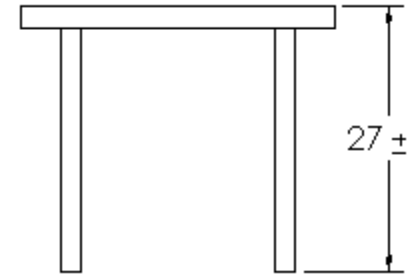# Geometric Dimensioning & Tolerancing (GD&T)

Geometric Dimensioning & Tolerancing (GD&T)

- GD&T is a system that uses standard symbols to indicate tolerances that are based on the feature's geometry.

- This allows a drawing to contain a more defined feature more accurately.

Inspection of Parts - For Example

Assume all 4 legs will be cut to length at the same time.

27 ±1

▶ Given Table Height

▶ However, all surfaces have a degree of waviness, or smoothness. For example, the surface of a 2 x 4 is much wavier (rough) than the surface of a piece of glass.

    ▶ As the table height is dimensioned, the following table would pass inspection.

27
26   28   or   27
                          26   28
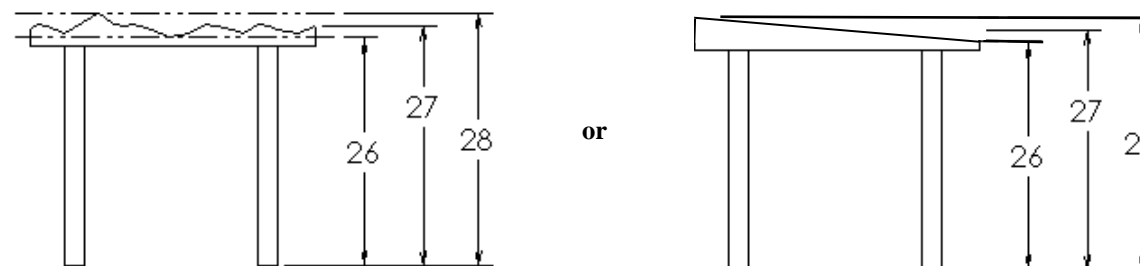
Example cont'd.

▶ The table height may pass inspection with any height between 26 and 28 inches.

▶ The table top must be flat within 1/16.
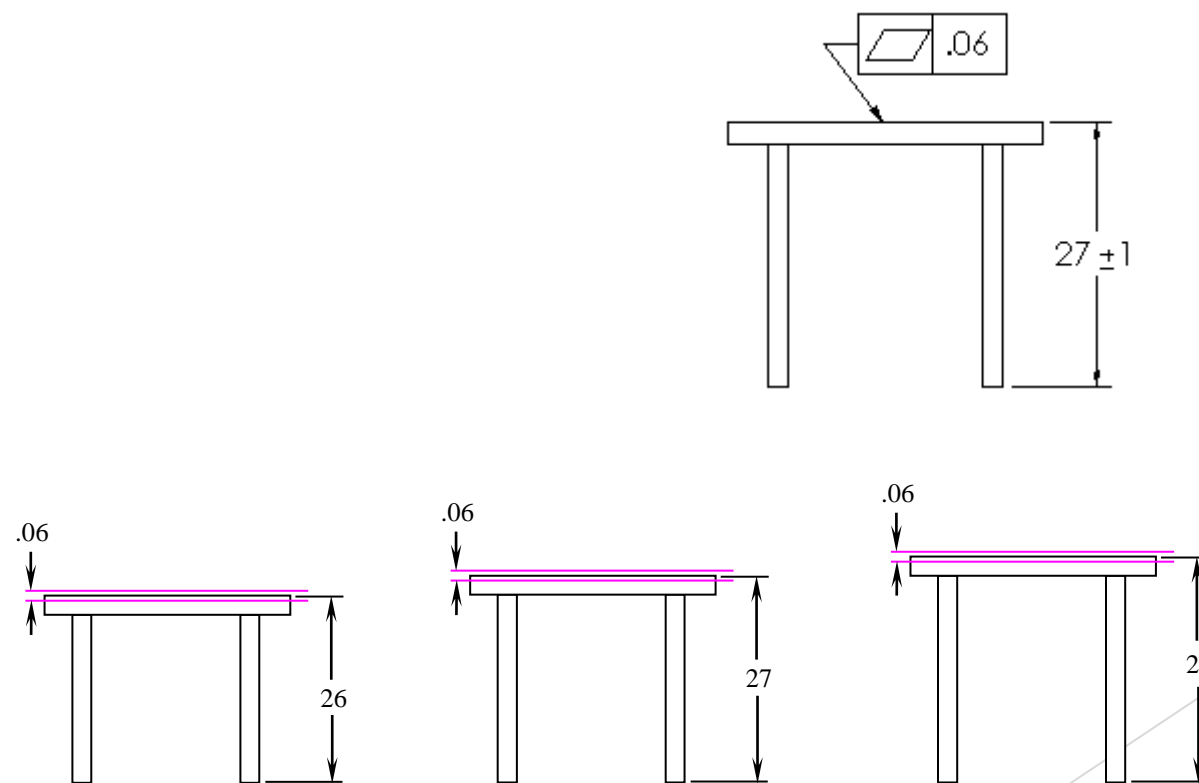
WHY IS GD&T IMPORTANT

