G-W
PUBLISHER

GD&T
APPLICATION AND INTERPRETATION

Eighth Edition

156

.003 Ⓜ A B Ⓜ

300

Bruce A. Wilson

Based on the ASME Y14.5-2018 standard

The author is GDTP certified by ASME in accordance with the qualifications of ASME 14.5.2 in the Senior level
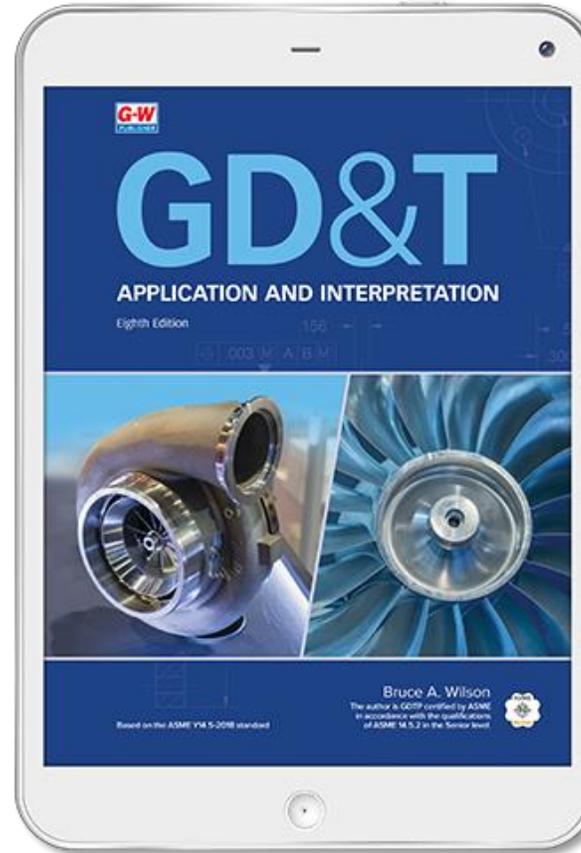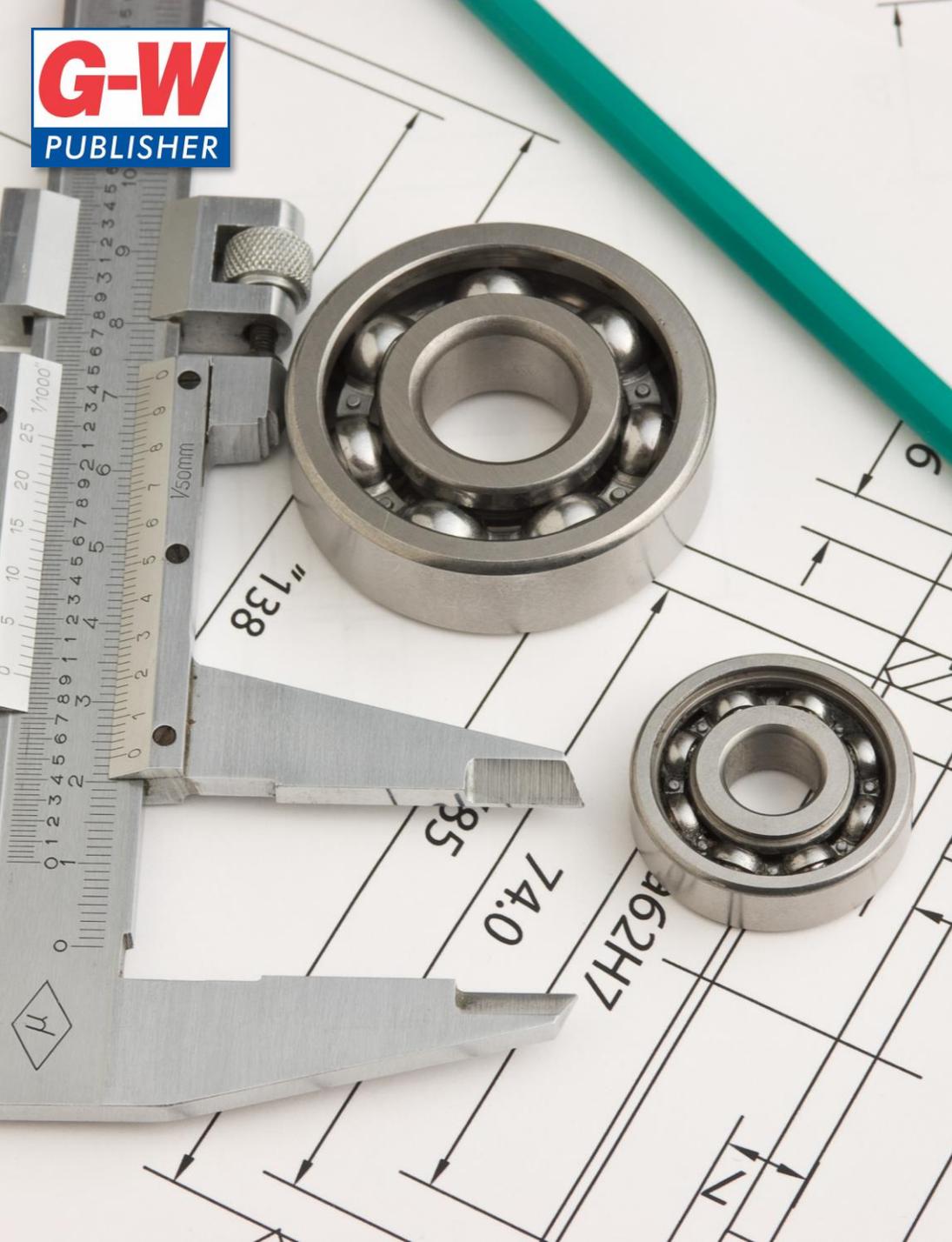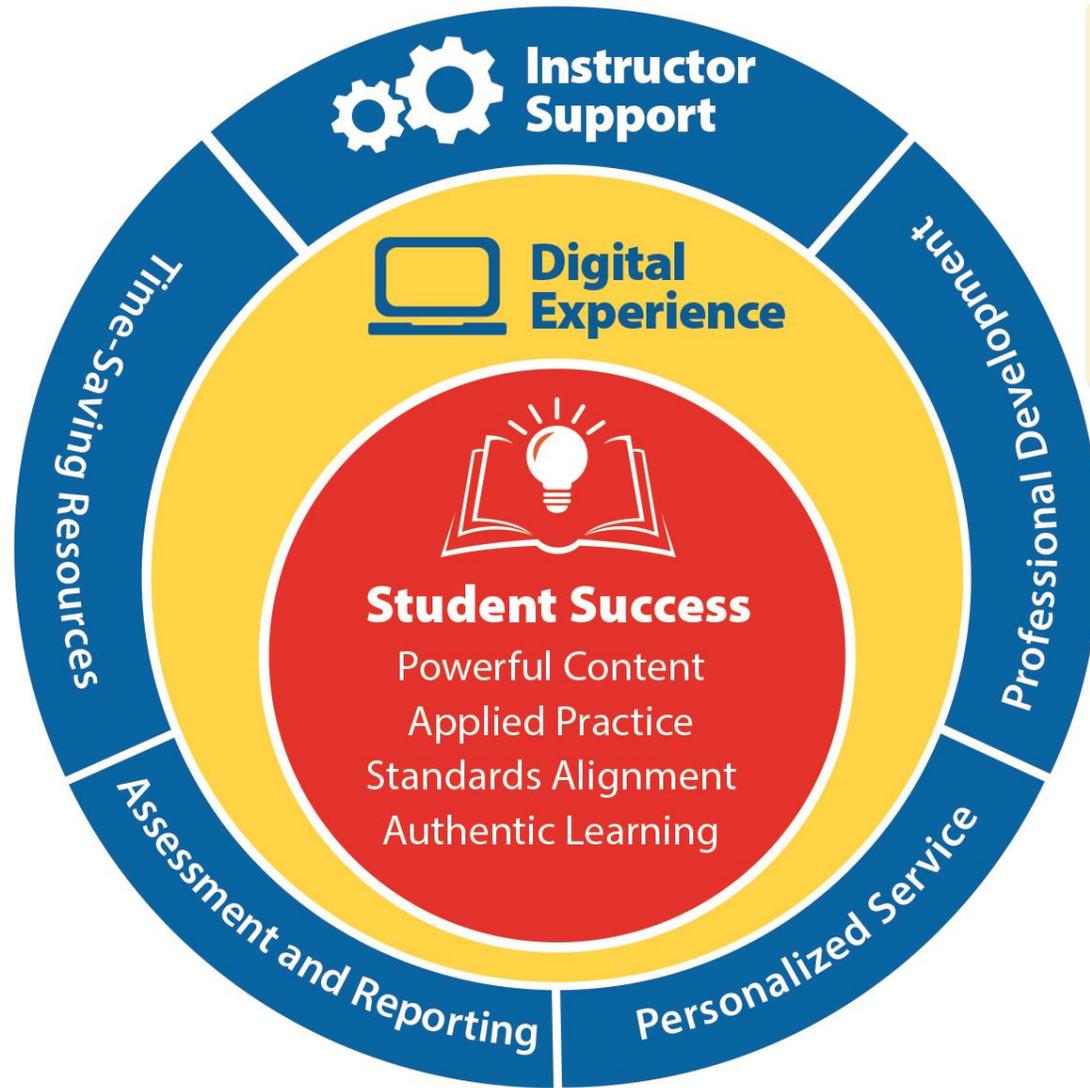
GD&T: Application and Interpretation

