Geometric Dimensioning & Tolerancing (Part 3)

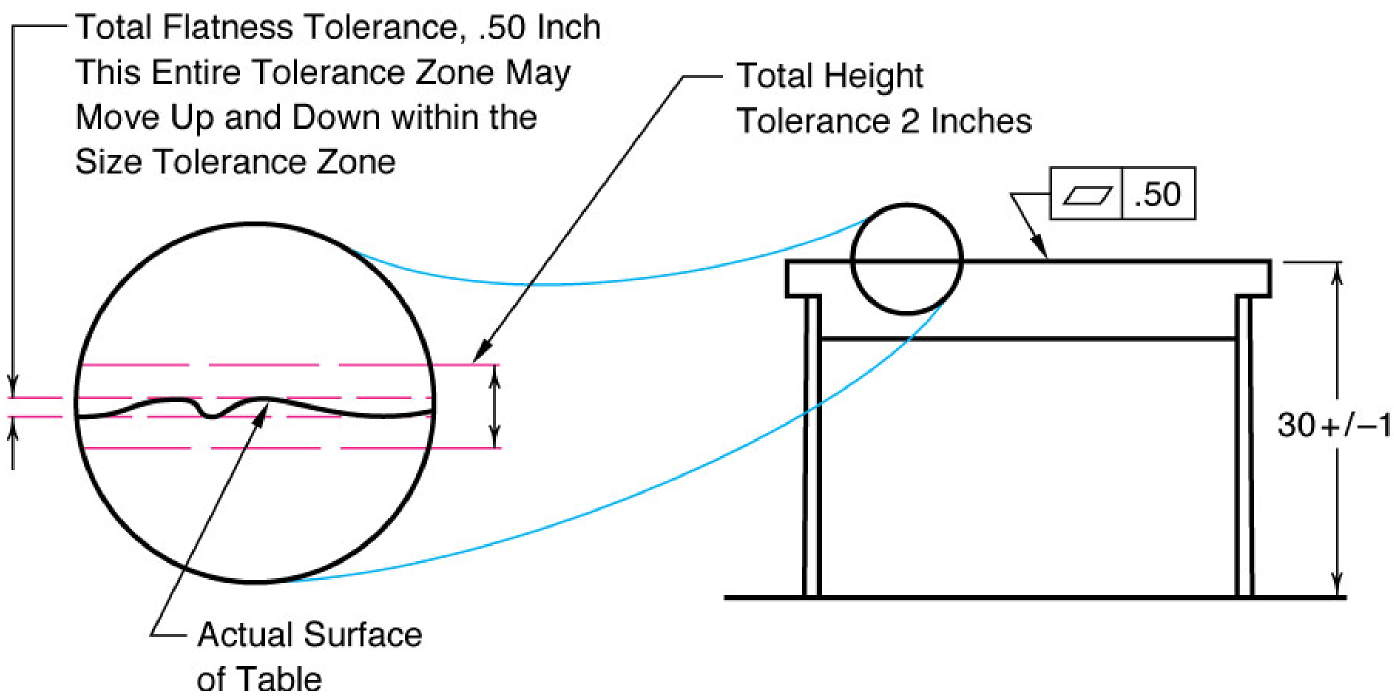
KCEC 1101

Introduction to GDT basic

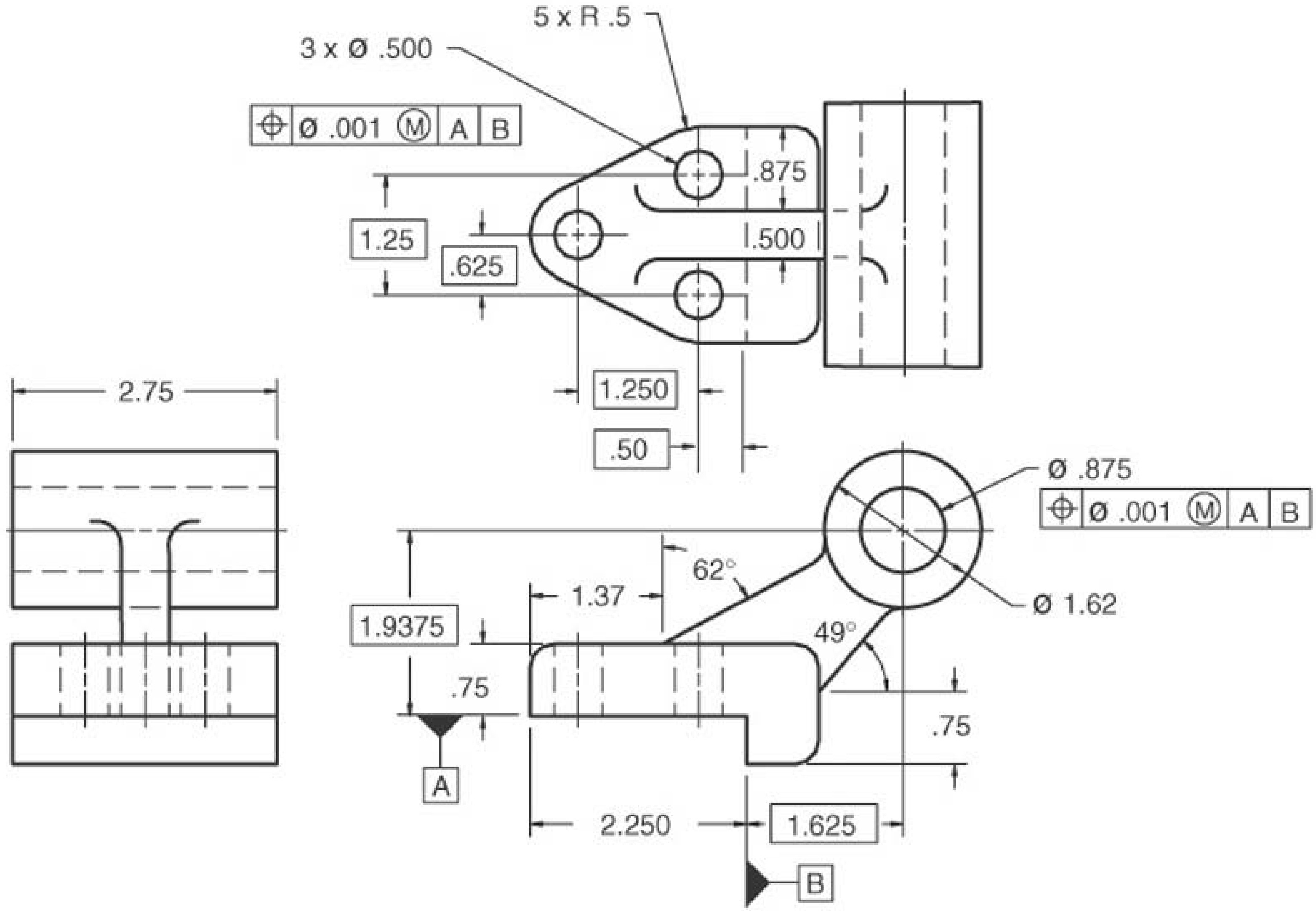
- **Geometric dimensioning and tolerancing** (GDT) was developed over the last forty years as a tool to define parts and features more efficiently.
- GDT takes the function of the part into consideration, as well as its fit with related parts.
- This allows the designer to define the parts features more accurately, without increasing the tolerances.

Overview on GDT

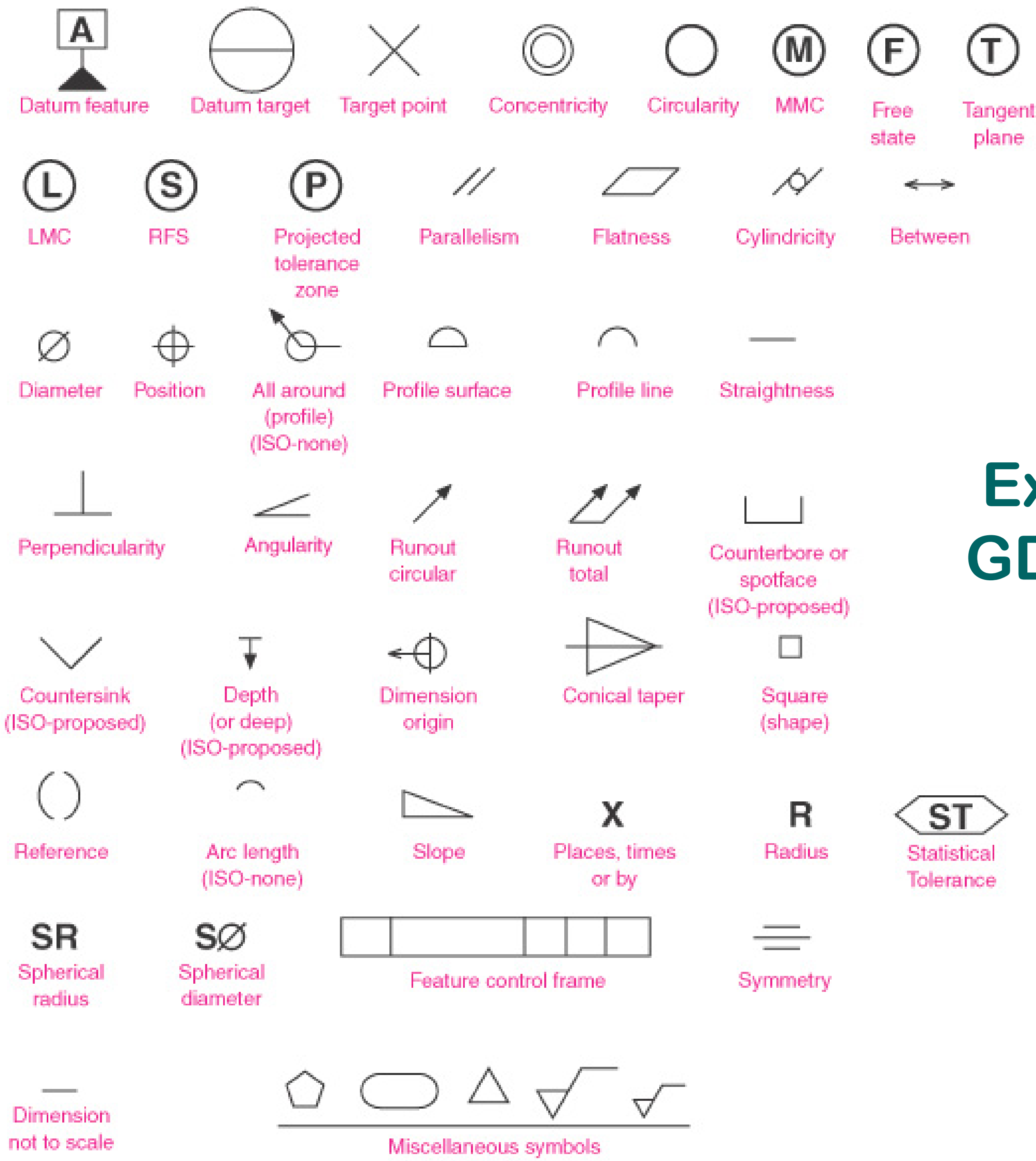
- Within the last 15 years there has been considerable interest in GDT, in part because of the increased popularity of **statistical process control**.
- This control process, when combined with GDT, helps reduce or eliminate inspection of features on the manufactured object.
- The flip side is that the part must be toleranced very efficiently; this is where GDT comes in.



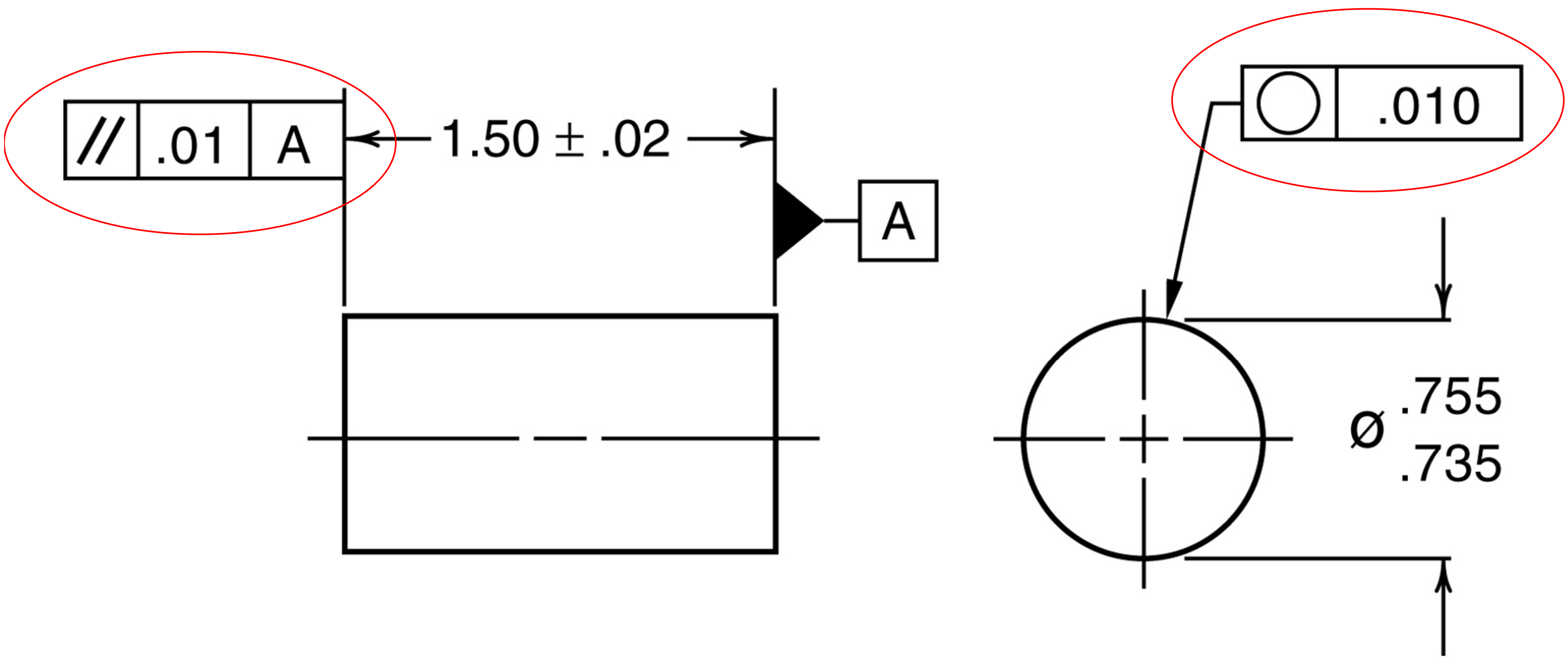
- **Another reason for the increased popularity of GDT is the rise of worldwide standards, such as ISO 9000, which require universally understood and accepted methods of documentation.**