- Saves money
  - For example, if large number of parts are being made – GD&T can reduce or eliminate inspection of some features.
- Ensures design, dimension, and tolerance requirements as they relate to the actual function
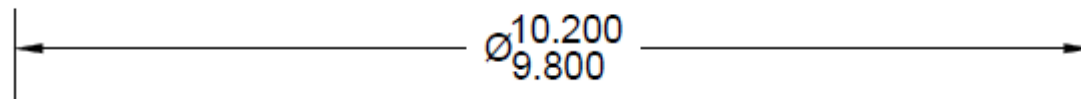- Provides uniformity

TERMINOLOGY REVIEW

▶ **Basic Dimension:** Nominal dimension from which tolerances are derived.

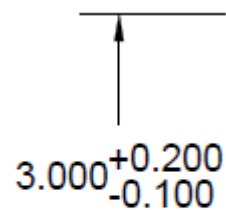▶ **Tolerance:** the permissible limit or limits of variation in a physical dimension.
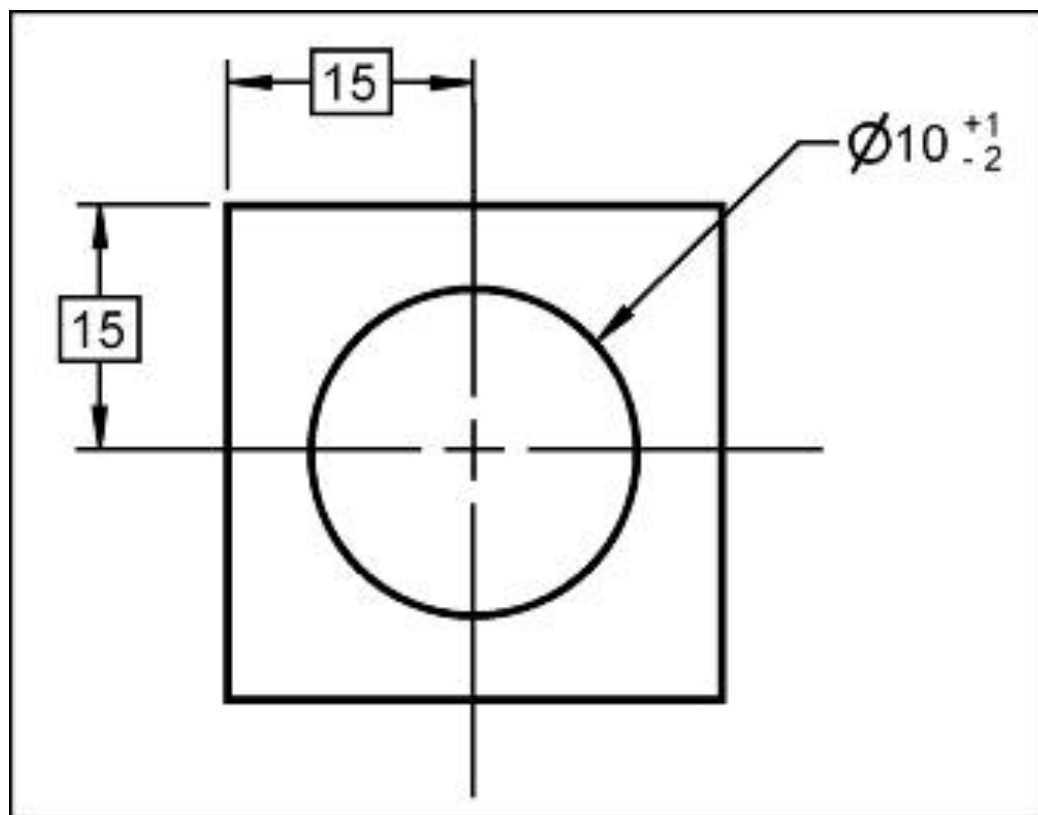
Limits, Deviations & Symmetrical Tolerances of Size
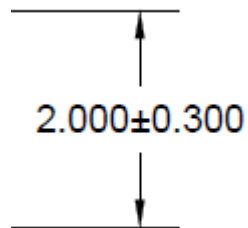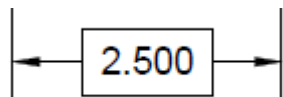
▶ Tolerances of size can be displayed as

