



Western Technical College

10606165 Geometric Dimensioning and Tolerancing

Course Outcome Summary

Course Information

Description	This course is designed to introduce the student to the fundamentals of Geometric Dimensioning and Tolerancing (GDT). Emphasis will be placed on how GDT controls proper fit and function between mating parts within an assembly from design to manufacturing through inspection. Terminology, rules and the geometric symbols will be covered. We will be using the coordinate measuring machine and related software to emphasize the GDT concepts by measuring dimension accuracy and geometric controls.
Career Cluster	Science, Technology, Engineering and Mathematics
Instructional Level	Associate Degree Courses
Total Credits	3
Total Hours	90

Textbooks

GD&T: Application and Interpretation. 7th Edition. Copyright 2021. Wilson, Bruce A. Publisher: Goodheart-Wilcox Co. **ISBN-13**: 978-1-63563-872-1. Required.

GD&T: Application and Interpretation – Study Guide. 7th Edition. Copyright 2021. Wilson, Bruce A. Publisher: Goodheart-Wilcox Co. **ISBN-13**: 978-1-63563-873-8. Required.

Learner Supplies

Scientific calculator (recommend T1-36x Solar). **Vendor**: Campus Shop. Required.

Success Abilities

1. Refine Professionalism: Improve Critical Thinking

Program Outcomes

1. Prepare detail and assembly drawings for documentation of mechanical components and products.
2. Analyze mechanic engineering problems.

3. Produce and revise supporting engineering documentation.
4. Interpret and communicate technical concepts, designs, and documentation.

Course Competencies

1. Develop techniques for dimensioning and tolerancing.

Assessment Strategies

- 1.1. Drawing Application

Criteria

You will know you are successful when

- 1.1. you differentiate dimensioning styles.
- 1.2. you identify drawing tolerances.
- 1.3. you apply dimensions and tolerances to part and assembly drawings.

Learning Objectives

- 1.a. Define each type of dimensioning style.
- 1.b. Apply each type of dimensioning style.
- 1.c. Identify the purpose of title block tolerances.

2. Interpret GD&T symbols and terminology.

Assessment Strategies

- 2.1. Drawing Application

Criteria

You will know you are successful when

- 2.1. you identify geometric tolerance symbols.
- 2.2. you define what each geometric tolerance symbol is used for.
- 2.3. you distinguish each part of a feature control frame.
- 2.4. you apply geometric tolerances to part and assembly drawings.

Learning Objectives

- 2.a. Identify, interpret and define geometric tolerance symbols.
- 2.b. Define each part of a feature control frame.
- 2.c. Apply geometric tolerances to drawings.
- 2.d. Apply geometric symbols in feature control frames.

3. Evaluate the effects of GD&T.

Assessment Strategies

- 3.1. Drawing Application

Criteria

You will know you are successful when

- 3.1. you explain the use of Virtual Conditions of mating parts.
- 3.2. you distinguish features of size and non-features of size.
- 3.3. you apply tolerance zones according to feature control frames on part and assembly drawings.

Learning Objectives

- 3.a. Define the difference between MMC and VC.
- 3.b. Explore the difference between VC of a hole and a shaft.
- 3.c. Describe a feature of size and a non-feature of size.
- 3.d. Apply VC to mating parts on a drawing.
- 3.e. Identify tolerance zones on parts.
- 3.f. Define the allowable part tolerance when VC is applied to mating parts.

4. Interpret and apply form tolerances.

Assessment Strategies

4.1. Drawing Application

Criteria

You will know you are successful when

- 4.1. you identify form tolerance symbols.
- 4.2. you explain how where each form tolerance is allowed.
- 4.3. you define what each form tolerance controls.
- 4.4. you calculate the permissible deviation of the feature the form tolerance is applied to.
- 4.5. you apply form tolerances on part and assembly drawings.

Learning Objectives

- 4.a. Examine each type of form tolerance.
- 4.b. Explore how each form tolerance is used.
- 4.c. Apply form tolerances to parts.
- 4.d. Apply form tolerances to feature control frames.

