GD&T Fundamentals

Learn the Simplified Framework Behind the Tolerances on your Prints

Section 1: Course Introduction
- What are the GD&T Standards
- Why Use GD&T

Section 2: GD&T Foundations
- Feature Control Frame
- Feature of Size Conditions
- Size Tolerance and Rule #1

Section 3: Datums
- Datums Introduction
- Primary Datum Controls
- DRF - Orthogonal Surfaces
- DRF - FOS Datums
- Datum Targets & Partial Datums

Section 4: Position
- GD&T Framework
- Position Tolerance
- Position Vs Coordinate Dimensions

Section 5: Modifiers
- Regardless of Feature Size (RFS)
- Maximum Material Condition (MMC)
- Least Material Condition (LMC)

Section 6: Profile Controls
- Surface Profile
- Profile of a Line

Section 7: Orientation Controls
- Perpendicularity
- Parallelism
- Angularity

Section 8: Form Controls
- Flatness (Surface & DMP)
- Straightness (Surface & DML)
- Circularity
- Cylindricity

Section 9: Runout Controls
- Circular Runout
- Total Runout

Section 10: Concentricity and Symmetry
- Concentricity
- Symmetry

Section 11: Conclusion & Exam
- Course Conclusion
- Final Exam

We promise, once you take our training & understand our GD&T framework, you will drastically improve the way you work with your drawings. We don't want you to just understand theory – we want you to apply what you learn!

GD&T Basics – Engineer Essentials ©

1-800-495-0991 info@gdandtbasics.com www.GDandTBasics.com

GD&T BASICS