

ETI's Fundamentals of GD&T Video Training DVD Content

Based on ASME Y14.5M-1994

The Fundamentals of GD&T Complete Course contains 28 lessons on 8 DVDs.
The Overview Set is comprised of DVDs 1, 2, 4, and 5. All DVDs can be ordered individually.

DVD 1: Terms, Symbols, Modifiers and Rules - Instructor's Segment, Engineering Drawings, Benefits of GD&T, Key Terms, Modifiers & Symbols, Rules of GD&T, Concepts of GD&T

DVD 2: Form Controls and Planar Datums - Flatness Control, Straightness Control, Circularity Control, Cylindricity Control, Planar Datum System

DVD 3: Datum Targets and Size Datums - Datum Targets, Feature of Size Datum Specifications at RFS, Feature of Size Datum Specifications at MMC

DVD 4: Orientation Controls - Perpendicularity, Angularity Control, Parallelism Control

DVD 5: Tolerance of Position, Part 1 - Tolerance of Position Fundamentals, Tolerance of Position Applications

DVD 6: Tolerance of Position, Part 2 - Cartoon Gages, Tolerance of Position – Special Applications, Calculating Part Distances, Fastener Formulas

DVD 7: Concentricity, Symmetry and Runout - Concentricity Control, Symmetry Control, Circular Runout, Total Runout

DVD 8: Profile Controls - Introduction to Profile, Profile of a Surface, Profile of a Line, Series Conclusion

DVD 1: Terms, Symbols, Modifiers, and Rules (Also included in Overview DVD Set)



Instructor's Segment

Lesson 1 – Engineering Drawings

Engineering Drawings
How Drawings Communicate
Consequences of Drawing Errors
Dimensions and Tolerances
Limit and Plus-Minus Tolerances
Metric Dimension Conventions
Interpreting Dimensional Limits
ASME Y14.5M-1994 Standard

Lesson 2 – Benefits of GD&T

Coordinate Tolerancing
Shortcomings of Coordinate Tolerancing
Geometric Tolerancing
Benefits of GD&T
How GD&T Eliminates the Shortcomings of Coordinate Tolerancing
Geometric Tolerancing vs. Coordinate Tolerancing

Lesson 3 – Key Terms

Feature
Feature of Size
Actual Local Size & Actual Mating Envelope
Maximum Material Condition & Least Material Condition
Regardless of Feature Size

Lesson 4 – Modifiers & Symbols

Modifiers
Radius and Controlled Radius
Geometric Characteristic Symbols
Five Types of Tolerances
Parts of the Feature Control Frame

Lesson 5 – Rules of GD&T

Rule #1
Using Rule #1
How to Override Rule #1
Rule #1 and the Interrelationship Between Features of Size
Exceptions to Rule #1
Go and No-Go Gages
Rule #2

Lesson 6 – Concepts of GD&T

Basic Dimensions
Inner and Outer Boundary
Worst-Case Boundary and Virtual Condition
Bonus Tolerance



www.etinews.com

Call 1-800-886-0909 for more information. Visit our website for a video sample or to order online.

DVD 2: Form Controls and Planar Datums (Also included in Overview DVD Set)



Lesson 7 – Flatness Control

Definition of Flatness
Rule #1 as an Indirect Flatness Control
Flatness Control Tolerance Zone
Legal Specification Test for Flatness
Verifying Flatness

Lesson 8 – The Straightness Control

Definition of Straightness
Tolerance Zone for Straightness Applied to a Surface
Rule #1 as an Indirect Straightness Control
Legal Specification Test for Straightness Applied to a Surface
Verifying Straightness Applied to Surface Elements
Straightness Applications
Determining if Straightness is Applied to a Surface or an Axis / Centerplane
Rule #1 as an Indirect Straightness Control (Axis)
Straightness Applications with Bonus
Legal Specification Test for Straightness Applied to an Axis / Centerplane
Verifying Straightness Applied to an Axis / Centerplane