Designed by Educators, Built for Educators

Student Success Is At the Heart of What We Do

Construction Careers:
Surveyor/Surveying Technician

- Establish official land boundaries and exact locations
- Must meet education and experience requirements
- Must pass licensing examination
- Work for architectural and engineering firms

Measuring and Recording Respirations (03:46)

g-wonlinetextbooks.com

Modern Residential Wiring 13e, Digital Companion

Collapse All

Interactive Glossary

1 Electrical Careers ▼

E-Flash Cards
Matching Activity
Vocabulary Game

1: E-Flash Cards

cable splicer

✓ Prepare for class
✓ Reinforce new concepts
✓ Assess learning

**Ready-to-Implement
Online Instructor Resources**

G-W
PUBLISHER

1 Introduction to Dimensioning and Tolerancing

1 Resources

Answer Key
Lesson Plan

1 Presentations

Instructor's Presentation
Instructor's Presentation for PowerPoint

2 Dimensioning and Tolerancing Symbology
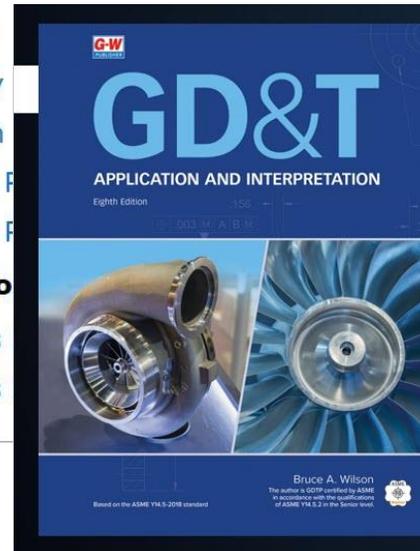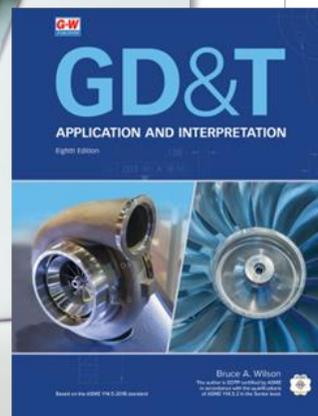
2 Resources

Answer Key
Lesson Plan
Application
Application

2 Presentations

Instructor's
Instructor's

Presentations for PowerPoint

GD&T
Application and
Interpretation

**Lesson Plans, PowerPoint
Presentations, and Answer Keys**

G-W Assessment
Assess learning • Gain insights • Build skills

- **Utilize a cloud-based assessment platform** that can seamlessly integrate with your LMS and Google Classroom

- **No more downloads, uploads, or imports** needed!

- **Access G-W Assessment with one quick click** from any device

NEW for 2026 Copyrights!
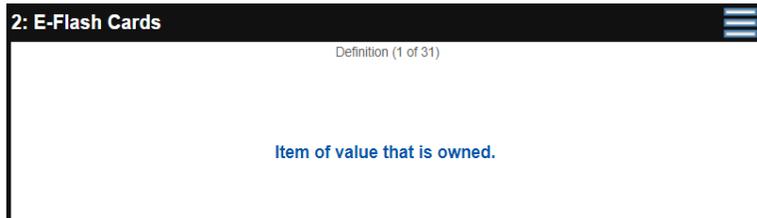
Print • Digital • Bundle Options Available

**LMS and CMS Integration
Easy Navigation**

G-W PUBLISHER

E-Flash Cards & Vocabulary Practice

2: E-Flash Cards
Definition (1 of 31)

Item of value that is owned.

2: E-Flash Cards
Term (1 of 31)

Select to flip

asset

Previous | Remove | Next

2: Vocabulary Game

Select a point value. Choose the term that matches the definition.

Score: 800

100	100	100	100
200	200	200	200
300	300	300	300
400	400	400	400

Definition: Act of giving money, goods, or services to meet the needs of others and supporting organizations and causes that are important to an individual.

- pay yourself first
- variable expense
- recordkeeping
- philanthropy

Check Answer

Interactive Activities

Chapter 3: General Dimensioning Requirements

Reading

Read Chapter 3 of the *GD&T: Application and Interpretation* textbook prior to completing the review exercises.

Learning Outcomes

A combination of activities is required to achieve the following outcomes. Completing the reading assignment and the following review exercises is an important part of achieving the outcomes. Familiarization with the outcomes prior to completion of the reading assignment and review exercises will make mastery of the outcomes easier. After completing the reading assignment and completing the review exercises, you will be able to:

3.1 Apply general dimensioning methods using the correct line types, lettering sizes, and arrowhead form.

3.2 Understand placement of dimensions and tolerances on annotation planes in models.

3.3 Apply dimensions using the best methods to achieve the desired fit and function.

3.4 Utilize preferred dimension placement to provide clear specification of part requirements.

3.5 Apply general and specific notes on a drawing.

3.6 Cite the general types of fit between mating parts.

Review Exercises

Place your answers in the spaces provided. Show all calculations for problems that require mathematical solutions.
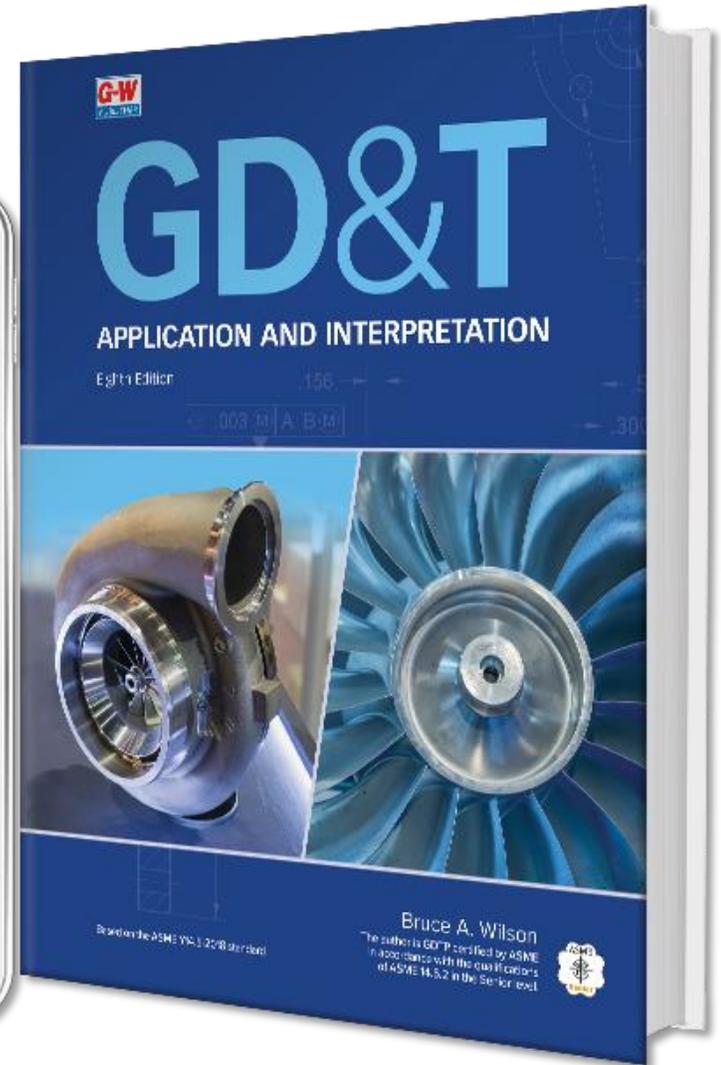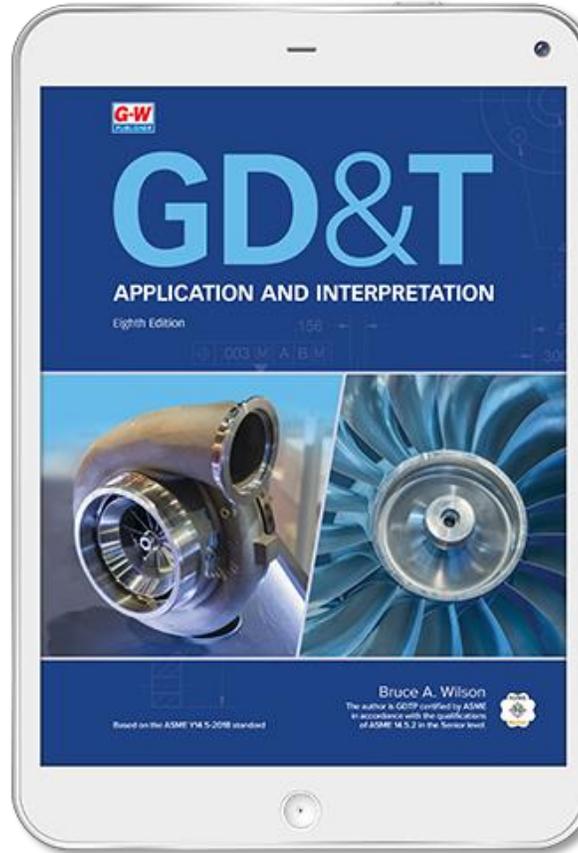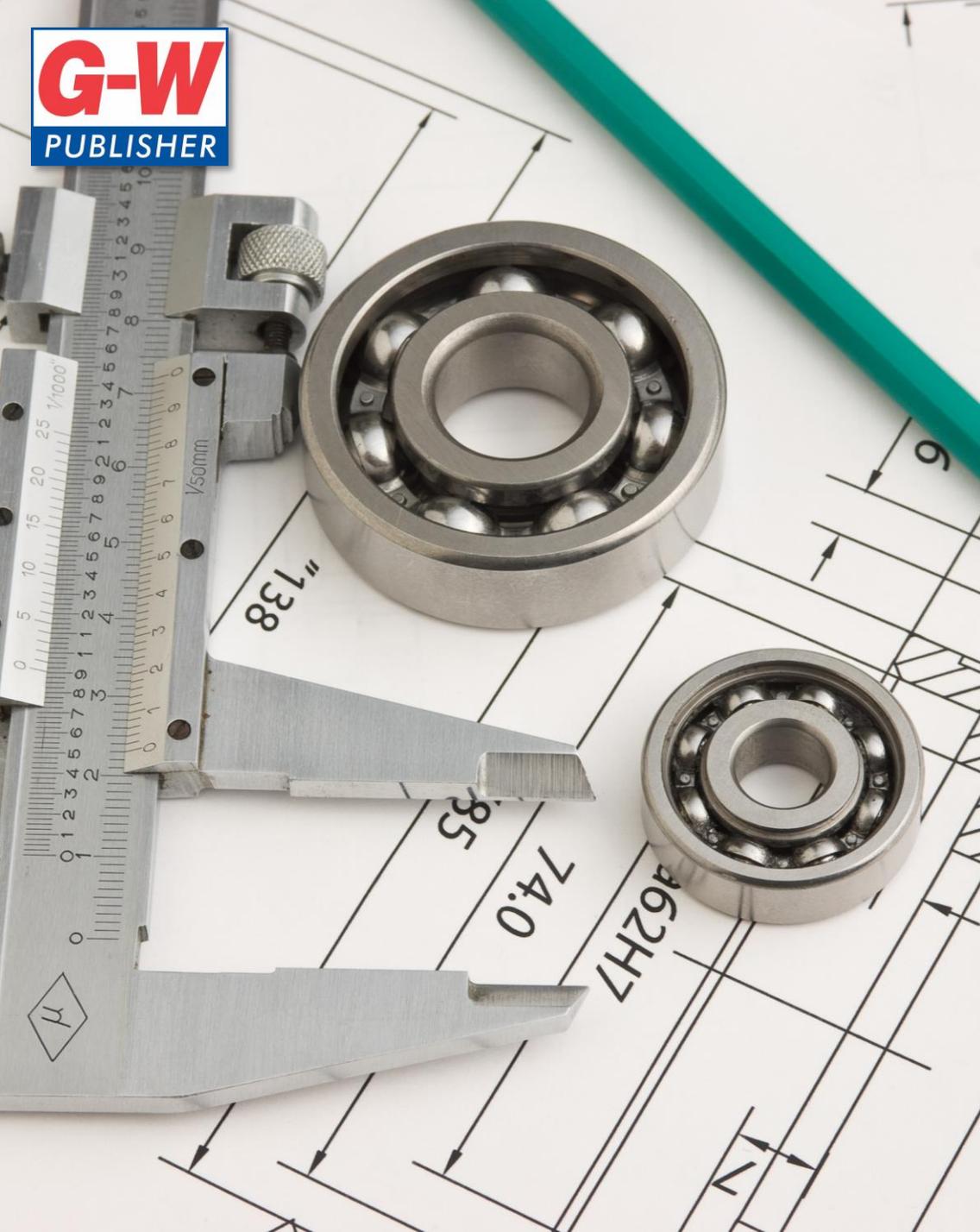
Multiple Choice

