

**ME493: 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	Marks Distribution				Total Marks
L	T	P		Theory Marks		Practical Marks		
			ESE	CE	ESE	CE		
3	0	2	5	70	30	30	20	150

**Course Contents:**

Unit No.	Topics	Teaching Hours
1	<p><b><u>Introduction:</u></b></p> <p>Geometric product definition principles; verification of position with open set-up; geometric characteristic symbols</p> <p>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 &amp; toleranced drawing; fits &amp; allowances, advantages of GD&amp;T.</p>	10
2	<p><b><u>MMC, LMC &amp; RFS:</u></b></p> <p>Maximum Material Condition (meaning &amp; use); Least Material Condition (meaning &amp; use); Regardless of Feature Size How to read a Feature Control Frame.</p>	06
3	<p><b><u>Size Control Form:</u></b></p> <p>The Taylors principle; Gauging size limits. Rules, concepts, Characteristics, and Untoleranced Dimensions: individual or related Datum's, Material Conditions; untoleranced dimensions.</p>	06
4	<p><b><u>Datums:</u></b> Datum features; oddly configured &amp; 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.</p> <p>Virtual Condition and Resultant condition Boundaries: Virtual condition (MMC concept &amp; a functional boundary). Effect of LMC; wall thickness calculation.</p>	05

<b>Unit No.</b>	<b>Topics</b>	<b>Teaching Hours</b>
5	<b><u>Datum Feature of Size Representation:</u></b> 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
<b>TOTAL</b>		<b>42</b>

**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