Workshop on Geometric Dimensioning & Tolerancing (GD&T – Basic)

Course Contents

❖ Introduction to GD&T (Session : 1)

- Review of Traditional Dimensioning
- Types of Dimensioning Systems
- Types of Tolerances & needs in part
- Tolerance Calculation
- Coordinate Dimensioning v/s Geometric Dimensioning
- GD&T : What, Why, When & How
- Types of Geometric Controls & Symbols
- Technical Standards for Dimensioning

❖ Important Concepts of GD&T (Session : 2)

- Part Features, Feature of Size, Non-Size Features
- Feature Control Frame (FCF)
- How to place & read FCF?
- Material Modifier Symbols : MMC, LMC & RFS
- Concepts of MMB, LMB & RMB
- Virtual Condition & Resultant Condition
- Bonus Tolerance Concept & Calculation
- Tolerance Zones
- Rule #1 or Taylor's Principle
- Rule #2

❖ Datums (Session : 3)

- Degree of Freedom (DOF)
- 3-2-1 Location Principle
- Importance of Part Location & Datum
- Datum Reference Frame (DRF)
- Datum Features
- Datum Feature Selection, Functional Hierarchy
- Datum Feature Identification / Placements and Interpretation
- Datum Plane, Datum Axis, Datum Centre Plane
- Physical Datum Feature Simulators
- Datum Median Plane
- Partial Datum Features

Workshop on Geometric Dimensioning & Tolerancing (GD&T – Basic)

❖ Form & Orientation Tolerances (Session : 4)

- Form tolerances – Flatness, Straightness, Circularity & Cylindricity
- Orientation tolerances – Parallelism, Perpendicularity & Angularity
- Tolerance zone shapes
- Straightness tolerances for line elements
- Straightness tolerances for cylindrical features
- Flatness tolerance for single planar feature
- When to use Form & Orientation Tolerance?
- Inspection methods to verify Form & Orientation tolerances

❖ Location Tolerances (Session : 5)

- Location tolerances – Position, Concentricity, Symmetry
- Tolerance zone shapes
- Position tolerance for cylindrical features on MMC, LMC and RFS
- Detailed table calculations for position tolerance on MMC, LMC and RFS
- Datums for Position Control
- Zero Position Tolerance at MMC
- When to use location Tolerance?
- Inspection methods to verify position tolerances

❖ Runout Tolerances (Session : 6)

- Runout tolerances – Circular Runout & Total Runout
- Tolerance zone shapes
- When to use Runout Tolerance?
- Inspection methods to verify runout tolerances

❖ Profile Tolerances (Session : 7)

- Profile tolerances – Profile of a line & Surface
- Tolerance zone shapes
- When to use profile Tolerance?
- Inspection methods to verify profile tolerances

❖ Summary & Test (Session : 8)

- Summary of all geometric symbols
- Explanation on various dimensional and modifier Symbols
- Explanation on below symbols:
  - Projected Tolerance Zone
  - Independency
  - Continuous feature

Framework Engineering, Pune

Workshop on Geometric Dimensioning & Tolerancing (GD&T – Basic)

- Controlled Radius
- Tangent Plane
- Free State
- Translation
- Statistical Tolerance

- Comparison of ASME & ISO Symbols
- Modification in ASME Y14.5 – 2018 Version

Note :

- ✓ Exercises throughout the workshop.
- ✓ Final tests on proficiency and application of GD&T.

- ✓ **Duration :** 3 Days (8 Hrs./Day)
- ✓ **Pre-requisite :** Basic understanding of part designing & drawings

Contact Details

Contact : Framework Engineering, Pune
Contact No. : 8905263772 (WhatsApp)
Email id : contact@framework-india.com