1. When a gap is used, extension lines begin approximately _____ from the dimensioned feature to provide a visible gap.
   A. .01–.02″
   B. .03–.06″
   C. .10–.12″
   D. .16–.18″
   Answer: _____

Study Guide

Integrate G-W Digital Resources

G-W
PUBLISHER

GD&T
APPLICATION AND INTERPRETATION
Eighth Edition

Bruce A. Wilson
The author is GDTP certified by ASME
in accordance with the qualifications
of ASME 14.5.2 in the Senior level.
Based on the ASME Y14.5-2018 standard

GD&T
APPLICATION AND INTERPRETATION
Eighth Edition

Bruce A. Wilson
The author is GDTP certified by ASME
in accordance with the qualifications
of ASME 14.5.2 in the Senior level.
Based on the ASME Y14.5-2018 standard

©2026, 8th edition, 484 pp.
by Bruce A. Wilson

2X Ø.328 $^{+.006}_{-.000}$
⊕ Ø .016 Ⓜ A Ⓜ B CⓂ

2X Ø.328 $^{+.006}_{-.000}$
⊕ Ø .016 Ⓜ A Ⓜ B CⓂ
SEP REQT ← Notation

B   Ø 900

1.350   .750   .250±.002
⊕ .008 Ⓜ A Ⓜ

2X Ø.390 $^{+.006}_{-.000}$
⊕ Ø .015 Ⓜ A Ⓜ B CⓂ

B   Ø.900

1.350   .750   .250±.002
⊕ .008 Ⓜ A Ⓜ

2X Ø.390 $^{+.006}_{-.000}$
⊕ Ø .015 Ⓜ A Ⓜ B CⓂ
SEP REQT ← Notation

Datum feature A simulator   Datum feature B simulator   Datum feature C simulator (.240)

Ø.312   Datum feature B simulator removed from this view

Ø.375   Four removable pins sized to the virtual condition of the holes

Datum feature A simulator   Datum feature B simulator   Datum feature C simulator (.240)

Ø.312   Datum feature B simulator removed from this view

Two removable pins sized to the virtual condition of the holes
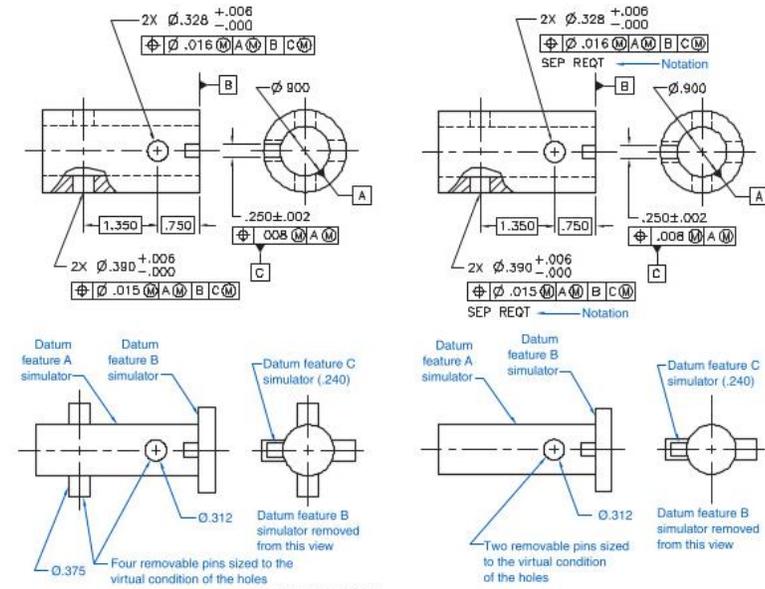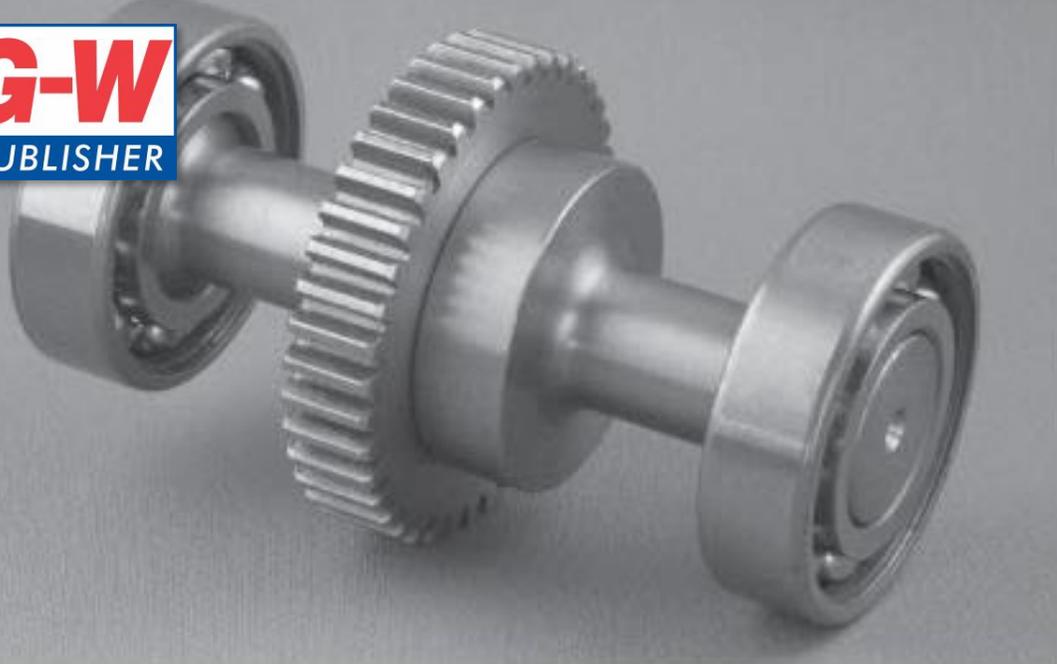
Goodheart-Willcox Publisher

Figure 9-19. Identical references to datum features of size require that the features be considered as a simultaneous requirement (single pattern).

holes to move relative to the datum features. The position tolerances on all the holes are simultaneously related to the same datum reference frame.

