



# GD&T Course Outline (3-day)

## Module 1: Introduction and Terms

- What is GD&T?
- How do we reference it?
- Why do we use GD&T?
- How does GD&T work?
- Reference dimensions
- Tolerances
- Basic dimensions
- Methods to identify basic dimensions
- Feature

## Module 2: Symbols

- Why do we use symbols?
- Radius tolerances
- Controlled radius
- Dimension origin symbol
- Geometric controls
- Repetitive features
- Feature control frame placement
- Screw threads

## Module 3: Fundamental Rules

- Application of tolerances
- Fundamental rules
- Tolerance rules
- Interpretation of limits
- Metric limits and fits

## Module 4: Features of Size

- Feature of Size
- Material condition
  - Maximum material condition
  - Least material condition
  - Regardless of feature size
- Modifier symbol application
- Functional gages
- When do we use material condition modifiers?
- Virtual condition
  - MMC virtual condition
  - LMC virtual condition

## Module 5: Form Tolerances

- Form Tolerances
- Rule 1: Perfect form at MMC
- Nullifying rule 1
- Exceptions to rule 1
- Straightness tolerance for line elements

- Straightness tolerance for cylinders
  - Applied RFS
  - Applied at MMC
  - Applied at LMC
- Flatness tolerance for a single planar feature
- Circularity tolerance
- Cylindricity tolerance
- Application over a limited length or area
- Application on a unit basis
- When do we use form tolerances?

## Module 6: Datuming

- What is a datum?
- Datum feature
- Datum feature symbol application
- True geometric counterpart (TGC)
- Datum reference frame (DRF) and three mutually-perpendicular planes
- Degrees of Freedom
- Feature of size referenced as a primary datum
  - At RFS
  - At MMC
- Feature of size referenced as a secondary datum
  - At RFS
  - At MMC
- Datum reference frame (DRF) displacement
- Simultaneous requirements
- Separate requirements
- Datum targets
  - Symbol
  - Identification
  - Application

## Module 7: Orientation Tolerances

- How to apply it
- Tolerance zones
  - Parallelism
  - Perpendicularity
    - Zero tolerance at MMC
    - Converting a zero tolerance at MMC
  - Angularity
- Applied to a planar feature
- Applied to a cylindrical or width feature
- Applied to line elements
- Angular tolerance zones
- Application of orientation tolerances

## Module 8: Position Tolerance

- Plus/Minus tolerancing
- How does position tolerancing work?
- Minimum tolerance between features
- How to apply it?
- Methods for establishing true positions
- Width type features
- Establishing true positions for angled features
- Virtual condition boundary for location
- Bi-directional position tolerancing
- Spherical tolerance zone
- Fixed fasteners
- Floating fasteners
- Zero tolerance at MMC
- Composite position
  - PLTZF virtual condition boundaries
  - FRTZF virtual condition boundaries
  - Simultaneous requirements
- Projected tolerance zone

## Module 9: Runout Tolerances

- Runout Tolerance
- Datums for runout tolerance
  - Single diameter
  - Coaxial features
  - Diameter and face features
- Circular runout
- Total runout
- When do we use runout tolerances?

## Module 10: Profile Tolerances

- Profile of a surface
- Profile tolerance zone
- Profile to control size and form
- Profile to control size, form and orientation
- Profile to control size, form, orientation and location
- Profile to control coplanarity
- Profile of a line

## Module 11: Symmetry Tolerances

- Concentricity
- Comparison of coaxial controls
- Symmetry