Quality LearningHub & Consultancy Singapore
22 Sin Ming Lane, #06-76 Midview City
Singapore 573969
Tel: (65) 6502 8238
Email: carina@qualitylearninghub.com.sg
Website: www.qualitylearninghub.com.sg

GD&T Master Class
(Basic GD&T + Advanced GD&T)

Introduction:

GD&T is a graphic language used by mechanical designers, engineers, and quality inspection operators to effectively and accurately communicate with one another through detailed drawings. It defines the geometry of mechanical parts without any ambiguities through symbols for geometric dimensions, tolerances and relative positions among features and datums. With GD&T defined drawings, desired forms and fits can be realised. This ensures the intended function of the parts and assemblies.

Course Contents:

GD&T: What, Where, When, Why.. ?
- Introduction to GD&T?
- Feature Control Frame (FCF) and its placement
- Part Features, Feature of Size, Non-size Features
- When to use MMC, LMC and RFS modifier in design
- Bonus tolerance calculations
- Rule # 1 and Rule # 2

Actual Mating Envelopes
- Related actual mating envelope
- Unrelated actual mating envelope

Virtual Condition & Resultant Condition Boundary
- Importance of Virtual condition & Resultant condition in designs
- Virtual condition and Resultant Condition calculations on MMC basis
- 100% interchangeability guarantee in design phase itself
- Exercise for virtual condition& Resultant condition

Datums
- What is datum?
- Six Degrees of Freedom
- 3-2-1 Principle
- Datum Reference Frame (DRF)
- Datum Feature Selection, Functional Hierarchy
- Datum feature identification / placements and interpretation
- Datum plane, datum axis, Datum center plane
- Datum feature simulators

- Datum feature qualification
- Datum Precedence
- Select datum features according to the design intent
- Partial Datum
- Coaxial and Co-planar datums
- New datum features: Conical, Linear extruded shape, Complex Shape
- Temporary datums
- Datum Targets: Points, Line, Area
- Individual datums (Repetitive Patterns of Features)
- Datum Feature Shift (or Datum Shift)

Form Tolerances

- Straightness tolerance for line elements
- Flatness tolerance for single planar feature
- Flatness tolerance with MMC modifier
- Flatness tolerance on unit-basis
- Circularity Tolerance
- Cylindricity Tolerance
- Inspection methods to verify form tolerances

Orientation Tolerances

- Perpendicularity tolerance
- Perpendicularity tolerances applied to feature of size
- Parallelism tolerance
- Tangent modifier
- Angularity tolerance
- Applied to a planar feature
- Orientation tolerances in multiple segment (combined) feature control frame
- Inspection methods to verify orientation tolerances

Location Tolerances:

A) Position Tolerance

- How to apply? Tolerance zone shapes
- Position tolerance for cylindrical features (holes / shafts) on MMC, LMC and RFS basis
- Detailed table calculations for position tolerance on MMC, LMC and RFS
- Position on Boundary basis for rectangular or oblong slots
- Zero tolerance at MMC basis
- Projected Tolerance Zone
- Simultaneous requirement

Quality LearningHub & Consultancy Singapore
22 Sin Ming Lane, #06-76 Midview City
Singapore 573969
Tel: (65) 6502 8238
Email: carina@qualitylearninghub.com.sg
Website: www.qualitylearninghub.com.sg

- Separate requirement
- Bi-Directional Tolerancing for pattern of features
- Composite Position tolerancing:
  - Pattern-locating Tolerance Zone Framework (PLTZF)
  - Feature-relating Tolerance Zone Framework (FRTZF)
- Combined position tolerance with other geometric symbols
- Floating Fastener Formula and calculations
- Fixed Fastener Formula and calculations
- Inspection methods to verify position tolerances
- Exercises for position

B) Runout Tolerances

- Circular Runout Tolerance
- Total Run out Tolerance for cylindrical feature
- Total Run out Tolerance for planar feature
- Runout tolerance with co-axial datum
- Inspection methods to verify runout tolerances

C) Profile Tolerances

- Profile of a Line Tolerance
- Profile of a Surface Tolerance
- 4 types of Profile Tolerance Zones
  - Bi-lateral
  - Unilateral – IN (Unequally Disposed)
  - Unilateral – OUT (Unequally Disposed)
  - Bi-lateral Un-equal (Unequally Disposed)
- Controlling extent of profile tolerance: Between symbol
- All-around and all-over modifiers
- Multi-segment or combined profile tolerance
- Composite tolerances: PLTZF & FRTZF
- FRTZF with no datum / 1 datum / 2 datums / 3 datums
- Profile on a Unit Basis
- Restraint notes on non-rigid parts - specification
- Inspection methods to verify Profile tolerances
- Exercise for profile

D) Concentricity & Symmetry Tolerances

- Concentricity Tolerance
- How to use position tolerance instead of concentricity
- How to use runout tolerance instead of concentricity
- Symmetry Tolerance

- How to use position tolerance instead of symmetry
- Inspection methods to verify concentricity & symmetry tolerances

Gauges to verify Geometrical tolerances:

- Functional / Fixed Gauges:
  - Go-Gauges / No Go-Gauges
  - Functional Gauges (Also known as Attribute / Fixed / Receiving / Qualifying gauges)
    - Gauge design exercises
- **Variable Gauges**
- **Various geometrical tolerance inspection tools**

Who Should Attend:

This course is beneficial for Manufacturing, Mechanical, Production, Design, Quality, Reliability, and Engineers, Managers, Designers, Inspectors, Supplier Quality Personnel and Technicians who need to specify, interpret and apply geometric tolerancing.

Pre-requisite:

Whilst no prior knowledge is required, some familiarity with part drawings and general manufacturing processes would be helpful.

Learning Methodology

Group discussion, case studies and hands-on exercises will be used to reinforce participants' understanding of each topic.

Award of Certificate:

Participants will be issued with a Certificate of Successful Completion upon meeting 75% of the required course attendance.

Duration:

5 days (35 hours)

Course Fee:

$1600 nett per trainee (GST is not applicable).

(Course fee is inclusive of all training materials and light refreshments.)