Separate Requirement Noted for References to Datum Features of Size

Groups of features may be specified to allow their TZFs to move independently (separately) even though datum feature references are identical. Exception to the simultaneous requirement is specified by placing a notation of SEPARATE REQUIREMENT or the abbreviation SEP REQT under the feature control frames of those items that are to act separately. See **Figure 9-20.** Although this figure contains the same part shown in **Figure 9-19,** the notation of separate requirement has been added beneath the two position tolerances.
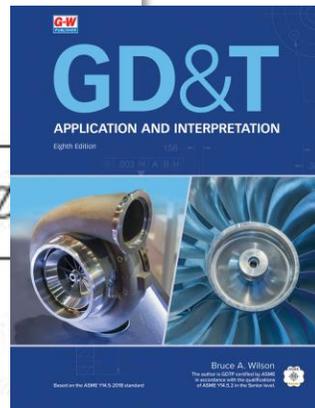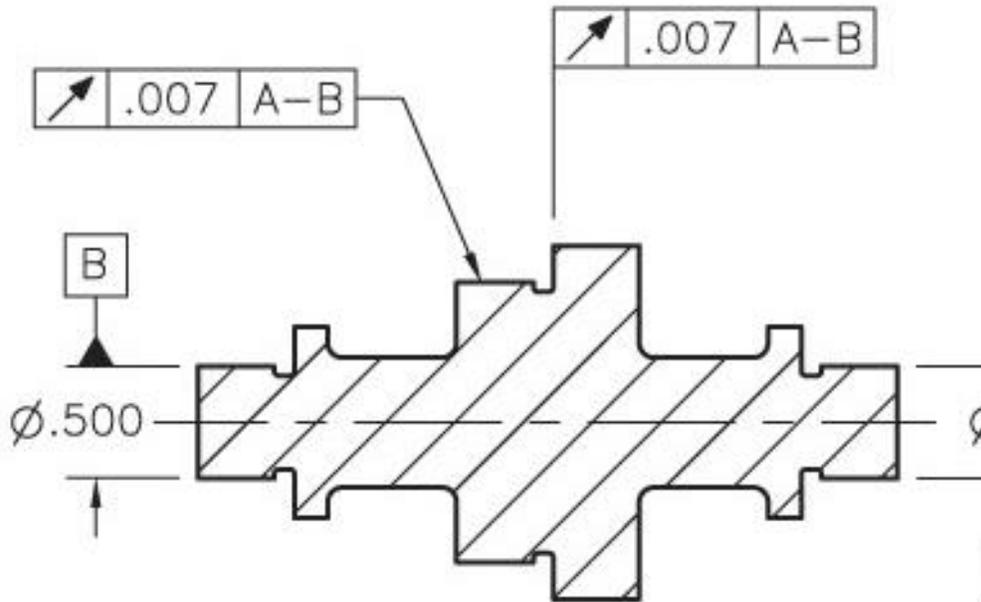
Both sets of holes are separately controlled due to the notation, and each set of holes has a TZF that is independent of the other. They are separate. The effect
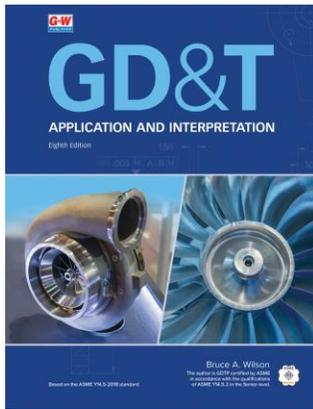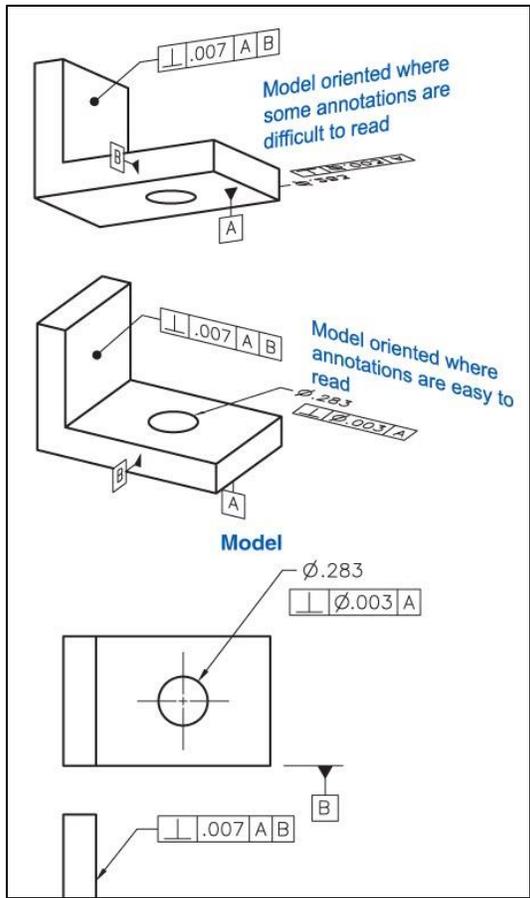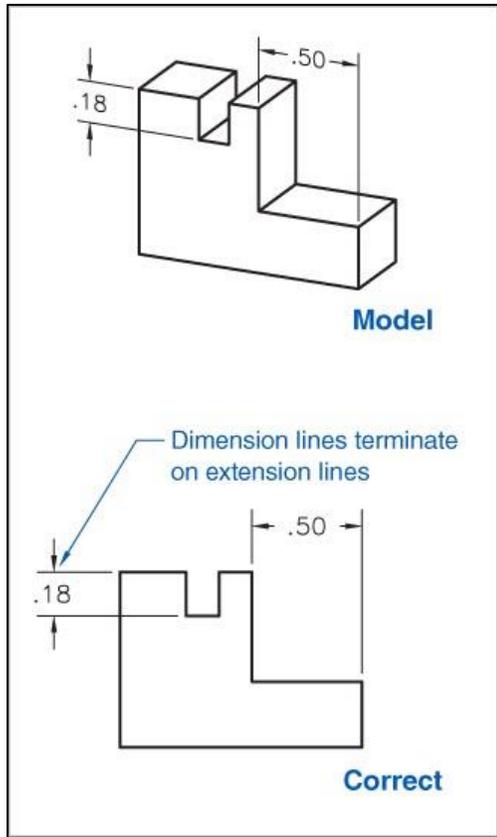
of this type of tolerance requirement is shown by the functional tools for inspection of the two hole patterns.

Two tools are shown for checking the two separate requirements. Each tool has datum feature simulators that are sized in the same manner as described

STANDARDS ADVISORY

Revisions of Standards

Specification of surface texture requirements changed significantly with the release of ASME Y14.36-2018. Including reference to the applicable standard is important to ensure correct interpretation of any surface texture requirements.

⬈ .007 | A–B

⬈ .007 | A–B

B

Ø.500

Goodheart-Willcox Publisher

G-W PUBLISHER

GD&T
APPLICATION AND INTERPRETATION
Eighth Edition
Bruce A. Wilson

Student Textbook

G-W
PUBLISHER

⊥ .007 A B

Model oriented where some annotations are difficult to read

⊥ .007 A B

Model oriented where annotations are easy to read

Ø.283
⊥ Ø.003 A

Model

Ø.283
⊥ Ø.003 A

⊥ .007 A B

B

.50

.18

Model

Dimension lines terminate on extension lines

.50

.18

Correct

do establish their own drafting and dimensioning requirements for special applications. When this is done, care should be taken to create company-specific standards such that no conflict is created with the national standards. As an example, an existing symbol should not be redefined for a different purpose. That would almost certainly result in confusion. It is also important to ensure these special company standards be made available to other companies that must work to the special requirements. It is proactive to communicate special needs with the appropriate ASME Y14 subcommittees to influence the content of future standards revisions.

STANDARDS ADVISORY

ISO Standards

As explained in Chapter 1, the ASME Y14.5 standard is the primary standard for dimensioning and tolerancing in the United States. However, other standards exist and it is important to be aware of them. Standards established by the International Organization for Standardization (ISO) are used in engineering and manufacturing and may be encountered in international work. ISO standards defining dimensioning and tolerancing requirements use many of the same symbols as ASME standards. However, ISO standards have a different meaning for some symbols, and that can result in significant differences in interpretation unless the applicable standards are known. It is essential that product documentation include a note invoking the applicable standard(s).

To ensure dimensions and tolerances are interpreted as intended when they are specified, it is essential to include a product documentation note invoking the applicable standard(s). If unable to comply with some requirements of an applicable standard, the note should be modified to take exception where needed. Failure to invoke the applicable standards has the potential to cause far-reaching negative effects that include (a) no defined meaning of the dimensions and tolerances and (b) the potential for conflicting requirements based on standards that may be imposed without the product definition explicitly invoking them. The following example invokes two standards in their entirety:

DIMENSIONS AND TOLERANCES SHALL BE IN ACCORDANCE WITH ASME Y14.5-2018 AND ASME Y14.41-2019.

Location and Size Dimensions

All dimensions applied to a part show size, profile, distance, angle, or location. See **Figure 3-1**. A *location dimension* (specified with an L in the figure) describes *where* a feature is, and a *size dimension* (specified with an S in the figure) indicates how large a feature is. Location and size must be controlled in all three axes.

