**BARTON COMMUNITY COLLEGE**

##### COURSE SYLLABUS

## GENERAL COURSE INFORMATION

Course Number: MSCT 1103

Course Title: Blueprint Reading/Geometric Dimensions & Tolerances (GD & T)  
Credit Hours: 2  
Prerequisite: None  
Division and Discipline: Workforce Training/Community Educ/Manufacturing Skills

Course Description: This course provides the study of basic blue print reading and reading of engineering drawings. This course will develop the student’s ability to locate and interpret dimensions in engineering.

###### INSTRUCTOR INFORMATION

###### COLLEGE POLICIES

Students and faculty of Barton Community College constitute a special community engaged in the process of education. The College assumes that its students and faculty will demonstrate a code of personal honor that is based upon courtesy, integrity, common sense, and respect for others both within and outside the classroom.

Plagiarism on any academic endeavors at Barton Community College will not be tolerated. The student is responsible for learning the rules of, and avoiding instances of, intentional or unintentional plagiarism. Information about academic integrity is located in the Student Handbook.

The college reserves the right to suspend a student for conduct that is determined to be detrimental to the College educational endeavors as outlined in the College Catalog, Student Handbook, and College Policy & Procedure Manual. (Most up-to-date documents are available on the College webpage.)

Any student seeking an accommodation under the provisions of the Americans with Disability Act (ADA) is to notify Student Support Services via email at [disabilityservices@bartonccc.edu](mailto:disabilityservices@bartonccc.edu).

## COURSE AS VIEWED IN THE TOTAL CURRICULUM

This course is one course that students complete in the pursuit of attaining the Manufacturing Skills Certification (MSC). This certificate curriculum was developed by the Kansas Institute for Technical Excellence (KITE) colleges in Kansas in collaboration with business and industry representatives within the manufacturing sector from the Central/South Central Kansas region.

## ASSESSMENT OF STUDENT LEARNING

Barton Community College assesses student learning at several levels: institutional, program, degree and classroom. The goal of these assessment activities is to improve student learning. As a student in this course, you will participate in various assessment activities. Results of these activities will be used to improve the content and delivery of Barton’s instructional program.

Course Outcomes, Competencies, and Supplemental Competencies:

1. Describe the background information found on engineering drawings, their role in manufacturing and the concept of multi-view representation of an object.
2. Discuss the purpose of engineering drawings.
3. Distinguish between detailed drawings and an assembly drawing.
4. Locate information in the title block, revision block and parts list.
5. Interpret the drawing scale.
6. Explain why engineering drawings are generally multi-view drawings.
7. Identify and explain the purpose of visible lines and hidden lines.
8. Explain the purpose of dimensions and tolerances.
9. Visualize a part in a multi-view drawing, explain how the principal views are derived through orthographic projection and how to identify related part features from view to view.
10. Identify the six principal views of an orthographic projection.
11. Determine whether a drawing is a third-angle or first-angle projection based on the ISO symbol.
12. Identify related part features in front, top and right side views.
13. Identify and interpret the following views: auxiliary, partial and enlarged.
14. Identify and explain the purpose of center lines.
15. Recognize and interpret common drafting conventions, including: line precedence, break lines, phantom lines and rotation of part features into alignment to avoid foreshortening.
16. Explain the purpose of sectional views, how they are obtained and their different types, along with some of the drafting conventions that are typically applied to sectional views.
17. Determine, from the cutting plane line, which portion of the part is shown in section.
18. Define the purpose of section lines.
19. Identify and interpret different types of sectional views, including: full sections, half sections, offset sections, aligned sections, broken-out sections, auxiliary sections, revolved sections and removed sections.
20. Identify and interpret the common drafting conventions applied to sectional views.
21. Identify dimensions and tolerances, the numeric elements of an engineering drawing, how dimensions and tolerances are commonly indicated on a drawing, and the different tolerancing methods.
22. Identify and explain the purpose of dimensions and tolerances.
23. Identify and explain the purpose of dimension lines, extension lines, leaders and notes.
24. Identify the unit of measure used on a drawing.
25. Identify reference dimensions.
26. Interpret the dimensions of angles, arcs and chords.
27. Identify and define a specified dimension.
28. Recognize and interpret tolerancing methods, including limit dimensioning, plus-and-minus tolerancing, bilateral tolerances and unilateral tolerances.
29. Calculate tolerances correctly.
30. Contrast dimensions and tolerances, dimensioning methods, maximum and least material condition classes of fit between mating parts, some of the symbols used in geometric dimensioning and tolerancing, and in surface finish specifications.
31. Identify and interpret three dimensioning methods: chain, baseline and direct.
32. Explain the origin of tolerance accumulation and state why it is a disadvantage of chain dimensioning.
33. Identify a datum feature and explain its purpose.
34. Define maximum material condition (MMC) and least material condition (LMC) and explain how they apply to internal and external features.
35. Determine whether mating parts will have a clearance, interference or transition fit by calculating allowance.
36. Identify a GD & T feature control frame and explain its basic components.
37. Recognize a surface finish symbol and identify its values.
38. Define roughness, waviness and lay.
39. Explain how the common part features are represented and dimensioned on a drawing.
40. Identify and interpret specifications for the following part features: hole, counterbore, knurl, countersink, keyseat, conterdrill, fillet, spotface, round, slot and screw thread.
41. Discuss the basic terms and principles of geometric dimensioning and tolerancing and why it has become an important engineering tool in manufacturing.
42. Define GD & T and explain how it is different from the conventional coordinate system of dimensioning and tolerancing.
43. Explain the benefits of GD & T.
44. Give examples of a geometric characteristic.
45. Explain how a geometric tolerance zone is different from a conventional tolerance zone.
46. Explain the purpose of a datum and identify datum features on a drawing.
47. Identify and define a basic dimension.
48. Explain the difference between maximum material condition and least material condition as they apply to external and internal part features.
49. Explain the difference between clearance, interference and transition fits.
50. Specify the general format of a GD & T specification and the meaning of the symbols it contains.
51. Explain the format of a feature control frame.
52. Identify the geometric characteristic being controlled.
53. Determine the size and boundaries of a geometric tolerance zone.
54. Determine whether a bonus tolerance is applicable and calculate the resulting geometric tolerance.
55. Determine which datum features are referenced.
56. Differentiate between the two types of geometric characteristics in detail: form tolerances and orientation tolerances.
57. Identify form or orientation tolerance being specified.
58. Define the size and boundaries of the geometric tolerance zone.
59. Determine whether a bonus tolerance is allowed and if so, how large.
60. Describe how each geometric tolerance if verified.
61. Evaluate the three types of geometric characteristics: profile tolerances, location tolerances and run out tolerances.
62. Identify the particular profile, run out, or location tolerance being specified.
63. Define the size and boundaries of the geometric tolerance zone.
64. Determine whether a bonus tolerance is allowed and if so, how large.
65. Describe how each geometric tolerance is verified.

## INSTRUCTOR'S EXPECTATIONS OF STUDENTS IN CLASS

## TEXTBOOKS AND OTHER REQUIRED MATERIALS

### REFERENCES

### METHODS OF INSTRUCTION AND EVALUATION

## ATTENDANCE REQUIREMENTS

## COURSE OUTLINE