



## Fundamentals of GD&T Workshop, based on ASME Y14.5-2009

Learn how to interpret engineering drawings produced in accordance with the ASME Y14.5-2009 standard. You will discover the terms, rules, symbols, and concepts of GD&T as prescribed in the standard, as well as gain an in-depth understanding of the geometric symbols, including each symbol's requirements, tolerance zones, and limitations. Numerous practice problems are assigned reinforcing and ensuring you retain the knowledge learned.

### Who Should Attend

This course is ideal for individuals who create or interpret engineering drawings including design, layout, assembly, checking and inspection personnel.

### Skill Level Needed

Students should have basic print reading skills.

### Course Agenda and Highlights

- **Introduction**  
Drawing Standards; Dimensions, Tolerances, and Notes Used on Drawings; Coordinate Tolerancing and GD&T; General Dimensioning Symbols.
- **Fundamentals**  
Key GD&T Terms; Symbols and Modifiers; GD&T Rules; GD&T Concepts.
- **Form Controls**  
Flatness Tolerances; Straightness Tolerances; Circularity Tolerances; Cylindricity Tolerances.
- **Datum's**  
The Datum System; Datum Targets; Size Datum Features (RMB); Size Datum Features (MMB).
- **Orientation Controls**  
Perpendicularity Tolerances; Angularity Tolerances; Parallelism Tolerances
- **Position Controls**  
Position Tolerance Introduction; Position Tolerance – RFS and MMC; Special Applications; Calculations and Formulas.
- **Runout, Concentricity and Symmetry Controls**
- Circular and Total Runout Tolerances; Concentricity and Symmetry Tolerances.
- **Profile Controls**  
Profile Basic Concepts; Profile Tolerance Applications.

### Learning Outcomes

You will learn the benefits of GD&T and leave with the ability to interpret GD&T on drawings. You will gain a thorough understanding of the fundamental concepts of GD&T ensuring you have a solid foundation for understanding more advanced topics.