Feature:

The two days Geometrical dimensioning and tolerancing training is an extensive program which gives complete understanding about ASME Y14.5M- 2009.The program is packed with number of exercises to practice, more than a dozen of case studies are discussed to give participants a hands-on practice to identify section in a particular scenario.

Course Objective:

1) To understand Geometric characteristics & their symbols
2) Other related symbols Datum feature symbol
   - ❖ Datum
   - ❖ Feature control frame &
3) Material Condition
   a. Maximum & least material condition, Regardless of feature size
4) Basic Dimension, Allowance, Clearance
5) ASME General rules
6) True position & Virtual condition
7) Bonus Tolerance

Who Should Attend?

Designers, QC inspectors, Product layout inspectors, New product development team, etc.

Course Duration:

2 Days

Course Content:

Day 1	
1) Introduction	9) Orientation
2) Fundamental Drawing Rules	10) Location
3) Units of measurement	11) Composite control
4) Types of Dimensions	12) Profile
5) General Tolerancing and Related Principles	13) Modifying Symbols
	14) Introduction to audit cases
6) Orthographic Projection	15) Datum Referencing
7) Geometric characteristics	16) Placement of Datums
8) Form	17) Feature control frame

Day 2	
1) Material Condition 2) ASME Rule 1-Individual Feature of Size 3) Geometric characteristics 4) Bonus Tolerances 5) Form Geometry 6) Way to specify Form Tolerance 7) Interpret limits 8) Inspection Method 9) The type of controls	10) Control of Orientation 11) Control of Location 12) Composite Control 13) Appendix 14) Position Tolerance Special Cases 15) Principal changes and improvements 16) Composite pattern of features 17) Projected Tolerances 18) Categories of surface roughness