

4ME59: GEOMETRIC DIMENSIONING AND TOLERANCING
CREDITS – 4 (LTP: 3, 0, 1)

Course Objective:

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

Teaching and Assessment Scheme:

Teaching Scheme (Hours per week)			Credits	Assessment Scheme				Total Marks
L	T	P		Theory Marks		Practical Marks		
			ESE	CE	ESE	CE		
3	0	2	4	60	40	20	30	150

Course Contents:

Unit No.	Topics	Teaching Hours
1	Introduction: Geometric product definition principles; verification of position with open setup; 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	MMC, LMC & RFS: Maximum Material Condition (meaning & use); Least Material Condition (meaning & use); Regardless of Feature Size How to read a Feature Control Frame	06
3	Size Control Form: The Taylors principle; Gauging size limits. Rules, concepts, Characteristics, and Untoleranced Dimensions: individual or related Datum's, Material Conditions; untoleranced dimensions	06
4	Datums: 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
5	Datum Feature of Size Representation: 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