Examples of GDT symbols



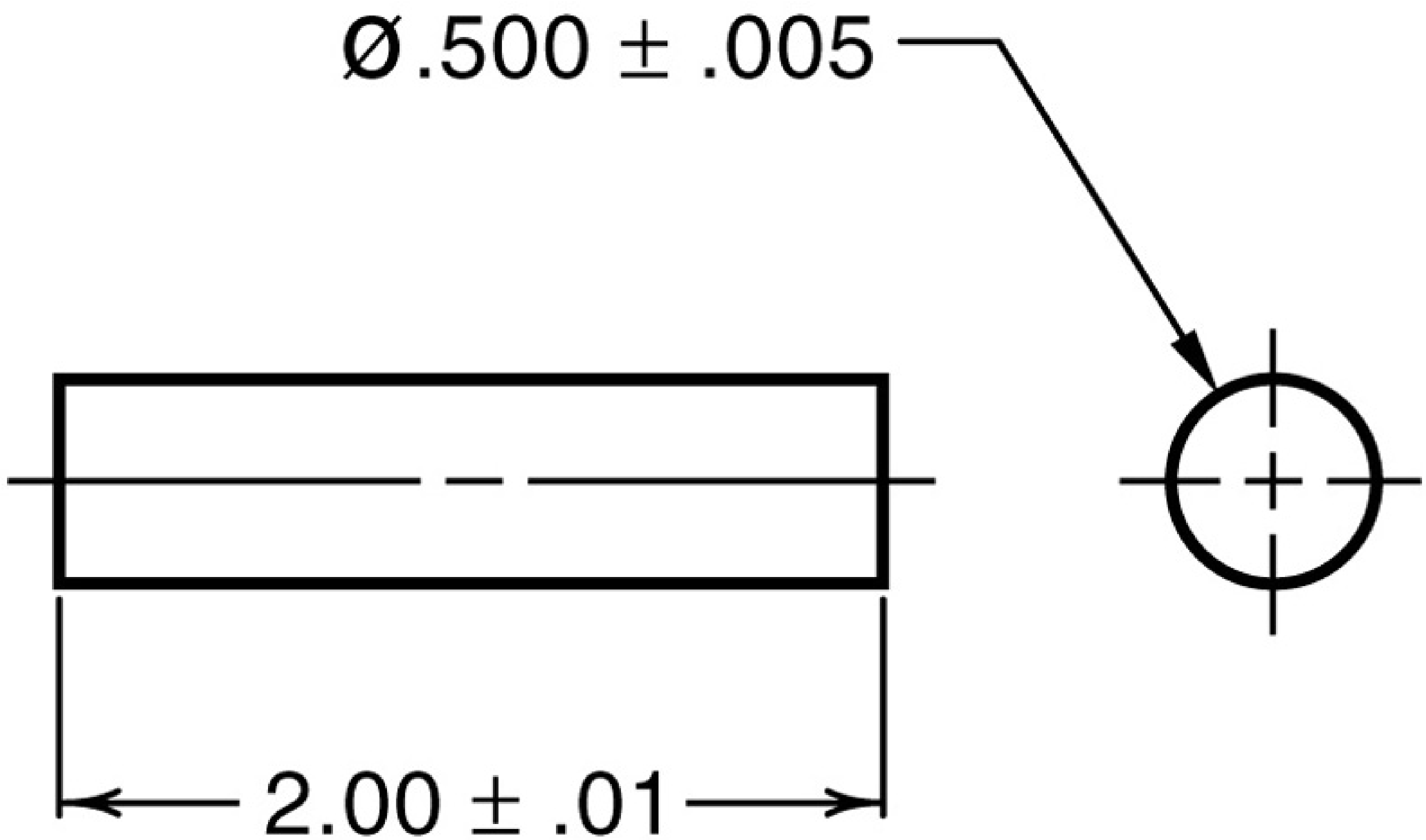
- At the heart of GDT is a rectangular box, called the **feature control frame**, in which the tolerancing information is placed.



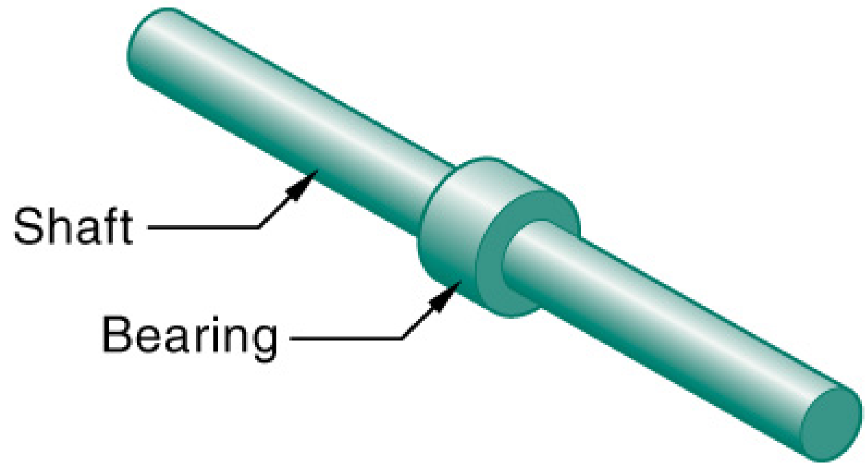
INDIVIDUAL FEATURE OF SIZE

- This rule covers the individual feature size and states:
  - Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form, as well as size, are allowed.
- The critical point is that GDT is concerned with both the **shape** and **position** of features.

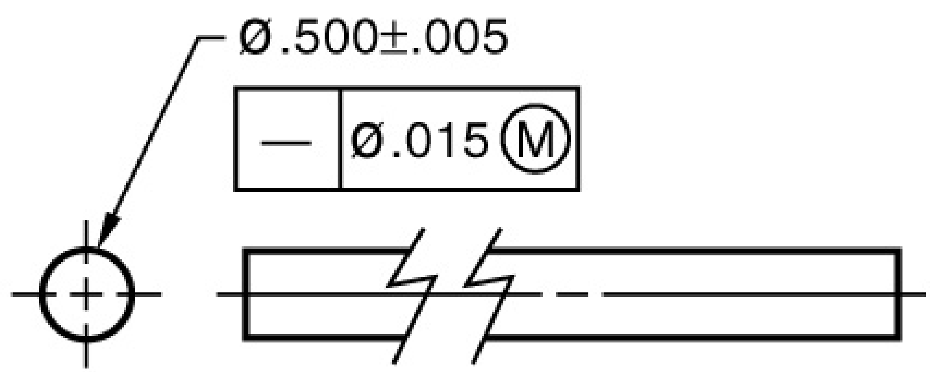
Individual feature of size



- If a shaft consider to mount liner positioning bearings which are used for sliding rather than rotation motion.
- In this case, the shaft must be highly accurate in size and roundness, but that same accuracy is not possible for the straightness.
- This situation is common in machine control design.



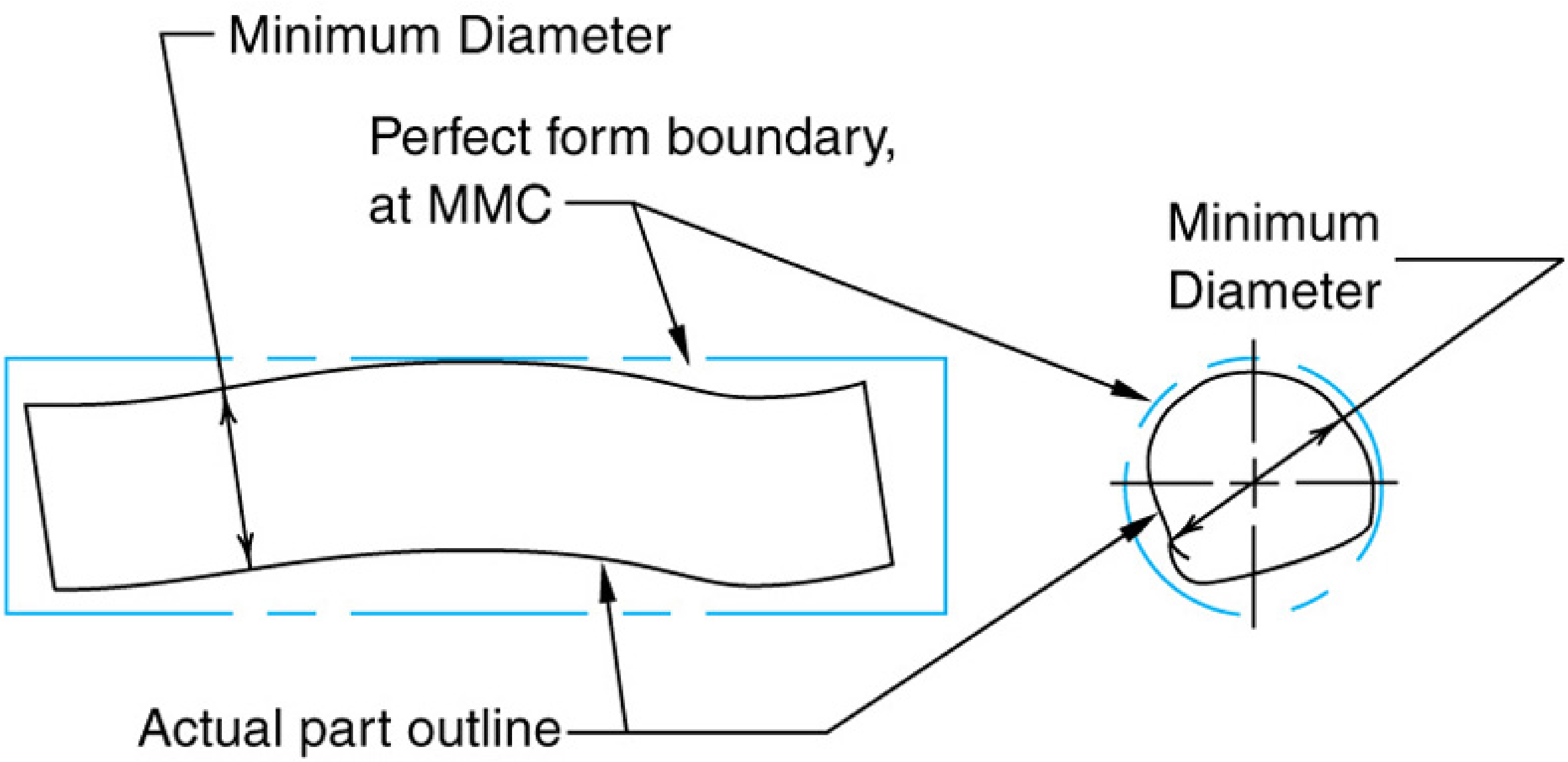
Shaft in Use



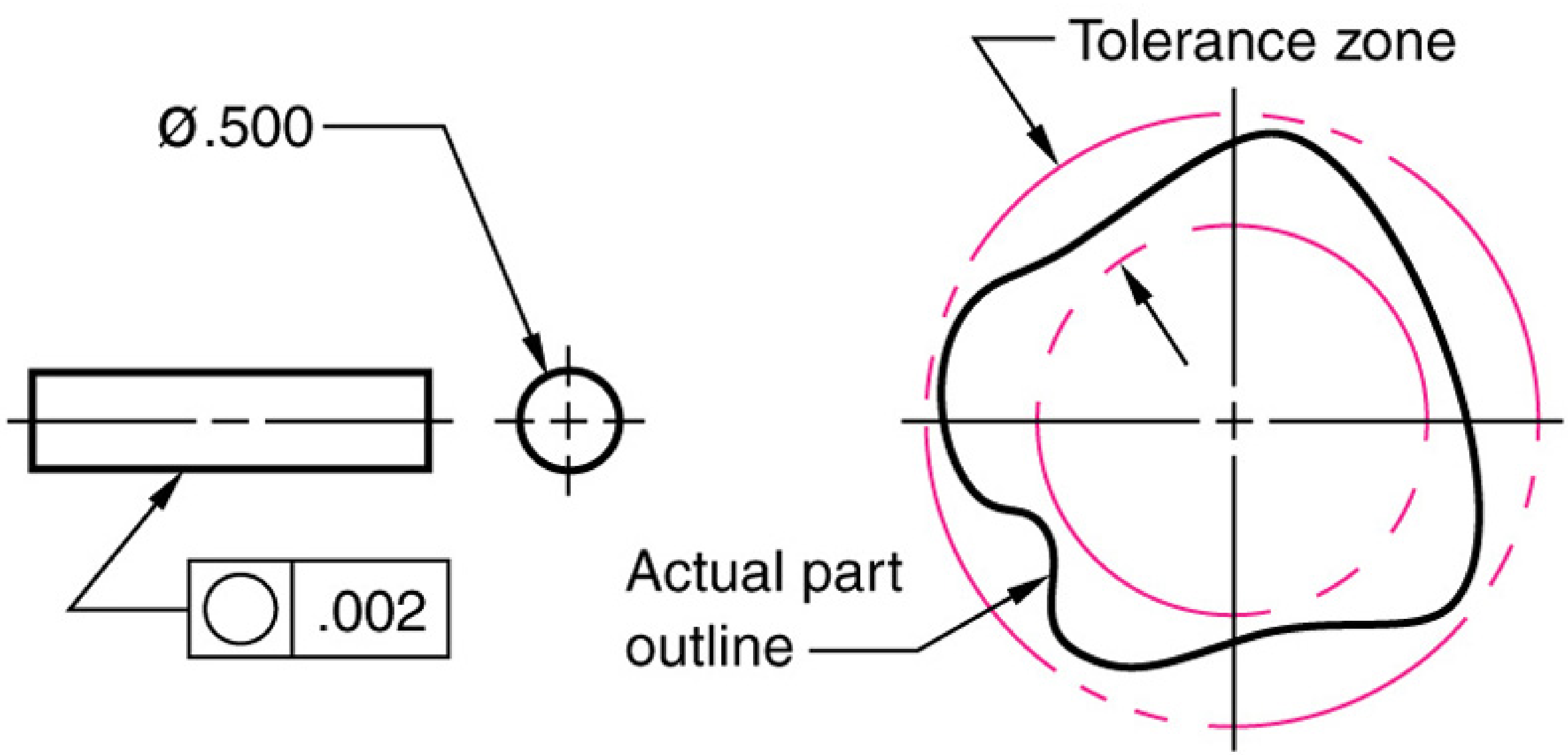
Drawing Callout

- **Maximum material condition (MMC) is the condition in which:**
  - **An *external* feature, like a shaft, is its *largest* allowable size.**
  - **An *internal* feature, like a hole, is its *smallest* allowable size.**
- **Taken together, it means the part will weigh its maximum.**
- **Three symbols pertaining to material conditions are:**
  - **M = Maximum material condition**
  - **L = Least material condition. This is the opposite of MMC (the part will weigh its minimum)**
  - **S = Regardless of material size. This indicates the material condition is not to be considered.**

- The **envelope principle** describes an idealized MMC of a part. Since any departure from MMC means less material, all parts should fit within an envelope described by MMC.



- **One of the keys to is that it allows separation of size from form. This added flexibility means that part can be manufactured at the highest allowable tolerance (and thus lowest cost) and still work. This two variables can be manipulated to meet the particular demands of the part functionality**



Inspection tools

- There are several inspection tools used in industry, and their use is important to the science of tolerancing.
  - Surface plate
  - Height Gage
  - Caliper
  - Micrometer
  - Precision spindle
  - Centers
  - Dial indicator
  - Coordinate measuring machine (CMM)

Surface plate

Figure 16.9

Surface plate

A surface plate is made of steel or granite and is used as a base for making inspections.

