COMP0205 Mechatronics and Making

Geometric Dimensioning & Tolerancing

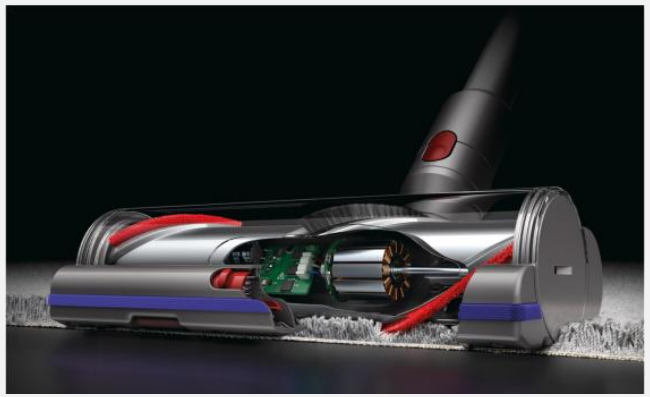
Dr. Chengxu Zhou
MEng Robotics and AI
UCL Computer Science

Today's Objectives

- To have an appreciation of the purpose and importance of Geometric Dimensioning and Tolerancing (GD&T)
- To understand and apply the **basic symbols and principles** of GD&T in engineering drawings.
- To be aware of the **key GD&T concepts** such as datums, tolerance zones, and feature control frames.
- To recognise and interpret **common tolerances** (e.g., straightness, flatness, circularity, cylindricity, perpendicularity, parallelism).

Assembly examples of multiple components

- An overall assembly will have slight differences from one assembly to another.
- These differences can be considered at the design stage in order to help ensure adequate and predictable performance.



How tolerancing Works?

- Engineering drawings need to show the dimensions for all features of a part. Next to the dimensions, a **tolerance** value needs to be specified with the **minimum** and **maximum** acceptable **limit**.
  - if we have a table that we would accept with a height between 750 mm and 780 mm, the tolerance would be 30 mm. (is it enough to produce an acceptable table?)

Geometric Dimensioning and Tolerancing

- GD&T is a system for defining and communicating **design intent** and **engineering tolerances** that helps engineers and manufacturers optimally control variations in manufacturing processes.
- GD&T is crucial for functional assemblies, multi-part products, or parts with complex functionality.
- **Describing product geometry** related to its intended functionality and manufacturing approach is ultimately simpler than having to describe everything in linear dimensions.

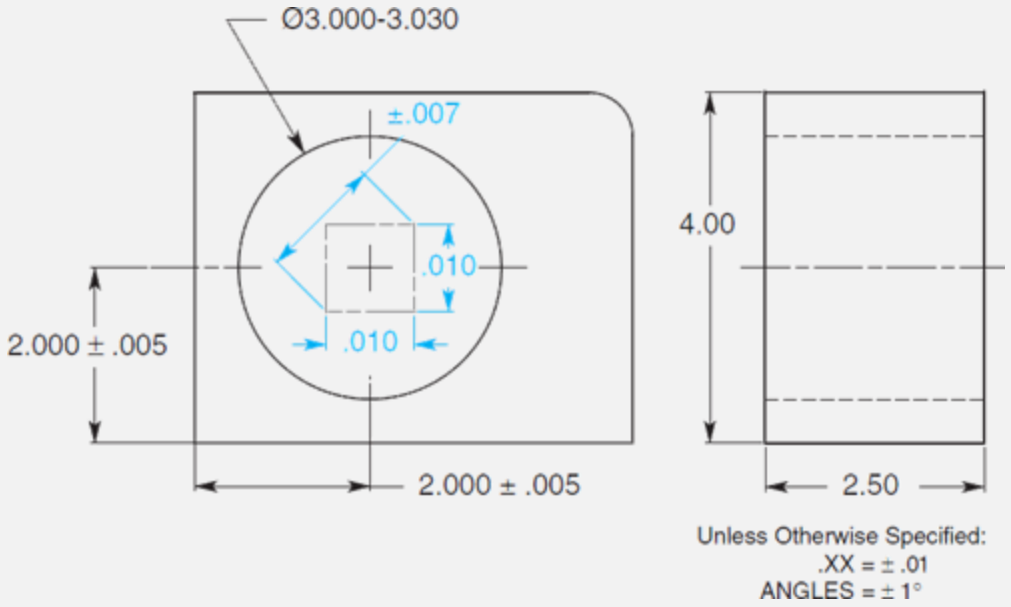
What Is GD&T?

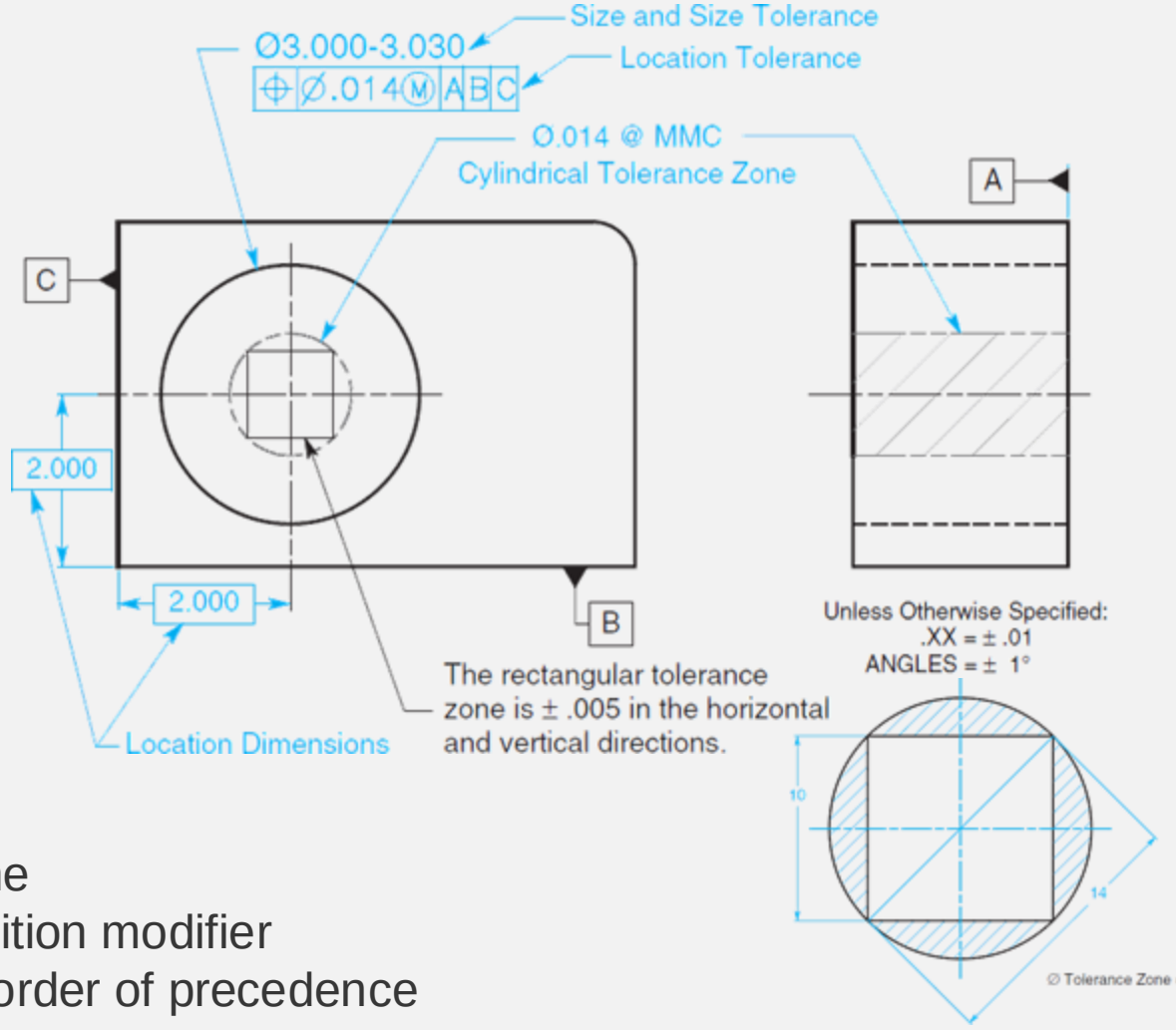
- GD&T is a **symbolic language.**
  - to specify the size, shape, form, orientation, and location of features on a part.
  - to ensure the proper assembly of mating parts, to improve quality, and to reduce cost.
- GD&T is a **design tool.**
  - serves as a checklist to remind the designer to consider all aspects of each feature
  - allows the designer to specify the maximum available tolerance and, consequently, design the most economical parts.
- GD&T **communicates design requirements.**
  - identifies all applicable datum features, and the features being controlled to these features
  - A properly toleranced drawing not only communicates the size and shape of the part but also tells a story that explains the tolerance relationships between features.

When Should GD&T Be Used?

- Because GD&T was designed to position features of size, the simplest answer is to locate **all features of size with GD&T controls**.
- Designers should tolerance parts with GD&T when:
  - Drawing delineation and interpretation need to be the same
  - Features are critical to function or interchangeability
  - It is important to stop scrapping perfectly good parts
  - It is important to reduce drawing changes
  - Automated equipment is used
  - Functional gaging is required
  - It is important to increase productivity
  - Companies want across-the-board savings

Advantages of **Geometric** D&T over Coordinate D&T

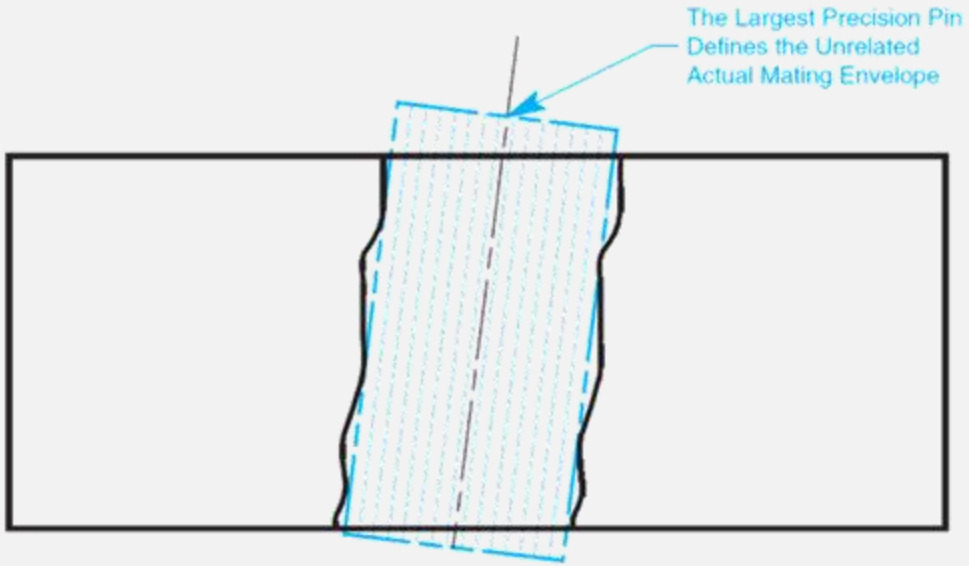


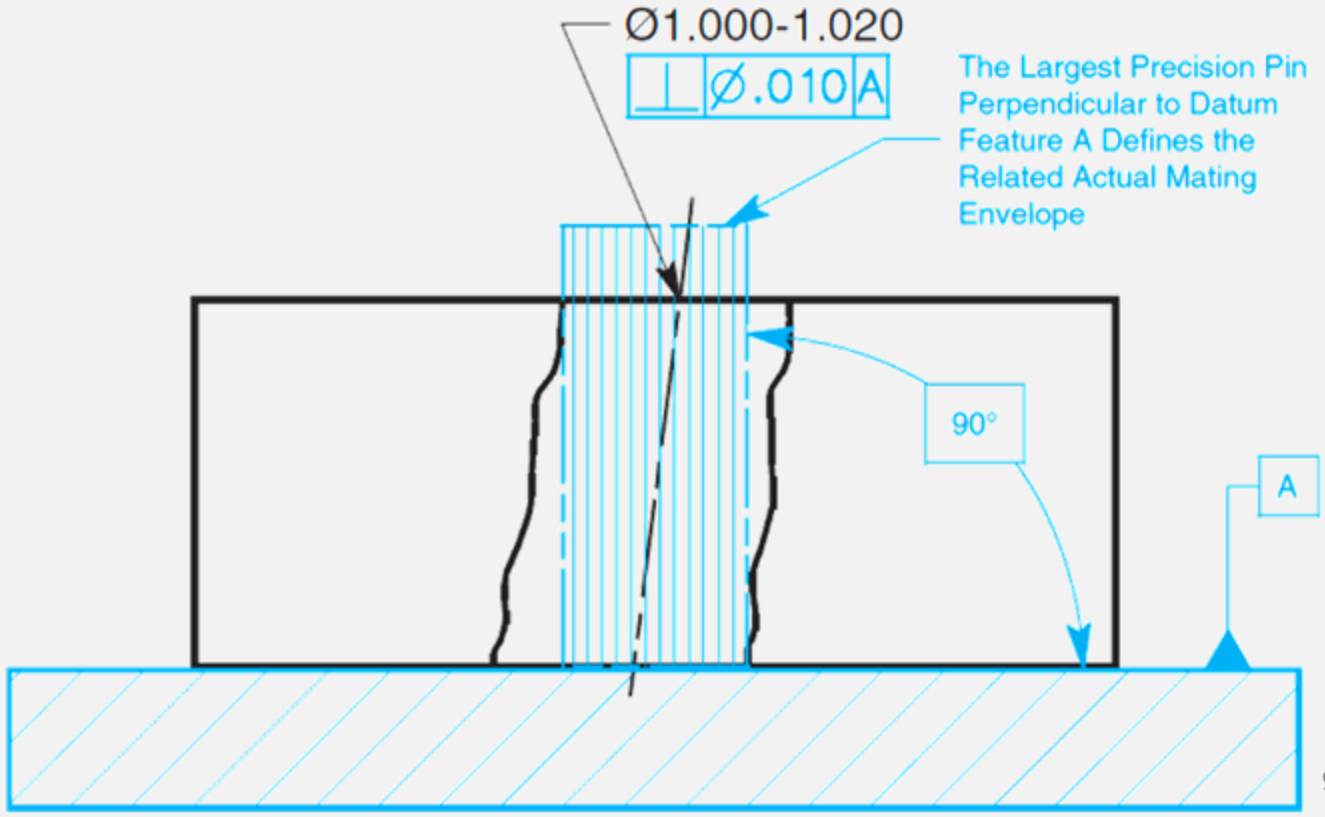


Advantages:
- The cylindrical tolerance zone
- The maximum material condition modifier
- Datum features specified in order of precedence

GD&T Terms

- Actual Mating Envelope
  - Unrelated and Related AME





GD&T Terms

- Actual Mating Envelope
  - Unrelated and Related AME
- Basic Dimension & Ture Position
- Datum
- Feature
- Limits of Size
  - Maximum Material Condition (MMC) and Least Material Condition (LMC)
- …

Specifying Linear Dimensions

- A **zero** is placed **before the decimal point** for values less than 1 mm.
- **Zeros are not added to the right of the decimal point** where dimensions are a whole number plus some decimal fraction of a millimeter.
- Neither a decimal point nor a zero is shown where the dimension is a whole number.

