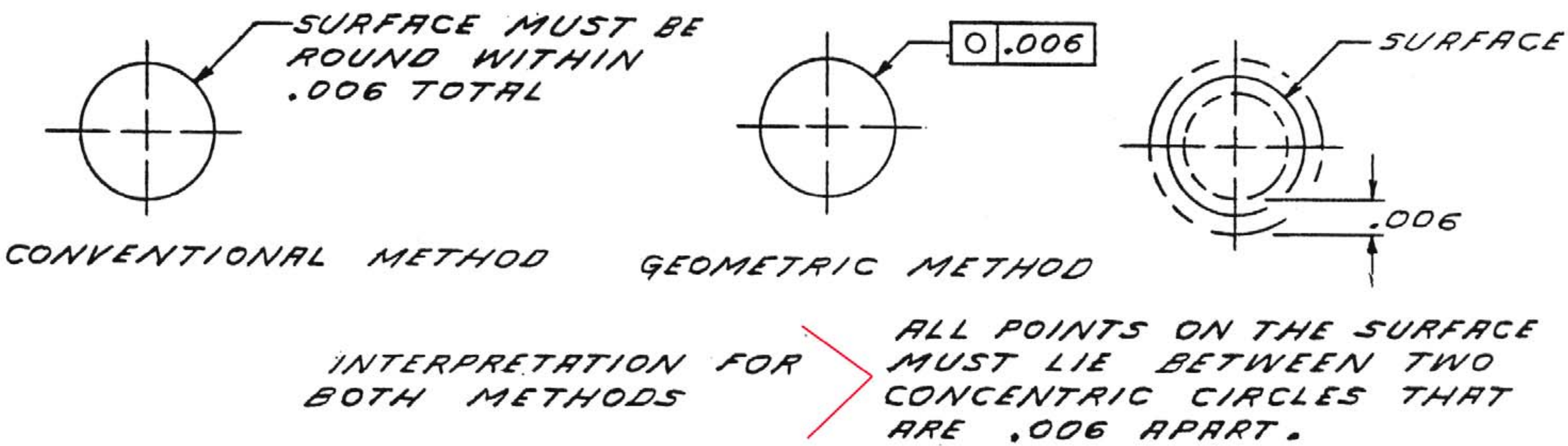
Geometric Dimensioning and Tolerancing

Geometric dimensioning and tolerancing (GDT) is

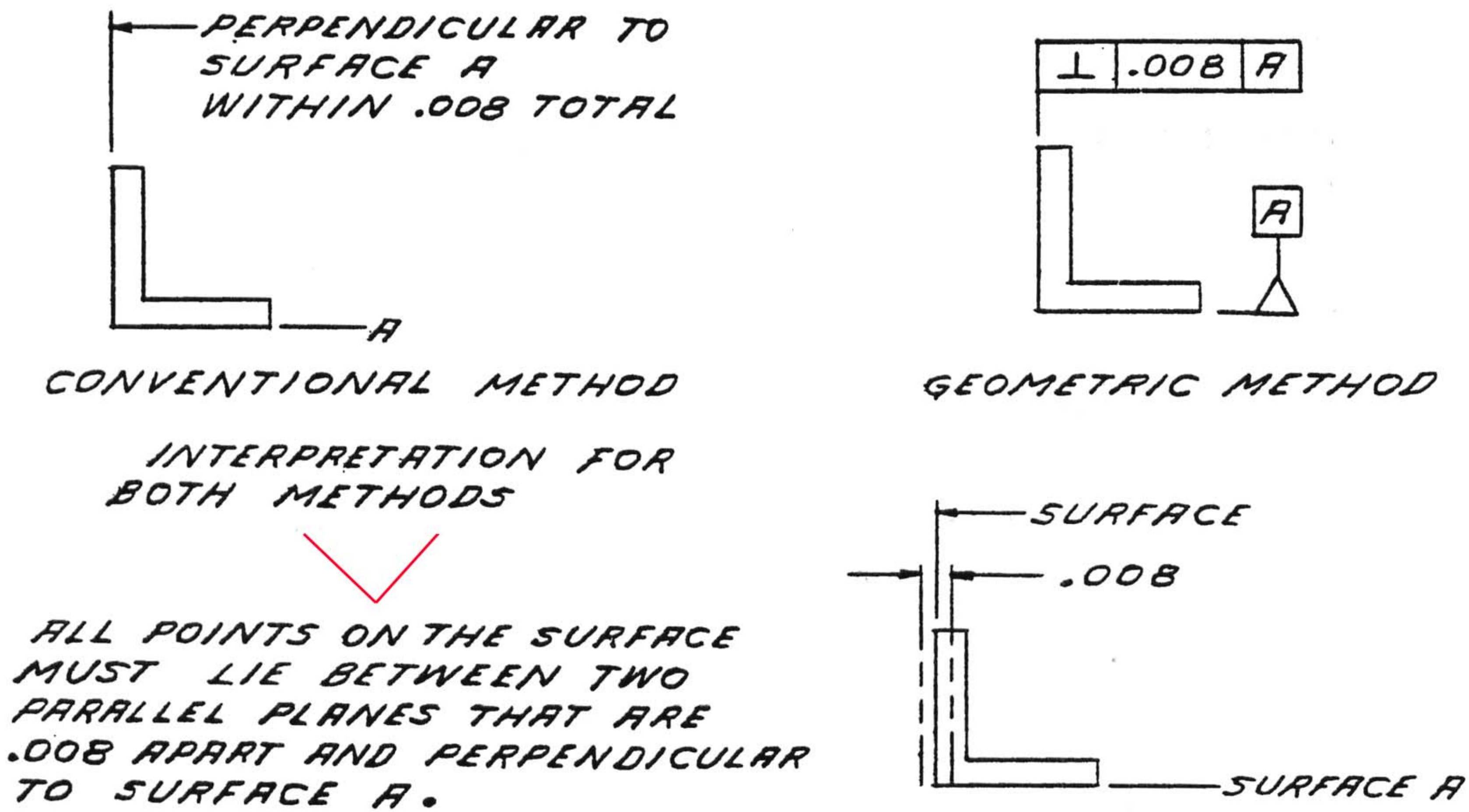
- a method of defining parts based on how they function, using standard ASME/ANSI symbols;
- a system of specifying certain types of dimensions and tolerances.

GDT is a combination of symbols and characters that supplements conventional dimensions and tolerances. Both systems – GDT and conventional – are used on the same drawing.

An example comparing conventional tolerancing to GDT, for a surface that must be round, is shown below

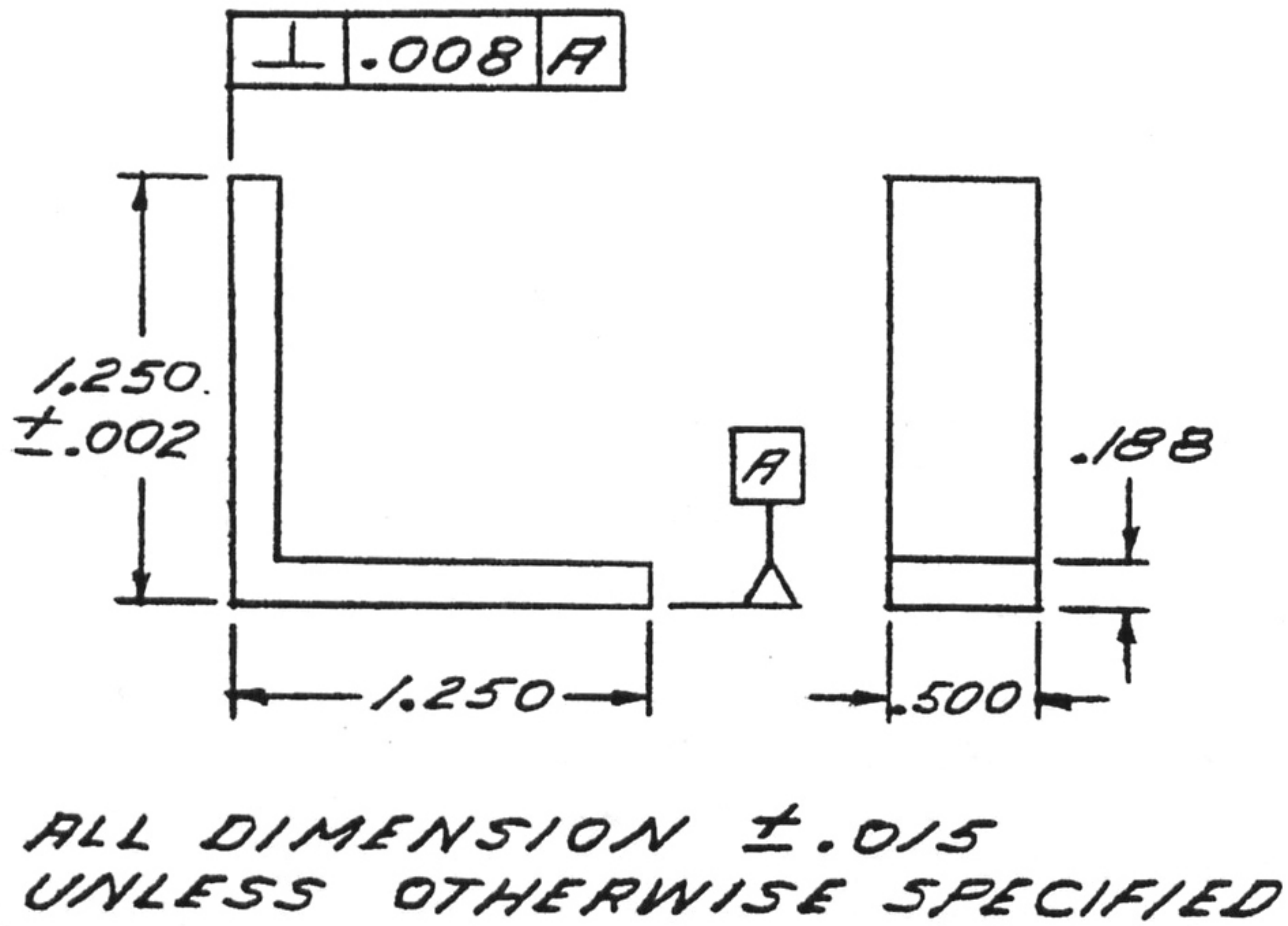


Another example of comparison between conventional tolerancing and GDT for specifying a surface that must be perpendicular:



All dimensions and tolerances that are shown on a print must be maintained.

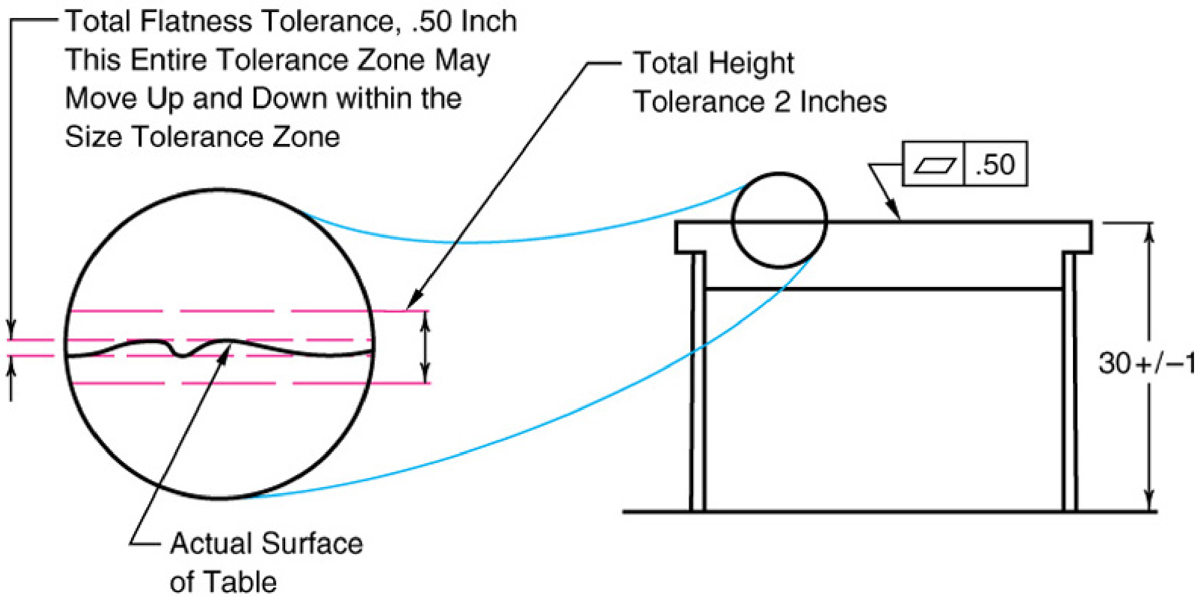
Parts that are made to the following figure must conform to both the conventional tolerances as well as the GDT tolerances:



Geometric dimensioning and tolerancing is used as a supplement to conventional dimensioning and tolerancing.