    ▶ Limits

$$\varnothing {}^{10.200}_{9.800}$$

    ▶ Deviations

$$3.000{}^{+0.200}_{-0.100}$$
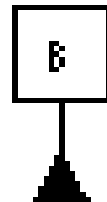
    ▶ Symmetrical

$$2.000\pm0.300$$

▶ If not toleranced, set to a Basic Dimension, used to describe theoretical exact size

2.500

Datums and Geometric Controls

- Datum References
    - Or just Datum, is some important part of an object - such as a point, line, plane, hole, or surface – that serves as a reference in defining the geometry of the object.
    - Define Datums using Datum Feature Symbols

- Geometric Controls
    - Describes the conditions and tolerances of a geometric control on a part's feature.
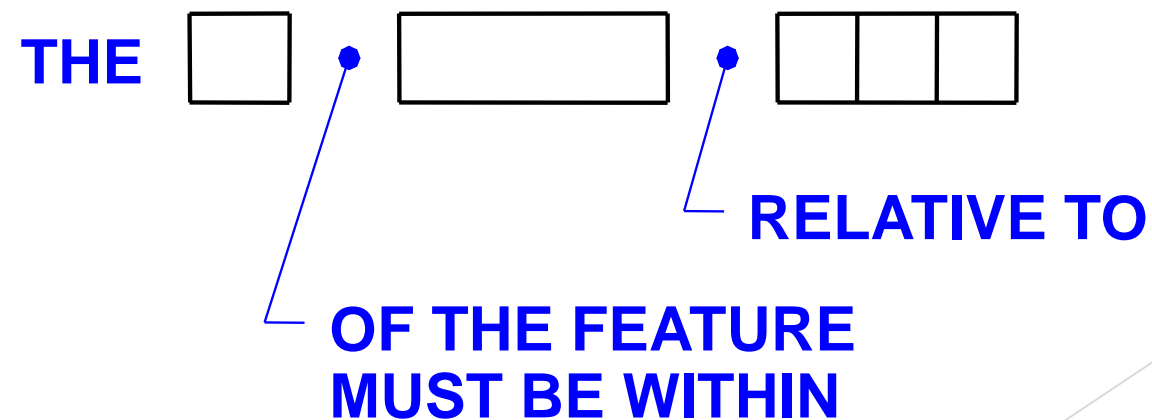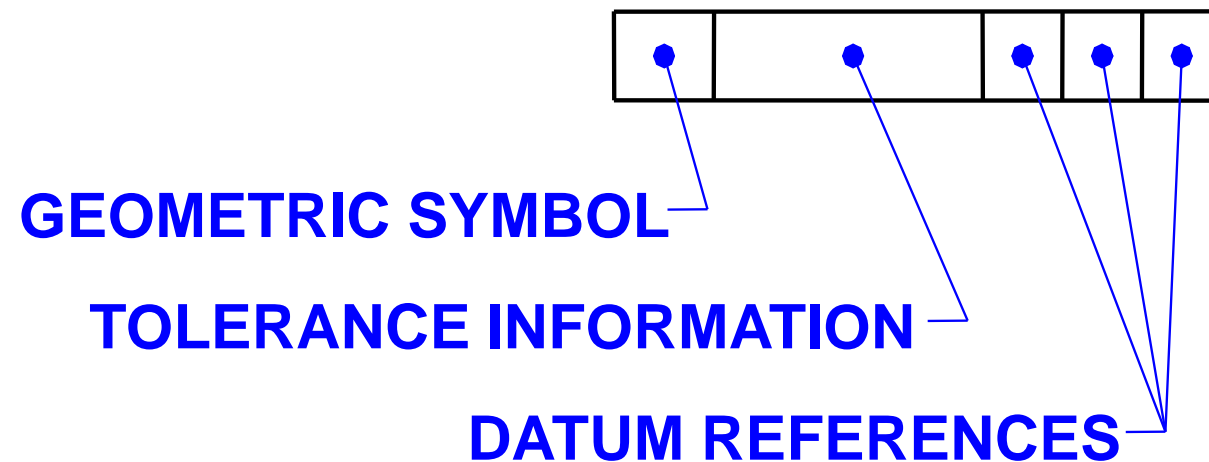   - Geometric Control Frame is used to describe these features