Correct (mm)	**Incorrect**
0.25	.25
4.5	4.500
4	4.000

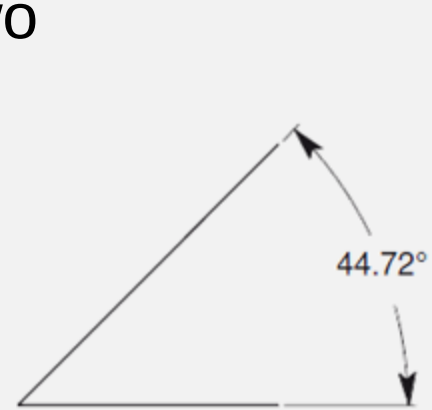
Specifying Linear Tolerances

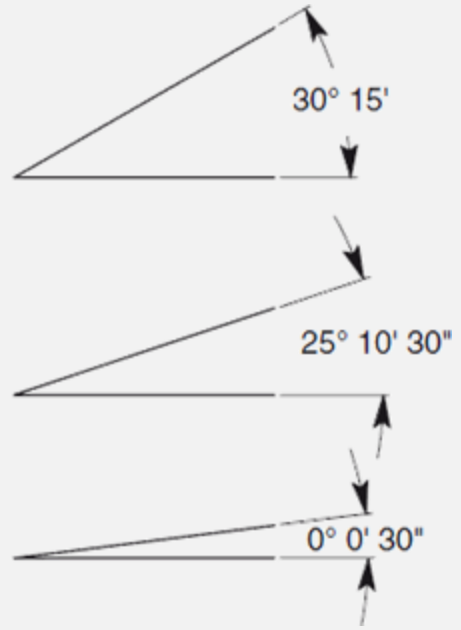
- Where a unilateral tolerance is specified and either the plus or minus limit is zero, a **single zero** is shown, and **no** plus or minus **sign** is used.
- Where unequal bilateral tolerancing is specified, both tolerance values have the **same number of decimal places**. Zeros are added where necessary.
- Where limit tolerancing is used, both values have the same number of decimal places, even if zeros need to be added after the decimal point.

Dimension	Tolerance	
	Correct	Incorrect
45	**0**	+.00
	-0.05	-0.05
60	**+0.25**	+.25
	-0.10	-.1
120	**+1.25**	+1.25
	-1.00	-1

Specifying Angular Dimensions

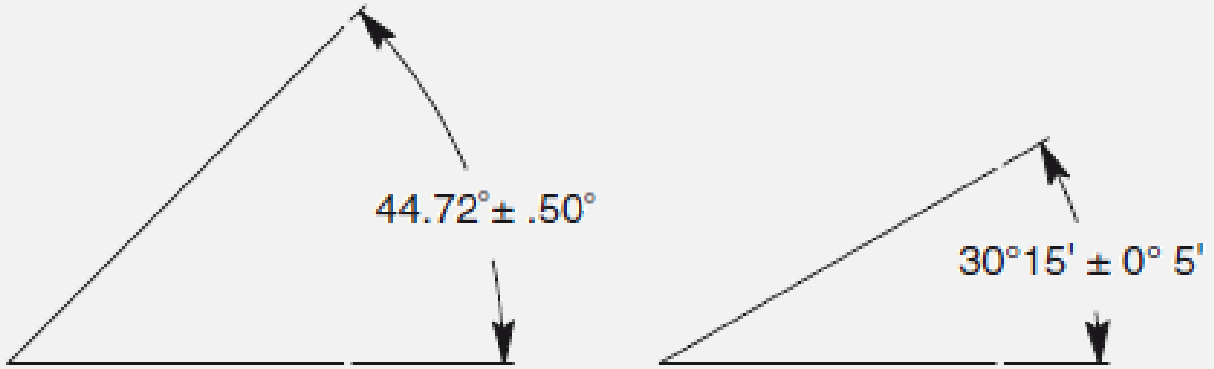
- Angular dimensions are specified in either of two conventions :
  - Degrees and decimal parts of a degree (44.72°)
  - Degrees (°), minutes (′), and seconds (″)
- Where only degrees are assigned, the value is followed with the degree symbol (60°).
- Where only minutes or seconds are specified, the number shall be preceded by zero degrees and/or zero minutes (0° 10′), (0° 0′ 30″).
- Features appearing to be 90° on the drawing are at an implied dimension of 90°.





Specifying Angular Tolerances

- Where specifying angle tolerances on drawings in terms of degrees and decimal fractions of a degree, the angle and the plus and minus tolerance values are written with the same number of decimal places.
- Where specifying angle tolerances in terms of degrees and minutes, the angle and the plus and minus tolerance values are written in degrees and minutes, even if the number of degrees is zero.



Geometric Charact. Symbols

- Geometric characteristic symbols are the essence of this graphic language. It is important not only to know each symbol but also to know how to apply these symbols on drawings.
- The **12** geometric characteristic symbols are divided into five categories.

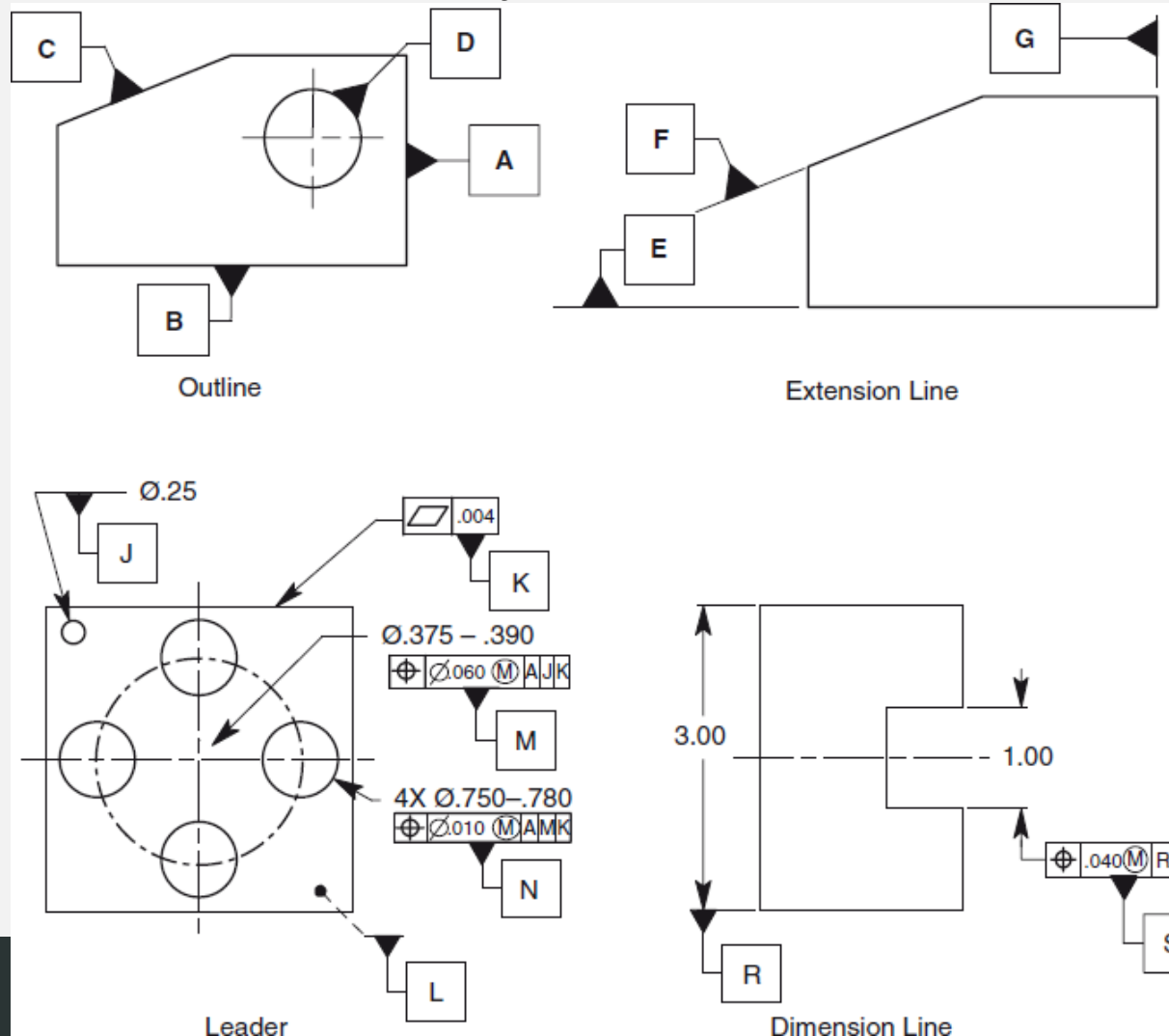
The concentricity and symmetry controls have been **deleted** from the ASME Y14.5-2018 standard.

Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
Individual Feature Only	**Form**	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY	
		CYLINDRICITY	
Individual Feature or Related Features	**Profile**	PROFILE OF A LINE	
		PROFILE OF A SURFACE	
Related Features	**Orientation**	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	Location	POSITION	
	Runout	CIRCULAR RUNOUT	
		TOTAL RUNOUT	

Datum Feature Symbol

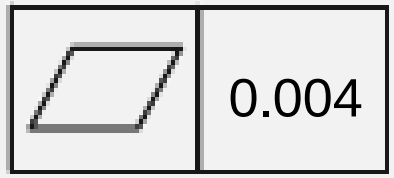
- The **datum feature symbol** is used to identify physical features of a part as datum features.
- The datum feature symbol consists of a **capital letter** enclosed in a **square box**. It is connected to a **leader** directed to the datum feature ending in a **triangle**.
- The datum feature identifying letters may be any letter of the alphabet except I, O, and Q
- Datum feature symbols **must not** be attached to centerlines, center planes, or axes

Methods of attaching datum feature symbols to datum features.



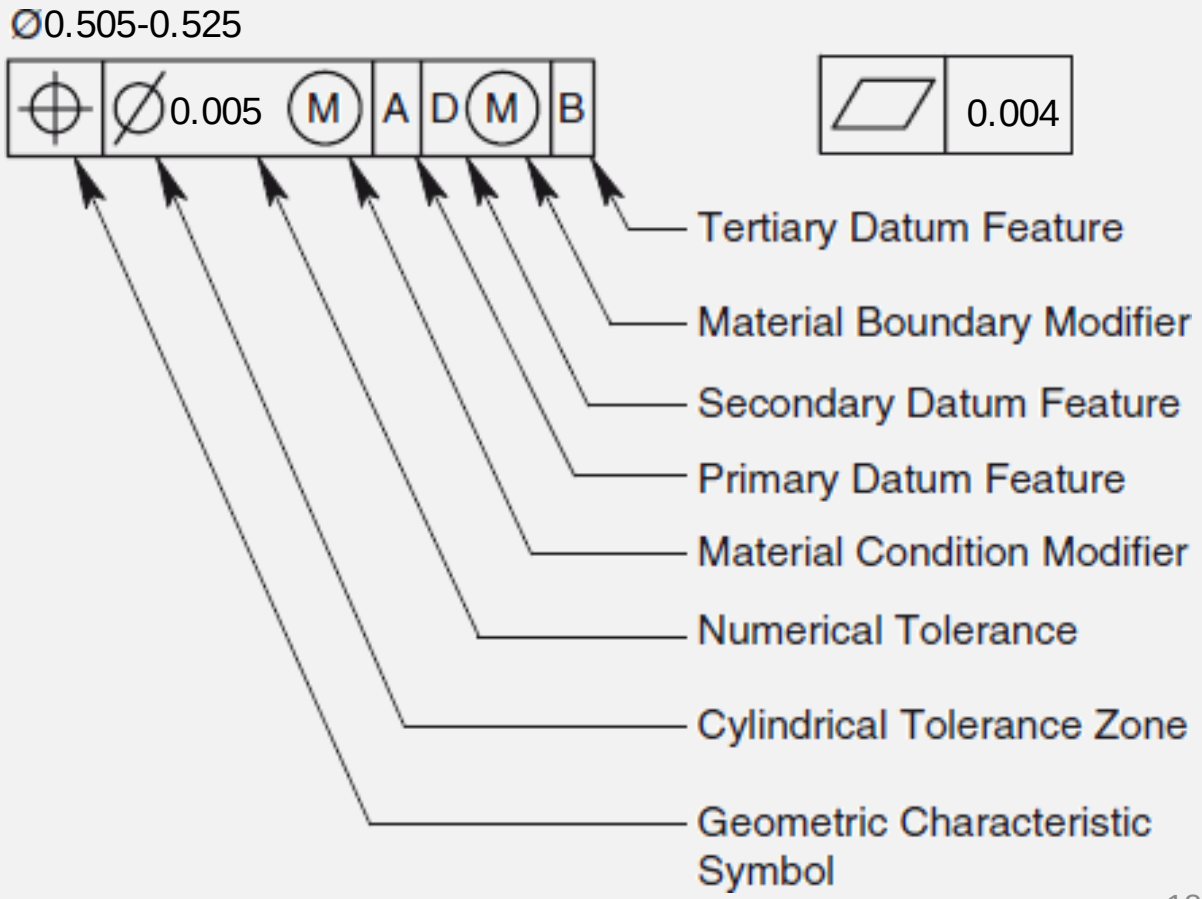
Feature Control Frame

- The feature control frame in the GD&T language is like a sentence in the English language—it is a complete tolerancing.
- All the geometric tolerancing for a feature, or pattern of features, is contained in **one or more** feature control frames.
- Just as in any other language, the feature control frame must be written **properly and completely**.
- The simplest feature control frame is one like the flatness control on the right. It consists of the geometric characteristic symbol for flatness and the flatness tolerance.

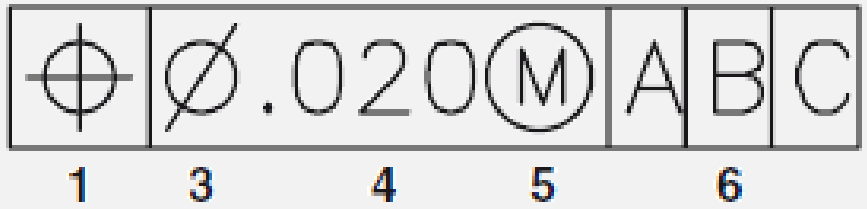


Reading the Feature Control Frame

- One of the 14 geometric characteristic symbols always appears in the first compartment of the feature control frame.
- The second compartment is the tolerance section.
- The final section of the feature control frame is reserved for datum features.