Size is dimensioned through one of several means depending on the shape. A rectangular prism is sized by dimensioning the height, width, and depth (three size dimensions). A hole can be sized by giving its diameter. If a hole does not go through a part, then its depth is dimensioned. If a hole goes through a part, depth is not needed. The part thickness defines the length of a through hole.

Location of a feature in a coordinate system requires three dimensions. However, most feature locations can be specified by showing one or two dimensions. This is possible because other features on the part define the remaining location dimension(s).
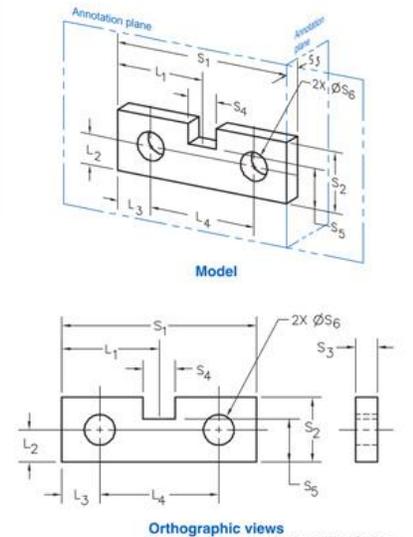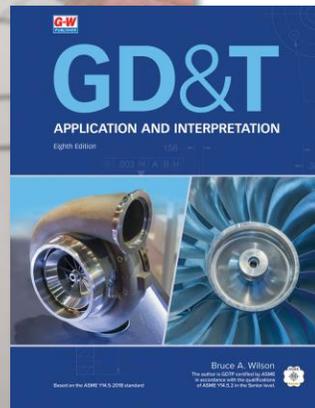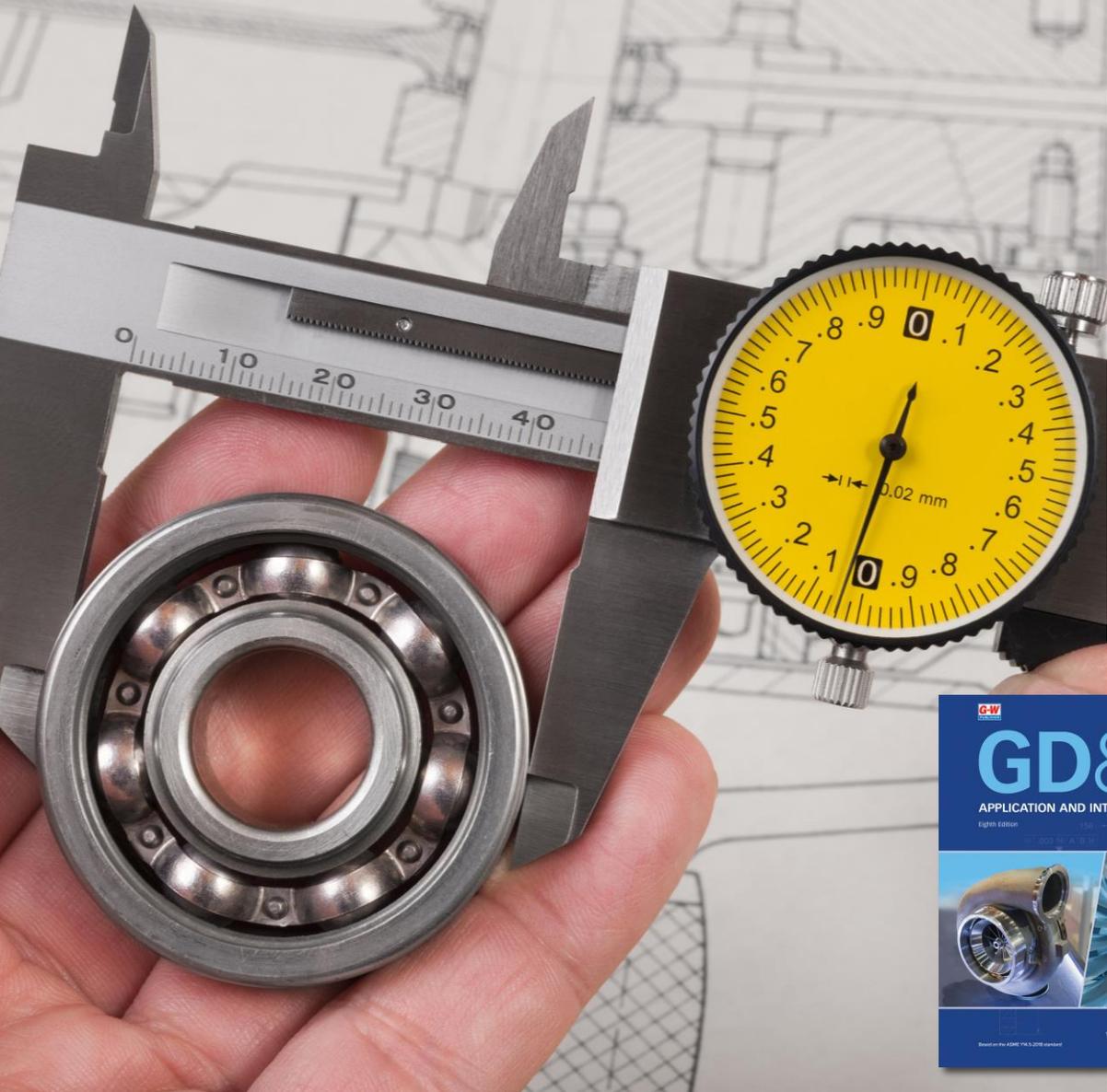
Model

Orthographic views
Goodheart-Willcox Publisher

Figure 3-1. The letter S indicates size dimensions, and the letter L indicates location dimensions. The view of the model shows the annotation planes used to place annotation.

G-W
GD&T
APPLICATION AND INTERPRETATION
Eighth Edition

Bruce A. Wilson

What's New to the Edition

Position Tolerancing— Expanded Principles

Learning Outcomes

Information in this chapter will enable you to:

9.1 Explain functional gaging concepts for checking hole position tolerances specified at MMC.

9.2 Specify and explain composite position feature control frames.

9.3 Explain the effect of using identical datum feature references in multiple position tolerance specifications.

9.4 Specify separate pattern requirements for groups of features not acting as a single pattern.

9.5 Specify position tolerances for coaxial holes and other coaxial features.

9.6 Specify position tolerances to control symmetry.

Technical Terms

actual mating envelope (AME)
composite position tolerance
concentricity
derived median line
derived median plane
feature-relating tolerance
feature-relating tolerance zone framework (feature-relating TZF)
pattern-locating tolerance
pattern-locating tolerance zone framework (pattern-locating TZF)
segment
simultaneous requirement
symmetry
tolerance zone framework (TZF)
unrelated actual mating envelope (unrelated AME)

Introduction

Position tolerancing methods permit a great deal of flexibility in the level of control that is specified in a model or on a drawing. The single-segment position tolerance specification presented in the previous chapter is adequate for many situations. However, additional methods are required to specify the greater levels of control necessary to meet complex design requirements. Composite position tolerances and multiple single-segment feature control frames provide a significant increase in the possible levels of control.

A special functional application of position tolerances is to control symmetry. Depending on the application, symmetry is accomplished with a position tolerance that is applicable RFS, MMC, or LMC.

Position tolerances may also be used to control the allowable variation between coaxial features. Coaxial features may also be controlled by profile and runout tolerances.

Chapter-Opening Material

The practice of using a symmetry tolerance symbol was included in the ASME Y14.5-2009 standard. A symmetry tolerance was used for control of a derived median plane. A symmetry tolerance applied RFS was different from a position tolerance applied RFS and was determined to be unnecessary. Past usage of the symmetry tolerance symbol based on the 1994 and 2009 standards is briefly explained later in this chapter.
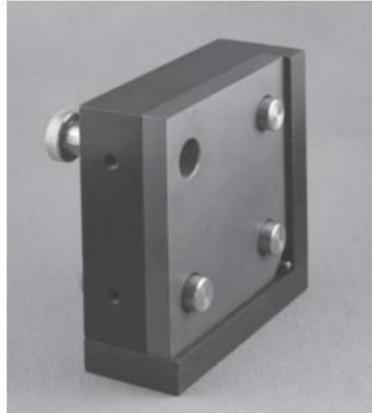
The practice of using a concentricity tolerance symbol was included in the ASME Y14.5-2009 standard and earlier standards. A concentricity tolerance was used for control of a derived median line. A concentricity tolerance applied RFS was different from a position tolerance applied RFS. The complex meaning of a concentricity tolerance was not what the term may intuitively seem to indicate and it was determined to be unnecessary. Past usage of the concentricity tolerance symbol based on the 2009 standard is briefly explained later in this chapter.

Relatively simple means of calculating the amount of position variation relative to the specified tolerances and paper gaging were presented in the last chapter. Those methods require data to be gathered and analyzed. Although paper gaging is a valuable process for many situations, gathering and analyzing data can be a significant expense if it is necessary to inspect a large quantity of parts.

Functional gages as shown in **Figure 9-1** provide another means of verifying that features are within specified position tolerances, and this chapter shows how they may be used. This chapter does not provide direction regarding how to calculate gage tolerances. Calculation of gage tolerances is an extensive subject and various methods are used depending on the inspection philosophy that is applicable. Any dimensions shown on functional gages in the figures of this textbook are intended to reflect the theoretically exact boundary that is to be verified on the part.