Height gage

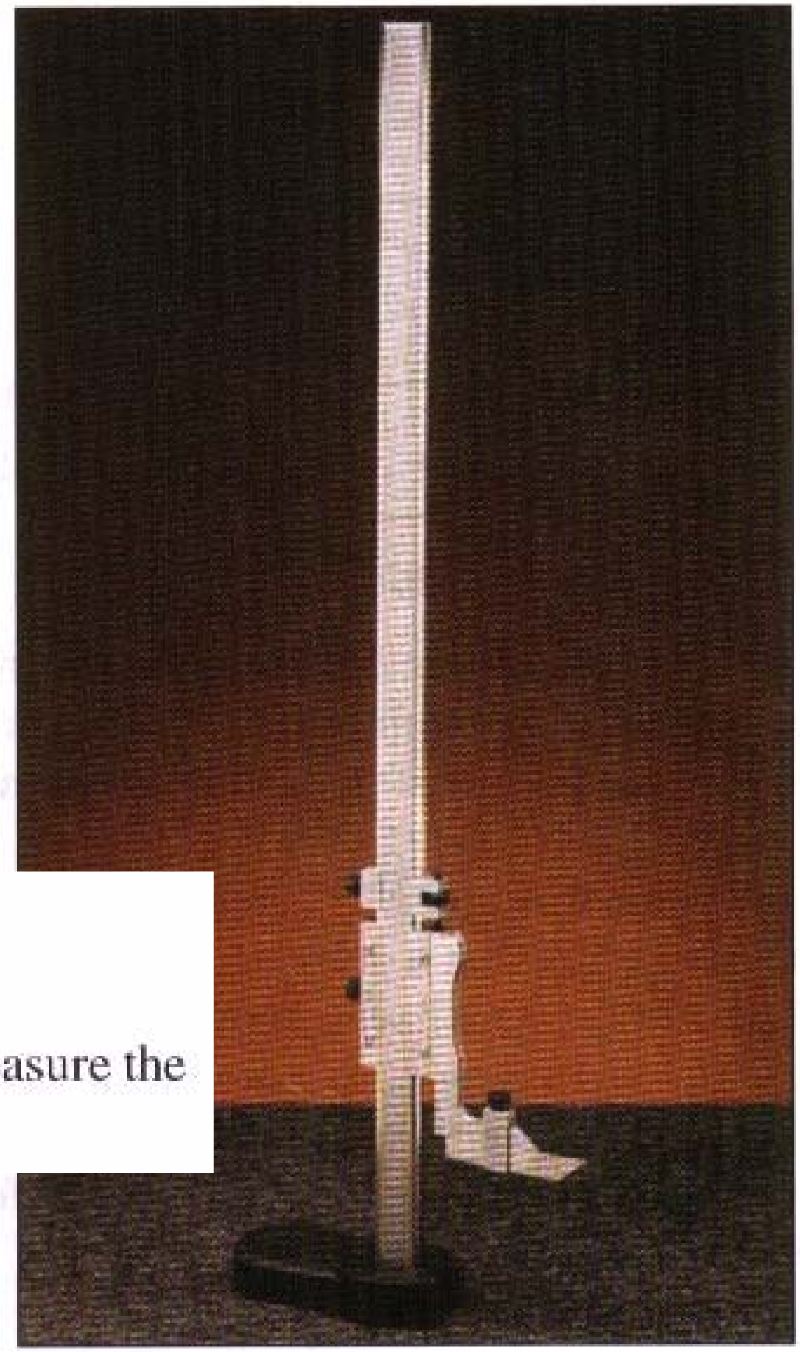


Figure 16.10

Height gage

A height gage is a precision instrument used to measure the heights of features from a surface plate.

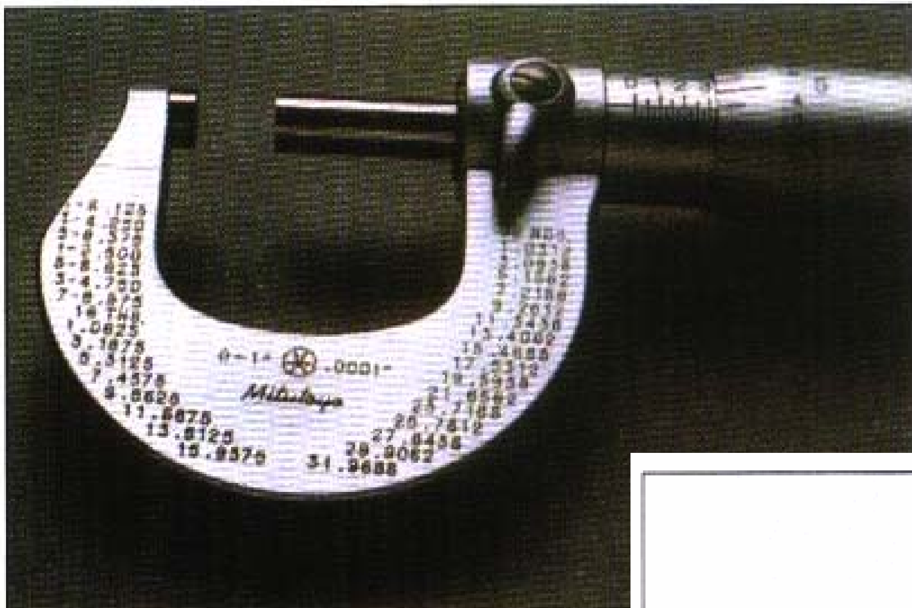
Caliper

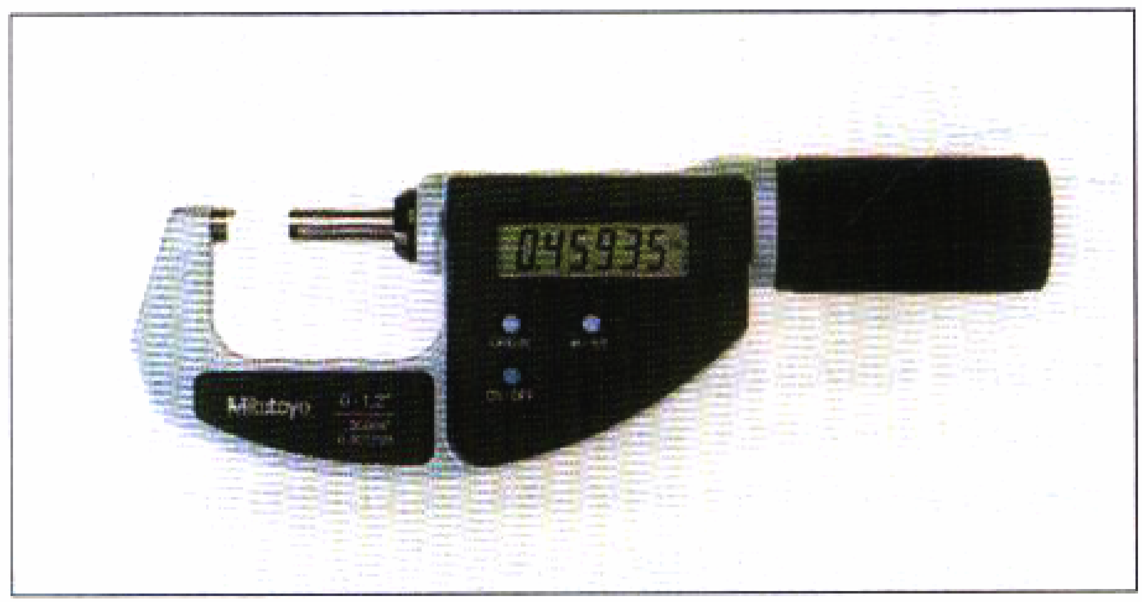
Figure 16.11

Caliper

A caliper is a hand-held instrument for measuring lengths, depths, and inside and outside diameters.

Micrometer





Dial indicator

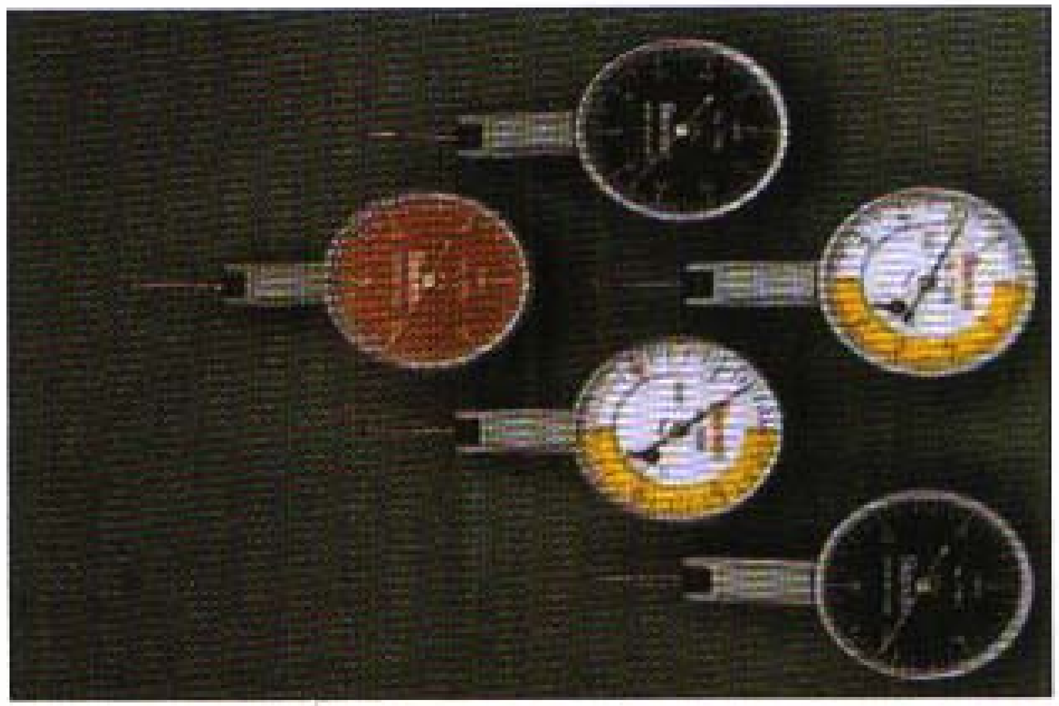


Figure 16.14

Dial indicators

A dial indicator is an instrument used to inspect and measure minute amounts of motion.

CMM

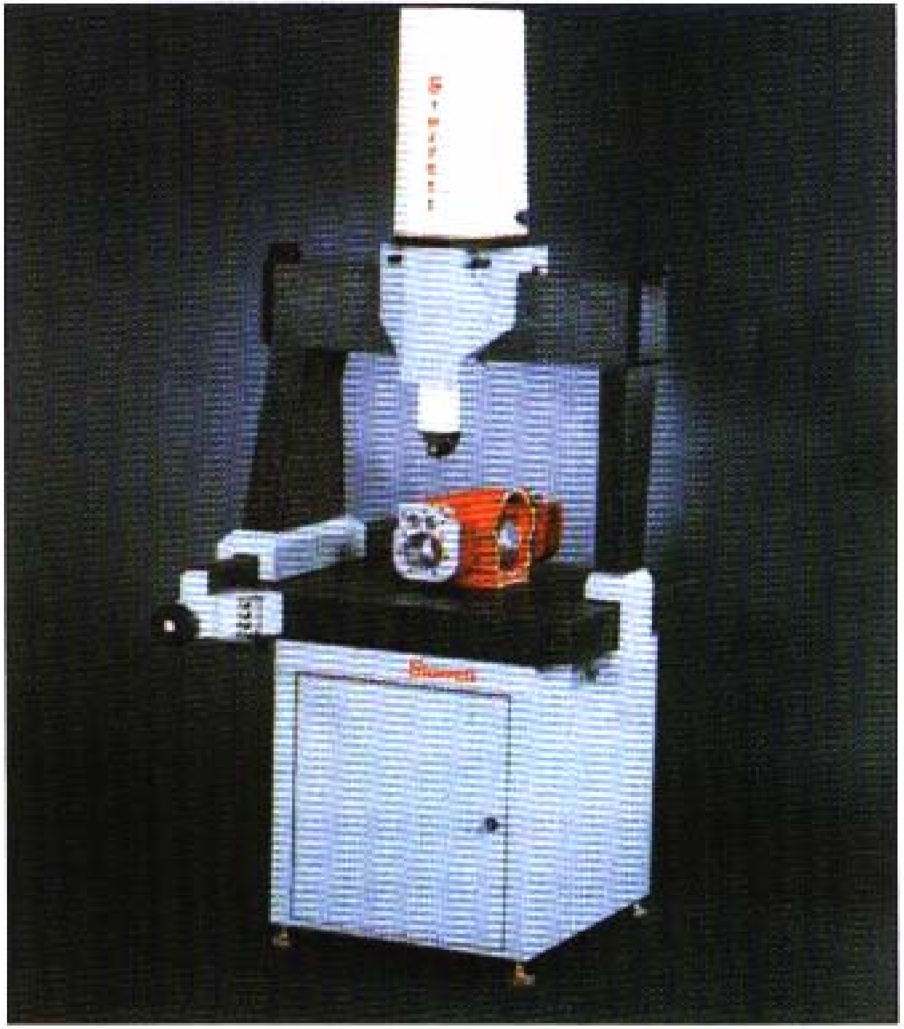


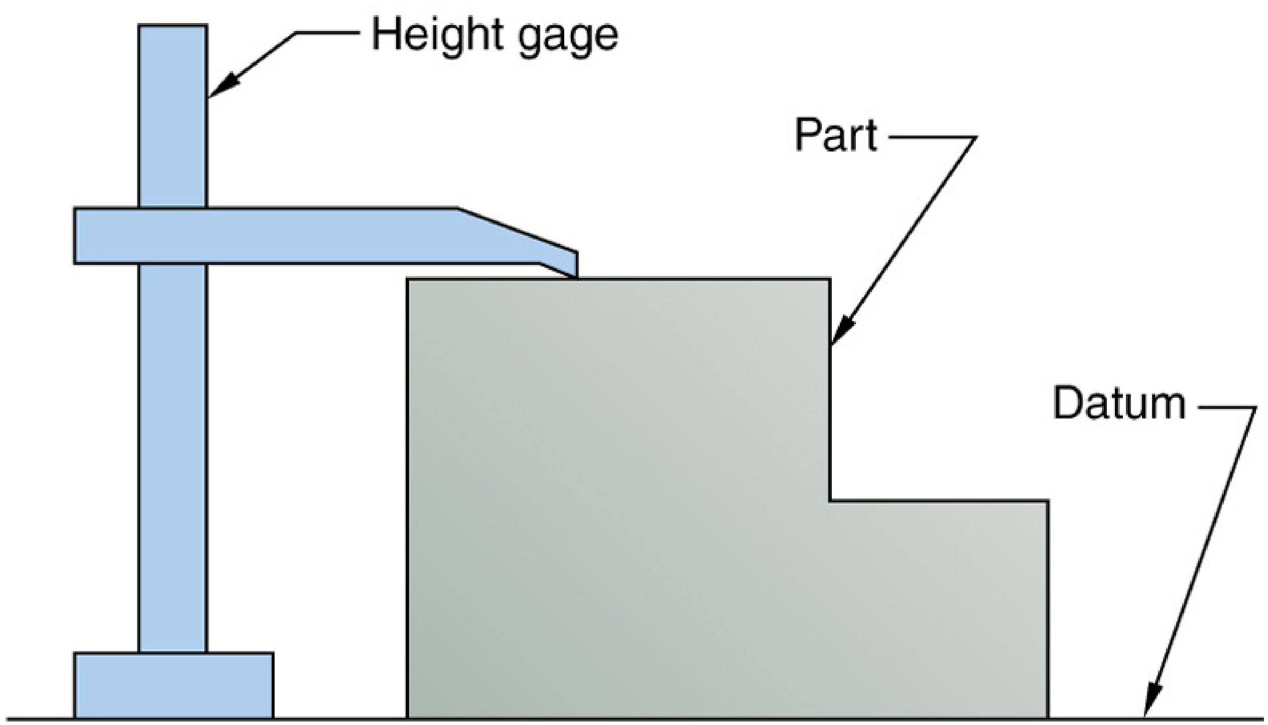
Figure 16.15

Coordinate measuring machine

A coordinate measuring machine (CMM) is an electromechanical device used to take precision measurements of parts.

DATUMS AND DATUM FEATURES

- A **datum** is a starting point for a dimension.
- Datums are theoretically ideal locations in space such as a plane, centerline, or point.
- A datum may be represented either directly or indirectly by an inspection device.



