THIRD EDITION

GEOMETRIC DIMENSIONING AND TOLERANCING

FOR MECHANICAL DESIGN

GENE R. COGORNO

McGraw Hill

Geometric Dimensioning and Tolerancing for Mechanical Design

This page intentionally left blank

Geometric Dimensioning and Tolerancing for Mechanical Design

Gene R. Cogorno

Third Edition

McGraw Hill

New York Chicago San Francisco
Athens London Madrid
Mexico City Milan New Delhi
Singapore Sydney Toronto

ISBN: 978-1-26-045379-9
MHID: 1-26-045379-0

The material in this eBook also appears in the print version of this title: ISBN: 978-1-26-045378-2,
MHID: 1-26-045378-2.

eBook conversion by codeMantra
Version 1.0

About the Author

Gene R. Cogorno is a professional educator, speaker, and author with more than 30 years of experience in education and training. He earned both his Bachelor's and Master's degrees in Industrial Education from San Jose State University. Mr. Cogorno developed and taught a practical training program in industrial education, and taught machine technology at San Jose State University. In 1984, he joined FMC Corporation as a Senior Technical Trainer. In 1992, Mr. Cogorno founded Technical Training Consultants, where he teaches courses in geometric dimensioning and tolerancing, tolerance analysis, and blueprint reading.

This page intentionally left blank

Contents

This page intentionally left blank

Preface

This book is written primarily for the learner who is new to the subject of geometric dimensioning and tolerancing (GD&T). The purpose of this book is to teach this graphic language in a way that the learner can easily understand and use in practical applications. This work is intended as a textbook to be used in colleges, universities, technical schools, and corporate training programs. It is intended for use in engineering, design, manufacturing, inspection, and drafting curriculums. This book is also appropriate for a self-study program.

The material in this book is written in accordance with the latest revision of the geometric dimensioning and tolerancing standard, ASME Y14.5-2018. GD&T is a graphic language; in order to facilitate the understanding of this subject, there is at least one drawing to illustrate each concept discussed. Drawings in this text are for illustration purposes only. In order to avoid confusion, only the concepts being discussed are completely toleranced. All of the drawings in this book are dimensioned and toleranced with the inch system of measurement because most drawings produced in the United States are dimensioned and toleranced with this system. The reader is expected to know how to read engineering drawings.

Organization

The discussion of each control starts with a definition and continues with how the control is specified, interpreted, and inspected. There are a sequential review, a series of study questions, and problems at the end of each chapter to emphasize key concepts and to serve as a self-test. This book is logically ordered so that it can be easily used as a reference text.

A Note to the Learner

To optimize the learning process, it is important for the learner to do the following:

1. Preview the chapter objectives, the subtitles, the drawing captions, and the summary.
2. Preview the chapter once again, focusing attention on the drawings and, at the same time, formulating questions about the material.
3. Read the chapter completely, searching for answers to your questions.
4. Underline or highlight important concepts.
5. Answer the questions and solve the problems at the end of the chapter.

Comprehending new information from the printed page is only part of the learning process. Retaining the new information in long-term memory is even more important. In order to optimize the learning process and to drive new information into long-term memory with the least amount of effort, it is suggested that the learner follow these steps:

1. Review all new information at the end of the day.
2. Review it again the next day.
3. Review it again the next week.
4. And, finally, review the new information again the following month.

Review is more than just looking at the information. Review includes rereading main ideas, speaking them out loud, and/or writing them. Some learners learn best by reading, others by hearing, and still others by writing or doing. Everyone learns differently, and some students may learn best by employing a combination of these activities or all three. Learners are encouraged to experiment to determine their own best method of learning. The answers to the questions and problems at the end of the chapters are available on the publisher's and author's websites shown below.

A Note to the Instructor

An Instructor's Guide is available. It includes the following:

1. A course calendar
2. Suggested lecture topics
3. Answers to questions and problems at the end of each chapter
4. Midterm examinations
5. A final examination
6. The answers for the midterm examinations and the final examination

Also, this book is organized in such a way that the instructor can select appropriate material for a more abbreviated course. This text can also be used as supplementary material for other courses, such as mechanical engineering, tool design, drafting, machining practices, and inspection. Using the Instructor's Guide with this text will greatly facilitate the administration of a course in GD&T.

