

ME450: GEOMETRIC DIMENSIONING AND TOLERANCING
CREDITS = 5 (L=3, T=0 P=2)

Course Objective:

To apply the concepts of GD&T for design, manufacturing and inspection.

Teaching and Assessment Scheme:

Teaching Scheme			Credits	Assessment Scheme				
L	T	P	C	Theory		Practical		Total Marks
				ESE	CE	ESE	CE	
3	0	2	5	70	30	30	20	150

Course Contents:

Unit No.	Topics	Teaching Hours
1	<u>Introduction:</u> Geometric product definition principles; verification of position with open set-up; geometric characteristic symbols Geometric Dimensioning and Tolerancing: an explanation of tolerance zone conversion; surfaces, features, features of size, datum features, datum features of size, and datum's; tolerances; components common to geometrically dimensioned & toleranced drawing; fits & allowances, advantages of GD&T	10
2	<u>MMC, LMC & RFS:</u> Maximum Material Condition (meaning & use); Least Material Condition (meaning & use); Regardless of Feature Size How to read a Feature Control Frame.	06
3	<u>Size Control Form:</u> The Taylors principle; Gauging size limits. Rules, concepts, Characteristics, and Untoleranced Dimensions: individual or related Datum's, Material Conditions; untoleranced dimensions	06
4	<u>Datums:</u> Datum features; oddly configured & curved surfaces as datum features; equalizing datum's; datum feature symbols; flexible parts; direct vs indirect tolerancing. MMC and its ramifications. Relations between individual features. Virtual Condition and Resultant condition Boundaries: Virtual condition (MMC concept & a functional boundary). Effect of LMC; wall thickness calculation.	05

Unit No.	Topics	Teaching Hours
5	<u>Datum Feature of Size Representation:</u> Modes of datum feature representation; angular orientation. Form Controls: flatness; straightness: circularity; free state variation; circularity Orientation Controls: orientation characteristics; angularity; perpendicularity Profile; line element controls Run out: circular & total Location: concentricity; the return of symmetry; position	05
6	A Logical Approach to part Tolerancing Dimensioning and Tolerancing Schemes Steps for the Development of a Dimensional Inspection Plan Paper Gauging and Functional Gauging	10
TOTAL		42

List of References:

1. James D Meadows, “*Geometric Dimensioning and Tolerancing*”, Marcel Dekker, Inc
2. James D Meadows, “*Measurement of Geometric Tolerances in Manufacturing*” Marcel Dekker, Inc
3. P S Gill, “*Geometric Dimensioning and Tolerancing*”, S K Kataria & sons, 2005-6

Course Outcomes (COs):

At the end of this course students will be able to

1. Contrast between conventional and GD&T tolerance zones
2. Explain MMC, LMC and RFS concepts
3. Explain Taylor’s principle of gauging
4. Assess the significance of selection of datum & datum features
5. Point out form, orientation, profile, runout and orientation controls
6. Explain the use of paper and functional gauging