



## ADVANCED GEOMETRIC DIMENSIONING & TOLERANCING

**Onsite: Two 8-hour days Optimum Class Size: 12 participants**

This 2 Day workshop deals with advanced concepts and builds on the basics of GD&T. GD&T is a system for defining and communicating engineering tolerances. It uses a symbolic language on engineering drawings and computer-generated three dimensional solid models that explicitly describe nominal geometry and its allowable variation. GD&T tells the manufacturing staff and machines what degree of accuracy and precision is needed on each controlled feature of the part. GD&T is used to define the nominal geometry of parts and assemblies, to define the allowable variation in form and possible size of individual features, and to define the allowable variation between features. The course is based on Dimensioning and Tolerancing ASME Y14.5 standard.

**Audience:**

Any individual involved in designing, manufacturing, inspecting, and purchasing parts and assemblies.

**Prerequisites:**

Basic math and basic understanding of engineering drawings is required. Blueprint reading skills and working understanding of common measuring devices suggested.

**Program Content:**

Students will get a thorough introduction to the Y14.5 standard. The principal objective is to give those whom are required to interpret blueprints the skills to understand the basics of geometric dimensioning and tolerancing including:

**Day One**

- Introduction to ASME Y14.5-2009
- GD&T Symbols
- Material Conditions
- Rules of GD&T
- Bonus Tolerance
- Datum Reference Frame
- Straightness
- Circularity
- Parallelism
- Angularity
- Concentricity
- Runout

• Coordinate Tolerancing vs. GD&T
• Basic Dimensions
• Feature Control Frame
• Virtual Condition
• The Datum System
• Degrees of Freedom
• Flatness
• Cylindricity
• Perpendicularity
• Position
• Symmetry
• Profile



## Day Two

### Advanced Topics

- Profile tolerance in depth
- Using Patterns as datums
- Auxiliary datums and inclined datum features
- Difference between simultaneous and separate requirements
- Composite position control
- Composite Gauging
- Using slots as datums
- How are Patterns toleranced and what does the callout mean
- Refinement of Orientation and Form in Position and Profile tolerance
- Concept of simultaneous requirements- What does it mean? What does it imply?
- Gauging with simultaneous requirements; Gauging with separate requirements
- Composite vs Multi segment position control
- Tolerancing of slots - Methods and their differences