PD570
Geometric Dimensioning & Tolerancing Fundamentals

CEUs: 1.5
PDHs: 15
Number of Days: 2

Not only is ASME's Y14.5 Standard considered the authoritative guideline for the design language of geometric dimensioning and tolerancing (GD&T); it is essential in ensuring that drawing information and symbols are being interpreted and communicated properly. Establishing uniform practices for stating and interpreting GD&T on engineering drawing and related documents is a critical component of the manufacturing of a part. These good design best practices are directly related to a product's innovation, beauty and success.

This official ASME course is based on the latest ASME Y14.5-2018 Standard and makes the GD&T concepts easy to learn and apply. By combining lecture with animated graphics and display models, this course aims to ensure that all students are engaged throughout.

By participating in this course, you will learn how to successfully:

- Demonstrate a thorough understanding of the fundamental geometric concepts
- Interpret and explain the application of geometric symbols on drawings
- Explain how to select the datum reference frames properly
- Calculate the position and profile tolerances.

Course Materials (included in purchase of course):

- A copy of the 400-page *GeoTol Pro 2020 Book and Pocket Guide*, by Scott Neumann and Al Neumann, which contains full color graphics and loaded with student exercises. The training materials will serve as a valuable resource long after the training is complete.

Who Should Attend?
The Designer, the approver/decision maker, and anyone who needs to read and/or interpret engineering drawings and their intent. Engineering, designing, drafting, quality control, procurement, tolling, production, purchasing, manufacturing, CAD inspection, and shop personnel.

Day One

- **Introduction to Geometric Dimensioning and Tolerancing**
  - General overview
  - Geometric characteristic symbols
  - Rules, terms and definitions
  - Introduction to measurement principles, open set-up and CMM (coordinate measuring machine)
- **Limits of Size**
  - Rule #1, size controls form
  - Features with & without size
  - Problems with plus/minus tolerancing
- **How the Geometric System Works:**
  - Introduction to the datum reference frame
  - Basic dimensions
  - Introduction to position tolerancing
  - MMC, LMC, RFS material condition modifiers
  - Introduction to profile tolerancing

Day Two

- **The Datum Reference Frame:**
  - Datums, datum features, true geometric counterparts
  - Holes, slots, shafts, widths as datum features
  - Connection between the theory and physical
  - Constraining the 6 degrees of freedom
  - Creating a complete datum reference frame
- Product Plans
  - Product definition drawing, manufacturing process plan, dimensional measurement plan
  - Position tolerancing and verification
  - Inspection and reporting of position tolerance RFS and at MMC
  - Paper gage
- Orientation Tolerances
  - Parallelism tolerance
  - Perpendicularity tolerance
  - Angularity tolerance
  - The hierarchy of tolerances: location, orientation, form
- Practical Application of GD&T (Case study examples)
  - Applying GD&T to blank drawings
  - Selecting a datum reference frame and calculating tolerances