5. Interpret and label datums.

Assessment Strategies

- 5.1. Drawing Application

Criteria

You will know you are successful when

- 5.1. you define how primary, secondary, and tertiary datums are used
- 5.2. you explain what primary, secondary, and tertiary datums control.
- 5.3. you distinguish the difference in application of datums on surfaces, extension lines, and dimensions.
- 5.4. you apply datums in a feature control frame.
- 5.5. you identify datum usages on part and assembly drawings.

Learning Objectives

- 5.a. Examine how primary, secondary, and tertiary datums are used.
- 5.b. Determine the difference in meaning for the placement of datum identifiers.
- 5.c. Apply datum identifiers to feature control frames.
- 5.d. Apply datum identifiers to parts and dimensions.
- 5.e. Identify how datum order affects part testing.

6. Interpret and apply orientation tolerances.

Assessment Strategies

- 6.1. Drawing Application

Criteria

You will know you are successful when

- 6.1. you identify orientation tolerance symbols.
- 6.2. you explain how where each orientation tolerance is allowed.
- 6.3. you define what each orientation tolerance controls.
- 6.4. you calculate the permissible deviation of the feature the orientation tolerance is applied to.
- 6.5. you apply orientation tolerances on part and assembly drawings.

Learning Objectives

- 6.a. Explore each type of orientation tolerance.
- 6.b. Determine how each orientation tolerance is used.
- 6.c. Apply orientation tolerances to parts.
- 6.d. Apply orientation tolerances to feature control frames.

7. Interpret and apply location tolerances.

Assessment Strategies

- 7.1. Drawing Application

Criteria

You will know you are successful when

- 7.1. you identify location tolerance symbols.
- 7.2. you explain how where each location tolerance is allowed.
- 7.3. you define what each location tolerance controls.
- 7.4. you calculate the permissible deviation of the feature the location tolerance is applied to.
- 7.5. you apply location tolerances on part and assembly drawings.

Learning Objectives

- 7.a. Examine each type of location tolerance.
- 7.b. Determine how each location tolerance is used.
- 7.c. Apply location tolerances to parts.
- 7.d. Apply location tolerances to feature control frames.

8. Interpret and apply profile tolerances.

Assessment Strategies

- 8.1. Drawing Application

Criteria

You will know you are successful when

- 8.1. you identify profile tolerance symbols.
- 8.2. you explain how where each profile tolerance is allowed.
- 8.3. you define what each profile tolerance controls.
- 8.4. you calculate the permissible deviation of the feature the profile tolerance is applied to.
- 8.5. you apply profile tolerances on part and assembly drawings.

Learning Objectives

- 8.a. Examine each type of profile tolerance.
- 8.b. Determine how each profile tolerance is used.
- 8.c. Apply profile tolerances to parts.
- 8.d. Apply profile tolerances to feature control frames.

9. Interpret and apply runout tolerances.

Assessment Strategies

- 9.1. Drawing Application

Criteria

You will know you are successful when

- 9.1. you identify runout tolerance symbols.
- 9.2. you explain how where each runout tolerance is allowed.
- 9.3. you define what each runout tolerance controls.
- 9.4. you calculate the permissible deviation of the feature the runout tolerance is applied to.
- 9.5. you apply runout tolerances on part and assembly drawings.

Learning Objectives

- 9.a. Examine each type of runout tolerance.
- 9.b. Determine how each runout tolerance is used.
- 9.c. Apply runout tolerances to parts.
- 9.d. Apply runout tolerances to feature control frames.

10. Identify inspection types.

Assessment Strategies

- 10.1. Drawing Application

Criteria

You will know you are successful when

- 10.1. you identify at least two types of inspection devices.
- 10.2. you explain how a specific inspection device is used.
- 10.3. you demonstrate the use of at least two inspection devices.

Learning Objectives

- 10.a. Identify inspection devices.

- 10.b. Discuss the use of a specific inspection device.
- 10.c. Demonstrate the use of inspection devices.
- 10.d. Determine which inspection device should be used for an application.