COMMON COURSE OUTLINE: Course discipline/number/title: PMT 1855: Introduction to Geometric Dimensioning and Tolerancing

A. CATALOG DESCRIPTION
1. Credits: 1
2. Hours/Week: 1
3. Prerequisites (Course discipline/number): None
4. Co-requisites (Course discipline/number): None
5. MnTC Goals (if any): NA

This course introduces the fundamental concepts of geometric dimensioning and tolerancing (GD&T) and describes the main types of tolerances.

B. DATE LAST REVISED (Month, year): April, 2009

C. OUTLINE OF MAJOR CONTENT AREAS:
1. What Is GD&T?
2. Background of GD&T
3. Datums and Features
4. GD&T vs. Coordinate Tolerancing
5. The Datum Reference Frame
6. The Order of Datums
7. Types of Tolerances
8. Straightness and Flatness
9. Circularity and Cylindricity
10. Profile of a Line and Surface
11. Angularity, Perpendicularity, and Parallelism
12. Position
13. Concentricity and Symmetry
14. Circular and Total Runout
15. Material Condition Modifiers
16. Bonus Tolerance
17. The Feature Control Frame
18. Advantages of GD&T

D. LEARNING OUTCOMES (GENERAL): The student will be able to:
Define GD&T, and correctly use GD & T to manufacture parts to required tolerances.

E. LEARNING OUTCOMES (MNTC): NA

F. METHODS FOR EVALUATION OF STUDENT LEARNING:
1. Online learning
2. Successful completion of online lessons
3. Successful completion of online assessments

G. SPECIAL INFORMATION (if any):
All PMT courses ending with a 5 will be online classes provided by Tooling U. Please see link below: