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

Course Contents

❖ **Introduction to GD&T (Session : 1)**
- Review of Traditional Dimensioning
- Types of Dimensioning Systems
- Types of Tolerances & needs in part
- Tolerance Calculations
- Co-ordinate Dimensioning Vs 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 read & place 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
- Exceptions to Rule #1
- Rule #2
- Actual Minimum Material Envelope vs. Actual Mating Envelope
- General Rules for Tolerances

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

Framework Engineering, Pune

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

- Coaxial Datums
- Co-planar Datums
- Inclined Datum Features
- Curved Surface as a Datum Feature
- Conical Datum Features
- Datum Targets : Points, Line, Area
- Temporary Datum Features
- Datum Shift
- Proper Sequence of Datums in applying GD&T
- Importance of Datum Precedence

❖ Form Tolerances (Session : 4)
- Form Tolerances – Flatness, Straightness, Circularity & Cylindricity
- Tolerance Zone Shapes
- Straightness of Line Elements
- Straightness of a Flat Surface
- Straightness of an Axis / Cylindrical Features
- Flatness of the Derived Median Plane
- Inspection Methods to verify Form Tolerances

❖ Orientation Tolerances (Session : 5)
- Orientation Tolerances – Parallelism, Perpendicularity & Angularity
- Tolerance Zone Shapes
- Perpendicularity of a Surface
- Perpendicularity applied to Feature of Size
- Perpendicularity with Multiple Datums
- Angularity of a Surface
- Angularity applied to Feature of Size
- Parallelism of a Surface
- Parallelism applied to Feature of Size
- Orientation Tolerances in multiple segment (combined) Feature Control Frame
- Inspection Methods to Verify Orientation Tolerances

❖ Location Tolerances (Session : 6)
- 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

Framework Engineering, Pune

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

- Zero Position Tolerance at MMC
- When to use Location Tolerance?
- Position – Boundary for slots
- Simultaneous Vs Separate Requirements
- Bi-directional Tolerancing Pattern of Features
- Composite Tolerances (Composite FCF)
- Concepts of PLTZF & FRTZF
- Pattern-Locating Tolerance Zone Framework (PLTZF)
- Feature-Relating Tolerance Zone Framework (FRTZF)
- FRTZF with No / One / Two / Three Datums
- Linear Coaxial Feature Alignment
- 2-Lines Composite Tolerances
- 3-Lines Composite Tolerances
- Rules of Composite Tolerances
- Combined Position Tolerances
- Floating Fastener Assembly Condition & Calculation
- Fixed Fastener Assembly Condition & Calculation
- Inspection Methods to Verify Position Tolerances

❖ **Runout Tolerances (Session : 7)**
- Runout Tolerances – Circular Runout & Total Runout
- Tolerance Zone Shapes
- When to use Runout Tolerance?
- Circular Runout Tolerance for Sphere
- Circular Runout Tolerance for Cone
- Circular Runout Tolerance for Curve
- Total Runout Tolerance with two Datums
- Inspection Methods to Verify Runout Tolerances

❖ **Profile Tolerances (Session : 8)**
- Profile Tolerances – Profile of a line & Surface
- Tolerance Zone Shapes
- Symbol for Unequal or Unilateral Profile Tolerancing
- Multi-Segment or Combined Profile Tolerances
- Profile on a Unit basis
- Restraint notes on Non-Rigid parts - Specification and Interpretation
- Datums for Profile Controls
- When to use Profile Tolerance?
- Inspection Methods to Verify Profile Tolerances

Framework Engineering, Pune

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

❖ **Dimensional & Modifier Symbols (Session : 9)**
- Explanation on various Dimensional Symbols
- Explanation on various Modifier Symbols
- Controlled Radius
- Tangent Plane
- Projected Tolerance Zone
- Independency Symbol
- Free State Symbol
- Translation Modifier
- Continuous Feature Symbol

❖ **Summary & Test (Session : 10)**
- Summary of all Geometric Symbols
- Comparison of ASME & ISO Symbols
- Common mistakes to avoid, when using GD&T
- GD&T best practices
- Test on GD&T

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

- ✓ **Duration** : 4 Days (8 Hrs./Day)
- ✓ **Pre-requisite :** Basic understanding of part designing & drawings

Contact Details

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

www.framework-india.com

Framework Engineering, Pune