When should be used GDT?

Example: a table height from the floor to the top is given as 30 inches:



Size tolerances alone are sometimes not enough to meet the design needs of a part. *Relationships between features* may also need to be controlled.

- ✓ Is the top of the table flat? It is, if the tolerance on the 30-inch height is, say, ±1 inch. The top could never be more than 31 inches or less than 29 inches. This means that the top must be flat within 2 inches.
- ✓ If the top must be flatter than that, a tighter tolerance might be, say, ± 1/4 inch. Now the top would be flat to within 1/2 inch. However, the height tolerance becomes too restrictive, causing the rejection of any table that is out of the height tolerance range, even if it is a good table.

We need two independent tolerances for these two features!

This is an example of trying to control the **form** of a part with a **size** tolerance.

Without GDT, the only way to separate the height tolerance from the flatness tolerance is with notes. The note for the table could read something like this:

NOTE 1. TABLE TOP TO BE FLAT WITHIN ½ INCH TOTAL.

Using GDT, we could

- return to the ±1 inch tolerance for the height and
- simply place a flatness control (total of .50 inch, in this example) on the top surface.

This would solve the problem and would communicate the design needs to the manufacturer and the inspector.

The symbols used in GDT create manufacturing and inspection definitions with a minimum of confusion and misinterpretation.

The questions that should be asked continuously during the design phase are:

- What kind of part would be rejected with these tolerances?
  - Will the rejected parts be unusable?
    - Will we reject all of the parts we cannot use?

For our table example, the answers are as follows:

1. Any table that is too high or too low (over 31 inches or under 29 inches), even if the top is perfectly flat.

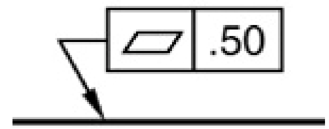
   What good is a perfectly flat table if it is only 4 inches off the floor?

2. Any table for which the top is not flat enough, even if the table is within height limits.

   What good is a 30-inch table if the top is too wavy to set a cup of coffee on it?

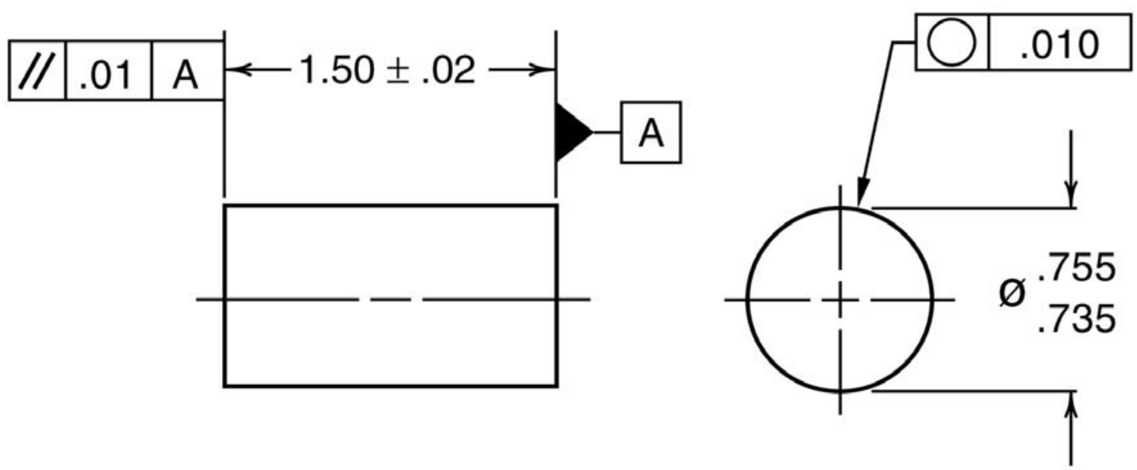
GDT Symbols

At the heart of GDT is a rectangular box, the **feature control frame**, where the tolerancing information is placed.



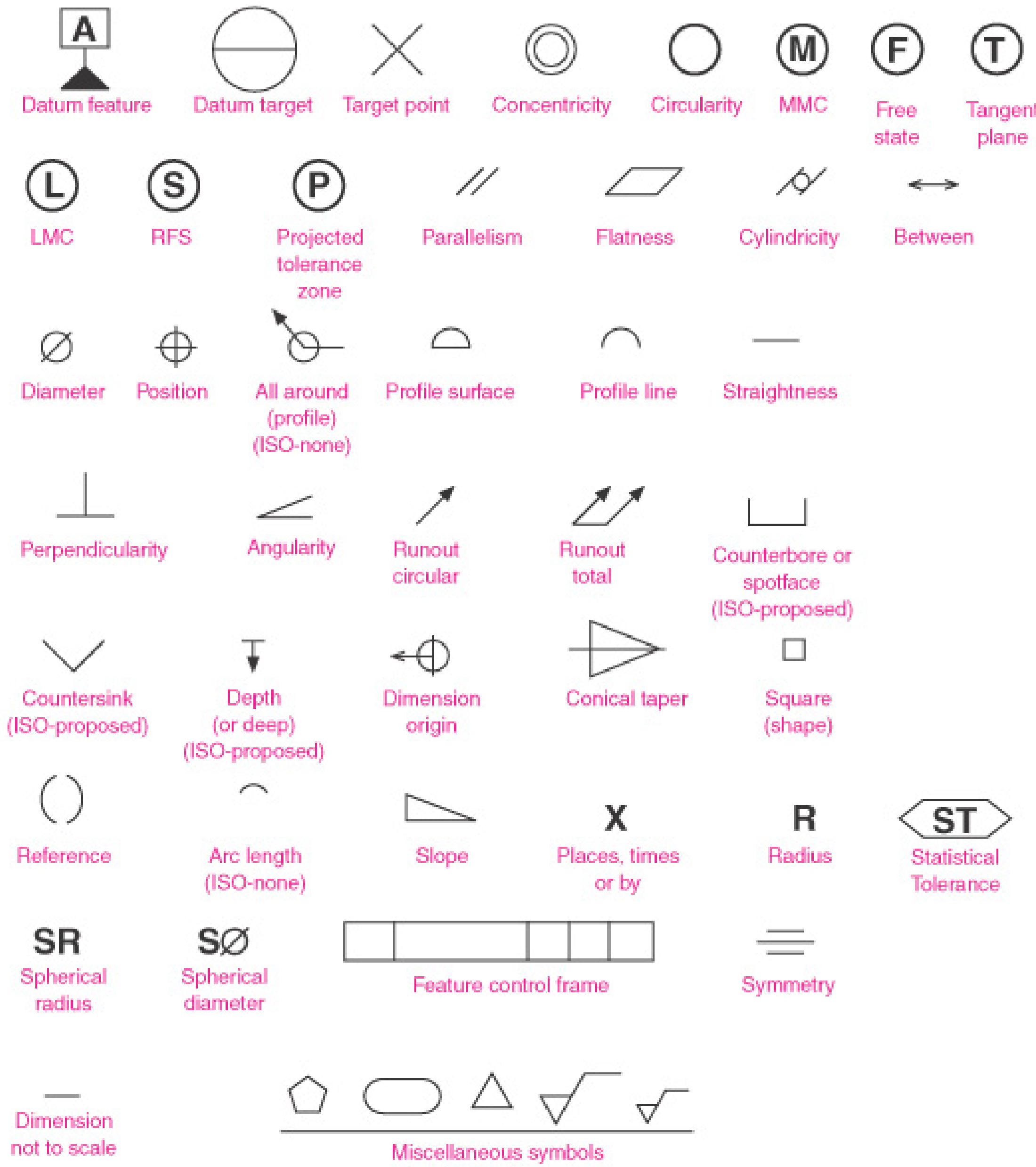
In the figure of the table example, the feature control frame is divided approximately in half. On the left side is the symbol for flatness, which states the actual geometric control for the table top. On the right side is the total size of the tolerance zone. In this case, the tolerance zone is 0.50 inch total.

Other examples of feature control frames:



The left example defines the parallelism between two surfaces, the right one shows the circularity.

Different **GDT symbols** for specifying compact requirements:



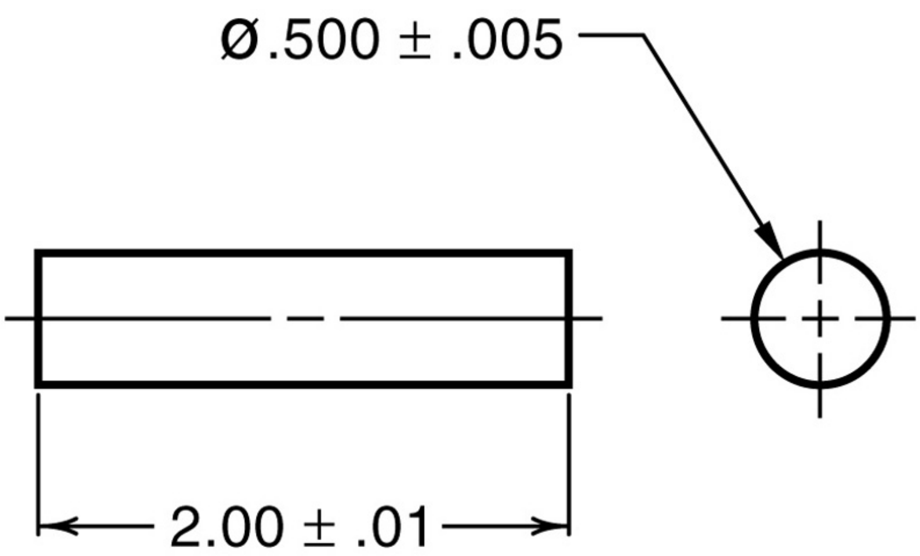
GDT Rule 1

Geometrics is the part of tolerancing that has to do with the shapes and positions of features. *All dimensions include geometric controls*, whether or not geometric tolerancing symbols are used.

This idea has been formulated in the **Rule 1:**
Where only a tolerance of size is specified, the limits of size of an individual feature prescribe simultaneously the extent to which variations in its geometric form, as well as size, are allowed.

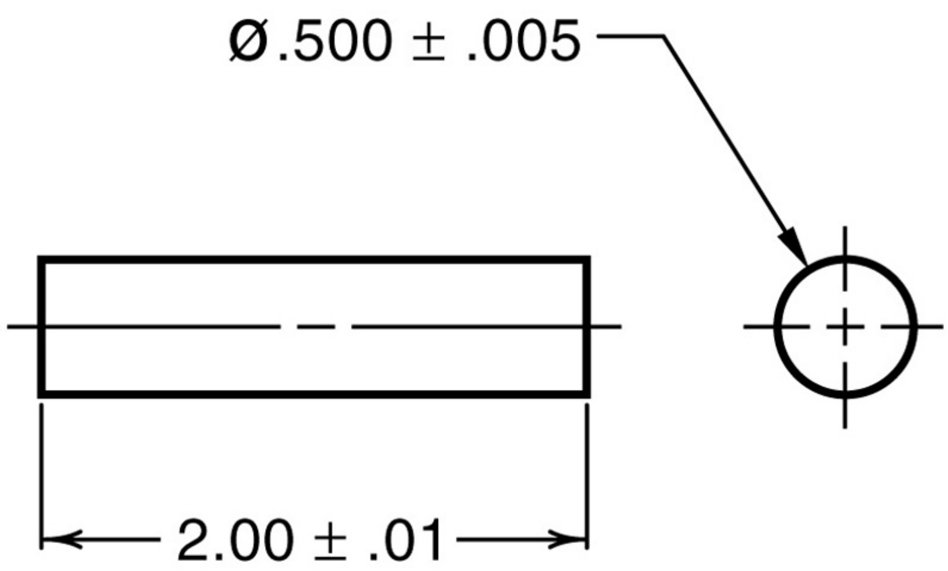
In other words: Size tolerance by default includes geometric tolerance of the same feature with the same values.

For example, if a shaft is dimensioned as 0.500 inch in diameter, this controls the circularity of the shaft:



In this case the circularity control is the equivalent of 0.005.

