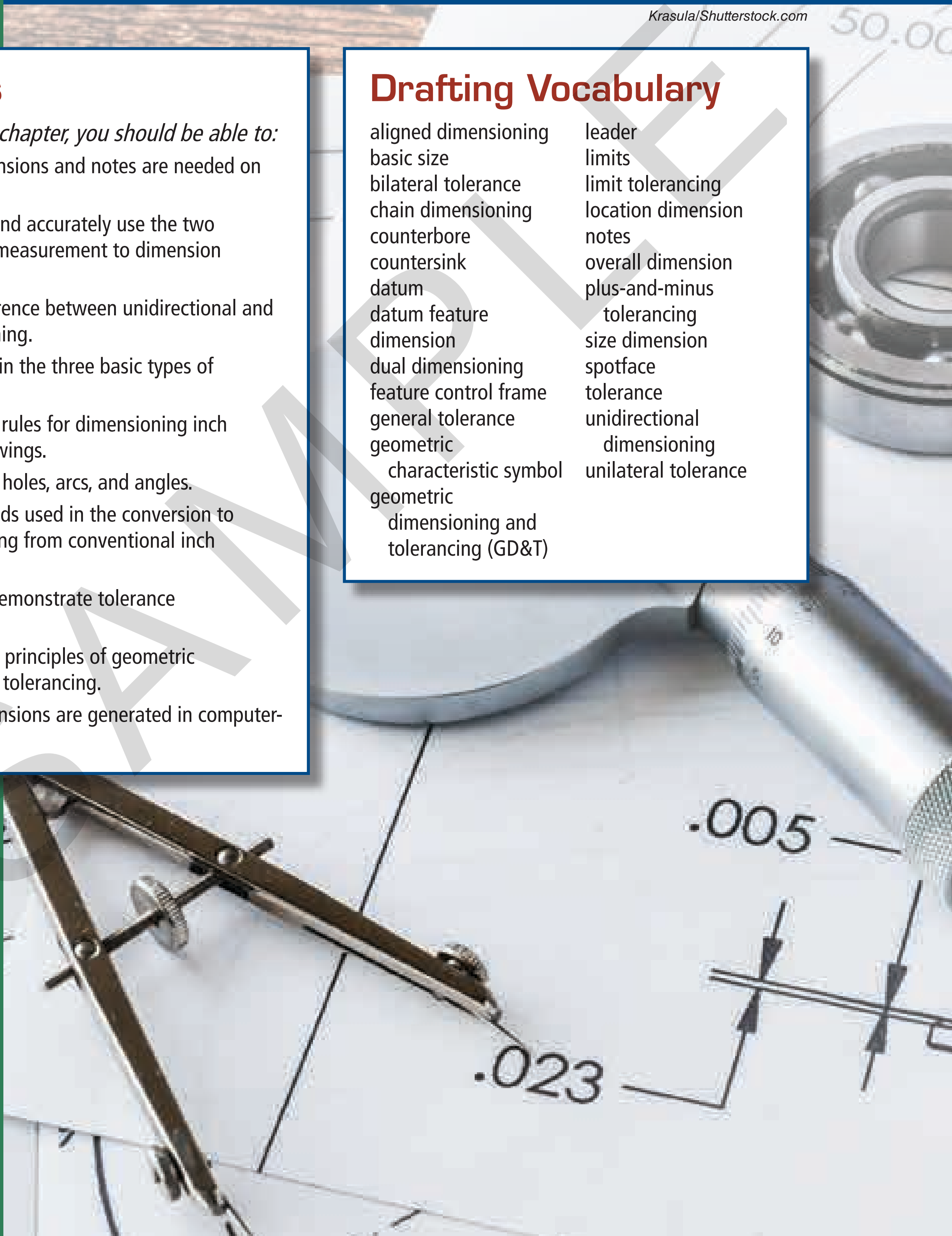
Chapter 10

Dimensioning



Krasula/Shutterstock.com

Objectives

After studying this chapter, you should be able to:

- Explain why dimensions and notes are needed on drawings.
- Identify, explain, and accurately use the two systems of linear measurement to dimension drawings.
- Describe the difference between unidirectional and aligned dimensioning.
- Identify and explain the three basic types of dimensions.
- Apply the general rules for dimensioning inch and/or metric drawings.
- Dimension circles, holes, arcs, and angles.
- Explain the methods used in the conversion to metric dimensioning from conventional inch dimensioning.
- Understand and demonstrate tolerance dimensioning.
- Describe the basic principles of geometric dimensioning and tolerancing.
- Explain how dimensions are generated in computer-aided drafting.

Drafting Vocabulary

aligned dimensioning
basic size
bilateral tolerance
chain dimensioning
counterbore
countersink
datum
datum feature
dimension
dual dimensioning
feature control frame
general tolerance
geometric characteristic symbol
geometric dimensioning and tolerancing (GD&T)
leader
limits
limit tolerancing
location dimension
notes
overall dimension
plus-and-minus tolerancing
size dimension
spotface
tolerance
unidirectional dimensioning
unilateral tolerance

Before You Begin

Arrange a study session to read the chapter aloud with a classmate. At the end of each section, discuss any words you do not know. Take notes of words you would like to discuss in class.

