DIMENSIONING & TOLERANCING

Deborah Munro, Ph.D.

Overview: Why do we dimension? Why do we tolerance? Why GD&T?

- Most machining, assembly, and construction is still done while not in front of a CAD model

- Provides information to the machinist as to our design intent

- Provides information as to how critical a dimension is to the design intent

- Provides information that cannot be conveyed by a CAD model

- Provides geometric intent not conveyed elsewhere
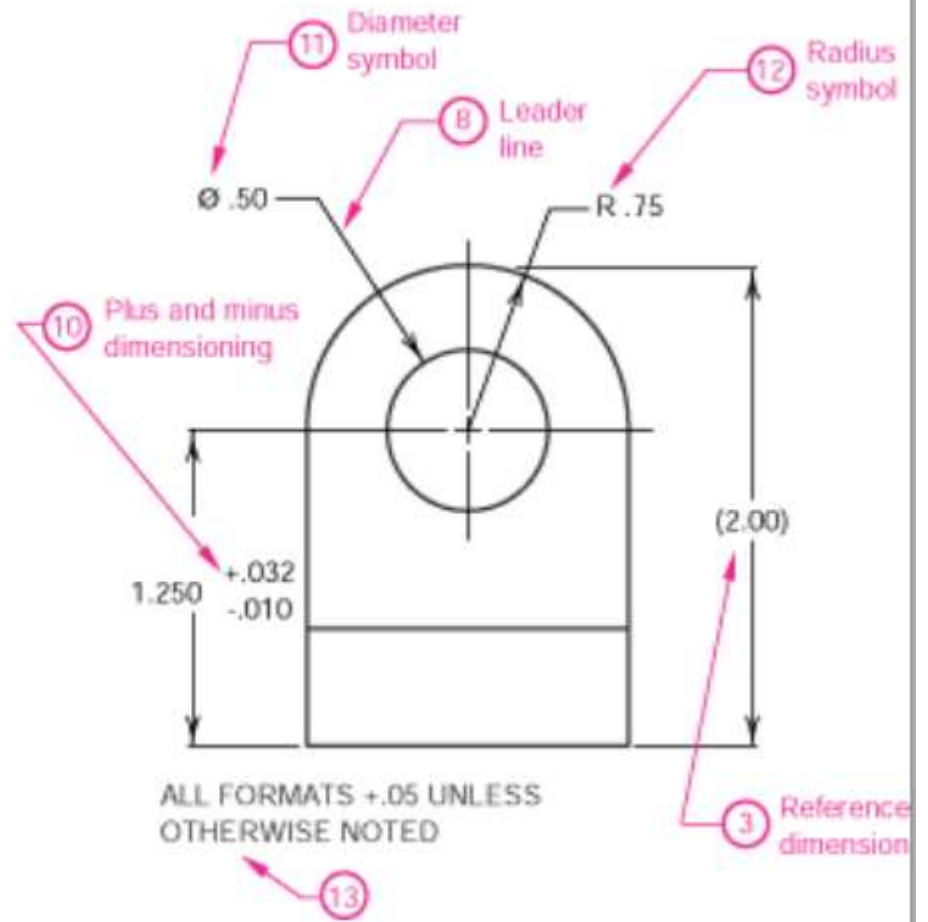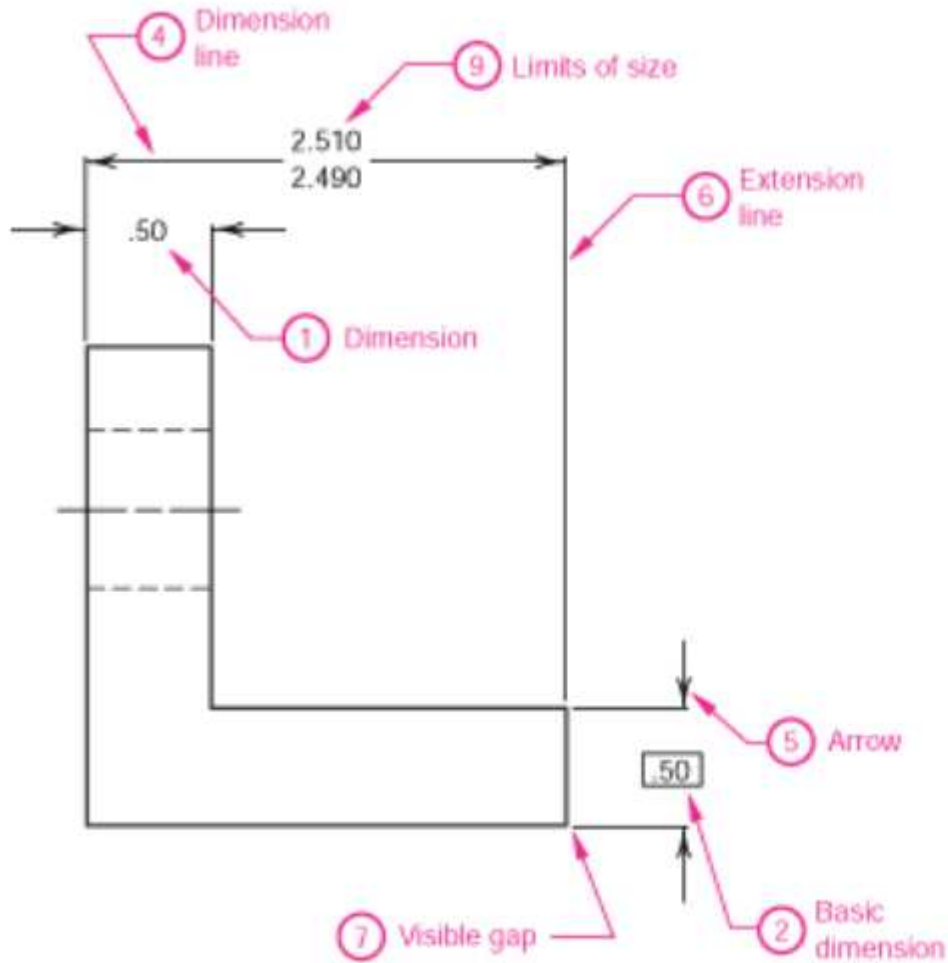
Basic Dimensioning Rules

