Tolerancing
Motivation

- Products are mass-produced
- Parts manufactured in different parts of world, brought together for assembly
- Parts must be interchangeable
- Variability always exists in manufacturing processes

Designer must consider:

- Allowable range of variation of part features
- How parts fit together in assembly
A \textit{tolerance} is the difference between the maximum and minimum size limits on a part.

- $3.25 \pm 0.03$
- lower limit = 3.22
- upper limit = 3.28
- tolerance = 0.06
Tolerances - 2

- Tolerance specifies range for size and location of features on a part that will allow part to function properly
- Used to control variance that exists on all manufactured parts
- State tolerances as generously as possible
  - Provides for a wider variety of processes that can be used to manufacture part
  - Keeps part cost low
Declaring Tolerances

Methods to indicate tolerances include:

- Direct tolerance methods
  - Limit tolerances
  - Plus and minus tolerances
- General tolerance notes
  - E.G., ALL DIMENSIONS HELD TO ± .05
- Geometric Dimensioning and Tolerancing (GDT)
Limit Tolerances

Limit tolerances state upper and lower limits for the dimension range.

The tolerance is the difference between the upper and lower limits.

\[ \varnothing 25.0 - 25.1 \]

80.000
79.970

35.1°
34.9°
Plus/Minus Tolerances

Plus/minus tolerances specify a range that is added or subtracted from a dimension value.

A. Unilateral Tolerancing

B. Bilateral Tolerancing
Tolerance Accumulation

- Tolerance for one dimension added to next dimension in a chain
- Results in a large variation in location of last feature in chain

Tolerance accumulation between X and Y is $\pm 0.03$
Baseline Dimensioning

- Locates a series of features from a common base feature
- Not as much tolerance stack up

Tolerance accumulation between X and Y is ± 0.02
Direct Dimensioning

If needed, can directly apply dimension between two surfaces

Tolerance accumulation between X and Y is ± 0.01
Fit

- Refers to how tightly (loosely) mating parts must fit together when assembled
- Tolerances specified based upon fit
Fit Types

- **Clearance** – internal member (e.g., shaft) always smaller than external member (e.g., hole)
- **Interference** – internal member always larger than external member
- **Transition** – either internal or external may be larger so that parts either slide together or can be forced together
- **Line** – one of the limits on both the hole and the shaft are equal → shaft and hole may have same size
Fit Types – Clearance, Interference

Clearance Fit

Interference Fit
Fit Types – Transition, Line

Transition Fit

Line Fit
Allowance

- Difference between smallest hole size and largest shaft size (tightest fit)
- Allowance is either minimum clearance (+) or maximum interference (-) between parts

**Clearance fit – allowance is positive**

**Interference fit – allowance is negative**
Tolerances Calculations

- Tolerances on hole, shaft
- Allowance
- Basic size - theoretical size from which a tolerance is assigned
- Type of fit
- Needed – method of calculation
Basic Hole System (English Units)

- Uses external member (often a hole) as basic size from which tolerances are determined
- Lower limit (smallest) hole used as basic size
  - A hole can be made larger, but not smaller
- Choose standard drill size for hole, turn down shaft to fit hole
- Most common technique
Basic Shaft System (English Units)

- Uses internal member (often a shaft) as basic size from which tolerances are derived.
- Largest shaft size chosen as basic size.
  - A shaft can be made smaller, but not larger.
- Choose standard shaft diameter, drill hole to suit fit.
- Less common than basic hole system.
Metric Fit Tables

- Hole-basis system
- Shaft-basis system
Tolerancing