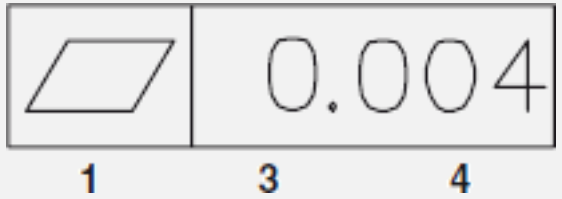


Reading the Feature Control Frame

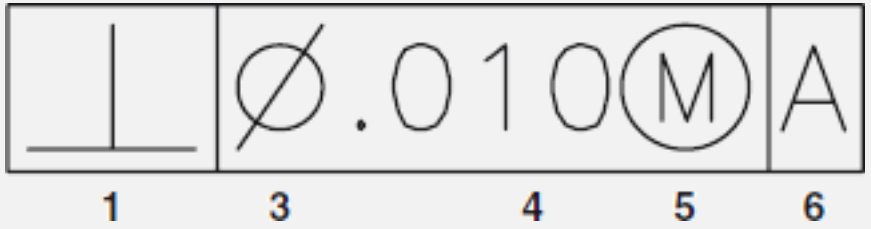


1. The geometric characteristic symbol
2. **The feature's controlled element** (determined by the geometry of the feature shown on the print, not in the feature control frame)
3. The tolerance zone shape
4. The tolerance zone size
5. Any tolerance zone modifier(s)
6. The datum feature(s)

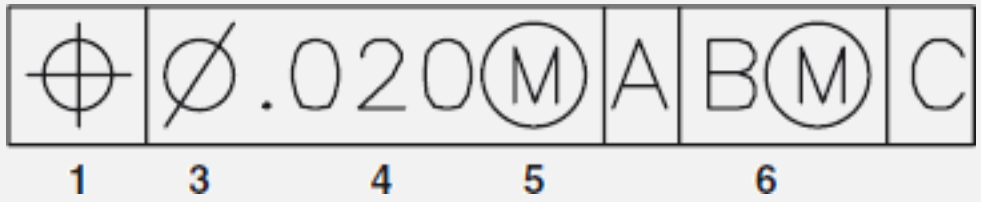
Can you read this?



Can you read this?



Can you read this?

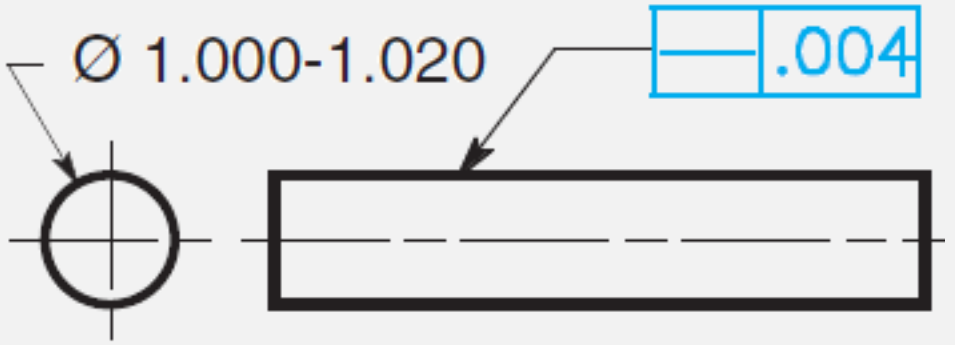


Form Tolerance

Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
Individual Feature Only	**Form**	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY	
		CYLINDRICITY	
Individual Feature or Related Features	**Profile**	PROFILE OF A LINE	
		PROFILE OF A SURFACE	
Related Features	**Orientation**	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	Location	POSITION	
	Runout	CIRCULAR RUNOUT	
		TOTAL RUNOUT	

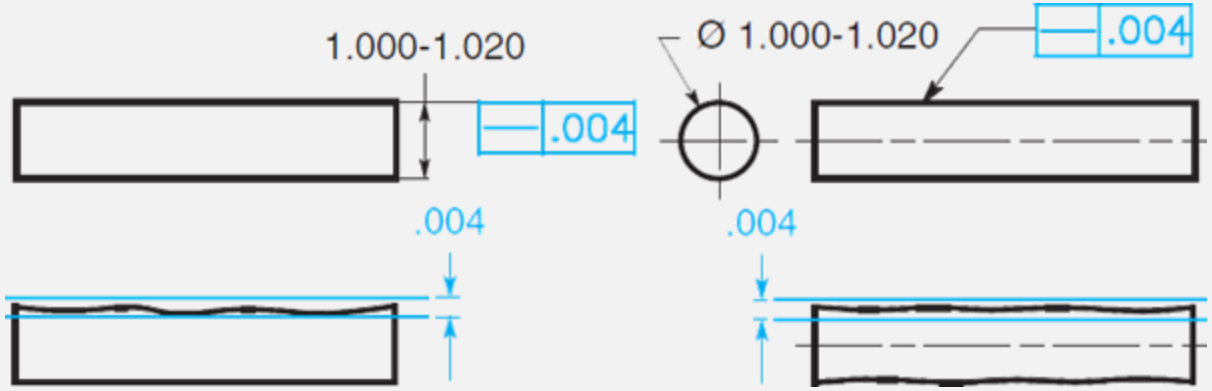
Straightness Tolerance

- Straightness is a condition where an element of a surface, or a derived median line, is a straight line.
- In a **view** where the **line** elements to be controlled **appear as a line**, a feature control frame is attached to the surface with a **leader or extension line**.
- Straightness tolerance is a refinement of the size tolerance and must be **less than** the size tolerance.



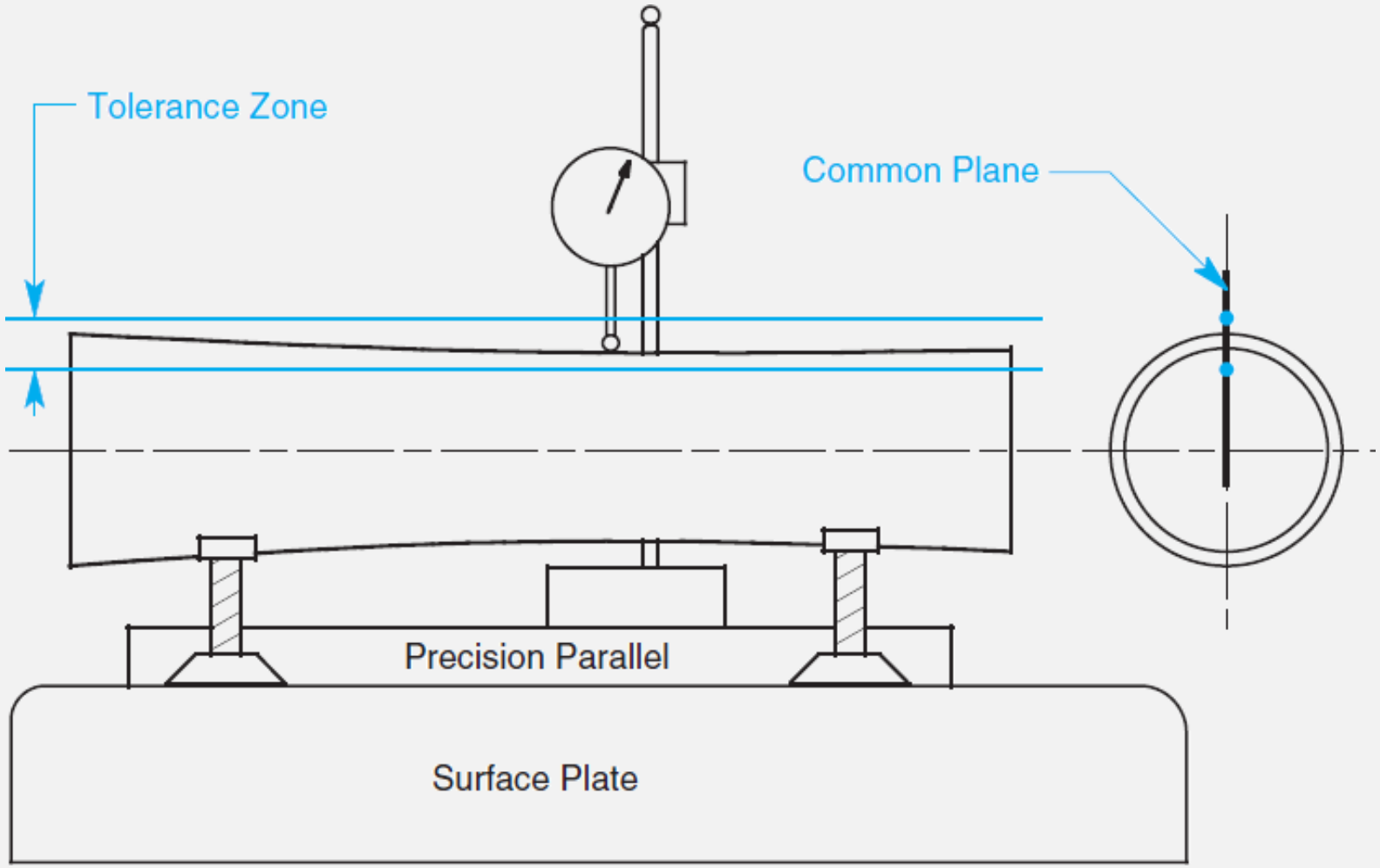
Interpretation of Straightness Tolerance

- The line elements being controlled must lie within a **tolerance zone** that consists of **two parallel lines** separated by the straightness tolerance of 0.004 specified in the feature control frame and **parallel to the view** in which they are specified.
- In addition, the line elements must fall within the size tolerance of 0.020.



Actual Size	Straightness Tolerance
1.010	
1.016	
1.018	
1.020	

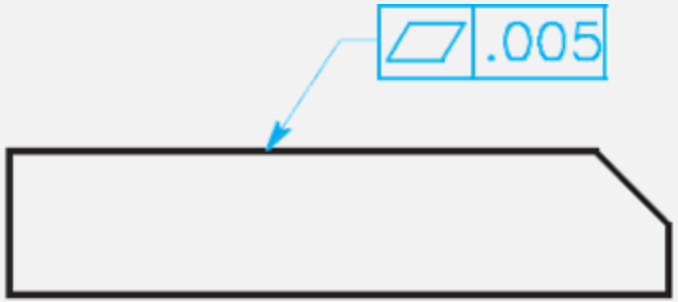
Inspection of Straightness Tolerance



There are an infinite number of line elements on any surface. The inspector must measure a sufficient number of line elements to be convinced that all line elements fall within the tolerance specified.

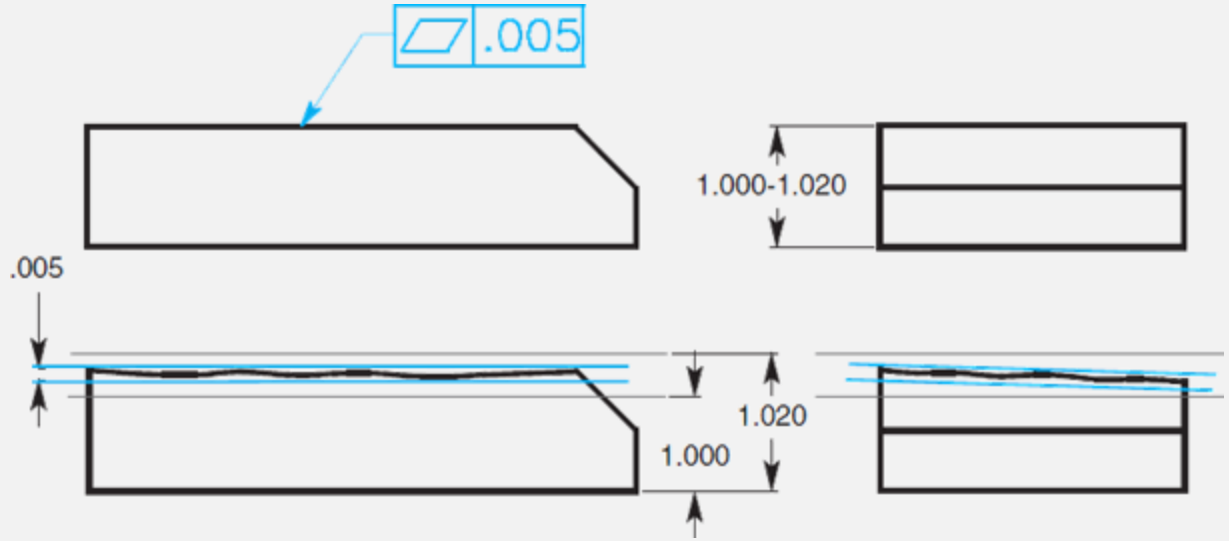
Flatness Tolerance

- Flatness is the condition of a surface or derived median plane having all elements in one plane.
- In a **view** where the **surface** to be controlled **appears as a line**, a feature control frame is attached to the surface with a **leader or extension line.**
- Flatness tolerance is a refinement of the size tolerance and must be **less than** the size tolerance.