GEOMETRIC CHARACTERISTIC CONTROLS
14 characteristics that may be controlled

TYPE OF FEATURE	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL
INDIVIDUAL (No Datum Reference)	FORM	FLATNESS	▱
		STRAIGHTNESS	—
		CIRCULARITY	◯
		CYLINDRICITY	⌀
INDIVIDUAL or RELATED FEATURES	PROFILE	LINE PROFILE	⌒
		SURFACE PROFILE	⌓
RELATED FEATURES (Datum Reference Required)	ORIENTATION	PERPENDICULARITY	⊥
		ANGULARITY	∠
		PARALLELISM	//
	RUNOUT	CIRCULAR RUNOUT	↗
		TOTAL RUNOUT	↗↗
	LOCATION	CONCENTRICITY	◎
		POSITION	⊕
		SYMMETRY	≡

Feature Control Frame

FEATURE CONTROL FRAME

GEOMETRIC SYMBOL

TOLERANCE INFORMATION

DATUM REFERENCES

THE

RELATIVE TO

**OF THE FEATURE
MUST BE WITHIN**

Feature Control Frame

▶ Uses feature control frames to indicate tolerance

Geometric Characteristic Symbol

Geometric Tolerance

Primary Datum

Secondary Datum

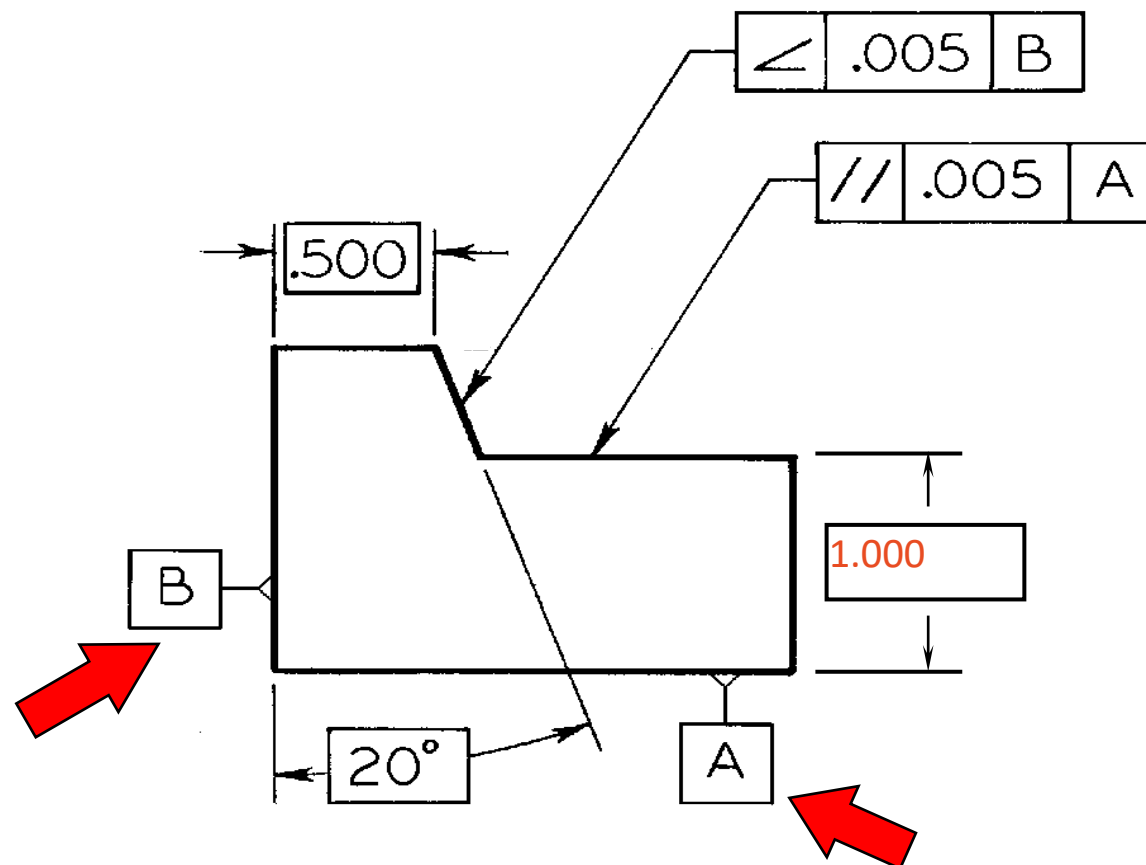$$\boxed{\phi} \quad \boxed{\varnothing\,0.5} \quad \boxed{A} \quad \boxed{B}$$
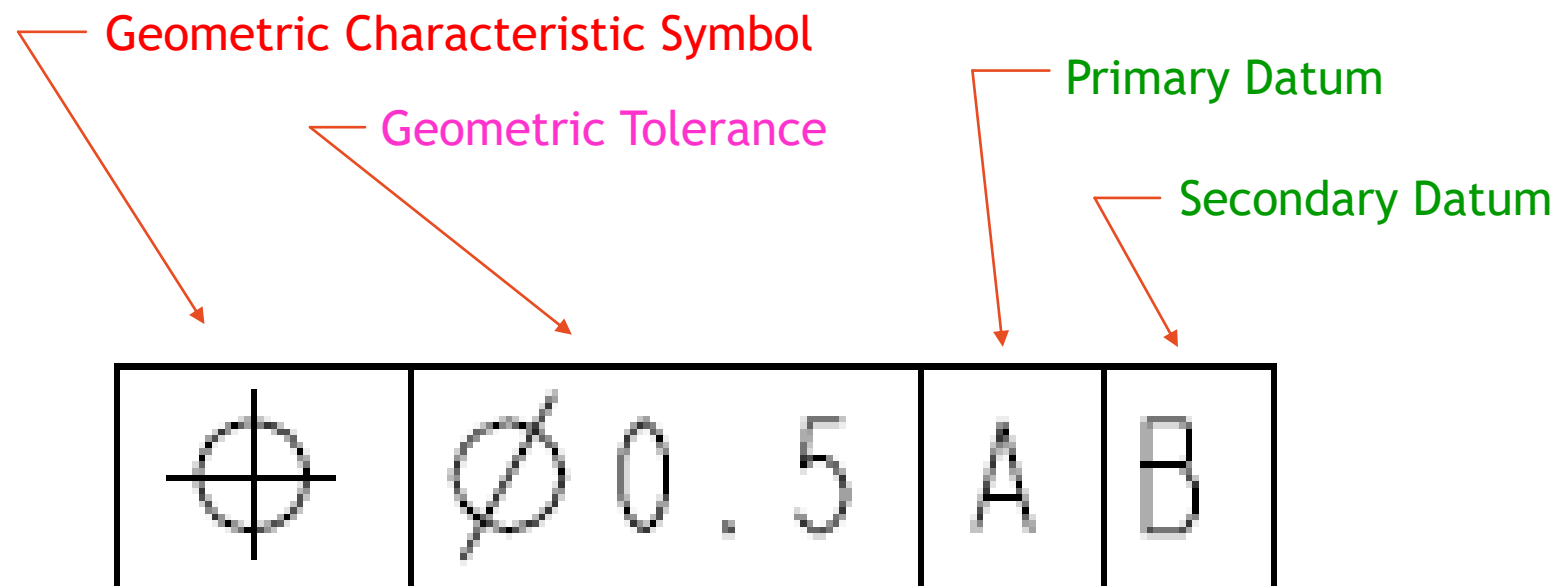
▶ Reads as: The position of the feature must be within a .5 diametrical tolerance zone relative to datums A, and B.
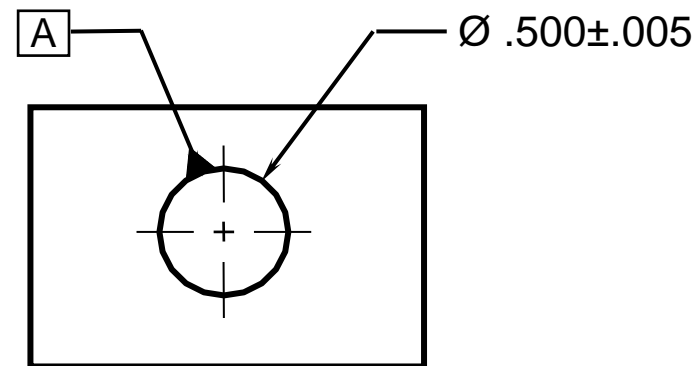
Datum