Access to the Instructor's Guide

- Publisher's website: https://www.mhprofessional.com/GDTMD3
- Author's website: http://www.ttc-cogorno.com

Gene R. Cogorno

Acknowledgments

I would like to express particular gratitude to my wife, Marianne, for her support of this project and for the many hours she spent reading and editing the manuscript. Also, thanks go to my son, Steven, for his efforts toward shaping the style of this book. I would like to acknowledge Robert Argentieri, my editor, and the McGraw-Hill Professional staff for their technical contributions and editorial comments. A special thanks goes to James D. Meadows, my first GD&T instructor, for his guidance and support throughout the years. Finally, thank you to the American Society of Mechanical Engineers for permission to reprint excerpts from Dimensioning and Tolerancing, ASME Y14.5-2018.

This page intentionally left blank

Geometric Dimensioning and Tolerancing for Mechanical Design

This page intentionally left blank

CHAPTER 1

Introduction to Geometric Dimensioning and Tolerancing

For many in the manufacturing sector, geometric dimensioning and tolerancing (GD&T) is a new subject. During World War II, the United States manufactured and shipped spare parts overseas for the war effort. Many of these parts, even though they were made to specifications, would not assemble. The military recognized that defective parts caused serious problems for military personnel. After the war, a committee representing government, industry, and education spent considerable time and effort investigating this defective parts problem; this group needed to find a way to ensure that parts would fit and function properly every time. The result was the development of GD&T.

Ultimately, the USASI Y14.5–1966 (United States of America Standards Institute, predecessor to the American National Standards Institute) document was produced based on earlier standards and industry practices. The following are revisions to that standard:

- ANSI Y14.5–1973 (American National Standards Institute)
- ANSI Y14.5M–1982
- ASME Y14.5M–1994 (American Society of Mechanical Engineers)
- ASME Y14.5–2009
- ASME Y14.5–2018

The 2018 revision is the current, authoritative reference document that specifies the proper application of GD&T.

Most government contractors are now required to generate drawings that are toleranced with GD&T. Because of tighter tolerancing requirements, shorter time to production, and the need to communicate design intent more accurately, many companies other than military suppliers are recognizing the importance of tolerancing their drawings with GD&T.

Traditional tolerancing methods have been in use since the mid-1800s. These methods do a good job of dimensioning and tolerancing the size of features and are still used in that capacity today, but they do a poor job of locating and orienting features of size. GD&T is used extensively for tolerancing size, shape, form, orientation, and location of features. Tolerancing with GD&T has a number of advantages over conventional tolerancing methods; three dramatic advantages are illustrated in this chapter.

The purpose of this introductory chapter is to provide an understanding of what GD&T is and why it was developed, when to use it, and what advantages it has over conventional tolerancing methods. With a knowledge of this subject, technical practitioners will be more likely to understand tolerancing in general. With this new skill, engineers will have a greater understanding of how parts assemble, do a better job of communicating design requirements, and ultimately be able to make a greater contribution to their companies' bottom line.

Chapter Objectives

After completing this chapter, the learner will be able to:

- *Define* GD&T
- *Explain* when to use GD&T
- *Identify* three advantages of GD&T over coordinate tolerancing

What Is GD&T?

GD&T is a symbolic language. It is used to specify the size, shape, form, orientation, and location of features on a part. Features toleranced with GD&T reflect the actual relationship between mating parts. Drawings with properly applied geometric tolerancing provide the best opportunity for uniform interpretation and cost-effective assembly. GD&T was created to ensure the proper assembly of mating parts, to improve quality, and to reduce cost.

GD&T is a design tool. Before designers can apply geometric tolerancing properly, they must carefully consider the fit and function of each feature of every part. GD&T, in effect, serves as a checklist to remind the designer to consider all aspects of each feature. GD&T allows the designer to specify the maximum available tolerance and, consequently, design the most economical parts. Properly applied geometric dimensioning and tolerancing ensures that every part will assemble every time.

GD&T communicates design requirements. This tolerancing scheme identifies all applicable datum features, that is, reference surfaces, and the features being controlled to these datum features. A properly toleranced drawing not only is a picture that communicates the shape and size of the part but also tells a story that explains the tolerance relationships between features.

When Should GD&T Be Used?

