



• WORKFORCE DEVELOPMENT SOLUTIONS •
CORE Manufacturing Skills Training

Includes ASME
Y14.5 GD&T
Pocket Guide

FUNDAMENTALS OF GEOMETRIC DIMENSIONING & TOLERANCING

November 14, 2018

8am – 4pm

Best Western - Manchester, NH

[Register Online](#)

This public workshop provides an introduction to geometric dimensioning and tolerancing. The course is based on ASME Y14.5-2009 standards.

Audience

Employees with a current understanding of basic blueprint reading and basic math skills, including experienced operators, machinists and quality control staff. This introductory class is not appropriate for experienced designers and design engineers, but could serve as an overview for newer engineers who have not had formal instruction in GD & T. While measuring methods and how they relate to drawing specifications are discussed this is not a course in inspection techniques.

Program Content

Students will be taught an overview of the core concepts of geometric tolerances and their applications. Case study engineering drawings are developed, using G D and T methods and symbols, with comparison to traditional (directly toleranced) dimensioning methods:

- What GD&T is and how it's different
- Symbols and Terms for GD&T
- "Rule # 1"- Limits of Size
- Feature control frame
- Datums and Datum Reference frame
- Order of precedence
- Feature Modifiers (MMC, LMC, RFS)
- Virtual Condition
- Position tolerances
- Form Tolerances
 - Flatness
 - Straightness
 - Circularity
 - Cylindricity
- Orientation Tolerances
 - Perpendicularity
 - Angularity
- Profile Tolerances