Geometric Dimensioning & Tolerancing Training & Certification Program - Technologist Level

3 Days, 2.25 CEUs, 22.5 PDHs on 1-3 August, 2011 at Mumbai

Organised by

CII

in association with ASME, NY

Why GD&T is so important for effective and efficient Product Development Life Cycle (PDLC)?

Manufacturing-based variations are not normally considered as a part of the design in the CAD systems, but are a critical part of the overall PDLC process. Companies are now realizing that a great design has to comprehend both functional design features and the associated manufacturing process capabilities in order to produce a robust product. But the constraint is that all current CAE and CAD geometry systems base all analysis on idealized manufactured and assembled conditions.

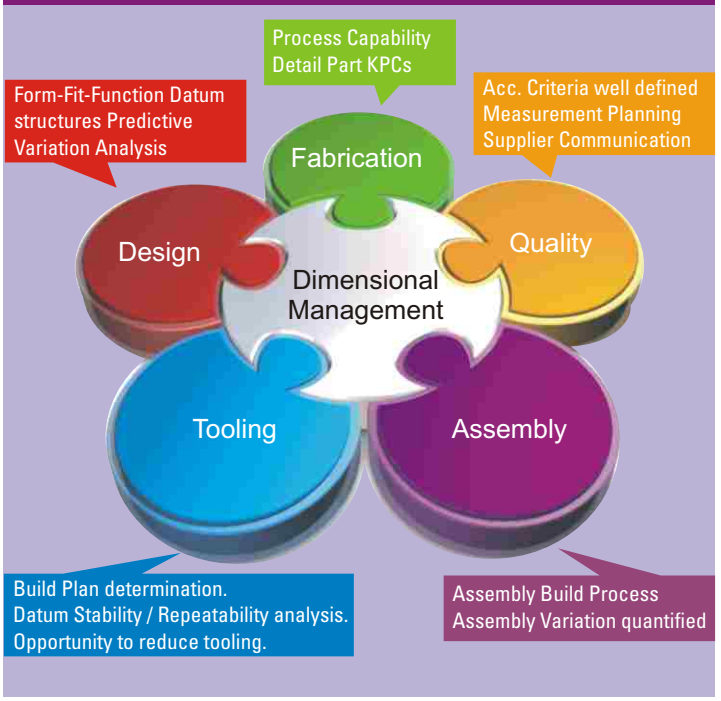
For example,

- Interference checking is conducted on a CAD model that is assumed to be "perfect."
- Structural analysis assumes a perfectly accurate form and correct location of parts in an assembly.

As a result, variations in products are introduced as functions of manufacturing processes used to produce parts that deviate from nominal on the surfaces of parts. Manufacturing variation then propagates or moves through assemblies as a function of the contacting surfaces in the assembly. This can result in significant cost and quality effects due to product failures or higher than necessary product costs.

Standardizing Geometric Dimensioning and in particular Tolerance analysis practices when used together with Variation Analysis tools and techniques is one way to understand, address and overcome this constraint. Wherein, Variation information is collected from a number of sources and is used to relate the variability of the manufacturing processes to the limits of the design tolerances. Tolerance analysis assumes that tolerances can be achieved by manufacturing followed by Variation behaviour based analysis that uses actual manufacturing process data as the source of variation values. The ASME GD&T Trainings is the 1st Step in this direction by ASME as a commitment to Knowledge dissemination in this area for Global Engineering workforce Development for effective usage of this best practices tool in Design and Manufacturing .

