



**Course name** Introduction to Geometric Dimensioning & Tolerancing (GD&T)

**Target Audience** Any Engineer, draft person, QA person or manufacturer required to interpret technical drawings.

**Prerequisites** A good understanding of general technical drawing techniques.

**Duration** 7.5 hours

**Delivery Mode** Instructor Led

### Course Objectives

Upon completion of this course the learner will be able to

- Explain the fundamentals of GD&T to anyone involved in the development or manufacturing process.
- Interpret the geometric tolerances to define the allowable variation in form and size of part features.
- Use the Datums to locate a part in space.
- Interpret the GD&T symbols defined by the international standards

### Course Outline

1. Introduction
  - Benefits
  - Difference between Geometric Tolerancing & traditional techniques
  - International standards & symbology
  - Tolerance Frame
2. Basic Dimensions
  - Describing the theoretical exact dimension
  - Using basic dimensions
  - Using the 3D model
  - Interpretation exercise
3. Datum Reference Frames
  - Locating a part in space
  - Datum features
  - Determining the Datum order
  - Datum Targets
  - Interpretation exercise
4. Tolerance Characteristics
  - Determining the shape of a tolerance zone
  - Determining the size of a tolerance zone
  - Form Tolerances
    - i. Straightness, Flatness, Circularity, Cylindricity
    - ii. Profile of a Line, Profile of a Surface
    - iii. Interpretation exercise
  - Orientation
    - i. Angularity, Parallelism, Perpendicularity
    - ii. Interpretation exercise
  - Location
    - i. Position, Concentricity, Symmetry
    - ii. Interpretation exercise
  - Alignment
    - i. Circular Run out, Total Run out
    - ii. Interpretation exercise



**Course name** Geometric Dimensioning & Tolerancing (GD&T) in Conception

**Target Audience** Mechanical engineers and drafters who are required to create mechanical drawings or specifications.

**Prerequisites** Introduction to Geometric Dimensioning & Tolerancing (GD&T)

**Duration** 7.5 hours

**Delivery Mode** Instructor Led

### Course Objectives

Upon completion of this course the learner will be able to

- Apply a balance of traditional dimensioning and GD&T to create simplified technical drawings with increased precision.
- Emphasise the critical dimensions and functional requirements of a part.
- Specify Statistical Tolerance Stacking in assemblies to allow larger tolerances.
- Combine large casting tolerances with precise machining tolerances using Target Datums to ensure functionality requirements are achieved.
- Apply minimal dimensioning to simplify drawings for complex CAD geometries and easily verify compliance in a CMM environment.

### Course Outline

1. Introduction
  - GD&T in mechanical engineering design
  - Determining the functional requirements of a feature
  - Applying the GD&T methodology
2. Setting the Datum reference frame
  - Select the Datums
  - Determine the order
  - Target Datums with castings
  - Application Exercise
3. Setting the tolerance requirements
  - Select the tolerance characteristic
  - Calculate the allowable variation
  - Separate GD&T requirements
  - Application Exercise
4. Tolerance Stacking
  - Calculate the tolerance chain
  - Evaluate worst case scenario
  - Use Statistical Tolerances if applicable
  - Application Exercise
5. Minimal Dimensioning
  - Complex geometries
  - Using CAD with GD&T
  - Application Exercise