Lesson 9 – The Circularity Control

Definition of Circularity
Tolerance Zone for Circularity
Rule #1 as an Indirect Circularity Control
Circularity Control Application
Legal Specification Test for Circularity
Verifying Circularity

Lesson 10 – The Cylindricity Control

Definition of Cylindricity
Tolerance Zone for Cylindricity
Rule #1 as an Indirect Cylindricity Control
Cylindricity Control Application
Legal Specification Test for Cylindricity
Verifying Cylindricity

Lesson 11 – The Planar Datum System

Implied Datums
Eliminating the Shortcomings of Implied Datums
Key Terms of the Datum System
Datum Symbols
Assigning Datum Features
Six Degrees of Freedom
Datum Reference Frame
3-2-1 Rule
Determining Which Dimensions are Measured from the Datum Reference Frame
Types of Datum Features: Inclined Datums, Multiple Datum Reference Frames, Coplanar Datum Features

DVD 3: Datum Targets and Size Datums



Lesson 12 – Datum Targets

Datum Targets
Datum Target Symbol
Datum Target Points
Datum Target Lines
Datum Target Areas
Datum Target Applications

Lesson 13 – Feature of Size Datum Specifications at RFS

Referencing Datum Features at MMC
MMC Datum References Simulated at Virtual Condition
Datum Shift
Datum Feature Simulators for External / Internal
Feature of Size Datum Features (MMC Primary)
Datum Feature Simulators for Datum Axis (MMC Secondary)

Lesson 14 – Feature of Size Datum Specifications at MMC

Referencing Datum Features at MMC
MMC Datum References Simulated at Virtual Condition
Datum Shift
Datum Feature Simulators for External / Internal Feature of Size Datum Features (MMC Primary)
Datum Feature Simulators for Datum Axis (MMC Secondary)
Datum Axis from a Pattern of Holes as a Datum Feature
Effects of Datum Sequence



DVD 4: Orientation Controls (Also included in Overview DVD Set)



Lesson 15 – The Perpendicularity Control

Implied 90° Angles
Common Tolerance Zones
Perpendicularity Applied to a Planar Surface
Multiple Datum References with Perpendicularity
Perpendicularity Applied to a Centerplane
Indirect Perpendicularity Controls
Legal Specification Test
Verifying Perpendicularity

Lesson 17 – The Parallelism Control

Implied Parallelism
Parallelism Applied to a Surface
Common Tolerance Zones
Parallelism Applied to a Feature of Size
Tangent Plane Modifier
Indirect Parallelism Controls
Legal Specification Test
Verifying Parallelism

Lesson 16 – The Angularity Control

Angularity Applied to a Surface
Common Tolerance Zones
Angularity Applied to a Feature of Size
Indirect Angularity Controls
Legal Specification Test
Verifying Angularity

DVD 5: Tolerance of Position, Part 1 (Also included in Overview DVD Set)



Lesson 18 – Tolerance of Position Fundamentals

True Position and Tolerance of Position
Implied Basic Relationships
Six Advantages of Tolerance of Position
Part Relationships Controlled with Tolerance of Position
Modifier Usage
Tolerance of Position Theories

Lesson 19 – Tolerance of Position Applications

Three Conditions of Tolerance of Position (RFS) Application
Common Tolerance Zones for an RFS Application
Hole Location Controlled with Tolerance of Position at RFS
Inspecting Tolerance of Position at RFS
Three Conditions of a Tolerance of Position MMC Application
Common Tolerance Zones for an MMC Application
Hole Location Controlled with Tolerance of Position at MMC
Location of a Hole Pattern Controlled with Tolerance of Position at MMC
Coaxial Diameter Location Controlled with Tolerance of Position at MMC
Legal Specification Test

DVD 6: Tolerance of Position, Part 2



Lesson 20 – Cartoon Gages

Functional Gage Definition
Cartoon Gages
How to Draw a Cartoon Gage

Lesson 22 – Calculating Part Distances

Tolerance Stacks Using Tolerance of Position at RFS
Tolerance Stacks Using Tolerance of Position at MMC
An Additional Stack Example