Many designers ask, when should I use GD&T? Because GD&T was designed to position features of size, the simplest answer is to locate all features of size with GD&T controls. Designers should tolerance parts with GD&T when:

- Drawing delineation and interpretation need to be the same
- Features are critical to function or interchangeability
- It is important to stop scrapping perfectly good parts
- It is important to reduce drawing changes
- Automated equipment is used
- Functional gaging is required
- It is important to increase productivity
- Companies want across-the-board savings

Advantages of GD&T over Coordinate Dimensioning and Tolerancing

Since the middle of the nineteenth century, industry has been using the plus or minus tolerancing system for tolerancing drawings. This system has several limitations. The plus or minus tolerancing system generates rectangular tolerance zones. A rectangular tolerance zone, such as the example in Fig. 1-1, is a boundary within which the axis of a feature that is in tolerance must lie. Rectangular tolerance zones do not have a uniform distance from the center to the outer edges. In Fig. 1-1, from left to right and top to bottom, the tolerance is ± .005; across the diagonals, the tolerance is ± .007. Therefore, when designers tolerance features with a plus or minus .005 tolerance, they must tolerance the mating parts to accept a plus or minus .007 tolerance, which exists across the diagonals of the tolerance zones.

With the plus or minus tolerancing system, features of size can be specified only at the *regardless of feature size* condition. *Regardless of feature size* means that the location tolerance remains the same, ± .005, no matter what size the feature happens to be within its size tolerance. If a hole, like the one in Fig. 1-1, increases in size, it actually has more location tolerance, but, with the plus or minus tolerancing system, there is no way to capture that additional tolerance.

Datum features are usually not specified where the plus or minus tolerancing system is used. Consequently, machinists and inspectors don't know which datum features apply or in what order they apply. In Fig. 1-1, measurements are taken from the lower and left sides of the part. The fact that measurements are taken from these sides indicates that they are datum features. However, since these datum features are not specified, they are called *implied datum features*. Where datum features are implied, the designer has not indicated which datum feature is more important and has not specified whether a third datum feature is included. It would be

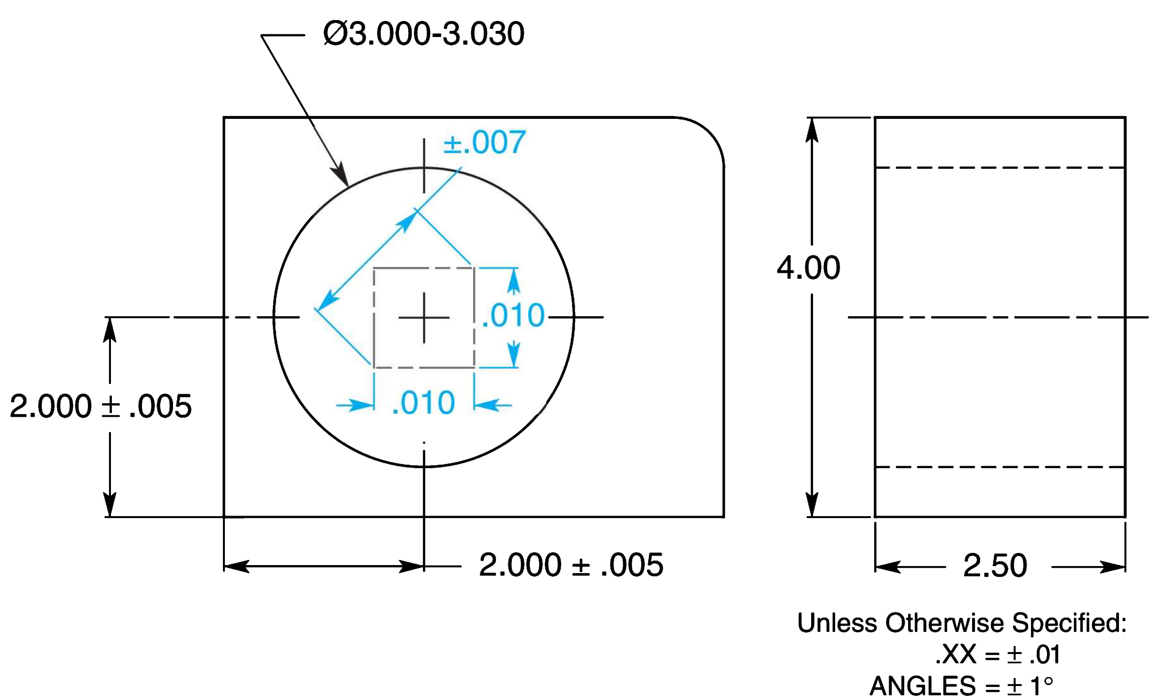

FIGURE 1-1 The traditional plus or minus tolerancing system. The axis of the 3-inch-diameter hole, to be in tolerance, must fall inside of the .010 square tolerance zone.

