



IFS Academy

Training for the future!!!

Geometric Dimensioning & Tolerances: Course Curriculum

- Importance and review of assembly and its functioning (Fits).
- Review of Manufacturing Cycle and Cost - Design, Manufacturing & Inspection - Tools Used & Important Parameters of the Processes (speed, feed, CMM programming)
- Need for tolerances - How they are and how they should be decided? (including statistical methods)
- Tolerance stack & process Planning.
- Co-ordinate method of tolerancing and its disadvantages.
- GD&T and its advantages - How to relax tolerances using GD&T.
- Types of tolerances - Size & Position.
- Different types of form tolerances & symbols: Straightness, Flatness, perpendicularity, roundness, cylindricity, Runout, concentricity, parallelism.
- Methodology of specifying tolerances - Feature Control Frame.
- Design Intent & GD&T - How to specify and interpret GD&T on models & Drawings (Pro/E WF, SolidWorks, CATIA).
- Interpretation of GD&T in Manufacturing & Inspection (How to 'read' GD&T).
- Important concepts in GD&T - Feature, Feature of Size, MMC & LMC, Tolerance zone, Virtual condition, Taylor's Principle, Bonus Tolerance.
- Importance of part location and Datum - DOF
- Introduction to functional gaging.
- Introduction to Computer Aided Inspection & modern inspection technologies & equipment.
- Introduction to Standards - ASME 14.5 (2D) & 14.41 (3D), ISO

Copyright 2010, IFS Academy, Pune

Phone: +91-20-6400 7296, Mob. No.: +91-98228 49628, Email: training@infinitesolutionsindia.com,

Visit Us At: www.ifsacademy.org