In addition, if the shaft drawing is fully dimensioned the following geometric controls would be required under Rule 1:

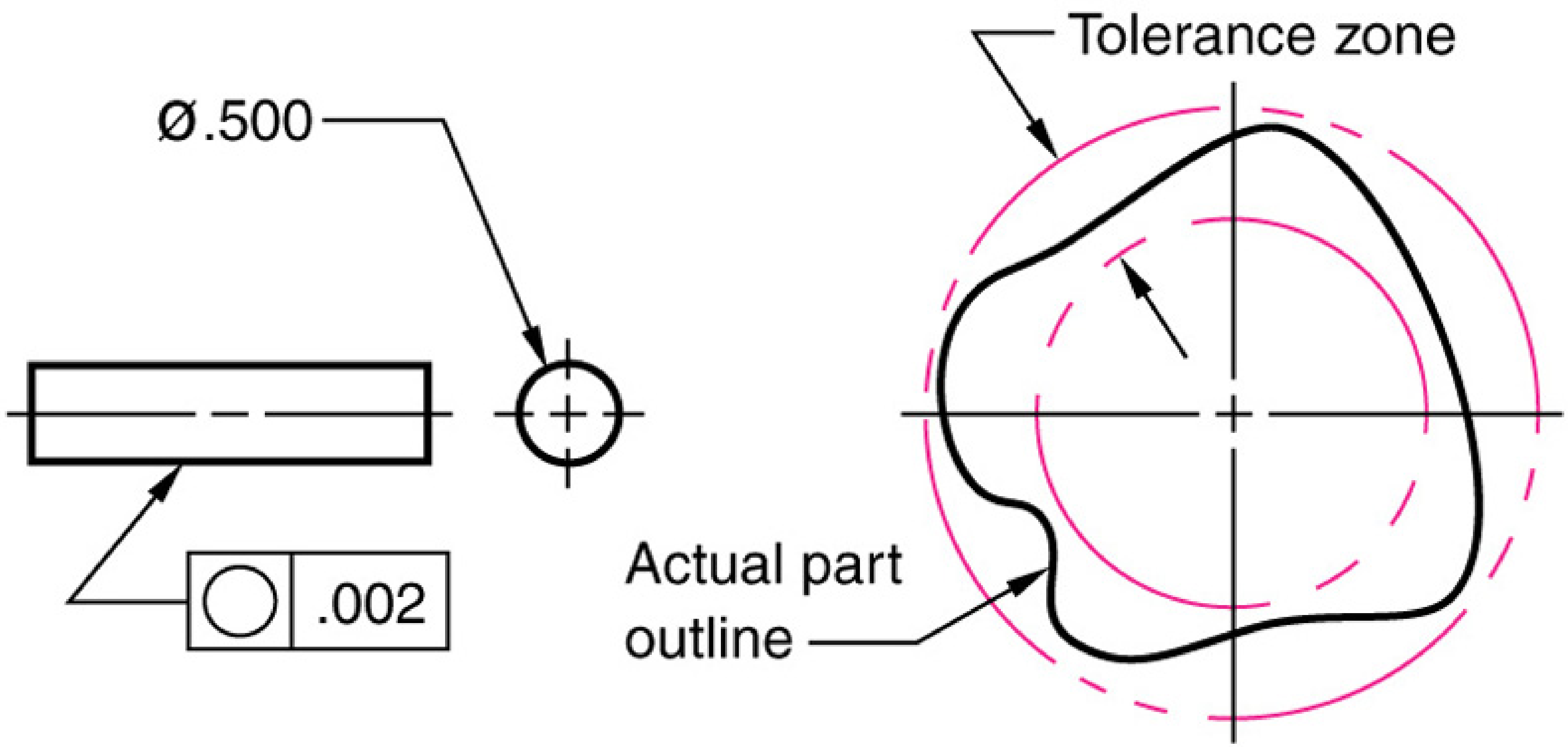


1. *Straightness of the line elements of the cylinder*. The line elements of the cylinder cannot be more “bent” than the total size tolerance.
2. *Flatness of the ends of the shaft.*
3. *Parallelism between any two opposite line elements.*

The line elements of the cylinder must not only be straight, they must also be parallel to one another, within the size tolerance of the cylinder.

The key element is as follows: All dimensions have built-in (natural) geometric control. Additional symbols should only be used when these natural controls must be refined.

For example, in the following figure the roundness of the shaft is controlled first by the diameter tolerance:



If the roundness control were removed, the shaft would still have to be round within the limits of the diameter tolerance, per Rule 1.

Maximum Material Condition

From the size tolerancing we know that *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 that the part will weigh its maximum.

There are three symbols in GDT relating to **material conditions**:

1. **M** - Maximum material condition.
2. **L** - Least material condition. This is the opposite of MMC (the part will weigh its minimum).
3. **S** - Regardless of feature size (RFS). This indicates the material condition is not to be considered. This is used where a tolerance is specified and the actual size of the controlled feature is not considered when applying a tolerance. (Example: cylindricity tolerance for the shaft is applied, but actual diameter is not considered). *In this case the GDT must be specified, since we don't care about the size, and size dimension doesn't have default GDT included.*

The term **Departure from MMC** is often used in GDT. It refers to how much *less* material a part has than its MMC. For a hole, this would mean how much larger it is from MMC. Departure from LMC would be the opposite.

Datums and Datum Features

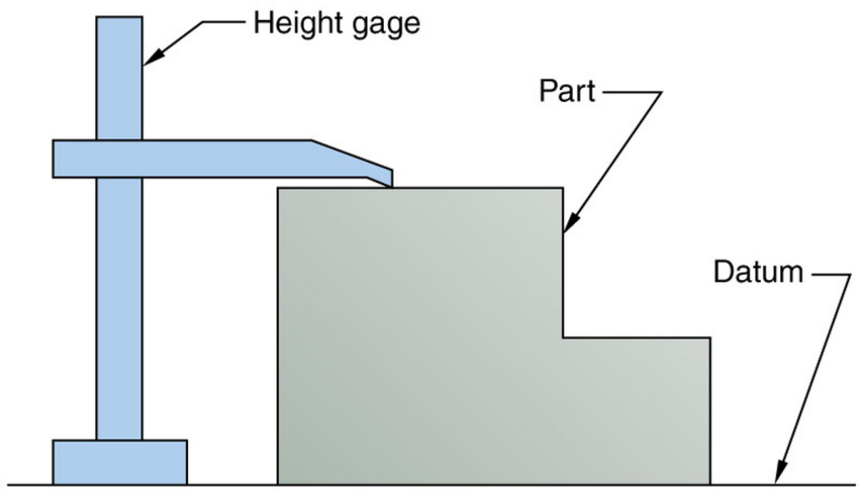
A **datum** is a starting point for a dimension. Datums are theoretically ideal locations in space such as a plane, centerline, or point.

Examples:

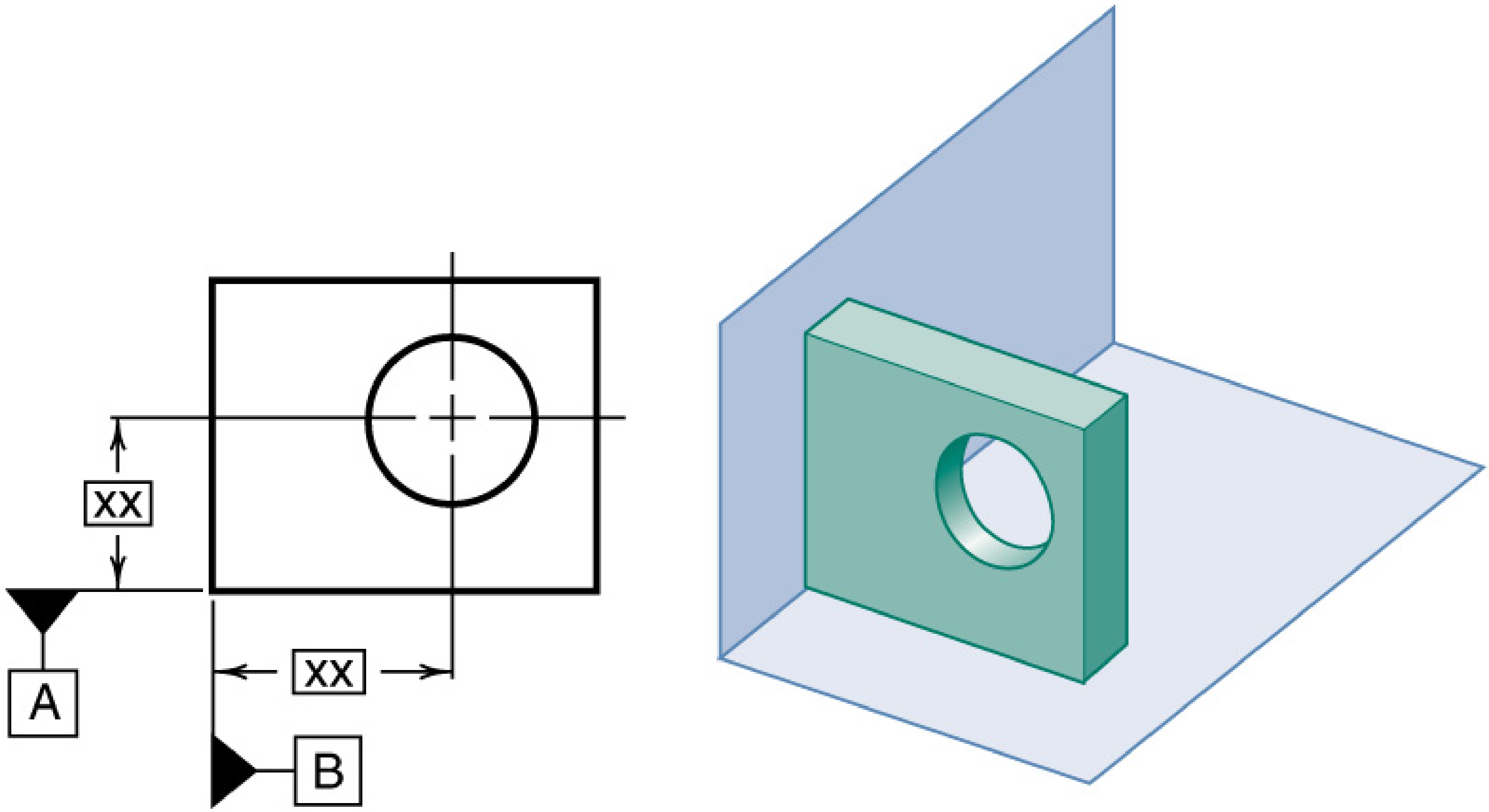
- Center line of a shaft
- The point at the center of a sphere
- Plane surface of a rectangular block or plate.

A datum may be represented either directly or indirectly by an inspection device.

The surface of the object which is placed on the inspection device representing the datum is called the **datum feature:**

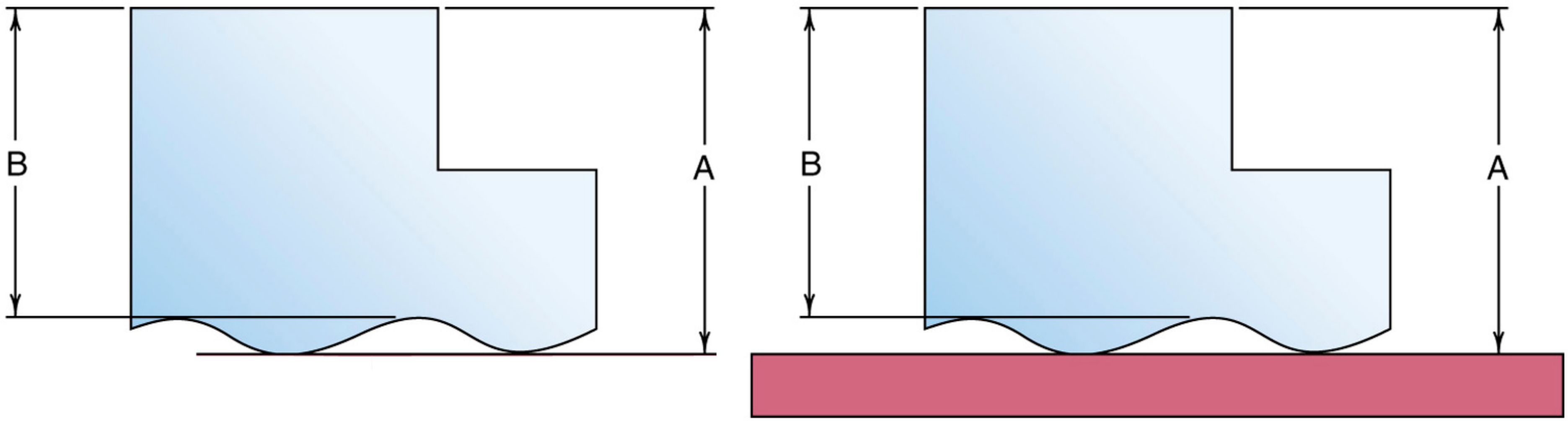


These datum features are clearly marked in the drawings to indicate which are the reference surfaces to make measurements from.



*These features should be machined firs*t. All dimensions should be shown from these surfaces.

