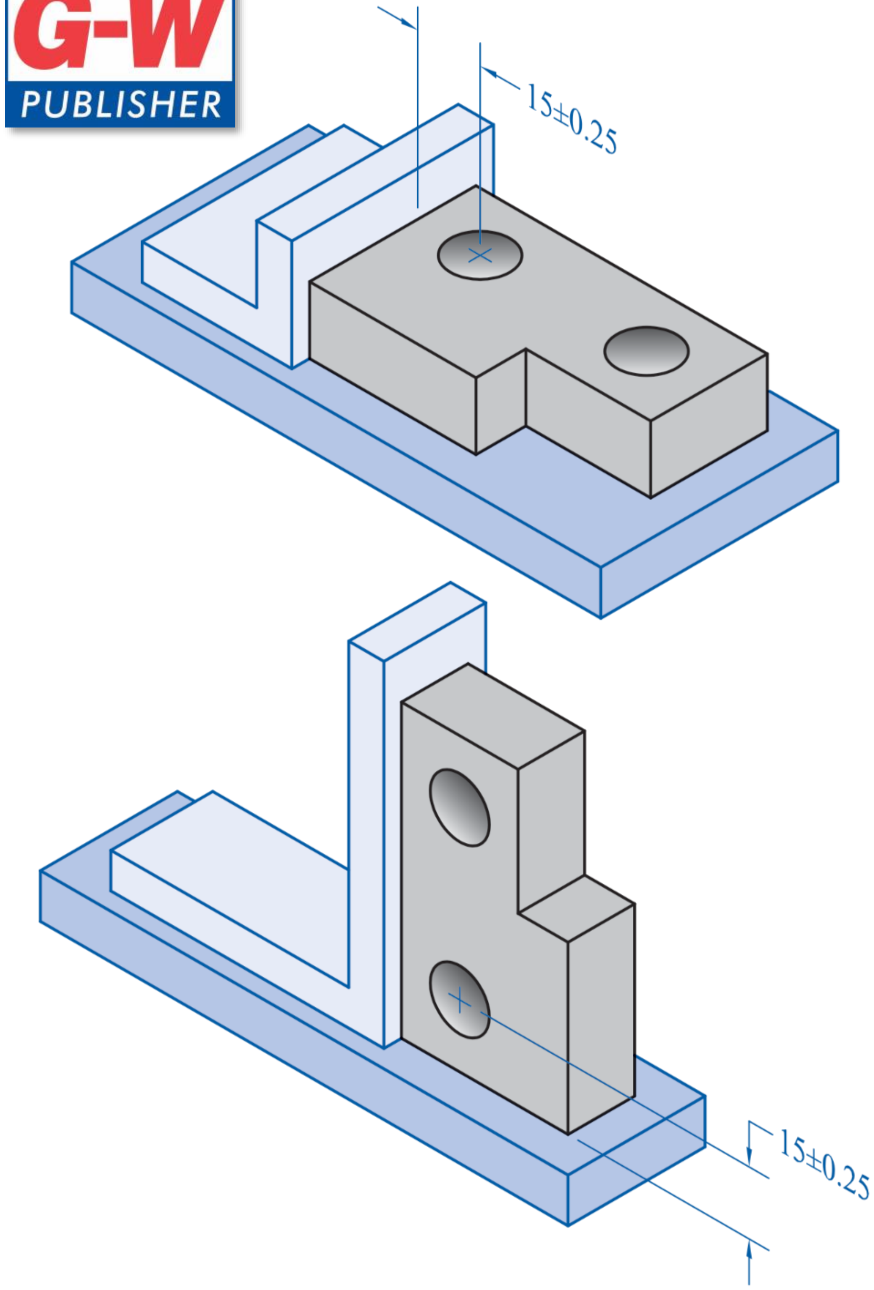
G-W PUBLISHER





# Geometric Dimensioning & Tolerancing:

## *Principles and Practices*


*Geometric Dimensioning and Tolerancing* Instructor Resources: Chapter 1 Answer Key

## Chapter 1: Geometric Dimensioning and Tolerancing Essentials

### Chapter Review

#### Know and Understand

***True/False***

1. True
2. False
3. True
4. False
5. True
6. False
7. True
8. False
9. False
10. True
11. True
12. False
13. False
14. False
15. True
16. False
17. True
18. False
19. True

Chapter 1 Geometric Dimensioning and Tolerancing Essentials

**Multiple Choice**

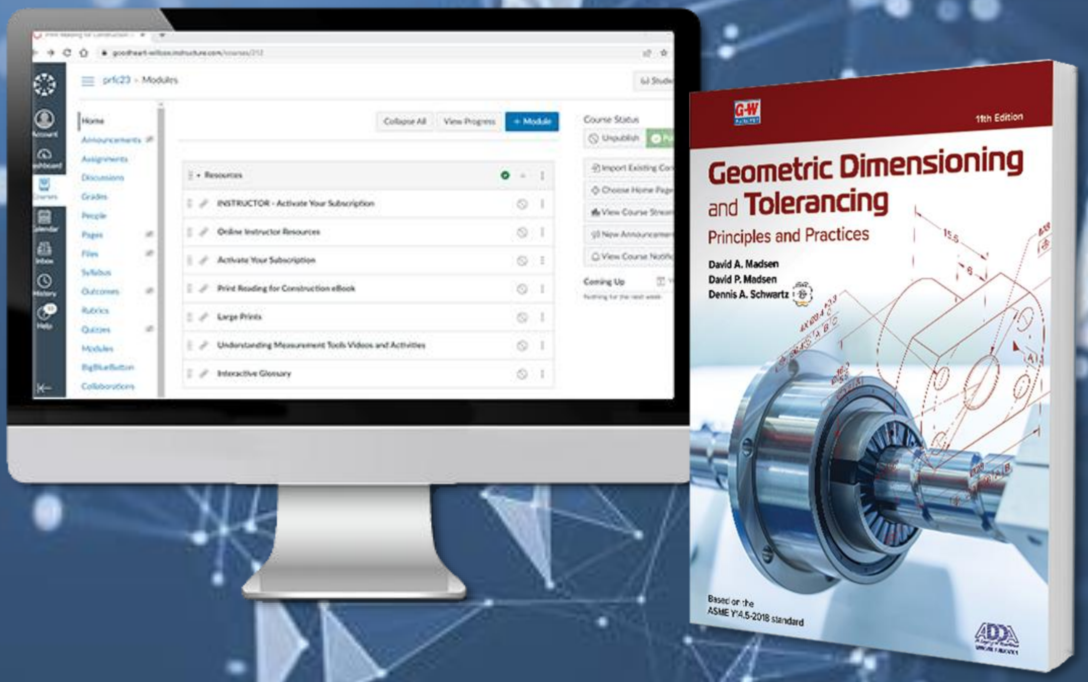
1. Which ASME standard establishes approved dimensioning and tolerancing practices for engineering drawings and related documentation practices?
A. Y14.2
*B. Y14.5
C. Y14.6
D. Y14.8
Title: Which ASME standard establishes approved...
Learning Outcome: 1.1
Reference: gdat11e_2027_exam01_001

