

SPRINGFIELD TECHNICAL COMMUNITY COLLEGE

ACADEMIC AFFAIRS

Course Number: MECH 226 Department: Mechanical Engineering
Tech.
Course Title: Metrology and Geometrics Semester: Spring Year: 1999

Objectives/Competencies

Course Objective	Competencies
1.Discuss the fundamental concepts of Geometric Dimensioning.	1.Discuss the three types of dimensions used on an engineering drawing. 2.Discuss the advantages of GD&T. 3.Differentiate between "feature" and "feature of size." 4.Recognize a location dimension. 5.Discuss the three fundamentals rules in geometric tolerancing. 6.Define Virtual condition. 7.Discuss functional dimensioning.
2.Understand the five categories of Geometric Characteristics.	1.Discuss Form tolerance and how it applies to geometric tolerancing. Identify which symbols are associated with form tolerancing. 2.Discuss Orientation tolerance and how it applies to geometric tolerancing. Identify which symbols are associated with orientation tolerancing. 3.Discuss Profile tolerance and how it applies to geometric

Course Objective	Competencies
	<p>tolerancing. Identify which symbols are associated with profile tolerancing.</p> <p>4. Discuss Runout tolerance and how it applies to geometric tolerancing. Identify which symbols are associated with runout tolerancing.</p> <p>5. Discuss Location tolerance and how it applies to geometric tolerancing. Identify which symbols are associated with location tolerancing.</p>
<p>3. Identify and use Datums.</p>	<p>1. Discuss how to establish datums and datum precedence.</p> <p>2. Describe the difference between functional and non-functional datums.</p> <p>3. Understand how nonfunctional datums are selected with regard to the manufacturing process.</p> <p>4. Discuss primary, secondary, and tertiary datums.</p> <p>5. Understand how inspection equipment is brought in contact with a part datum feature to establish datum points, lines, or planes for the purpose of part inspection.</p>
<p>4. Create and use Feature Control Frames and Datums.</p>	<p>1. Understand the concepts of the Feature Control Frame.</p> <p>2. Understand the symbology for specifying Datums.</p> <p>3. Apply the ASME Y14.5M - 1994 Standards when using Datum Features.</p> <p>4. Understand where to place Feature Control Frames on a drawing.</p> <p>5. Understand where to place Datum.</p>
<p>5. Apply Tolerance of Form.</p>	<p>1. Understand the concept of Flatness tolerance and the</p>

Course Objective	Competencies
<p>6. Apply Tolerances of Orientation.</p>	<p>flatness tolerance zone.</p> <ol style="list-style-type: none"> 2. Understand the Material Condition Symbols - Maximum Material Condition (MMC), Least Material Condition (LMC), and Regardless of Feature Size (RFS) and how they apply to features subject to variation in size. 3. Understand the concept of Straightness tolerance and how it is affected by MMC, and RFS. 4. Measure Straightness tolerances using direct and indirect measurement methods. 5. Understand the concept of Circularity and how it differs from Cylindricity. 6. Verify cylindricity using total runout to centers. <ol style="list-style-type: none"> 1. Understand the various Parallelism tolerance zones for different applications. 2. Explain how parallelism of a surface is not additive to associated size tolerances. 3. Understand the various perpendicularity tolerance zone for a given surface or axial application. 4. Identify for the perpendicularity of plan surfaces, the importance of a secondary alignment. 5. Understand how vertical measurement systems work to measure perpendicularity. 6. Understand the various Angularity tolerance zones. 7. Use a sine bar or angle plate in conjunction with gage blocks to inspect Angularity.
<p>7. Apply Tolerances of Runout.</p>	<ol style="list-style-type: none"> 1. Understand the differences between Circular Runout and

Course Objective	Competencies
<p>8. Apply Tolerances of a profile.</p>	<p>Total Runout. 2. Inspect circular runout on a diameter or an end surface. 3. Inspect circular runout on a cone. 4. Understand the impact that Circular runout has on Roundness and Concentricity.</p> <p>1. Understand the concept of Profile of a line Measurement. 2. Understand the difference between Profile of a line element and the Profile of a surface Tolerance. 3. Identify the differences between Profile controls that use datums and those that do not. 4. Understand why basic dimensions are required to define the true profile. 5. Understand how to use mechanical gaging for profile measurement.</p>
<p>9. Apply Tolerances of Location.</p>	<p>1. Understand how Positional tolerances effect the location of holes, pins, bosses and tabs. 2. Understand the use of datums and basic dimensions to define Positional Locations. 3. Understand the use of MMC and LMC when applied to Positional Locations and the effects they have on location with respect to Bonus tolerances. 4. Understand Symmetry and how it applies to manufactured parts. 5. Understand Concentricity and how it applies to features such as holes, cylinders, cones, and hexagons that share a common axis or center.</p>