logical to assume that a third datum feature does exist because the datum reference frame consists of three mutually perpendicular planes, even though a third datum feature is not implied. When locating features with GD&T, there are three important advantages over the coordinate tolerancing system:

- The cylindrical tolerance zone
- The maximum material condition modifier
- Datum features specified in order of precedence

The Cylindrical Tolerance Zone

The cylindrical tolerance zone is located and oriented to a specified datum reference frame. In Fig. 1-2, the tolerance zone is oriented perpendicular to datum plane A and located with basic dimensions to datum planes B and C. There are no tolerances directly associated with a basic dimension; consequently, basic dimensions eliminate undesirable tolerance stack-up. Because the cylindrical tolerance zone is established at a basic 90° angle to datum plane A and extends through the entire length of the feature, it easily controls the orientation of the axis.

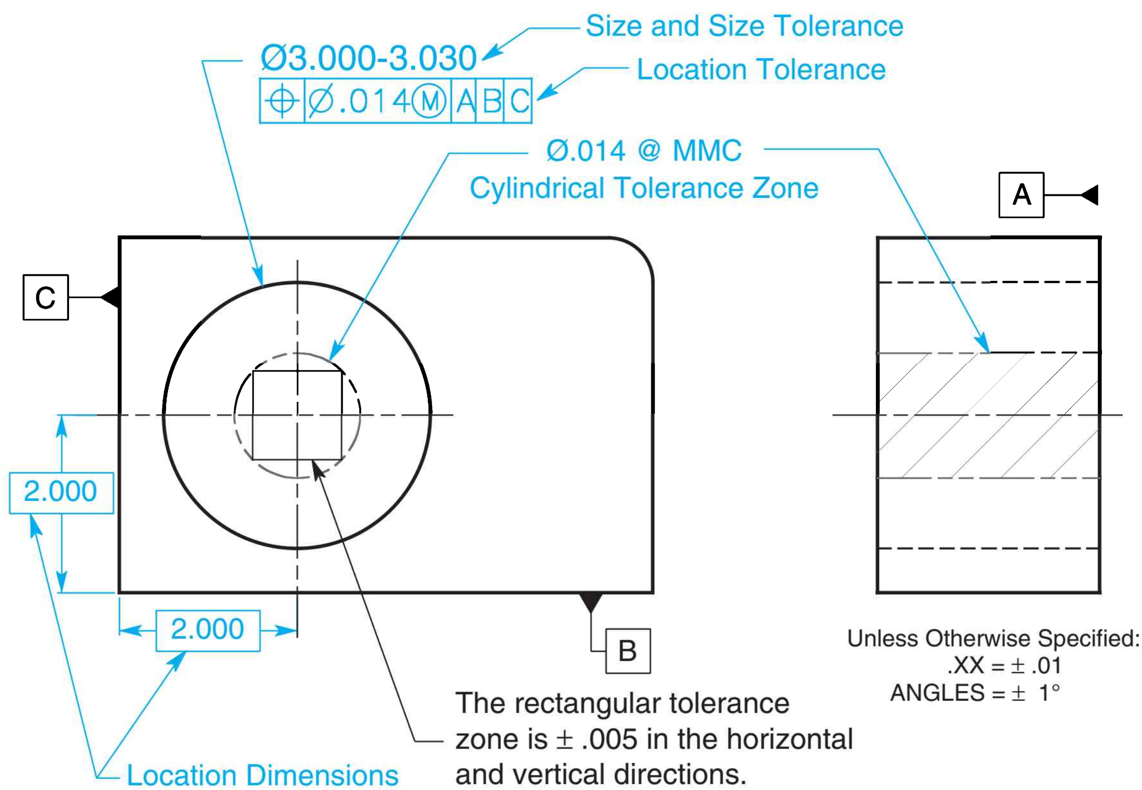

FIGURE 1-2 The cylindrical tolerance zone compared with the rectangular tolerance zone.

