TOLERANCES AND ALLOWNANCES

Accuracy Versus Precision in Processes

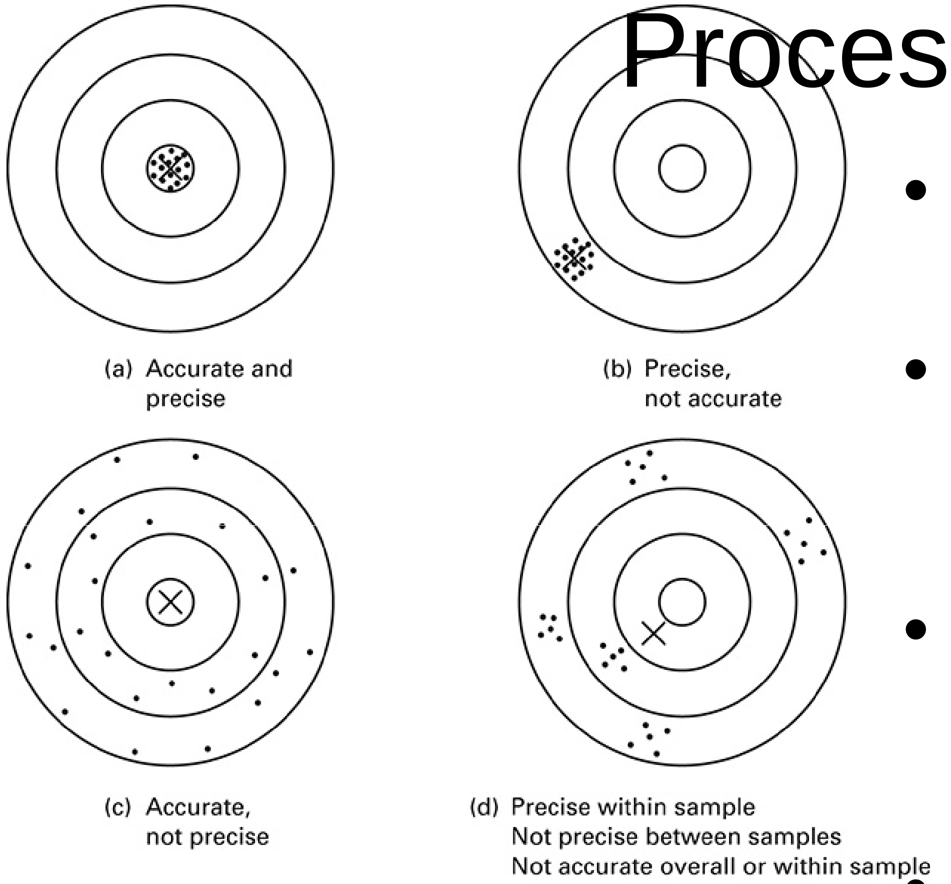


Figure 10-6 Accuracy versus precision. Dots in targets represent location of shots. Cross (X) represents the location of the average positions of all shots.

- Accuracy- ability to hit what is aimed at
- Precision- repeatability of the process
- Measuring devices must be both precise and accurate
- Skill of the operator may also have to be taken into account for measurements

MEAN VALUE

$$\mu = \frac{\sum_{i=1}^{n} X_i}{n}$$

- X_i-measurement I
- n-total number of measurements
- μ-mean value

STANDARD DEVIATION

$$\sigma = \sqrt{\frac{\sum_{i=1}^{n}(X_i - \mu)^2}{n}}$$

Tolerances

- A ***tolerance*** is an acceptable amount of dimensional variation that will still allow an object to function correctly.
- Tolerance- undesirable but permissible deviation from a desired dimensions
  - No part can be made exactly to a specified dimension
  - Necessary to permit the actual dimension to deviate from the theoretical (nominal) dimension

TOTAL TOLERANCE

- The total tolerance is a value that describes the maximum amount of variation.
- Tolerance = □.010
- **.500**
- **.020**
- **.490**
- **.510**