Physical functional gages are declining in usage, because parts are controlled through process control and the gaging function can now be performed through inspection software. But the concept is the same. If the physical gage is understood, then what must be achieved by the software is also understood.

Position tolerancing methods are intended to provide a means of specifying tolerances that reflect the functional requirements of the parts. Expressing functional requirements through properly applied tolerances not only maximizes the available tolerance and improves the producibility of the parts, but it can also make it possible to check the parts with


Goodheart-Willcox Publisher

Figure 9-1. The upper photograph shows a functional gage and a part adjacent to it. The lower photograph shows the part in the functional gage with three of the four pins extended through the holes to confirm position tolerances for those holes.

Third angle projection symbol
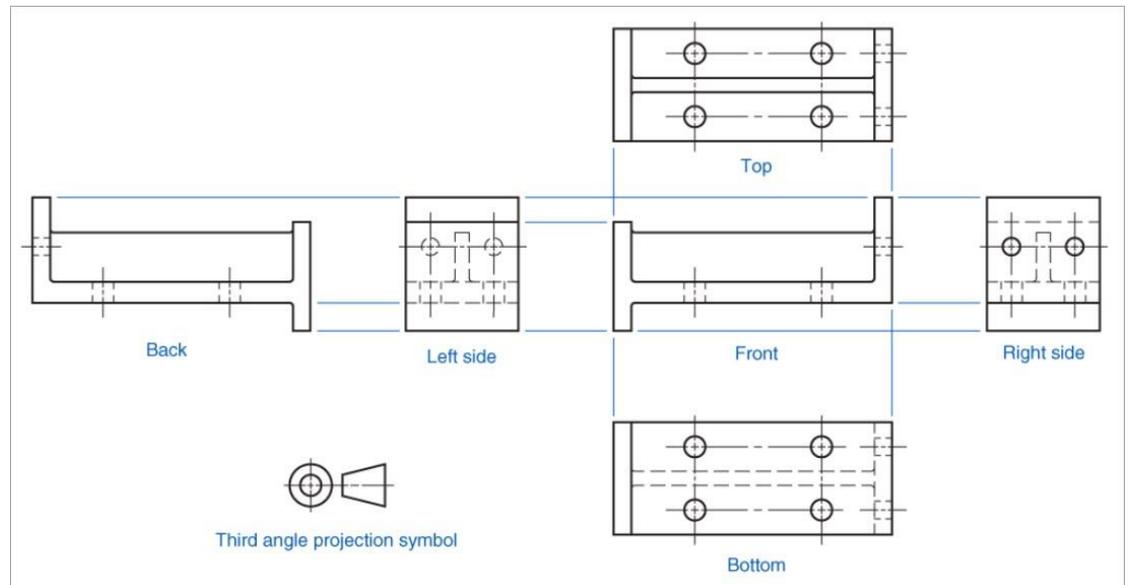
NOTE

The constraints on a tolerance zone framework relative to a datum reference frame are always explained in regard to "translational and rotational degrees of freedom." The tolerancing terms *position*, *location*, and *orientation* are used in regard to the control and variation of part features within the tolerance zones.

PRO TIP

Contoured Surfaces as Datum Features at RMB

When a profile tolerance is applied to a contoured surface, referencing that surface as a primary datum RMB creates a requirement that is difficult to simulate. The datum feature simulator is supposed to progress through the tolerance zone until maximum possible contact is achieved. Determining where maximum possible contact takes place on an imperfect contoured surface can be difficult to do. To avoid this difficult simulation requirement, the datum feature reference may be made at basic if the curvature does not make it impossible to contact the datum feature simulator. Another possibility is to apply datum targets on the surface and locate them with basic dimensions so that the datum tar-
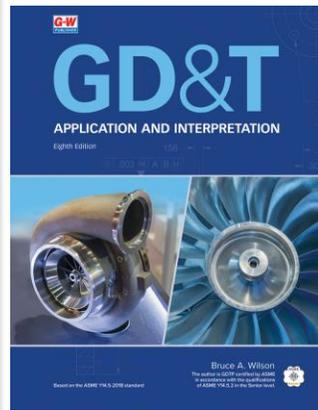
Textbook Features

End-of-Chapter Content

Chapter Summary

→ Composite position tolerances create pattern-locating and feature-relating tolerances.

→ The specified pattern-locating tolerance value of a composite position tolerance specification is always larger than the specified feature-relating tolerance value.

→ Paper gaging of the first segment in a composite tolerance is completed in the same manner as for a single-segment position tolerance.

→ The second segment of a composite position tolerance is paper gaged by plotting the feature position variations of the holes relative to the feature-relating TZF and then placing a set of concentric circles over the plotted variations. There is no requirement to center the concentric circles on the origin for the plotted values.

→ A functional gage for the first segment of a composite tolerance is the same as a functional gage for a single-segment position tolerance.

→ A functional gage for the second segment of a composite tolerance must check the feature-to-feature locations and the orientation of the feature-relating TZF relative to the referenced datums.

→ Pattern-locating tolerance zones are located on a pattern-locating TZF. This framework is defined by the basic dimensions between the features and the basic dimensions that locate the TZF relative to datums.

→ Feature-relating tolerance zones are located on a feature-relating TZF. This framework is defined by the basic dimensions that give the true positions between the features and establish the orientation of the TZF relative to the referenced datums.

→ All features that have position or profile tolerances specified with the exact same datum feature references create a simultaneous requirement (a single pattern). All features that are included in a simultaneous requirement are checked relative to a single datum reference frame.

→ Separate pattern requirements for groups of features can be specified by placing the notation SEPARATE REQUIREMENT or SEP REQT under the feature control frames of the items not acting as a single pattern.

→ Coaxial hole position tolerance requirements may be controlled with a composite position tolerance. The first segment controls the hole locations relative to the pattern-locating TZF that is constrained in translation and rotation relative to the referenced datums. The second segment controls the hole locations relative to the feature-relating TZF that is constrained in rotation relative to the referenced datums.

→ A pattern of symmetrically located features may be position toleranced to control symmetry relative to specified datums.

→ Position or runout tolerances may be used for coaxial features to control the feature axis or surface location relative to a datum axis.

Review Questions

Answer the following questions using the information provided in this chapter. Additional review questions and problems are available in the Study Guide.

Multiple Choice

1. A composite position tolerance has _____ tolerance symbol(s).
   A. one
   B. two
   C. either one or two
   D. None of the above.

2. Datum feature references in the first segment of a composite position tolerance constrain _____ of the pattern-locating TZF relative to the datum reference frame.
   A. translation and rotation
   B. translation
   C. rotation
   D. None of the above.

3. The pattern-locating TZF must be constrained in translation and rotation relative to _____.
   A. the specified datum reference frame
   B. the feature-relating tolerance zones
   C. the production equipment
   D. None of the above.

20. *True or False?* Hole size dimensions are not a good indicator of which holes form a complete pattern.

21. *True or False?* The position tolerance for two groups of holes of different sizes may be checked with one gage if simultaneous requirements apply to the holes.

22. *True or False?* Coaxiality of features, such as the cylinders on a stepped shaft, may be controlled with a position tolerance.

23. *True or False?* Symmetry exists when a feature or a group of features is dimensioned with an offset to one side of a centerline or center plane.

24. *True or False?* The center plane and derived median plane are different geometric entities.

25. *True or False?* Location accuracy requirements for symmetrical features may be specified using position or runout tolerances.

26. *True or False?* The datum feature simulator for a secondary datum feature reference specified RMB must force the workpiece into alignment with the simulator, even if this pulls the workpiece off the primary datum feature simulator.

Fill in the Blank

27. The _____ segment of a composite position tolerance specifies the feature-to-feature location requirements.

28. When paper gaging, _____ circles are used to represent tolerance zone diameters.

29. A functional gage for checking hole locations should include pins that are sized to the _____ of the holes.

30. A notation that states _____ may be placed under single-segment feature control frames if it is necessary to have groups of features act as separate patterns.

31. Coaxial hole patterns on parts such as hinges have holes that lie on a common _____.

32. A condition in which features on each side of a centerline or center plane are equal is known as _____.

33. When a position tolerance is applicable RFS on a symmetrically located rectangular feature of size (such as a slot or rail), the position tolerance defines the allowable variation of the feature's _____ plane.

34. Coaxial tolerancing requirements may be specified with a _____ tolerance at RFS or MMC, a profile tolerance, or a runout tolerance, depending on design function.

35. A secondary or tertiary datum feature of size that is referenced at MMB requires the datum feature simulator be sized to the applicable _____ of the datum feature.

Short Answer

36. Two TZFs are created by a two-segment composite position tolerance. What is the name of the framework created by the first segment of a composite position tolerance?

37. Describe the requirements of the first segment in a composite position tolerance.

38. What is the effect of referencing one or more datum features in the second segment of a composite position tolerance that is applied to coaxial holes?

39. When using a functional gage to inspect a part with holes produced to a position tolerance specified as applicable at MMC, what two conditions must be established by gage pins to verify that the holes are in acceptable locations?

40. How is a datum feature of size simulated if it is referenced as primary, the datum feature reference includes an MMB modifier, and the datum feature has no form tolerance applied?

Application Problems

Some of the following problems require that a sketch be made. All sketches should be neat and accurate. Some problem descriptions require the addition of dimensions for completion of the problem. Apply all required dimensions in compliance with dimensioning and tolerancing requirements. Show any required calculations.