DATUMS AND DATUM FEATURES

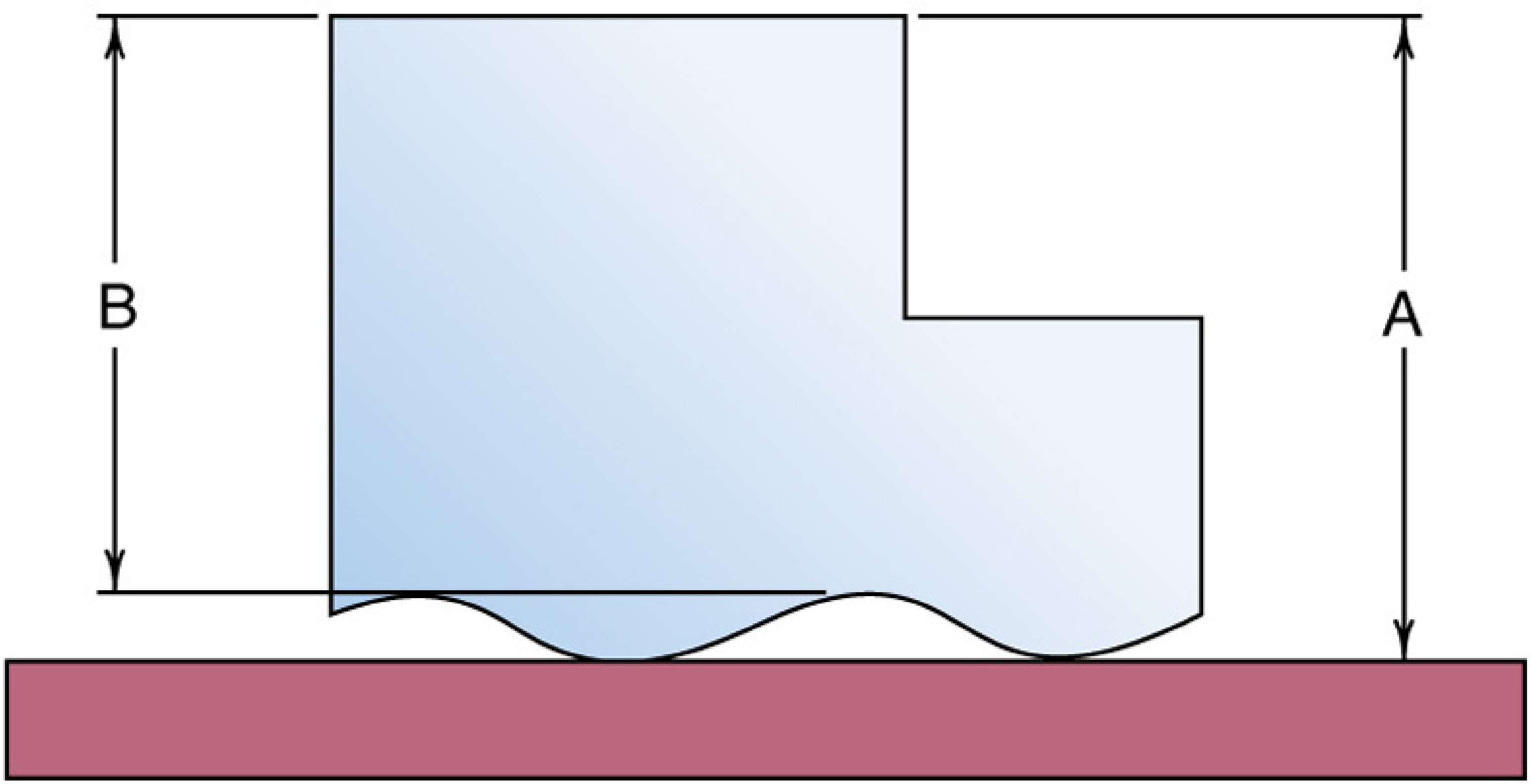
- The surface of the object which is placed on the inspection device representing the datum is called the **datum feature.**
- These datum features are clearly marked in the drawings to indicate which are the reference surfaces to make measurements from.

Datum Uses

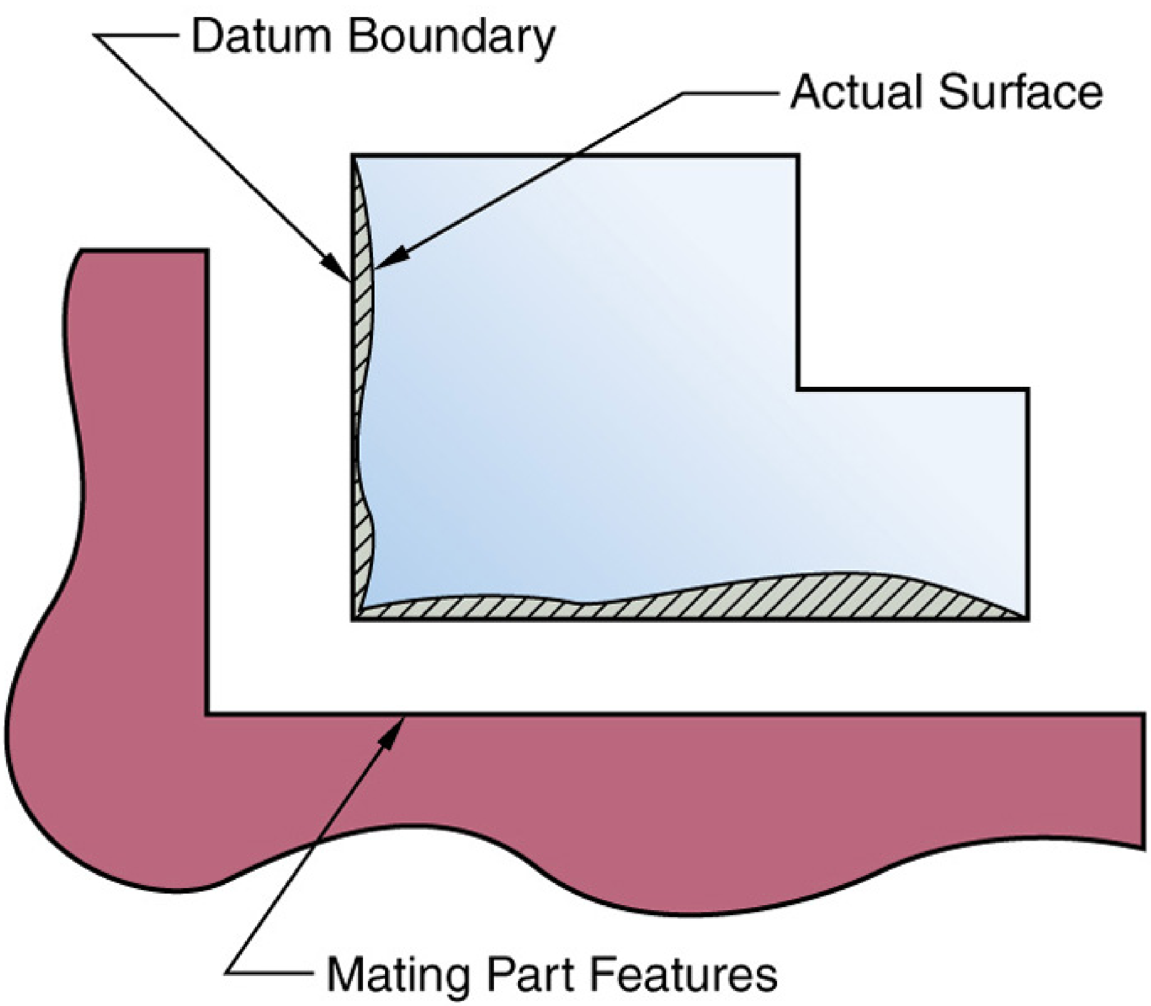
- Once a datum is established, the measurements can be taken from it rather than features on the object.
- That is to say, the object feature representing the datum is aligned/placed on the inspection device and the measurement is taken.
- The datum establishes the method of locating other features on the object relative to each other.

Datum Uses

- This figure shows the difference in measuring fro a surface plate (datum) versus from the part itself



Datums and assembly

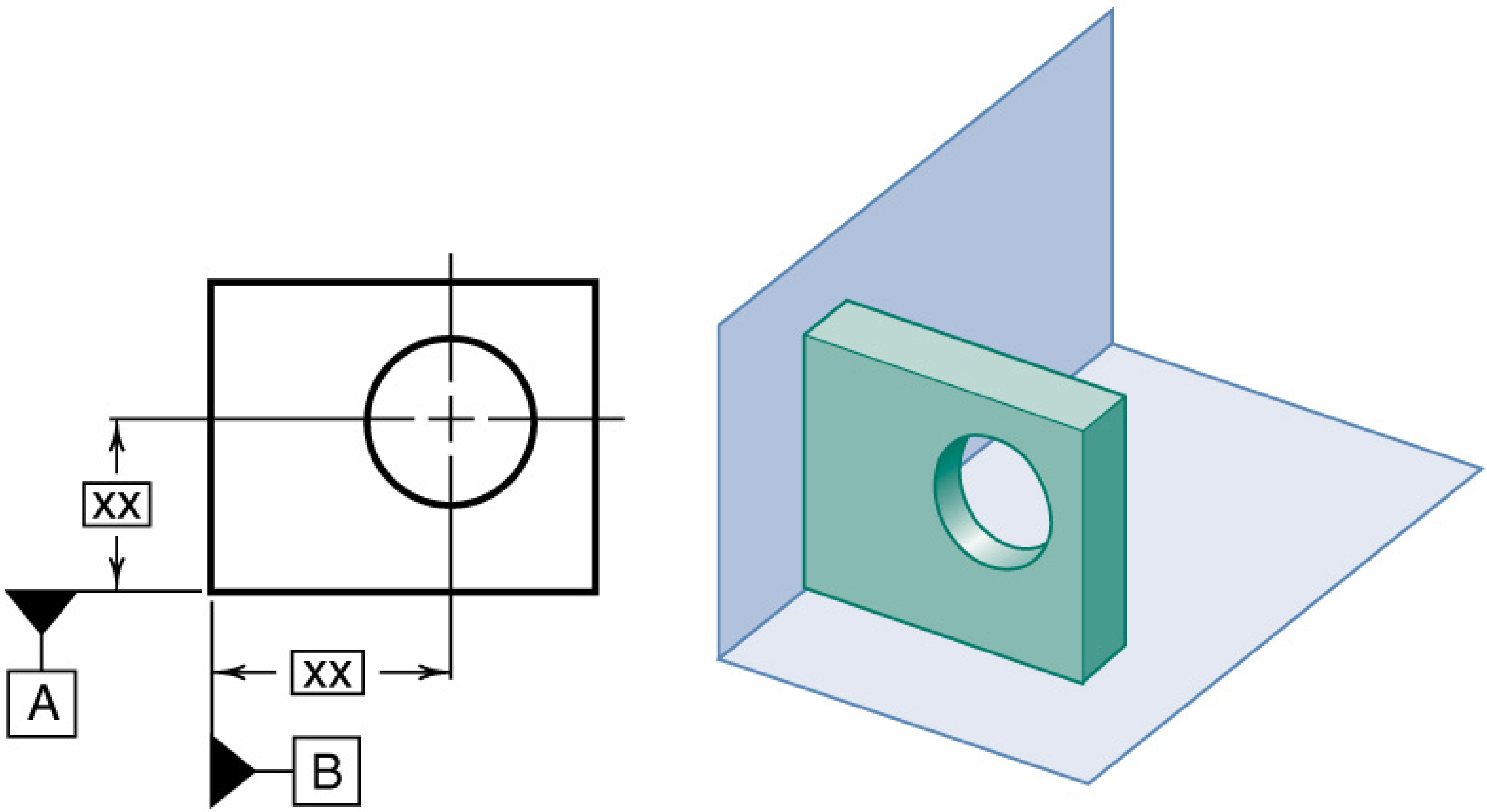


- **Datums are not only used internal to a part but, more importantly, in relation to mating parts in an assembly.**

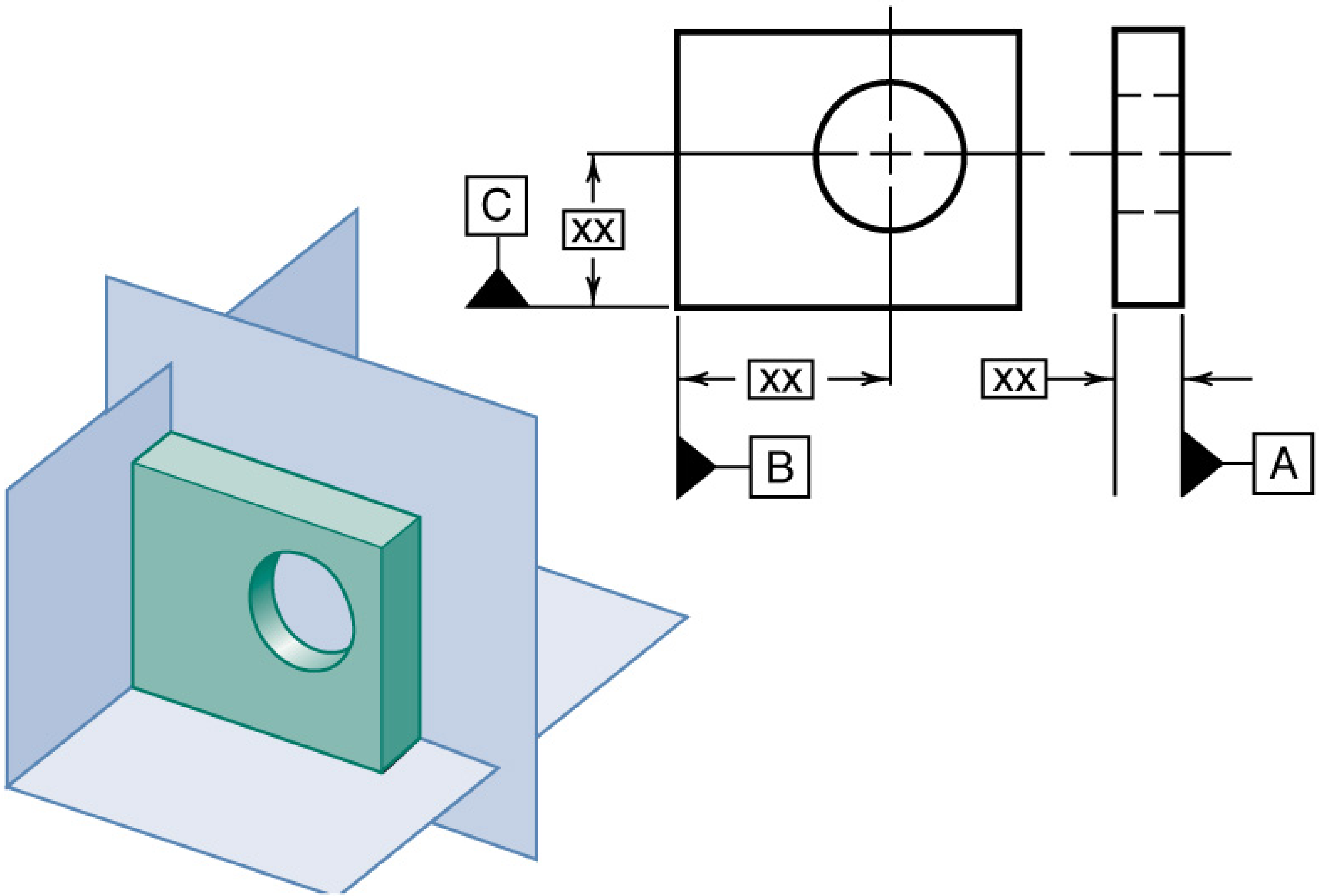
Datum reference frame

- Datums are the locators and the **datum reference frame** is the six degrees of freedom of movement from the datum.
- The six degrees of freedom are the plus and minus directions along the three Cartesian coordinate axes.
  - Move up, down, left, right, forward & backward.
- Another way of looking at the frame is as three orthogonal planes.