Unlike the rectangular tolerance zone, the cylindrical tolerance zone defines a uniform distance from true position, the theoretically perfect center of the hole, to the tolerance zone boundary. When a .014-diameter cylindrical tolerance zone is specified about true position, there is a

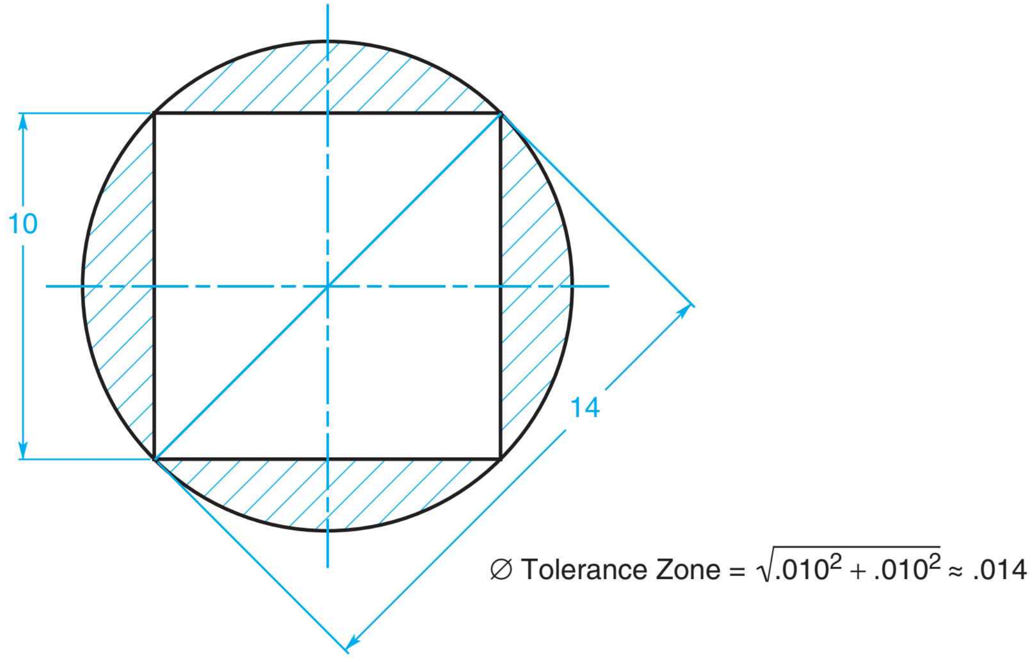

FIGURE 1-3 A cylindrical tolerance zone provides a uniform distance from the axis to the tolerance zone boundary.

tolerance if .007 from true position in all directions. A cylindrical tolerance zone circumscribed about a square tolerance zone, such as the one in Fig. 1-3, has 57% more area than the square tolerance.

The Maximum Material Condition Modifier

The maximum material condition symbol (circle M) in the feature control frame is a modifier. It specifies that as the hole in Fig. 1-2 increases in size, a bonus tolerance is added to the tolerance stated in the feature control frame.

The limit tolerance in Fig. 1-4 indicates that the hole size can be as small as 3.000 (maximum material condition) and as large as 3.030 (least material condition) in diameter. The geometric tolerance specifies that the hole be positioned with a cylindrical tolerance zone of .014 in diameter when the hole is produced at its maximum material condition size. The tolerance zone is oriented perpendicular to datum plane A and located with basic dimensions to datum planes B and C. Since the .014-diameter tolerance is specified with a maximum material condition modifier, circle M, a bonus tolerance is available. As the hole size in Fig. 1-2 departs from maximum

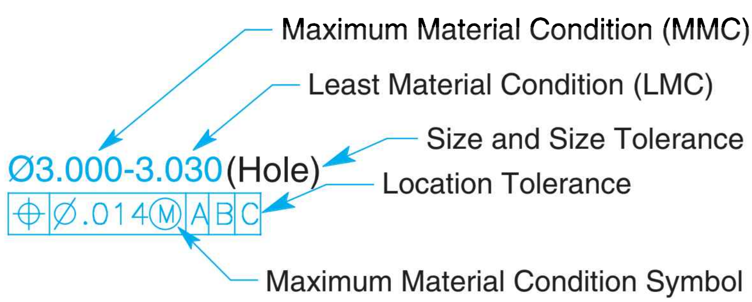

FIGURE 1-4 The size, size tolerance, and feature control frame for the hole in Fig. 1-2.

material condition toward least material condition, additional location tolerance, called *bonus tolerance*, is allowed in the exact amount of such departure. If the hole specified by the feature control frame in Fig. 1-4 is actually produced at a diameter of 3.020, the total available tolerance is a diameter of .034.