- A dimension can only be called out <u>once</u>.

- Smaller lengths should be closer to the part, larger ones farther out.

- Dimension from one side or a key location (hole center) to avoid error stack up.

- Do not dimension to hidden lines!
  - Use section views as needed to avoid this

- Dimensions should be logically laid out and grouped by features

- Dimension only on part drawings, not assemblies

Basic Tolerancing Rules

- Use the default tolerance blocks as much as possible, with decimal places for tolerancing

- Be as generous as possible with your tolerances

- Use non-symmetric tolerancing to show design intent

- When in doubt, ask a machinist what tolerances they can hold!

Types of Dimensions

Dimension line (4)

Limits of size (9)

2.510
2.490

.50

Extension line (6)

Dimension (1)

Arrow (5)

.50

Visible gap (7)

Basic dimension (2)

Diameter symbol (11)

Leader line (8)

Radius symbol (12)

Ø .50

R .75

Plus and minus dimensioning (10)

1.250 +.032 -.010

(2.00)

ALL FORMATS +.05 UNLESS OTHERWISE NOTED (13)

Reference dimension (3)
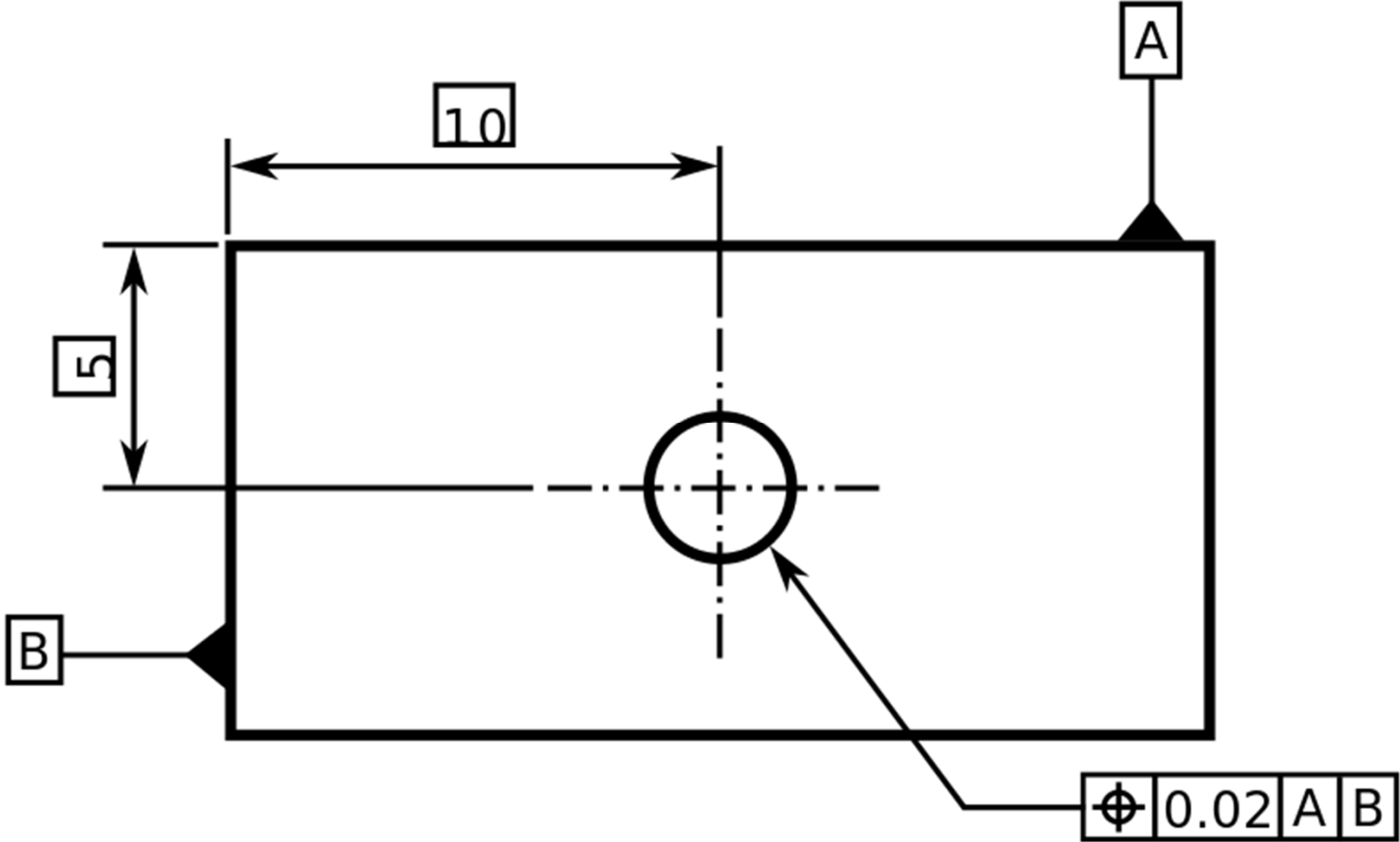
Geometric Dimensioning & Tolerancing