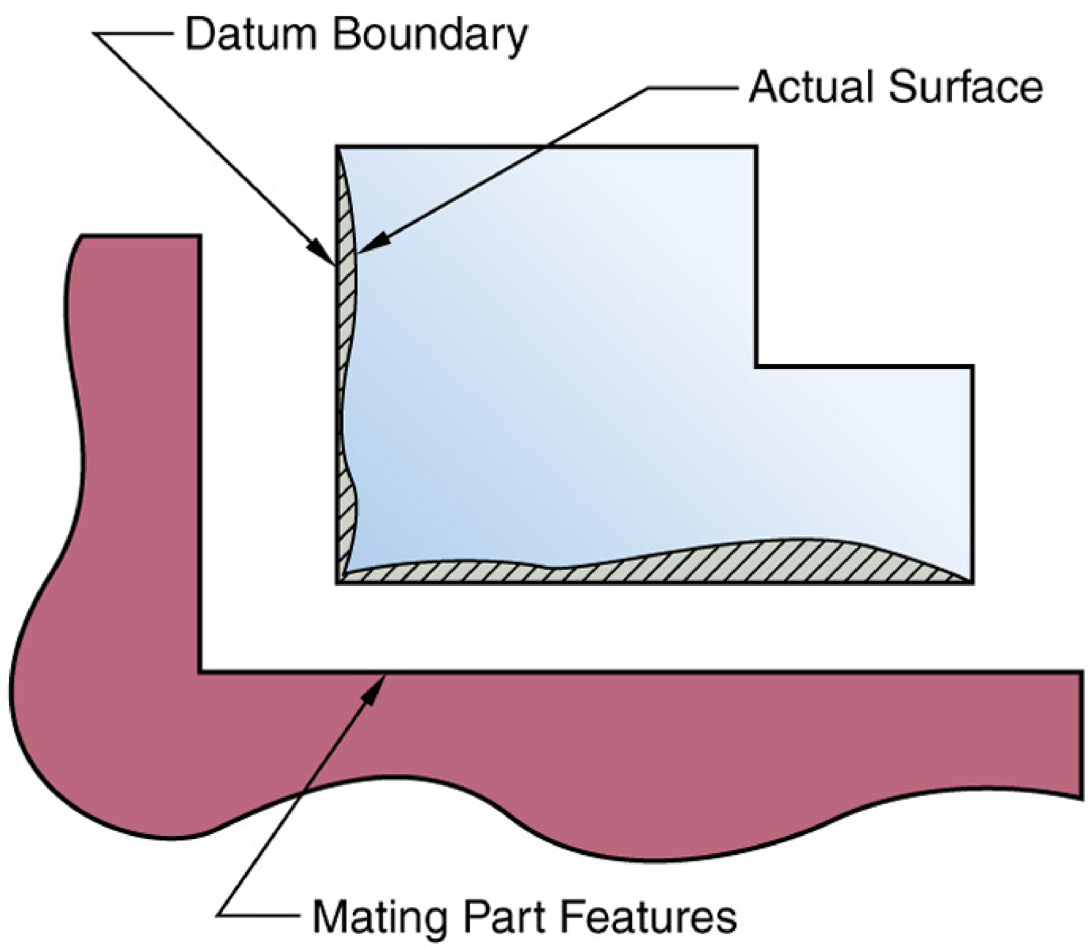
Once a datum is established, the measurements can be taken from it rather than from the feature. The importance of doing this is illustrated by the figure:



When measured at location A, the part seems to be within tolerance; however, measuring at location B creates a problem because the measurement is being taken at a high spot of the surface roughness, which may make the part appear to be unacceptably out of tolerance.

A better approach is to take a measurement of the largest distance, not a spot location. This distance can be derived by laying the part on a surface plate and measuring from the plate, rather than from the part itself.

Datums are not only used internal to a part but, more importantly, in relation to mating parts in an assembly. If one part is mounted on another, the mating surface need not be perfect. The two parts will fit well if the mating surfaces (the chosen datum features) have no protrusions that extend beyond the design planes (datums).



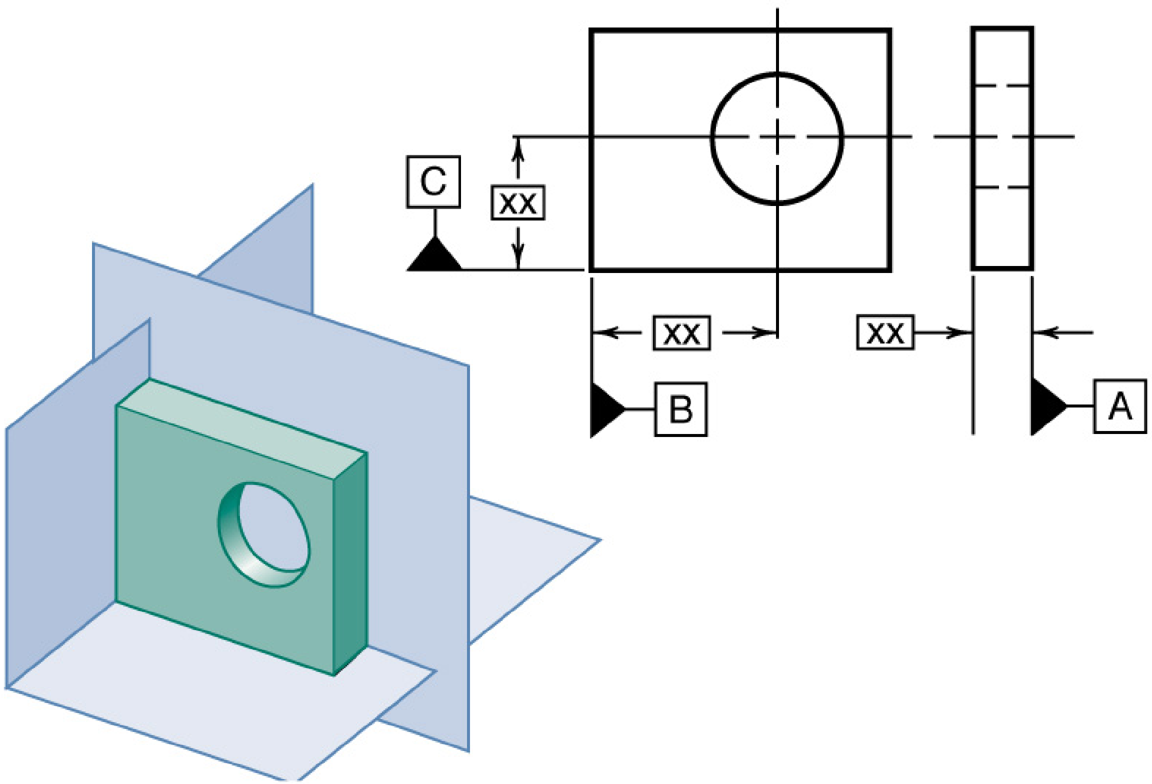
However, it may be acceptable for the surfaces to have depressions, which will not interfere with the mating of the parts.

Space has six degrees of freedom. Part may move up, down, left, right, forward, and backward.

It is necessary to locate the part in space while it is being made, inspected & used.

Datums are the locators and the **datum reference frame** is the six-direction locator of a part in space.

The six degrees of freedom are the plus and minus directions along the three Cartesian coordinate axes. Another way of looking at the frame is as three orthogonal planes. On all stages of design, manufacturing and inspection the same datums should be used for measuring.

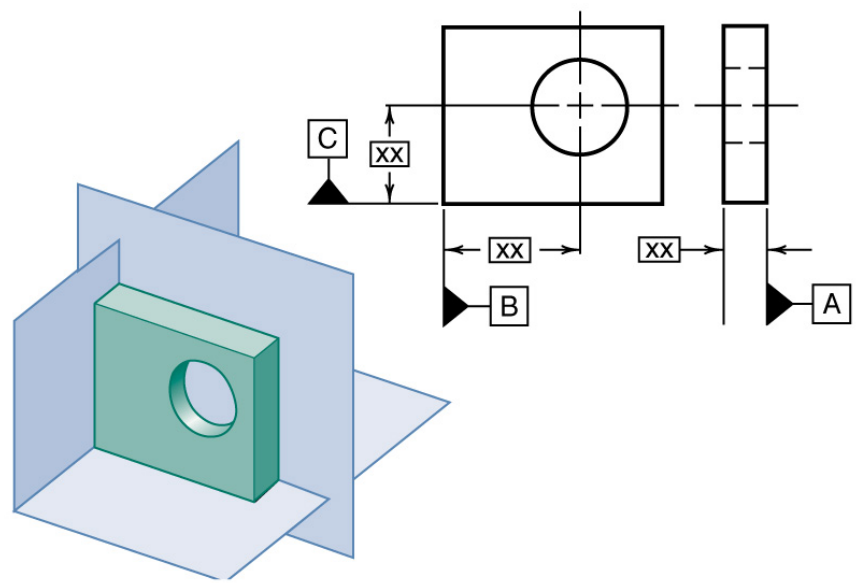


The critical part of datum selection is identification of the **primary datum**. In many cases, this will be the only datum, and it has to be chosen based on a number of criteria:

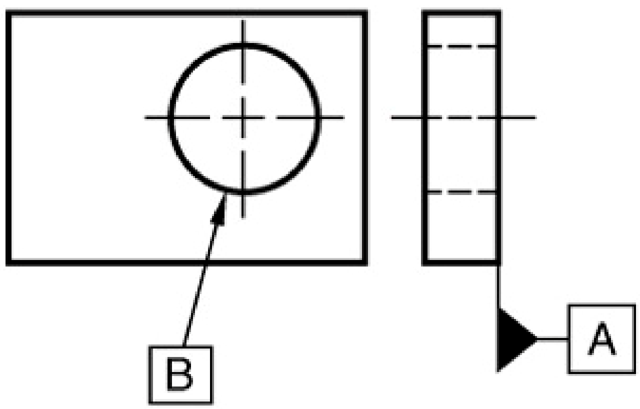
- **Stability**. This is the most important feature. Often dictating that *the largest, flattest surface is chosen*, which is critical for dimensioning and tolerancing. Dimensioning quality depends on the quality of datum.
- **Functional relationship**. How does the feature mate/interact with other features? *Mating surfaces preferably should be assigned as datums*.
- **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.

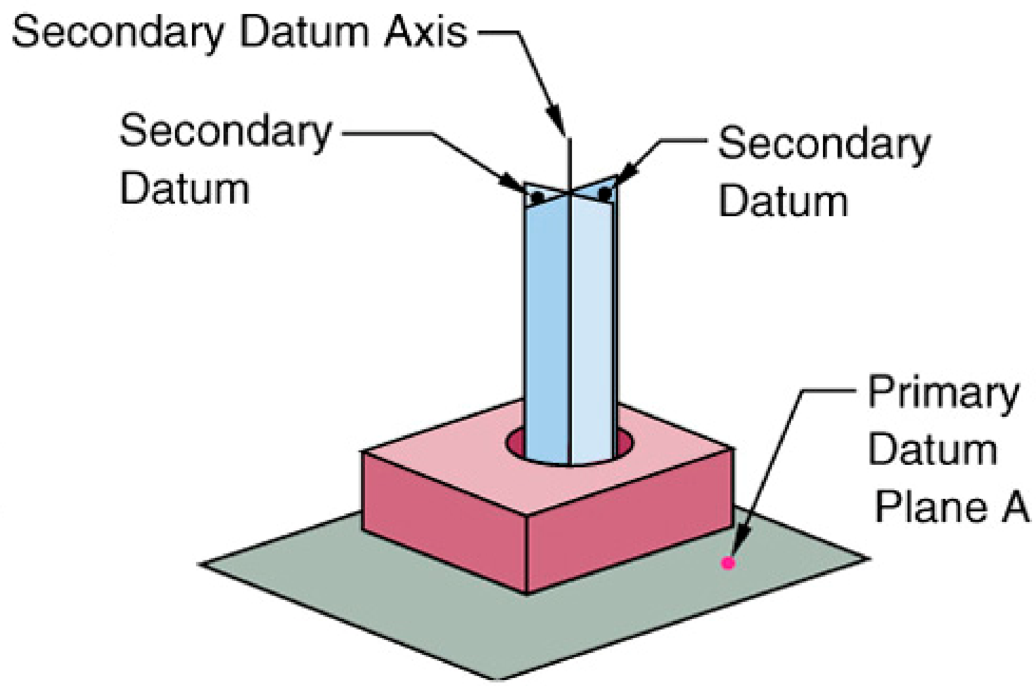
Secondary and **tertiary datums**, if needed, should be located mutually perpendicular to each other and to the primary datum. The secondary datum should be a functional feature, and it must be perpendicular to the primary feature. The tertiary datum must be perpendicular to both the primary and the secondary.

The primary, secondary, and tertiary datums are identified with the letters A, B, and C, respectively.



The most useful secondary datum feature can be a hole that is perpendicular to the primary datum. The surface creates the primary datum, and the hole creates two perpendicular planes.





This reference frame is the most commonly used and is called the *plane and cylinder reference frame*. Here the primary datum plane is identified with the letter A, and secondary cylinder surface is identified with the letter B.

