

Geometric Dimensioning and Tolerancing (GD&T)

Course Description

Industry uses GD&T to specify contractual requirements. Unfortunately, many people within industry have a flawed or incomplete understanding of the subject. Our basic course builds competence to correctly apply and interpret the rules, definitions, principles, and symbols per the American National Standard, ASME Y14.5 – 2009, in only two days! This course provides a common language to improve communication, so set-up requirements and tolerance zones are clearly understood by all.

If you fixed communications in your company, how many of your problems would go away overnight?

Course Outline

1. Definitions and Fundamental Rules
2. Dimensions, Tolerances and GD&T
3. Datum Features and Datum Schemes
4. Form Tolerances
5. Orientation Tolerances
6. Location Tolerances
7. Profile Tolerances
8. Runout Tolerances
9. Putting It All Together
10. A Business Case for GD&T

Key Course Objectives

- ✓ Learn the rules, definitions, and symbols of ASME Y14.5 – 2009.
- ✓ Interpret specified feature control frames.
- ✓ Interpret and simulate specified datum reference frames.
- ✓ Describe the shape, size, and location of tolerance zones.
- ✓ Calculate feature size requirements given specified tolerances.