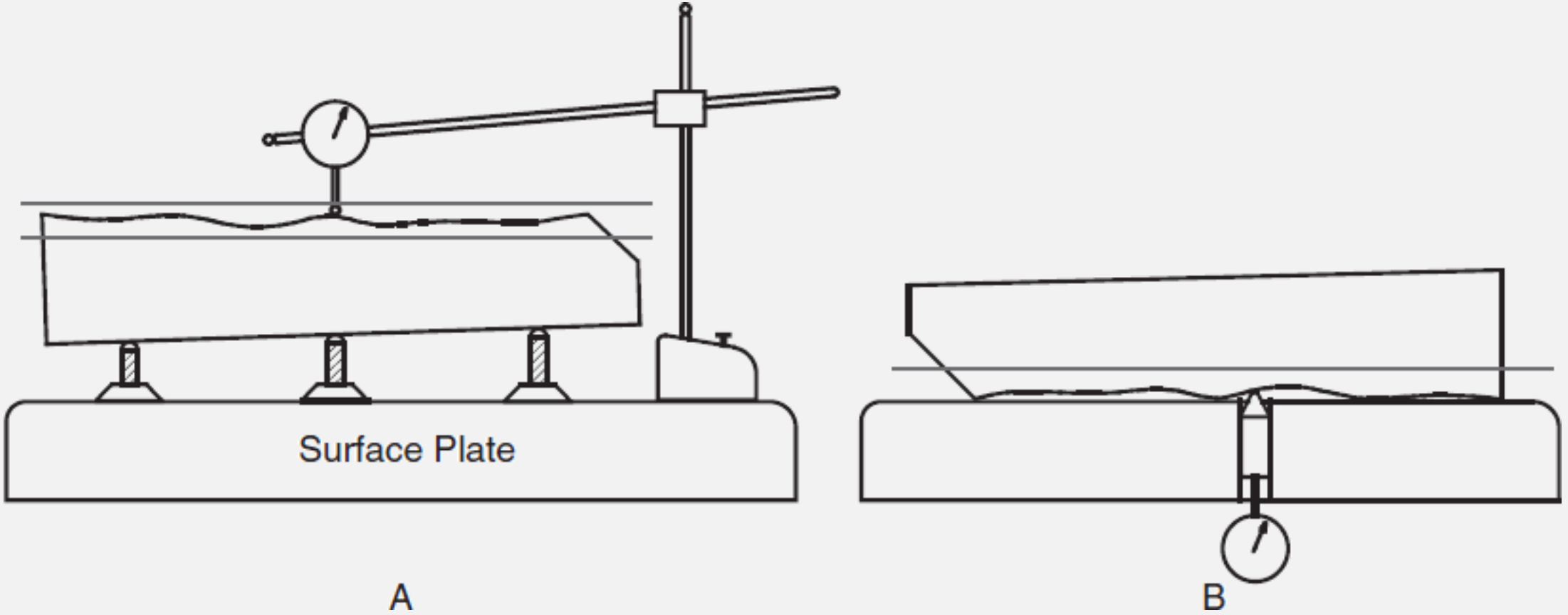
Interpretation of Flatness Tolerance

- The surface being controlled for flatness must lie between two **parallel planes** separated by the flatness tolerance of 0.005 specified in the feature control frame.
- In addition, the surface must fall within the size tolerance, the two parallel planes 0.020 apart.



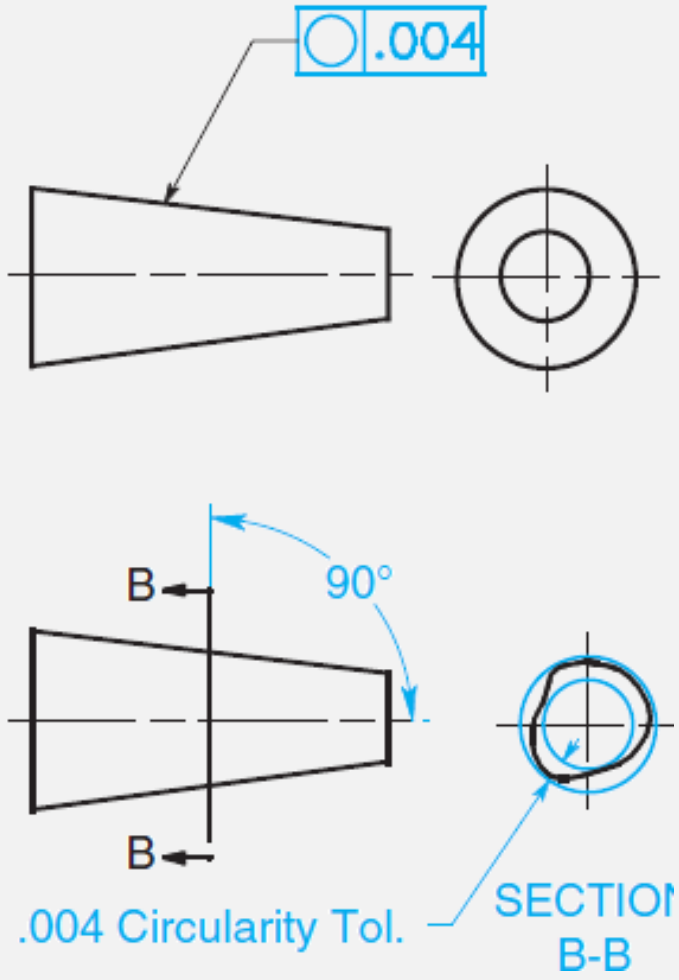
Actual Size	Flatness Tolerance
1.000	
1.008	
1.016	
1.020	

Inspection of Flatness Tolerance

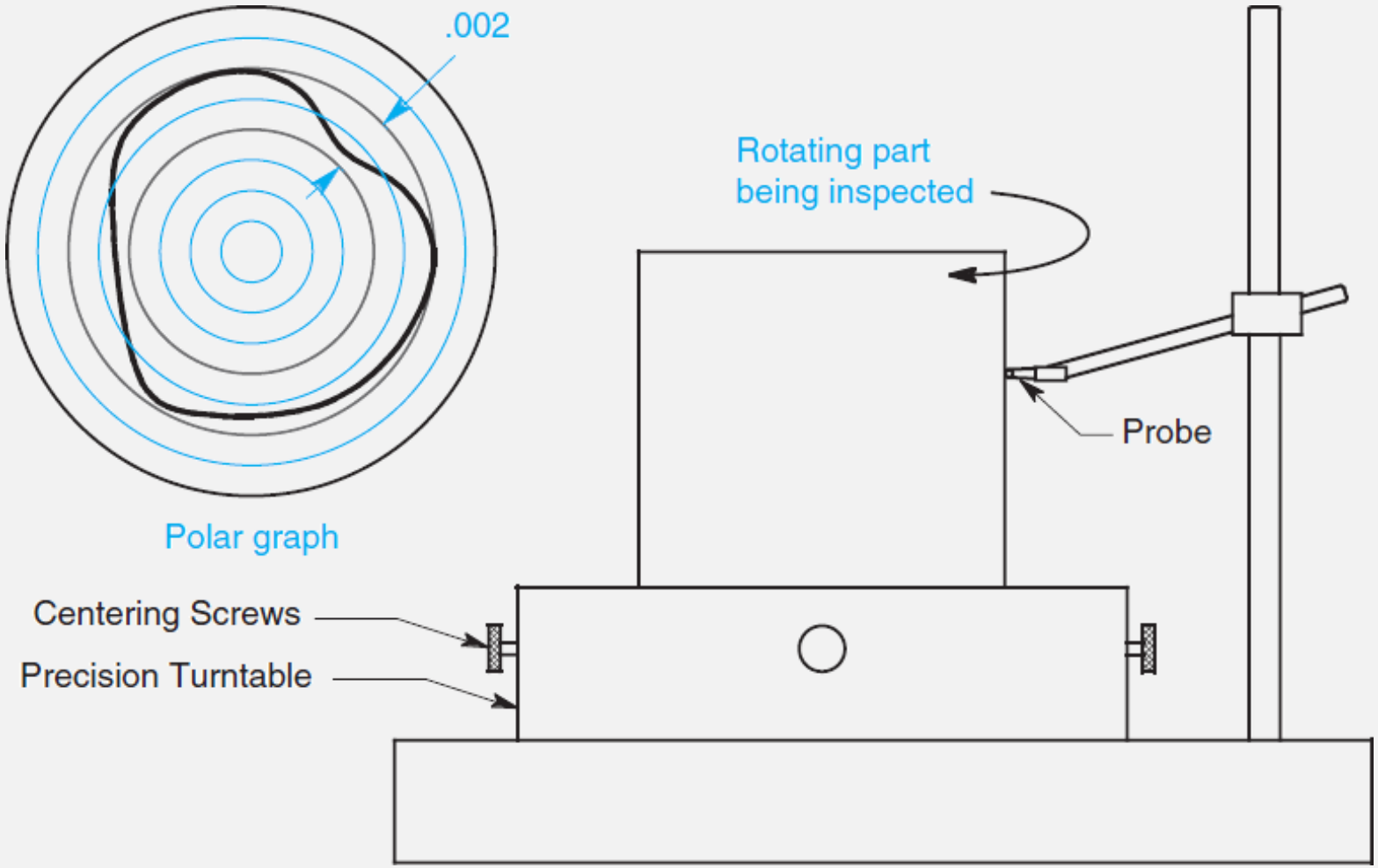


Circularity Tolerance

- Circularity has two definitions:
  - For a surface of revolution, all points of the surface intersected by a plane perpendicular to an axis or spine (curved line) are equidistant from that axis or spine.
  - For a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.
- Circularity tolerance is a refinement of the size tolerance and must be **less than** the size tolerance

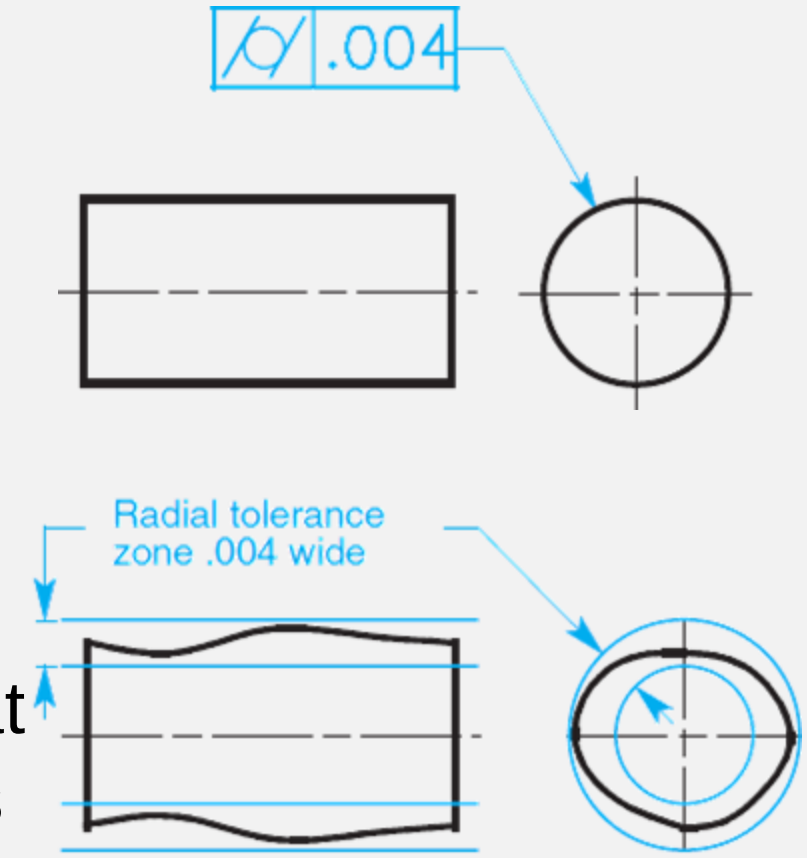


Inspection of Circularity Tolerance

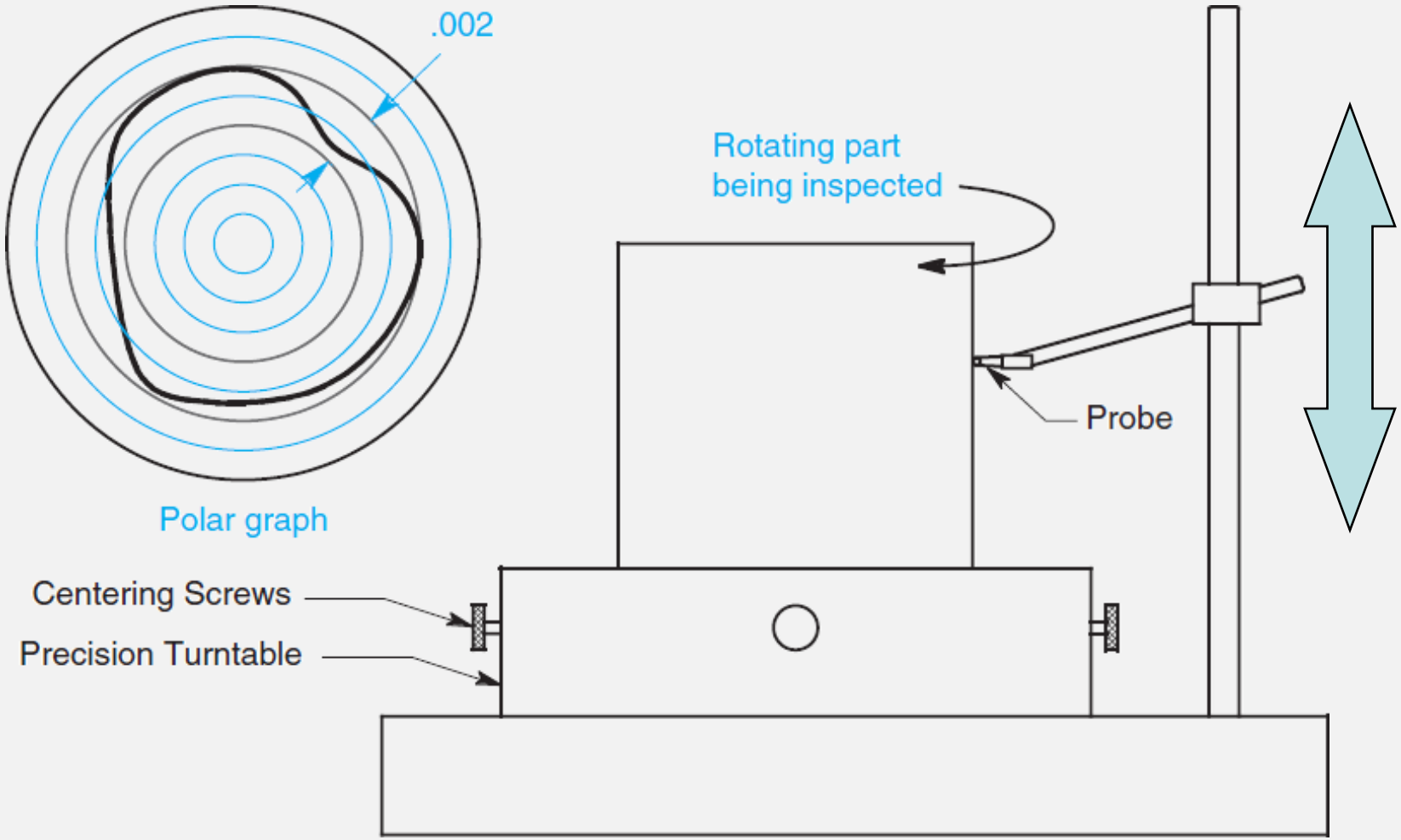


Cylindricity Tolerance

- Cylindricity is a condition of the surface of a cylinder where all points of the surface are equidistant from a common axis.
- Unlike circularity, the cylindricity tolerance applies to **both circular and longitudinal** elements of the surface at the same time.
- Cylindricity is a **composite form tolerance** that simultaneously controls circularity, straightness of a surface, and taper of cylindrical features.