Geometric Controls

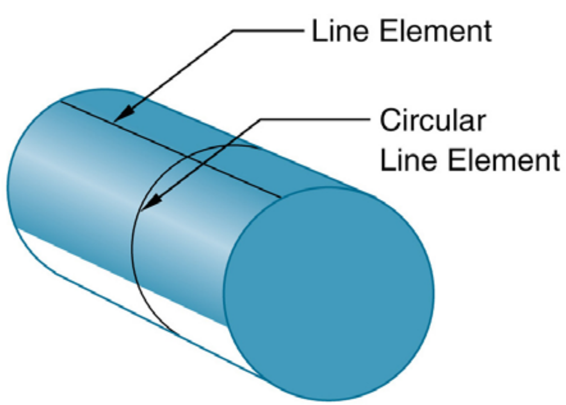
Geometric controls fall into three major categories:

- ✓ **Form**
- ✓ **Orientation**
- ✓ **Position**

1. Form controls are a comparison of an actual feature to a theoretically perfect one.

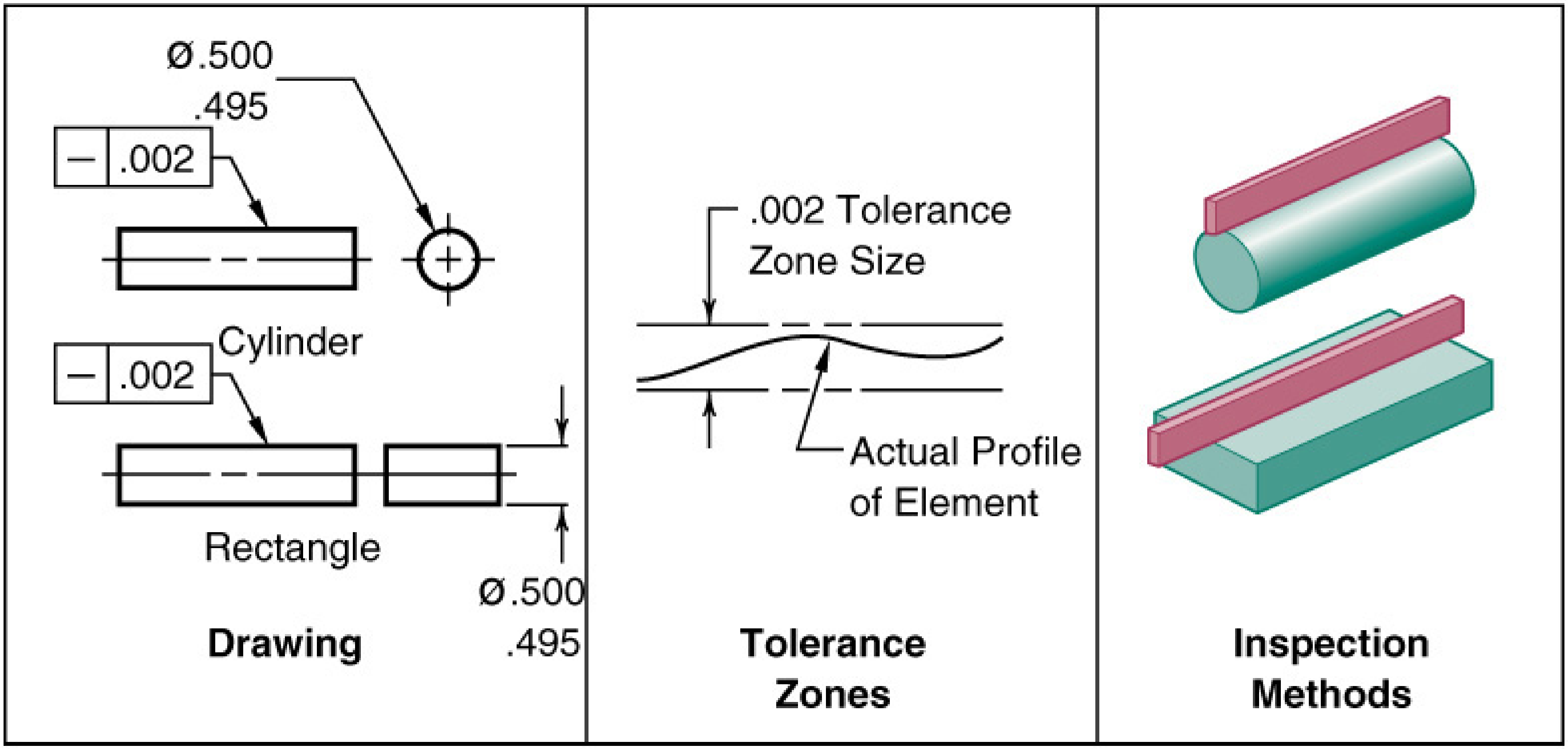
These controls include:

- **Straightness**. All form controls are variations and combinations of straightness. Straightness itself is based on a *line element*. A line element is any single line on any surface, in any direction, (ex., cylinder generating line or circular line):

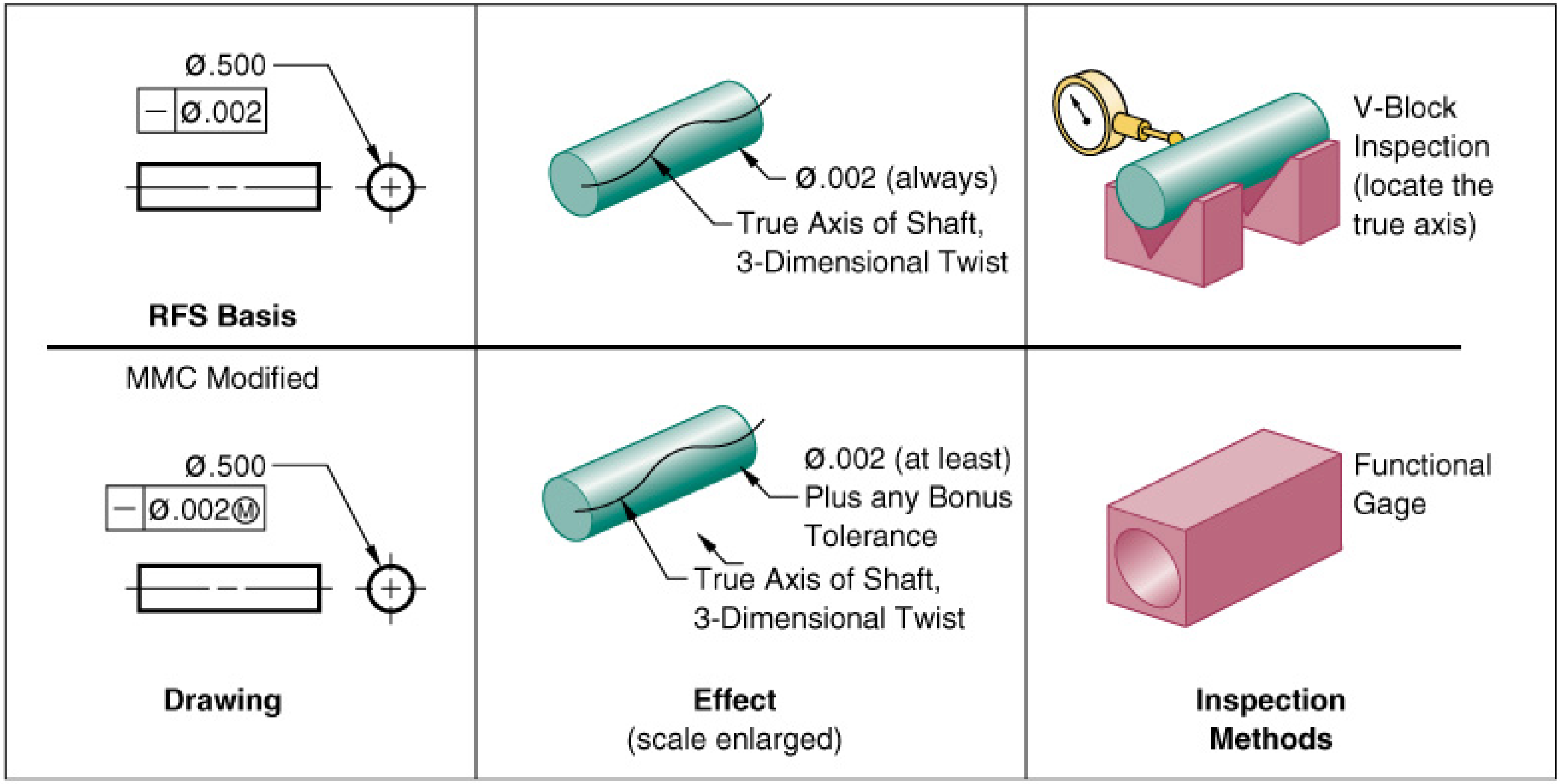


Straightness has two distinct variations:

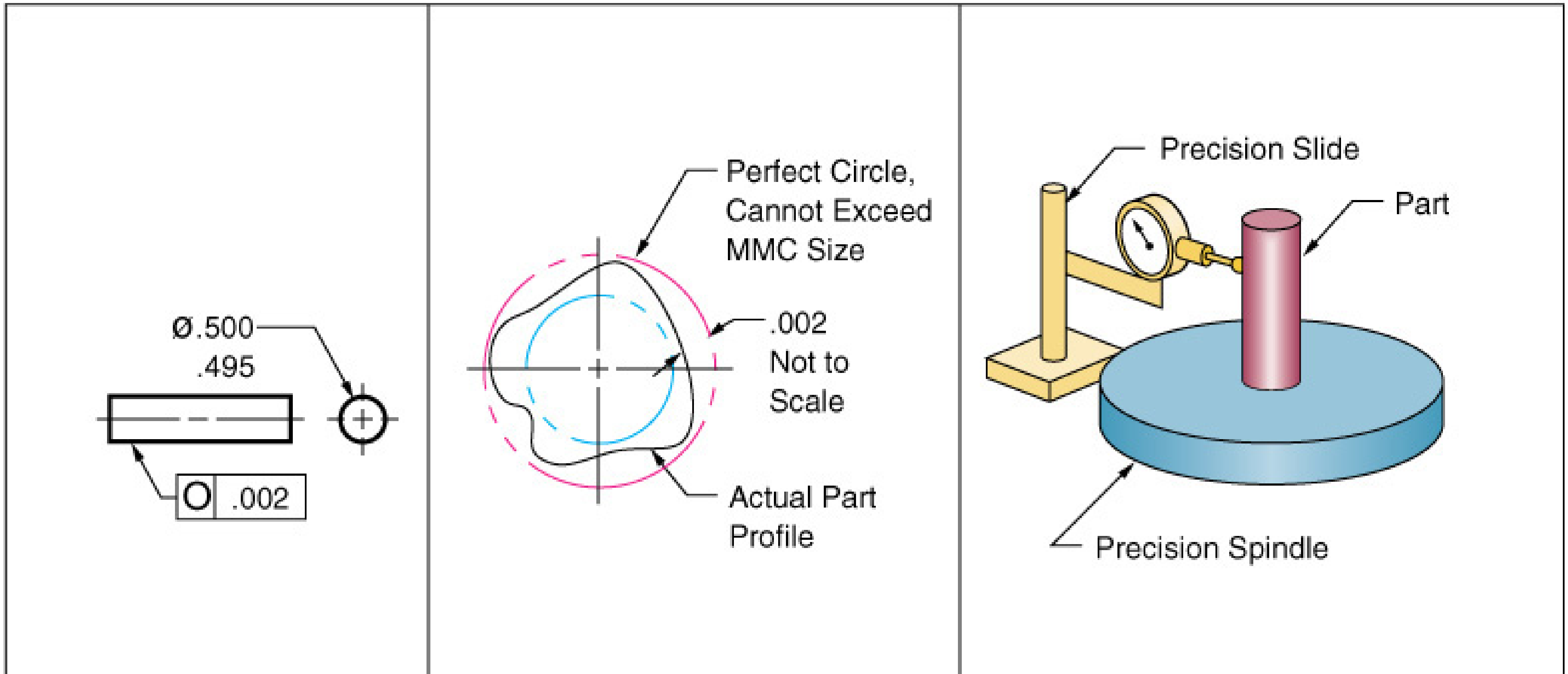
- **Line element straightness**. This compares a line on the part to a perfectly straight line. If the line is on a flat surface, the direction must be identified.



- **Axis straightness**. This compares the axis of a cylindrical feature to a perfectly straight line.

