

GD&T Introduction

- What is GD&T? Why use GD&T?
- How does GD&T work?
- Comparison of GD&T with CD&T (Coordinate Dimensioning and Tolerancing)

Part Features, Feature of Size, Non-size Features

- What are Part Features?
- Features of Size
- Non-Size Features

14 GD&T Symbols

- Feature Control Frame and its placement
- How to read FCFs?
- Modifier symbols: MMC, RFS
- When to use MMC, RFS modifier in design (correct applications)
- Bonus tolerance calculations
- Rule # 1 and Rule # 2
- Tolerances Zones

Datums

- What is datum?
- Six Degrees of Freedom
- Datum Reference Frame (Cartesian coordinate system)
- Datum Feature Selection, Functional Hierarchy
- Datum feature identification / placements and interpretation
- Datum plane, datum axis, Datum center plane
- Physical datum feature simulators
- 3-2-1 principle

Form Tolerances

- Flatness tolerance
- Straightness tolerance
- Circularity Tolerance
- Cylindricity Tolerance
- Inspection methods to verify form tolerances

Location Tolerances

A) Position Tolerance

- How it works? How to apply it?
- Tolerance zone shapes
- Position tolerance for cylindrical features on MMC and RFS basis
- Detailed table calculations for position tolerance on MMC and RFS
- Datums for Position Control
- Inspection methods to verify position tolerances
- Class room exercise for position

B) Runout Tolerances

- How it works? 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
- Bi-lateral Profile Tolerance Zone
- Controlling extent of profile tolerance
- Inspection methods to verify Profile tolerances

D) Concentricity & Symmetry Tolerances

- How it works? How to apply it?
- Concentricity Tolerance
- Symmetry Tolerance
- Tolerance zone shapes
- Datums for concentricity & symmetry controls
- Inspection methods to verify concentricity & symmetry tolerances

Orientation Tolerances

- Perpendicularity tolerance
- Parallelism tolerance
- Angularity tolerance
- Applied to a planar feature
- Applied to feature of size
- Inspection methods to verify orientation tolerances

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 !