2. The total amount a dimension or feature is permitted to vary is called the ____.
A. code
B. basic dimension
*C. tolerance
D. annotation
Title: The total amount a dimension or feature...
Learning Outcome: 1.1
Reference: gdat11e_2027_exam01_002

3. What is the general term that refers to the types of tolerances used to control form, profile, orientation, location, and runout?
A. Descriptive geometry
B. Directly toleranced dimensioning
C. Coordinate tolerancing
*D. Geometric tolerancing
Title: What is the general term that refers to the types of tolerances...
Learning Outcome: 1.1
Reference: gdat11e_2027_exam01_003

4. GD&T relates to design applications and processes used in ____ drafting and manufacturing.
A. architectural
B. civil
*C. mechanical
D. structural
Title: GD&T relates to design applications and processes...
Learning Outcome: 1.1

# Lesson Plans, Assessments, and Answer Keys




# Print • Digital • Bundle Options Available

Blackboard
D2L BRIGHTSPACE
canvas
moodle
schoology
Additional LTI-Compliant Platforms
Clever
ClassLink
1EDTECH CERTIFIED
Google Classroom

# LMS and CMS Integration
# Easy Navigation

G-W PUBLISHER

## 1: E-Flash Cards

Term (1 of 53)

Select to flip

2D drawing

Previous | Remove | Next

## Digital Activities

## 2: Matching Activity

**Match the term with the correct definition.**

- actual local size
- assembly drawing
- datum
- equal bilateral tolerance
- fair curve
- free state
- free state variation
- locational fits
- master layout
- radius
- stock size
- template
- undimensioned drawing
- unequal bilateral tolerance
- unilateral tolerance

A. The measured size of a produced feature or part, obtained by taking any cross-sectional measurement at any two adjacent points.

B. A full-size reproduction defining the true profile of a part.

C. A smooth curve without sharp changes in direction over any portion of the curve's length.

D. A plane, point, line, or axis, or a combination thereof. A datum is theoretically exact and is derived from the true geometric counterpart.

E. The distance from the center of a circle to the outside of the circle.

F. The distortion of a part after removal of forces applied during manufacturing.

G. A drawing of a product showing how the parts of the product fit together. Typically includes a parts list.

H. A tolerance where the variation is permitted in one direction from the true profile or specified dimension.

I. An exact-scale drawing from which the designed part and associated tooling are produced by photography or other processes.

J. The condition of a part in which no external forces are applied except gravity.

K. A group of fits intended to determine only the location of mating parts.

L. A commercial or premanufactured size, such as a particular size of square, round, or hex steel bar.

M. A tolerance where the variation from the true profile or specified dimension is not the same in both directions.

N. A full-size pattern drawing created for fabrication.

O. A tolerance where the variation from the true profile or specified dimension is the same in both the + and − directions.

