Course ID
GD&T
Course Duration
2 days

Course Title
Geometric Dimensioning and Tolerancing

Related Courses

- Geometric Dimensioning & Tolerancing (GD&T) Workshop (GDT3D, 3 days)
- Tolerance Stack Analysis Using GD&T (TOL-GDT, 2 days)
- Statistical Tolerance Analysis: A Comprehensive Workshop (S-TOL-ANAL, 2 days)

Aimed At

This course is aimed at all who use GD&T to design, produce, and inspect parts, including engineers, designers, managers, production planners, manufacturing supervisors, machinists, quality engineers, inspectors, and supplier quality specialists.

Group Size

5-25

Prerequisites

While there are no formal prerequisites, the course assumes a design, process, industrial, manufacturing, or engineering background.

Course in a Nutshell

Geometric dimensioning and tolerancing represents a major leap forward in accurately defining product configurations in engineering drawings. The geometric dimensioning and tolerancing approach overcomes shortfalls inherent in traditional engineering drawings, and it has become a common language for communicating product configuration requirements. It provides a communications baseline between engineering, design, manufacturing, quality assurance, procurement, and other organizations, greatly reducing the need for supplementary drawing notes, and providing a consistent approach for gages, fixtures, and assembly jigs.

The *Geometric Dimensioning and Tolerancing* course is an intense training program focused on developing requisite skills for understanding and defining dimensions, datums, form and orientation controls, and tolerancing. The course provides a common understanding and a working knowledge of geometric dimensioning and tolerancing principles.

The *Geometric Dimensioning and Tolerancing* course includes lectures, interactive discussions, group exercises, and numerous examples to assure thorough understanding of the symbols, terminology, and rules defined in the current ANSI Y14.5 2009 standard.

Customize It!

Whatever the nature of your systems, products, and objectives, we will customize the course to meet your specific needs and concerns. Here are some of the ways in which we can tailor the course to help you get more out of it:

- Additional "workshop days" to allow the participants to work together to address geometric dimensioning and tolerancing topics specific to your organization. The workshop day can be scheduled a few weeks after the course to allow time for applying the technologies presented in class under an experienced engineer.
- Schedule post-class follow-up consultation for continuing in-house geometric dimensioning and tolerancing implementation.

Learn How To

- Work together to implement geometric design and tolerancing concepts.

Course Outline

Day 1: Geometric Dimensioning and Tolerancing Concepts

- **Dimensioning and Tolerancing Introductory Concepts.** Dimensional control history. Dimensioning standardization and standards. The nature of variation, including process, tool wear, operator error, operator bias, material variations, ambient conditions, processing equipment differences, maintenance, inspection variation, and assembly variation.

  Plus/minus (±) tolerances and geometric tolerances. Title block tolerances. Local ± tolerances. Geometric dimensioning and tolerancing. Geometric dimensioning and tolerancing advantages.

- **Dimensioning and Tolerancing Symbology.** General symbols and abbreviations. Symbol application. Geometric tolerancing symbols. Symbol shapes and sizes. Feature control frames. Feature control frame composition. Feature control frame placement. Surfaces, features of size, threads, gears and splines, and datums. Datum feature and target symbols. Datum targets. Basic dimensions and basic dimension symbol.

  Group exercises.

- **Basic Geometric Dimensioning Concepts.** Form, orientation, location, and profile. Dimensioning rules, features, feature of size, local size, mating envelope, mating size, modifiers and symbols, feature control frames, Rules 1, 2, and 2A, bonus tolerance, virtual condition, and inner and outer boundaries.

  Group exercises.

- **Datums and Datum References.** Datum description, technology, and terminology. The datum reference frame. Datum identification. Datum

functions and references. Datums applied to planes, cylindrical features, slopes, and other features. Datum features of size. Datum simulation. Datum order of precedence. Datum references in feature control frames.

Group exercises.

- **Form and Orientation Controls.** Flatness, straightness, circularity, cylindricity. Surface, axis, and centerplane applications. Perpendicularity, parallelism, and angularity. When and how to apply orientation controls. Modifiers. Tangent plane, surface, axis, and center plane applications. Controlling hole axis orientation and virtual condition of a hole. Orientation as a form control. Gaging principles.

Group exercises.

Day Two: Tolerancing and Advanced Concepts

Geometric Tolerancing. Position and position tolerancing. Feature control frame position tolerance feature. Implied basic relationships. Material condition modifier application and Rule 2. Position tolerance zone. Basic dimensions and true positions. Relationships to datum reference frames. Material condition modifier effects. Regardless of feature size. Maximum material and least material conditions. Tolerance calculation. Fasteners. Bonus tolerances. Threads. Projected tolerance zones.

Group exercises.

- **Position Tolerancing Advanced Concepts.** Concentricity and symmetry Controls. Concentricity and symmetry definitions. Composite position tolerances. Repeated versus nonrepeated datum referencing. Tolerance zones. Two line feature control frames. Comparisons. Position control. Coaxial and radial hole patterns.

Group exercises.

- **Runout Controls.** Circular runout. Feature control frame. Circular runout applications as applied to cylinders, noncylindrical features, and face surfaces. The datum axis. Single and simultaneous datum features. Limited tolerance zone applications. Total runout and applications. Total runout applied to cylinders and face surfaces. Combined effects of size and runout tolerances.

Group exercises.

- **Profile Tolerances.** Profile specification. Line and surface profiles. Limits of application. All around application. Bidirectional and unidirectional control and tolerance. Achievable control levels. Control of

surface features, including form, orientation, and location. Controls on coplanar features. Coplanarity, orientation, and location. Coplanarity applied to conical surfaces. Composite profile tolerances.

Group exercises.

- **Tolerance Analysis.** Worst case tolerance analysis in the geometric dimensioning and tolerancing world. Tolerance analysis assumptions. ASME and ISO dimensioning and tolerancing standards. Converting geometric dimensioning and tolerancing into equal-bilateral plus/minus tolerances.

  Group exercises.

- **Course Wrap-Up.** Course review. Questions and answers. Plans for future actions.

  Course critique.

How You Will Learn

- A seasoned consulting engineer-instructor will present this course in interactive lecture/workshop format.
- Along with the lectures, we use exercises, puzzles, case studies, and interesting group activities to enrich the instruction and drive home the essential points.
- You will receive a printed Participant Handbook that includes all materials presented in class, which will help you remember and retain what you learned in class and apply it when you are back on your job.
- You will learn geometric dimensioning and tolerancing concepts from a theoretical and practical perspective.

Website: www.eogogics.com **or** www.gogics.com
E-mail: sales@eogogics.com
Tel. +1 (703) 281-3525
USA 1 (888) 364-6442