MACH A175: DIMENSIONING AND TOLERANCING

Item	Value
Top Code	095630 - Machining and Machine Tools
Units	3 Total Units
Hours	54 Total Hours (Lecture Hours 54)
Total Outside of Class Hours	0
Course Credit Status	Credit: Degree Applicable (D)
Material Fee	No
Basic Skills	Not Basic Skills (N)
Repeatable	No
Grading Policy	Standard Letter (S)

Course Description

A course in reading and using Reference (SME Y-14.5-1994) standards for geometric dimensioning and tolerancing. ADVISORY: MACH A120. Transfer Credit: CSU.

Course Level Student Learning Outcome(s)

1. Identify Geometric Dimensioning and Tolerancing symbols according to ASME – Y1M, 1994 standard.
2. Identify the correct usage and placement of datum on an orthographic engineering drawing.
3. Calculate tolerances for a fixed fastener problem and for a floating fastener problem and correctlyplace them in a feature control frame.
4. Convert a conventional engineering drawing to a Geometric Dimensioning and Tolerancing formatand determine on the correct tolerances for all the associated feature control frames.

Course Objectives

- I Recognize the importance of a datum to dimensioning practices.
- II Apply MMC, LMC and RFS to established dimensions.
- III Define a projected tolerance zone.
- IV Describe how a datum is positioned for measurement purposes.
- V Describe and measure the following part forms:
- V. 1. Roundness
- V. 2. Flatness
- V. 3. Straightness
- V. 4. Perpendicularity
- V. 5. Angularity
- V. 6. Parallelism
- V. 7. Profile
- V. 8. Runout
- V. 9. True position
- VI Interpret the application of a form tolerance to a part and calculate the possible part variance.
- VII Explain which measuring tool and process should be used for each form.
- VIII Describe the accuracy requirements for the measurements to be made.
- IX Calculate free state variation and average diameter.

Lecture Content

History of Coordinate System Definitions Datum Tolerances Positional tolerance theory and comparison with coordinate tolerance Bonus tolerance (MMC RFS) Fixed and floating fastener conditions Zero positional tolerance Combination and composite tolerances Projected tolerance zone Symmetry non circular features Coaxial controls Tolerance of Form General rules, perfect form @ MMC and virtual condition Straightness Line elements Axes Flatness Roundness and cylindricity Perpendicularity, the five interpretations Angularity Surface Axes Parallelism, the three interpretations Profile tolerances Surface Line Coplanarity Runout Circular Total Free state variation and average diameter

Method(s) of Instruction

- Lecture (02)
- DE Live Online Lecture (02S)

Instructional Techniques

Lecture

Reading Assignments

.

Writing Assignments

Short answer quizzes

Out-of-class Assignments

.

Demonstration of Critical Thinking

Quizzes, tests, class participation, and final examination

Required Writing, Problem Solving, Skills Demonstration

Short answer quizzes

Textbooks Resources

1. Required Foster, Lowell W.. Geometrics III, latest ed. New York: Prentice-Hall, 1994 Rationale: -

Other Resources

1. Selected handout materials provided and distributed by the instructor.