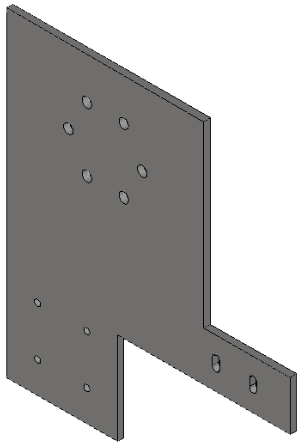
Check Answers

## 3D PDF Prints

Geometric Dimensioning and Tolerancing: Principles and Practices

### Drafting Problem 20

| | |
|---|---|
| NAME | MODULAR CHASSIS PLATE |
| MATERIAL | SAE 30308, 1.5 THICK |

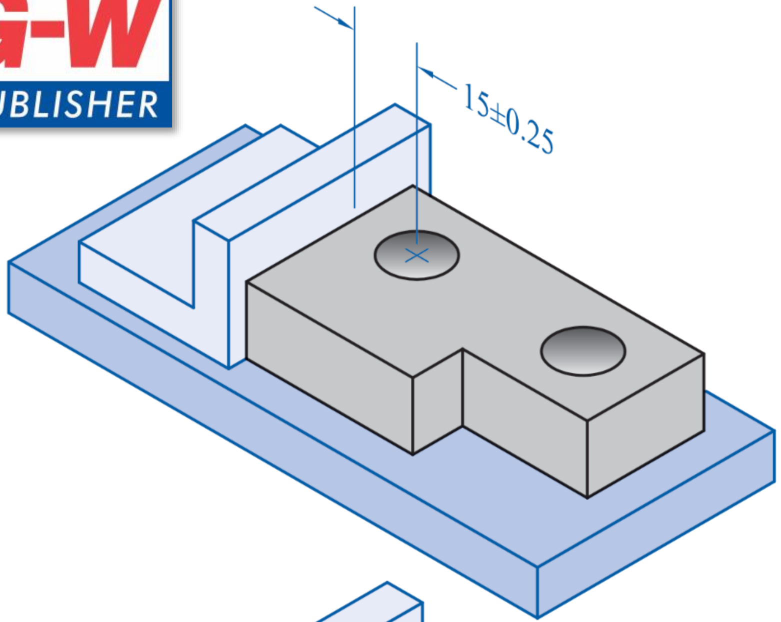


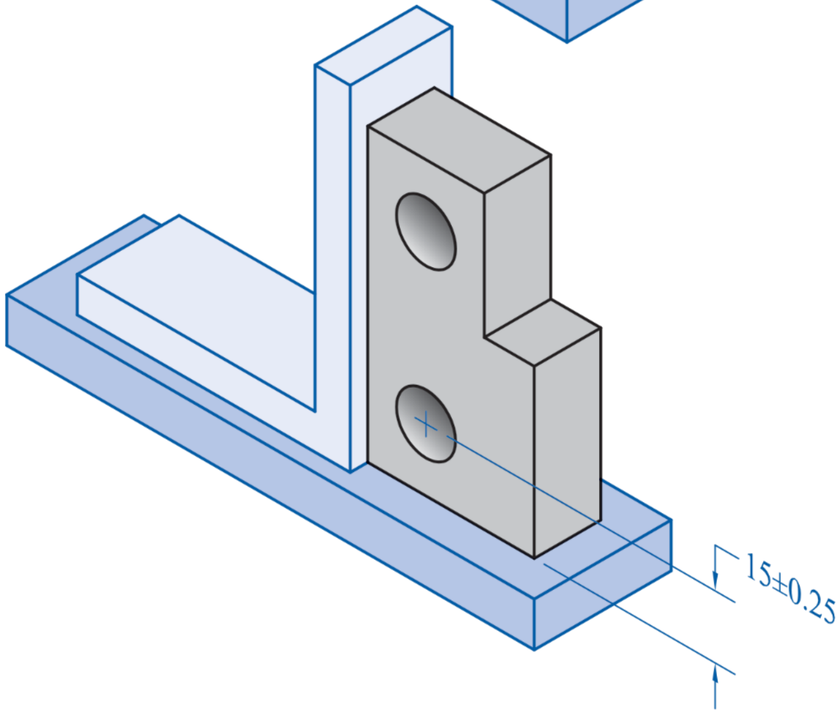
Isometric | Front | Top | Right

G-W PUBLISHER



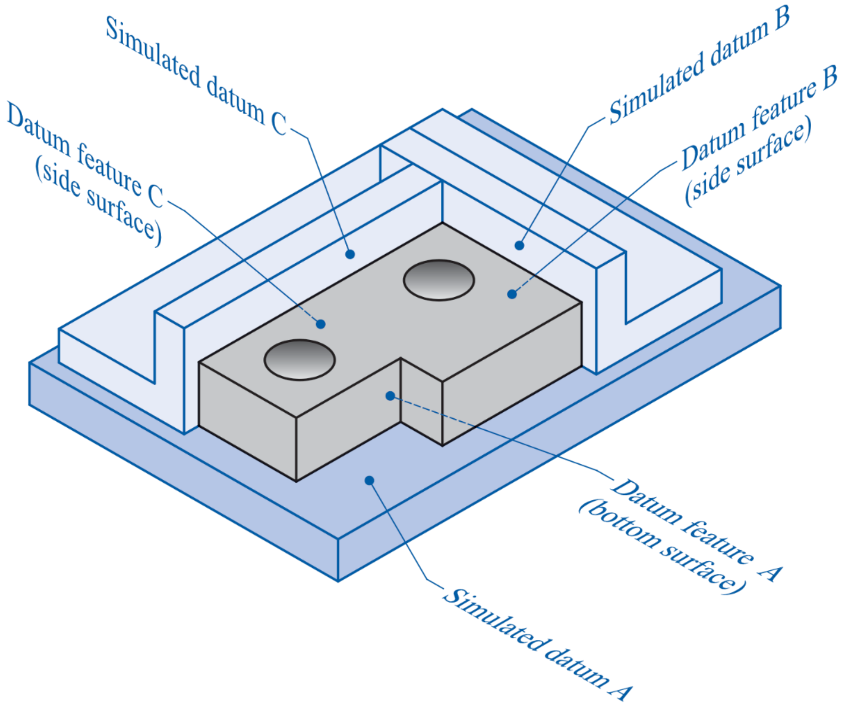
# Integrate G-W Digital Resources



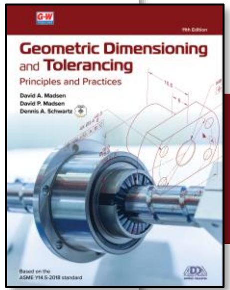


**©2027, 11th edition, by David A. Madsen, David P. Madsen, and Dennis A. Schwartz**




# Brief Contents






**Student Textbook**

G-W PUBLISHER

CHAPTER 2

# Dimensioning and Tolerancing Practices

## LEARNING OUTCOMES

*After studying this chapter, you will be able to:*

**2.1** Interpret dimensions on drawings based on ASME standards.
**2.2** Describe and use common practices for placing metric and inch dimensions.
**2.3** Identify and use fundamental dimensioning rules.
**2.4** Define common dimensioning and tolerancing terms.
**2.5** Describe and use fundamental tolerancing practices.
**2.6** Identify standard types of limits and fits between mating parts.

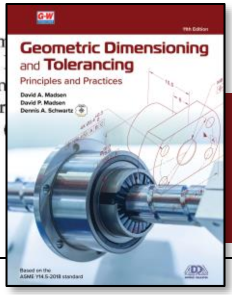
## TECHNICAL TERMS

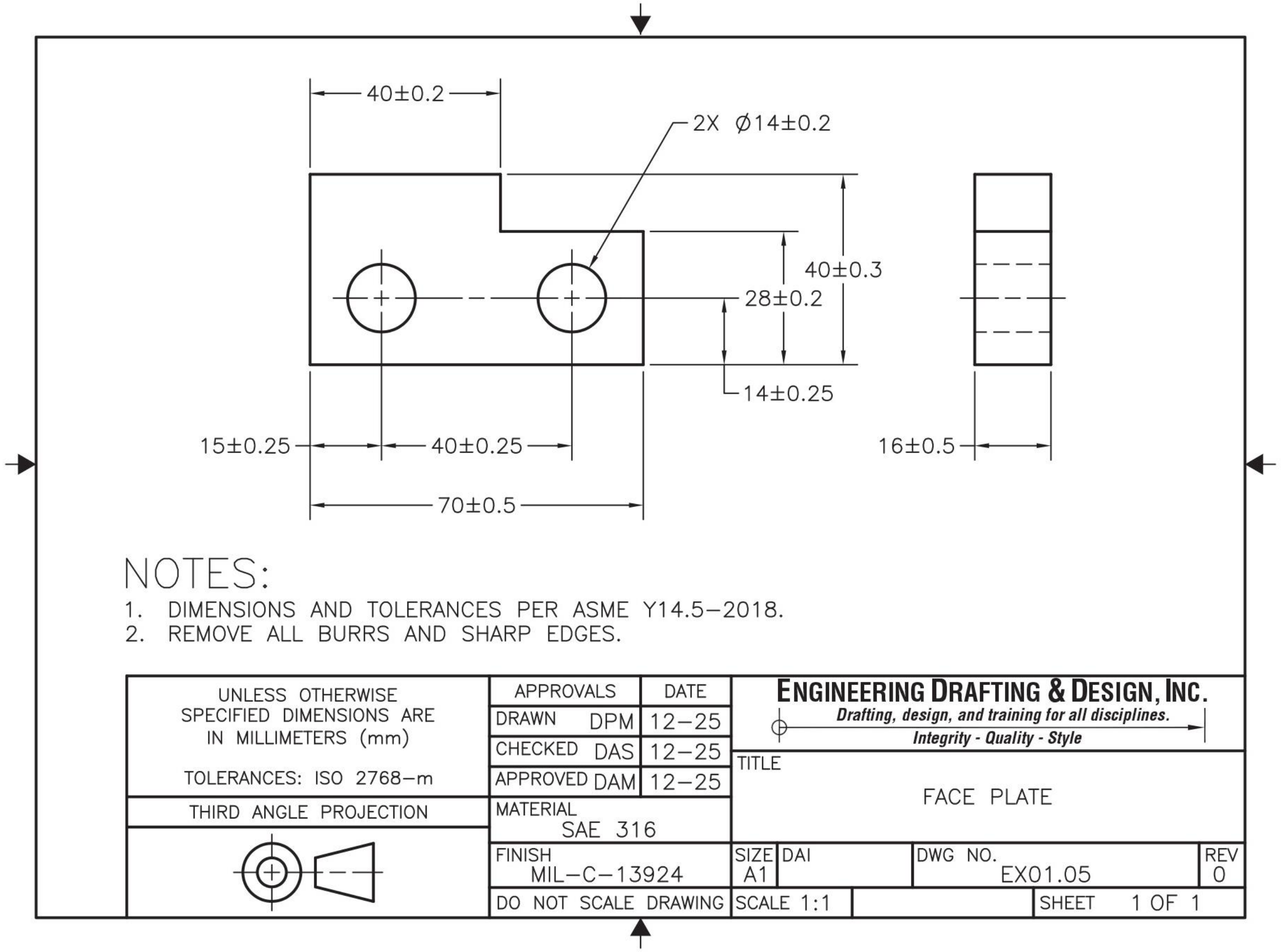
actual local size
actual mating envelope
allowance
assembly drawing
basic dimension
bilateral tolerance
clearance
clearance fits
controlled radius
datum
degree
diameter
equal bilateral tolerance
extreme form variation
fair curve
feature
feature of size
feature without size
force fits (FN)
free state
free state variation
gage
interference fits
irregular feature of size
least material condition (LMC)
limit dimensioning
limits
limits of size
locational fits
loft drawing
master layout
maximum material condition (MMC)
nominal size
nonrigid parts
parts list
plus and minus tolerancing
printed wiring
radius
reference dimension
regular feature of size
running and sliding fits (RC)
shrink fits
single limits
specified dimension
stock size
template
tolerance
transition fits
undimensioned drawing
unequal bilateral tolerance
unilateral tolerance

