
Geometric Dimensioning and Tolerancing with Y14.5-2009 On-Site Seminar

Geometric Dimensioning and Tolerancing (GD&T) was developed as an international language to communicate exactly how a part should be made to achieve the required functionality. When properly applied, GD&T is a powerful tool for achieving quality and for reducing costs in design, manufacturing, and inspection.

The American Society of Mechanical Engineers (ASME) document Y14.5-2009 *Dimensioning and Tolerancing* is the latest U.S. standard on this language. Our On-Site Seminar, *Geometric Dimensioning and Tolerancing with Y14.5-2009*, provides in-depth information on the use and interpretation of this important standard.

The Seminar is designed for presentation to a mixed audience of Engineering, Manufacturing, Purchasing, and Quality Department personnel. The program can be customized to meet the specific needs of:

- Engineering, Manufacturing, Purchasing, or Quality Managers
- Design, Manufacturing, or Production Engineers
- Part Programmers
- Fixture Designers
- Production Supervisors, Setup Personnel, or Machine Operators
- Quality Engineers, Gage Designers, or Inspectors

Geometric Dimensioning and Tolerancing with Y14.5-2009 consists of 14 instructional units and requires approximately 32 to 40 hours for presentation. Participants receive a Seminar Binder containing copies of all presentation materials, copies of selected illustrations, reprints of key articles and technical papers, and additional materials. Various presentation schedules are available to meet your specific requirements.

Call us to learn how you can put the power of Geometric Dimensioning and Tolerancing to work for your company.

Seminar Outline:

Unit 1 - Background

- Introduction
- Scope
- References
- Definitions
- Fundamental Rules
- Units of Measure
- Types of Dimensioning
- Application of Dimensions
- Dimensioning Features
- Location of Features

Unit 2 - General Tolerancing and Related Principles

- General
- Direct Tolerancing Methods
- Tolerance Expression
- Interpretation of Limits
- Single Limits
- Tolerance Accumulation
- Limits of Size
- Applicability of Modifiers on Geometric Tolerance Values and Datum Feature References
- Screw Threads
- Gears and Splines
- Boundary Conditions
- Angular Surfaces
- Conical Tapers
- Flat Tapers
- Radius
- Tangent Plane
- Statistical Tolerancing

Unit 3 - Symbology

- General
- Use of Notes to Supplement Symbols
- Symbol Construction
- Feature Control Frame Symbols
- Feature Control Frame Placement
- Identification of the Tolerance Zone
- Tabulated Tolerances

Unit 4 - Datum Reference Frames

- General
- Degrees of Freedom
- Degrees of Freedom Constrained by Primary Datum Features Regardless of Material Boundary
- Constraining Degrees of Freedom of a Part
- Datum Feature Simulator
- Theoretical and Physical Application of Datum Feature Simulators
- Datum Reference Frame
- Datum Features
- Datum Feature Controls
- Specifying Datum Features in an Order of Precedence
- Establishing Datums

Innovations PLUS



11 Helmsford Way
Penfield, NY 14526-1971
Phone: 585-377-8340
www.iplusnet.com

Unit 4 - Datum Reference Frames (continued)

- Multiple Datum Features
- Mathematically Defined Surface
- Multiple Datum Reference Frames
- Functional Datum Features
- Rotational Constraint About a Datum Axis or Point
- Application of MMB, LMB, and RMB to Irregular Features of Size
- Datum Feature Selection Practical Application
- Simultaneous Requirements
- Restrained Condition
- Datum Reference Frame Identification
- Customized Datum Reference Frame Construction
- Application of a Customized Datum Reference Frame
- Datum Targets

Unit 5 - Tolerances of Form

- General
- Form Control
- Specifying Form Tolerances
- Form Tolerances
- Application of Free-State Symbol

Unit 6 - Tolerances of Orientation

- General
- Orientation Control
- Orientation Symbols
- Specifying Orientation Tolerances
- Tangent Plane
- Alternative Practice

Unit 7 - Tolerances of Location

- General
- Positional Tolerancing
- Positional Tolerancing Fundamentals: I
- Positional Tolerancing Fundamentals: II
- Pattern Location
- Coaxial Feature Controls
- Tolerancing for Symmetrical Relationships

Unit 8 - Tolerances of Profile

- General
- Profile
- Tolerance Zone Boundaries
- Profile Applications
- Material Condition and Boundary Condition Modifiers as Related to Profile Controls
- Composite Profile
- Multiple Single-Segment Profile Tolerancing
- Combined Controls

Unit 9 - Tolerances of Runout

- General
- Runout
- Runout Tolerance
- Types of Runout Tolerances
- Application
- Specification

Unit 10 - Principal Changes and Improvements

- General
- Standard Format
- Section Changes

Unit 11 - Formulas for Positional Tolerancing

- General
- Formula Symbols
- Floating Fastener Case
- Fixed Fastener Case When Projected Tolerance Zone is Used
- Provision for Tilting of the Axis or Center Plane When Projected Tolerance Zone is Not Used
- Coaxial Features
- Limits and Fits

Unit 12 - Form, Proportion, and Comparison of Symbols

- General
- Form and Proportion
- Comparison

Unit 13 - Former Practices

- General
- Specification of RFS for Position Tolerances
- Specification of Straightness to Control the Flatness of a Derived Median Plane
- MMC, LMC, and RFS for Datum Features

Unit 14 - Decision Diagrams for Geometric Control

- Purpose
- Functional Requirements
- Reference to Standard
- Geometric Controls
- Choosing Other Controls
- Use of Modifiers
- Datums

Note: For companies implementing ASME Y14.5M-1994 (*reaffirmed 2004*), please see the data sheet for our On-Site Seminar *Geometric Dimensioning and Tolerancing with Y14.5M-1994*.

For more information on our complete range of products and services, please visit us on the web, call, or write...

Innovations PLUS



11 Helmsford Way
Penfield, NY 14526-1971
Phone: 585-377-8340
www.iplusnet.com