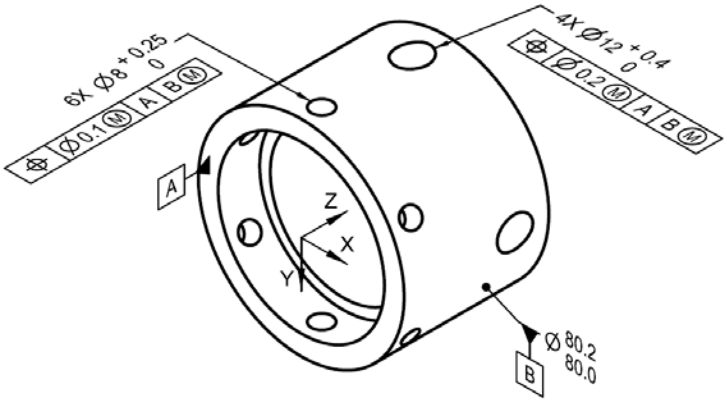

Overview and Purpose

Light on lecture, heavy on hands-on practice, this introductory 2 day workshop allows participants to practice interpreting dimensioning and tolerancing, including Geometric Dimensioning and Tolerancing (GD&T). Participants learn how to properly interpret datums, dimensions, and geometric tolerances from their own product and process segments and families.

Participants are encouraged to bring in drawings; each workshop is tailored to the drawings and projects the students are currently working on.

Who Should Attend

This introductory course is intended for personnel who are not already familiar with the GD&T principles and practices as related in ASME Y14.5M and ISO 1101 and related standards. The course is designed for junior engineering, design, manufacturing, quality, supply chain, DFM, or other related personnel who need to better understand the interpretation of dimensions, tolerances, and GD&T on existing designs.

Prerequisite: None

Here's What You'll Do

- Interpret datums, dimensions, and tolerances on existing drawings.
- Identify and correct dimensioning and tolerancing errors on existing designs.
- Identify and interpret the most common geometric tolerances of form, profile, orientation, location, and runout.

About Your Instructor

Mr. Bauer, an ASME Certified Senior GD&T Professional and Certified Six Sigma Master Black Belt (CSSMBB) who holds a Master of Science in Industrial Operations and is fluent in Spanish, serves as president and principal consultant with ITR. He also holds Lean Certification through the University of Michigan College of Engineering. With experience supporting over 450+ different product design and manufacturing launch programs in the past 25+ years in North America, Europe, and Asia, he specializes in providing training and engineering services in the areas of Dimensional Management, Six Sigma, Design for Lean Six Sigma, design, engineering, quality, performance and process improvement, and productivity. He has personally serviced over 250 different customers in the automotive, aerospace/aviation, robotics, consumer products/durable goods, telecommunications, defense, health care, medical devices, agricultural and construction, power transmission and distribution, oil and gas and energy industries. Dan specializes in training and implementation of AS9145, AS13000, AS13002, AS13003, AS13004, and AS13006. He can be reached at dan.bauer@itr-sa.com or by phone at (915) 867-4500.

Since 2007 Mr. Bauer has provided training, coaching, and consulting to Schlumberger engineering, design, manufacturing, quality, and supply chain personnel in various areas including Geometric Dimensioning and Tolerancing (GD&T), Tolerance Stack Analysis (TSA), and Design for Manufacturing and Design for Assembly (DFM/DFA).

GD&T Introduction Outline

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

Module 1 - Introduction
- Pre-Assessment
- Administrative Items
- Objectives
- 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
- Introduction 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
- Material Conditions
- Maximum Material Condition
- Least Material Condition
- Symbols and Modifiers
- MMC Modifier Application
- LMC Modifier Application
- Bonus Tolerance Summary
- RFS Material Condition
- Rule #1 – Size Controls Form
- Exceptions to Rule #1
- Rule #2 – RFS Implied
- Basic Dimensions
- Measurement and Gaging Methods

Module 3 – Form Controls
- Objectives
- Form Symbols
- Straightness Applied to a Surface
- Size Controls Straightness
- Straightness of a Line Element
- Inspection Guidelines
- Flatness
- Size Controls Flatness
- Flatness Interpretation
- Inspection Guidelines
- Circularity
- Size Controls Circularity
- Circularity Interpretation
- Inspection Guidelines
- Cylindricity
- Size Controls Cylindricity
- Cylindricity Interpretation
- Inspection Guidelines

Module 4 – Datums
- Objectives
- Datums Establish Relationships
- Planar Datums
- Datum Terms and Definitions
- Datum Reference Frame
- Axis and Center plane Datums
- Datums and Datum Features
- Datum Targets
- Datum Sequence

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
- 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
- Perpendicularity of a Surface
- Axis Perpendicularity RFS and MMC
- Centerplane Perpendicularity RFS and MMC
- Inspection of Perpendicularity
- Angularity of a Surface
- Axis Angularity RFS and MMC
- Inspection of Angularity
- Implied Parallelism
- Parallelism of a Surface
- Axis Parallelism RFS and MMC
- Inspection of Parallelism

Module 7 – Location Tolerances
- Objectives
- Location Symbols
- Definition of Location Controls
- Location Tolerance Zones
- Location Tolerance Facts
- Implied Basic Relationships
- 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
- Symmetry
- Symmetry Tolerance Facts
- Symmetry Interpretation
- Inspection Guidelines

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 Interpretation
- Inspection Guidelines
- Total Runout on a Diameter
- Total Runout Interpretation
- Inspection Guidelines
- Circular vs. Total Runout
- Course Summary and Q&A
- Post Assessment
- Course Evaluation