## Introduction

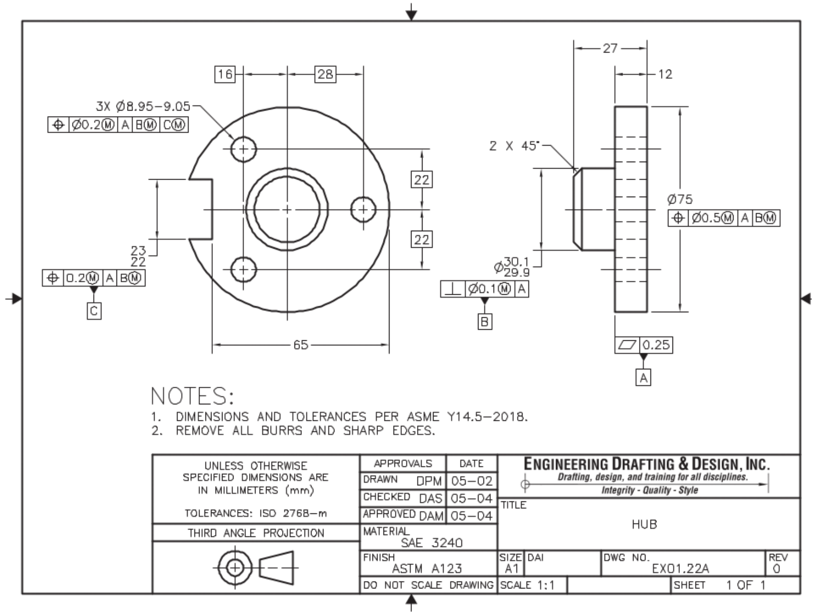
This chapter provides fundamental information about a variety of din erancing practices and systems used in the manufacturing industry. facturing technology student, you will read and interpret information prints. If you are a drafting technology student, you will read industr and dimension two-dimensional (2D) drawings or three-dimensional



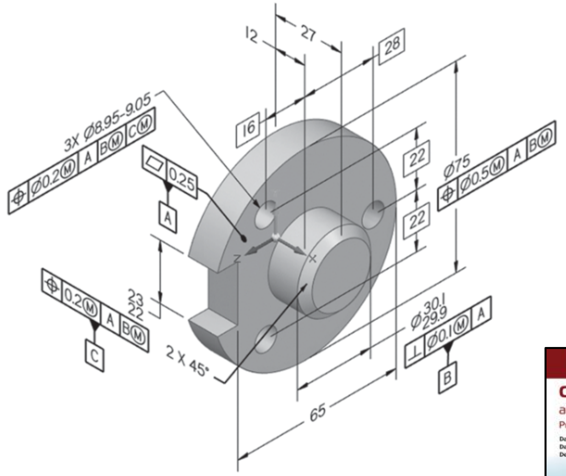



**Chapter Opening Materials**



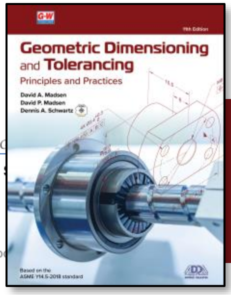

A



B

**Example 1-22.** A—A metric unit 2D multiview drawing of a hub. B—A 3D
the hub with GD&T specifications defined within the model.








dimensioning with directly toleranced dimensions. Coordinate tolerancing uses the rectangular coordinate system to locate the features of a part. In this system, dimensions specify linear distances from the X, Y, and Z axes.

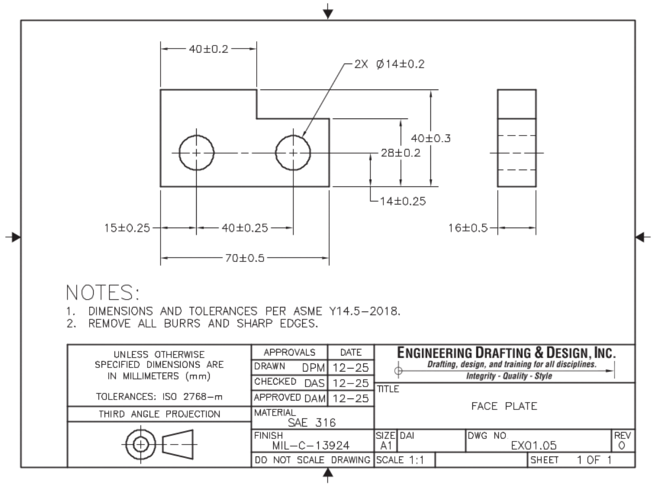
**Example 1-5** shows a metric unit drawing of a face plate dimensioned using coordinate tolerancing. Notice that plus and minus tolerances are directly applied to the dimensions.

| ISO Standard | Title |
|---|---|
| **GPS Standards** | |
| ISO 286-1 | *ISO code system for tolerances on linear sizes—Part 1: Basis of tolerances, deviations and fits* |
| ISO 286-2 | *ISO code system for tolerances on linear sizes—Part 2: Tables of standard tolerance classes and limit deviations for holes and shafts* |
| ISO 1101 | *Geometrical tolerancing—Tolerances of form, orientation, location and run-out* |
| ISO 1660 | *Geometrical tolerancing—Profile tolerancing* |

| ISO Standard | Title |
|---|---|
| **General Tolerance Standards** | |
| ISO 2768-1 | *General tolerances—Part 1: Tolerances for linear and angular dimensions without individual tolerance indications* |
| ISO 2768-2 | *General tolerances—Part 2: Geometrical tolerances for features without individual tolerance indications* |

*Goodheart-Willcox Publisher*

**Example 1-4.** ISO standards related to dimensioning and tolerancing practices.




*Goodheart-Willcox Publisher*

**Example 1-5.** A metric unit drawing of a face plate defined using coordinate tolerancing. The linear dimensions are directly toleranced rectangular coordinate dimensions.



