



Basic Geometric Dimensioning & Tolerancing Reading

(Onsite or Virtual – 1 Days - 8 hours total, 1 – 12 Participants)

Training Description:

Geometric Dimensioning & Tolerancing (GD&T) is the symbolic, mathematical language used on drawings to clearly define design intent, describe part geometry, and define the allowable variation to the size, form, orientation, and feature location of a part based on how it functions.

The instructor will utilize a projected and visual-aided presentation of the basic GD&T symbols and their definitions and uses based on the ASME Y14.5-2009 standard. The optional use of client-provided drawing examples can be introduced to provide examples of GD&T use to define features and features of size of parts recognizable to students.

Training Objective:

Students will get a thorough introduction to the ASME Y14.5-2009 standard. The principal objective is to give those who are required to interpret blueprints the skills to understand the basics of geometric dimensioning and tolerancing.

Target Audience:

Student candidates are recommended to include any individual involved in interpreting drawings, manufacturing, inspecting, and purchasing parts and assemblies or serve as a good refresher for engineering staff.

Recommended Prerequisites:

Basic math and basic understanding of engineering drawings.

Blueprint reading skills and working understanding of common measuring devices suggested.

Skill Attainment:

- A copy of the course presentation.
- Exercise booklet.
- A copy of “The Ultimate GD&T Pocket Guide” ASME &14.5-2009 edition, published by SAE International.

These skills are transferable within the company, industry and are highly desirable by any manufacturer.



Agenda:

Basic topics discussed are:

- Introduction to ASME Y14.5-2009
- Basic Dimensions
- Coordinate Tolerancing vs. GD&T
- General GD&T Symbols
- Defining Features and Features of Size
- Material Conditions – Maximum, Least, & Regardless of Feature Size (MMC, LMC, RFS)
- Modifiers & Feature Control Frames
- GD&T Rule 1 (Limits of Size) & Rule 2 (Modifier Symbols)
- Virtual Condition
- Bonus Tolerance
- Form Tolerances & Their Measurement Methods (Straightness, Flatness, Circularity, Cylindricity)
- The Datum System
- Datum Reference Frame - Degrees of Freedom
- Orientation Tolerances & Their Measurement Methods (Parallelism, Perpendicularity, Angularity)
- Location Tolerances & Their Measurement Methods (Position, Concentricity, Symmetry)
- Runout Tolerances & Their Measurement Methods (Circular Runout, Total Runout)
- Profile Tolerances & Their Measurement Methods (Profile of a Line, Profile of a Surface)
- Client drawing review/ Q&A (optional)