Inspection of Cylindricity Tolerance



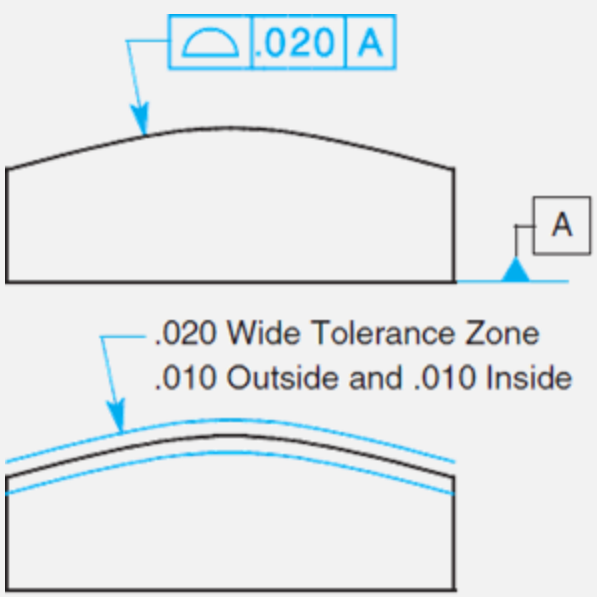
Profile Tolerance

Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
Individual Feature Only	**Form**	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY	
		CYLINDRICITY	
Individual Feature or Related Features	**Profile**	PROFILE OF A LINE	
		PROFILE OF A SURFACE	
Related Features	**Orientation**	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	Location	POSITION	
	Runout	CIRCULAR RUNOUT	
		TOTAL RUNOUT	

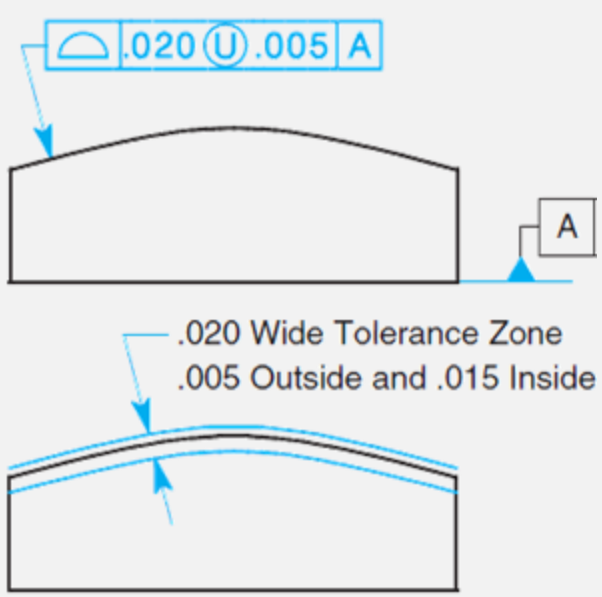
Profile Tolerance

- **Profile of a line** is the outline of an object in a plane as the plane passes through the object. **Profile of a surface** is the result of projecting the profile of an object on a plane or taking cross sections through the object at various intervals.
- **Profile tolerances** are used to define a tolerance zone to control size and form or combinations of size, form, orientation, and location of a feature(s) relative to a true profile.
  - The true profile is shown in a profile view or a section view of a part and may be dimensioned with basic **size dimensions**, basic **coordinate dimensions**, basic **radii**, basic **angular dimensions**, **formulas**, **mathematical data**, or **undimensioned** drawings.
- Profile controls are **surface controls**.
  - The shape of the tolerance zone is the exact shape of the profile, and it is as wide as the tolerance specified in the feature control frame.

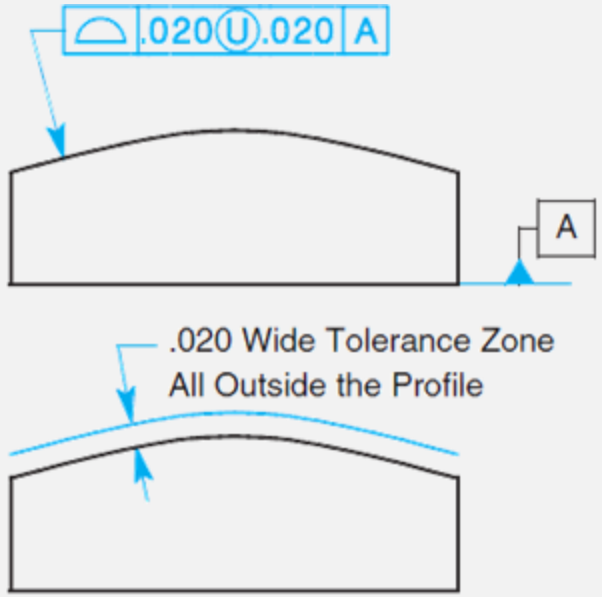
Specifying Profile Tolerance



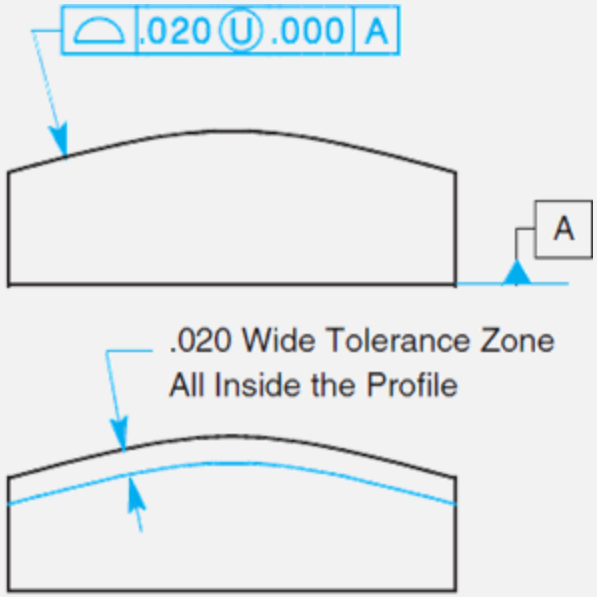
A. Bilateral Tolerance



B. Unequally Distributed Bilateral Tolerance



C. Unilateral Tolerance Outside



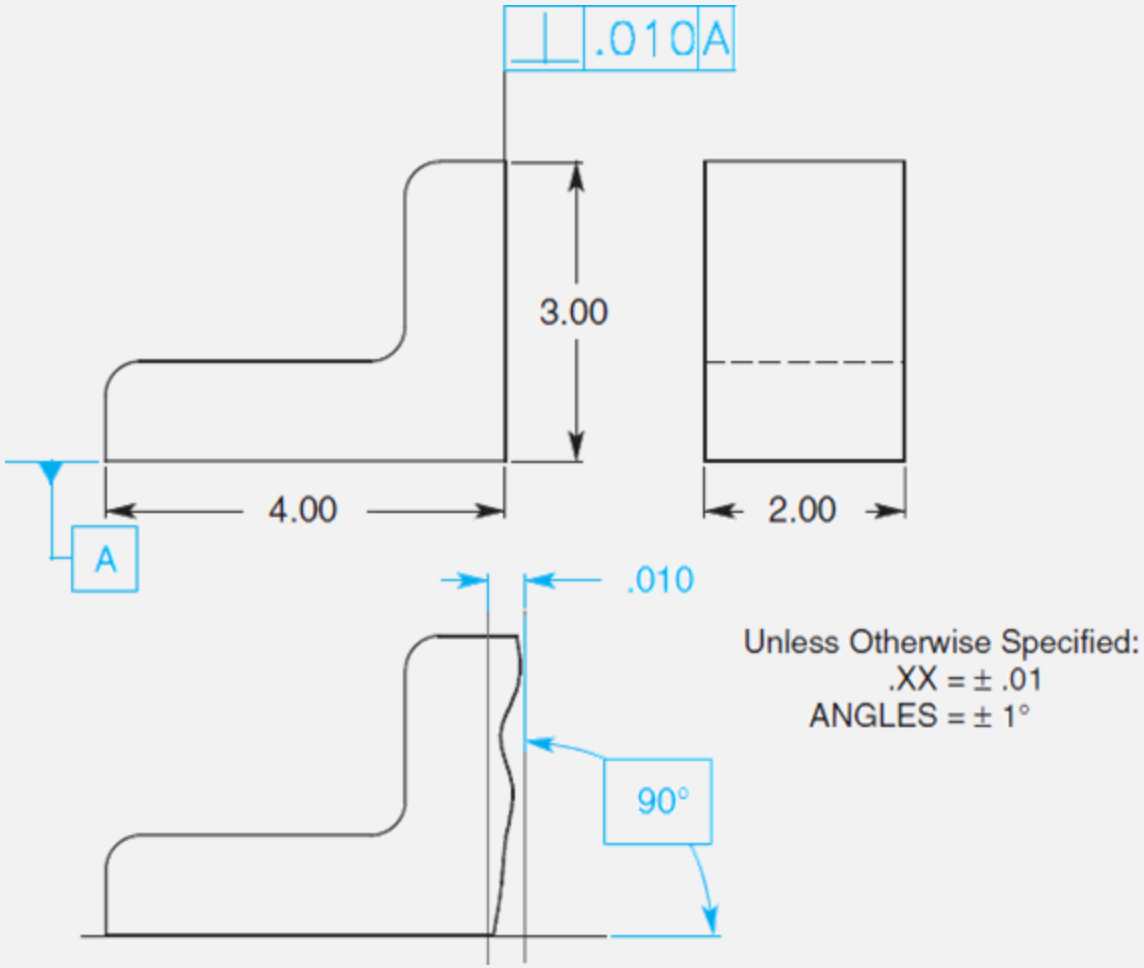
D. Unilateral Tolerance Inside

Orientation Tolerance

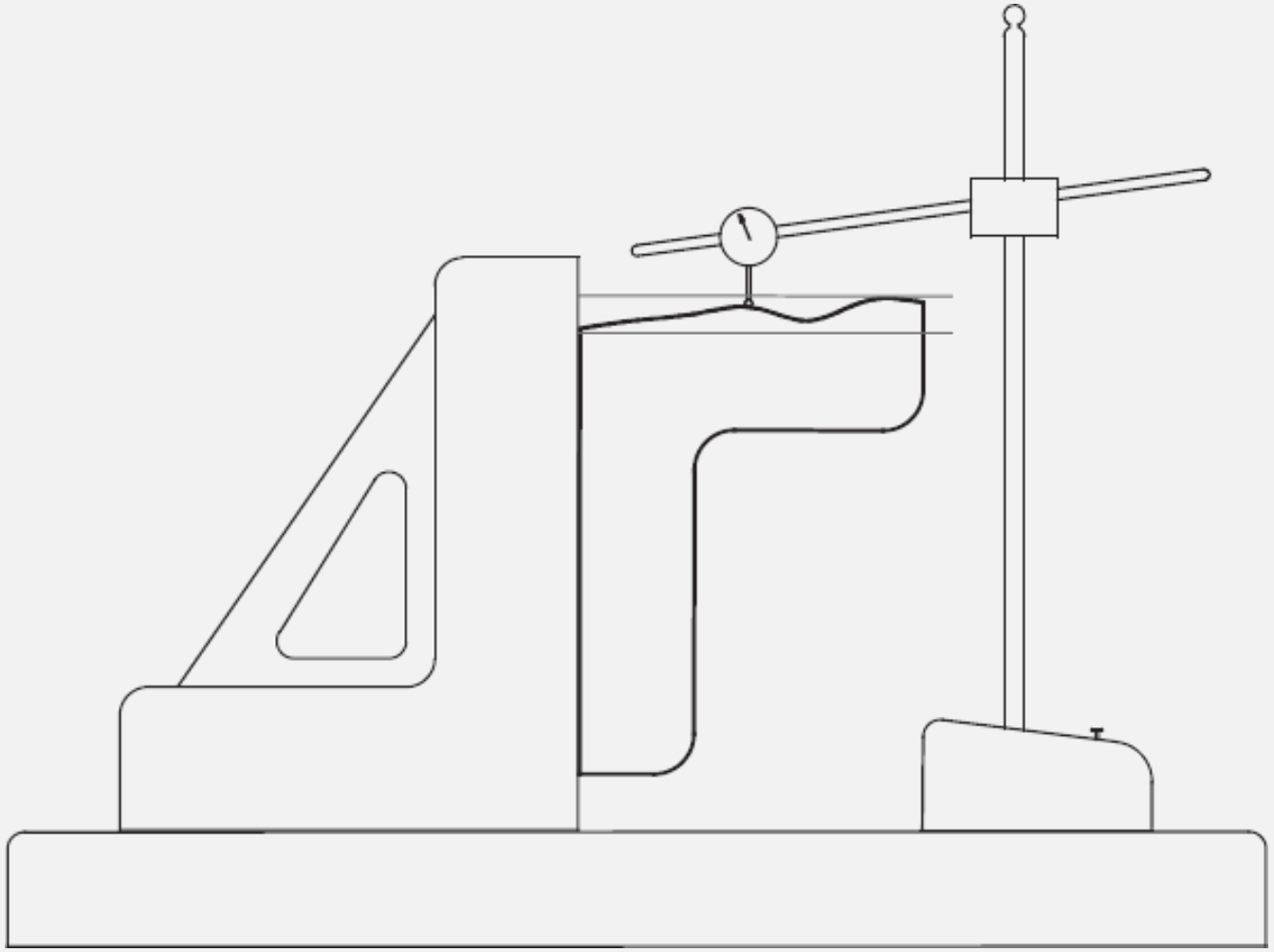
Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
Individual Feature Only	**Form**	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY	
		CYLINDRICITY	
Individual Feature or Related Features	**Profile**	PROFILE OF A LINE	
		PROFILE OF A SURFACE	
Related Features	**Orientation**	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	Location	POSITION	
	Runout	CIRCULAR RUNOUT	
		TOTAL RUNOUT	

Perpendicularity Tolerance

- Perpendicularity is the condition of a surface, axis, or center plane that is at a 90° angle to a datum plane or datum axis.
- The feature control frame contains a perpendicularity symbol, a numerical tolerance, and at least **one datum feature.**