# Visual Examples

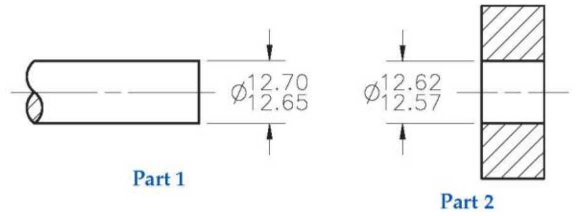
## Force Fit

A force fit is also referred to as an interference fit or a shrink fit. This is where two mating parts must be pressed or forced together. Because of the tolerance on each part, the shaft is larger than the hole, as shown in **Example 2-18**. At any produced size within the stated tolerance, the shaft is larger than the hole. The smallest amount of interference is:

LMC SHAFT = 12.65
–LMC HOLE = 12.62
MIN INTERFERENCE = 0.03

The greatest amount of interference is:

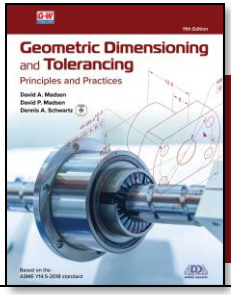
MMC SHAFT = 12.70
–MMC HOLE = 12.57
MAX INTERFERENCE = 0.13




**Example 2-18.** Force fit between two parts.

**NOTE**

Drawings are dimensioned and toleranced until there is a full understanding of the characteristics of each feature. Each dimension has a tolerance except for dimensions identified as reference, maximum, minimum, or stock. The tolerance can be applied directly to the dimension, applied indirectly as with basic dimensions, indicated by a general note, or located in the tolerance block of the sheet. Examples found throughout this textbook show specific content being described and may not be completely dimensioned. Keep in mind that real-world drawings representing the same features would be completely dimensioned to follow the requirements explained throughout this textbook.




**NOTE**

ASME Y14.5-2018 eliminates plus and minus tolerancing and limit dimensioning for location dimensions. These methods are documented as alternative practices when applied to surfaces. Location dimensions that apply to regular features of size are basic from datum features with positional tolerances. Location dimensions between surfaces are basic from datum features with profile tolerances. Chapter 4 covers datums. Chapters 8 and 9 cover positional tolerancing. Chapter 10 covers profile tolerancing applications. Only size dimensions have plus and minus tolerances or limits defined by limit dimensions.

**NOTE**

For plus and minus tolerancing, the specified dimension is given first, followed by the plus/minus symbol, followed by the tolerance value. For example, .50±.01. For limit dimensioning, the maximum value is placed over the minimum value. For example, $\frac{.505}{.500}$. When the limit values are on one line, the minimum value is placed in front of the maximum value with the values separated by a dash. For example, .500–.505.

# Enhance Chapter Content with Notes

# Chapter Review

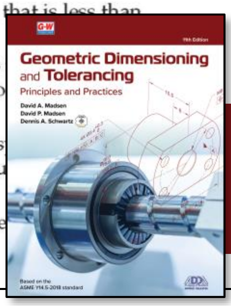
Name ___________________________________________

## Know and Understand

*Answer the following questions using the information provided in this chapter.*

### True/False

1. _____ *True or False?* The general note UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN MILLIMETERS should be placed on the drawing when dimensions are in millimeters. (2.1)
2. _____ *True or False?* The general note UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN INCHES should be placed on the drawing when dimensions are in inches. (2.1)
3. _____ *True or False?* The decimal point and zero are omitted when the metric dimension is a whole number. (2.2)
4. _____ *True or False?* A zero precedes a decimal millimeter dimension that is less than one. (2.2)
5. _____ *True or False?* When the metric dimension is greater than a whole number by a fraction of a millimeter, the last digit to the right of the decimal point is followed by one or more zeros depending on the tolerance. (2.2)
6. _____ *True or False?* Plus and minus values of a metric tolerance have the same number of decimal places, and zeros are added to fill in where needed. (2.2)
7. _____ *True or False?* Metric limit dimension values have the same number of decimal places. (2.3)
8. _____ *True or False?* When limit dimension values are displayed on one line, the lower value follows the higher value, and the values are separated with a dash. (2.3)
9. _____ *True or False?* When using unilateral tolerances in metric dimensioning, a single 0 is used without a + or – sign for the 0 part of the value. For example, $24^{\ 0}_{-0.25}$ and $24^{+0.25}_{\ 0}$. (2.3)
10. _____ *True or False?* Where basic dimensions are given, zeros are added to the dimension value to match the associated geometric tolerance. (2.3)
11. _____ *True or False?* Nominal size refers to a dimension used for general identification, such as stock size or thread diameter. (2.4)
12. _____ *True or False?* A zero precedes a decimal inch dimension that is less than one. (2.2)
13. _____ *True or False?* A specified inch dimension is expressed to of decimal places as its tolerance, and zeros are added to decimal point if needed. (2.3)
14. _____ *True or False?* Unilateral inch tolerances use the + and – s value has the same number of decimal places as the valu less than 0. (2.3)
15. _____ *True or False?* Inch limit dimension values are not require same number of decimal places. (2.3)




## Analyze and Apply

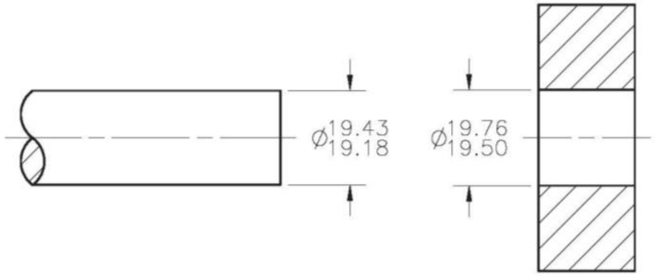
*Answer the following questions using the information provided in this chapter.*

1. A(n) ____________ dimension is a dimension that is considered theoretically perfect in size, profile, orientation, or location. (2.4)
2. ____________ is a general term applied to describe a physical portion of a part. (2.4)
3. ____________ is the total amount a dimension or feature can vary. (2.4)
4. Give the general note that should be placed on all drawings based on the standard used throughout this textbook. (2.1)
5. Give the basic rules for displaying metric limit dimension values correctly. (2.3)
6. Give an example of metric limit dime
7. Give an example of stacked metric lir
8. Give the basic rules for displaying inc
9. Give an example of inch limit dimens
10. Give an example of stacked inch limi

## Critical Thinking

*Answer the following questions using the information provided. Show all calculations.*

1. Referring to the specifications given for the two parts shown, answer the following questions. Calculate the values indicated where required and label each value. Show all calculations. (2.6)