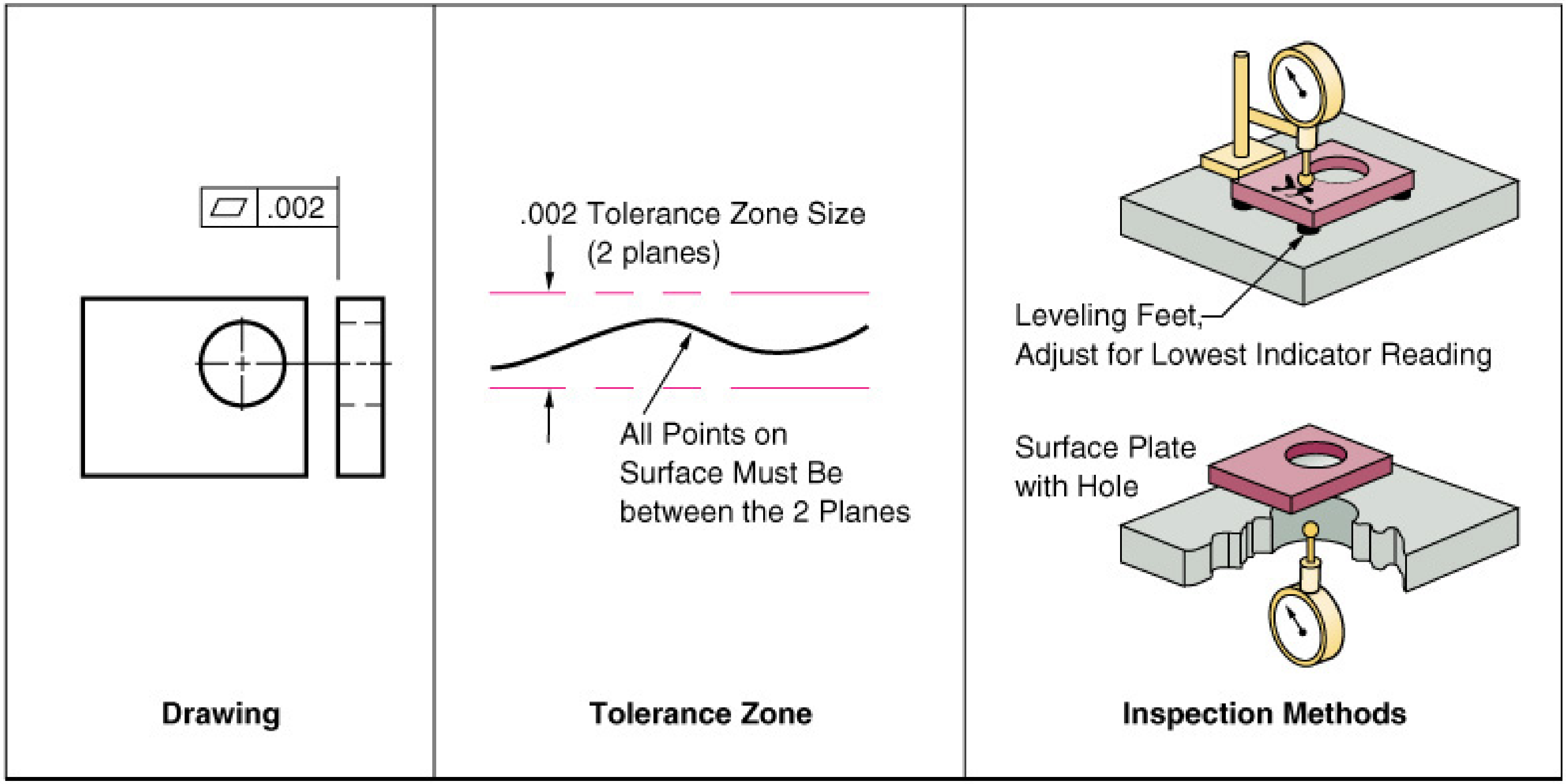


- **Roundness (or circularity)**. This compares a circular element on a feature to a perfect circle. Roundness could be considered straightness bent into a circle. Note that the circle is being measured for form only, not for size (i.e. no MMC is applied).



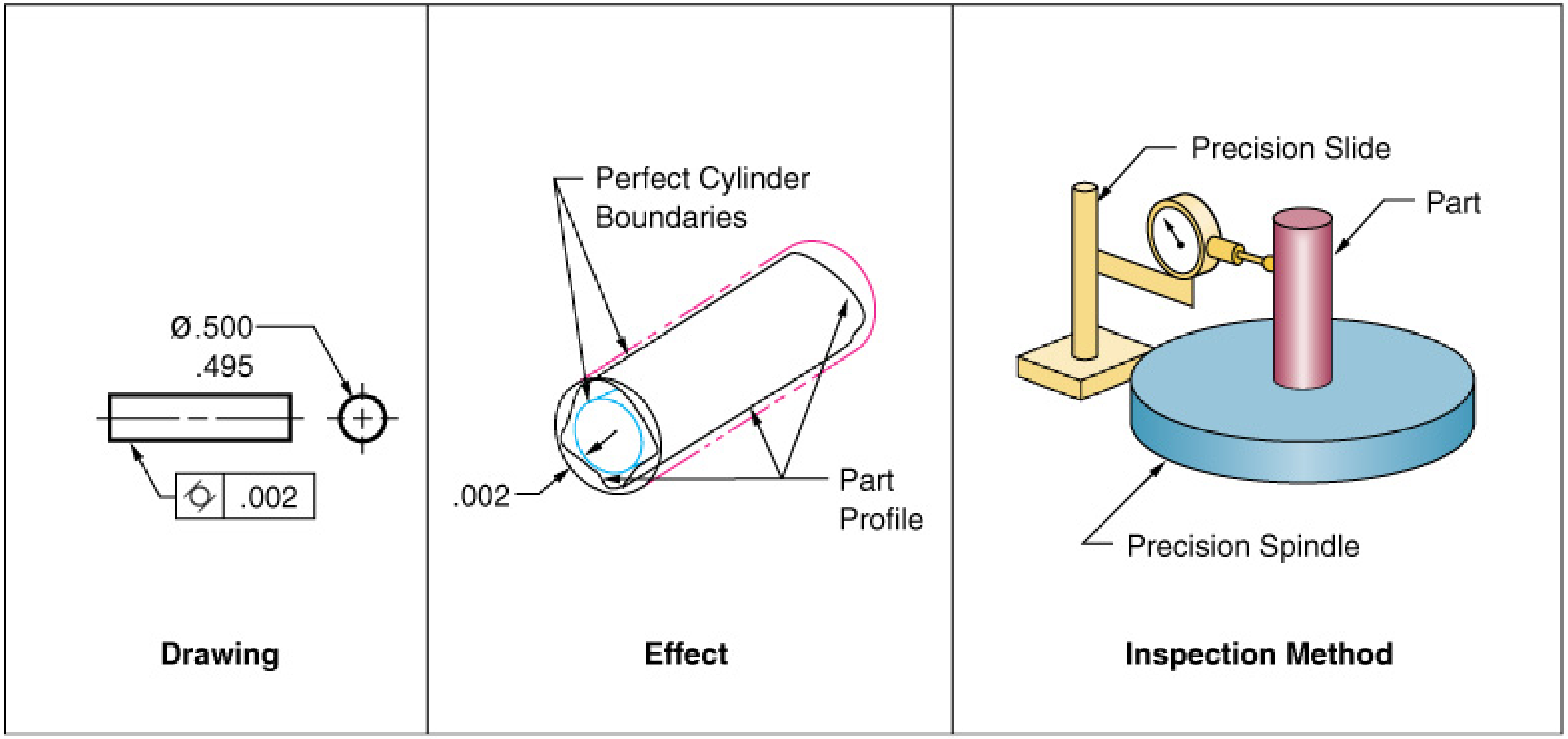
Drawing | **Tolerance Zone** | **Inspection Method** (check only one slice at a time)

- **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).

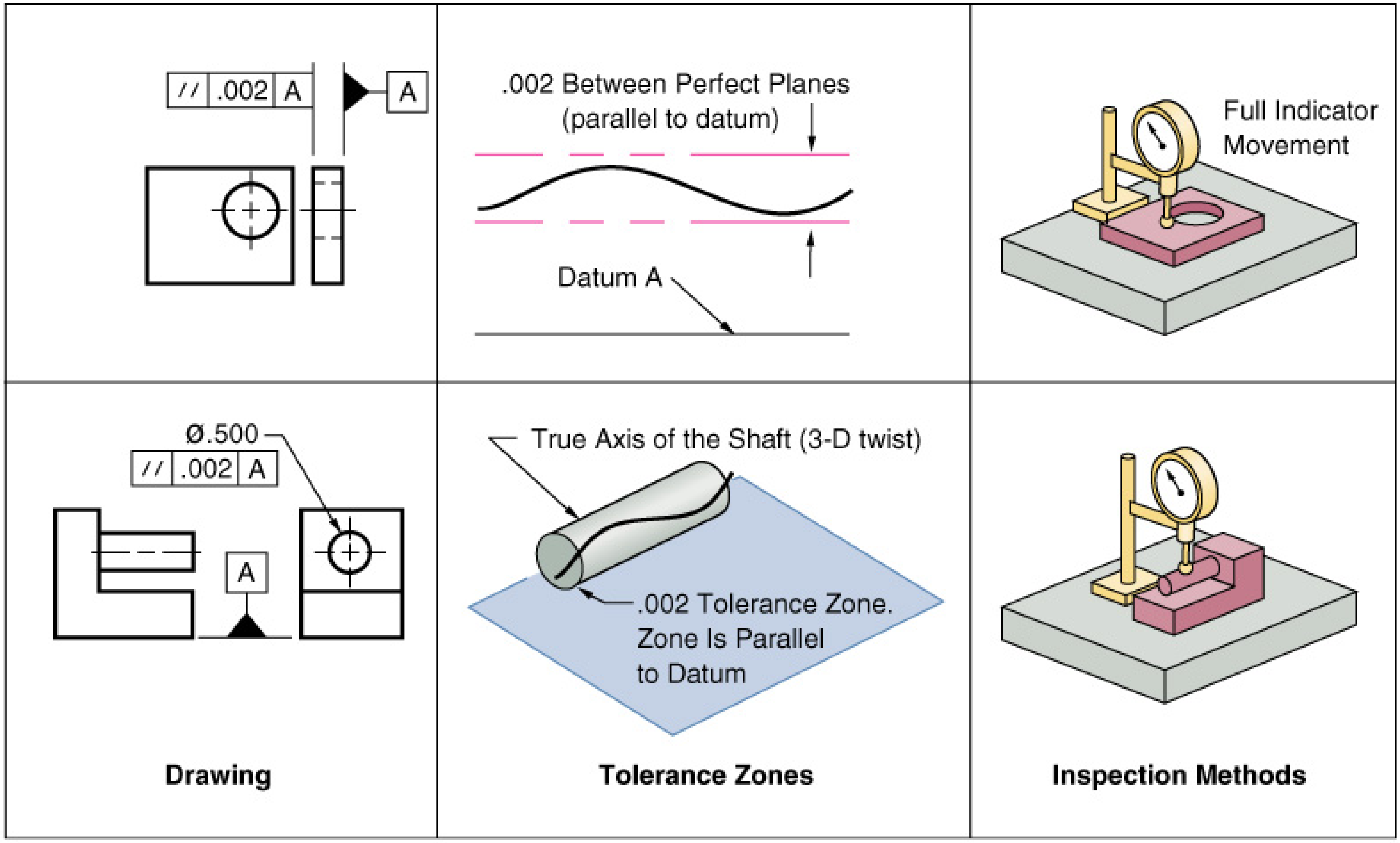
- **Cylindricity**. In comparing a feature to a perfect cylinder, three factors are being considered:
    - o straightness of all line elements,
    - o roundness of all circular elements, and
    - o taper (comparison of circular elements to each other).



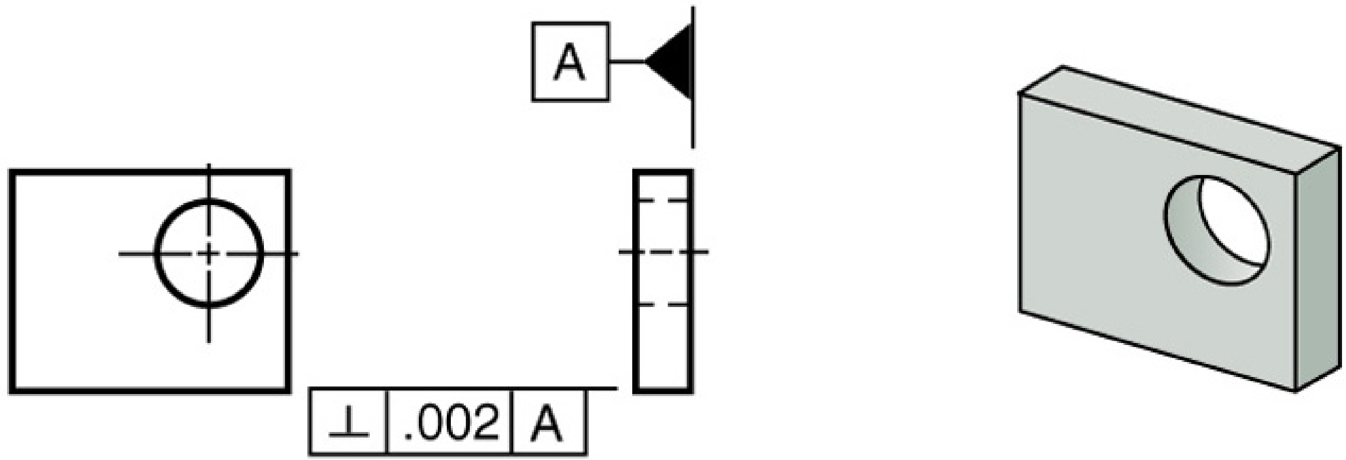
This is probably the most expensive control due to its difficulty in measuring.

