



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

Target Audience Any Engineer, draft 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
 - Feature Control 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.
- Use Modifiers to tailor requirements to the design intent of a part and reduce costs by better defining the allowable precision.
- 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. Using Modifiers
 - Max & Min material conditions
 - Projected tolerance zone
 - Free State
 - Tangent Plane
 - Statistical Tolerance
 - Application Exercise
5. Tolerance Stacking
 - Calculate the tolerance chain
 - Evaluate worst case scenario
 - Use Statistical Tolerances if applicable
 - Application Exercise
6. Minimal Dimensioning
 - Complex geometries
 - Using CAD with GD&T
 - Application Exercise