PD570: Geometric Tolerancing Fundamentals 1 (2 Days)

Day 1

- **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 and without size; regular and irregular features of size; limits and fits
- **How the Geometric System Works** – Problems with plus/minus tolerancing; introduction to the datum reference frame; datum precedence, basic dimensions; position and profile; MMC, LMC, RFS feature modifiers; Form tolerances, flatness, straightness, roundness, circularity; Orientation tolerances, parallelism, perpendicularity, angularity

Day 2

- **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; Perpendicularity as a refinement of position
- **The Datum Reference Frame** – Datums, datum features, datum feature simulators; Holes, slots, shafts, tabs, widths, hole patterns as datum features; Connection between the theory and physical; Constraining the degrees of freedom; Datum feature simulator requirements; Advanced datum reference frames; Datum targets; Partial datum features; Datum feature modifiers – MMB, LMB, RMB
- **Profile and Runout Tolerances -** Profile of a surface and line; Using profile to control size, form, orientation and location; Bilateral and unilateral tolerances; Application and verification principles; Circular Runout; Total Runout.