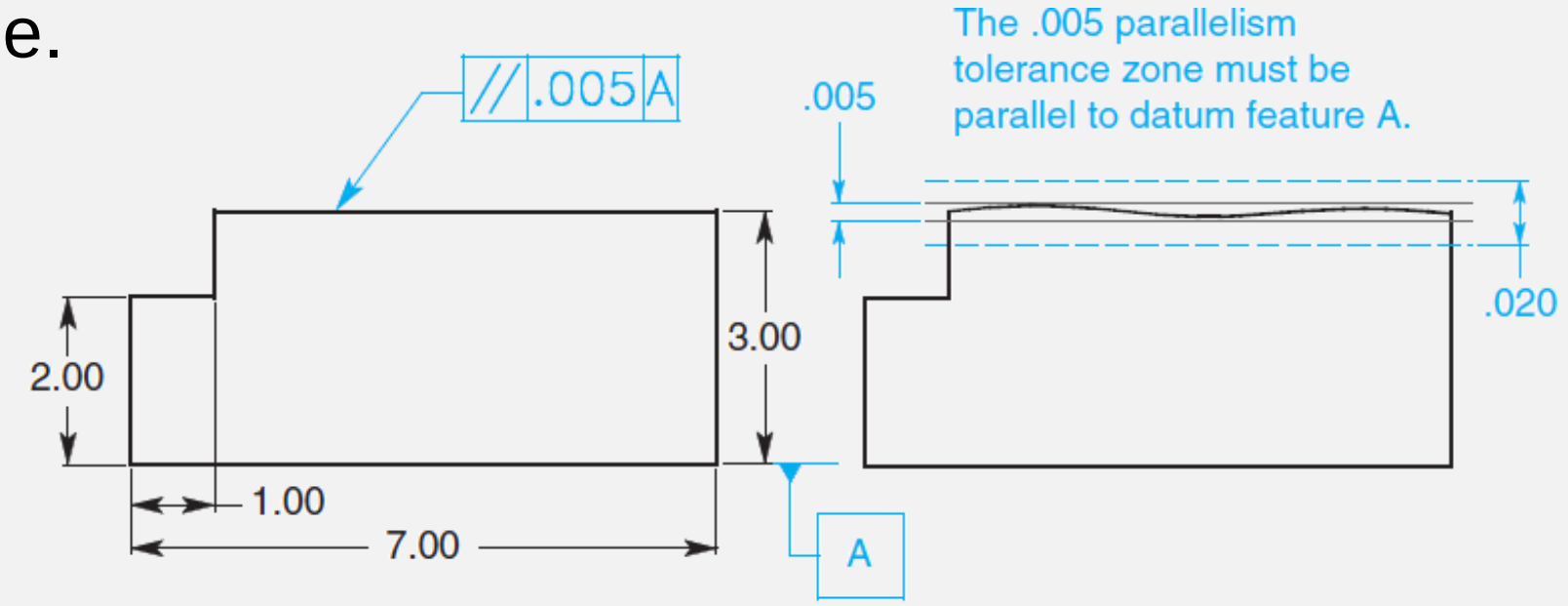


Inspection of Perpendicularity Tolerance

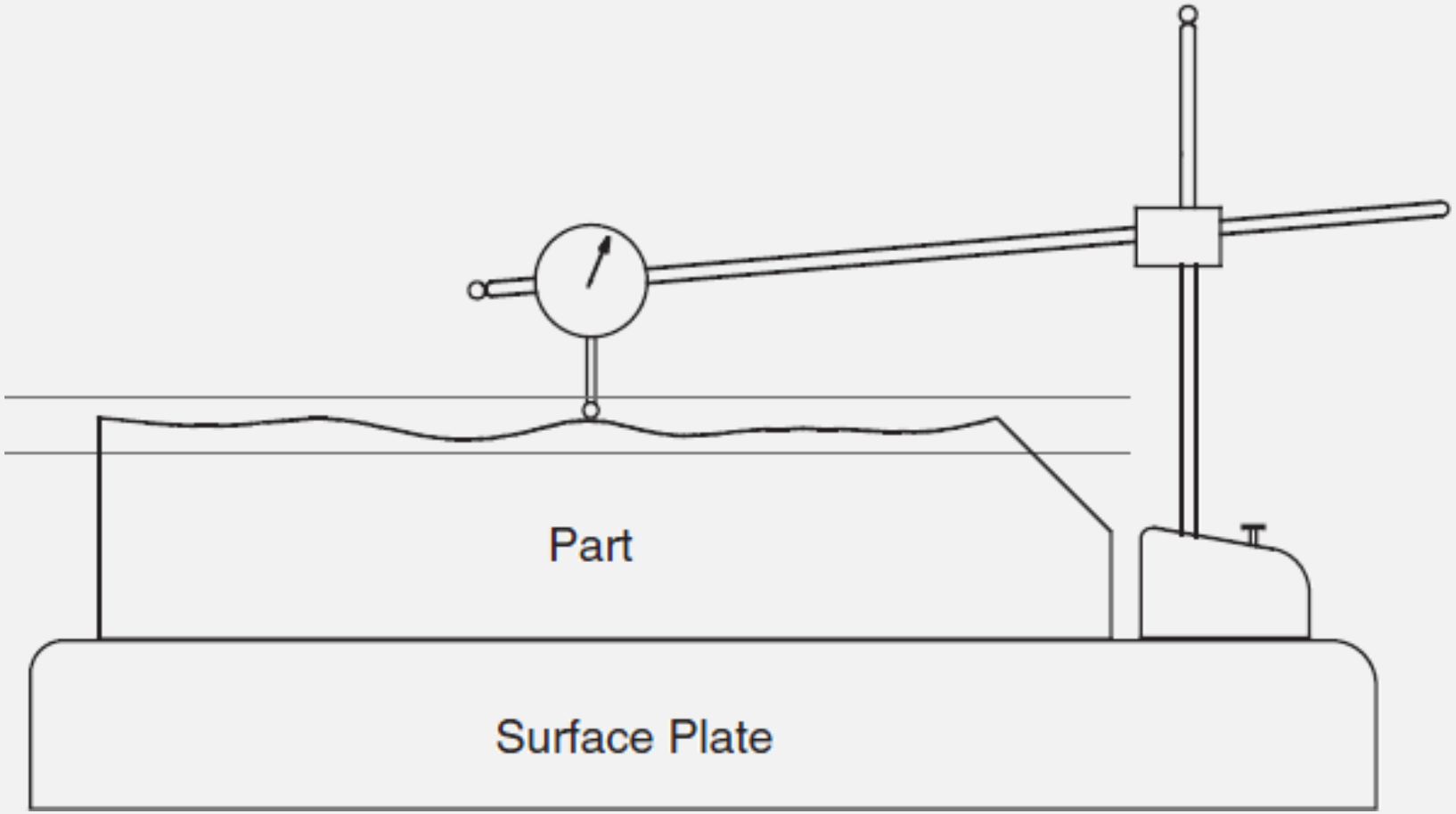


Parallelism Tolerance

- Parallelism ensures a surface, centre plane, or axis remains **equidistant** from a specified datum.
- It is a refinement of the size tolerance and must be **less than** the size tolerance.

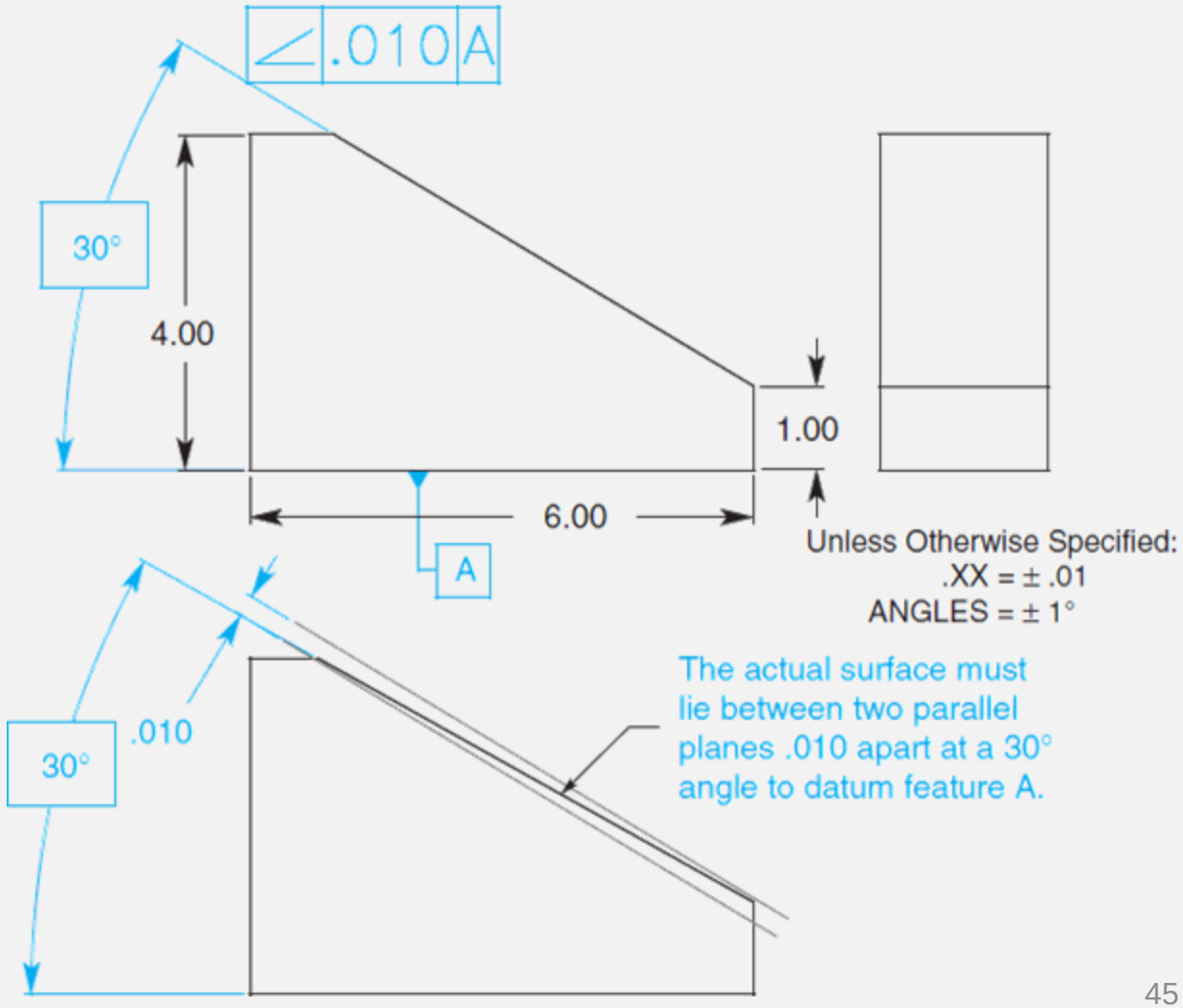


Inspection of Parallelism Tolerance

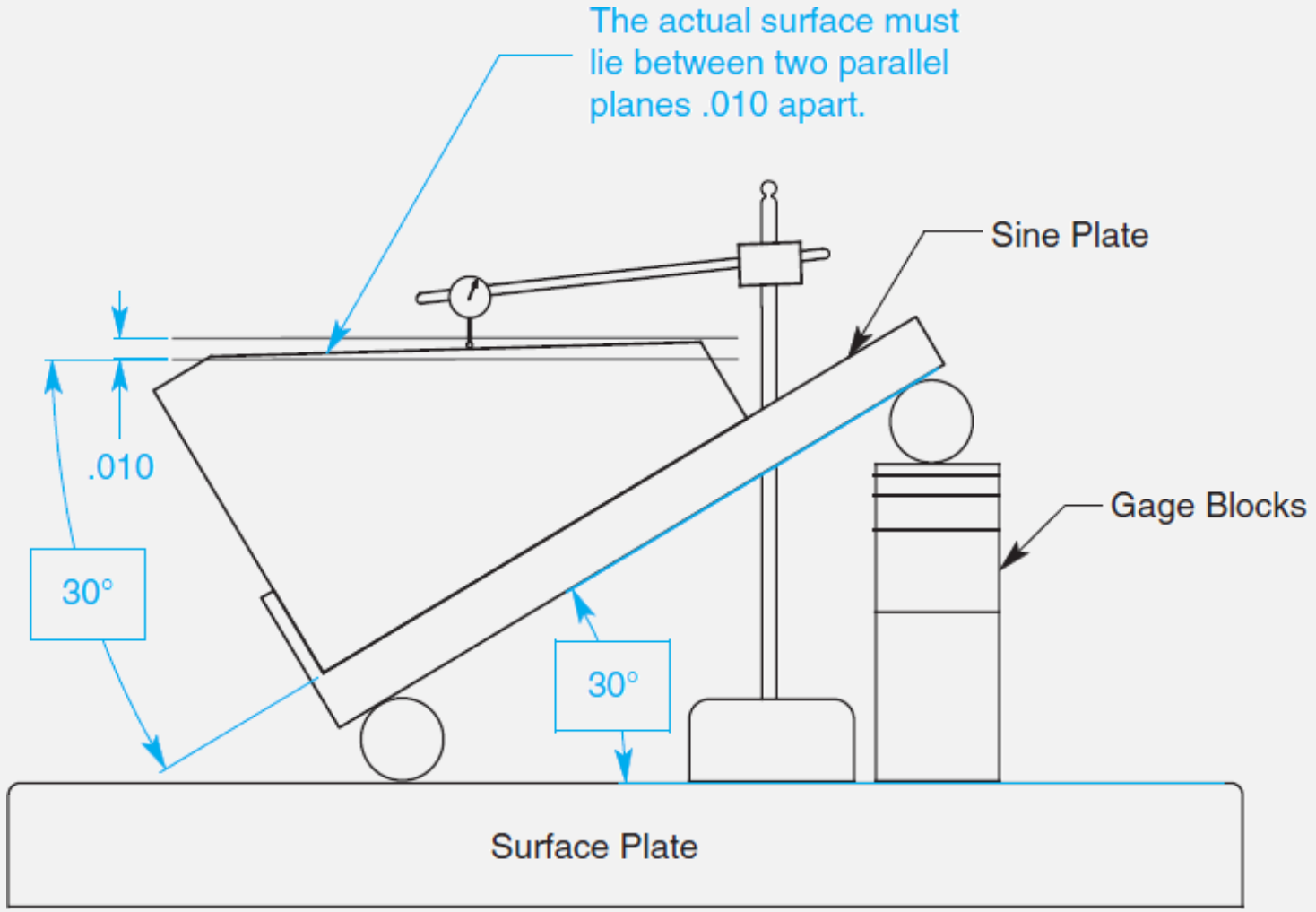


Angularity Tolerance

- Angularity is the condition of a surface, axis, or center plane at any specified angle from a datum plane or datum axis.
- The numerical tolerance for the surface being controlled is specified as a linear dimension because it generates a uniform shaped tolerance zone.