- Datum Placements
  - Placements of datums are used with datum triangles that touch the feature, with a straight leader line to the datum identifier box
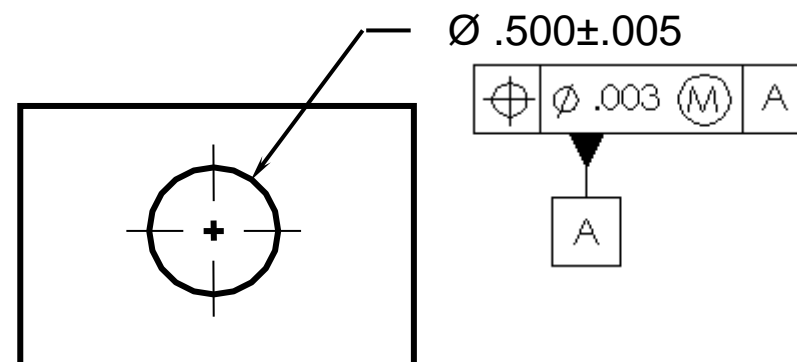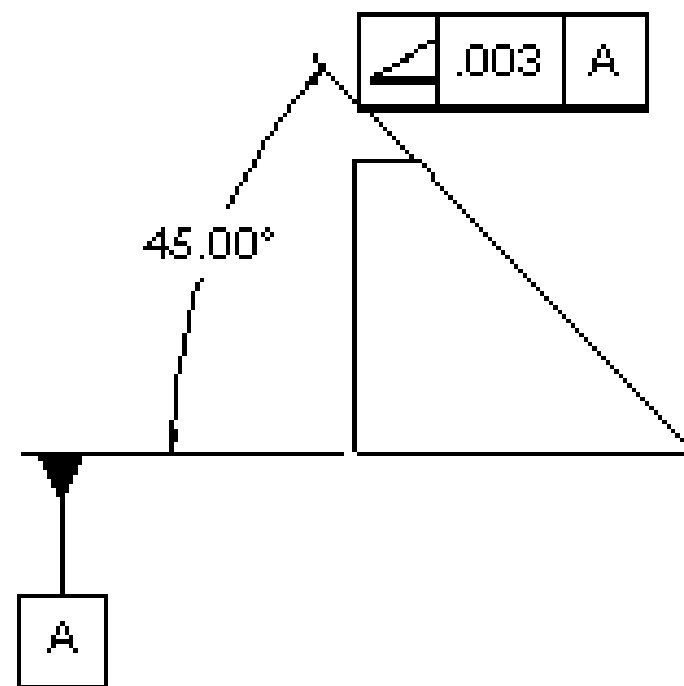
Placement of Datums

