GD&T (Geometric Dimensioning and Tolerancing)
2 Day Course Outline

Course Description:

This course will identify the many benefits of the geometric system of tolerancing based on ASME Y14.5M-2009 standard. Participants will discover cost savings at the design, manufacturing and inspection stages of the organization as a result of the proper application and implementation of the GD&T system. This course will enable participants to understand and speak the "language" of GD&T.

> **NOTE:** This 16-hour course will include the same topics as the more comprehensive 24-hour course but will not allow for thorough discussions related to each topic due to the tighter time restraint.

Learning Objectives:

- Recognize and apply bonus geometric tolerances
- Easily rework parts to meet requirements
- Use functional gauges to effectively verify geometric requirements
- Recognize extra tolerances from referenced datums
- Inspect geometric tolerances using conventional measuring equipment
- Recognize why good parts may be rejected when using the coordinate measuring machine (CMM)
- Identify which standard was used in creating the drawing, and identify the main differences between the most common standards used.

Course Outline:

Course Introduction

- Course conventions
- Why GD&T is important to my company
- Why GD&T is important to me
- Course expectations.

Engineering Drawing Standards

- What an engineering drawing is
- Importance of an engineering drawing
- ASME related to engineering drawings
- Identifying applicable drawing standards

Dimensions, Tolerances, and Notes Used on Drawings

- Purpose of tolerances
- Dimensioning conventions
- Dimension types
- Tolerance types
- General notes
- Using tolerances on CAD models

Comparison of GD&T and Coordinate Tolerancing

- Why coordinate tolerancing is UNSAFE
- What geometric tolerancing is
- Six major components of GD&T
- Benefits of GD&T

General Dimensioning Symbols

- Radius
- Controlled radius
- Spherical radius
- Diameter
- Spherical diameter
- Square
- Counterbore
- Spotface
- Depth
- Countersink
- "by"
- Maximum dimension
- Minimum dimension
- Reference
- Dimension origin

Key Terms Used in GD&T

- Size and actual local size
- Actual mating envelope
- Axis
- Center plane
- Complex feature
- Feature
- Feature of size
- Irregular feature of size
- Least material condition
- Maximum material condition
- Non-opposed
- Opposed
- Partially opposed
- Pattern
- Regardless of feature size
- Related actual mating envelope
- Simultaneous requirement
- Size dimension
- Unrelated actual mating envelope

GD&T Symbols and Modifiers

- Geometric characteristic symbols
- Five categories
- Five geometric attributes
- Twenty-one geometric modifying symbols
- Feature control frame
- Feature control frame placement
- How the continuous feature modifier affects a feature and feature of size.

GD&T Rules

- 16 fundamental dimensioning rules
- Rule #1
- Rule #1 applied to a feature of size
- Exceptions to Rule #1
- Overriding Rule #1
- Independency concept
- Rule #2
- GO/NOGO gage
- How to inspect Rule #1

GD&T Concepts
- Virtual condition boundary
- Worst-case boundary
- Virtual condition calculations
- Bonus tolerance concept
- Bonus tolerance calculations
- Verification principles for a virtual condition boundary

Flatness Tolerance
- Derived median plane
- Tolerance zones
- Rule #1 as a flatness control
- Real-world applications
- Applied to a planar surface and feature of size
- Bonus tolerance (at MMC)
- Inspection methods

Straightness Tolerance
- Derived median line
- Tolerance zones
- Rule #1 as a straightness control
- Real-world applications
- Applied to a surface and a feature of size
- Bonus tolerance (at MMC)
- Inspection method

Circularity Tolerance
- Tolerance zones
- Rule #1 as a circularity control
- Real-world applications
- Inspection methods

Cylindricity Tolerance
- Tolerance zones
- Rule #1 as a cylindricity control
- Inspection methods.

The Datum System
- Implied datums
- Benefits
- Terminology
- Datum reference frame
- Six degrees of freedom
- Datum reference frame symbol
- Coplanar datum features
- Multiple datum reference frames
- Datum related dimensions

Datum Targets
- The datum target symbols
- Where datum targets are used
- Requirements of datum targets
- Point datum target
- Line datum target
- Datum target simulators
- Movable datum targets
- Datum target applications

Size Datum Features: RMB
- Terms
- Methods to specify a feature of size as a datum feature
- Effects of a datum feature (RMB)
- Coaxial datum features of size
- Datum feature simulator for coaxial datum features of size (RMB)

Size Datum Features: MMB
- Maximum material boundary (MMB)
- Effects of using MMB
- Datum shift
- Datum feature simulators
- Using a hole pattern as a datum feature
- Datum sequence

Perpendicularity Tolerance
- Implied 90° angles
- Perpendicularity and perpendicularity tolerance
- Common tolerance zones
- Indirect perpendicularity tolerances
- Modifiers used with perpendicularity tolerances
- Real-world applications
- Inspection methods

Parallelism Tolerance
- Implied parallel relationships
- Parallelism and parallelism tolerance common tolerance zones
- Indirect parallelism controls
- Modifiers used with parallelism tolerances
- Real-world applications
- Inspection methods

Angularity Tolerance
- Angularity and angularity tolerance
- Common tolerance zones
- Indirect angularity controls
- Modifiers used in angularity tolerances
- Real-world applications

Position Tolerance Introduction
- True position
- Common tolerance zones
- Implied relationships
- Advantages
- Surface and axis interpretations
- Real-world applications

Position Tolerance: RFS & MMC
- Conditions of RFS applications
- Conditions of MMC applications
- Inspection methods

Position Tolerance: Special Applications
- Projected tolerance zone
- Bi-directional control
- LMC application
- Multiple single-segment application
- Elongated hole application
- Zero tolerance at MMC

Position Tolerance Calculations
- Floating fastener assembly
- Fixed fastener assembly
- Floating fastener formula
- Fixed fastener formula
- Calculating position tolerance values
- Limitations of the fixed and floating fastener formulas

Circular & Total & Runout Tolerances
- Circular runout tolerance and total runout tolerance
- Tolerance zone shapes
- Three ways to establish a datum axis
- Real-world applications
- Interpret circular runout tolerance
- Interpret total runout tolerance
- Circular and total runout comparison
- Inspection methods

Concentricity & Symmetry Tolerances
- Median points on centricity tolerance and symmetry tolerance
- Common tolerance zones
- Real-world applications
- Interpreting a concentricity and symmetry tolerances
- Concentricity and total runout comparison
- Inspection methods

Profile Tolerances: Introduction

- Profile
- True profile
- Part characteristics affected by profile tolerances
- Effects of datum references
- Advantages
- Tolerance zone options
- Tolerance zone extents

Profile Tolerances: Applications

- Real-world applications
- Planar surface application
- Coplanar surface application
- Closed polygon application
- Conical surface application
- Multiple single-segment application
- Inspection methods

Class Closing

- Course summary
- Course evaluation
- Customer drawing review (optional)

At the end of each chapter participants will answer questions associated with the topics covered, followed by group discussions and the correct answers revealed.

Location:

Public Course or On-Site

Duration:

Two (2) Days (16 Hours)