- Used when it's "not the tolerance" that you want to control

- Parallel, Perpendicular, Concentric, Flat, Position

- Requires a datum reference

- Requires a tolerance on GD&T

- Modifiers: MMC, LMC

- http://en.wikipedia.org/wiki/Geometric_dimensioning_and_tolerancing

```
⫽ | .005 | A
```
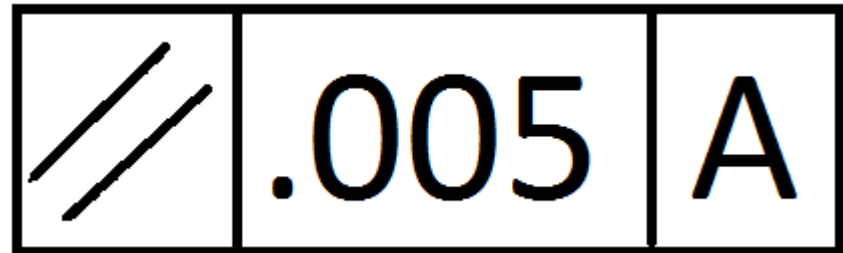
Example: Pistons
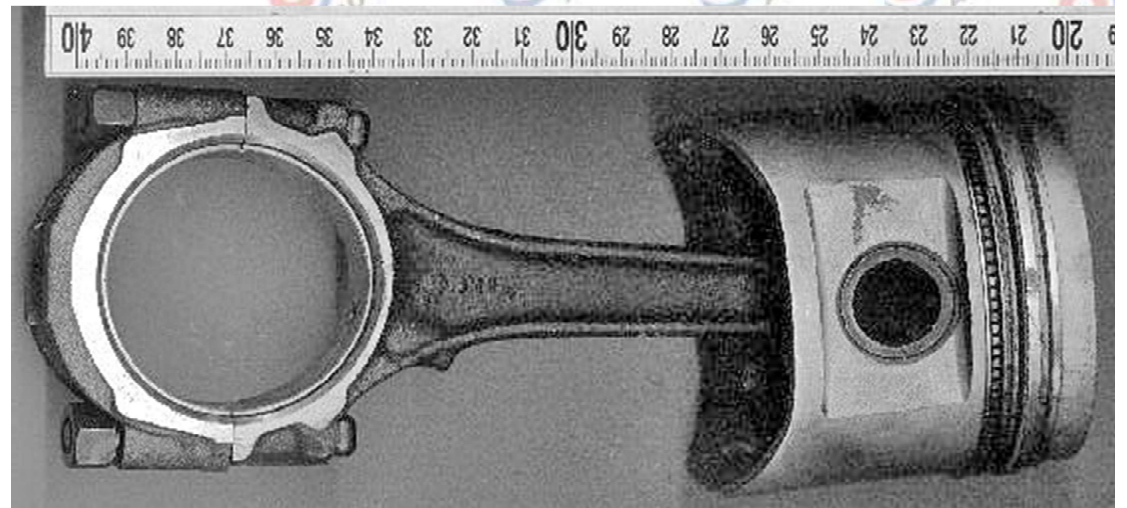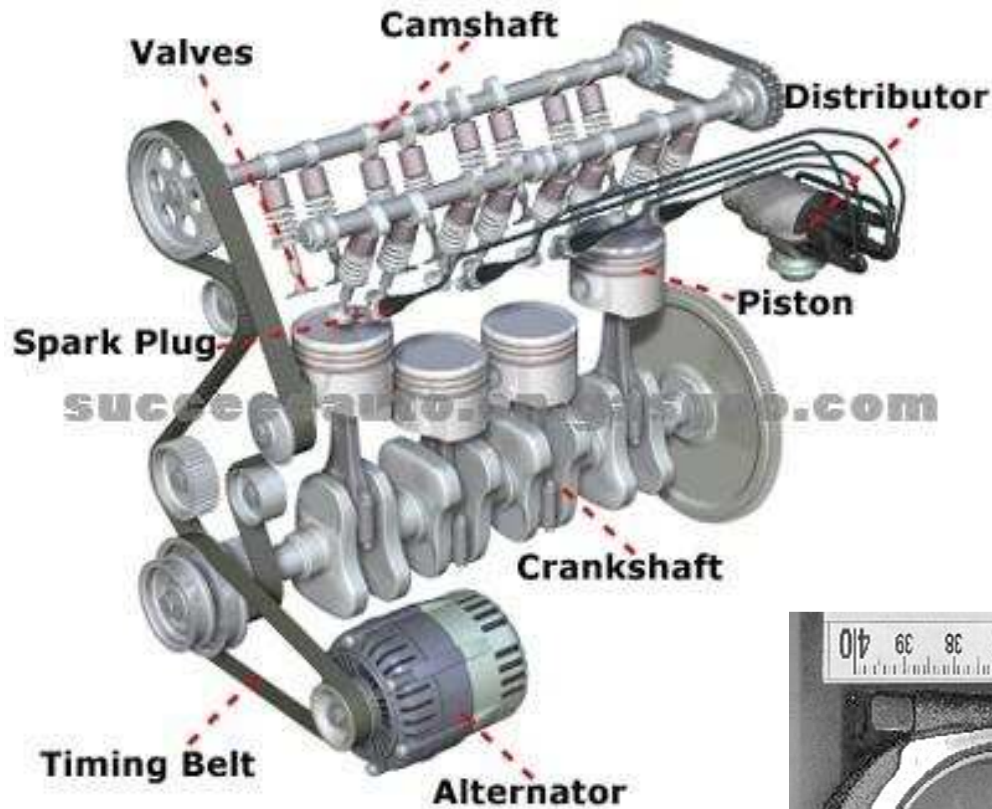
(True) Position

Definition of Positional Tolerance

Positional tolerance controls the location of an axis, median plane or surface of a feature. This is a term used for the positioning of cylindrical features (holes, shafts etc.), Line features, Point features and Slot features etc. These features are always related to basic dimensions or datum within the stated tolerance and generally at maximum material condition (MMC).

Positional tolerance is normally used for the interchangeability characteristic of non-spinning mating parts for maximum productivity. It defines a zone within which the axis, median plane or surface of a feature is permitted to vary from a true position.

True position is established with basic dimensions from a datum reference frame or from other features that are related to the datum reference frame. All dimensions must be basic and are considered theoretically exact. The tolerance for the deviation of the feature from basic is found in the position feature control frame.

Summary

- Only dimension once

- Dimension everything

- Tolerance beyond decimal places only when needed

- Use GD&T to control things tolerances won't

- Use notes and callouts when appropriate

- Don't dimension to hidden lines

- Provide overall part size reference dimensions