Employability

Email Etiquette

In everyday life and in business, there is a proper way to behave. This is known as *etiquette*. Rules of etiquette have been adapted to today's business needs, including proper conduct when writing email. In many companies, employees may write email messages daily to coworkers, vendors, and customers. It is important to follow the proper etiquette when you write messages that represent your company. The rules of email etiquette include:

- Use a tone that is appropriate to your relationship and to the writing situation.
- Do not use emoticons in a business email message.
- Let the reader know if you are sending an attachment by mentioning it in the message.
- Do not send or forward personal messages, jokes, chain letters, or spam.
- Never use profanity or any other type of derogatory language.
- Never respond in anger to an email message. Wait until you are calm enough to respond in a professional manner.
- Avoid overuse of email. Stop to think whether a phone call or personal visit would be more productive.
- Use the "blind carbon copy" window for large external mailings to protect recipients' privacy.
- Avoid using all capital letters in your message. All capital letters implies shouting.

Activity

Imagine that you have received an email message from a client. In the message, the client notes that he has received your latest drawing and writes several paragraphs about how poor it looks. Many of the items the client objects to are things that he originally wanted to be included in the drawing. He also threatens to speak to your supervisor about your work. You know that you based the drawing on the client's original wishes and that it is as good as possible given the client's directions. Write an email message back to the client. Maintain a professional manner and try to keep the client from taking his business elsewhere.

If an object is to be manufactured according to the designer's specifications, the person making the product usually needs more information than that furnished by a scale drawing of its shape. Dimensions and drawing notes help provide this information, **Figure 10-1**.

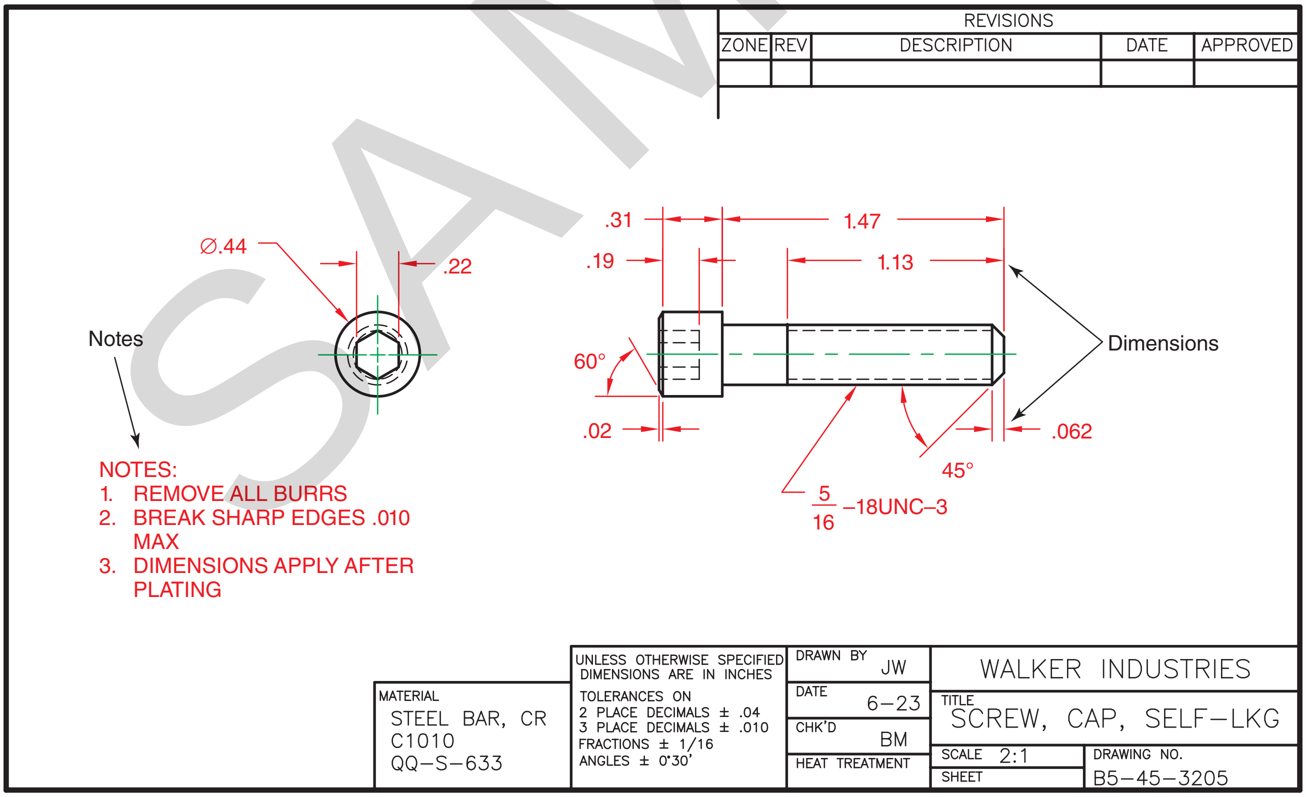
It is important to realize that the main purpose for creating, dimensioning, and noting a drawing is to communicate the size and shape of the product so that the person making the product can do so as easily and accurately as possible. Hence, the most pertinent information needed for assembly must be provided. It is the job of the drafter to define what that information is and to provide it in the clearest and most efficient way.