NATURAL TOLERANCE LIMITS

- Upper (UNTL) and Lower (LNTL) natural tolerance limits

$$\mu + 3\sigma = UNTL$$

$$\mu - 3\sigma = LNTL$$

Specifying Tolerance

- Tolerance can be specified in four ways
  - Bilateral, unilateral, limits and geometric
- Bilateral
  - Plus or minus deviation from the nominal size
- Unilateral
  - Deviation is in one direction from the nominal size
- Limits
  - Maximum and minimum dimensions

Bilateral Tolerance

- A ***bilateral tolerance*** exists if the variation from a target dimension is shown occurring in both the positive and negative directions

SPECIFYING TOLERANCES:BILETERAL

$$10.0mm^{+0.1mm}_{-0.2mm}$$

$$3.000in^{+0.002in}_{-0.004in}$$

Unilateral Tolerance

-

- A ***unilateral tolerance*** exists when a target dimension is given along with a tolerance that allows variation to occur in only one direction.

SPECIFYING TOLERANCES:UNILETERAL

$$10.0mm^{0.0}_{-0.1}$$

$$3.000in^{0.000in}_{-0.002in}$$

Limit Dimensions

-
- ***Limit dimensions*** are two dimensional values stacked on top of each other. The **dimensions** show the largest and smallest values allowed. Anything in between these values is acceptable.

Limit Dimensions

- 500-target dimension
- .020-total tolerance
- .490
- .510

UNSPECIFIED TOLERANCES

- If no tolerances are specified at the dimension level, then general tolerances may be applied by deliberately controlling the number of values past the decimal point on each dimension.

A measuring device

- A measuring device should be able to accurately measure within 1/10th of the total blueprint tolerance identified.

Geometric Tolerances

- Maximum allowable deviation of a form or position from the perfect geometry
- Maximum material condition indicates that a part is made with the maximum allowable material
- Least material condition indicates that a part is made with the minimum allowable material
- Geometric tolerances are specified with respect to a datum or reference surface
- Four tolerances
  - Flatness, straightness, roundness, and cylindricity

Geometric Tolerances

	Tolerance	Characteristic	Symbol
Individual features	Form	Straightness	⏤
		Flatness	▱
		Circularity	○
		Cylindricity	⌭
Individual or related features	Profile	Line	⌒
		Surface	⌓
Related features	Orientation	Angularity	∠
		Perpendicularity	⊥
		Parallelism	//
	Location	Position	⌖
		Concentricity	◎
	Runout	Circular runout	↗
		Total runout	⌰

Notes	∅	Ⓜ	Ⓛ	Ⓢ
	DIA	MMC	LMC	RFS

(a)

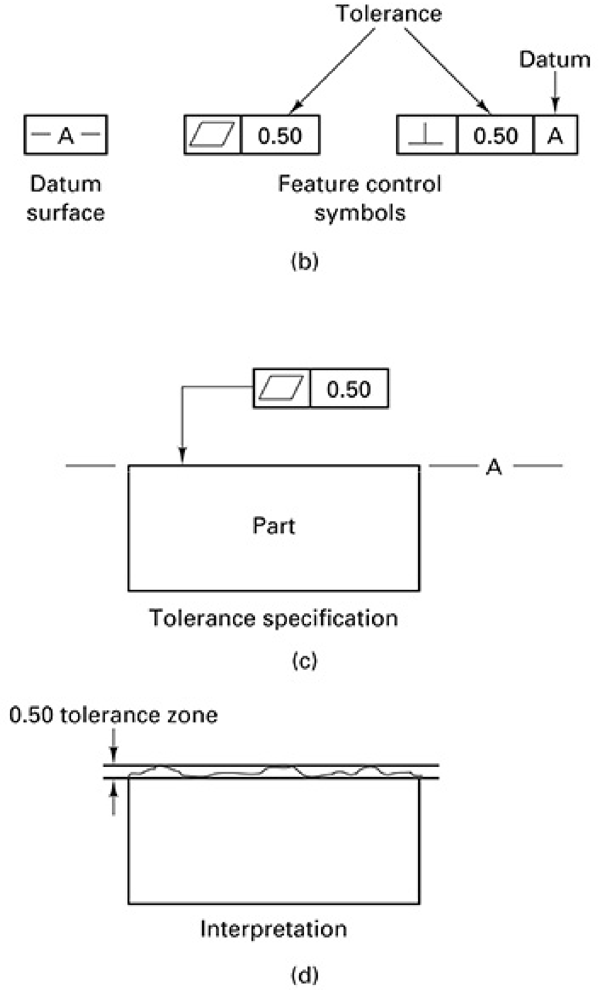


Figure 10-11 (a) Geometric tolerancing symbols; (b) feature control symbols for part drawings; (c) how a geometric tolerance for flatness is specified; (d) what the specification means.

