Aditi Consultancy Services

#2, 'ANANDA', 19th Cross, 24th Main, J P Nagar 5th Phase, Bangalore - 560 078
Phone: **080-26580711;** Mobile: **9449612671**

Email**: training@aditiconsultancy.com; aditiconsultancy@gmail.com**
Web **- http://www.aditiconsultancy.com**

ASME GDTP Certification – Technologist Level

All the major topics defined in the body of knowledge of ASME GDTP Technologist Level certification (Appendix A) are included in our discussions as listed below:

1 Scope, Definitions and General Dimensioning

- General information about Y14.5 Standard
- Definitions
- Fundamental rules
- Units of measurement
- Millimeter / decimal inch dimensioning
- Dimensioning, application of dimensions
  - Dimensioning features
  - Location of features
    - rectangular coordinate dimensioning
    - rectangular coordinate dimensioning without dimension lines
    - tabular dimensioning
    - polar coordinate dimensioning
    - repetitive features or dimensions
    - use of "X" to indicate "BY" or "NUMBER OF PLACES"

2 General Tolerancing Related Principles and Former Practices

- Application of tolerances
- Expression of Tolerance
- Interpretation of limits
- Accumulation of Tolerance
- Limits of size, feature of size (Rule 1), exceptions to Rule 1
- Applicability and effect of RFS, MMC, LMC
- Effect of zero tolerance at MMC and LMC
- Application of geometric tolerance to screw threads
- Application of geometric tolerance to gears and splines
- Virtual/resultant condition - MMC & LMC
- Datum features at virtual condition
- Angular surfaces
- Conical and flat tapers
- Radius and Controlled Radius Tolerance
- Statistical tolerancing
- Former practices of Y14.5 standard

3 Symbology

- Geometric characteristic symbols
- Datum feature symbol
- Relating datum symbol frame to datum feature
- Datum target symbol
- Basic dimension symbol
- Counterbore, Countersink, depth, square, dimension origin and all around Symbols
- Modifying symbols
- Feature control frame incorporating one, two and three datum references
- Feature control frame placement
- Definition of the Tolerance zone

4 Datum Referencing

- Definitions
- Datum simulator
- Datum reference frame
- True geometric counterparts
- Datum feature order of precedence
- Establishing datums from datum features
- Parts with inclined datum features
- Datum features not subject to size variations
- Datum features subject to size variations
- Specifying datum features RFS, at MMC and at LMC
- Multiple datum features
- Pattern of features
- Screw threads, gears, and splines
- Partial surface as datum features
- Mathematically defined surface
- Multiple datum reference frames
- Simultaneous versus separate requirements
- Simultaneous requirements and composite feature control
- Datum targets: Points, Lines and Areas
- Datum target dimensions
- Datum planes established by datum targets
- Methods of establishing a primary datum axis
- Equalizing datums
- Datums established from complex or irregular surfaces

5 Tolerances of Location

- Types of location tolerances
- Position tolerancing
- Basic dimensions: application to base line and chain dimensioning
- Effect of material condition
- Zero positional tolerancing at MMC
- Multiple patterns of features located by basic dimensions relative to common datums
- Simultaneous requirements — RFS
- Simultaneous requirements — MMC
- Feature pattern location
- Feature Relating Tolerance Zone Framework (FRTZF)

- Pattern Locating Tolerance Zone Framework (PLTZF)
- Composite positional tolerancing
- Projected tolerance zone
- Nonparallel holes
- Counter bored holes
- Closer control at one end of a feature
- Bi-directional positional tolerancing of features
- Noncircular features
- Coaxiality controls
- Concentricity
- Differences between coaxiality controls and concentricity
- Positional tolerancing for symmetrical relationships
- Symmetry tolerancing
- Spherical features

6 Tolerances of Form, Profile, Orientation and Runout

Meaning of the symbols, modifiers and relationships as applied to engineering drawings and related documentation of the following:

- Form **tolerance**
  - Straightness
  - Flatness
  - Circularity
  - Cylindricity
- Orientation
  - Angularity
  - Parallelism
  - Perpendicularity
- Profile
  - Profile of a line
  - Profile of a surface
- Runout
  - Circular Runout
  - Total Runout

The approximate distribution of questions is as follows:

1) **10%** on Scope, Definitions and General Dimensioning
2) **10%** on General Tolerancing and Related Principles and Former Practices of Y14.5
3) **5%** on Symbology
4) **15%** on Datum Referencing
5) **30%** on Tolerances of Location
6) **30%** on Tolerances of Form, Profile, Orientation and Runout