GD&T uses an international language of symbols to detail a product in a three-dimensional perspective. It describes the form, orientation, and location of a part’s features within a zone of tolerance.

Symbols and Definitions
- MMC-maximum material condition
- Feature contains the maximum amount of material
- RFS-regardless of feature size
- Tolerance of form, size or runout must be met regardless of where the feature lies within its size tolerance
- LMC-least material condition
- Feature contains the least amount of material

Symbols and Definitions
- Projected tolerance zone
- Used to control 90° angle of a hole; extends tolerance zone for a distance beyond the surface of the part
- Basic dimension
- Theoretical value used to describe the exact size, shape, or location of a feature. Perfect; a tolerance required is always required

Virtual condition
- Constant worst case boundary generated by the collective effects of MMC or LMC and the geometric tolerance for that material condition
- Holes=MMC-tolerance or LMC+tolerance; shafts=MMC-tolerance or LMC-tolerance
- Full indicator movement
- Total indicator reading when applied to a feature same as FIR or TIR

Symbols and Definitions
- Datum feature symbol
- Identifies the feature from which functional relationships are established
- Feature control frame
- Boxed expression containing GDT symbol and all other references and modifiers that may apply
- Tolerance zone
- Total allowable change from desired form, attitude, runout, or location of the feature

Virtual condition
- Virtual condition
- Constant worst case boundary generated by the collective effects of MMC or LMC and the geometric tolerance for that material condition
- Holes=MMC-tolerance or LMC+tolerance; shafts=MMC-tolerance or LMC-tolerance
- Full indicator movement
- Total indicator reading when applied to a feature same as FIR or TIR

Symbols and Definitions
- Symbol
- Identifies the feature from which functional relationships are established
- Feature control frame
- Boxed expression containing GDT symbol and all other references and modifiers that may apply
- Tolerance zone
- Total allowable change from desired form, attitude, runout, or location of the feature

Virtual condition
- Virtual condition
- Constant worst case boundary generated by the collective effects of MMC or LMC and the geometric tolerance for that material condition
- Holes=MMC-tolerance or LMC+tolerance; shafts=MMC-tolerance or LMC-tolerance
- Full indicator movement
- Total indicator reading when applied to a feature same as FIR or TIR

Symbols and Definitions
- Symbol
- Identifies the feature from which functional relationships are established
- Feature control frame
- Boxed expression containing GDT symbol and all other references and modifiers that may apply
- Tolerance zone
- Total allowable change from desired form, attitude, runout, or location of the feature

Virtual condition
- Virtual condition
- Constant worst case boundary generated by the collective effects of MMC or LMC and the geometric tolerance for that material condition
- Holes=MMC-tolerance or LMC+tolerance; shafts=MMC-tolerance or LMC-tolerance
- Full indicator movement
- Total indicator reading when applied to a feature same as FIR or TIR
MFG 316 Chapter 13
Geometric Dimensioning and Tolerancing

- **Figures**

MFG 316 Chapter 13
Geometric Dimensioning and Tolerancing

- **Flatness**
  - All elements of a surface are in the same plane
  - Specified a tolerance zone bordered by two parallel planes, within which the entire surface must lie
  - No datum used

MFG 316 Chapter 13
Geometric Dimensioning and Tolerancing

- **Straightness**
  - Condition where a portion of a surface or an axis is a straight line
  - Specified a cylindrical tolerance zone within which the entire axis or element must lie
  - No datum used

MFG 316 Chapter 13
Geometric Dimensioning and Tolerancing

- **Cylindricity**
  - All points on the surface of a cylinder are equal distances from the axis
  - Tolerance zone bounded by two concentric cylinders
  - No datum used

MFG 316 Chapter 13
Geometric Dimensioning and Tolerancing

- **Circularity (roundness)**
  - All points on the surface intersected by any plane at right angles to an axis have all points an equal distance from the axis
  - Tolerance zone bounded by two concentric circles, within which all elements of the surface must lie
  - No datum used

MFG 316 Chapter 13
Geometric Dimensioning and Tolerancing

- **Profile of a line or surface**
  - Used to specify an allowable deviation from the desired profile where other geometric controls cannot be used
  - Specifies a uniform boundary along the desired profile within which the elements of the surface or line must lie
  - Datum may or may not be required
  - Rule 1 may not apply
**Perpendicularity**
- Surface, median plane, or axis is exactly 90° from a datum plane or datum axis
- Always requires a datum
- Controls flatness to an extent

**Parallelism**
- Surface, line, or axis is equidistant from a datum line or plane at all points
- Always requires a datum
- Controls flatness to an extent

**Position**
- Specify how far a feature of size may vary from the theoretically exact location on the part drawing
- Defines a zone within which the axis or center plane of a feature of size can vary
- Datums always required

**Angularity**
- Surface or axis must be at a specified angle from a datum plane or axis.
- Tolerance zone defined by two parallel planes at a specified basic angle from one or more datum planes or axes within which the feature must lie
- Datum is always used; basic angle is required

**Runout**
- Deviation from the desired form of a part surface of revolution when the part is rotated 360° around a datum axis
- Tolerance specifies the max FIM allowed
- Always RFS; always requires a datum

**Concentricity**
- Two or more features have a common axis
- Specifies the condition in which all median points of all diametrically opposed elements of a figure of revolution are congruent with the axis
- Datums required
Symmetry

- Specifies the condition in which all median points of all opposed or correspondingly located elements of two or more feature surfaces are congruent with the axis or center plane of a datum feature.
- Datums required

Three plane concept

- 321 method of surface location
- Established by or relative to part features

Figures

Positional tolerance of zero!