

GD&T Introduction Webinar Outline

Note: Course outline will be customized to accommodate customer-specific products, processes, and applications.

Module 1 - Introduction

- Pre-Assessment
- Objectives
- Elements of Engineering Drawings
- Dimensions and Tolerances
- Size, Location, Orientation, and Form
- Tolerancing Options and Expressions
- Coordinate Tolerance Expressions
- Metric Conventions
- Inch Conventions
- Interpretation of Limits
- Evolution of GD&T
- Dimensioning Standards
- Fundamental Rules from ASME Y14.5
- Basic Concepts from ISO 1101
- Mini-stapler Magazine Case Study
- Coordinate System Weaknesses
- Square vs. Cylindrical Tolerance Zones
- Ambiguous Inspection Setups
- Introduction to GD&T
- Inspection of Position w/ MMC after Conversion
- Major Elements of GD&T
- Advantages of GD&T
- New Terms and Definitions
- Exercises

Module 2 – Symbols, Concepts, and Rules

- Objectives
- GD&T Symbols
- Feature Control Frame Construction
- How to Read a Feature Control Frame
- Features and Features of Size
- Identifying Features of Size Using the Caliper Test
- Material Conditions
- Maximum Material Condition
- Least Material Condition
- Symbols and Modifiers
- MMC Modifier Application
- Bonus Tolerance Summary
- RFS Material Condition
- Rule #1 – Size Controls Form, Envelope Principle
- ISO Independency Principle
- Rule #2 – RFS Implied
- Basic Dimensions

- Virtual and Resultant Conditions, Bonus Tolerance
- Worst Case Boundaries RFS
- Summary of Formulas
- Measurement and Gaging Methods
- Verifying Geometric Tolerances at MMC or LMC
- New Terms and Definitions
- Exercises

Module 3 – Form Controls

- Objectives
- Form Symbols
- Straightness Applied to a Surface
- ASME Size Controls Straightness
- ASME Straightness of a Line Element
- ISO Straightness of a Line Element
- Inspection Guidelines
- Flatness
- ASME Size Controls Flatness
- ASME Flatness Interpretation
- ISO Flatness Interpretation
- Inspection Guidelines
- Circularity (Roundness)
- ASME Size Controls Circularity
- ASME Circularity Interpretation
- ISO Circularity Interpretation
- Inspection Guidelines
- Cylindricity
- ASME Size Controls Cylindricity
- ASME Cylindricity Interpretation
- ISO Cylindricity Interpretation
- Inspection Guidelines
- New Terms and Definitions
- Exercises

Module 4 – Datums

- Objectives
- Datums Establish Relationships
- Typical Types of Datums
- Planar Datums
- Datum Reference Frame
- Axis and Center Plane Datums
- Datums and Datum Features
- Datum Targets
- Datum Target Areas, Lines, and Points
- Datum Target Reference Frame
- New Terms and Definitions
- Exercises

Module 5 – Profile Tolerances

- Objectives
- Profile Symbols
- Profile Can Control SLOF
- Typical Profile of a Surface Tolerancing
- More Profile Facts
- Equal Bilateral Tolerance Default
- Default Profile Coverage
- Delimited Profile Coverage
- All Around Profile Coverage
- Locating Surface with Profile
- Coplanarity with Profile
- Inspection Guidelines
- Profile of a Line
- New Terms and Definitions
- Exercises

Module 6 – Orientation Tolerances

- Objectives
- Orientation Symbols
- Perpendicularity Applications
- More Orientation Facts
- Definitions and Tolerance Zones
- Implied 90 degree Angles
- Perpendicularity Misinterpretation
- Perpendicularity of a Surface
- Axis Perpendicularity RFS and MMC
- Centerplane Perpendicularity RFS and MMC
- Inspection of Perpendicularity
- Unclear Angular Specification
- Angularity of a Surface
- Inspection of Angularity
- Implied Parallelism
- Parallelism via Rule #1
- Parallelism of a Surface

- Inspection of Parallelism
- New Terms and Definitions
- Exercises

Module 7 – Location Tolerances

- Objectives
- Location Symbols
- Definition of Location Controls
- Position Tolerance Facts
- Position Misinterpretation without GD&T
- Advantages of Position Tolerancing
- Position of an Axis RFS and MMC
- Position of a Centerplane RFS and MMC
- Inspection Guidelines and Options
- Concentricity
- Concentricity Tolerance Facts
- ASME Concentricity Interpretation vs. Position
- Inspection of Concentricity
- ISO Concentricity Options and Interpretation
- New Terms and Definitions
- Exercises

Module 8 – Runout Tolerances

- Objectives
- Runout Symbols
- Runout Tolerance Zones
- Datum Referencing Options
- Circular Runout on a Diameter
- Maximum Allowable Position Error
- Maximum Allowable Circularity Error
- Circular Runout Interpretation
- Inspection Guidelines
- Total Runout on a Diameter
- Total Runout Interpretation
- Inspection Guidelines
- Circular vs. Total Runout
- Application Guidelines
- New Terms and Definitions
- Exercises
- Course Summary and Q&A
- Post Assessment
- Course Evaluation