinder. The combined effect of the MMC size and the permitted axis error is known as the virtual condition.

Geometrical Tolerances (Form)

○ Circularity (Roundness)



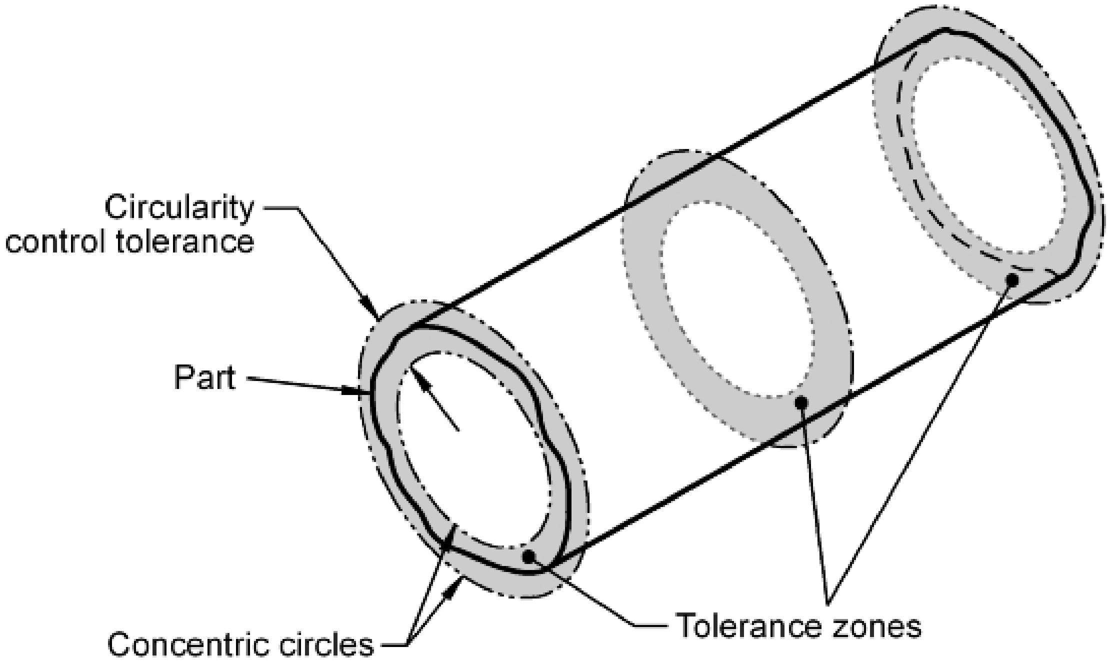
If the indicated surface were sliced by any plane perpendicular to its axis, the resulting outline must be a perfect circle, within the specified tolerance zone.

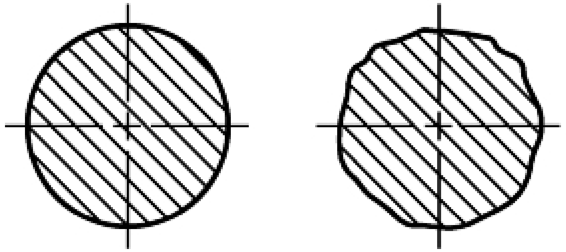
The **circularity control** defines how much each circular cross-sections of a cylinder, sphere or cone may deviate from its perfect circular form.

Circularity

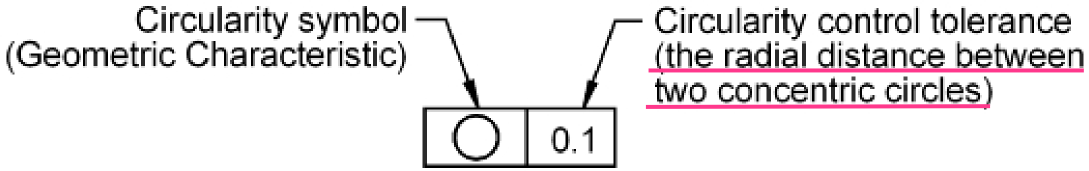
Circularity Tolerance Zone

The area between two concentric circles.

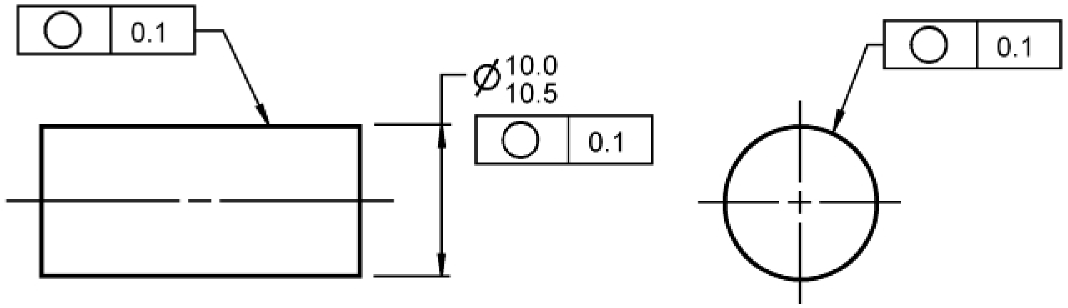




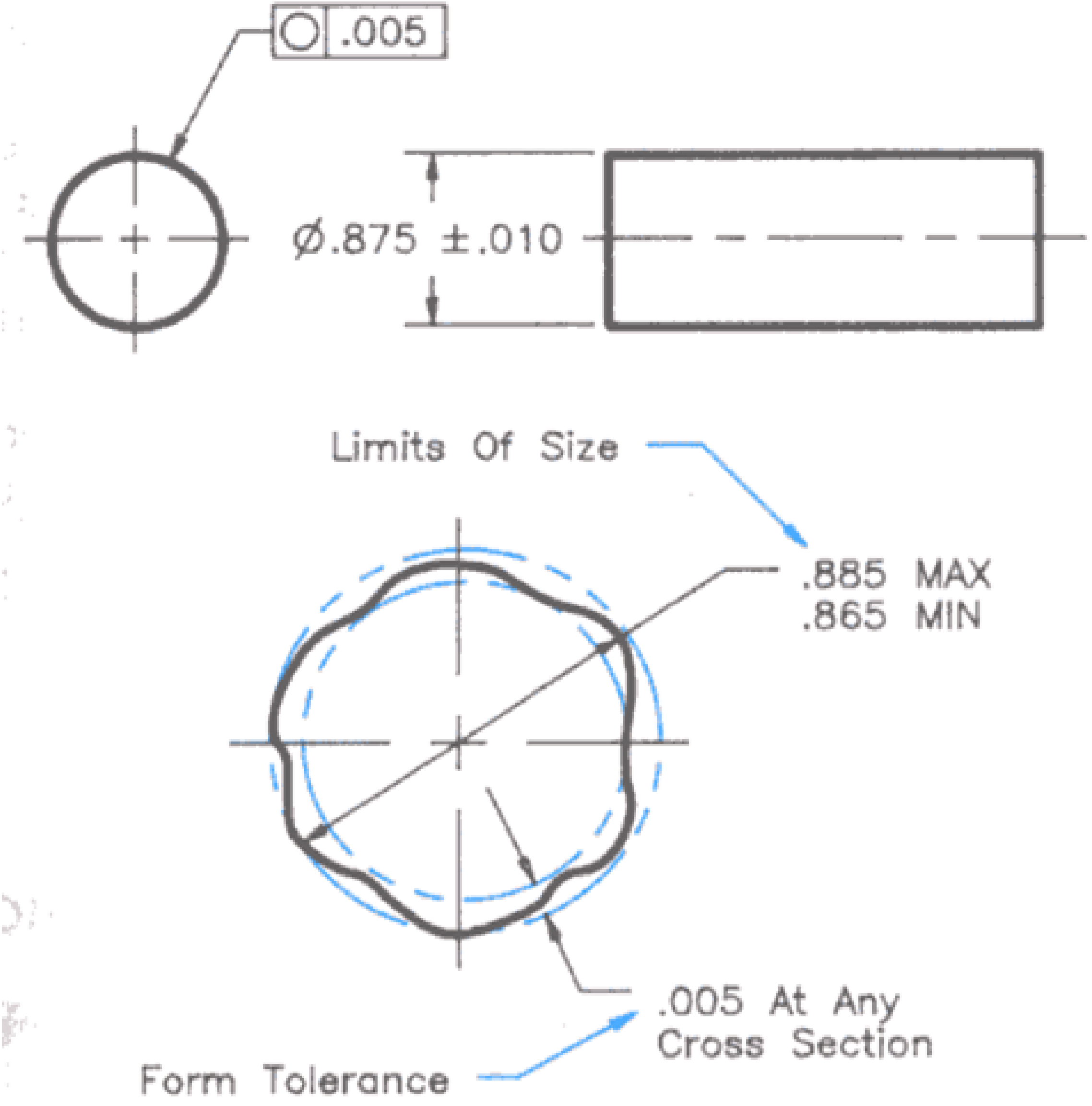
Feature Control Frame



Feature Control Frame Placement

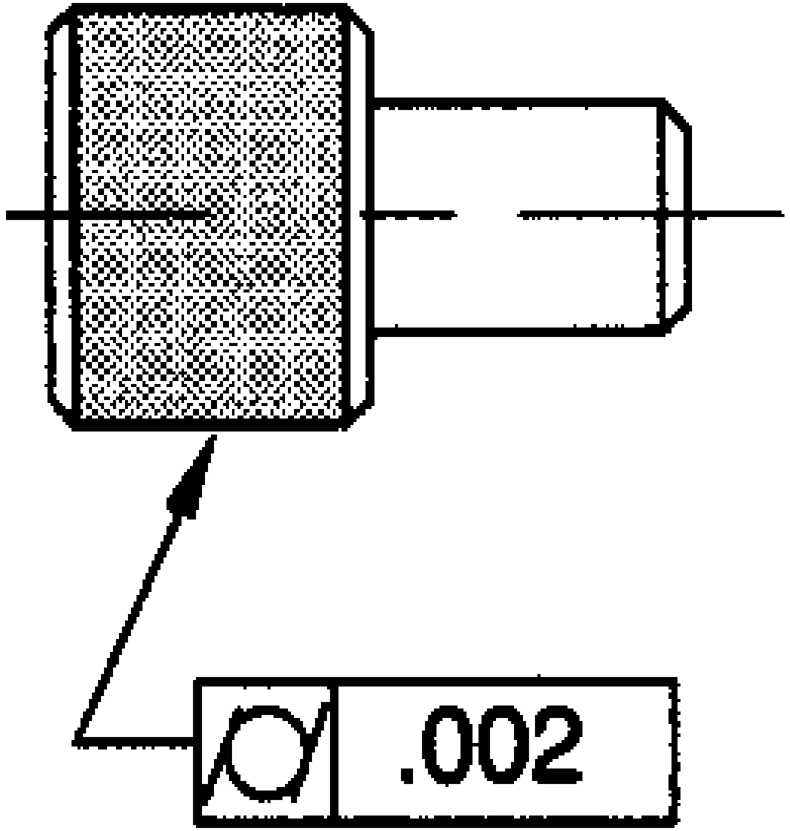


Circularity



Geometrical Tolerances (Form)

