Geometric Dimensioning and Tolerancing On-Site Seminar

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

The American Society of Mechanical Engineers (ASME) document *Y14.5M-1994 Dimensioning and Tolerancing* is the current US standard on this language. Our On-Site Seminar *Geometric Dimensioning and Tolerancing*, 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 consists of 16 instructional units and requires approximately 32 to 40 hours for presentation. Participants receive a copy of ASME Y14.5M-1994 (Reaffirmed 2004), 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 you.

Seminar Outline:

Unit 1 - Background
- Introduction
- References
- Definitions
- Fundamental Rules
- Units of Measurement
- Types of Dimensioning
- Application of Dimensions
- Dimensioning Features
- Location of Features

Unit 2 - General Tolerancing
- Direct Tolerancing Methods
- Tolerance Expression
- Interpretation of Limits
- Single Limits
- Tolerance Accumulation
- Limits of Size
- Applicability of RFS, MMC, and LMC
- Screw Threads
- Gears and Splines
- Virtual/Resultant Condition
- Angular Surfaces
- Conical Tapers
- Flat Tapers
- Radius
- Statistical Tolerancing

Unit 3 - Symbology
- Use of Notes to Supplement Symbols
- Symbol Construction
- Geometric Tolerance Symbols
- Feature Control Frame Placement
- Identification of the Tolerance Zone
- Tabulated Tolerances

Unit 4 - Datums
- Immobilization of Part
- Datum Features
- Specifying Datum Features in a Order of Precedence
- Establishing Datums
- Datum Targets

Innovations PLUS

11 Helmsford Way
Penfield, NY 14526-1971
Phone: 585-377-8340
Fax: 585-377-7569
www.iplusnet.com

Printed in USA.

Unit 5 - Positional Tolerancing

- Positional Tolerancing
  - Method
  - Application to Base Line and Chain Dimensioning
- Fundamental Explanation of Positional Tolerancing
  - Material Condition Basis
  - MMC as Related to Positional Tolerancing
  - Zero Positional Tolerance at MMC
  - RFS as Related to Positional Tolerancing
  - LMC as Related to Positional Tolerancing
  - Multiple Patterns of Features Located by Basic Dimensions Relative to Common Datums

Unit 6 - Feature Pattern Positional Tolerancing

- Composite Positional Tolerancing
- Projected Tolerance Zone
- Nonparallel Holes
- Counterbored Holes
- Closer Control at One End of a Feature
- Bidirectional Positional Tolerancing of Features

Unit 7 - Other Positional Tolerancing

- Noncircular Features
- Coaxiality Controls
- Concentricity
- Positional Tolerancing for Symmetrical Relationships
- Symmetry Tolerancing to Control the Median Points of Opposed or Correspondingly Located Elements of Features
- Spherical Features

Unit 8 - Form Tolerancing

- Form and Orientation Control
- Specifying Form and Orientation Tolerances
- Form Tolerances
  - Straightness
  - Flatness
  - Circularity (Roundness)
  - Cylindricity

Unit 9 - Profile Tolerancing

- Profile Control
  - Profile Tolerancing
  - Tolerance Zone
  - Explanation of Profile Tolerance
  - Application of Datums
  - Combined Controls
  - Coplanarity
  - Profile Tolerance for Plane Surfaces
  - Profile Tolerance for a Conical Feature
  - Composite Profile

Unit 10 - Orientation Tolerancing

- Specifying Orientation Tolerances in Relation to Datum Features
- Angularity
- Parallelism
- Perpendicularity

Unit 11 - Runout Tolerancing

- Runout Tolerance
- Free State Variation

Unit 12 - Principal Changes and Improvements

Unit 13 - Formulas for Positional Tolerancing

- Formula Symbols
- Floating Fastener Case
- Fixed Fastener Case When Projected Tolerance Zone is Used
- Provision for Out-of-Squareness When Projected Tolerance is Not Used
- Coaxial Features
- Limits and Fits

Unit 14 - Form, Proportion, and Comparison of Symbols

Unit 15 - Former Practices

- Definition for Feature of Size
- Applicability of RFS, MMC, and LMC
- Tangent Radii
- Datum Feature Symbol
- Projected Tolerance Zone

Unit 16 - Decision Diagrams for Geometric Control

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

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
Fax: 585-377-7569
www.iplusnet.com

Printed in USA.