- **Two-plane datum reference frame in which the hole is dimensioned from the left and bottom**



- **Three-plane datum reference plane**

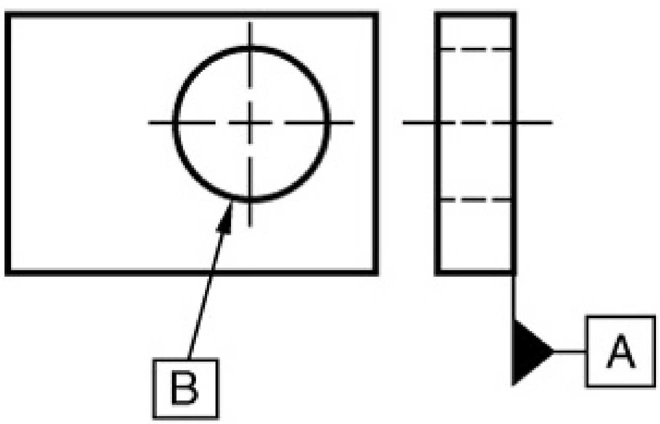


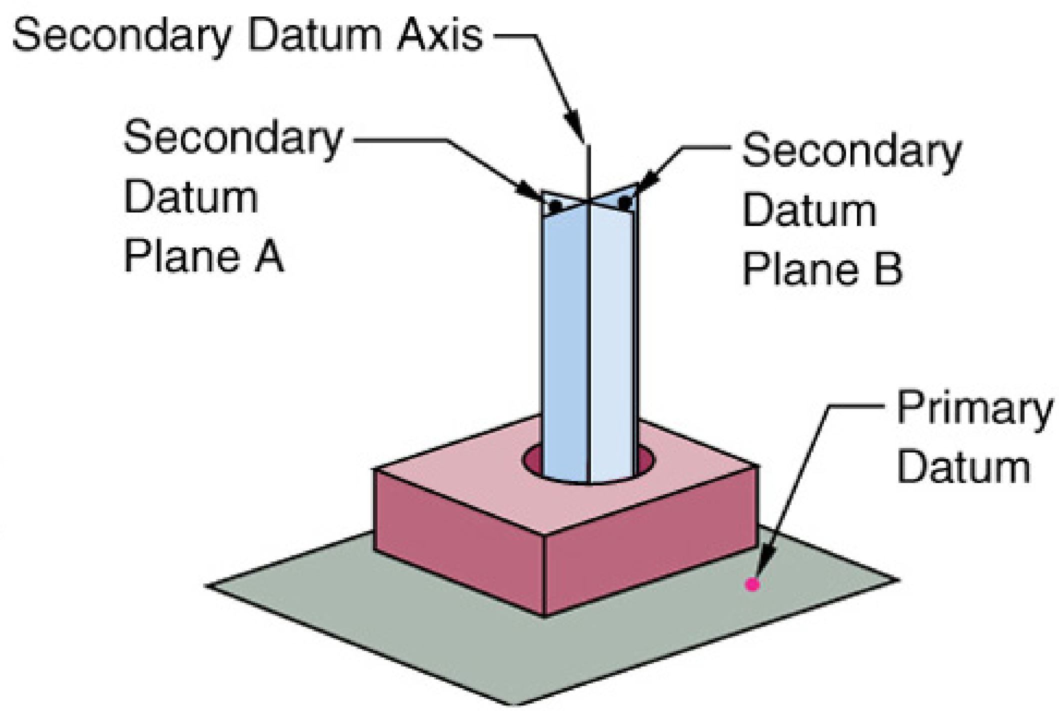
Primary datum

- At a minimum, a part will have a **primary datum**. This datum will be chosen based on a number of criteria:
  - **Stability**. This is the most important feature. Often dictating that the largest, flattest surface is chosen.
  - **Functional relationship**. How does the feature mate/interact with other features?
  - **Accessibility**. Can the part be mounted and measured on the inspection device via this feature?
  - **Repeatability**. Variations in the datum feature due to manufacturing should be predictable so they can be accounted for.

Secondary/tertiary datums

- **Secondary** and **tertiary datums**, if needed, should be located mutually perpendicular to each other and to the primary datum.





Geometric control

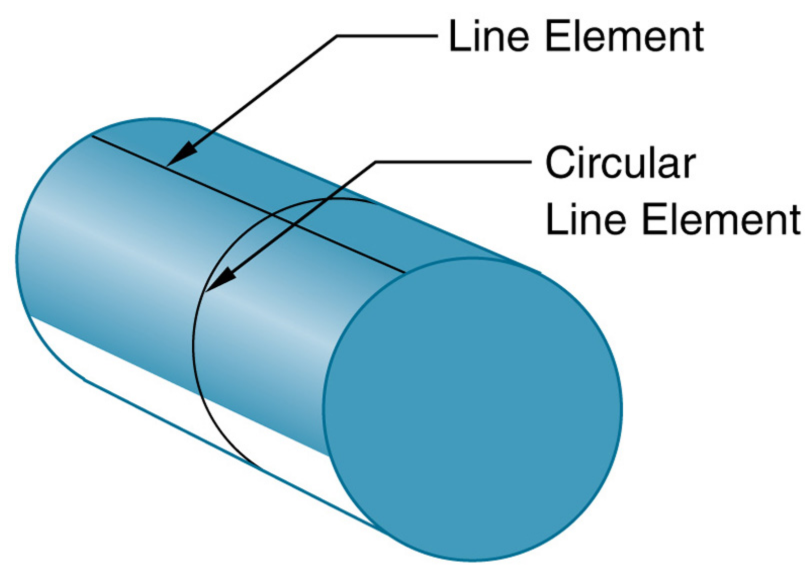
- **Geometric controls fall into three major categories:**
  - **Form controls**
  - **Orientation controls**
  - **Position controls**

Form controls

- **Form controls** are a comparison of an actual feature to a theoretically perfect one.
- The controls include:
  - Straightness
  - Roundness,
  - flatness
  - Cylindricity

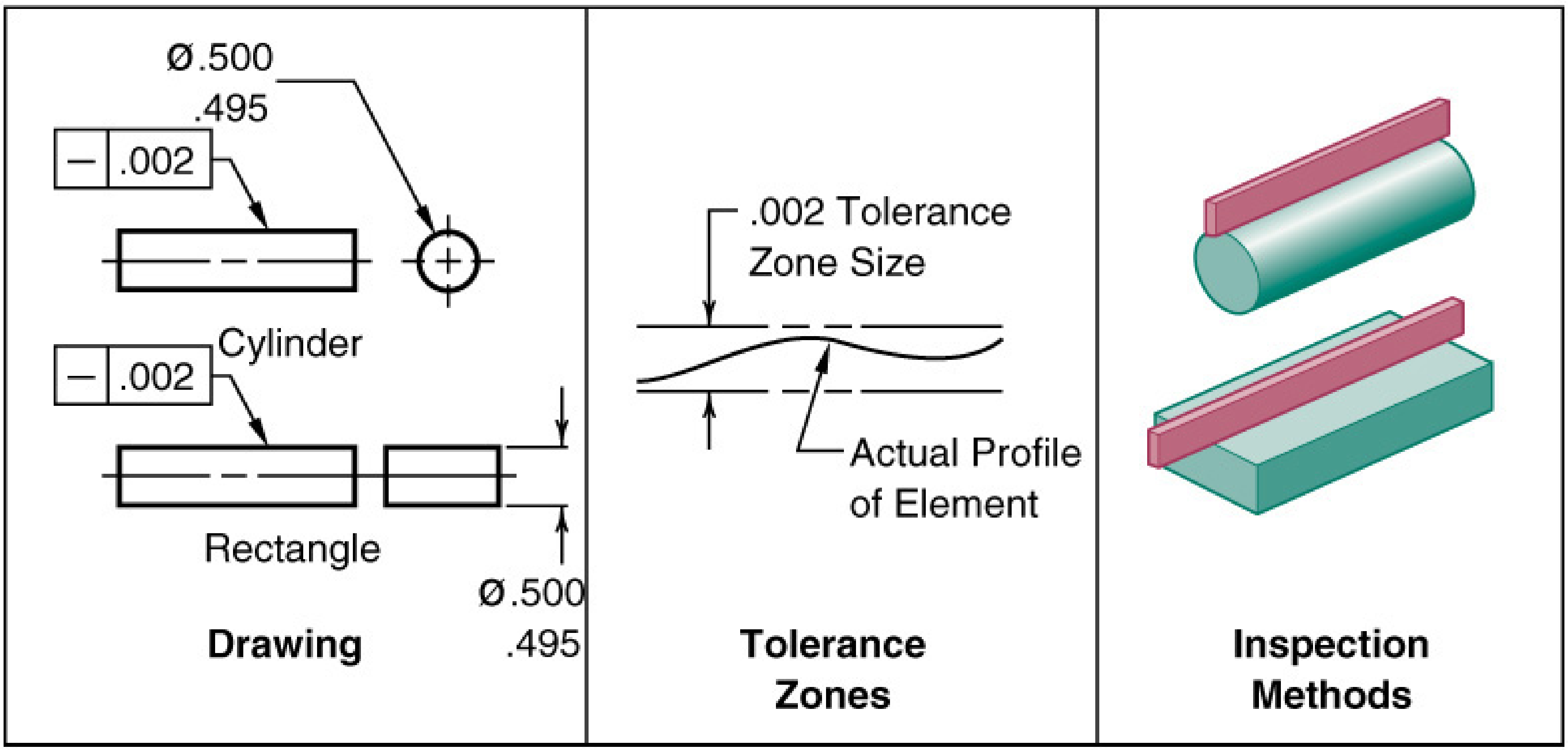
Form control: Straightness

- **Straightness. All form controls are variations and combinations of straightness.**
- **Straightness itself is based on a *line element*.**
- **Straightness has two distinct variations:**
  - **Line element straightness.**
  - **Axis straightness.**



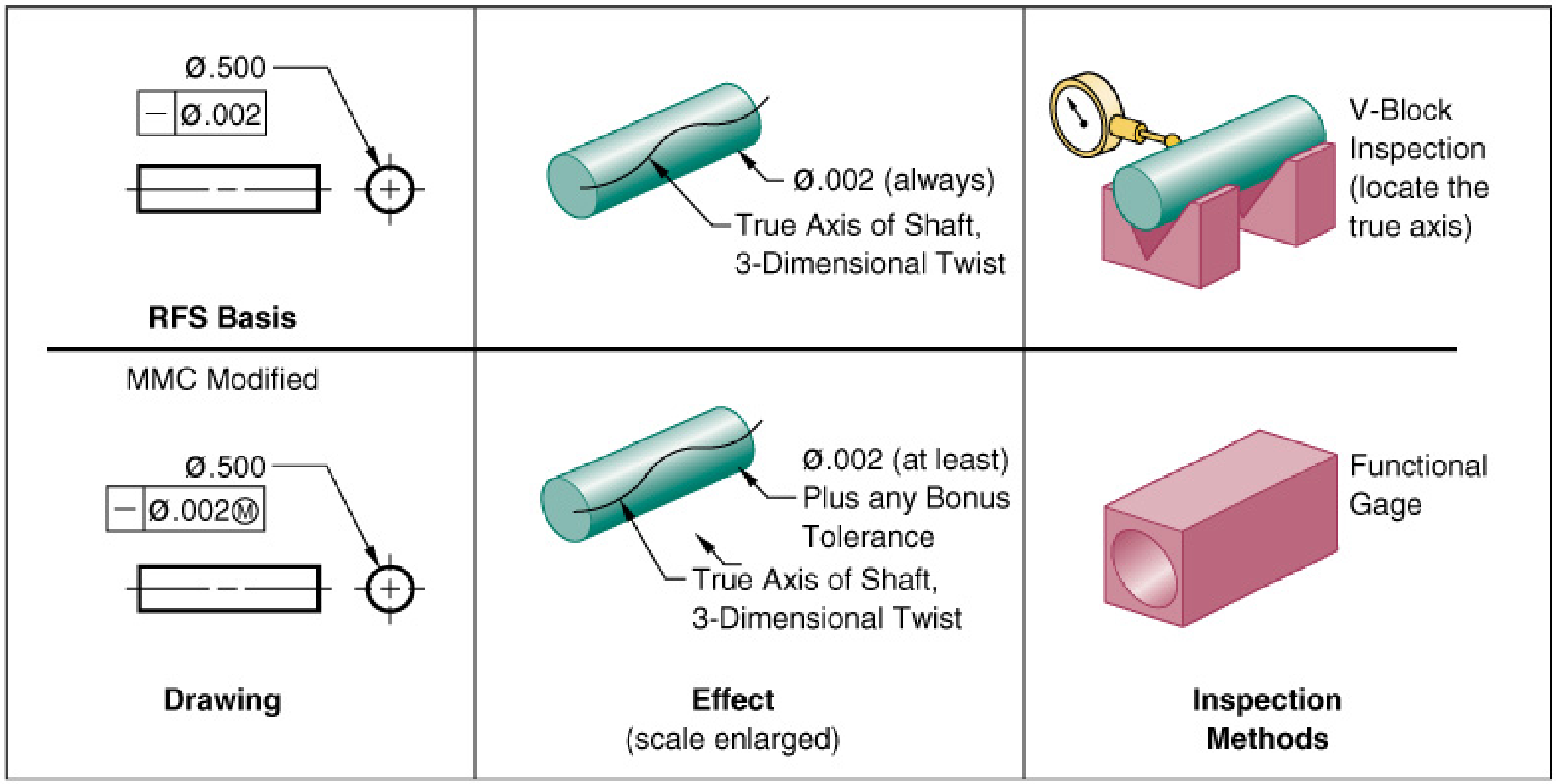
Form control: Straightness

- **Line element straightness.** This compares a line on the part to a perfectly straight line. If the line is on a flat surface, the direction must be identified.



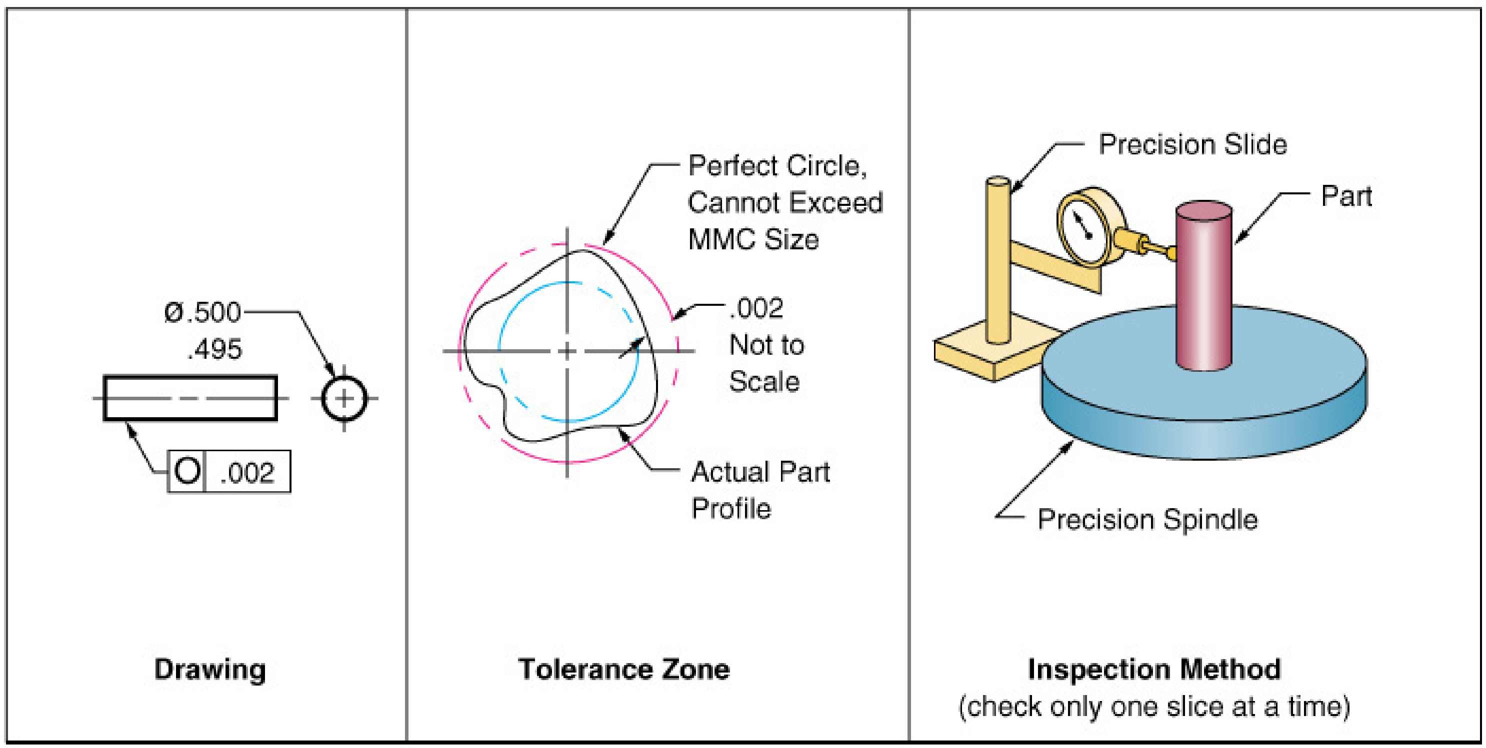
Form control: Straightness

- **Axis straightness. This compares the axis of a cylindrical feature to a perfectly straight line. The axis can also define other symmetric forms such as a square tube.**