Inspection of Angularity Tolerance



Location Tolerance

Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
Individual Feature Only	**Form**	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY	
		CYLINDRICITY	
Individual Feature or Related Features	**Profile**	PROFILE OF A LINE	
		PROFILE OF A SURFACE	
Related Features	**Orientation**	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	Location	POSITION	
	Runout	CIRCULAR RUNOUT	
		TOTAL RUNOUT	

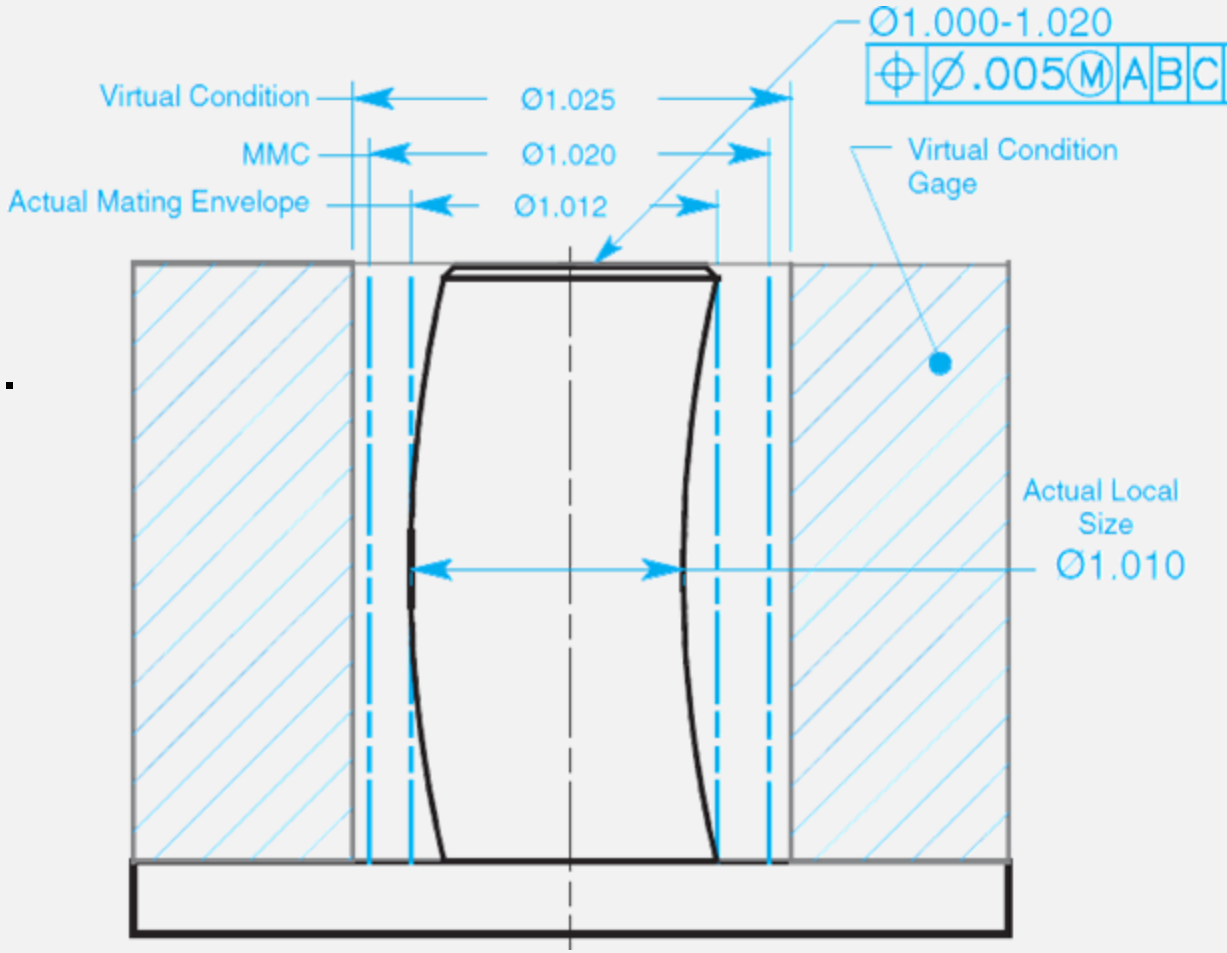
The **concentricity** and **symmetry** controls have been **deleted** from the ASME Y14.5-2018 standard.

Position Tolerance

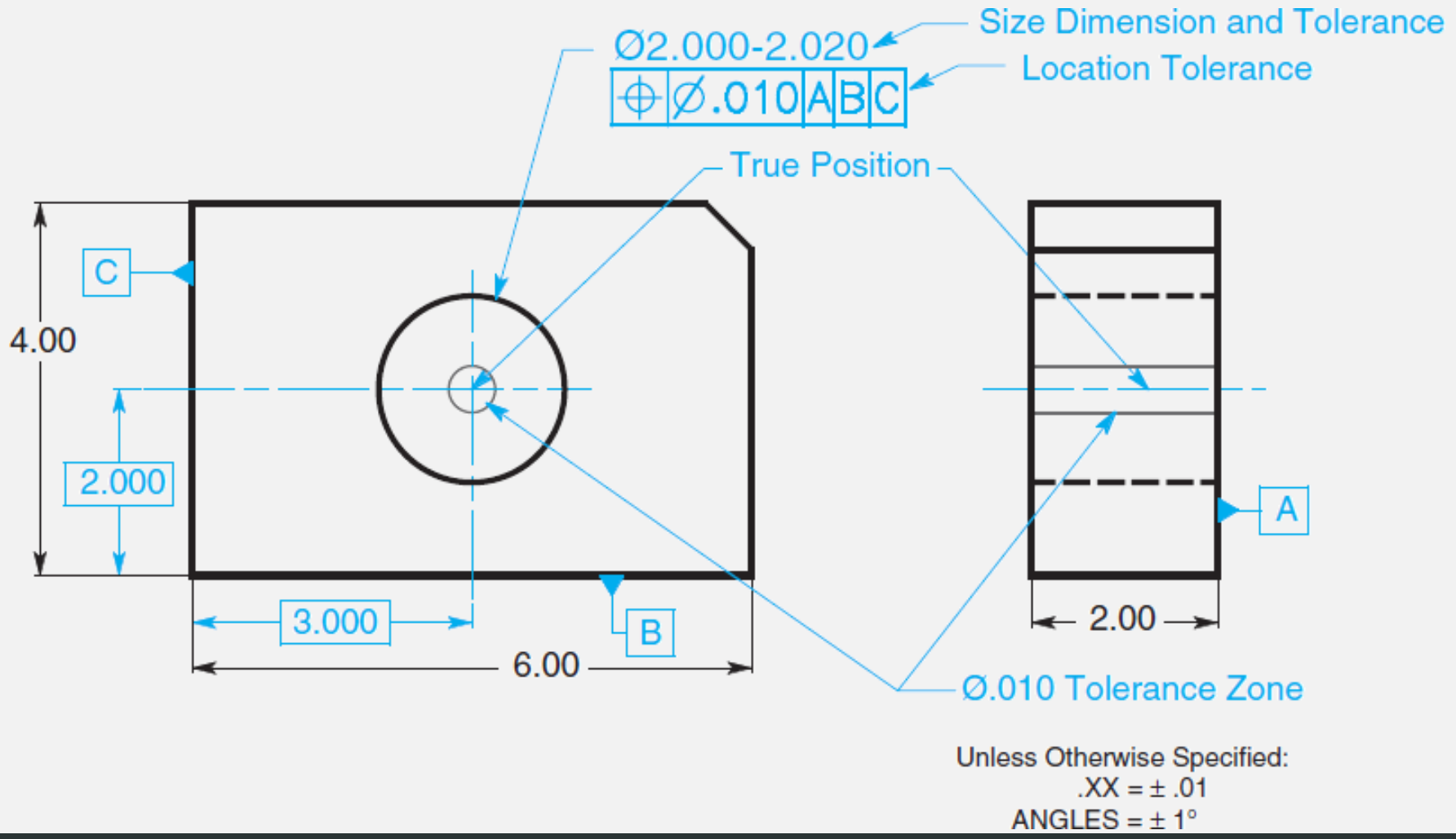
- Position tolerance is a **composite tolerance** that controls both the **location** and the **orientation** of features of sizes at the same time.
- The tolerance of position may be viewed in either of two ways:
  - A **theoretical tolerance zone** of the toleranced feature located at true position within which the center point, axis, or center plane of the feature may vary from true position.
  - A **virtual condition boundary** of the toleranced feature, when specified at MMC or LMC and located at true position, which may not be violated by the surface or surfaces of the considered feature of size.

Position Tolerance

- A feature of size has four geometric characteristics that must be controlled.
  - **size, form, orientation**, and **location**.
- The **limits of size** control **size** and **form**.
- The **position tolerance** controls **orientation** and **location**.
- The sum of the actual local size and any form error equals the actual mating envelope.

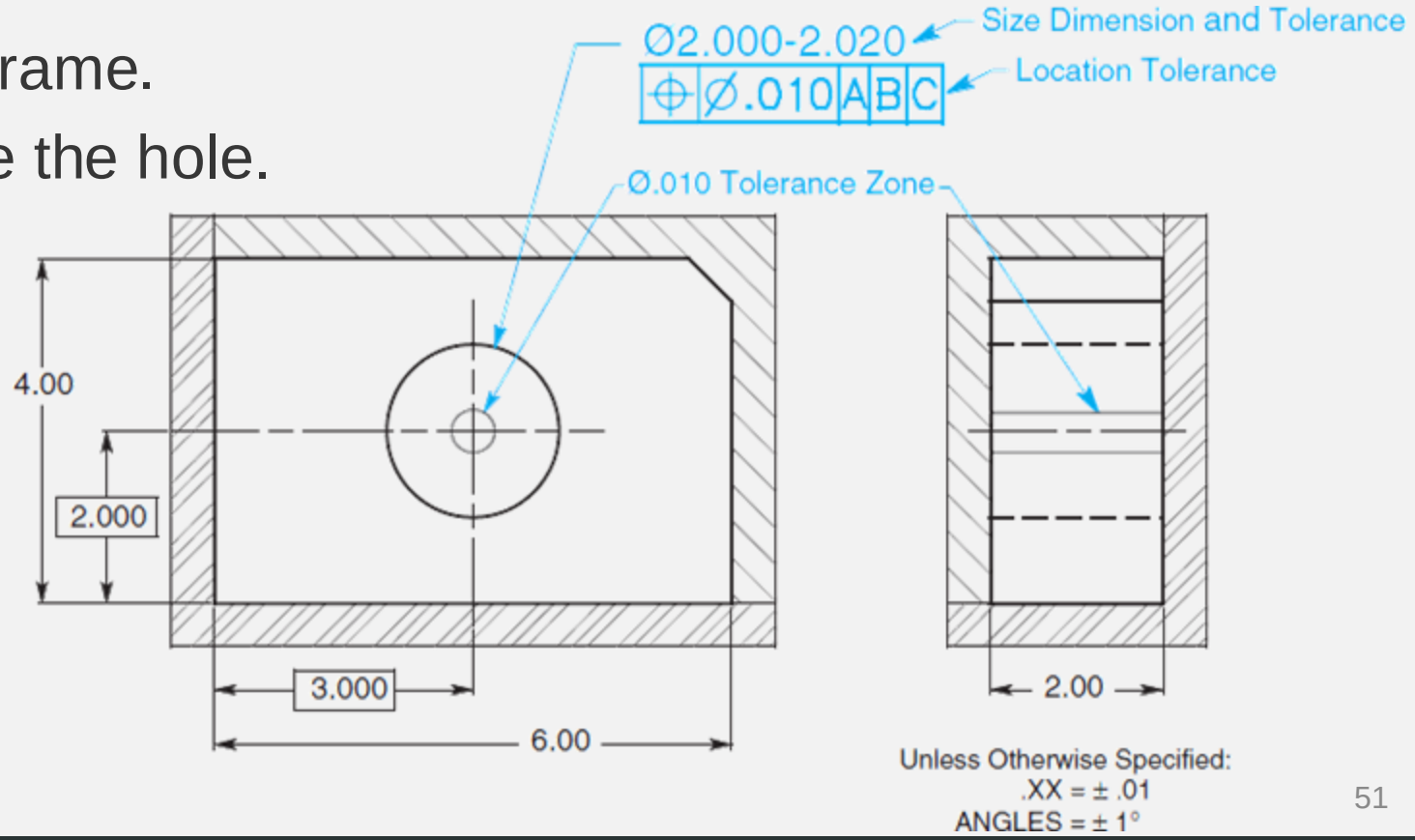


Specifying Position Tolerance



Inspection of Position Tolerance

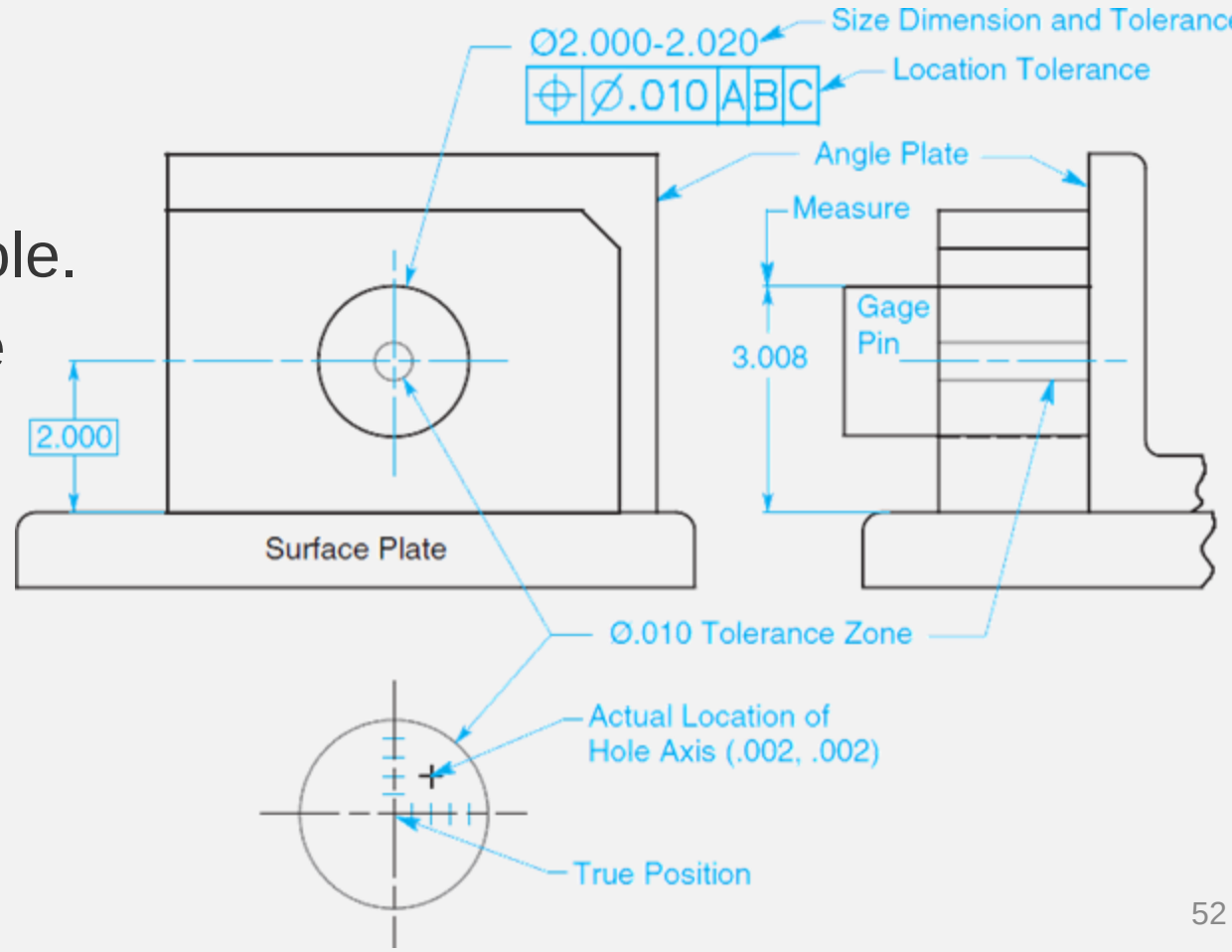
1. measure the hole diameter.
2. clamped in a datum reference frame.
3. use largest pin gage to fit inside the hole.



