

Quick Recap: Basics of GD&T (Only around 1 hour)

- Revision of GD&T fundamentals, rules
- Feature of size and non-size features
- Virtual condition
- 14 GD&T Symbols
- Applications of Material Condition Modifier
- Basic Datums

16 Fundamental Rules of Dimensioning and Tolerancing

Virtual Condition & Resultant Condition Boundary

Tolerances Zone Shapes

Form Tolerances

- Straightness tolerances for line elements
- Straightness tolerances for cylindrical feature
- Straightness tolerance with MMC modifier
- Flatness tolerance for single planar feature
- Flatness tolerance with MMC modifier
- Tangent modifier
- Circularity Tolerance
- Cylindricity Tolerance
- Inspection methods to verify form tolerances

Gauges to verify Geometrical tolerances

- Functional / Fixed Gauges:
 - Go-Gauges / No Go-Gauges
 - Attribute / Fixed gauges
 - Gauge size calculations and gauge-makers tolerances
 - Gauge design exercises
- Variable Gauges
- Various geometrical tolerance inspection tools
- Hand calculation for position tolerance using Pythagoras theorem

Datums

- 4 criterion to select datums
- 3-2-1 Principle
- Datum center plane
- Partial Datum
- Coaxial and Co-planar datums
- Inclined Datum Features
- New datum features: Conical, Linear extruded shape, Complex Shape
- Identification of coordinate system location correctly
- Datum feature simulators
- Datum Targets: Points, Line, Area
- Fixture designs using datum targets
- Customized datum reference frame
- Temporary Datums
- Datum with translation modifier
- Datum shift (Datum feature shift)
- MMB calculations with 1 datum / 2 datums / 3 datums
- Cartoon gage sketch to verify datum shift
- Proper Sequence in applying GD&T

Orientation Tolerances

- How to apply?
- Perpendicularity tolerances with multiple datums
- Perpendicularity tolerances applied to feature of size
- Parallelism tolerances applied to feature of size
- Angularity tolerances applied to a feature of size
- Applied to a cylindrical or width type feature
- How to make 3D orientation tolerances to 2D
- Orientation tolerances in multiple segment (combined) feature control frame
- Inspection methods to verify orientation tolerances

Location Tolerances

A) Position Tolerance

- Position – LMC Basis
- Position –Boundary (for slots)
- Zero Tolerance at MMC
- Projected Tolerance Zone
- Position with no datums to establish primary datum axis from multiple features
- Simultaneous Vs Separate requirements
- Bi-Directional Tolerancing for pattern of features
- Composite Position tolerance:
 - Pattern-locating Tolerance Zone Framework (PLTZF)
 - Feature-relating Tolerance Zone Framework (FRTZF)
 - Composite tolerance with no datum / 1datum / 2datums / 3datums
 - 2-tier composite tolerance
 - 3-tier composite tolerance
 - 4-tier composite tolerance
 - Composite tolerance for Linear Coaxial Feature Alignment
- Rules of composite tolerances
- Combined position tolerances
- Use position to control co-axial features
- Datums for Position Control
- Inspection methods to verify position tolerances

B) Runout Tolerances

- Circular Runout Tolerance for sphere
- Circular Run out Tolerance for cone
- Circular Run out Tolerance for curve
- Total Run out for flat surfaces
- Total runout applied on partial length
- Total Run out Tolerance with two datums

C) Profile Tolerances

- Profile of a plane or tapered surface
- Multi-segment or combined profile tolerance
- Composite tolerances:
 - Composite tolerance with no datum / 1datum / 2datums / 3datums
 - 2-tier composite tolerance
 - 3-tier composite tolerance
- Profile on a Unit Basis
- Use of profile from flushness and gap analysis point view
- How to control co-planarity using profile tolerance
- Restraint notes on non-rigid parts – specification and interpretation
- Datums for Profile Controls
- Inspection methods to verify Profile tolerances

D) Concentricity & Symmetry Tolerances

- Concentricity Tolerance
- How to use position tolerance instead of concentricity
- How to use runout tolerance instead of concentricity
- Symmetry Tolerance
- How to use position tolerance instead of symmetry

Common mistakes to avoid when using GD&T on drawings

- Incorrect datum usage
- Incorrect geometrical control
- Incorrect modifier usage
- Incorrect calculation of virtual condition

Case Studies & Review of “Your Company” drawings

- Case studies in general and specific to your company / products
- Review existing Your Company drawings
- Discussion on the correct and incorrect use of GD&T on your existing drawings
- GD&T best practices for inspection and manufacturing

Contact

Mr. Swapnil Gujarathi

Senior GD&T Trainer: ASME-GD&T & ISO-GPS
Certified ASME GD&T Senior-Level Professional

CADSaga Consultancy Services

7, Vihar Sankul, Tidake Nagar,
Untwadi, Nashik - 422009

 +91 90110 81591

 swapnil@cadsaga.com

 www.cadsaga.com

Thank You !