Cylindricity



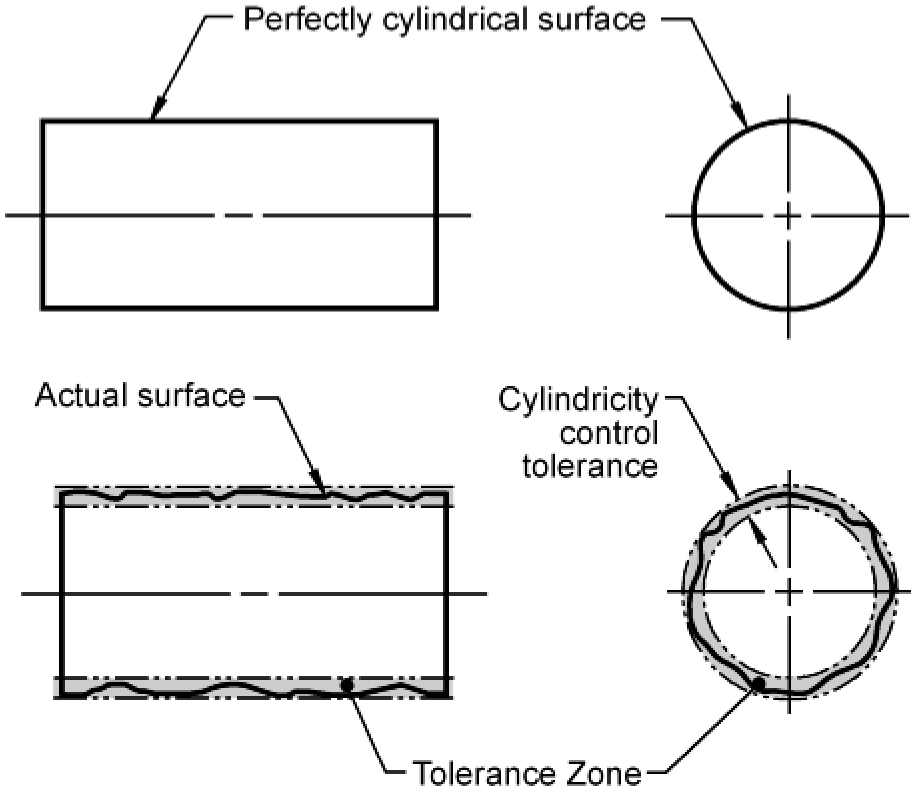
All points on the indicated surface must lie in a perfect cylinder around a center axis, within the specified tolerance zone.

The **cylindricity control** defines how much a cylindrical surface on a real part may vary from an ideal cylinder that is perfectly round, perfectly straight and has no taper.

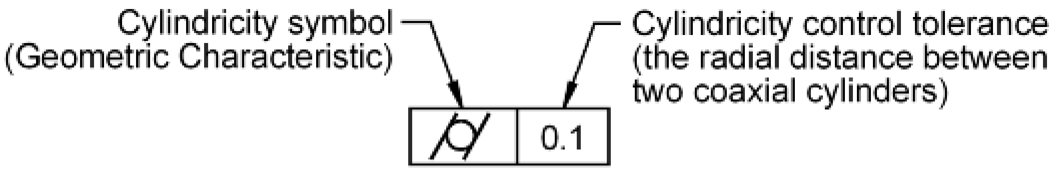
Cylindricity

Cylindricity Tolerance Zone

(No real surface is perfectly cylindrical.)

