



IFS ACADEMY

Training For The Future!!

Geometric Dimensioning & Tolerancing (GD&T)

Course Curriculum (Duration: 50 Hrs.)

Chapter 1: What is GD&T

- a. Review of traditional dimensioning
- b. What is GD&T?
- c. Use of GD&T in Drawings
- d. Technical Standards
- e. GD&T vs CD&T (Coordinate Dimensioning & Tolerancing)
- f. ASME Y14.5-2009

Chapter 2: Drawing

- a. What is Drawing
- b. Important areas of Drawing
- c. Dimensioning Types
- d. Tolerances & its types

Chapter 3: Dimensioning System

- a. Types of Dimensions
- b. Difference and Benefits
- c. Tolerance Calculation
- d. Examples / Exercises
- e. Functional Dimensions

Chapter 4: GD&T Symbols

- a. Modifier Symbols: MMC, LMC & RFS
- b. When to use MMC, LMC & RFS modifier in Design
- c. Bonus Tolerance Calculation
- d. Part Features, Feature of Size, Non-Size Features
- e. Tolerance Zone

Chapter 5: Form Tolerances

- a. Form tolerances
- b. Straightness tolerances
- c. Circularity Tolerance
- d. Cylindricity Tolerance
- e. When to use Form Tolerance?
- f. Design recommendation on why to use form tolerances only when required to ensure successful design intent

Chapter 6: Datums

- a. What is datum?
- b. Datum Reference Frame (Cartesian Coordinate System)
- c. Datum Feature Selection, Functional Hierarchy
- d. Datum feature identification / placements and interpretation
- e. Datum plane, datum axis, datum center plane
- f. Physical datum feature simulators
- g. Select datum features according to the design intent

Chapter 7: Orientation Tolerances

- a. How it works? How to apply it?
- b. Perpendicularity tolerances
- c. Parallelism tolerances
- d. Angularity tolerances
- e. Applied to a planar feature
- f. Datums for Orientation Control

- g. Inspection methods to verify orientation tolerances

Chapter 8: Location Tolerances

8.1 Position Tolerance

- i. How it works? How to apply it?
- ii. Tolerance zone shapes
- iii. Position tolerance for cylindrical features on MMC, LMC and RFS basis
- iv. Detailed table calculations for position tolerance on MMC, LMC and RFS
- v. Datums for Position Control
- vi. Inspection methods to verify position tolerances
- vii. Class room exercise for position

8.2 Runout Tolerances

- i. How it works? How to apply it?
- ii. Circular Runout Tolerance
- iii. Total Run out Tolerance
- iv. Datums for Run out Controls
- v. Inspection methods to verify runout tolerances

8.3 Profile Tolerances

- i. How it works? How to apply it?
- ii. Profile of a Line Tolerance
- iii. Profile of a Surface Tolerance
- iv. Profile Tolerance Zone: Bi-lateral type
- v. Datums for Profile Controls
- vi. Class room exercise for profile

8.4 Concentricity & Symmetry Tolerances

- i. How it works? How to apply it?
- ii. Concentricity Tolerance
- iii. Symmetry Tolerance
- iv. Tolerance zone shapes
- v. Datums for concentricity & symmetry controls
- vi. Why to **avoid** using Concentricity and Symmetry for small and inexpensive part designs

M: +91-98228 49628 / +91-9922440102; T: +91-20-25430338

E: training@ifsacademy.org