10.3 Allowance and Tolerance

- Allowance- intentional, desired difference between two mating parts
  - Determines the condition of tightest fit
  - May be specified for clearance or interference
- Tolerance- undesirable but permissible deviation from a desired dimensions

Allowance and Tolerance

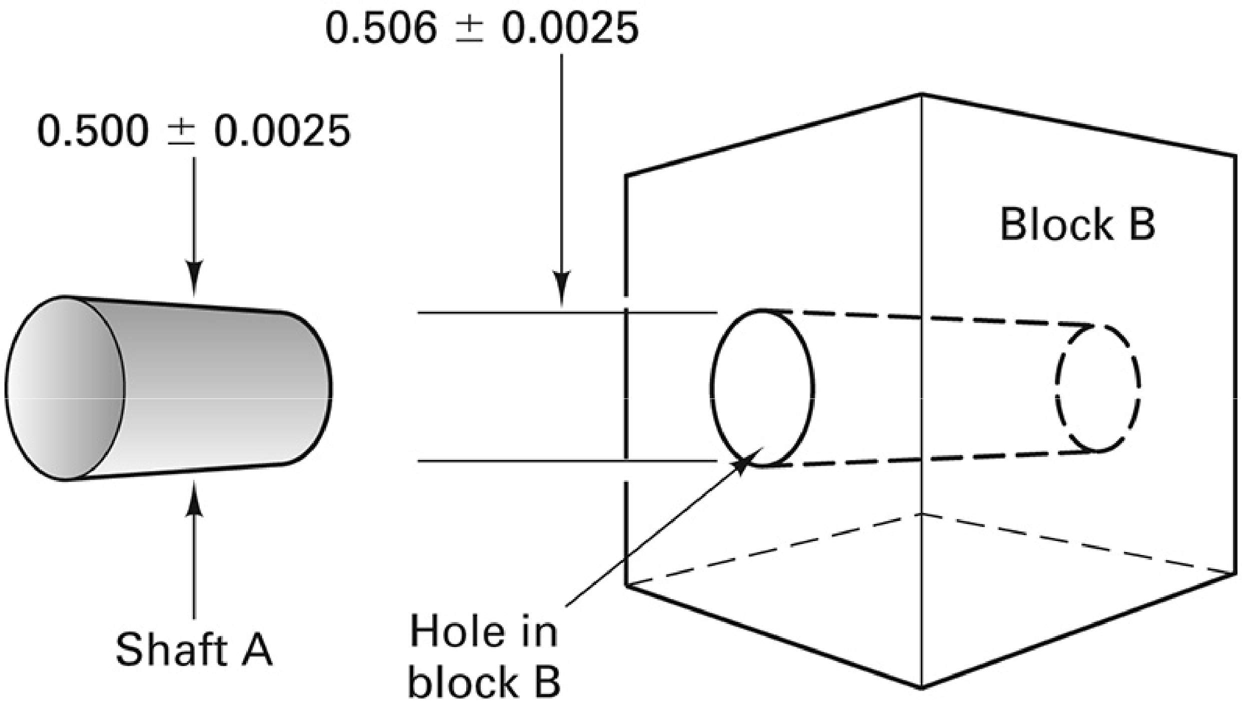


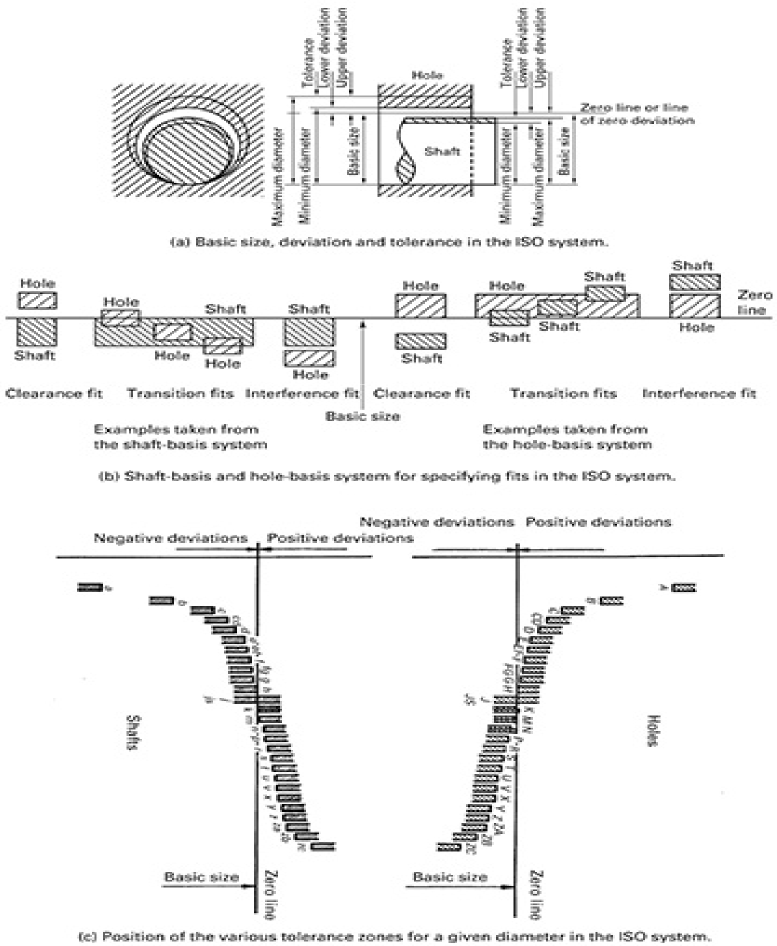
Figure 10-7 When mating parts are designed, each shaft must be smaller than each hole for a clearance fit.

Three Types of Fit

- There are three types of fit that should be considered when working with tolerances.

- Clearance Fit- have limits of size so prescribed that a clearance always results when mating parts are assembled.
- Interference Fit- have limits of size so prescribed that an interference always results when mating parts are assembled.
- Transition Fit- have limits of size indicating that either a clearance or an interference may result when mating parts are assembled.