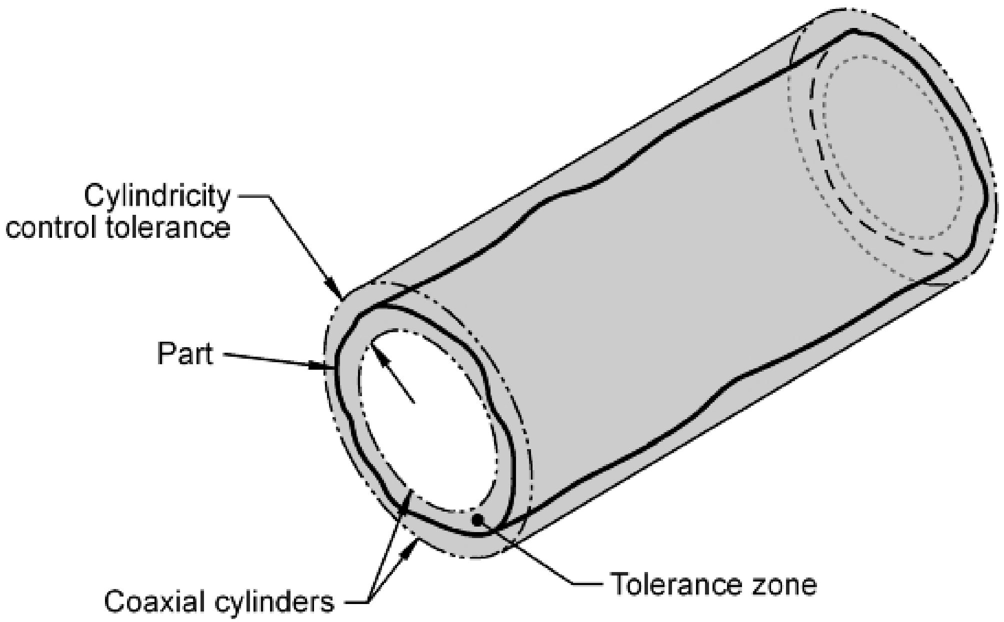


Feature Control Frame

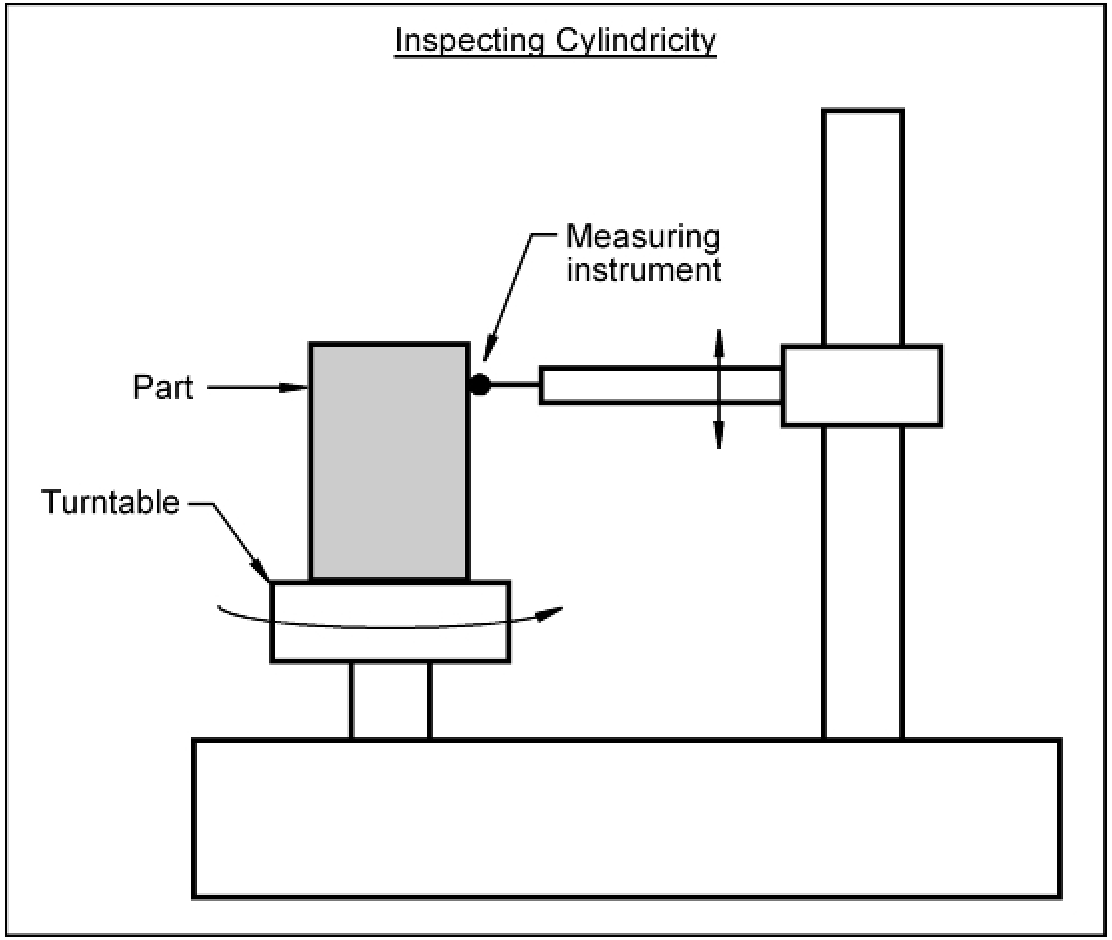


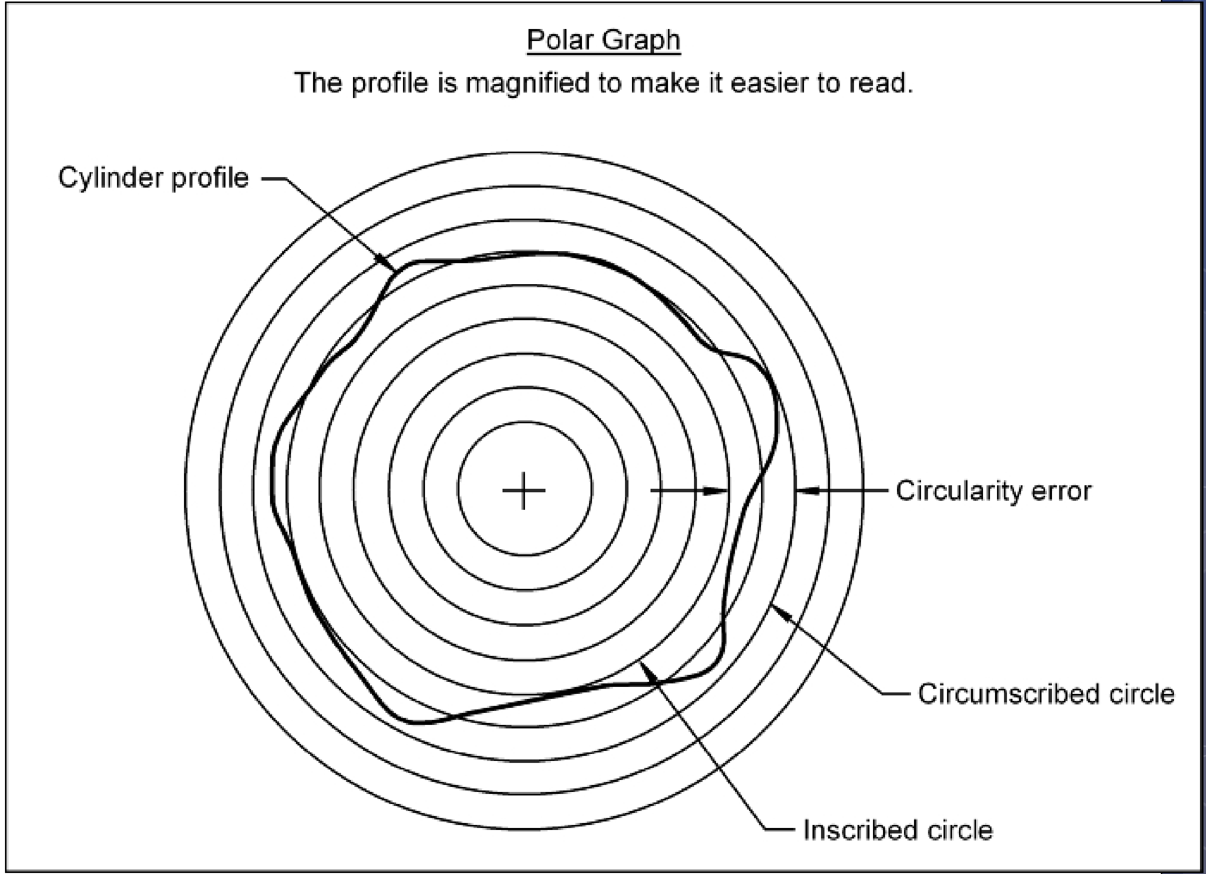
Cylindricity Tolerance Zone

The volume between two coaxial cylinders.



Cylindricity





CMM Video: https://www.youtube.com/watch?v=m5be1CWYRc0

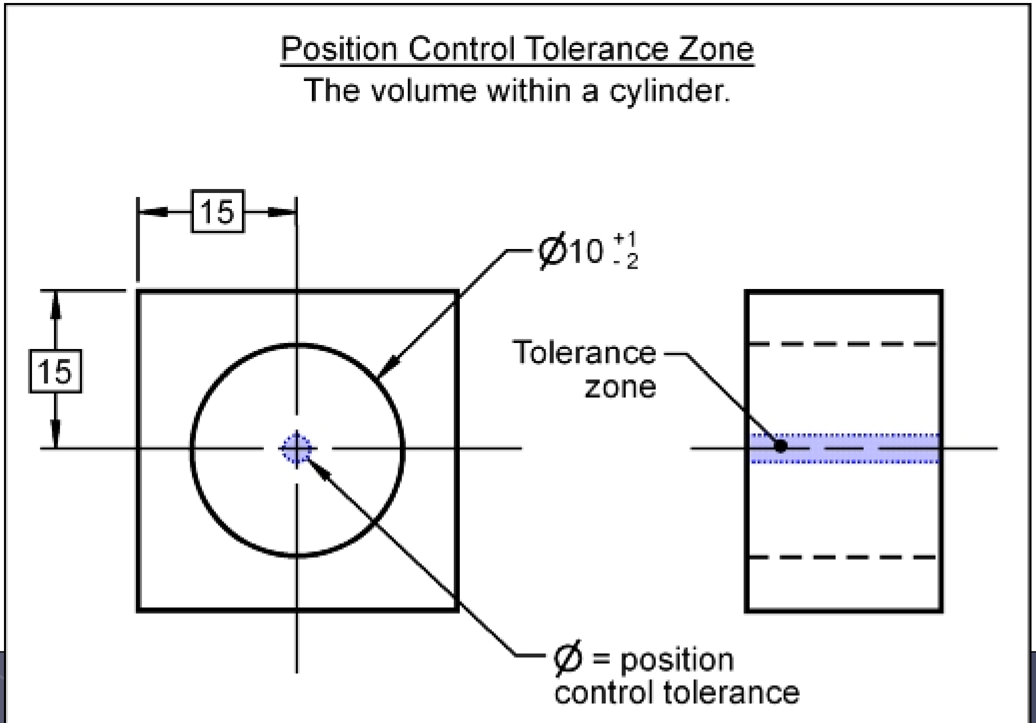
Geometrical Tolerances (Location)

Position

The position tolerance can control a *center point*, *axis*, or *center plane*.

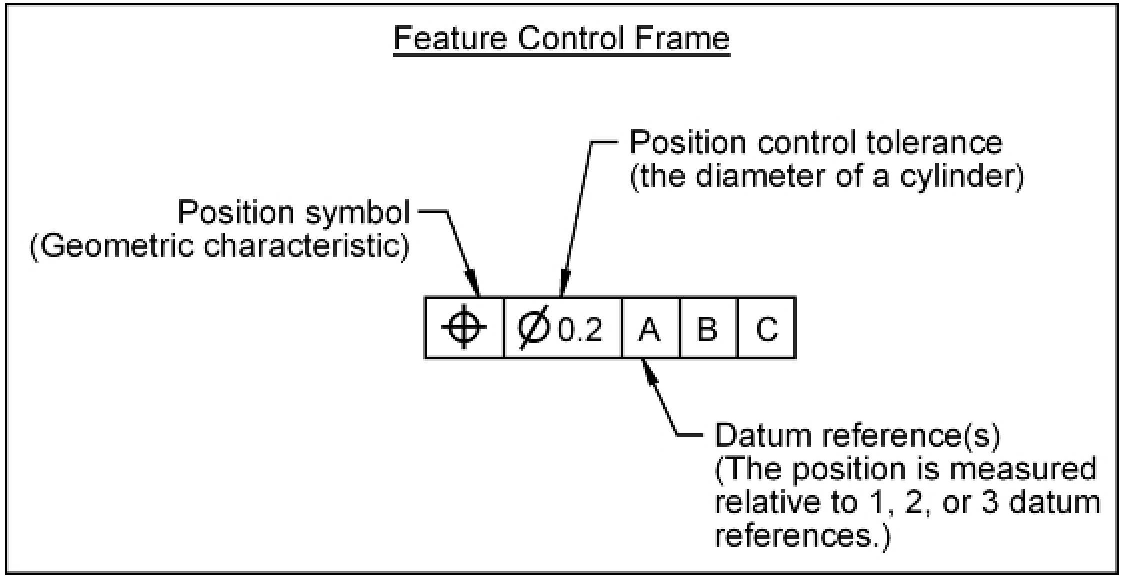
The **position tolerance** defines how much the location of a feature may deviate from its true position.

The **position tolerance zone** for a controlled axis is the volume defined by a cylinder. The diameter of the cylinder is the stated value of the **position control tolerance**. The axis being controlled must lie with in the volume defined by the tolerance zone.



Position Control Tolerance Zone

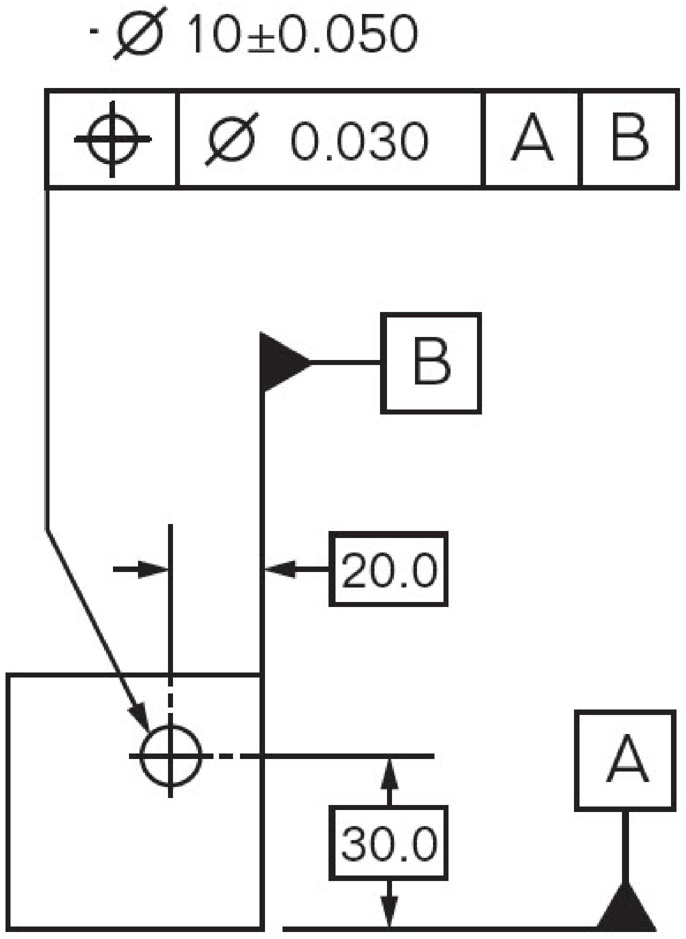
The volume within a cylinder.



