GD&T Advanced Webinar Outline

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

Module 1 - Introduction

- Pre-Assessment
- Objectives
- GD&T Application Guidelines
- Basic Steps to Functional Dimensioning and Tolerancing
- Appropriate Mix of Coordinate and Geometric Tolerances
- Dimensional Mapping
- 10 Cs of Applying Dimensioning and Tolerancing
- Customer Requirements
- Context
- Contact
- Critical Characteristics
- Constraint
- Consistency
- Connections
- Calculations
- Consensus
- Confirmation
- Mini-Stapler Case Study
- 5 Basic Steps to Functional Dimensioning and Tolerancing
- Coordinate System Weaknesses
- Square vs. Cylindrical Tolerance Zones
- Ambiguous Inspection Setups
- Introduction to GD&T
- Conversion from Coordinate to GD&T
- Inspection of Position with MMC
- New Terms and Definitions
- Exercises

Module 2 – GD&T Review (Optional)

- Objectives
- GD&T Symbols
- ISO Form Tolerances
- ASME Form Tolerances
- Straightness of an Axis MMC
- ASME vs. ISO Form Tolerancing Practices
- Datum Terms and Definitions
- Datum Shift
- Datum Feature Symbols
- Profile Tolerances
- Orientation Tolerances
- Position Tolerances
- Concentricity and Symmetry Tolerances
- Runout Tolerances
- Review of New Terms and Definitions
- Exercises

Module 3 – Datums

- Objectives
- Typical Types of Datums
- Model-Based Datum Features
- Datum Reference Frame
- Axis and Center Plane Datums
- Axis and Center Plane Datums RFS
- Axis and Center Plane Datums MMC
- Explicit Gage Feature Values
- Datum Shift
- Shift Applied to Patterns
- Datums and Datum Features
- Explicit Degrees of Freedom
- Datum Targets
- Datum Target Areas, Lines, and Points
- Datum Target Reference Frame
- Datums Terms and Dimensions
- Datums and Dimensions
- Datum Selection Guidelines
- Functional Part-to-Part Analysis
- Manufacturing Datum Selection
- Local Datum References
- Modifier Application Guidelines
- Datum Sequence
- New Terms and Definitions
- Exercises

Module 4 – Advanced Position Applications

- Objectives
- ANSI/ASME Limits and Fits
- ISO Limits and Fits
- ASME Symbols and Modifiers
- ISO Symbols and Modifiers
- Checking Limits of Size
- Zero Tolerance at MMC
- Rule #1 Exceptions
- ASME vs. ISO Size-to-Form Relationships
- Implied Basic Dimensions
- Boundaries and Bonus Tolerance
- Summary of Formulas
- Independency Modifier with Straightness

- Per Unit Straightness
- Straightness of an Axis MMC
- Profile to Define Features of Size
- Unequal and Unilateral Profile Tolerances
- Profile Coverage Area Options
- Composite Profile Tolerances
- Dynamic Profile Tolerance
- Profile with Position
- Profile Design for Inspection Strategies
- Perpendicularity of a Surface with 2 Datums
- Illegal Use of Parallelism
- Tangent Plane Modifier
- Tolerancing Options w/ Same Datums & Dimensions Using GD&T
- Comparison of Coaxiality Options
- Comparison of Circular vs. Total Runout
- Analysis of Zero Tolerance at MMC
- Position LMC
- Typical Uses for the LMC Modifier
- Basis for Projected Tolerances
- Projected Tolerance Examples
- Projected Tolerance Calculations
- Fixed Fastener Designs
- Fixed Fastener Calculations
- Floating Fastener Designs
- Floating Fastener Calculations
- Threads, Gears, Splines
- Pitch Diameter Rule
- Best Practices on Thread Specifications
- Composite Tolerance Examples
- New Terms and Definitions
- Exercises
- Course Summary and Q&A
- Post Assessment
- Course Evaluation

Appendices

- Mini-Stapler Assembly
- Test Cap Assembly
- Speed Pickup Assembly
- Application Flowcharts and Guidelines
- Tolerance Stack Formulas
- Glossary