Training Title: 5-Day "GD&T - Pro"

(Basics of GD&T + Advanced GD&T + Tol. Stack-up)

==========

"Basics of GD&T + Advanced GD&T" syllabus

GD&T: What, Where, When, Why..?

- Introduction to GD&T. What is GD&T and NOT?
- Why use GD&T? When do we use GD&T?

14 GD&T Symbols

- Feature Control Frame(FCF) and its placement
- How to read FCFs?
- Part Features, Feature of Size, Non-size Features
- Modifier symbols: MMC, LMC, RFS
- When to use MMC, LMC and RFS modifier in design (correct application of it)
- Bonus tolerance calculations
- Rule # 1 and Rule # 2

Virtual Condition & Resultant Condition Boundary

- Importance of Virtual condition & Resultant condition in designs
- Virtual condition and Resultant Condition calculations on MMC basis,
- Use virtual condition for mating parts as well to design fixed gauges
- 100% interchangeability guarantee in design phase itself by using virtual condition correctly
- Class room exercise for virtual condition & Resultant condition

Tolerances Zones

- Derived Element (Axis, Planes, Median point)
- Tolerance Zone Shapes

Datums

- What is datum?
- Six Degrees of Freedom
- 3-2-1 Principle
- Datum Reference Frame (DRF)
- Datum Feature Selection, Functional Hierarchy
- Datum feature identification / placements and interpretation
- Datum plane, datum axis, Datum center plane
- Datum feature simulators
- 3-2-1 principle
- Datum feature qualification
- Importance of Datum Precedence
- Select datum features according to the design intent
- Class room exercise for datums
- 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 Targets: Points, Line, Area
- Fixture designs using datum targets
- Custom datum reference frame
- Datum shift (or Datum feature shift)
- MMB calculations with 1 datum / 2 datums / 3 datums

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
- When to use Form Tolerance?
- Inspection methods to verify form tolerances

Orientation Tolerances

- How to apply?
- Perpendicularity tolerance
- Perpendicularity tolerances with multiple datums
- Perpendicularity tolerances applied to feature of size
- Parallelism tolerance
- Parallelism tolerances applied to feature of size
- Angularity tolerance
- Angularity tolerances applied to a feature of size
- Applied to a planar feature
- Applied to feature of size
- Applied to a cylindrical or width type feature
- How to make 3D orientation tolerances to 2D
- When do we use Orientation Tolerance?
- Datums for Orientation Control
- Orientation tolerances in multiple segment (combined) feature control frame
- Inspection methods to verify orientation tolerances
- Difference between parallelism and Flatness

Location Tolerances:

A) Position Tolerance

- How to apply?
- Tolerance zone shapes
- Position tolerance for cylindrical features (holes / shafts) on MMC, LMC and RFS basis
- Detailed table calculations for position tolerance on MMC, LMC and RFS
- Position on Boundary basis for rectangular or oblong slots
- Zero tolerance at MMC basis
- 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 / 1 datum / 2 datums / 3 datums
    - 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
- Class room exercise for position

B) Runout Tolerances

- Why do we use it?
- How does it work?
- How to apply it?
- Circular Runout Tolerance
- Total Run out Tolerance
- Datums for Run out Controls
- Inspection methods to verify runout tolerances

C) Profile Tolerances

- How it works?
- How to apply it?
- Profile of a Line Tolerance
- Profile of a Surface Tolerance
- 4 types of Profile Tolerance Zones
    - Bi-lateral
    - Unilateral – IN (Unequally Disposed)
    - Unilateral – OUT(Unequally Disposed)
    - Bi-lateral Un-equal(Unequally Disposed)
- Controlling extent of profile tolerance
- Multi-segment or combined profile tolerance
- Composite tolerances:
    - Composite tolerance with no datum / 1 datum / 2 datums / 3 datums
    - 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
- Class room exercise for profile

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
- Tolerance zone shape
- Datums for concentricity & symmetry controls
- Inspection methods to verify concentricity & symmetry tolerances

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
- Understand the internal competition between 14 GD&T symbols and select the most appropriate and inexpensive GD&T symbol.

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**

Fundamental Rules of Dimensioning / Tolerancing

Common mistakes to avoid when using GD&T on drawings:

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

==========

"Tolerance Stack-up Analysis" syllabus

- Importance of Tolerance Stack-up Analysis: Why, when and how
    - Introduction to Tolerance Stack-up Analysis:
        - Coordinate dimensioning and tolerancing stacks
        - GD&T stacks
        - Statistical tolerancing stacks: RSS and MRSS
    - Methodical steps involved in any stack-up analysis
        - Loop diagram creation
        - Universal sign language of tolerance stack-up analysis
        - identification of correct dimensional contributors (vectors)
        - Drawing improvements to reduce tolerance stack-up
    - Understanding Tolerance Stack-up and Tolerance allocation difference
    - Considerations of process capable tolerances

- **Part Level Stack-up Analysis:**
    - Part stacks using coordinate dimensioning and tolerancing
    - Part stacks using position
        - Review of Virtual Condition (VC) Boundary & Resultant Condition (RC) Boundary concepts
        - IB (inner boundary) and OB (outer boundary) calculations
    - Part stacks using profile
    - Part stacks using runout
    - Part stacks using bonus (planar & RFS datums)
    - Part stacks using position with bonus tolerance
    - Part stacks using combined geometric tolerances

- **Assembly Level Stack-up Analysis:**
    - Assembly stacks using coordinate dimensioning and tolerancing
    - Assembly stacks using position:
        - Floating fastener assembly stack-up
        - Fixed fastener assembly stack-up
        - Floating and Fixed fastener formulas
    - Assembly stacks using profile

- Assembly stacks using runout
- Assembly stacks using combined geometric tolerances
- Stacks using form controls applied to a feature
- Stacks using orientation controls applied to a feature without size
- Stacks using orientation controls applied to a feature of size (FOS)
- Introduction to Statistical Tolerancing:
    - RSS (Root Sum Square) Stacks
    - MRSS (Modified Root Sum Square) Stacks
- Comparison between Worst-case method, RSS & MRSS methods
- Live demo of Stack-up analysis using Excel sheet templates:
    - Don's and Dont's of Excel sheet stack-up analysis

Contact:

Mr. Swapnil Gujarathi
ASME-USA Certified, Senior GD&T Trainer

CADSaga Consultancy Services
L-8, Suyojit Sankul, Sharanpur Road
Nashik - 422002

90110 81591

www.cadsaga.com swapnil@cadsaga.com