Feature Control Frame

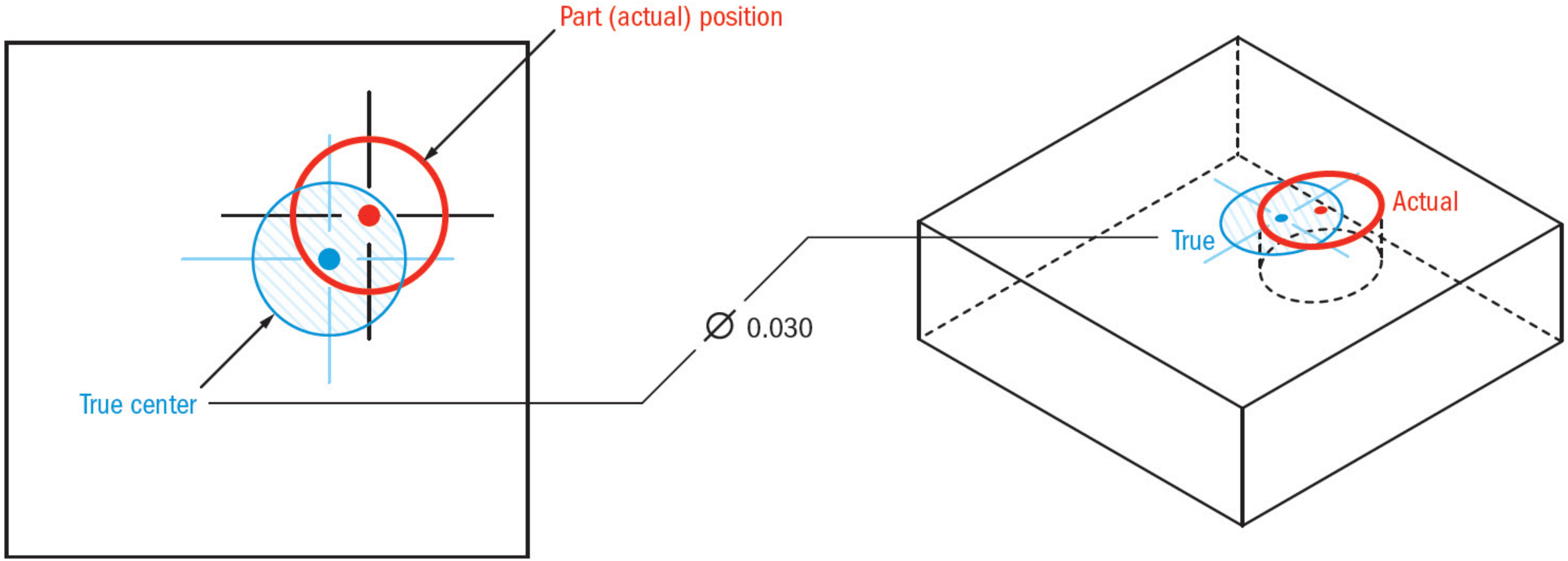
Geometrical Tolerances (Location)

Position



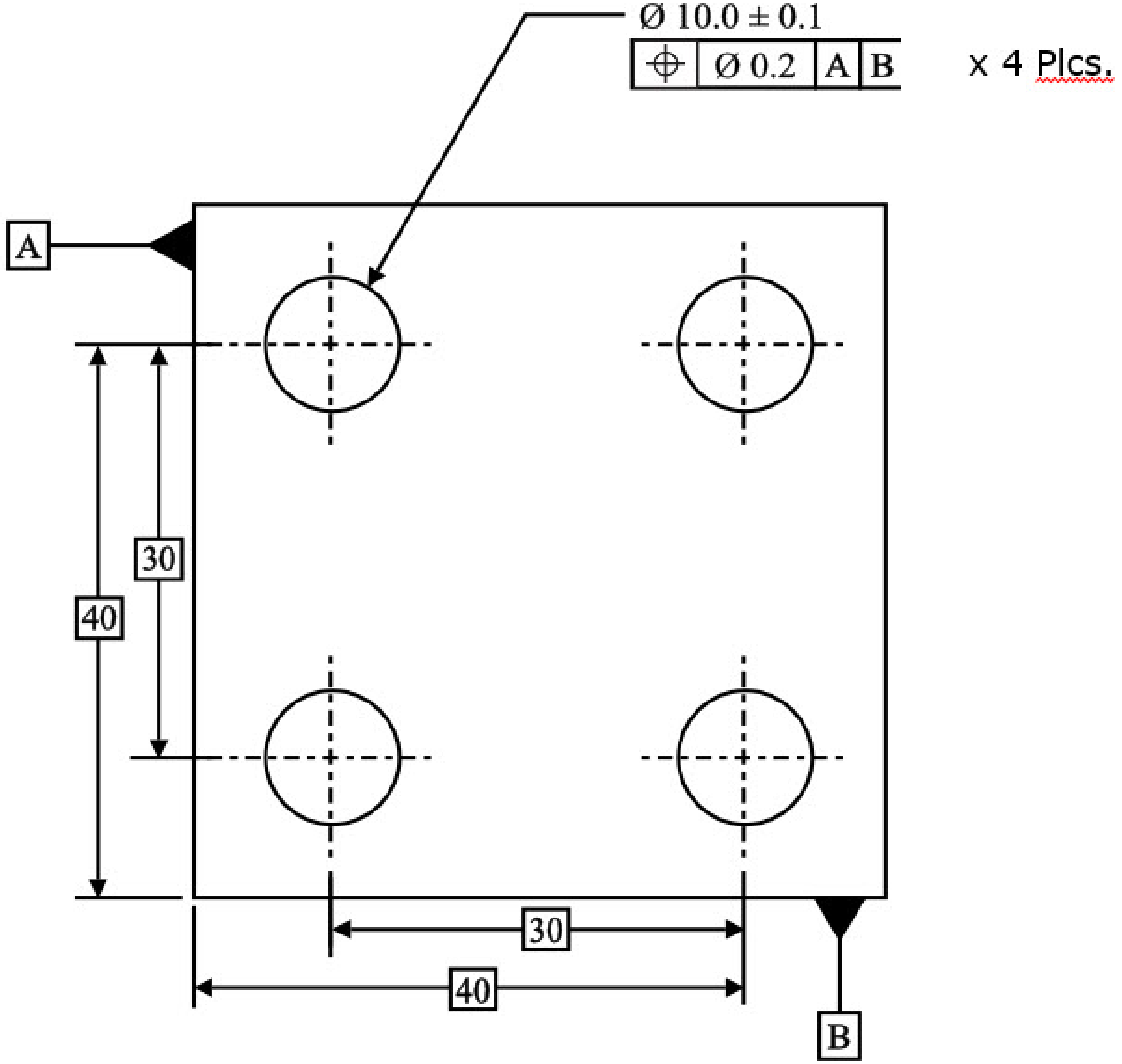
Geometrical Tolerances (Location)

Position



Geometrical Tolerances (Location)

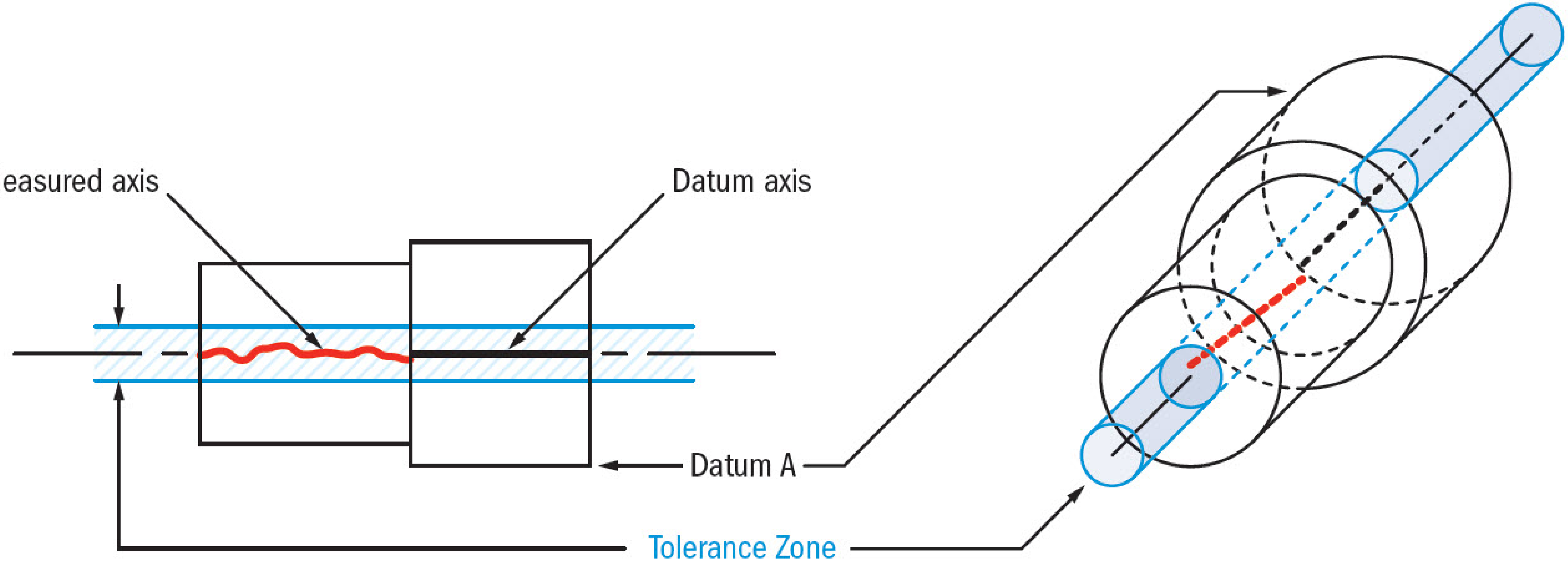
Position



Geometrical Tolerances (Location)