Lesson 21 – Special Applications

Locating Non-Parallel Holes with Tolerance of Position
Bi-Directional Tolerance of Position Control
Locating an Elongated Hole with Tolerance of Position
Projected Tolerance Zone
Controlling a Symmetrical Relationship with Tolerance of Position
Using the Least Material Condition Modifier
Using Tolerance of Position to Control the Spacing and Orientation of a Pattern of Holes
Multiple Single-Segment Tolerance of Position Controls
Tolerance of Position with Zero Tolerance at MMC

Lesson 23 – Fastener Formulas

Fixed Fastener Assembly
Fixed Fastener Formula
Calculating Tolerance of Position Tolerance Values for Fixed Fasteners
Floating Fastener Assembly
Floating Fastener Formula
Calculating Tolerance of Position Tolerance Values for Floating Fasteners
A Warning About Using the Fixed and Floating Fastener Formulas

DVD 7: Concentricity, Symmetry, and Runout



Lesson 24 – The Concentricity Control

Definitions
Concentricity Tolerance Zone
Concentricity Application
Differences Between Concentricity, Runout, and Tolerance of Position RFS
Legal Specification Test
Verifying Concentricity

Lesson 25 – The Symmetry Control

Symmetry and the Symmetry Control
Symmetry Tolerance Zone
Symmetry Application
Differences Between Symmetry and Tolerance of Position RFS
Legal Specification Test
Verifying Symmetry

Lesson 26 – Circular Runout

Three Ways to Establish a Datum Axis
Definitions
Circular Runout as a Composite Control
Circular Runout Applications
Legal Specification Test
Verifying Circular Runout

Lesson 27 – Total Runout

Definition of Total Runout
Total Runout Tolerance Zone
Total Runout as a Composite Control
Total Runout Application
Verifying Total Runout
Comparison Between Circular and Total Runout
Runout Calculations

DVD 8: Profile Controls



Lesson 28 – Introduction to Profile

Definitions
Effects of Using Profile With and Without Datum References
Four Part Characteristics that can be Controlled with Profile
Profile Tolerance Zones
Tolerance Zone Coverage
Between Symbol
All Around Symbol
Advantages of Profile Controls

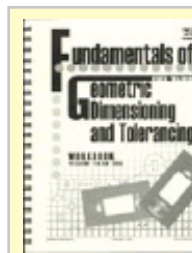
Lesson 30 – Profile of a Line

Definition of Profile of a Line
Shape and Size of the Tolerance Zone
Using Profile of a Line in a Multiple Single-Segment Control
Profile of a Line with Coordinate Tolerances
Verifying Profile of a Line
Profile Calculations

Lesson 29 – Profile of a Surface

Definition of Profile of a Surface
Profile of a Surface – Location Example
Profile of a Surface – Polygon Example
Conical Feature Application
Profile of a Surface Applied to Coplanar Surfaces
Using Profile of a Surface in a Multiple Single-Segment Control
Legal Specification Test
Verifying Profile of a Surface

Series Conclusion



The video training series is cross-referenced with ETI's Video Workbook.

Reinforce what you learn with exercises, quizzes, diagrams, tips, charts, and key points from the videos.

Product 4010 - GD&T Video Training Full Course - 8-DVD Set - ASME Y14.5M-1994

The full course contains 28 lessons covering the basic rules, definitions, and concepts of GD&T. The total running time is approximately 14 hours; however, we recommend that you allocate 20-24 hours to get the most out of this program.

Product 4014 - GD&T Video Training Overview Course - 4-DVD Set - ASME Y14.5M-1994

The overview course contains 16 lessons covering the basic rules, definitions, and concepts of GD&T. The total running time is approximately 7 hours; however, we recommend that you allocate 15-17 hours to get the most out of this program.

Product 4015 - GD&T Video Training Individual Module - 1-DVD - ASME Y14.5M-1994

The individual video training modules are an economical means of reviewing individual topics that need more study. Each DVD contains 2-6 lessons covering a variety of GD&T topics. Running time for each DVD varies.

