



INTRODUCTION TO GEOMETRIC TOLERANCING TRAINING COURSE

COURSE LENGTH: 3 Days

OVERVIEW: In this class students will be shown how to interpret design drawings and CAD representations of product definitions that use modern geometric tolerance standards. The class includes step-by-step procedures to apply the practices and allows dimensioning and tolerancing professionals to express their design requirements clearly. After completing this course, users will be able to create production representations that are able to be specific in conveying tolerancing needs for products that can be more easily manufactured and with appropriate inspection techniques clarified.

PREREQUISITES: Users who take this class should have familiarity with basic drawing creation for design and manufacturing.

OBJECTIVES:

- Symbols, rules and principles instituted by ASME Standards
- Basic GD&T
- Advanced GD&T
- Inspection Techniques and Sources of Measurement Uncertainty
- Tolerancing Mating Parts and Assemblies

REQUIRED BOOKS: (1) Geometric Dimensioning and Tolerancing – Applications, Analysis & Measurement (2009), by James D. Meadows and (2) Workbook and Answer book for Geometric Dimensioning and Tolerancing (2009), by James D. Meadows



INTRODUCTION TO GEOMETRIC TOLERANCING

COURSE MODULES

- 1 Symbols, Rules, Charts
- 2 Selecting a Tolerancing Approach
- 3 Datum Feature Simulators
- 4 Boundaries and Material Condition Symbols, MMC, LMC & RFS
- 5 Major Concepts of Geometric Dimensioning and Tolerancing
- 6 Form
- 7 Orientation
- 8 Profile
- 9 Runout
- 10 Concentricity and Symmetry
- 11 Datums
- 12 Centerplane Datums
- 13 Datum Targets