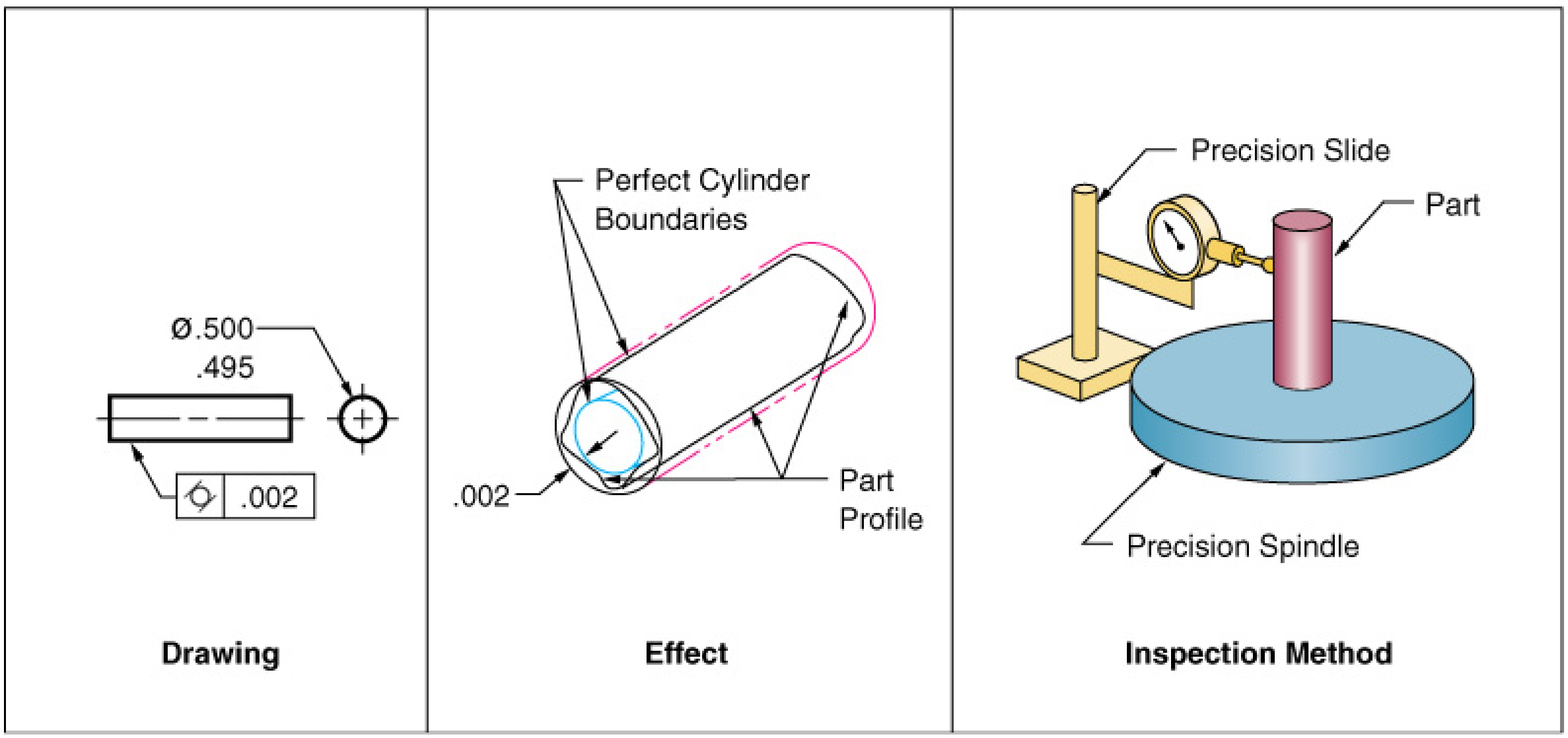
Form control: Roundness

- **Roundness. This compares a circular element on a feature to a perfect circle. Roundness could be considered straightness bent into a circle. Note that the circle is being measured for form only (i.e. no MMC is applied).**



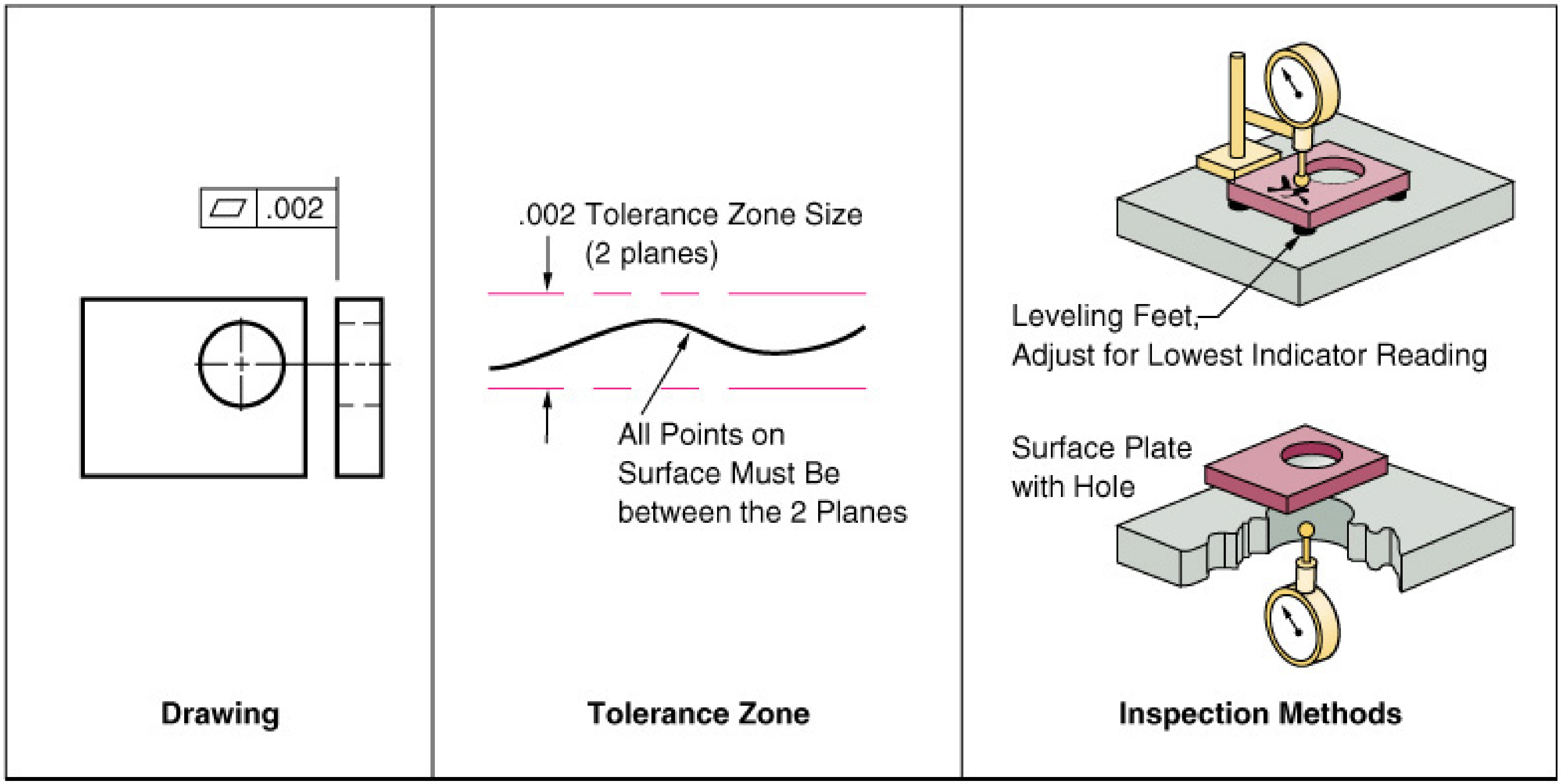
Form control: Cylindricity

- **Cylindricity**. In comparing a feature to a perfect cylinder, three factors are being considered: straightness of all line elements, roundness of all circular elements, and taper (comparison of circular elements to each other). This is probably the most expensive control due to its difficulty in measuring.



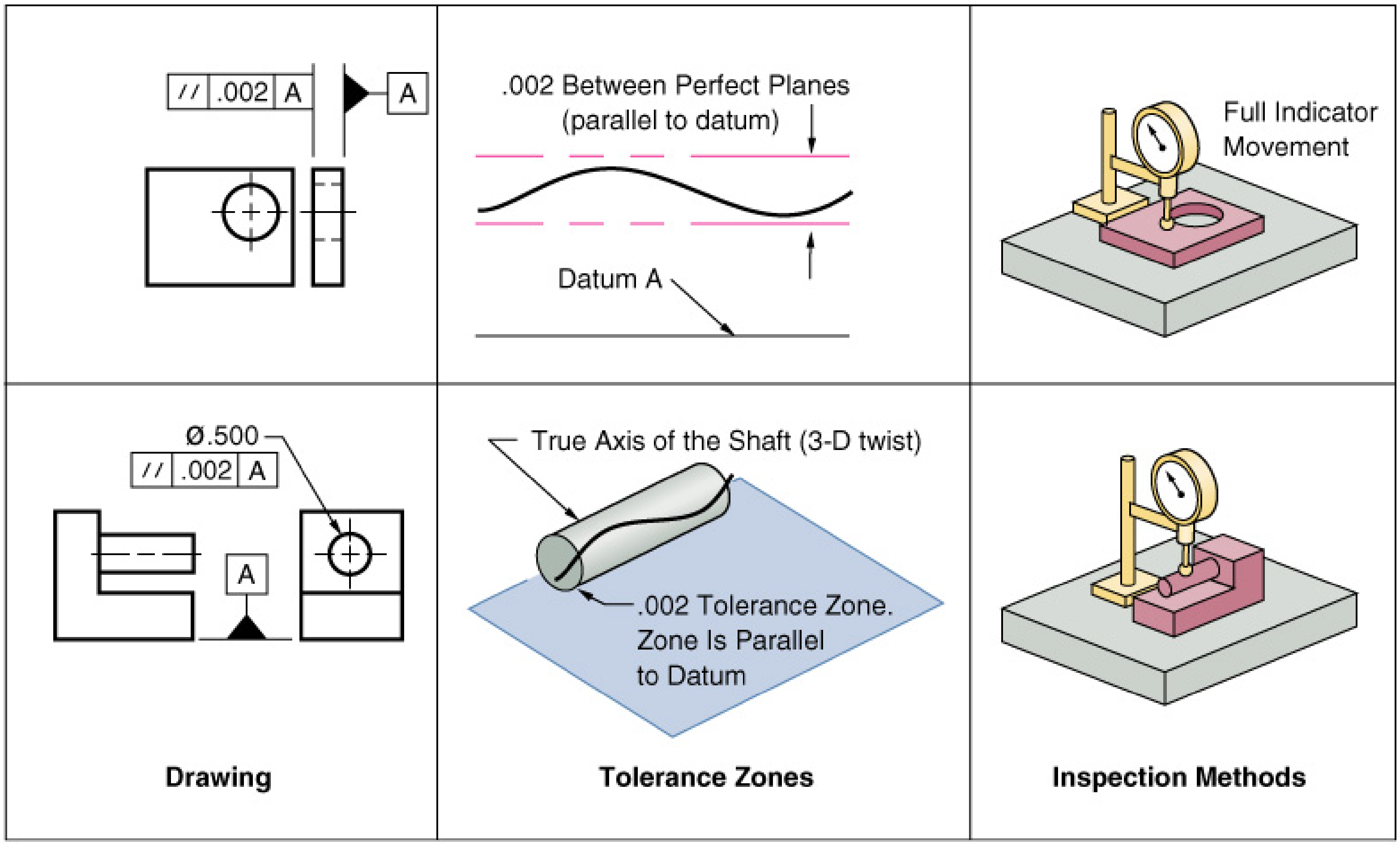
Form control: Flatness

- **Flatness. Evaluates the highest and lowest point on a surface. That is, the surface is compared to a perfect plane (straightness applied in all directions).**



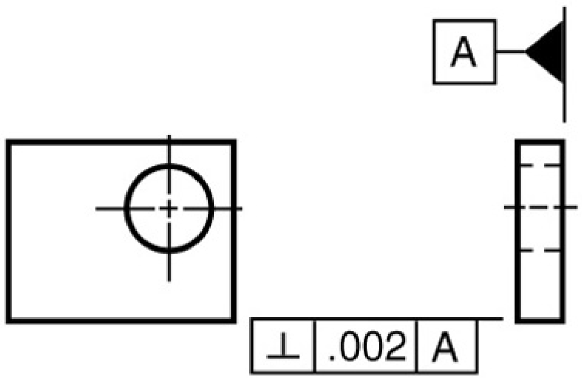
Orientation controls: Parallelism

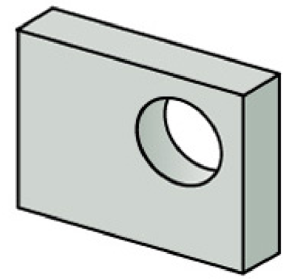
- **Parallelism. This could be considered flatness at a distance or straightness of an axis at a distance.**

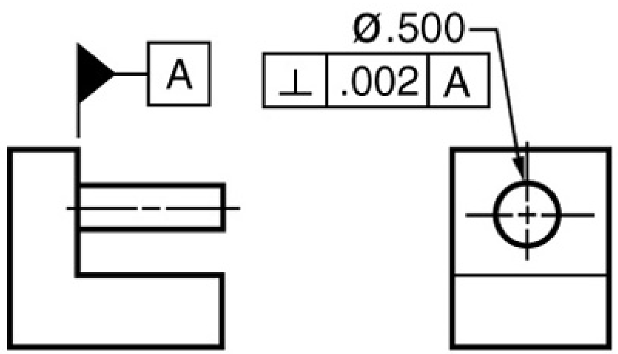


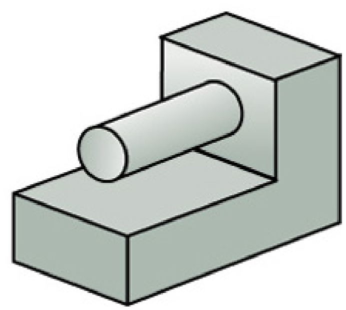
Orientation controls: Perpendicularity

- **Perpendicularity**. This could be considered flatness or straightness of an axis 90 degrees to a datum.