Inspection of Position Tolerance

1. measure the hole diameter.
2. clamped in a datum reference frame.
3. use largest pin gage to fit inside the hole.
4. measure the distance from the surface plate to the top edge of the pin gage.
5. calculate the actual axis position
6. repeat above with another datum.

3.008 – 0.5*2.012 – 2.000 = 0.002



Runout Tolerance

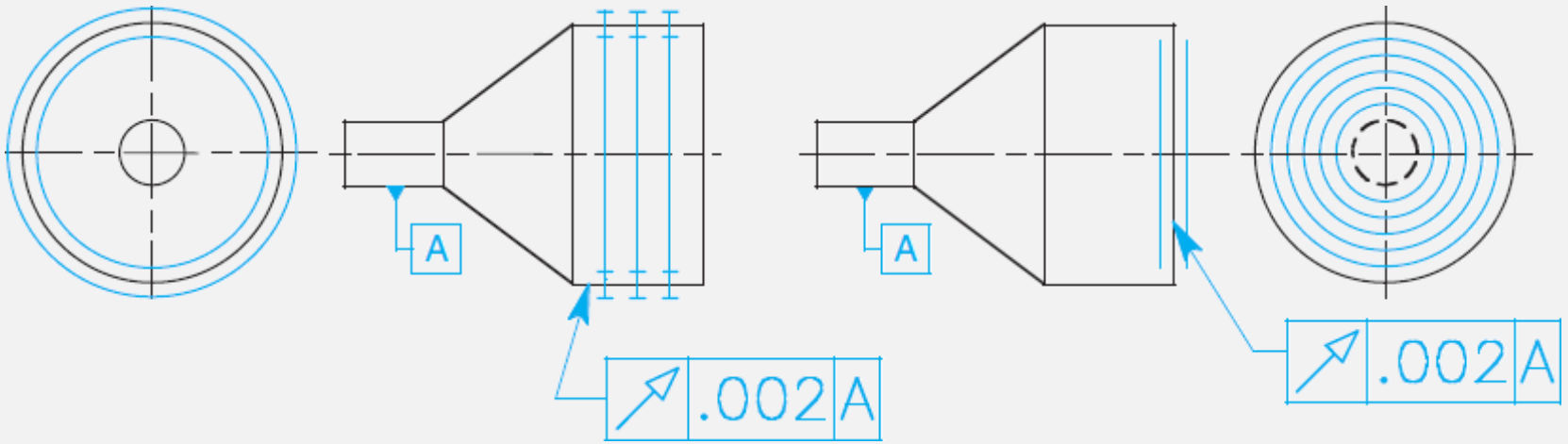
Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
Individual Feature Only	**Form**	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY	
		CYLINDRICITY	
Individual Feature or Related Features	**Profile**	PROFILE OF A LINE	
		PROFILE OF A SURFACE	
Related Features	**Orientation**	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	Location	POSITION	
	Runout	CIRCULAR RUNOUT	
		TOTAL RUNOUT	

Runout Tolerance

- Runout is a **surface control.** It controls **surfaces** constructed **around a datum axis** and surfaces constructed **perpendicular to a datum axis**.
- Runout is a **composite tolerance** used to control the functional relationship of one or more features of a part to a datum axis established from a datum feature.
- Circular runout  and Total runout 
- It may not exceed the boundary of perfect form at MMC.

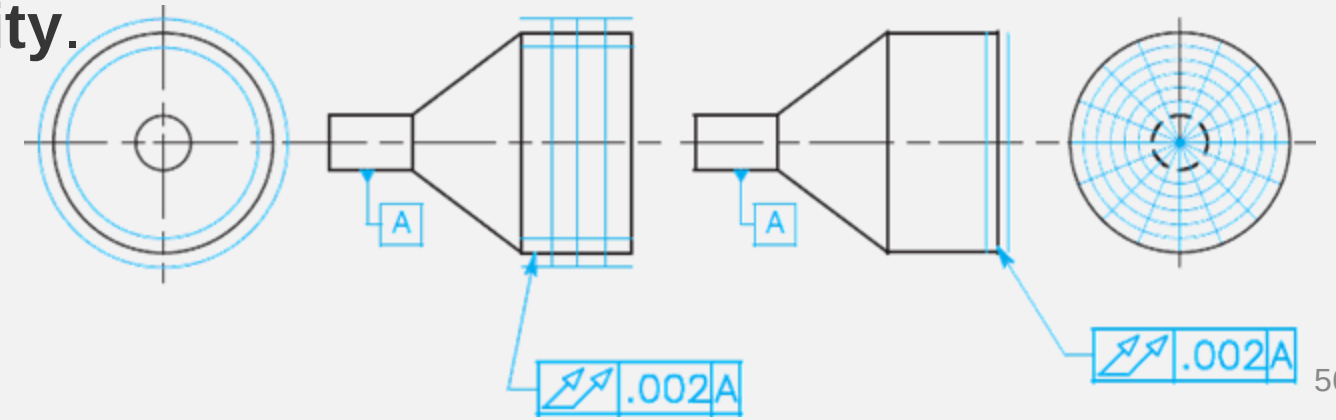
Circular Runout Tolerance

- Circular runout applies **independently** to each circular element on the surface of a part either constructed around its datum axis or perpendicular to its datum.
- where applied to surfaces of revolution, circular runout controls a combination of variations in **circularity and coaxiality**.
- where applied to surfaces perpendicular to a datum axis, circular runout controls **wobble**.

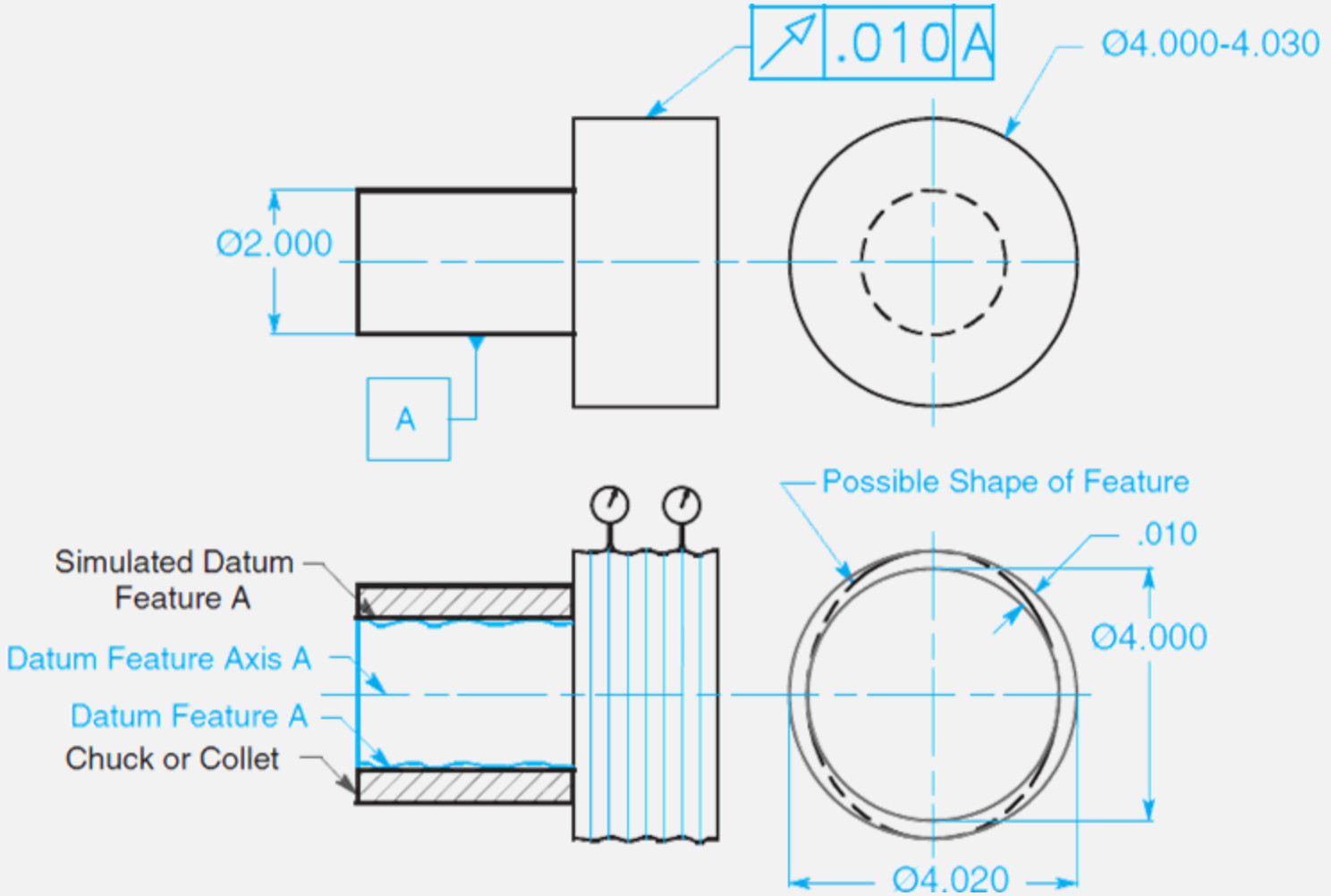


Total Runout Tolerance

- Total runout is a **compound tolerance** that provides control of all surface elements of a feature.
- where applied to surfaces constructed around a datum axis, total runout controls a combination of surface variations, such as **circularity, straightness, coaxiality, angularity, taper, and profile.**
- where applied to surfaces perpendicular to a datum axis, total runout controls **wobble and concavity or convexity**.



Inspection of Runout Tolerance



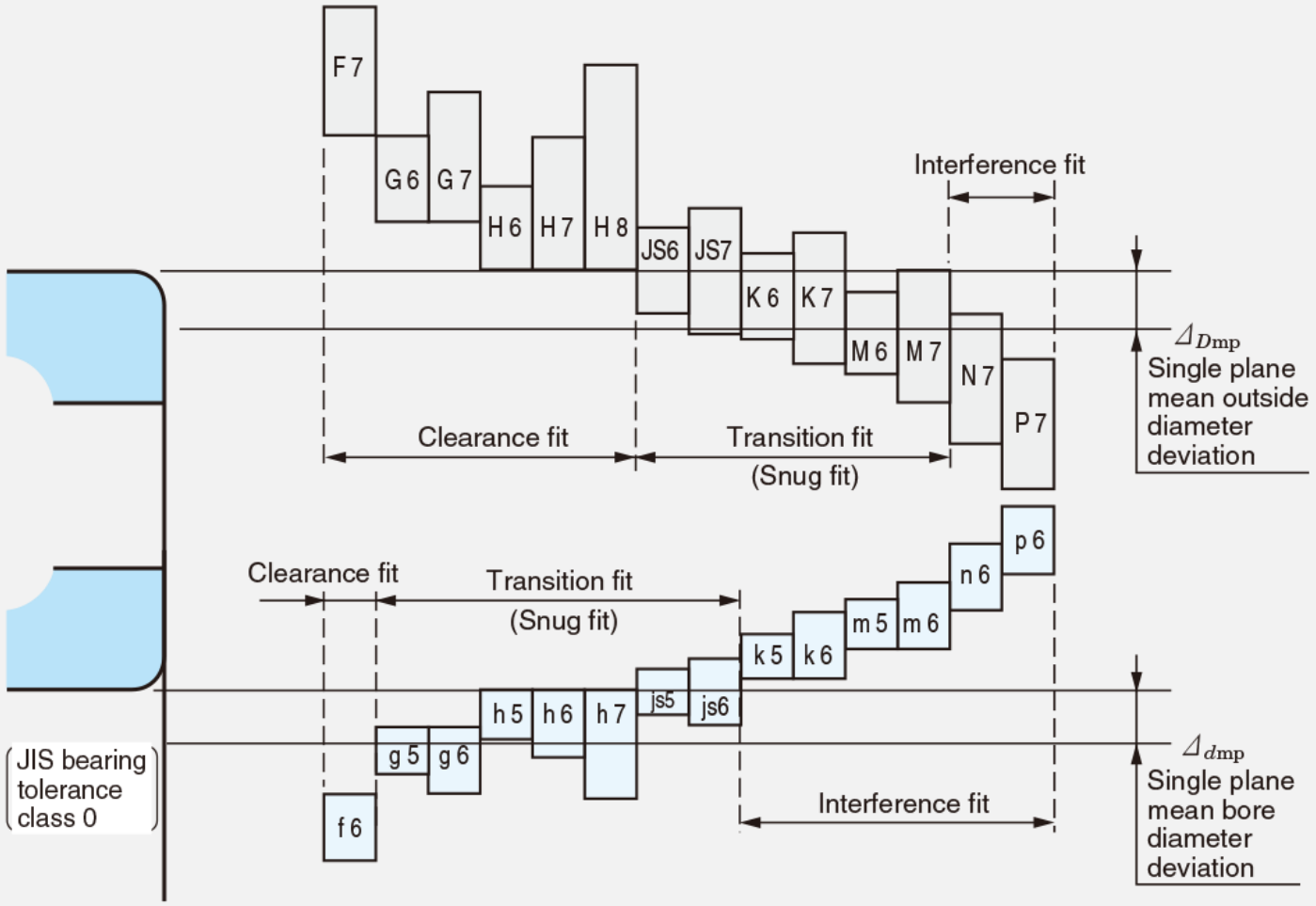
GD&T Tolerancing Guidelines

An engineering drawing has to accurately convey the product without adding unnecessary complexity or restrictions.

- **Clarity** of a drawing is the most important, even more so than its accuracy and completeness.
- Each feature **must** be toleranced and completely understood without ambiguity.
- Always **design for the loosest feasible tolerance** to keep costs down.
- Use a **general tolerance** defined at the bottom of the drawing for all dimensions of the part.
- **Tolerance functional features** and their interrelations **first**, then move on to the rest of the part.
- Whenever possible, leave GD&T work to the manufacturing experts and **do not describe manufacturing processes** in the engineering drawing.
- Do not specify a 90-degree angle since it is assumed.
- Dimensions and tolerances are valid at 20 ° C / 101.3 kPa unless stated otherwise.

Bearing tolerance and fit for shaft & housing

ISO 286



Example drawing of a shaft