	Actual Mating Envelope	3.020
Minus	Maximum Material Condition	−3.000
	Bonus Tolerance	.020
Plus	Geometric Tolerance	+.014
	Total Positional Tolerance	.034

The maximum material condition modifier allows the designer to capture all of the available tolerance.

Datum Features Specified in Order of Precedence

Datum features are not usually specified on drawings toleranced with the coordinate dimensioning system. The lower and left edges on the drawing in Fig. 1-5 are implied datum features because the holes are dimensioned from these edges. But which datum feature is more important, and is a third datum plane included in the datum reference frame? A rectangular part such as this is usually placed in a datum reference frame consisting of three mutually perpendicular intersecting planes. When datum features are not specified, machinists and inspectors are forced to make assumptions that could be very costly.

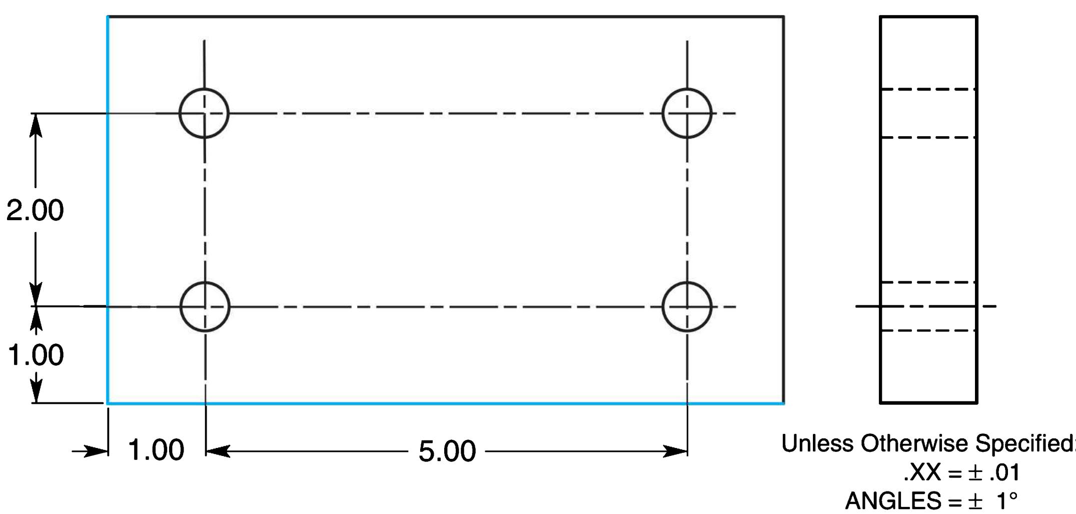

Figure 1-5 A conventional drawing with no datum features specified.

The parts placed in the datum reference frames in Fig. 1-6 shows two interpretations of the drawing in Fig. 1-5. With the traditional method of tolerancing, it is not clear whether the lower edge of the part should be resting against the horizontal surface of the datum reference frame as in Fig. 1-6*A* or if the left edge of the part should be contacting the vertical surface of the datum reference frame as in Fig. 1-6*B*.

Manufactured parts are not perfect. It is clear that, when drawings are dimensioned with traditional tolerancing methods, a considerable amount of information is left to the machinists'