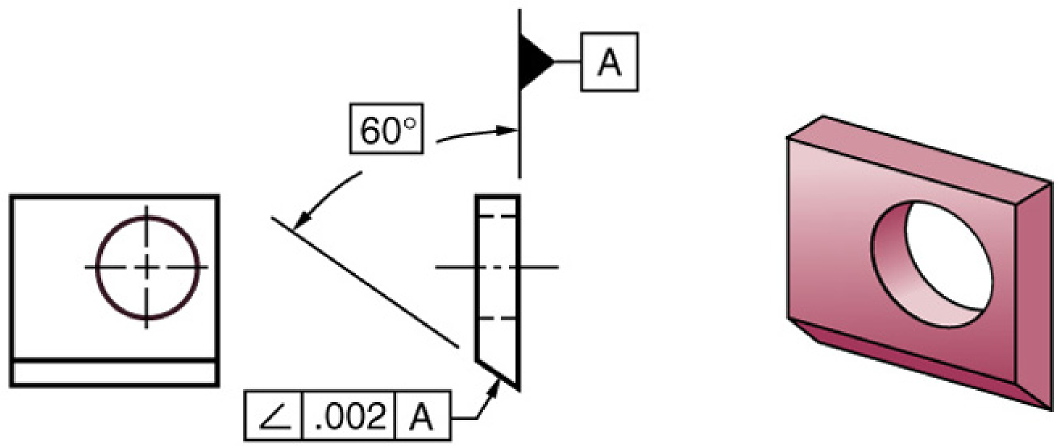


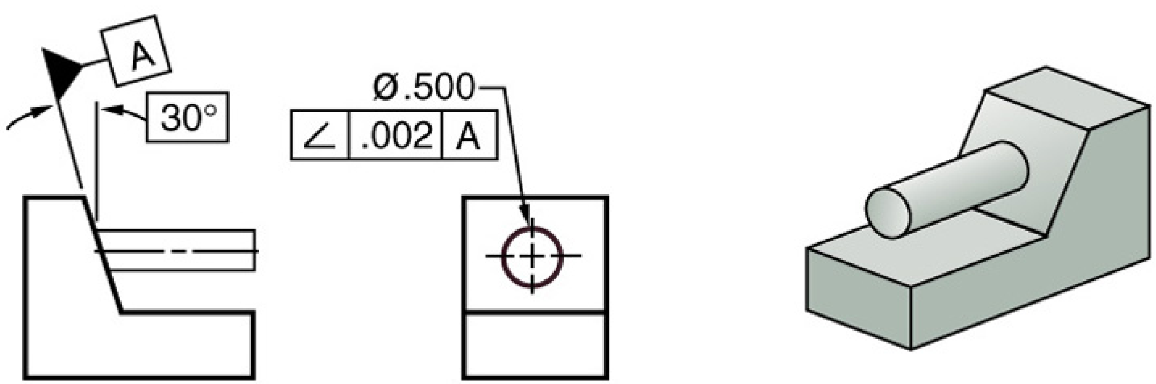




Orientation controls: Angularity

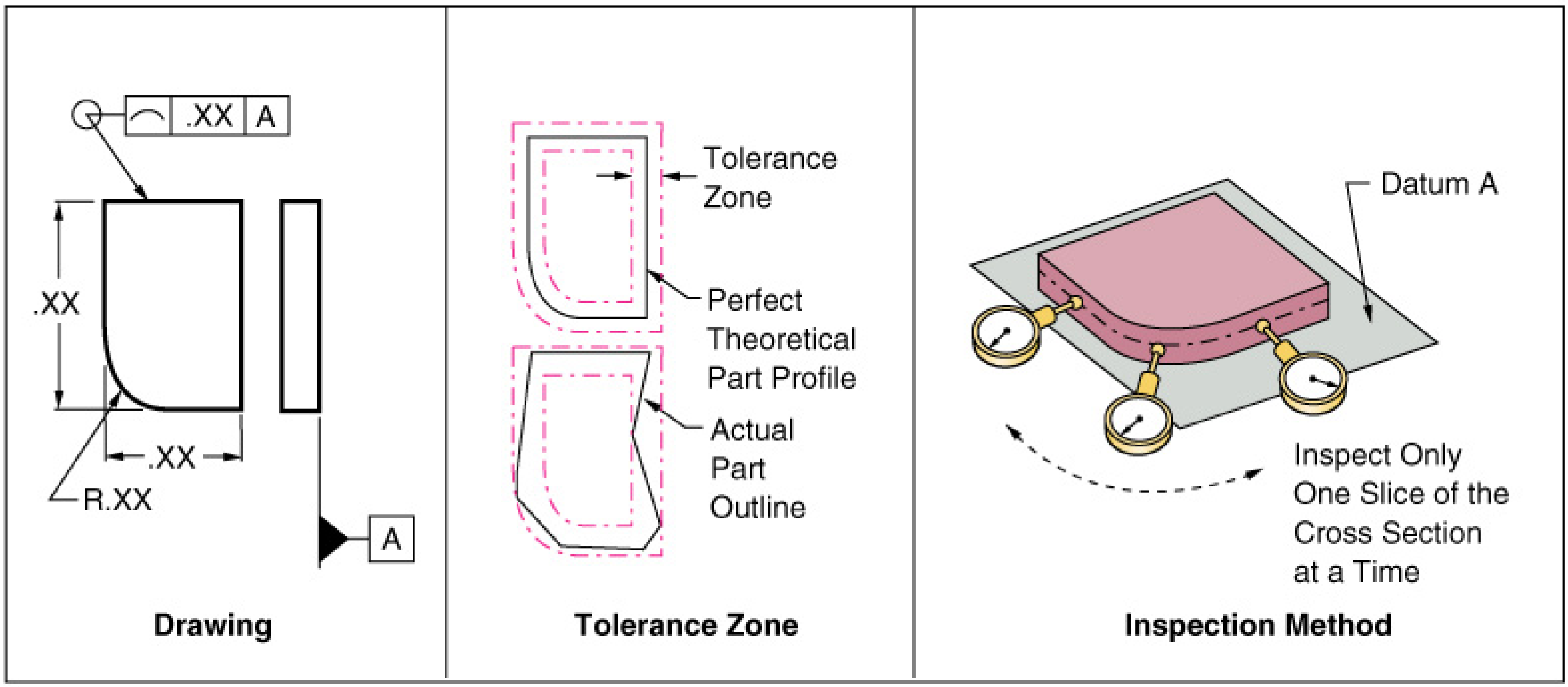
- **Angularity.** This could be considered flatness or straightness of an axis at some angle to a datum.





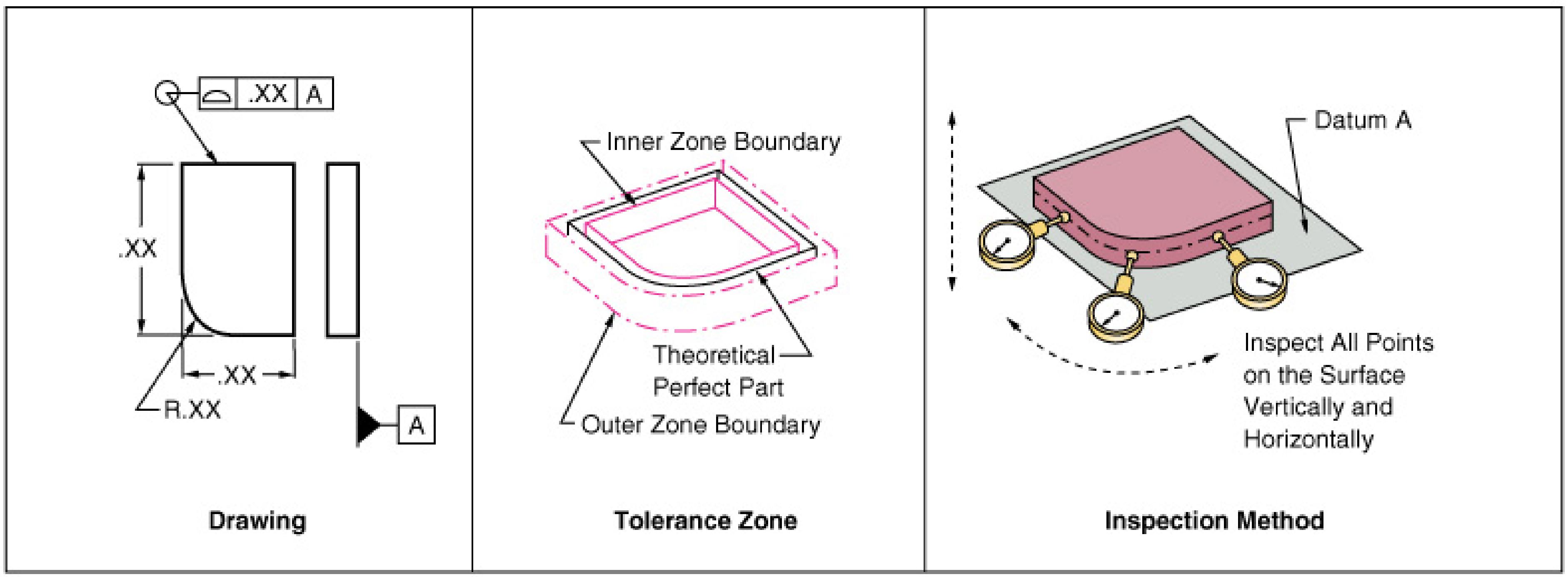
Orientation controls: Line profile

- **Line profile**. This takes a cross-sectional slice or slices of a feature and compares it to an ideal shape. The control shape is usually some contiguous collection of mathematically defined line elements (e.g. straight lines, circular arcs, elliptical curves, etc.).



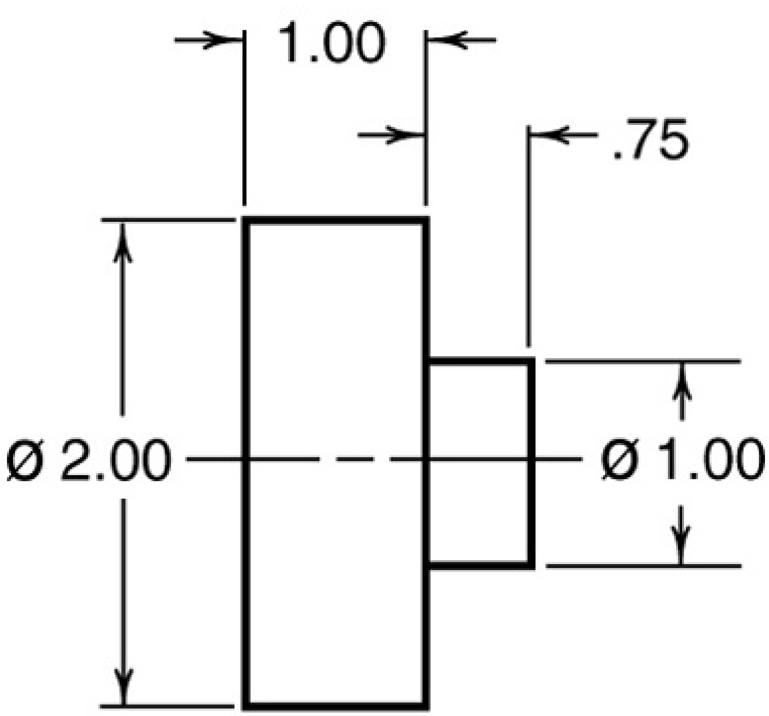
Orientation controls: Surface profile

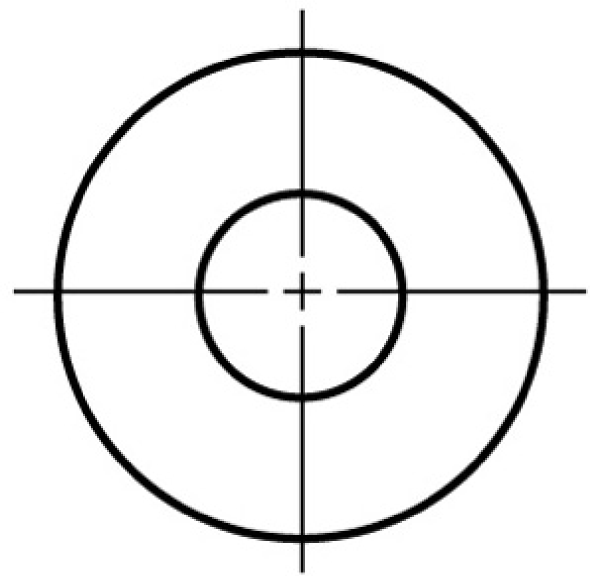
- **Surface profile.** This profile is constructed by stacking line profiles into a 3-D surface.



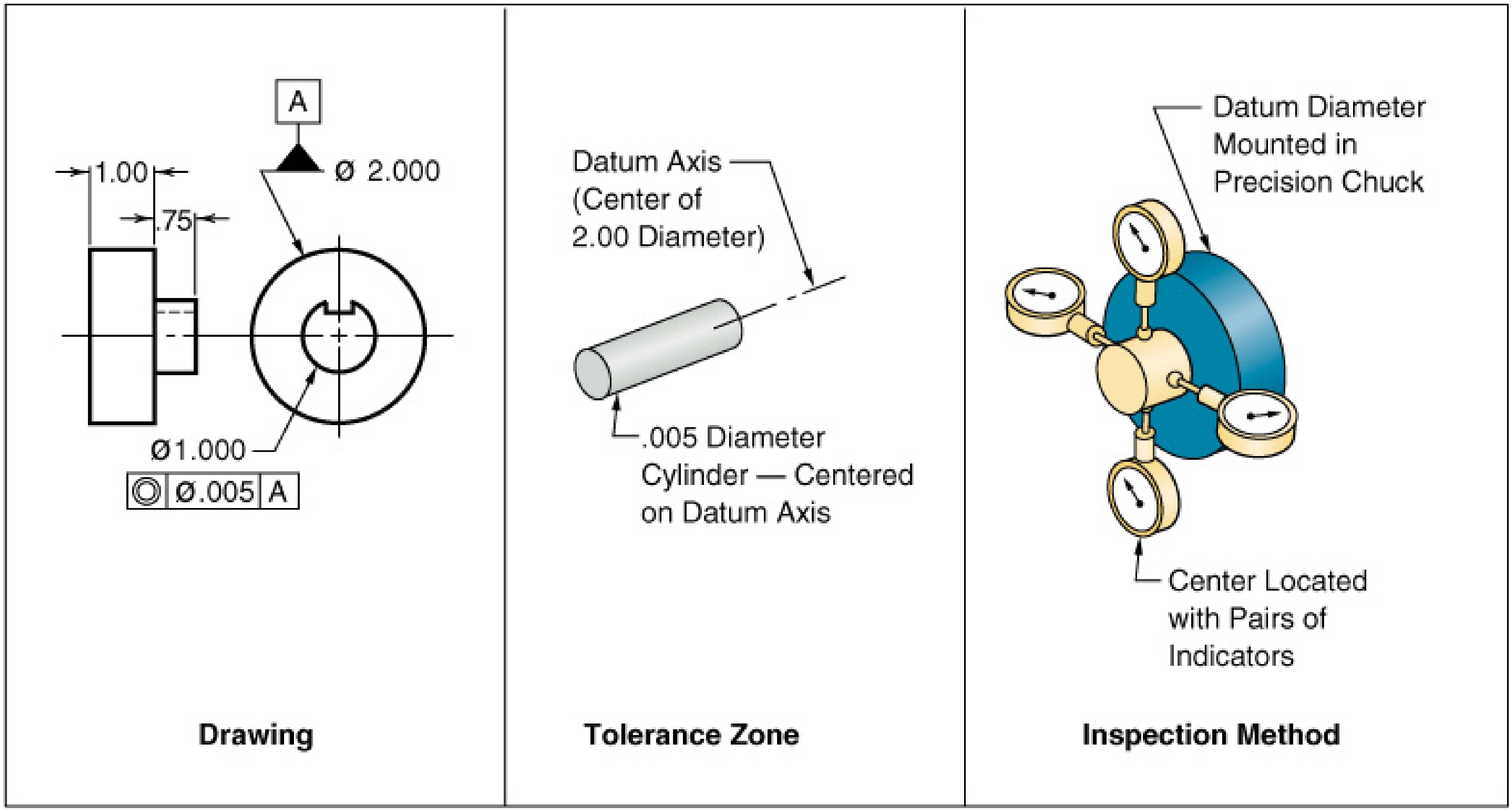
Location controls: Concentricity

- **Concentricity**. The condition in which all cross sectional elements share the same datum axis. This control is important for spinning parts where dynamic balance is important. Though concentricity is a control of position, it is also concerned with shape since the shape can affect the location of the axis.