41. Draw a composite position tolerance feature control frame that requires a .027″ diameter pattern-locating tolerance that is related to primary datum feature A, secondary datum feature D, and tertiary datum _____ require a .012″ diameter feature _____ tolerance that is related to p_____ feature A.

Application Problems

Each of the following problems requires that a model, drawing, or sketch be completed as assigned by your instructor. Sketches should be neat and sufficiently accurate to clearly show your ability to apply the required dimensions or tolerances. Dimensions must be applied and tolerance values added where required by the problem description. Unspecified dimensions may be approximated using the given scale for the grid. Dimensions for approximated sizes and locations will be evaluated on the basis of how the dimension is applied, not on the accuracy of the approximated dimension value. Some of the problems require calculations. Dimensions applied on the basis of calculations must be the correct values.

44. Make a drawing or sketch of the given views at full scale. Apply all dimensions.

.188 WIDE X .09 DEEP KEYSEAT
#6 COUNTERDRILL
Grid spacing = .250
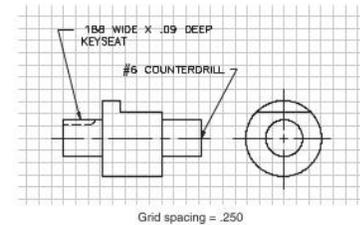
45. Make a model, drawing, or sketch of the given part at full scale. Apply all dimensions. Hole sizes provided in the table are to be applied using leader lines and notations. Either the inch or metric part may be completed. Values shown are not conversions.
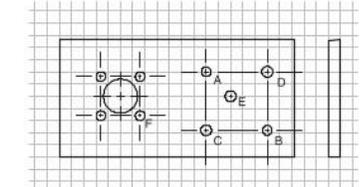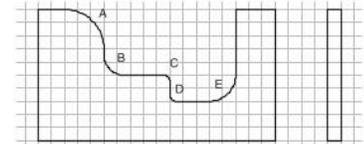

Grid spacing = .250
or
Grid spacing = 10.0 mm

Inch and metric parts are different sizes. Values given are not conversions.

Hole Table

Hole	Inch	Millimeter
A	.250	9.0
B	.250	9.0
C	.280	10.0
D	.280	10.0
E	.280	10.0
F	.188	8.0

46. Make a model, drawing, or sketch of the given part at full scale. Apply all radius dimensions to the given figure. Either the inch or metric part may be completed. Values shown are not conversions.

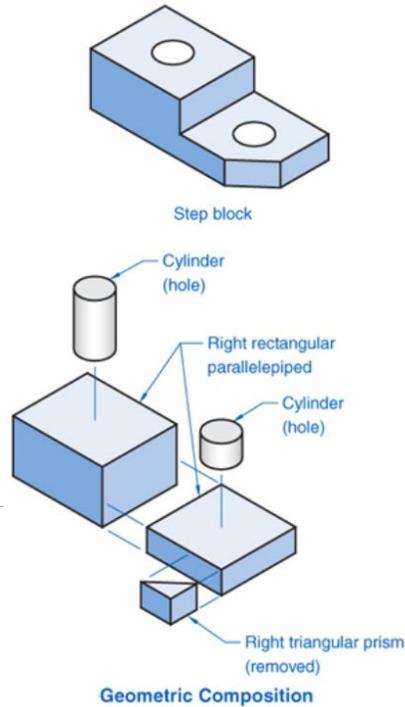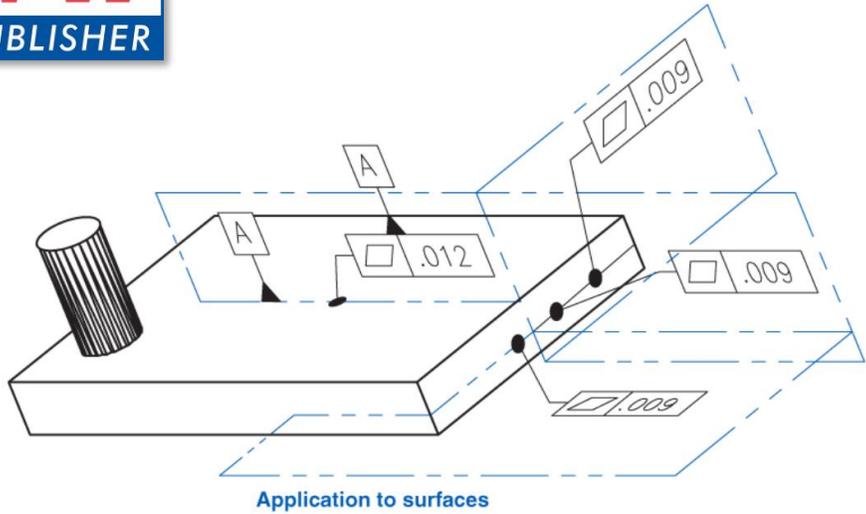| Radius	A	B	C	D	E
Inch	.750	.375	.125	.125	.500
Millimeter	16.0	12.0	2.0	2.0	8.5

Grid spacing = .250
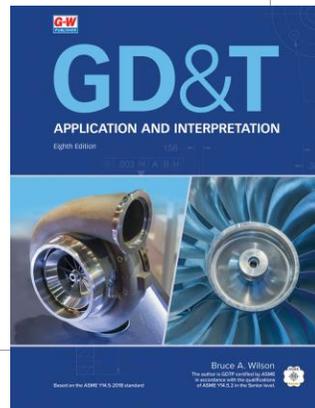or
Grid spacing = 10 mm

HISTORY BRIEF

Fractional Dimensions

The use of common fractions was included in USASI Y14.5-1966. Fractions were widely used in industry and some old drawings may contain them. Fractions should not be used on new product definition that is intended to comply with current standards.

.009

A

A

.012

.009

.009

Application to surfaces

Step block

Cylinder
(hole)

Right rectangular
parallelepiped

Cylinder
(hole)

Right triangular prism
(removed)

Geometric Composition

Activities ▼

GD&T
APPLICATION AND INTERPRETATION
Eighth Edition

Bruce A. Wilson

Based on the ASME Y14.5-2018 standard

Chapter 7: Orientation Tolerances

Reading

Read Chapter 7 of the *GD&T: Application and Interpretation* textbook prior to completing the review exercises.

Learning Outcomes

A combination of activities is required to achieve the following outcomes. Completing the reading assignment and the following review exercises is an important part of achieving the outcomes. Familiarization with the outcomes prior to completion of the reading assignment and review exercises will make mastery of the outcomes easier. After completing the reading assignment and completing the review exercises, you will be able to:

7.1 Identify, apply, and interpret orientation tolerances.

7.2 Complete orientation tolerance specifications including one or two datum feature references.

7.3 Explain the effects of material condition modifiers when orientation tolerances are applied to features of size.

7.4 Calculate the virtual condition for internal and external features of size to which an orientation tolerance is applied.

7.5 Complete tolerance specifications that include orientation and form requirements on a single feature.

Review Exercises

Place your answers in the spaces provided. Show all calculations for problems that require mathematical solutions.
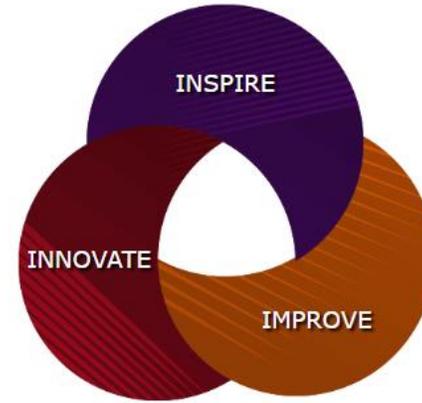
Multiple Choice

1. Which of the following is *not* considered an orientation tolerance?

    A. Angularity
    B. Circular runout
    C. Parallelism
    D. Perpendicularity
    Answer:

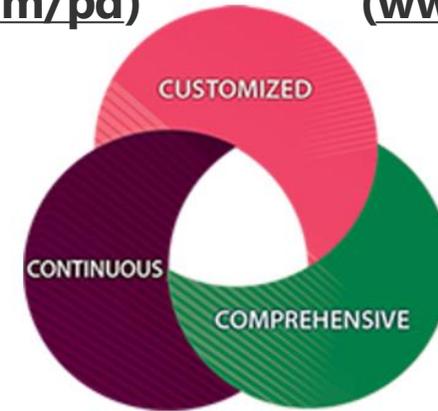2. An orientation tolerance applied to a flat surface or feature of size with parallel

Study Guide Assignments

Tools for Educators:
Activities, Posters, and Professional Development Videos

G-W PUBLISHER

From Here to Career: IT

Type: Infographic / Poster
Grade: 13-14
Subject: Computer Science and Information Technology

Download
Learn More

From Here to Career: Early Childhood Education

Type: Infographic / Poster
Grade: 9-14
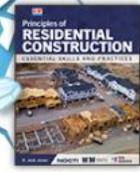Subject: FCS-Education and Training

Download
Learn More

From Here to Career: Automotive

Type: Infographic / Poster
Grade: 13-14
Subject: Transportation, Distribution, and Logistics

Download
Learn More

Snap, Clap, Pop: Construction

Type: Downloadable Activities
Grade: 9-12
Subject: Architecture and Construction

Download
View Product
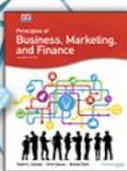
Snap, Clap, Pop: Business, Marketing, Finance

Type: Downloadable Activities
Grade: 9-12
Subject: Business, Marketing, and Finance

Snap, Clap, Pop: Health Science

Type: Downloadable Activities
Grade: 9-12
Subject: Health and Physical Education

---

www.g-w.com/tools-for-educators

G-W PUBLISHER

6-12 Education ▾   College & Career ▾   Resources ▾   Support ▾

Enter Title, Keyword, or ISBN   🔍

Welcome to the G-W Training Center!

