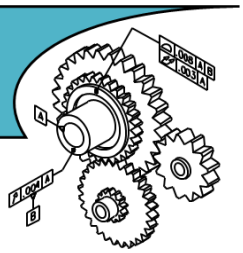
MechSigma GD&T Course Topics

- Based on ASME Y14.5-2018 and ASME Y14.41-2019



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?
- Terminology
  - Dimensions
  - Tolerances
  - Reference dimensions
  - Basic dimensions
  - Methods to identify basic dimensions
- Feature
  - Feature of size
    - Regular feature of size
    - Irregular feature of size
  - Non-size feature
- Actual local size
- Actual mating envelope
- Actual minimum material envelope
- Continuous feature

Module 2: Symbols and Feature Control Frames

- Why do we use symbols?
- Symbols
  - Diameters
  - Radius tolerances
  - True radius
  - Controlled radius
  - Spherical radiusSquare symbol
  - Statistical tolerance symbol
  - Documenting statistical tolerances
  - Dimension origin symbol
  - Counterbores/countersinks
  - Spotfaces
  - Angular surfaces
  - Conical taper/flat taper
- Geometric controls/symbols
- Feature control frame
- Feature control frame placement
  - On orthographic views (drawings)
  - On models
- Repetitive features

Module 3: Fundamental Rules and General Tolerancing Applications

- Application of tolerances
- Fundamental rules
- Nonrigid parts
- Specifying restraint
- Free state
- Tolerance rules
- Interpretation of limits
- Metric limits and fits

Module 4: Form Controls

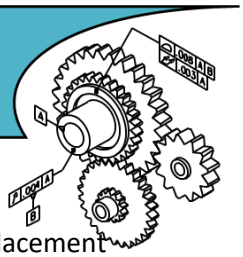
- How to apply them
- Rule 1: Perfect form at MMC
- Nullifying rule 1
- Exceptions to rule 1
- Straightness tolerance for line elements
- Straightness tolerance for cylinders
  - Applied regardless of feature size (RFS)
  - Applied at maximum material condition (MMC)
  - Applied at least material condition (LMC)
- Flatness tolerance for a single planar feature
- Flatness tolerance for a width-type feature
- Circularity tolerance
- Cylindricity tolerance
- Circularity or cylindricity tolerance with average diameter
- Applied over a limited length/area
- Applied on a unit basis
- When do we use form tolerances?

Module 5: Datuming (Part 1)

- What is a datum?
- Datum feature
- Datum feature symbol application
  - Identifying datum features on orthographic views (drawings)
  - Identifying datum features on models
- Establishing datums from planar features
- Establishing datums from coplanar features
- Datum reference frame (DRF) and three mutually-perpendicular planes
- Degrees of Freedom
- True geometric counterpart
- Datum planes not at 90 degrees
- Rocking datums
- Datum targets
  - Symbol
  - Identification
- Target applications
- Moveable datum targets
- Selecting the best datum targets

MechSigma GD&T Course Topics

- Based on ASME Y14.5-2018 and ASME Y14.41-2019



Module 6: Orientation Controls

- How to apply them
- Tolerance zones for planar features
  - Parallelism
  - Perpendicularity
  - Angularity
- Applied to a tangent plane
- Applied to line elements
- Angular (wedge shaped) tolerance zones
- Applied to cylindrical features (RFS)
- Applied to width-type features (RFS)
- Application of orientation tolerances
- Replacing perpendicularity and parallelism symbols with angularity symbol

Module 7: Features of Size

- Material conditions
  - Regardless of feature size
    - Internal feature RFS
    - External feature RFS
- Maximum material condition
  - Internal feature at MMC
  - External feature at MMC
- Least material condition
  - Internal feature at MMC
  - External feature at MMC
- When to use each material condition modifier?
- MMC Virtual condition
  - Internal feature at MMC
  - External feature at MMC
  - When to calculate MMC virtual condition
- Zero perpendicularity at MMC
- LMC Virtual condition
  - Internal feature at LMC
  - External feature at LMC
  - When to calculate LMC virtual condition

Module 8: Datuming (Part 2)

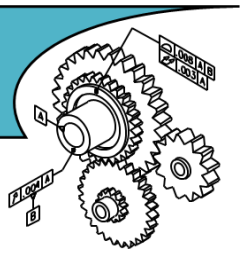
- Material boundaries
  - Regardless of material boundary (RMB)
  - Maximum material boundary (MMB)
  - Least material boundary (LMB)
- Internal feature of size at RMB
  - Primary / Secondary / Tertiary
- Translation modifier
- External feature of size at RMB
  - Primary / Secondary / Tertiary
- Internal feature of size at MMB
  - Primary / Secondary / Tertiary
- External feature of size at MMB
  - Primary / Secondary / Tertiary
- Datum reference frame (DRF) displacement
- Simultaneous requirements
- Separate requirements
- Internal feature of size at LMB
  - Primary / Secondary / Tertiary
- External feature of size at LMB
  - Primary / Secondary / Tertiary
- Pattern of holes
- Mathematical surfaces
- Offset datum features
- Note to establish a DRF
- Restrained datum features
- Releasing degrees of freedom with customized datum reference frames

Module 9: Position Control

- Plus/Minus tolerancing
- How does position tolerancing work?
- Minimum tolerance between features
- How to apply it?
- Methods for establishing true positions
- Axis interpretation vs. surface interpretation
- Width type features
- Establishing true positions for angled features
- Bi-directional position tolerancing
- Spherical tolerance zone
- Fixed fasteners
- Floating fasteners
- Position for coaxiality
- Virtual condition boundary for location
- Zero tolerance at MMC
- Screw thread rule
- Gears and spline rule
- Composite position
  - PLTZF and FRTZF
  - Simultaneous requirements
  - Separate requirements
  - Prohibiting customized datum reference frame
- Stacked single segment feature control frames
- Projected tolerance zone

MechSigma GD&T Course Topics

- Based on ASME Y14.5-2018 and ASME Y14.41-2019



Module 10: Runout Controls

- How to apply them
- Datums for runout tolerance
  - Single diameter
  - Coaxial features
  - Diameter and face features
- Circular runout
- Total runout
- Runout on a tangent plane
- Runout applied to an interrupted feature
- Using the between symbol with runout
- Runout on an assembly
- When do we use runout tolerances?

Module 11: Profile Controls

- How to apply them
- 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 on a tangent plane
- Dynamic profile
- Non-uniform tolerance zone
- Profile of a line
- Profile of a line with a customized datum reference frame
- Represented line element
- Profile “All Around”
- Profile “Between”
- Profile “From—To”
- Profile ”All Over”
- Profile applied to continuous features
- Composite profile
- When do we use profile tolerances?

Module 12: Symmetry Controls (per ASME Y14.5-2009 and prior)

- How to apply them
- Concentricity
- Comparison of coaxial controls
- Symmetry