Concentricity

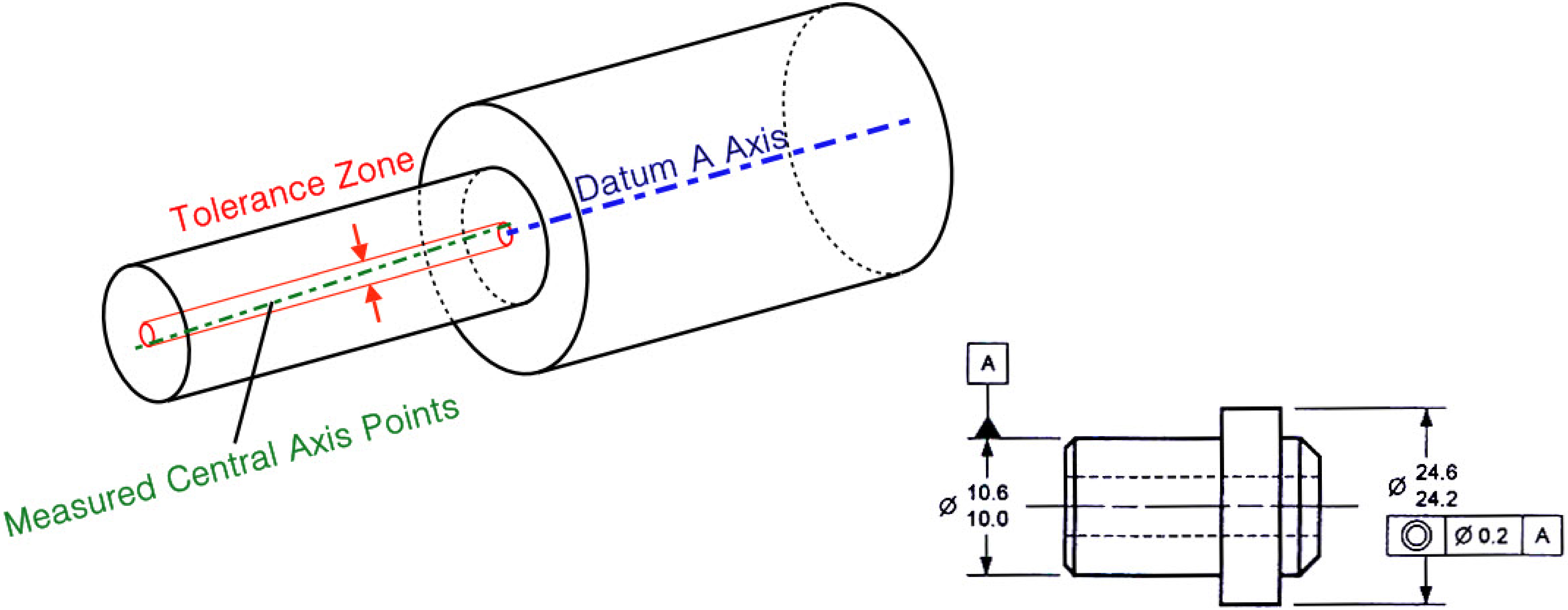
Concentricity, sometimes called coaxially, is a tolerance that controls the central axis of the referenced feature, to a datum axis.



Geometrical Tolerances (Location)

Concentricity

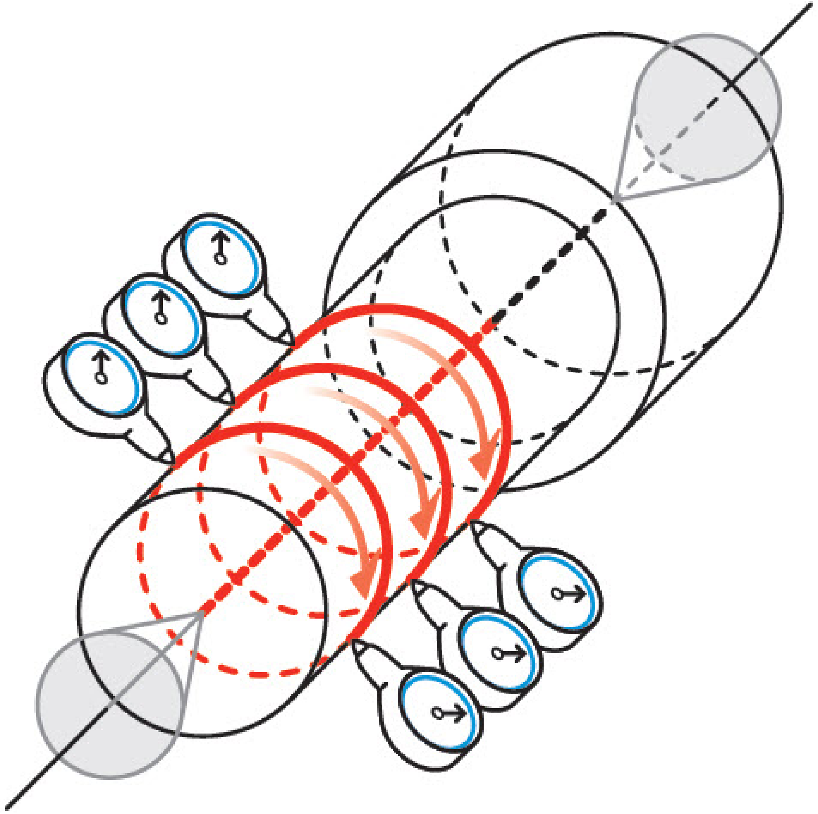
A concentricity tolerance specifies a cylindrical tolerance zone whose axis coincides with a datum axis and within which all cross-sectional axes of the feature being controlled must lie.



Geometrical Tolerances (Location)

Concentricity

A concentricity tolerance specifies a cylindrical tolerance zone whose axis coincides with a datum axis and within which all cross-sectional axes of the feature being controlled must lie.



The following is usually done with a CMM:

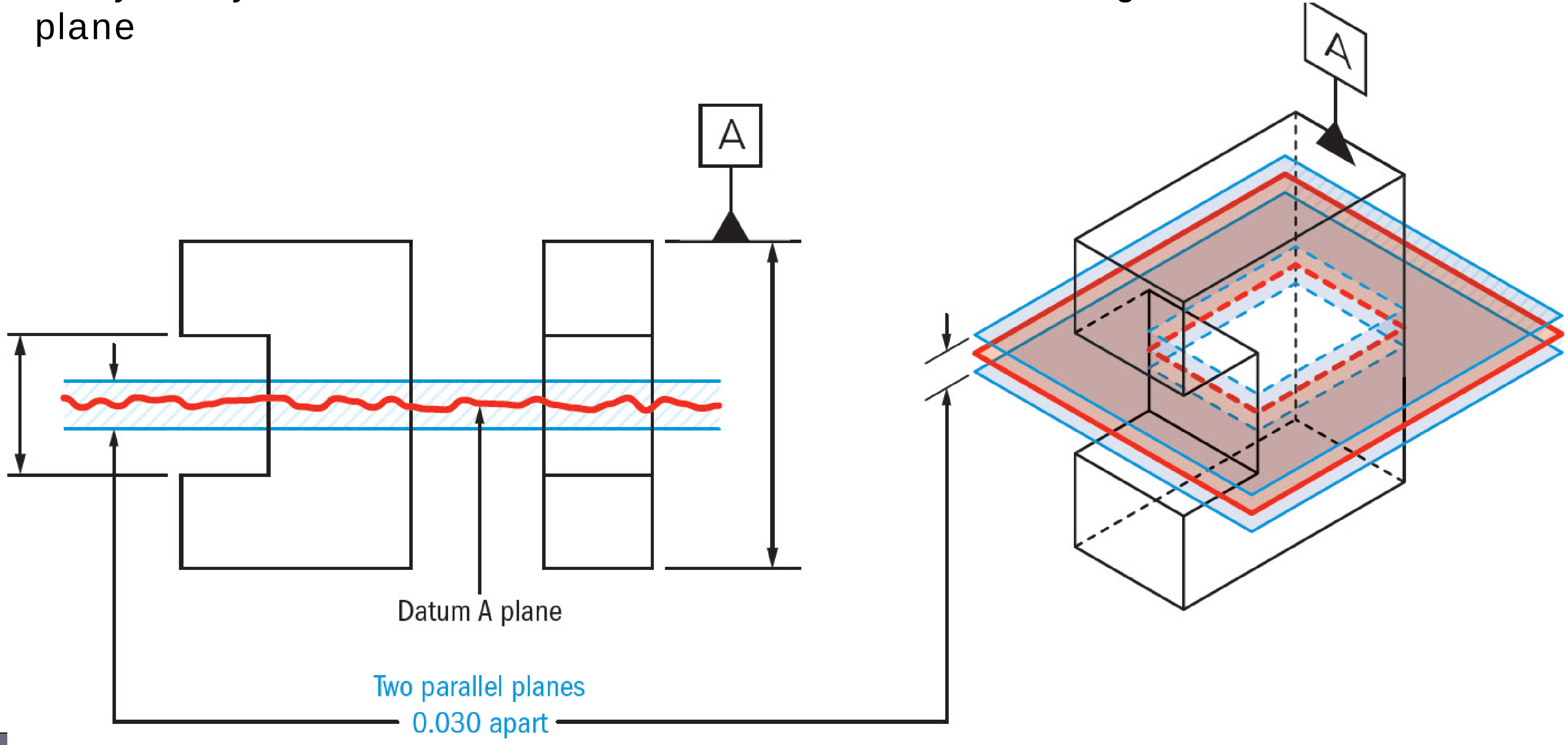
1. Determine Datum axis
2. Measure referenced surface
3. Determine if
   central axis fall in TZ

Geometrical Tolerances (Location)