Interrelation of Dimensional Management Process with other PLM functions



ASME Personnel Qualification Program comprising of 3 days of Training on Basic GD&T Practices and Tools serves both as a knowledge based cum proficiency building Preparation course for appearing for GDTP –Technologist Level Certification , read on…

Geometric Dimensioning & Tolerancing -Preparatory Course for GDTP Technologist Level Certification Examination

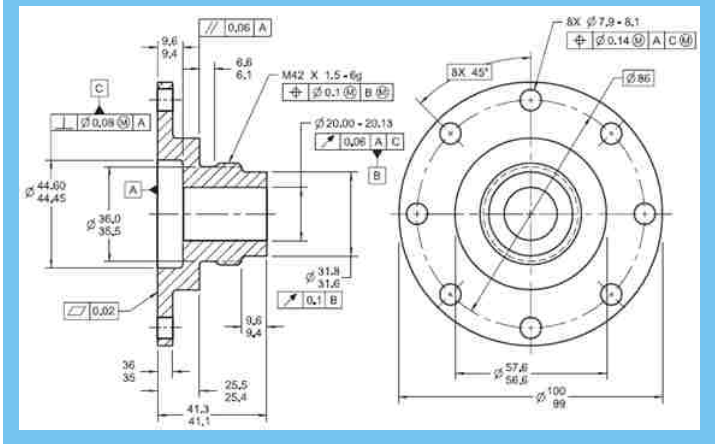
About the course

Benefits

- Improve your work skills at your own pace and convenience
- Learn-by-doing enhance learning experience
- Acquire practical tools that you can apply on the job
- Network with others in your field
- Earn 2.25 CEUs

Target Audience

The course is suited for beginners with minimal GD&T experience & experienced professional who are looking to get a refresher in ASME GD&T 2009 standard. The course material will also provide questions developed to prepare individuals for Geometric Dimensiong & Tolerancing Professional (GDTP) Technologist Level certification.



Course Outline

This course provides the most comprehensive GD&T knowledge and practices used in industry. The theoretical and practical concepts of each of the geometric controls are explained relative to the areas of design, tooling, production, and inspection.

Instructor Profile

Raj Verma is Dimensional Management Process Scientist in Dimensional Management group at Spirit Aero systems, an aerospace company based in Kansas, US. He has broad variation analysis experience of 15 years in automotive, aerospace, and biomedical industry. Raj is also recognized as a Senior Geometric Dimensioning & Tolerancing Professional (GDTP-S) by ASME and holds certification in Six Sigma Black Belt (SSBB). Raj has extensively used DM (Vis-VSA and Sigmund) software to conduct up-front assembly build analysis to predict assembly build variation and eliminate build issues in manufacturing. He has also been a member of the design team for the development of Sigmund products and is a co-patent holder in Tolerance Simulation Analysis. Raj earned his MS in Industrial Engineering from University of Oklahoma, US.

For Nominations and Enquiries, PI contact :

Mr. S. D. Puranik, Executive Director
CII Naoroji Godrej Centre of Excellence, Godrej Station-Side Colony,
Opp. Railway Station, Vikhroli (East), Mumbai – 400 079
Tel: (022) 2574 5146 / 5148 Fax : (022) 25743361
Email: s.d.puranik@cii.in / ciicoe@vsnl.net

Participation Fee - Rs. 30000/- (on non-residential basis) + 10.30% towards service tax i.e. a total of Rs. 33090/-. Payment to be drawn in favour of 'Confederation of Indian Industry' payable at Mumbai.

Course content

Introduction to GD&T
1. Reasons to use GD&T
2. GD&T symbols
3. Conventional v/s Advanced Tolerancing Methods
4. Use of Basic Dimensions

Concepts of GD&T
1. Features of Size
2. Actual Envelopes (Related and Unrelated)
3. Material Conditions of Features of Size
4. Individual Features of Size (Rule # 1)
5. Inner and Outer Boundary, Virtual Condition Calculations

Datums Introductory concepts
1. Degrees of Freedom
2. Datum Feature Simulators
3. Datum Precedence
4. Multiple Datum Features
5. Datum Feature Selection
6. Datum Targets

Datums Advanced concepts
1. Affect of Modifiers on datums
2. Hole Pattern as Datum Features
3. Simultaneous Requirement
4. Free state v/s Restrained datums
5. Movable datum targets
6. Customized Datum Reference Frames