▶ Feature sizes, such as holes
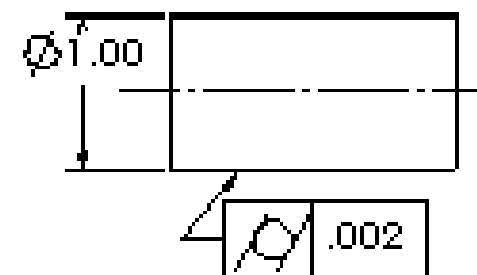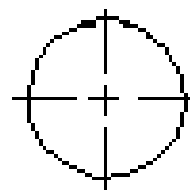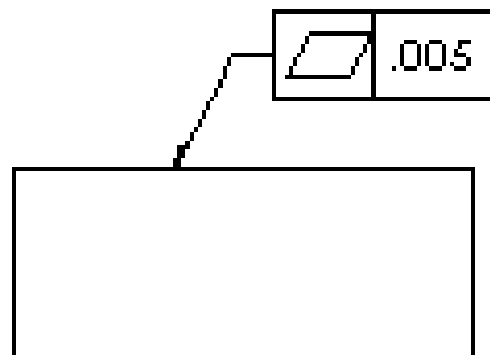

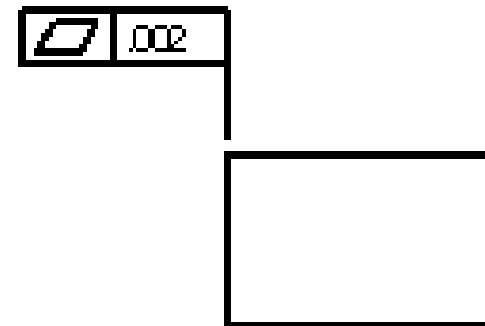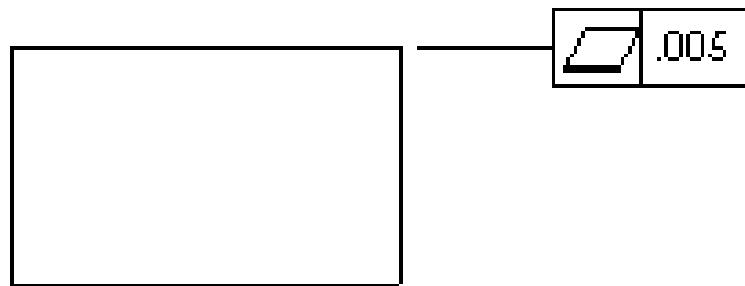▶ Sometimes a feature has a GD&T and is also a datum

Placement of Feature Control Frames

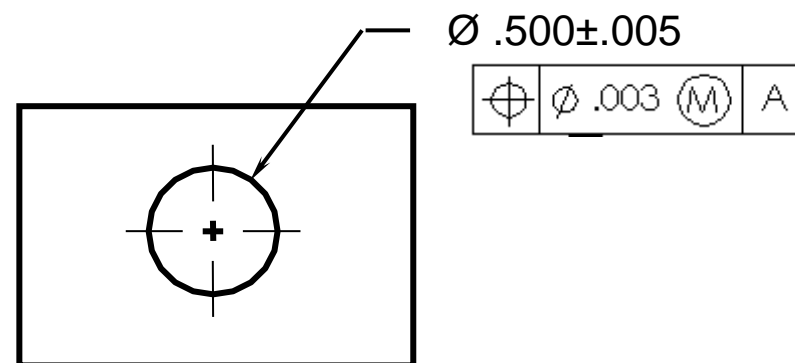- May be attached to a side, end or corner of the symbol box to an extension line.