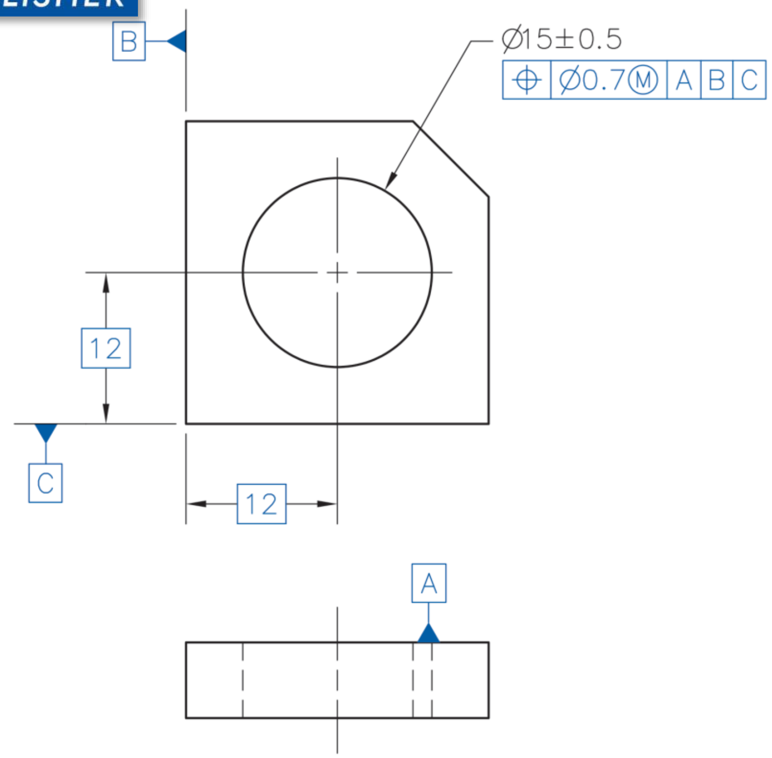



A. Is the fit between the two parts a clearance fit or a force fit?

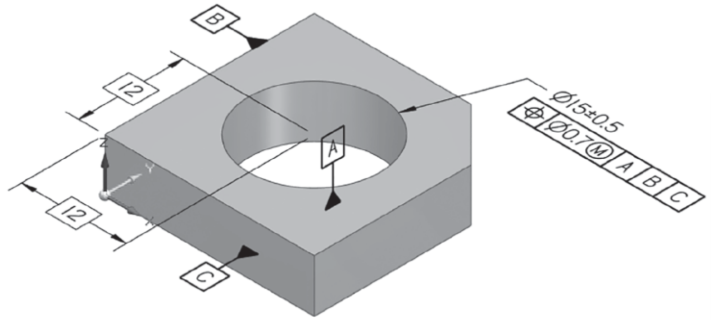
B. What is the allowance between the two parts? Show the formula and your calculations.

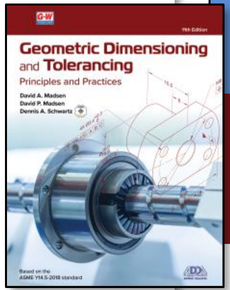
# End-of-Chapter Material

## Orthographic Views




## Model View






18. Refer to the 2X 0.76 X 45° dimension: (2.1)
    A. What is the tolerance for the 0.76 dimension
    B. What is the tolerance for the 45° dimension
19. Refer to the $\varnothing^{4.834}_{4.763}$ dimension: (2.1)
    A. What is the MMC? ______
    B. What is the LMC? ______
    C. What is the tolerance? ______

**Refer to the print of the MOUNTING PLATE (UP found on page 525.**

20. What do the parentheses around the 1.875 dim

______

21. Refer to the 1.60 dimension: (2.1)
    A. Where is the tolerance found? ______
    B. What is the tolerance? ______
    C. What are the limits? ______
    D. What is the MMC? ______
    E. What is the LMC? ______

**Refer to the print of the HYDRAULIC VALVE fou**

22. Refer to the $\varnothing^{.961}_{.959}$ dimension: (2.1)
    A. What is the tolerance? ______
    B. What is the MMC? ______
    C. What is the LMC? ______
23. Refer to dimension "A" for part number 1 MS
    A. What is the MMC of this dimension? ______
    B. What is the LMC of this dimension? ______
    C. What is the tolerance for this dimension? ______
24. What is the tolerance for the 45° dimension fo

______

**Refer to the print of the COVER, CAGE-INNER A**

25. What does "SR" mean on the SR7.500 dimens

______

**Refer to the print of the FEMORAL A-P SAW GU**

26. Are the dimensions given in inches or millime

______

27. Give the tolerances for the following unspecifi
    A. .x ______
    B. .xx ______
    C. .xxx ______

Name ______

## Print Reading Problems

The following print reading problems are provided for manufacturing technology students and are optional for drafting technology students as determined by course objectives. The print reading problems can be considered helpful for drafting technology students in understanding more about the information provided on prints. The print reading problems give you the opportunity to read prints illustrating dimensioning and GD&T concepts related to the chapter. These problems require you to apply concepts from the chapter, interpret information on prints, and use problem-solving skills.

The print reading problems use actual industry prints with related questions that require you to read specific dimensioning and GD&T representations. The answers should be based on previously learned content in this book. The prints used are based on ASME standards. However, company standards can differ slightly. When reading these prints, or any other industry prints, a degree of flexibility is required to determine how individual applications correlate with the ASME standards.

**Refer to the print of the SLEEVE-DEWAR REIMAGING found on page 521.**

1. Are the dimensions given in inches or millimeters? (2.2)

   ______

2. What does the print say about burrs and sharp edges? (2.1)

   ______

3. Refer to the 1.914 dimension: (2.1)
   A. Where is the tolerance specified? ______
   B. What is the tolerance? ______
   C. What is the maximum limit? ______
   D. What is the minimum limit? ______
4. Refer to the Ø.8740±.0005 dimension: (2.1)
   A. Where is the tolerance specified? ______
   B. Is the tolerance unilateral or bilateral? ______
   C. What is the tolerance? ______
   D. What is the MMC? ______
   E. What is the LMC? ______
5. Refer to the Ø.7500 dimension: (2.1)
   A. Where is the tolerance specified? ______
   B. What is the tolerance? ______
   C. What is the MMC? ______
   D. What is the LMC? ______
6. Give the complete specifications associated with the Ø.107±.001 hole. (2.1)