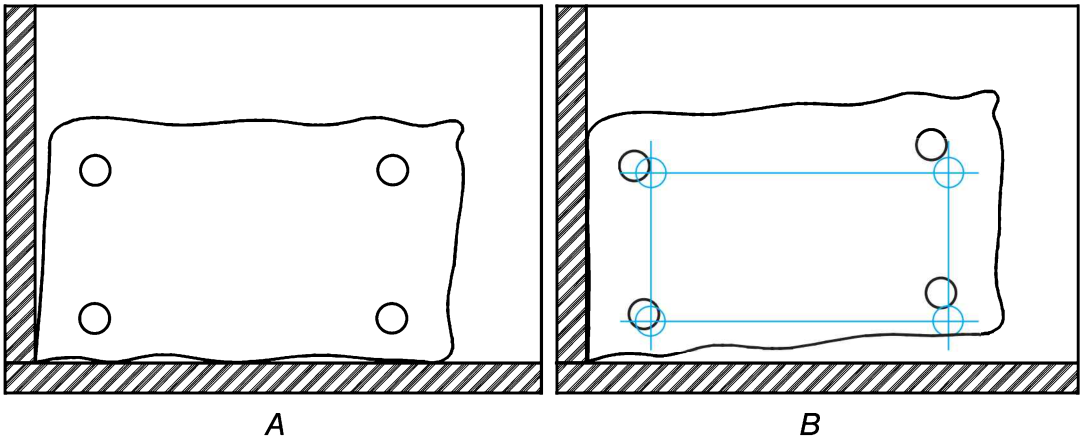

Figure 1-6 Possible datum feature interpretations of the drawing in Fig. 1-5.

and inspectors' judgment. If a part is to be inspected the same way every time, the drawing must specify how the part is to fit in the datum reference frame. Each datum feature must be specified in the feature control frame in its proper order of precedence.

Summary

- GD&T is a symbolic language used to specify the size, shape, form, orientation, and location of features on a part.
- GD&T was created to ensure the proper assembly of mating parts, to improve quality, and to reduce cost.
- GD&T is a design tool.
- GD&T communicates design requirements.
- This text is based on the standard *Dimensioning and Tolerancing* ASME Y14.5–2018.
- The cylindrical tolerance zone defines a uniform distance from true position to the tolerance zone boundary.
- The maximum material condition symbol in the feature control frame is a modifier that allows a bonus tolerance.
- All of the datum features must be specified in order of precedence.

Chapter Review

1. ________________________________ is the current, authoritative reference document that specifies the proper application of GD&T.

2. GD&T is a symbolic language used to specify the ______________, ______________, ______________, ______________, and ______________ of features on a part.

3. Features toleranced with GD&T reflect the ________________________________ between mating parts.

4. GD&T was created to ensure the proper assembly of ________________, to improve ________________, and to reduce ________________.

5. Geometric tolerancing allows the maximum available ________________ and, consequently, the most ________________ parts.

6. Plus and minus tolerancing generates a ________________ shaped tolerance zone.

7. ________________ generates a cylindrical shaped tolerance zone to control an axis.

8. If the distance across a square tolerance zone is ± .005, or a total of .010, what is the approximate distance across the diagonal? ________________

9. The ________________ defines a uniform distance from true position to the tolerance zone boundary.

10. Bonus tolerance equals the difference between the actual mating envelope size and the ________________.

11. While processing, a rectangular part usually rests against a ________________ consisting of three mutually perpendicular intersecting planes.

CHAPTER 2

Dimensioning and Tolerancing Fundamentals

Many people know how to design parts and make drawings, yet they lack the basic knowledge to produce engineering drawings that conform to industry standards. Nonconforming drawings can be confusing, cause misunderstanding, and produce unacceptable parts. This chapter will familiarize the reader with some of the lesser-known but important standards-based dimensioning and tolerancing practices. All of the drawings in this book are dimensioned and toleranced with the inch system of measurement because most drawings produced in the United States are dimensioned with this system. Metric dimensioning is shown for illustration purposes only.

Chapter Objectives

After completing this chapter, the learner will be able to:

- *Identify* fundamental drawing rules
- *Demonstrate* the proper way to specify units of measurement
- *Demonstrate* the proper way to specify dimensions and tolerances
- *Interpret* limits
- *Explain* the need for dimensioning and tolerancing on CAD/CAM database models

Fundamental Drawing Rules

Dimensioning and tolerancing must clearly define engineering intent and shall conform to the following rules:

1. Each feature must be toleranced. Those dimensions specifically identified as reference, maximum, minimum, or stock do not require the application of a tolerance.
2. Dimensioning and tolerancing must be complete so that there is full understanding of the characteristics of each feature. Values may be expressed in an engineering drawing or in a CAD product definition data set specified in ASME Y14.41. Neither scaling nor assumption of a distance or a size is permitted.
3. Each necessary dimension of an end product must be shown or defined by model data. No more dimensions than those necessary for complete definition shall be given. The use of reference dimensions on a drawing should be minimized.