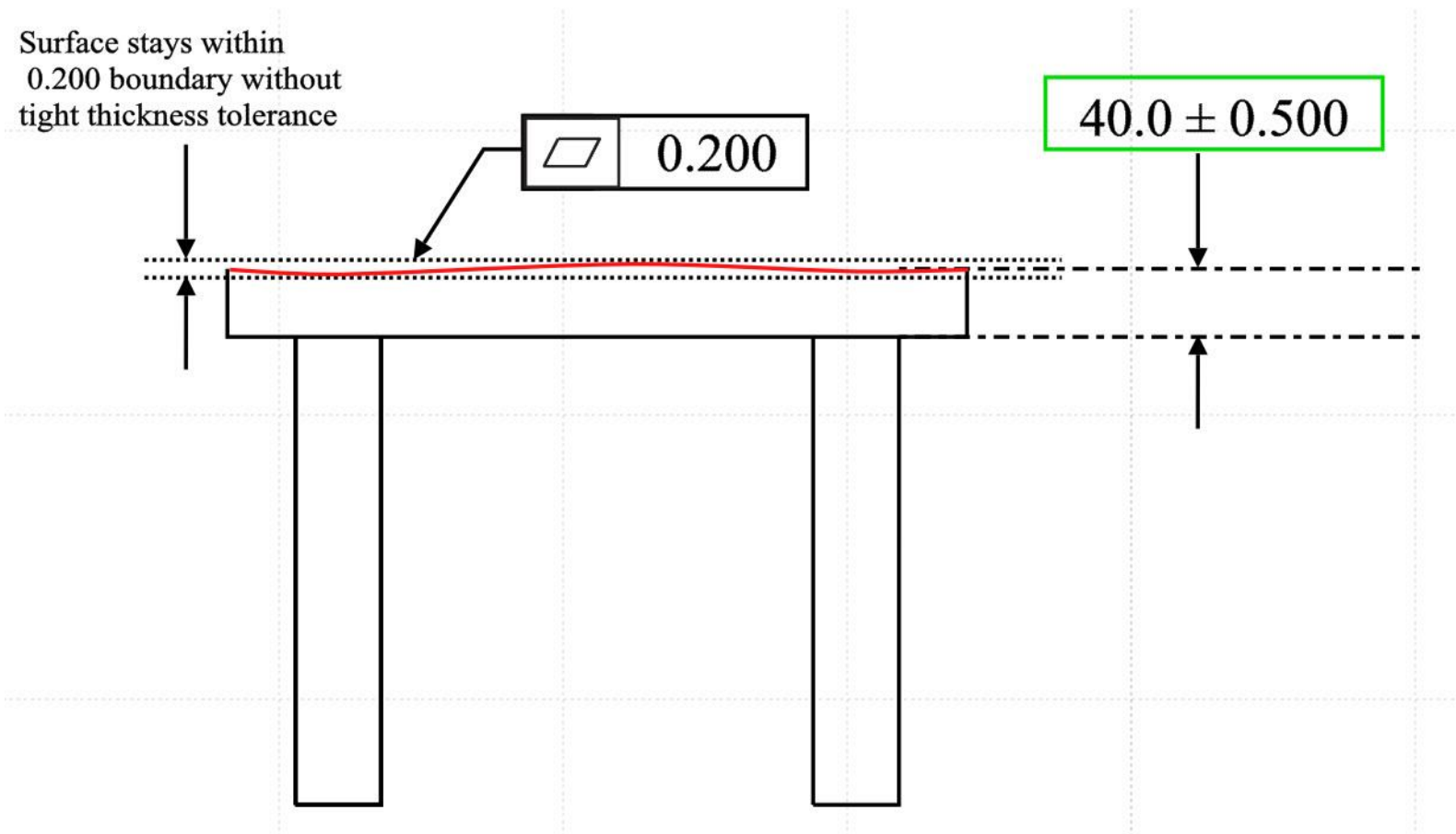# Placement of Feature Control Frames

▶ Applied to a surface

Placement of Feature Control Frames

▶ May be below or closely adjacent to the dimension or note pertaining to that feature.

Ø .500±.005

⊕ | Ø .003 Ⓜ | A

Geometric Control Example: Flatness

▶ Straight forward. References how flat a surface is.

  ▶ Flatness as stated on drawing: The flatness of the feature must be within .200 tolerance zone.

Surface stays within
0.200 boundary without
tight thickness tolerance

⬭ | 0.200

40.0 ± 0.500

- Would Pass Inspection