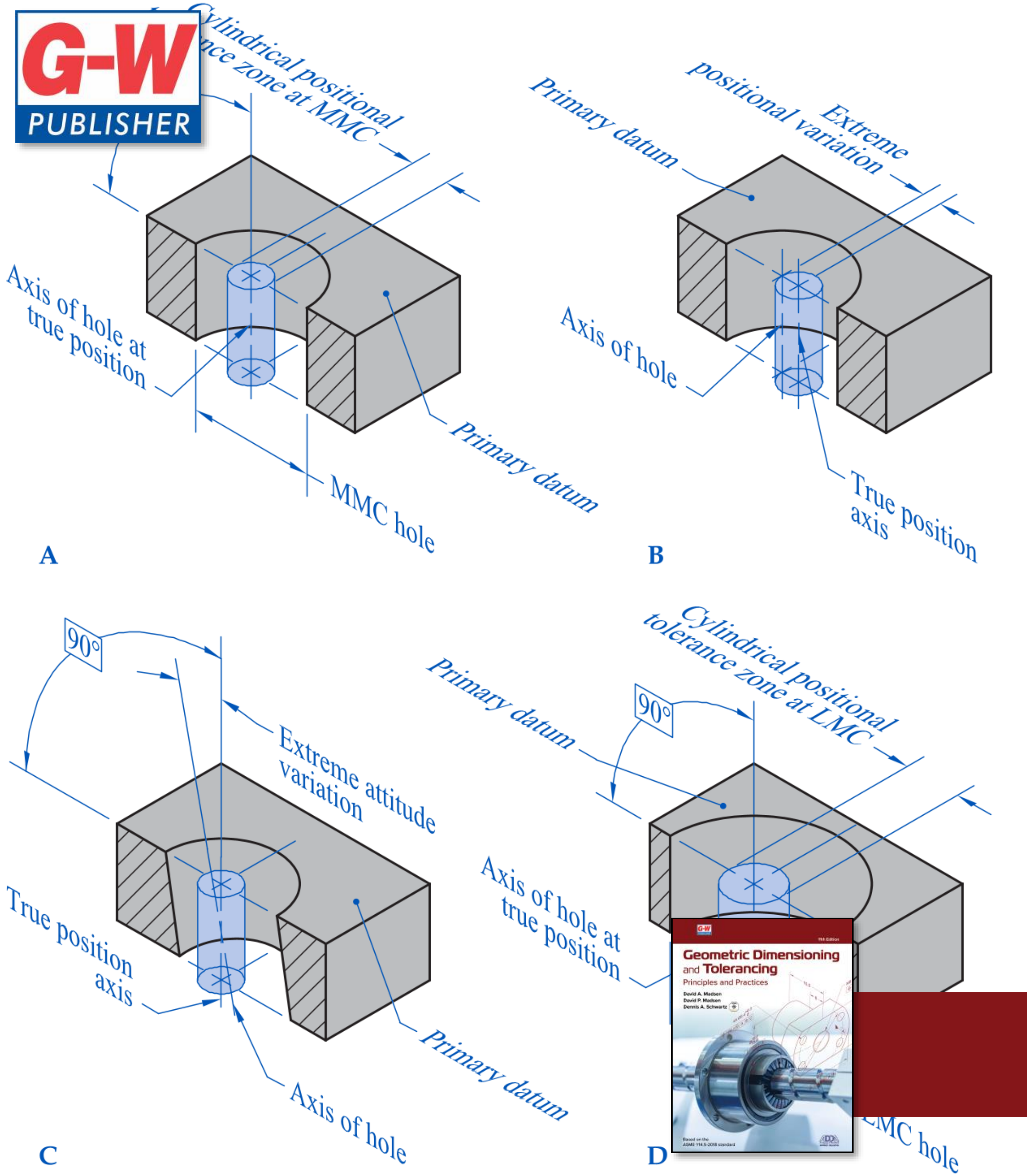
   ______

   ______

7. What does the box around the .469 dimension mean? (Look at **Example 3-11** in Chapter 3.) (2.1)

   ______

# Print Reading Problems






## Drafting Problems

Drafting problems are provided for drafting technology students in the Drafting Problems section of this textbook. Drafting problems are presented as pictorial illustrations or design layouts. This requires you to determine the correct views, dimensions, and GD&T placement. Drafting problems can be completed by creating and dimensioning orthographic views, or by creating 3D models as determined by your course objectives.

The Drafting Problems section of this textbook contains Problems 1–21. Problems 22–39 are included in the instructor's resources for this textbook. Ask your instructor or refer to the course outline for specific assignments and problems to create for this chapter.

Instructions for completing problems are given in the Drafting Problems section of this textbook. Drafting problems for Chapter 2 are drawn without placing GD&T symbols unless otherwise assigned by your instructor. Draw the appropriate orthographic views with dimensions or create 3D models as assigned by your instructor. The following are drafting problems recommended for this chapter:

Drafting Problems 1 through 8, 10, 11, 13, 14, 15, 16, 17, 18, 20, 21, 22, 23, and 24.



## Drafting Problems

Drafting problems are provided for drafting technology students in the Drafting Problems section of this textbook. Drafting problems are presented as pictorial illustrations or design layouts. This requires you to determine the correct views, dimensions, and GD&T placement. Drafting problems can be completed by creating and dimensioning orthographic views, or by creating 3D models as determined by your course objectives.

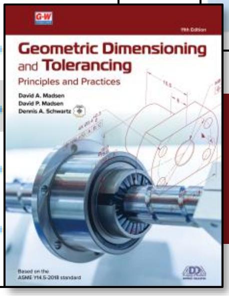
The Drafting Problems section of this textbook contains Problems 1–21. Problems 22–39 are included in the instructor's resources for this textbook. Ask your instructor or refer to the course outline for specific assignments and problems to create for this chapter.

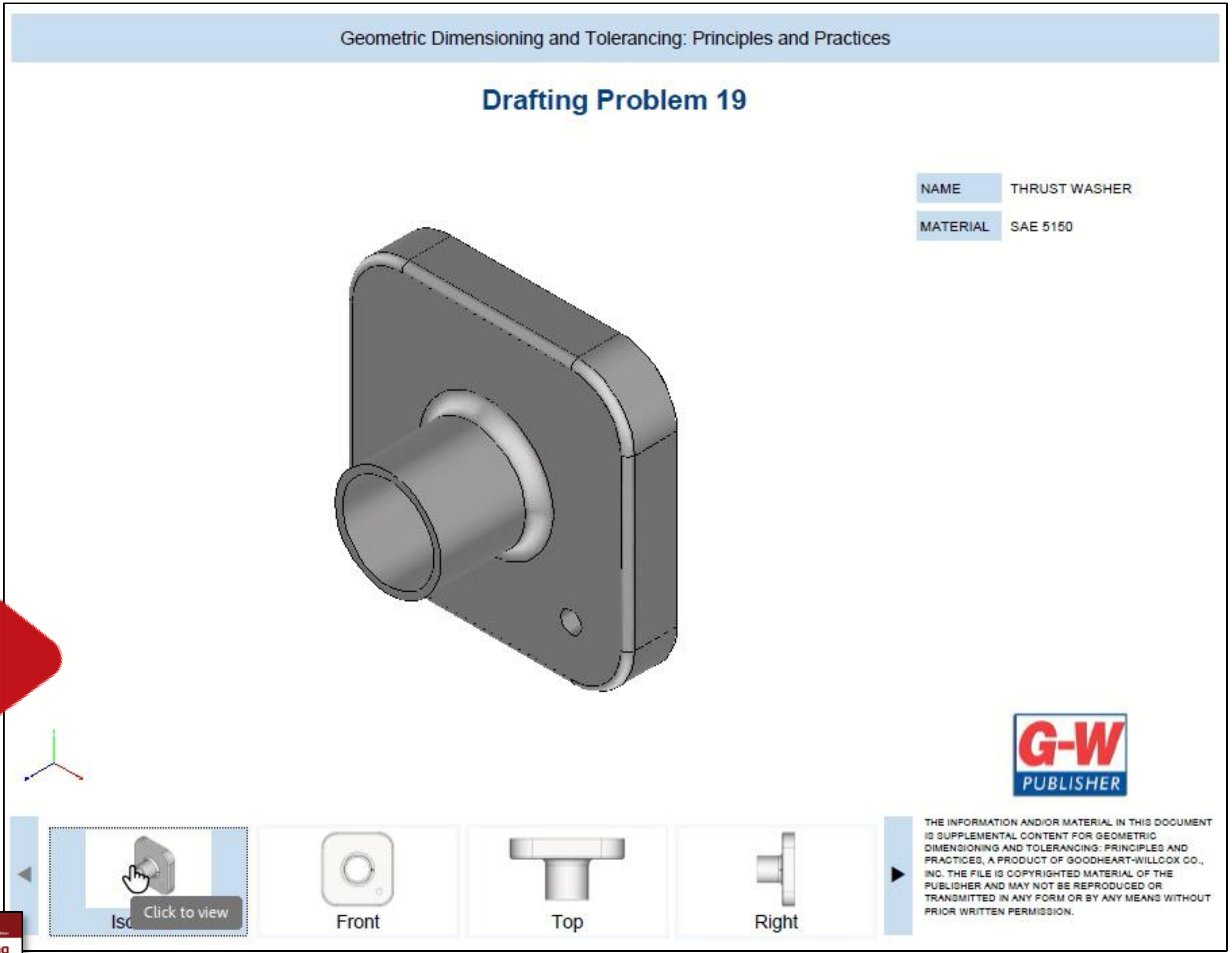
Instructions for completing problems are given in the Drafting Problems section of this textbook. Drafting problems for Chapter 8 are drawn by placing all dimensions, notes, and GD&T symbols unless otherwise assigned by your instructor. Draw the appropriate orthographic views with dimensions or create 3D models as assigned by your instructor. The following are drafting problems recommended for this chapter:

