Content and/or textbook subject to change without notice

Pennsylvania College of Technology Workforce Development and Continuing Education

Geometric Dimensioning & Tolerancing
INT 727

Course Outline

Course Description: Gain technical knowledge in interpreting geometric dimensioning and tolerancing standards used in industry. Topics include basic principles and applications, geometric symbols, ANSI and ISO drawing standards, datums, form tolerances, runout, and profile control

Textbook: Fundamentals of Geometric Dimensioning & Tolerancing

Prerequisites: Basic understanding of mechanical drawings

Course Length: 30 hours

Course Objectives: Participants will gain technical knowledge in interpreting geometric dimensioning and tolerancing standards.

Course Outline:

- **Definitions**
  - Absolute limits
  - Tolerance accumulation
  - Envelope of perfect form at MMC
  - Form control
- **Geometric Tolerance Basics**
  - Definition
  - Advantages
  - When to use geometric tolerances
  - Symbols
  - Modifiers
  - Basic dimensions
  - Feature control frame
  - Datum feature symbol
  - Symbol placement

- **Form Tolerances**
  - Roundness
  - Cylindricity
  - Straightness
  - Flatness

- **Datums**
  - Datum types
  - Datum targets
  - Three plane datum concept
  - Datum selection

- **Orientation Tolerances**
  - Parallelism
  - Perpendicularity
  - Angularity

- **Profile Tolerances**
  - Profile of a surface/line
  - Profile of tolerancing of a conical feature
  - Profile of tolerancing of coplanar surfaces (coplanarity)

- **Runout Tolerance**
  - Circular runout
  - Total runout

- **Location Tolerances**
  - Concentricity
  - Position