PD570 Geometric Dimensioning & Tolerancing Fundamentals I

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
- Limits of Size
  - Rule #1
  - Features with & without size
  - Problems with plus/minus tolerancing
  - Limits & fits
- How the Geometric System Works:
  - Introduction to the datum reference frame
  - Datum precedence
  - Basic dimensions
  - Introduction to Position Tolerancing
  - MMC, LMC, RFS feature modifiers
  - Introduction to Profile Tolerancing

Day Two

- Position Tolerancing and Verification
  - Verifying position
  - More on MMC, LMC, RFS feature modifiers
  - Paper gage
- Product Plans and Virtual Condition
  - Product definition drawing, manufacturing process plan, dimensional measurement plan
  - Introduction to boundaries
  - Calculating virtual size
  - Introduction to Datum Modifiers
  - Perpendicularity as a refinement of position
- The Datum Reference Frame:
  - Datums, datum features, datum feature simulators
  - Holes, slots, shafts, tabs, widths as datum features
  - Connection between the theory and physical
  - Datum feature precedence
  - Constraining the degrees of freedom
  - Datum feature simulator requirements
  - Partial datum features
  - Creating a Complete Datum Reference Frame
- Practical Problems