Drafting Problems 1, 7, 8, 10, 11, 13, 14, 15, 18, 19, 20, 21, 22, 23, 24, 25, 26, 31, 32, 33, 34, 35, 38, and 39 (use content learned in this chapter and previous chapters for Problem 39).

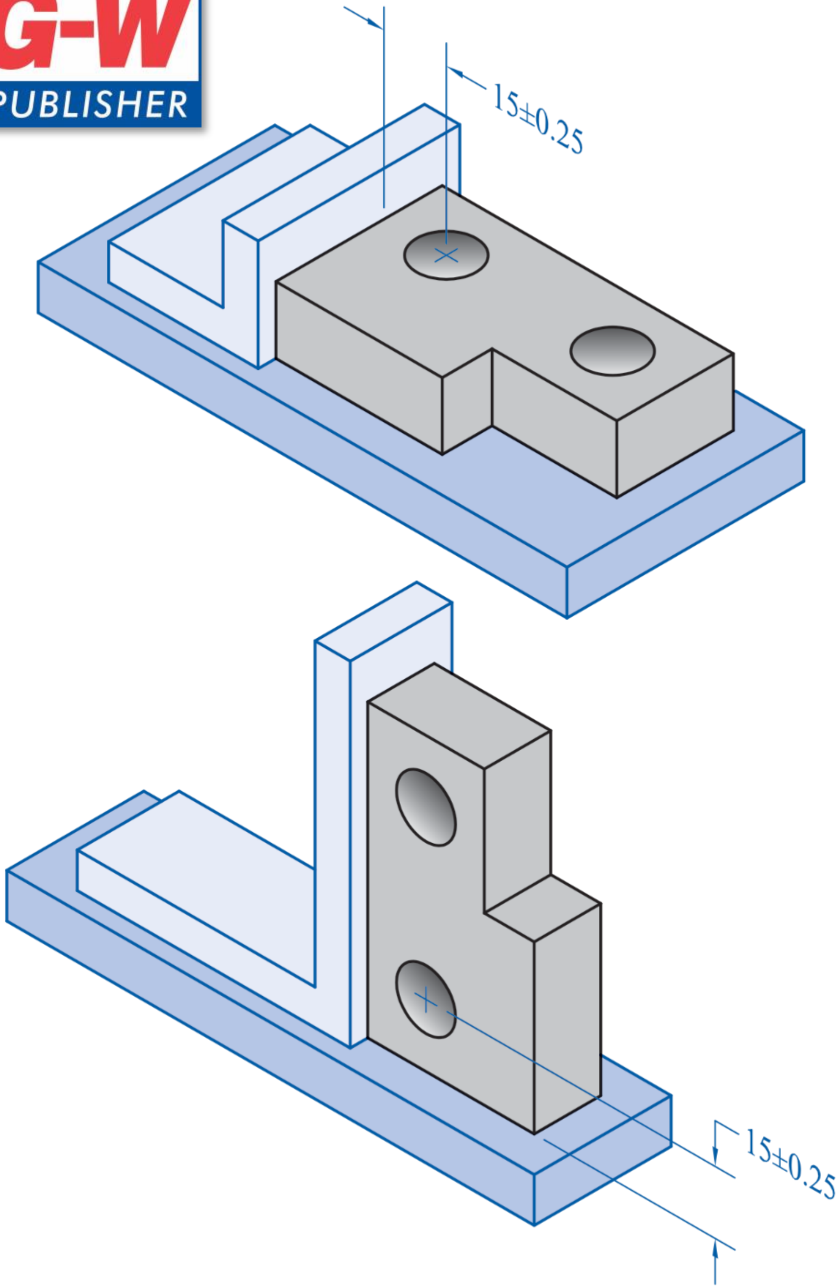
# Drafting Problems

| | Type |
|---|---|
| 27_dp01.pdf | Adobe Acrobat Document |
| gdtpp2027_dp02.pdf | Adobe Acrobat Document |
| gdtpp2027_dp03.pdf | Adobe Acrobat Document |
| gdtpp2027_dp04.pdf | Adobe Acrobat Document |
| gdtpp2027_dp05.pdf | Adobe Acrobat Document |
| gdtpp2027_dp06.pdf | Adobe Acrobat Document |
| gdtpp2027_dp07.pdf | Adobe Acrobat Document |
| gdtpp2027_dp08.pdf | Adobe Acrobat Document |
| gdtpp2027_dp09.pdf | Adobe Acrobat Document |
| gdtpp2027_dp10.pdf | Adobe Acrobat Document |
| gdtpp2027_dp11.pdf | Adobe Acrobat Document |
| gdtpp2027_dp12.pdf | Adobe Acrobat Documen |
| gdtpp2027_dp13.pdf | Adobe Acrobat Docu ent |
| gdtpp2027_dp14.pdf | Adobe Acrobat Doc ment |
| gdtpp2027_dp15.pdf | Adobe Acrobat Document |
| gdtpp2027_dp16.pdf | Adobe Acrobat Document |
| gdtpp2027_dp17.pdf | Adobe Acrobat Document |
| gdtpp2027_dp18.pdf | Adobe Acrobat Do |
| gdtpp2027_dp19.pdf | Adobe Acrobat Do |
| gdtpp2027_dp20.pdf | Adobe Acrobat Do |
| gdtpp2027_dp21.pdf | Adobe Acrobat Do |




# 3D Prints


15±0.25
15±0.25


11th Edition
Geometric Dimensioning and Tolerancing
Principles and Practices
David A. Madsen
David P. Madsen
Dennis A. Schwartz
Based on the ASME Y14.5-2018 standard