Form Tolerances
1. Flatness
2. Straightness
3. Circularity
4. Cylindricity
5. Tolerance zones, Application and Inspection methodology for Form tolerances

Orientation Tolerances
1. Perpendicularity
2. Parallelism
3. Angularity
4. Tolerance zones, Application and Inspection methodology for Form tolerances

Profile Tolerances
1. Line and Surface Profile
2. Profile as General Requirement
3. Unequally Disposed Profile tolerances
4. Unilateral Profile tolerances
5. Composite Profile tolerances
6. Coplanarity
7. Profile per Unit length
8. Tolerance zones, Application and Inspection methodology for Form tolerances

Location Tolerances
1. Position, Symmetry and Concentricity
2. Composite Position tolerancing
3. Projected Tolerance zone
4. Tolerance zones, Application and Inspection methodology for Location tolerances

Runout Tolerances
1. Circular and Total Runout
2. Tolerance zones, Application and Inspection methodology for Runout tolerances

GDTP Technologist Body of Knowledge covered in the course

Scope, Definitions, and General Dimensioning
1. Y14.5 background information.
2. Definitions required for certification exam.
3. Fundamental rules
4. Measurement units
5. Types of dimensioning
6. Application of dimensions
7. Dimensioning features (diameters, radii, chords, arcs, angles, holes (round, slotted, counterbored, countersunk), spotfaces, chamfers, keyseats
8. Location of features

General tolerancing and related principles
1. Application of tolerances
2. Tolerance expression
3. Interpretation of limits
4. Plated and Coated parts
5. Single Limits
6. Tolerance accumulation
7. Limits of size
8. Relationship between features
9. Applicability of RFS, MMC, and LMC
10. Geometric application to screw threads, gears, and splines
11. Virtual / Resultant condition
12. Datum features at virtual condition
13. Angular surfaces, Conical and Flat tapers.
14. Statistical tolerancing identification

Symbology
1. Fourteen geometric characteristic symbols
2. All symbols of GD&T language in ASME standard

Datum referencing
1. Definitions
2. Immobilization of part
3. Datum feature identification
4. Datum feature controls
5. Datum feature order of precedence
6. Establishing datums from datum features
7. Datum targets

Tolerances of location
1. Position tolerancing
2. Feature Pattern location
3. Bi-directional positional tolerancing of features
4. Non-circular features
5. Coaxiality controls
6. Concentricity
7. Positional Tolerancing for symmetrical features
8. Symmetry tolerancing
9. Spherical features

Tolerances of form, profile, orientation, and runout
1. Form tolerances Straightness, Flatness, Circularity, Cylindricity
2. Orientation Angularity, Parallelism, Perpendicularity
3. Profile Line, Surface
4. Runout Circular, Total

GDTP Certification

Technologist Level:

Technologist Geometric Dimensioning and Tolerancing Professional (GDTP) Certification, achieved by passing a computer-based, multiple choice examination, provides an objective measure of an individual's ability to understand drawings which have been prepared using the language of Geometric Dimensioning and Tolerancing (GD&T), as defined in the ASME Y14.5 Standard.

The ASME GDTP Technologist symbol is for the sole use of those individuals who have demonstrated the required qualifications in accordance with the ASME Y14.5.2 Standard for the Certification of Geometric Dimensioning and Tolerancing Professionals (GDTP). The symbol was developed for the benefit of those who met the qualifications, and for recognition of their achievement within the field of engineering.

Venue: **CII Naoroji Godrej Centre of Excellence, Godrej Station-Side Colony, Vikhroli (East), Mumbai – 400 079**

ASME
SETTING THE STANDARD

Venue: CII Naoroji Godrej Centre of Excellence, Godrej Station-Side Colony, Vikhroli (East), Mumbai – 400 079