4. Dimensions must be selected and arranged to suit the function and mating relationship of a part and must not be subject to more than one interpretation.
5. The drawing should define a part without specifying manufacturing methods.
6. Nonmandatory processing dimensions must be identified by an appropriate note, such as NONMANDATORY (MFG DATA).
7. Dimensions should be arranged to provide required information for optimum readability.
8. Dimensions in orthographic views should be shown in true profile views and refer to visible outlines. When dimensions are shown in models, the dimension must be applied in a manner that shows the true value.
9. Wires, cables, sheets, rods, and other materials manufactured to gage or code numbers must be specified by linear dimensions indicating the diameter or thickness. Gage or code numbers may be shown in parentheses following the dimension.
10. An implied 90° angle always applies where centerlines and lines depicting features are shown on orthographic views at right angles and no angle is specified.
11. An implied 90° basic angle always applies where centerlines of features or surfaces shown at right angles on an orthographic view are located or defined by basic dimensions and no angle is specified.
12. A zero basic dimension always applies where axes, center planes, or surfaces are shown coincident on orthographic views and geometric tolerances establish the relationship between the features.
13. Unless otherwise specified, all dimensions and tolerances are applicable at 68°F (20°C). Compensation may be made for measurements made at other temperatures.
14. Unless otherwise specified, all dimensions and tolerances apply in a free-state condition, except for restrained nonrigid parts.
15. Unless otherwise specified, all tolerances and datum features apply for the full depth, length, and width of the feature.
16. Dimensions and tolerances apply only at the drawing level where they are specified. A dimension specified for a given feature on one level of a drawing is not mandatory for that feature at any other level.
17. Unless otherwise specified by a drawing/model note or reference to a separate document, the as-designed dimension value does not establish a functional or manufacturing target.
18. Where a coordinate system is shown on the orthographic views or in the model, it must be right-handed unless otherwise specified. Each axis must be labeled and the positive direction shown.
19. Unless otherwise specified, elements of a surface include surface texture and flaws. All elements of a surface must be within the applicable specified tolerance zone boundaries.

Units of Linear Measurement

Units of linear measurement are typically expressed in either the inch system or the metric system. The system of measurement used on the drawing must be specified in a note, usually in the title block. A typical note reads, UNLESS OTHERWISE SPECIFIED, ALL

DIMENSIONS ARE IN INCHES (or MILLIMETERS, as applicable). Some drawings have both inch and metric systems of measurement on them. On drawings dimensioned with the inch system where some dimensions are expressed in millimeters, the millimeter values are followed by the millimeter symbol, mm. On drawings dimensioned with the millimeter system where some dimensions are expressed in inches, the inch values are followed by the inch symbol, IN.

Specifying Linear Dimensions

Where specifying decimal inch dimensions on drawings as described in Table 2-1:

- A zero *is never placed* before the decimal point for values less than 1 inch. Some designers routinely place zeros before the decimal point. This practice is incorrect and confusing for the reader who knows the proper convention.
- A dimension is specified with the same number of decimal places as its tolerance even if zeros need to be added to the right of the decimal point.

	Decimal Inch Dimensions		**Millimeter Dimensions**	
	Correct	**Incorrect**	**Correct**	**Incorrect**
1.	**.25**	0.25	**0.25**	.25
2.	**4.500 ± .005**	4.5 ± .005	**4.5**	4.500
3.			**4**	4.000

Table 2-1 Decimal Inch and Millimeter Dimensions

Where specifying millimeter dimensions on drawings as described in Table 2-1:

- A zero *is placed* before the decimal point for values less than 1 millimeter.
- Zeros *are not added* to the right of the decimal point where dimensions are a whole number plus some decimal fraction of a millimeter. (This practice differs where tolerances are written bilaterally or as limits. See "Specifying Linear Tolerances" below.)
- Neither a decimal point nor a zero is shown where the dimension is a whole number.

Specifying Linear Tolerances

There are three types of direct tolerancing methods:

- Limit tolerancing
- Plus and minus tolerancing
- Geometric tolerancing directly applied to features

Where using limit dimensioning, the high limit or largest value is placed above the lower limit. If the tolerance is written on a single line, the lower limit precedes the higher limit separated by a dash. With plus and minus dimensioning, the dimension is followed by a plus and minus sign and the required tolerance.