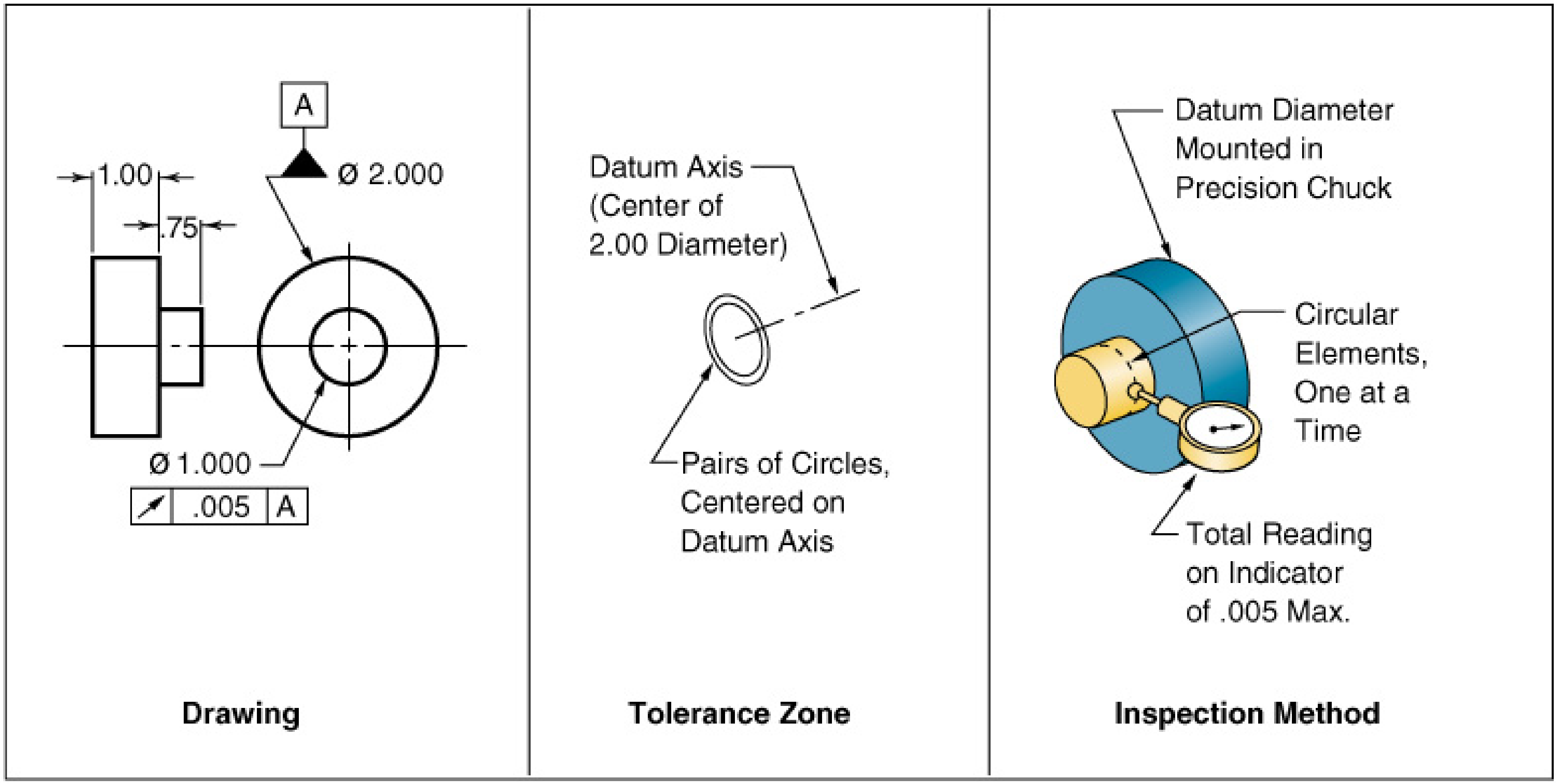
Location controls: Concentricity



Location controls: Runout

- **Runout**. There are two types of runout: single element and total. This control is best described by its method of measurement.
- Single element runout places a gage on the rotating part and measures the amount of fluctuation. With a perfectly centered cylinder, the gage would not fluctuate.
- Total runout has the gage move up and down along the central axis to measure all possible cross sections.

Location controls: Runout



Location controls: Position

- Position is the single most valuable, flexible, and versatile geometric control available.
- A few of the things this control can do is:

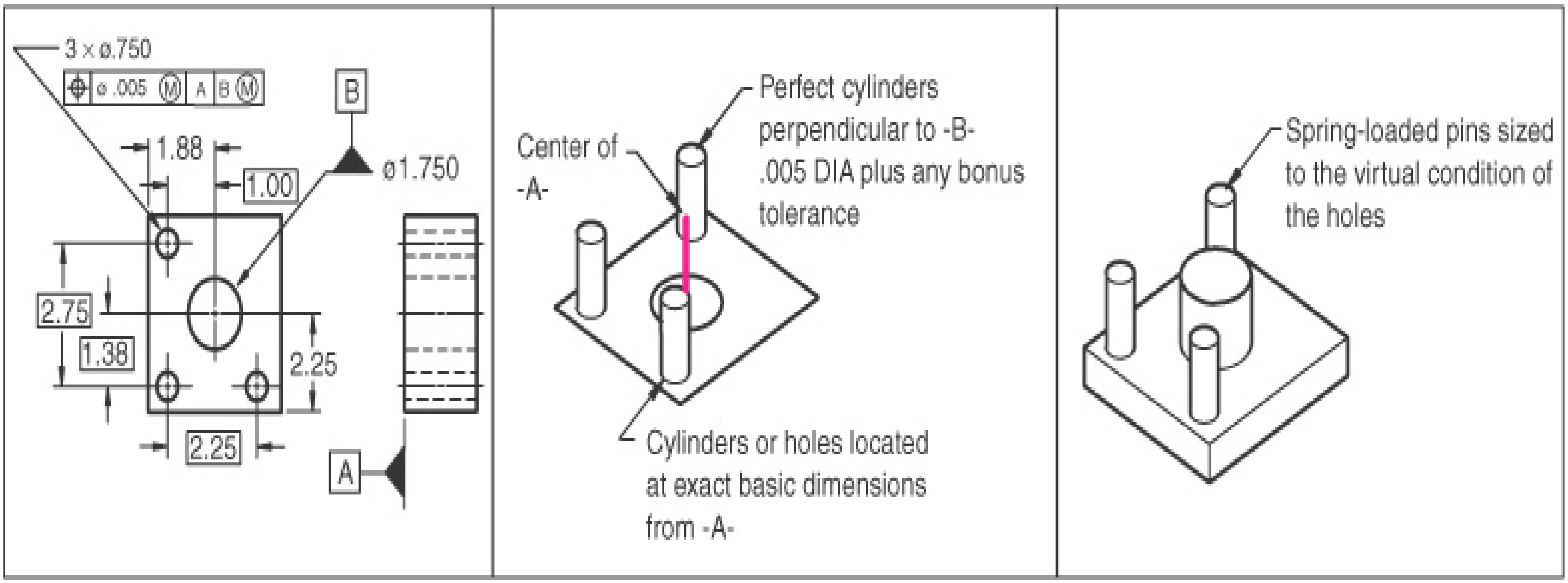
1. Locate holes or a pattern of holes.
2. Locate holes or a pattern of holes.
3. Locate the center of a feature.
4. Keep holes or other features perpendicular or parallel to other features.
5. Keep features straight and round.
6. Allow loose tolerances on the sizes of features while maintaining close control on their locations.

Location controls: Position

- **Hole location from edges.** This is used so that multiple holes are located accurately enough that mating parts with the same number of matching pins will assemble properly.
- A functional gage designed for this use would have perfect cylinders placed at the exact basic dimension from the surfaces representing the datum(s) and from each other.
- This positional tolerance would not have any meaning without taking MMC into account.

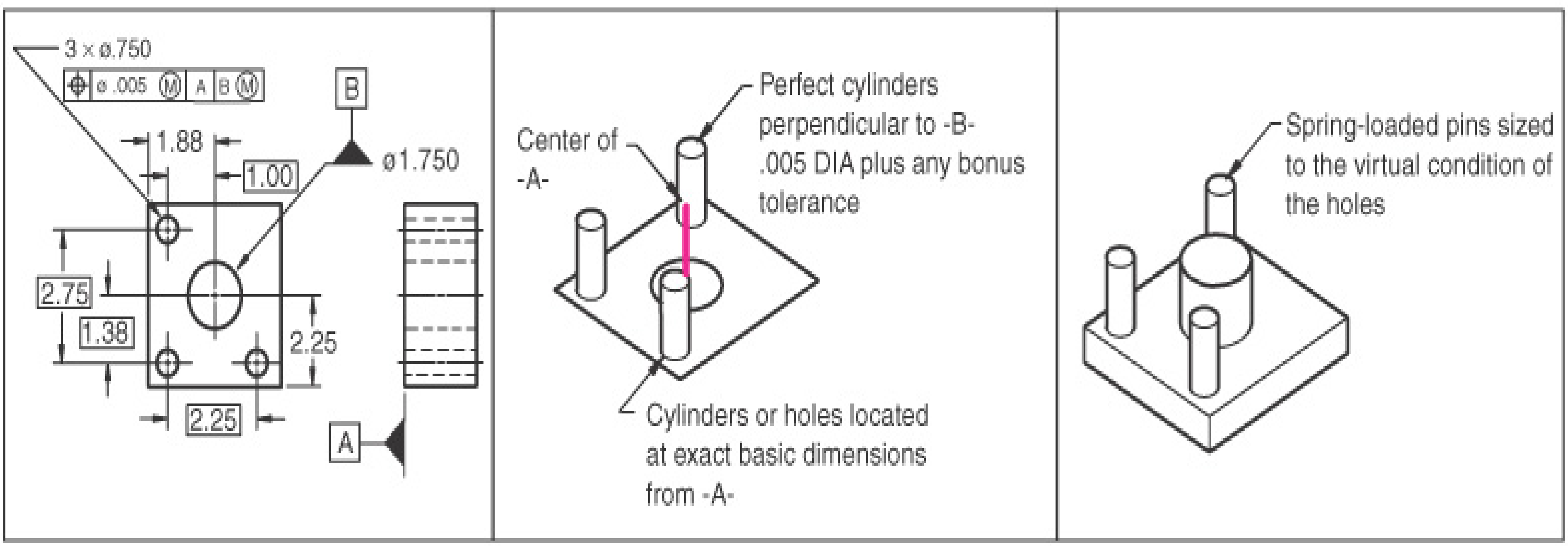
Location controls: Position

- **Hole location from edges.**



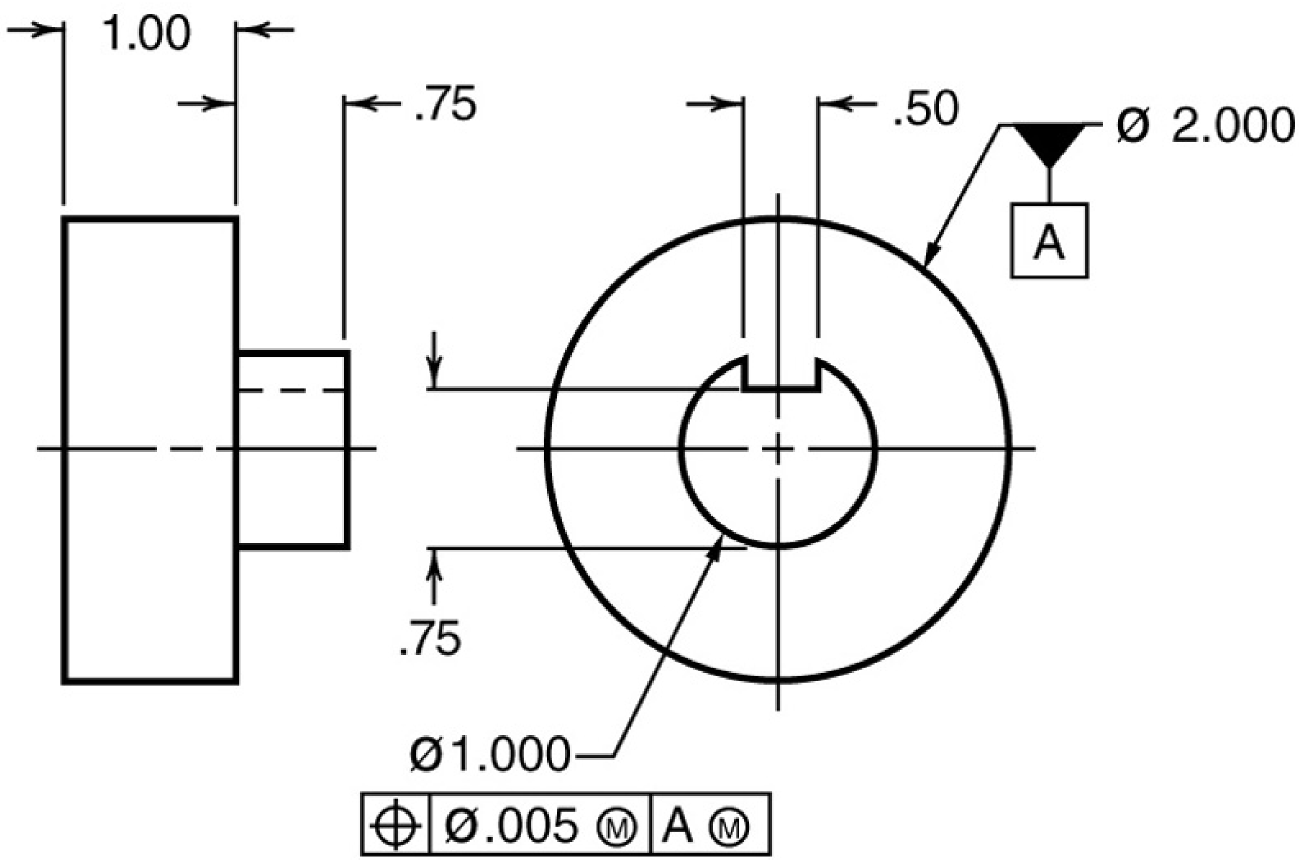
Location controls: Position

- **Hole location from hole.** The key to this control is that holes are located relative to another hole, not the edge of the part.



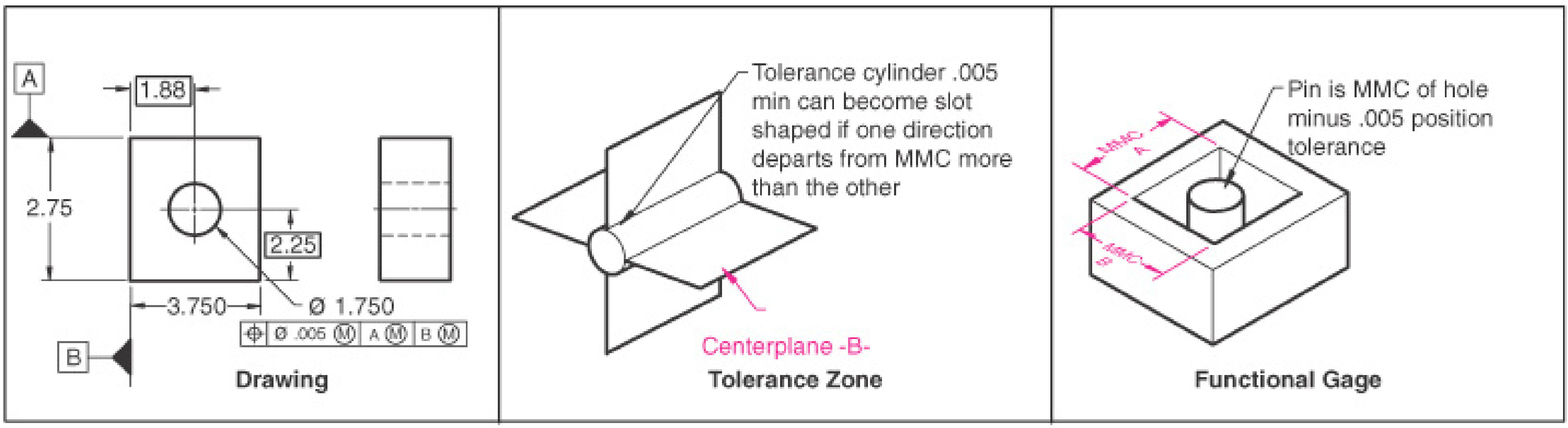
Location controls: Position

- **Coaxial datum and feature. This control ensures that a smaller, internal diameter is centered on a larger, external diameter.**



Location controls: Symmetry

- **Symmetry**. This control ensures that the feature is centered on the datum.



Five step to geometric control

1. Isolate and define the functions of the features/part: Break the part down to its simplest functions.
2. Prioritize the functions:
3. Identify the datum reference frame based on functional priorities.
4. Select the proper control(s).
5. Calculate the tolerance values.