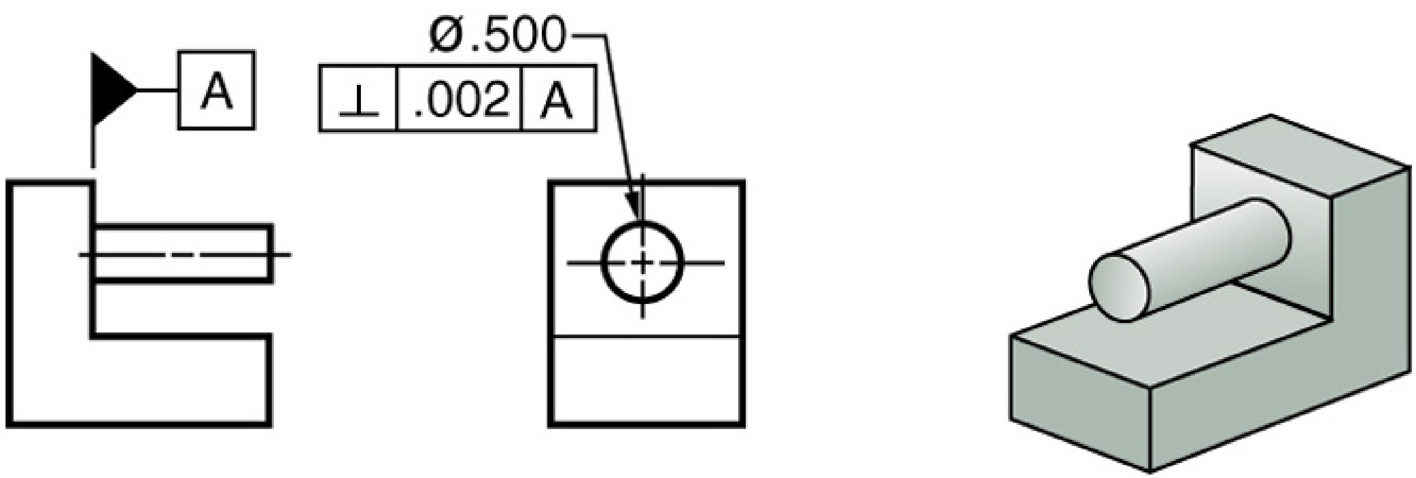
2. Orientation controls include:

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

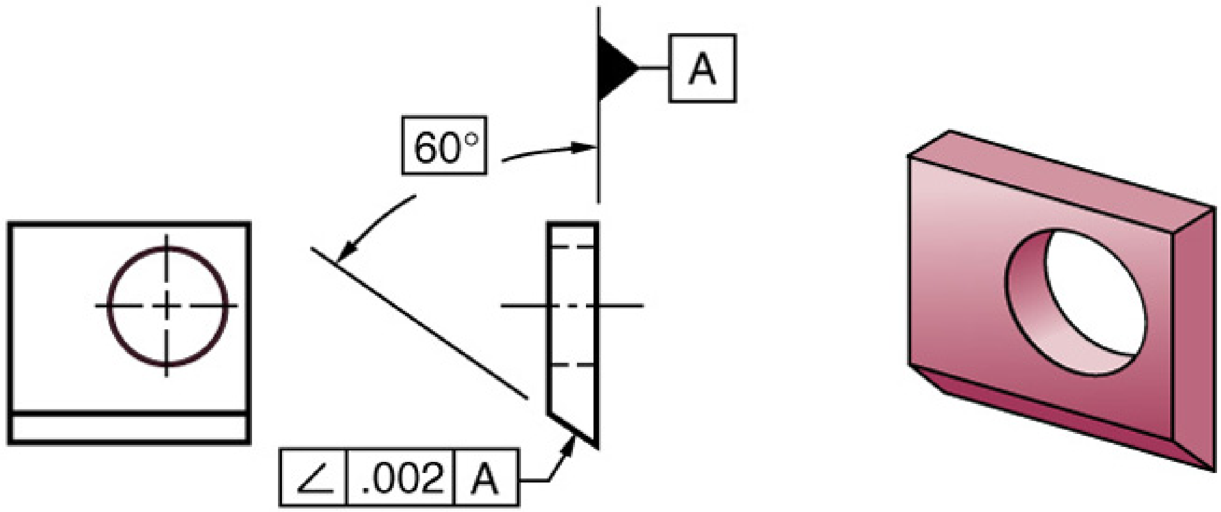


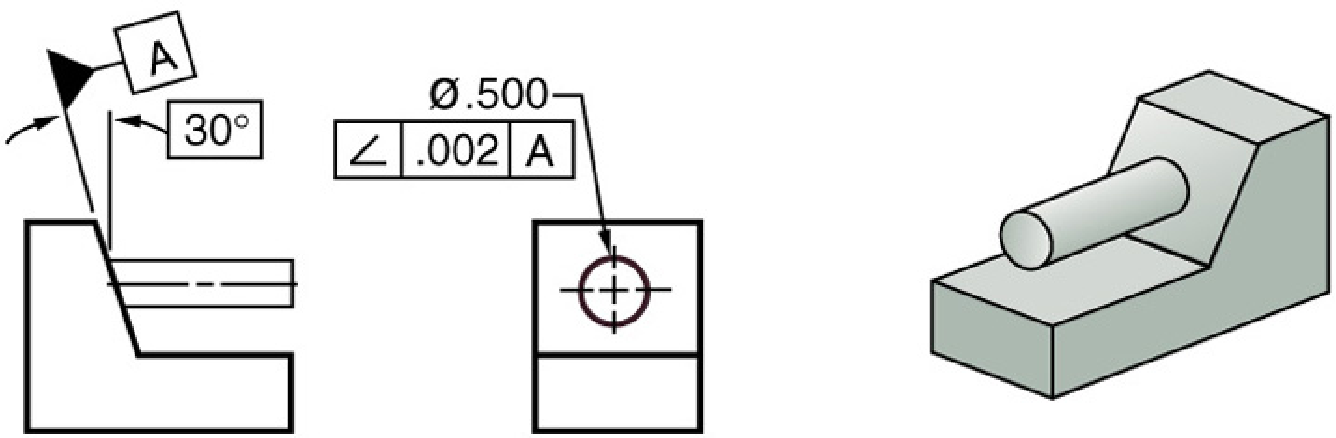
- **Perpendicularity**. This could be considered flatness or straightness of an axis 90 degrees to a datum.



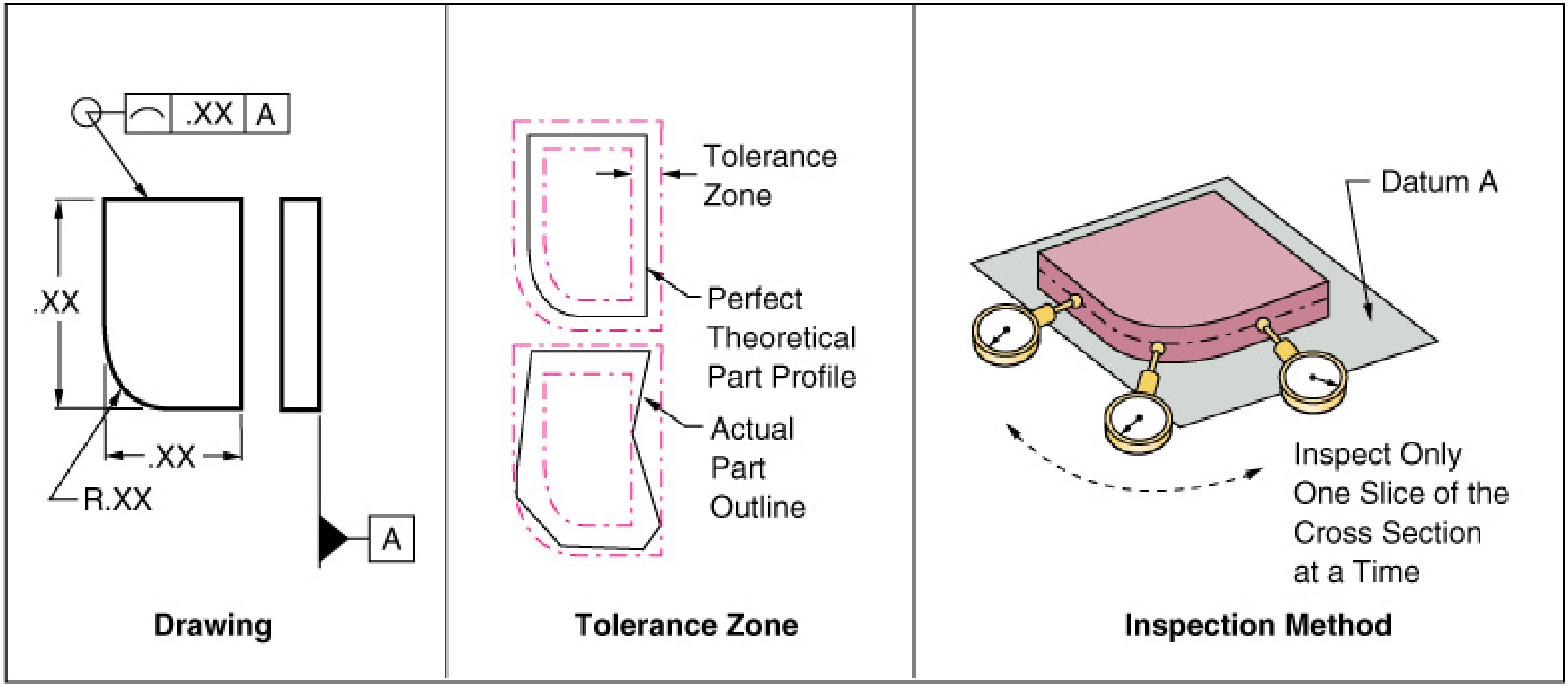


- **Angularity**. This could be considered flatness or straightness of an axis at some angle to a datum.



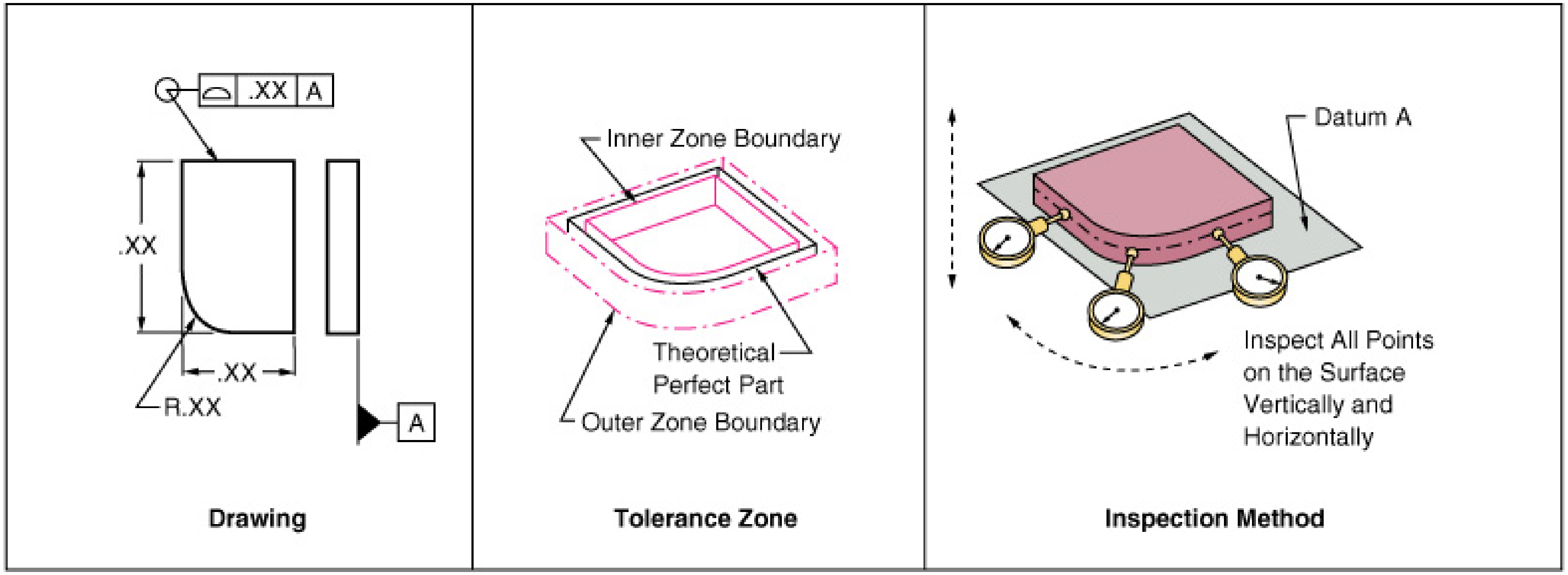


- **Line profile**. This takes a cross-sectional slice or slices of a feature and compares it to an ideal shape.



