Fundamentals of GD&T

In today's modern and technically advanced design, engineering and manufacturing world, effective communication is required to ensure the design and manufacture of successful products. Success oriented organizations, which require accurate and common lines of communications between engineering, design, manufacturing and quality consider knowledge of GD&T as an essential skill for design professionals.

The best design in the world ***is worthless if it cannot be produced!***

In this Course we teach you the GD&T as prescribed in the ASME Y14.5-2009 Standard which is the need of the hour in Industries.

You'll get an in-depth explanation of geometric symbols, including each symbol's requirements, tolerance zones, and limitations. The class includes a comparison of GD&T to coordinate tolerancing; an explanation of tolerance zones; Rules #1 and #2; form and orientation controls; tolerance of position; runout and profile controls.

Course Agenda

Introduction

- GD&T importance
- Fundamental Rules
- Coordinate tolerancing and its Short Comings
- What geometric tolerancing is, and its benefits
- Coordinate tolerancing Vs Geometric tolerancing

Key Terms Used in GD&T

- Size and actual local size, actual mating envelope, axis, center plane
- Complex feature, feature, feature of size, irregular feature of size
- Least and maximum material condition
- Non-opposed, opposed, partially opposed, pattern
- Regardless of feature size
- Related and unrelated actual mating envelope
- Feature control frame and placement

GD&T Symbols & Modifiers

- Geometric characteristic symbols
- Geometric modifying symbols

GD&T Rules

- Rule #1: applied to a feature of size, exceptions, overriding, inspecting
- Independency concept,
- Rule #2: All applicable rule

GD&T Concepts

- Virtual condition, calculations, and worst-case boundary
- Bonus tolerance concept and calculations
- Verification principles for a virtual condition boundary

Straightness Tolerance

- Derived median line, tolerance zones,
- Rule #1 as a straightness control
- Applied to a surface and a feature of size
- Bonus tolerance (at MMC)

Flatness Tolerance

- Derived median plane, tolerance zones
- Rule #1 as a flatness control
- Applied to a planar surface and feature of size
- Bonus tolerance (at MMC)

Circularity Tolerance

- Tolerance zones
- Rule #1 as a circularity control

Cylindricity Tolerance

- Tolerance zones
- Rule #1 as a cylindricity control

The Datum System

- Implied datums, benefits, terminology
- Datum reference frame and symbol

- Six degrees of freedom
- Coplanar datum feature
- Multiple datum reference frames

Datum Targets

- The datum target symbols, usage, requirements
- Point datum target, line datum target, datum target simulators, movable datum targets
- Datum target applications

Size Datum Features: RMB & MMB

- Terms, methods to specify a feature of size as a datum feature
- Effects of a datum feature (RMB)
- Maximum material boundary (MMB) and effects
- Datum shift, datum sequence, datum feature simulators

Perpendicularity Tolerance

- Perpendicularity and perpendicularity tolerance
- Common tolerance zones
- Modifiers used with perpendicularity tolerance

Parallelism Tolerance

- Parallelism and parallelism tolerance
- Common tolerance zones
- Modifiers used with parallelism tolerance

Angularity Tolerance

- Angularity, angularity tolerance
- Common tolerance zones
- Modifiers used in angularity tolerances

Position Tolerance Introduction

- True position and common tolerance zones
- Implied relationships and advantages
- Conditions of RFS and MMC applications
- Projected tolerance zone
- Zero tolerance at MMC

- Floating fastener and fixed fastener assemblies, formulas and limitations
- Calculating position tolerance values

Circular & Total & Runout Tolerances

- Circular runout and total runout tolerance,
- Interpretation, and comparison
- Tolerance zone shape

Concentricity & Symmetry Tolerances

- Concentricity and Symmetry tolerance
- Tolerance zone shapes

Profile Tolerances: Introduction

- Profile and true profile
- Part characteristics affected by profile tolerances
- Effects of datum references