

GD&T Basic/Fundamentals Course Outline

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

Module 1 - Introduction

- Pre-Assessment
- Administrative Items
- Objectives
- Elements of Engineering Drawings
- Impact of Engineering Drawings
- Cost of Drawing Errors
- 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
- Coordinate System Weaknesses
- Square vs. Cylindrical Tolerance Zones
- Ambiguous Inspection Setups
- Introduction to GD&T
- Conversion from Coordinate to GD&T
- 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
- Features and Features of Size
- Identifying Features of Size
- Actual Size and Actual Local Size
- Actual Mating Envelope
- Material Conditions
- Maximum Material Condition
- Least Material Condition
- Symbols and Modifiers
- MMC Modifier Application
- LMC Modifier Application
- Bonus Tolerance Summary
- RFS Material Condition
- MMC, LMC, and RFS Examples

- Dimension Origin Symbol
- Rule #1 – Size Controls Form
- Exceptions to Rule #1
- Rule #2 – RFS Implied
- Threads, Gears, and Splines
- Basic Dimensions
- Boundaries and Bonus Tolerance
- Virtual and Resultant Conditions
- Worst Case Boundaries RFS
- Summary of Formulas
- Measurement and Gaging Methods
- New Terms and Definitions
- Exercises

Module 3 – Form Controls

- Objectives
- Form Symbols
- Straightness Applied to a Surface
- Size Controls Straightness
- Straightness of a Line Element
- Application Guidelines
- Inspection Guidelines
- Straightness Applied to a FOS
- Straightness of and Axis MMC
- Application Guidelines
- Inspection Guidelines
- Flatness
- Size Controls Flatness
- Flatness Interpretation
- Application Guidelines
- Inspection Guidelines
- Circularity
- Size Controls Circularity
- Circularity Interpretation
- Application Guidelines
- Inspection Guidelines
- Cylindricity
- Size Controls Cylindricity
- Cylindricity Interpretation
- Application Guidelines
- Inspection Guidelines
- New Terms and Definitions
- Exercises

Module 4 – Datums

- Objectives
- Datums Establish Relationships
- Planar Datums
- Datum Terms and Definitions
- Datum Reference Frame
- Special Datum References
- Axis and Centerplane Datums
- Axis and Centerplane Datums RFS
- Axis and Centerplane Datums MMC
- Datum Shift
- Datums and Datum Features
- Datum Targets
- Datum Target Areas, Lines, and Points
- Datum Target Reference Frame
- Datums and Dimensions
- Datum Application Guidelines
- Logical Datum Selection
- Functional Analysis
- Functional Datum Selection
- Manufacturing Datum Selection
- Modifier Application Guidelines
- Datum Sequence
- New Terms and Definitions
- Exercises

Module 5 – Profile Tolerances

- Objectives
- Profile Symbols
- Profile Controls SLOF
- Profile of a Surface Tolerancing
- Orientation Using Profile
- More Profile Facts
- Equal Bilateral Tolerance Default
- Unequal and Unilateral Tolerancing
- Default Profile Coverage
- Delimited Profile Coverage
- All Around Profile Coverage
- Locating Surface with Profile
- Coplanarity with Profile
- Profile on Offset Surfaces
- Composite Profile Tolerance
- Profile with Position
- Profile on a Conical Feature
- Application Guidelines
- Inspection Guidelines
- Profile of a Line
- Refinement Using Profile of a Line

- New Terms and Definitions
- Exercises

Module 6 – Orientation Tolerances

- Objectives
- Orientation Symbols
- More Orientation Facts
- Definitions and Tolerance Zones
- Implied 90 degree Angles
- Perpendicularity Misinterpretation
- Dimension Origin Symbol
- Perpendicularity of a Surface
- Perpendicularity with 2 Datums
- Axis Perpendicularity RFS and MMC
- Centerplane Perpendicularity RFS and MMC
- Inspection of Perpendicularity
- Unclear Angular Specification
- Dimension Origin Symbol
- Angularity of a Surface
- Angularity with 2 Datums
- Axis Angularity RFS and MMC
- Angularity Refinement
- Inspection of Angularity
- Implied Parallelism
- Parallelism via Rule #1
- Parallelism of a Surface
- Axis Parallelism RFS and MMC
- Inspection of Parallelism
- Tangent Plane Modifier
- Application Guidelines
- New Terms and Definitions
- Exercises

Module 7 – Location Tolerances

- Objectives
- Location Symbols
- Definition of Location Controls
- Location Tolerance Zones
- Location Tolerance Facts
- Implied Basic Relationships
- Position Misinterpretation
- Advantages of Position Tolerancing
- Position of an Axis RFS and MMC
- Position of a Centerplane RFS and MMC
- Inspection Guidelines and Options
- Application Guidelines
- Concentricity
- Concentricity Tolerance Facts

- Concentricity Interpretation
- Inspection of Concentricity
- Application Guidelines
- Symmetry
- Symmetry Tolerance Facts
- Symmetry Interpretation
- Inspection Guidelines
- Application Guidelines
- New Terms and Definitions
- Exercises
- Application Guidelines
- New Terms and Definitions
- Exercises
- Course Summary and Q&A
- Post Assessment
- Course Evaluation

Module 8 – Advanced Position Applications

- Objectives
- Zero Tolerance at MMC
- Analysis of Zero Tolerance at MMC
- Position LMC
- Basis for Projected Tolerances
- Projected Tolerance Examples
- Projected Tolerance Calculations
- Fixed Fastener Designs
- Fixed Fastener Calculations
- Floating Fastener Designs
- Floating Fastener Calculations
- Advance Position Applications
- Composite Tolerance Examples
- Tolerance Stacks with Position
- Stacks with Position RFS, MMC, and LMC
- New Terms and Definitions
- Exercises

Module 9 – Runout Tolerances

- Objectives
- Runout Symbols
- Definitions of Runout Controls
- Runout Tolerance Zones
- Datum Requirements
- Datum Referencing Options
- Circular Runout on a Diameter
- Circular Runout on a Face
- Circular Runout on a Revolved Irregular or Tapered Surface
- Circular Runout Interpretation
- Inspection Guidelines
- Total Runout on a Diameter
- Total Runout on a Face
- Total Runout Interpretation
- Inspection Guidelines
- Circular vs. Total Runout