Symmetry

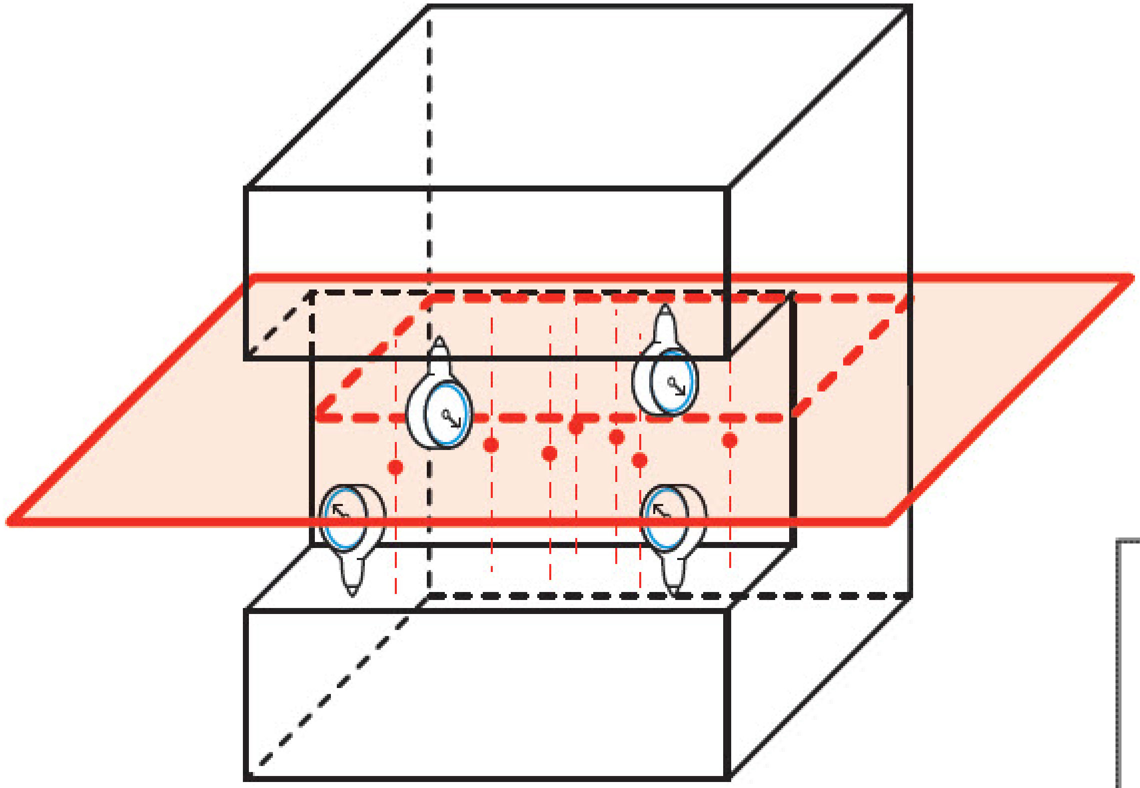
Symmetry is a 3-Dimensional tolerance that is used to ensure that two features on a part are uniform across a datum plane.

Symmetry is very difficult to measure. Due its tolerance zone being constrained to a virtual plane



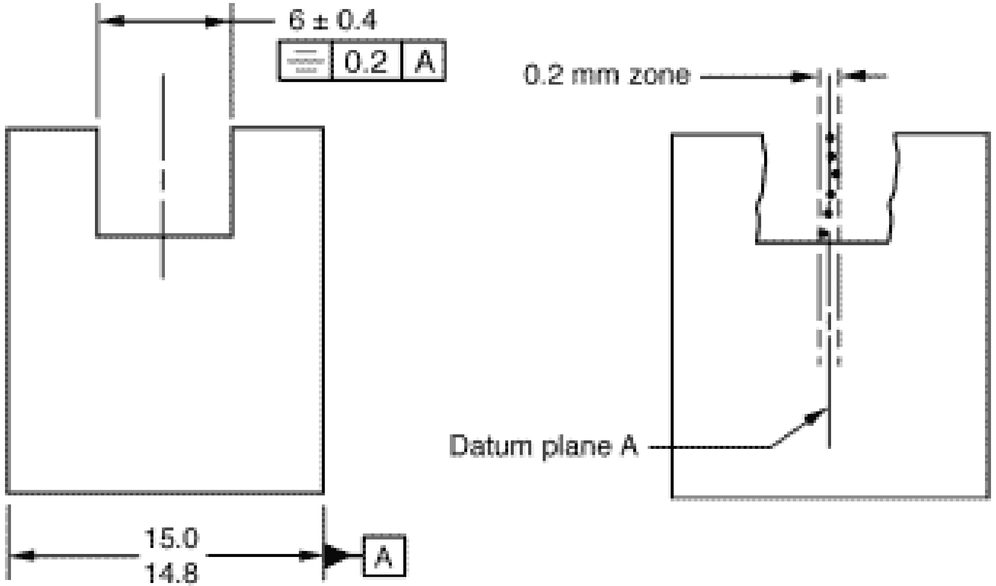
Symmetry

Symmetry is very difficult to measure. Due its tolerance zone being constrained to a virtual plane



The following is usually done with a CMM:

1. Determine Datum plane
2. Measure both surfaces of features
3. Determine if midpoints fall in TZ



Geometrical Tolerances (orientation)

Parallelism Tolerance

A parallelism tolerance is measured relative to a datum specified in the control frame.

If there is no material condition (i.e.. regardless of feature size), then the tolerance defines parallel planes that must contain the maximum and minimum points on the face.

It relates the orientation of one surface plane parallel to another datum plane in a 3-Dimensional tolerance zone. The tolerance indirectly controls the 0° angle between the parts by controlling where the surface can lie based on the datum.

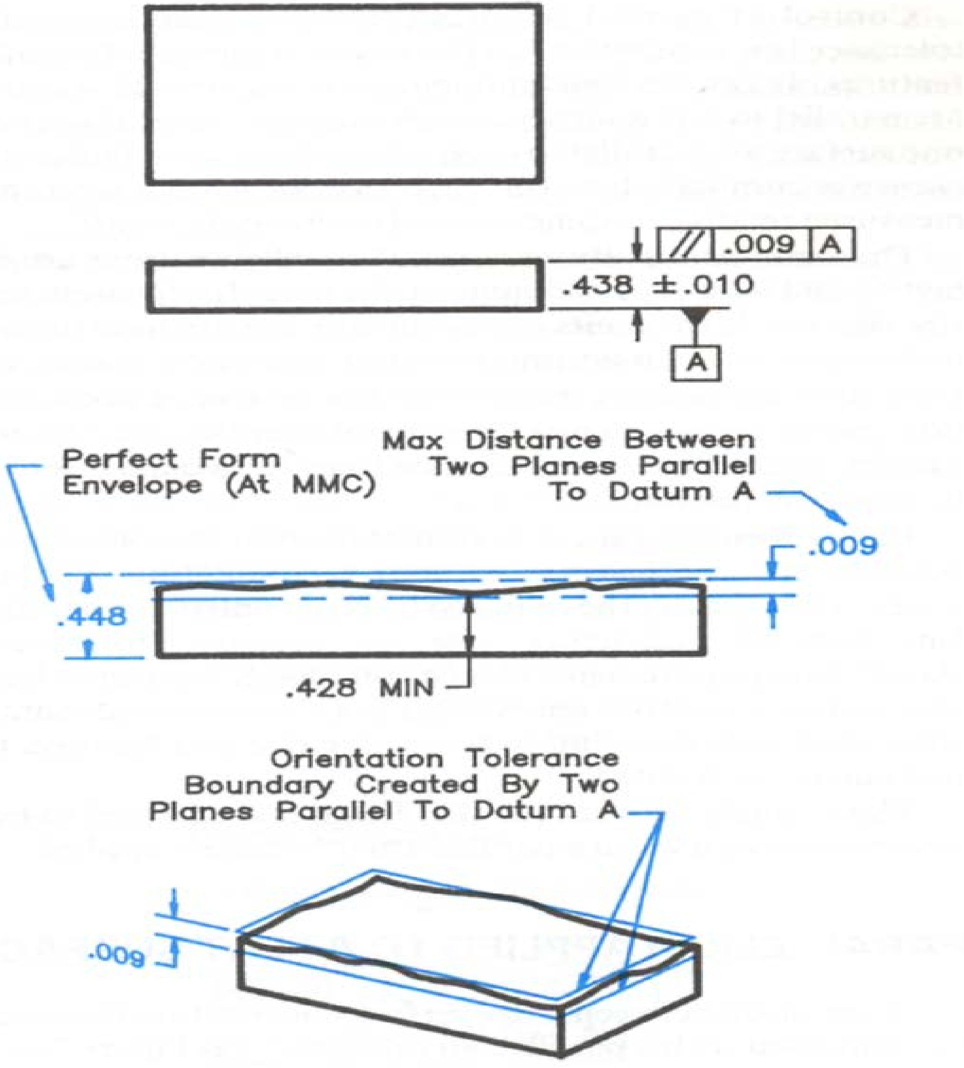
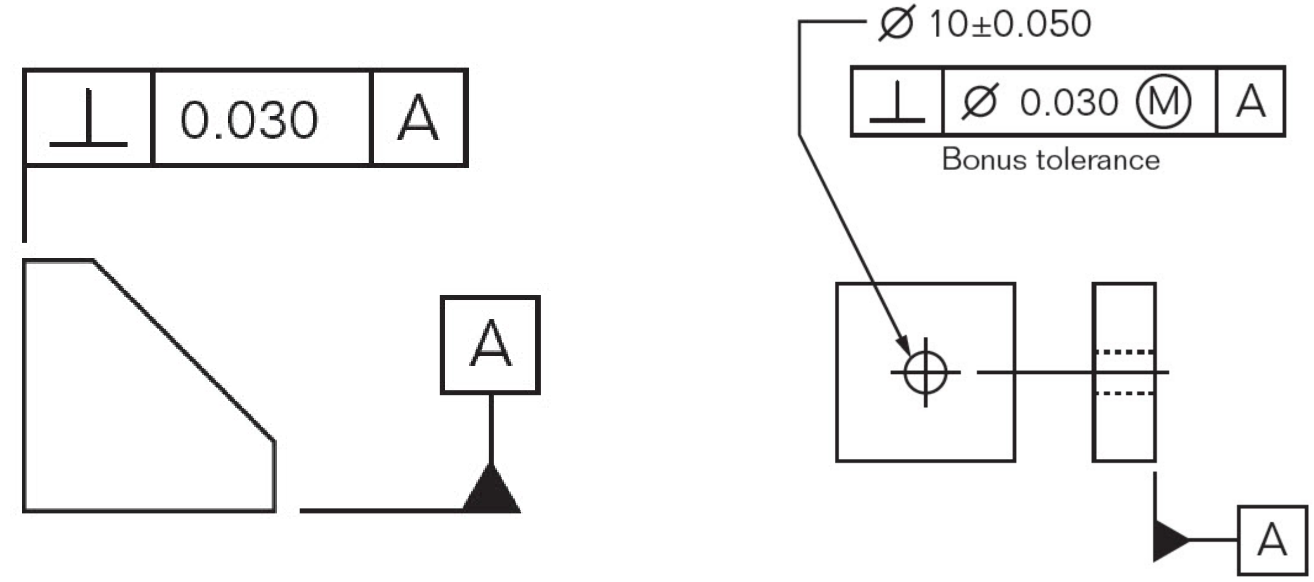


Figure 7-6. Two planes form the boundary for a parallelism tolerance.