Profile is usually used to control shapes that are combinations of contiguous straight lines, circular arcs, and other curves.

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

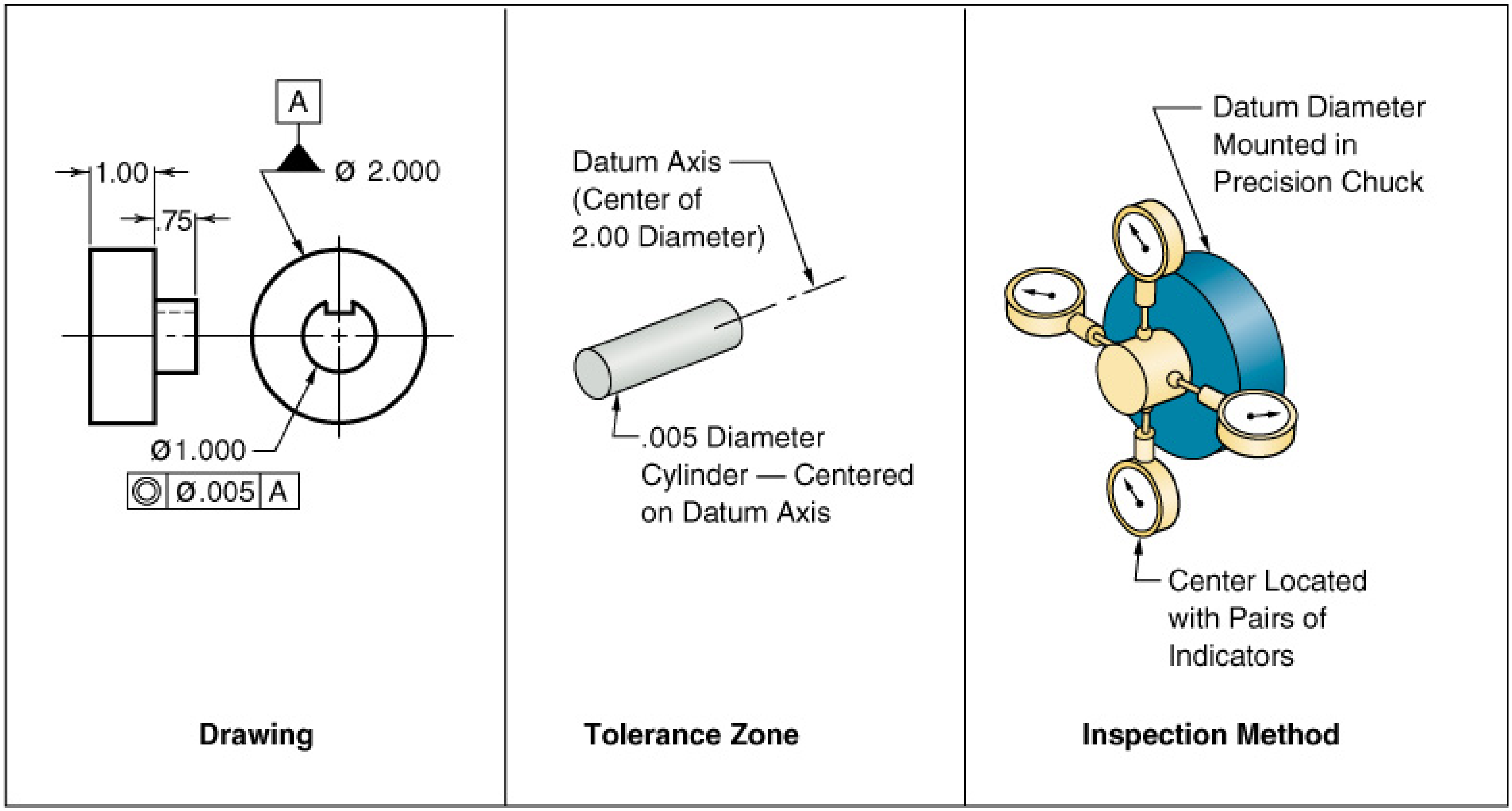


Drawing | Tolerance Zone | Inspection Method

In line profile control we have to inspect only one slice along the edge – horizontally, but in surface profile control we inspect different slices – all points both horizontally and vertically.

3. Position controls include:

- **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.

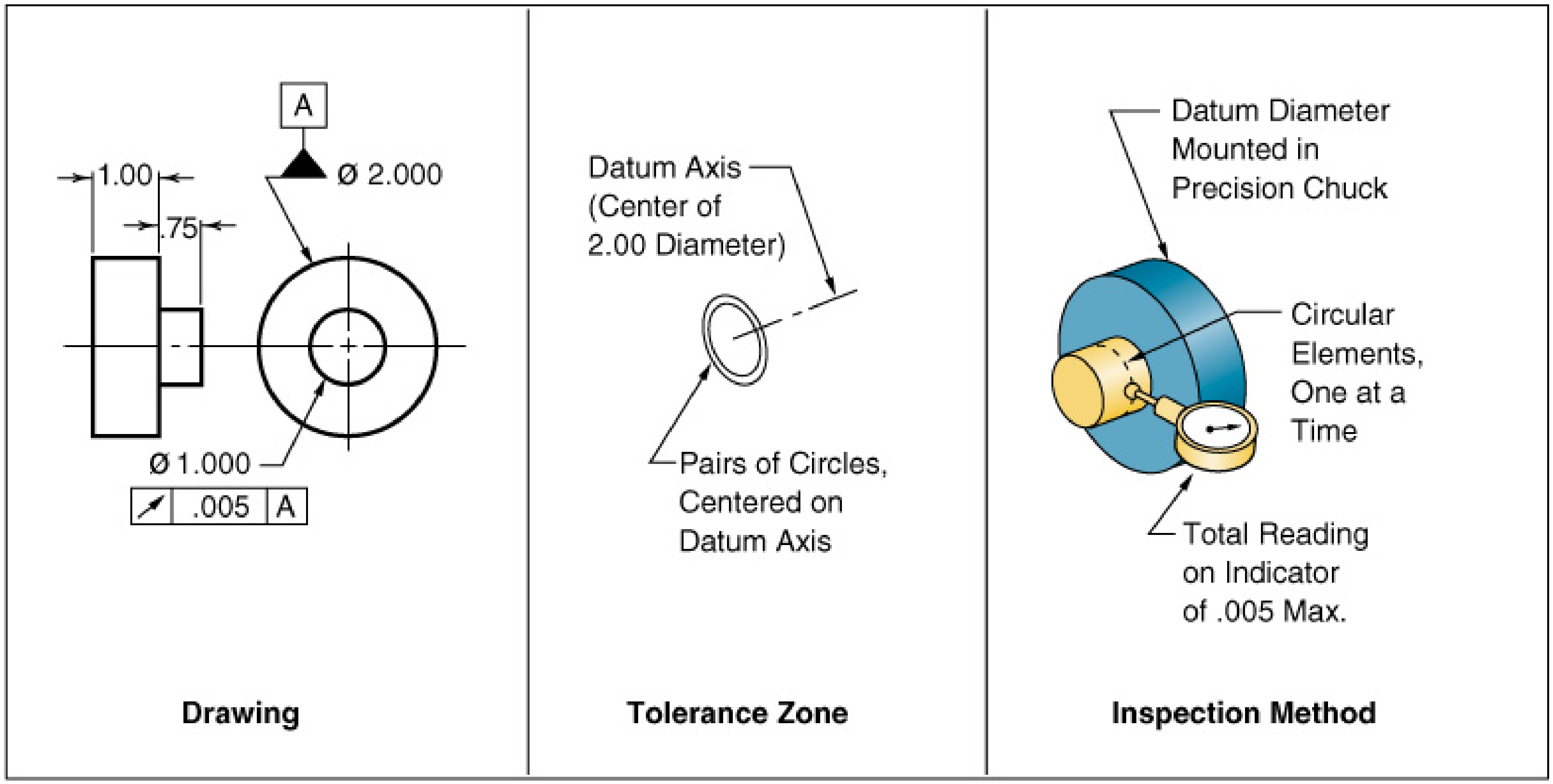


Drawing | Tolerance Zone | Inspection Method

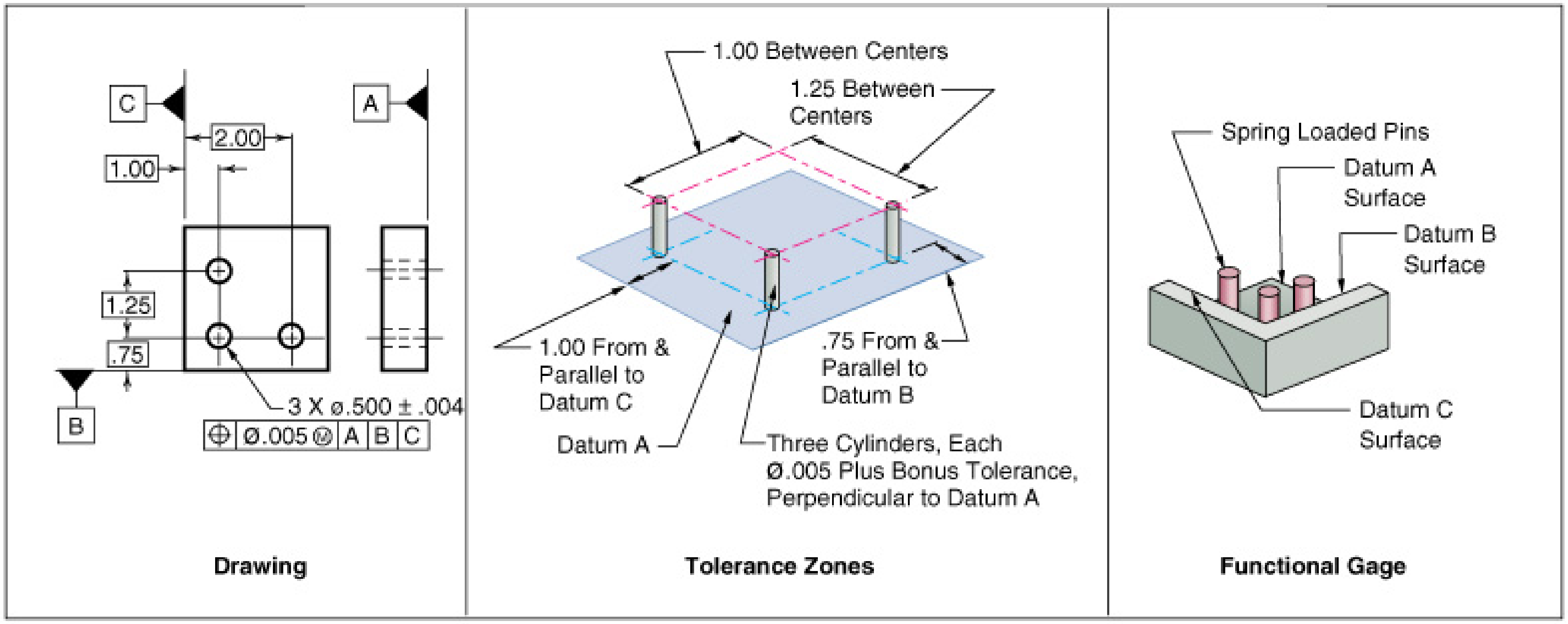
- **Runout**. There are two types of runout: single circular element (slice) and total (cylinder).

An excessive runout reading on a shaft could be caused by

- a bent shaft,
- an out-of-round surface (i.e. any imperfection, such as scratches, burrs, keysets), or
- an eccentrically placed feature.



- **Position.** This is the more flexible and versatile control. A few of the things this control can do is:
  - Locate holes or a pattern of holes.
  - Locate the center of a feature.
  - Keep holes or other features perpendicular or parallel to other features.
  - Keep features straight and round.



Other types of position controls you will find in machinery books.

Summary

- Geometric dimensioning and tolerancing (GDT) is essential to the modern manufacturing environment.
- The first rule of GDT is that size control of a feature (as is given in a dimension) inherently includes controls of form.
- Because of Rule 1 the need for GDT callouts on technical drawings is minimized.
- The goal of GDT is to carefully evaluate the functionality of a part and its features and only control the geometry of those features necessary for the proper functioning of the part.