DE ANZA COLLEGE
APPLIED TECHNOLOGY DIVISION
COURSE OUTLINE

Degree Applicable

MANUFACTURING AND CNC TECHNOLOGY 72 Effective Quarter Fall 2004

I. Catalog Information

MCNC 72 Applied Geometric Inspection Dimensioning and Tolerancing (ASME Y14.5M); Coordinate Measuring Machines (CMM) 3 Units

Advisories: English Writing 100B and Reading 91 (or Language Arts 100), or English as a Second Language 4; experience in blueprint reading

Six hours lecture-laboratory

Interpretation of specifications and inspection procedures related to current ASME Y 14.5 Geometric Dimensioning and Tolerancing (GD&T) standards. Applications and capabilities of precision measuring tools, including the computer-aide Coordinate Measuring Machine (CMM), used in manufacturing environments to inspect discrete complex parts. Machine and inspected part set-up for measuring form, orientation, and position callouts.

II. Course Objectives

The student will:

A. Explain the rationale, advantages and imitations of the GD&T system.
B. Interpret various form tolerances that do not have datums.
C. Describe orientation tolerances and methods of inspecting them.
D. Explain the true position tolerance concept and interpret tolerances.
E. Understand the applications of coordinate measuring systems to inspect form, orientation and position callouts of mechanical parts.

III. Essential Student Materials

None

IV. Essential College Facilities

Metrology lab with a computer-aide Coordinate Measuring Machine

V. Expanded Description: Content and Form

A. Introduction to ISO and ASME Y 14.5M Geometric Dimensioning and Tolerancing (GD&T)
   1. System rationale, advantages, limitations and relationship to function
   2. Geometric dimensioning and tolerancing characteristics and symbols
   3. Maximum and minimum material principle
   4. Datums
   5. Standard rules
   6. Distinction between profile, form, orientation, runout and location tolerances

B. Form tolerances without datums
   Interpretation, application and methods of inspection for each of the following:
   1. Flatness
   2. Straightness
      a. Surface elements

   b. Axis, with and without modifiers
3. Roundness (circularity)
   a. Cylinders
   b. Cones
   c. Spheres
4. Cylindricity

C. Orientation tolerances
   Interpretation, application and methods of inspection for each of the following:
   1. Datum definitions and applications
   2. Parallelism
      a. Surface, cylindrical size feature and axis
      b. Effect of modifiers
   3. Perpendicularity
      a. Surfaces, cylindrical size feature and axis
      b. Effect of modifiers
   4. Angularity
      a. Surface
      b. Axis
   5. Profile
      a. Surface
      b. Surface all around
      c. Coplanar surfaces
   6. Runout
      a. Circular
      b. Total
      c. Axis, diameter and face datums

D. Location tolerance (true position)
   1. Concept, advantages and disadvantages
   2. Three plane concept of datums
      a. Specified datums
      b. Implied datums
   3 . Position tolerance
      a. For holes in relationship to feature size
      b. For non- cylindrical features
      c. Or mating parts
      d. For coaxial gages
      e. Non-cylindrical gages
   4. Position and coordinate tolerance zone conversions
   5. Concentricity
   6. Symmetry

E. Coordinate measuring machines
   1. Types of coordinate measuring machines
      a. Utility grade
      b. Direct computer controlled
      c. High accuracy
      d. Sources of error
   2. Structure of the coordinate measuring machine
      a. Scales
      b. Axis
      c. Surface plate
      d. Environment
   3. Probing systems
      a. Solid probes

   b. Touch trigger
   c. Analog probes
   d. Sources of error
4. Scanning vs point to point measurement, datum simulation and assembly
5. Alignment
   a. Right hand rule
   b. Primary secondary and tertiary datum alignment
6. Measuring form
   a. Flatness
   b. Straightness
   c. Circularity
   d. Cylindricity
7. Measuring orientation
   a. Angularity
   b. Perpendicularity
   c. Parallelism
8. Measuring location
   a. Position
   b. Concentricity
9. Measuring profile
10. Assembly of a measurement plan

VI. Assignments

A. Reading from textbook and references
B. Laboratory measurement assignments
C. Measurement plans

VII. Methods of Evaluating Objectives

A. Midterm and comprehensive final exam
B. In class lab assignments
C. Measurement plan assignment

VIII. Texts and Supporting References

A. Texts

1. Foster, Lowell. *Geo-Metrics III*. New York: Addison-Wesley, 1998

B. Reference:
1. Busch, Ted. *Fundamentals of Dimensional Metrology.* New York: Delmar, 1999.
2. ASME. *American Society of Mechanical Engineers Dimensioning and Tolerancing Y14.5M.* New York, ASME, 1994