Perpendicularity

It describes the orientation of one surface plane perpendicular to another datum plane. The tolerance of the perpendicularity indirectly controls the 90° angle between the parts by controlling the location where the surfaces have to lie.



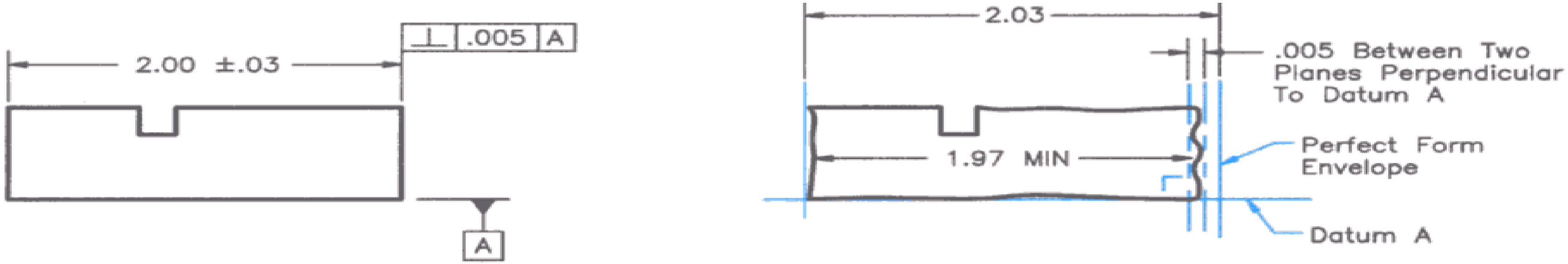
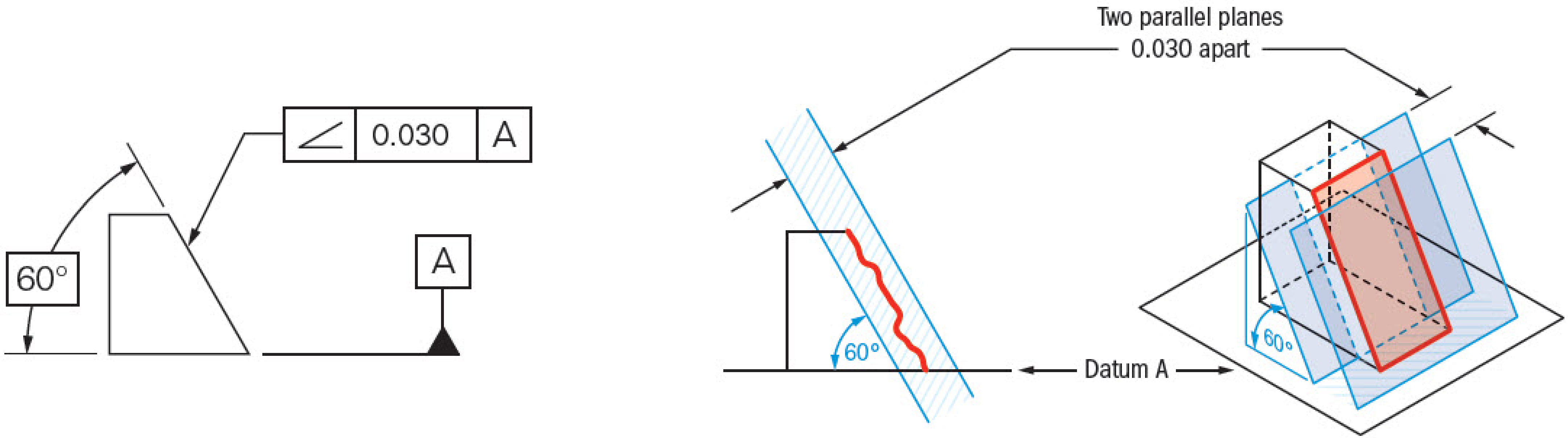


Figure 7-12. The tolerance zone for a perpendicularity tolerance on a flat surface is bounded by two planes.

Angularity

Angularity is the symbol that describes the specific orientation of one feature to another at a referenced angle

Angularity does not directly control the angle of the referenced surface; it controls the envelope (like flatness) that the entire surface can lie.



Angularity

Angularity does not directly control the angle of the referenced surface; it controls the envelope (like flatness) that the entire surface can lie.

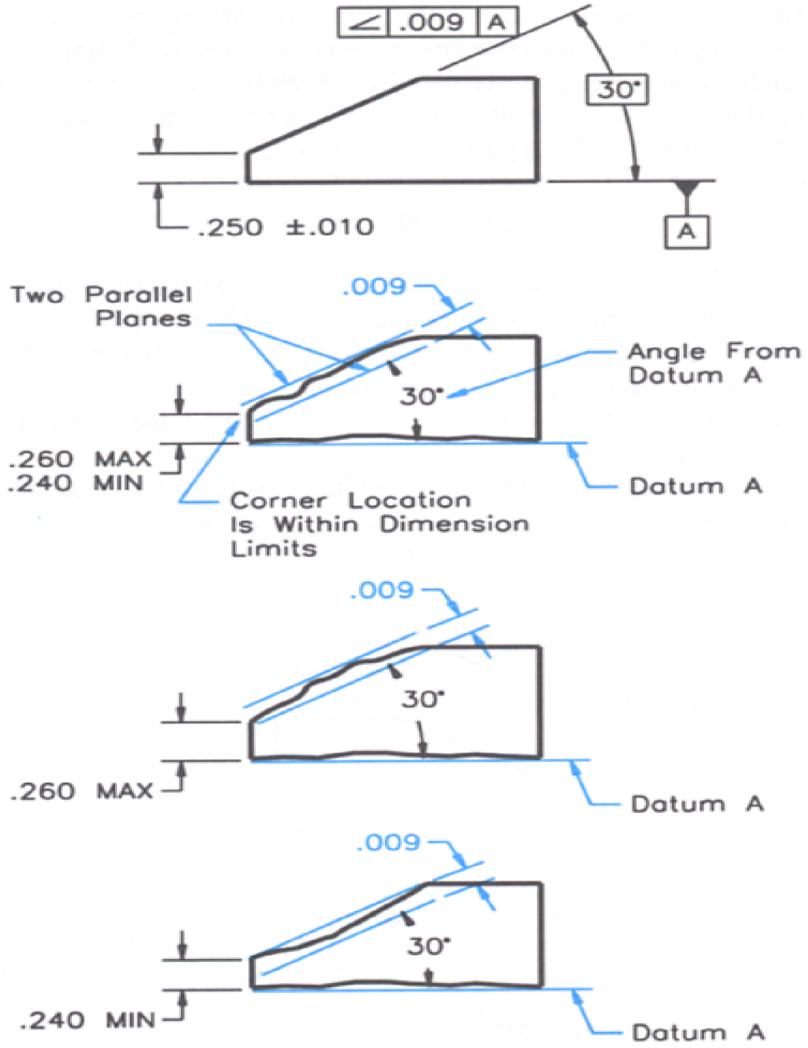
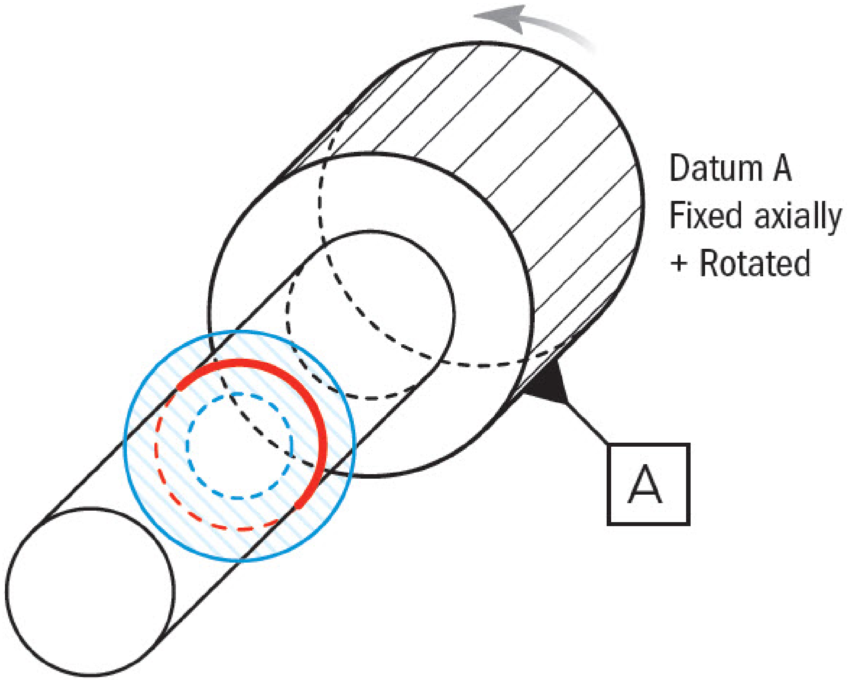


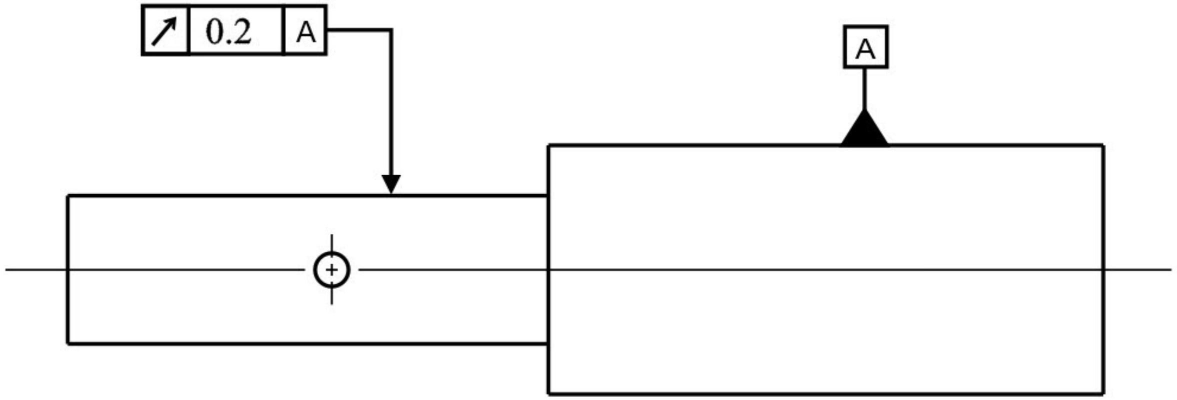
Figure 7-27. The angle dimension defining the surface orientation must be basic when an angularity tolerance is applied.