A well-prepared educator is at the cornerstone of every successful learning environment—and G-W is here to support you every step of the way. Our resources equip teachers with the knowledge and skills necessary to maximize the impact of our programs in your classrooms. By providing tools that range from simple, user-friendly training resources that directly align to your textbook to high-impact instructional practices, we want every educator to feel confident using G-W resources as a powerful gateway to student-centered learning.

Learn more about Knowledge-Base Articles and using training resources.

Questions? Connect with Support

www.g-w.com/training

G-W needs instructors for:

- Content Reviews
- Digital Reviews
- Authors
- Subject Matter Experts
- Advisory Boards
- Focus Groups

www.g-w.com/engage-with-us

Engage with Us

GoodheartWillcox

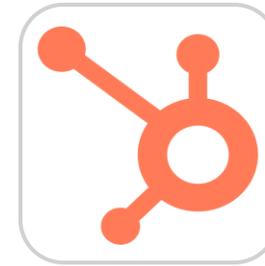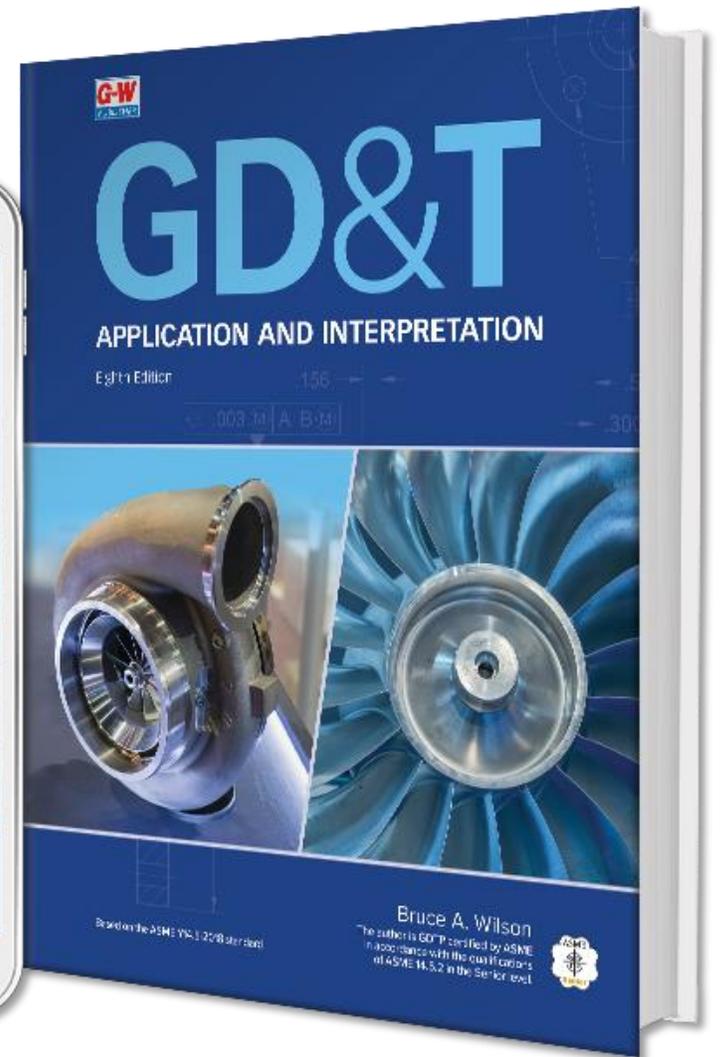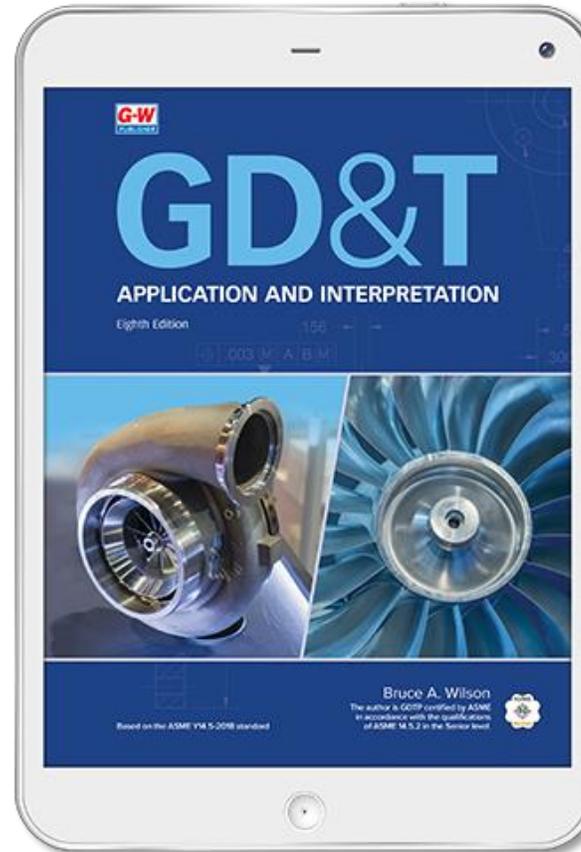**GWPublisher**

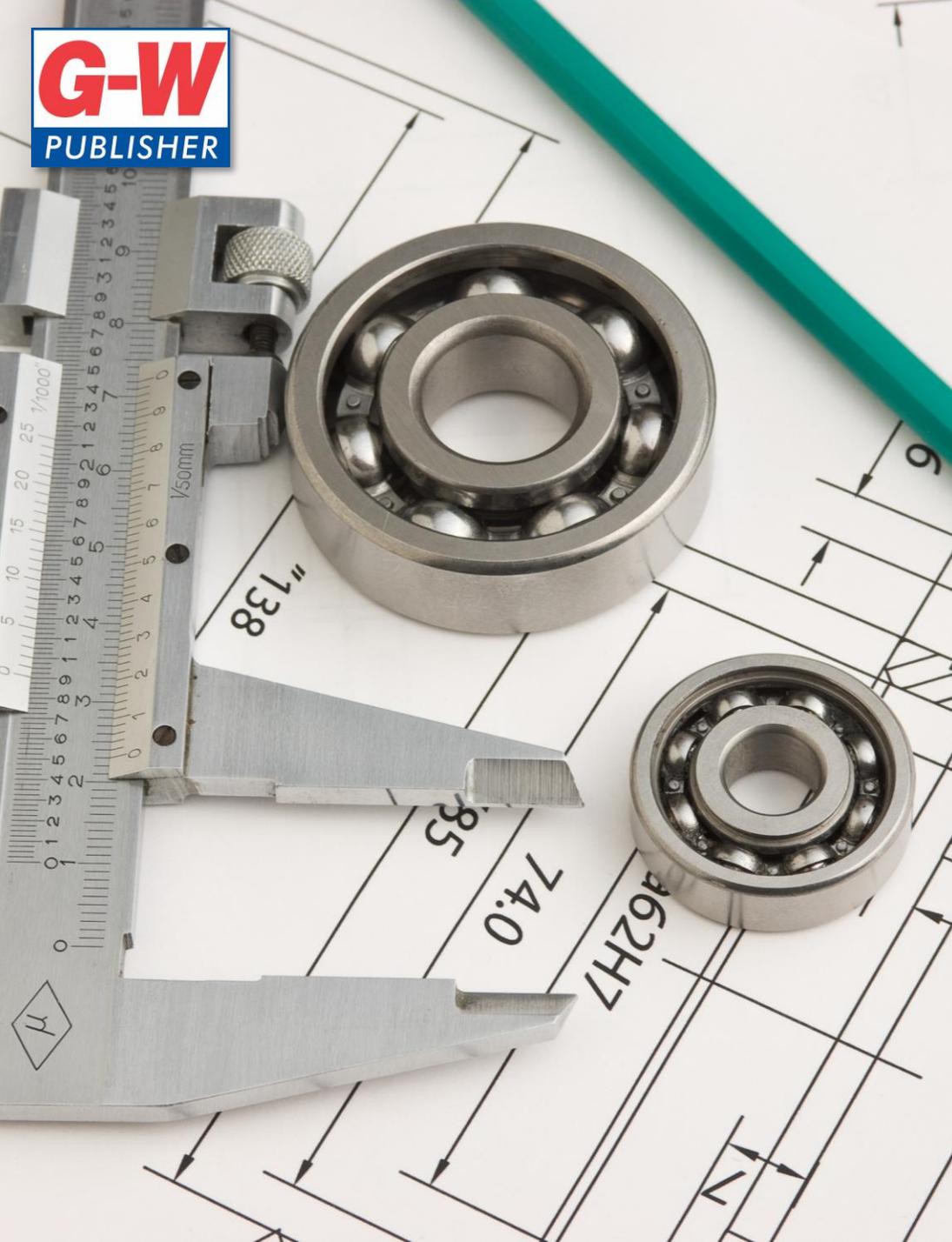**company/gwpublisher**

GWPublisher

Connect with G-W!

blog.g-w.com

GoodheartWillcox

Connect with G-W!

Learn more at <u>**www.g-w.com**</u>