SHAFT VS HOLE

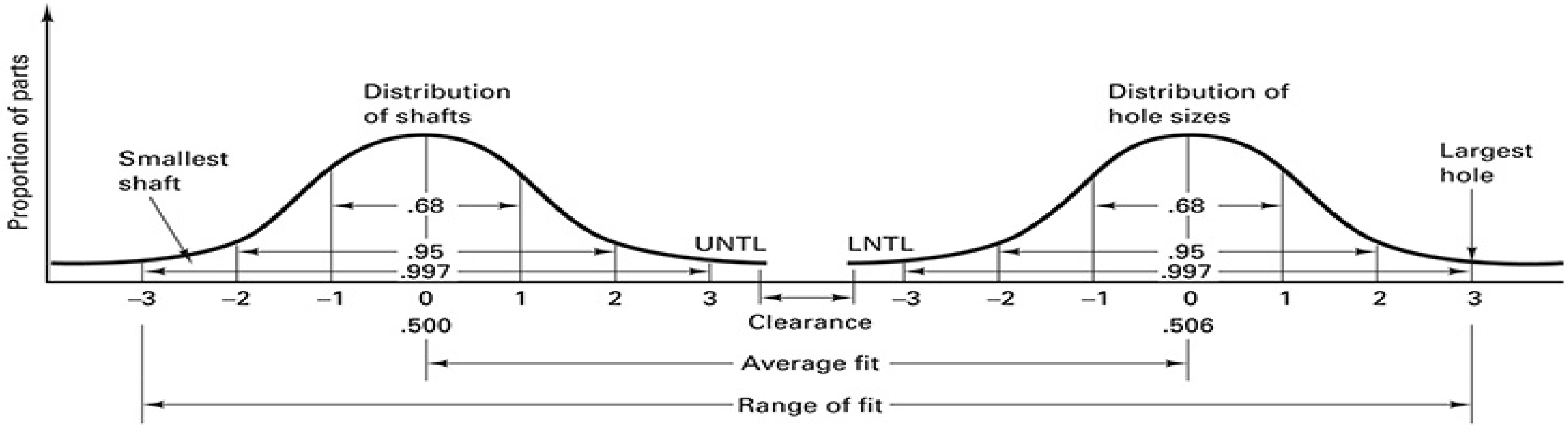


ANSI Classes of Fits

- Class 1: Loose fit
- Class 2: Free fit
- Class 3: Medium fit
- Class 4: Snug fit
- Class 5: Wringing fit
- Class 6: Tight fit
- Class 7: Medium force
- Class 8: Heavy force and shrink fits

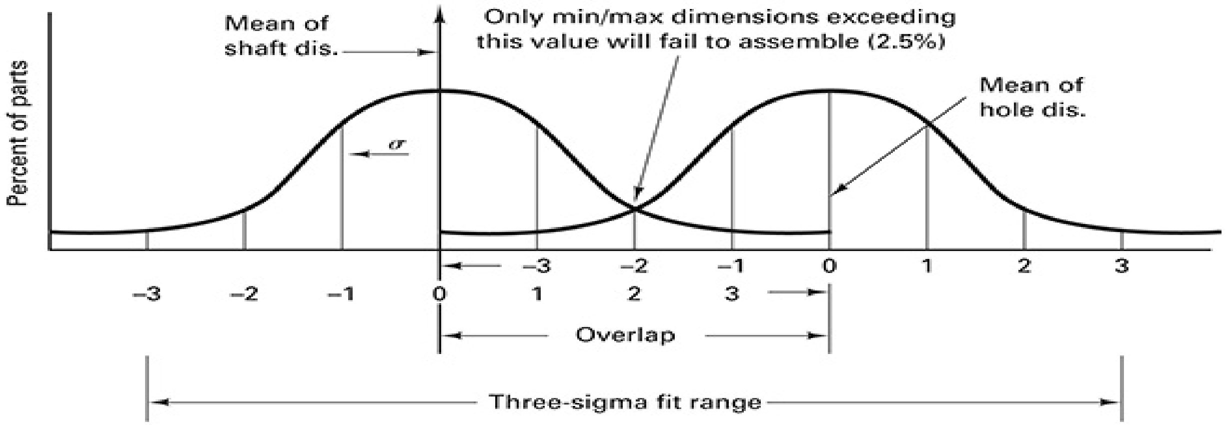
ISO System of Limits and Fits

- Used in metric countries
- Each part has a basic size and each limit is defined by its limit from that size
  - Difference is called the tolerance
- Three classes of fits
  - Clearance
  - Transition
  - Interference
- Tolerances may be specified with respect to zero deviation



Proportion of parts
Distribution of shafts
Distribution of hole sizes
Smallest shaft
Largest hole
.68
.95
.997
UNTL
LNTL
−3 −2 −1 0 1 2 3
Clearance
.500
.506
Average fit
Range of fit

(a)



Percent of parts
Mean of shaft dis.
Only min/max dimensions exceeding this value will fail to assemble (2.5%)
Mean of hole dis.
σ
−3 −2 −1 0 1 2 3
Overlap
Three-sigma fit range

(b)

Testing

- Destructive testing
  - Components are subjected to conditions to induce failure
- Proof testing
  - Product is subjected to a load or pressure of some known and determined magnitude to simulate product life
- Hardness tests
- Nondestructive testing
  - Products are examined in a way that it can still be used

Dormant versus Critical Flaws

- Most materials have flaws of some magnitude
- The extent and possible severity of flaws is important in determining if the flaws in the product can be tolerated
- Larger defects may grow or propagate under cyclic loading
- Identify the conditions below which the flaw remains dormant and above which it becomes critical

Summary

- Measurement and inspection is an important aspect of quality control
- There is a wide variety of techniques that can be employed to make measurements
- The correct technique depends on the application, available equipment, and necessary accuracy
- Cost may play a role in determining which technique is appropriate