Runout

Runout is how much one given reference feature or features vary in its form and orientation with respect to another datum when the part is rotated 360° around the datum axis. It is essentially a control of a circular feature, and how much variation it has with the rotational axis.

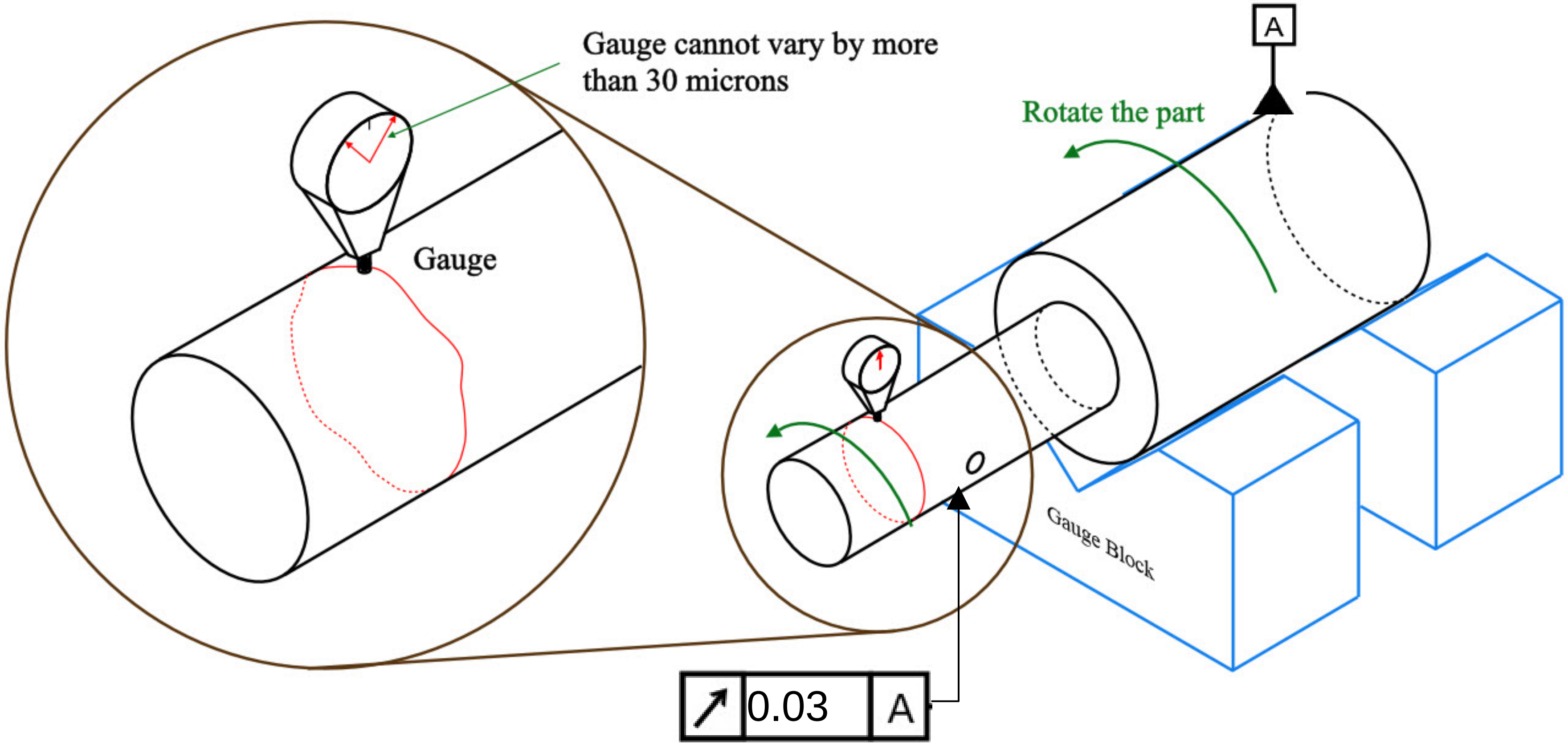
Tolerance zone is defined by a datum axis where all points on the called surface must fall into. The zone is a direct reference to the datum feature. Runout is the total variation that the reference surface can have, when the part is rotated around the datum's true axis.





Runout

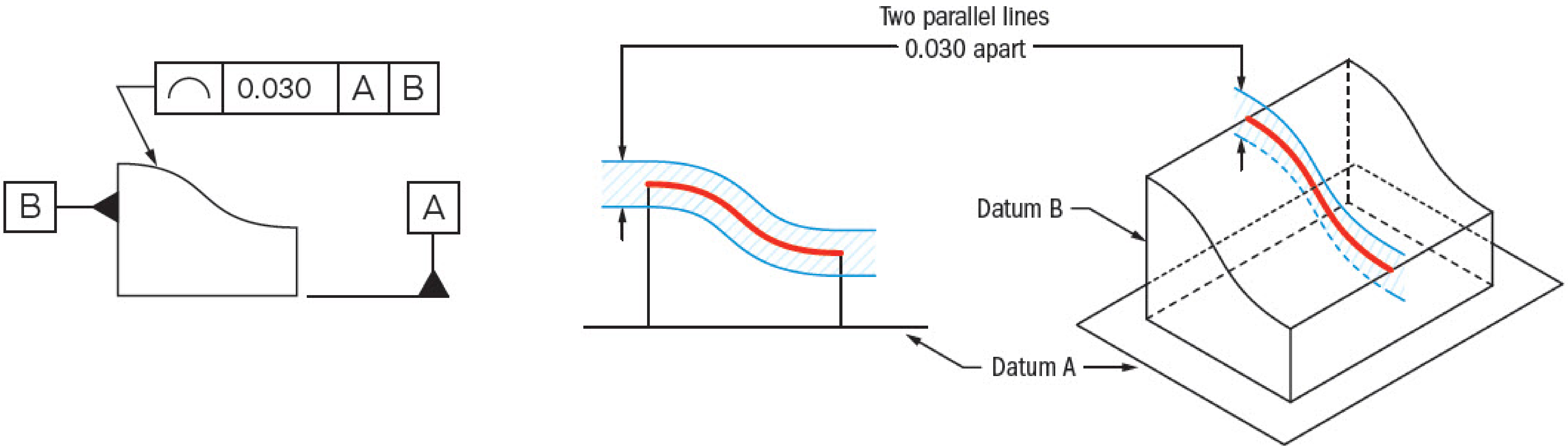
Tolerance zone is defined by a datum axis where all points on the called surface must fall into. The zone is a direct reference to the datum feature. Runout is the total variation that the reference surface can have, when the part is rotated around the datum’s true axis.



Profile of a Line

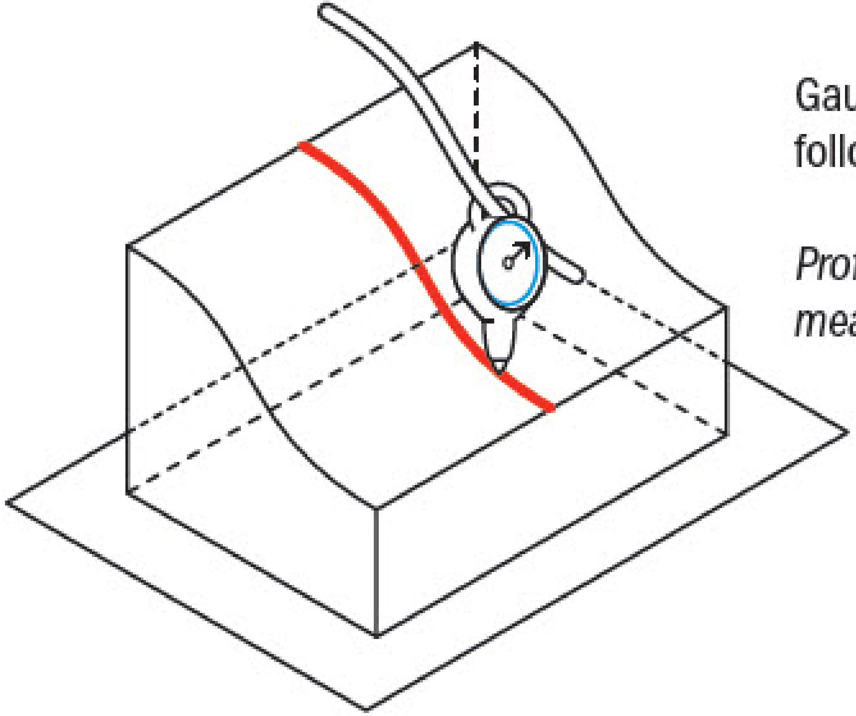
Profile of a line describes a tolerance zone around any line in any feature, usually of a curved shape.

Tolerance zone is defined by two parallel curves that follow the contour of the true surface profile.



Profile of a Line

Profile of a line is measured using a gauge that is referenced to the true profile at the given specific cross section.



Gauge must
follow true profile.

*Profile is usually
measured with a CMM.*