Dimensions define the size, orientation, and location of the features of an object and give the overall size of the object. ***Notes*** provide additional information not found in the dimensions. In manual drafting, dimensions and notes are added to a drawing by hand. In CAD drafting, drawings are dimensioned using CAD software commands and tools.

Systems of Measurement Used in Dimensioning

As discussed in Chapter 4, measurements on drawings are made in US Customary or SI Metric units. Dimensions are placed on drawings using the appropriate units in one of the two systems of measurement. Sometimes, drawings are dimensioned using both systems. In some cases, drawings are completed using one system for most of the dimensions and the other system for selected dimensions. When this occurs, special rules apply. This is discussed later in this chapter.

When using the US Customary system, drawings are dimensioned in inches and feet. The dimensions may be represented in one of two ways. They may be decimal inch dimensions (such as 1.5 or 2.75) or fractional inch dimensions (such as 1-1/2 or 2-3/4). Decimal dimensioning is the standard practice recommended by the ASME drafting standard. In mechanical drafting, drawings based on the US Customary system are most commonly



Goodheart-Willcox Publisher

Figure 10-1. A typical drawing from industry with dimensions and notes.

dimensioned in decimal inches. Decimal inch dimensioning is convenient because decimals are easier to add, subtract, multiply, and divide. However, fractional inch dimensioning is still widely employed. This type of dimensioning is primarily used on architectural drawings, but it is also sometimes used on engineering drawings. Decimal inch and fractional inch dimensioning are both used in examples in this text.

Fractional inch units are often used when making drawings of objects made from materials that typically cannot be machined to very fine tolerances (such as wood). Decimal inches are generally used for drawings of objects made from materials such as metal or plastic. These materials can be machined to very fine tolerances.

When using the SI Metric system, drawings are commonly dimensioned in millimeters (mm). There are special rules for dimensioning metric drawings. Metric dimensioning is discussed later in this chapter.

Unidirectional and Aligned Dimensioning

Dimensions are placed on the drawing in one of two orientations. They are drawn in either a unidirectional or aligned manner, **Figure 10-2**. Unidirectional dimensioning is preferred.

In ***unidirectional dimensioning***, dimensions are placed horizontally so that they are read from the bottom of the drawing. In ***aligned dimensioning***, dimensions are placed parallel to the dimension line. The numerals are read from the bottom and from the right side of the drawing.

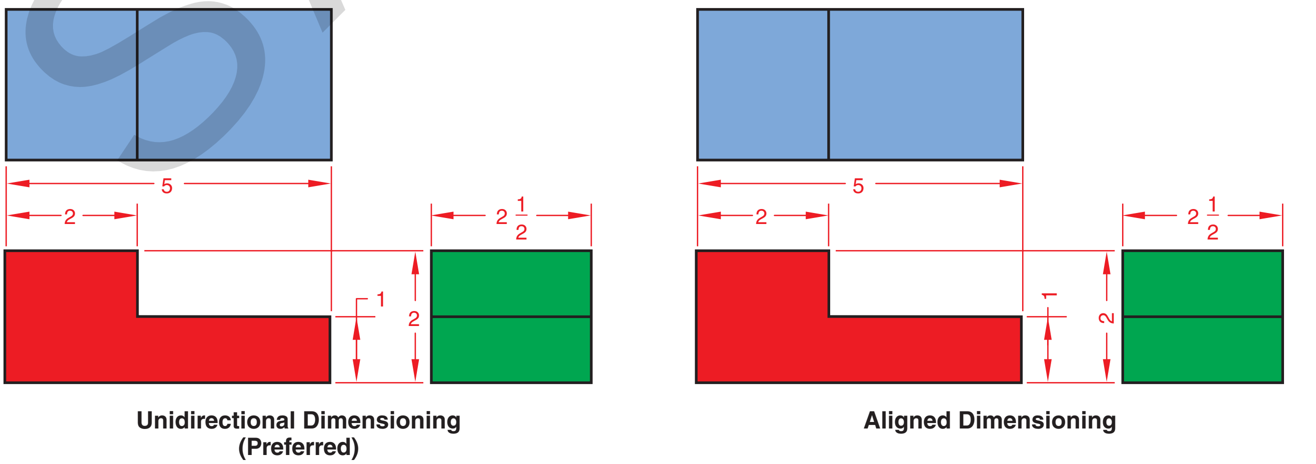
Regardless of the method used, dimensions shown with leaders are lettered parallel to the bottom of the drawing. The same is true for all notes.

Dimensioning a Drawing

From your study of the alphabet of lines, you will remember that special lines are used for dimensioning, **Figure 10-3**. These include dimension lines, extension lines, and leaders.

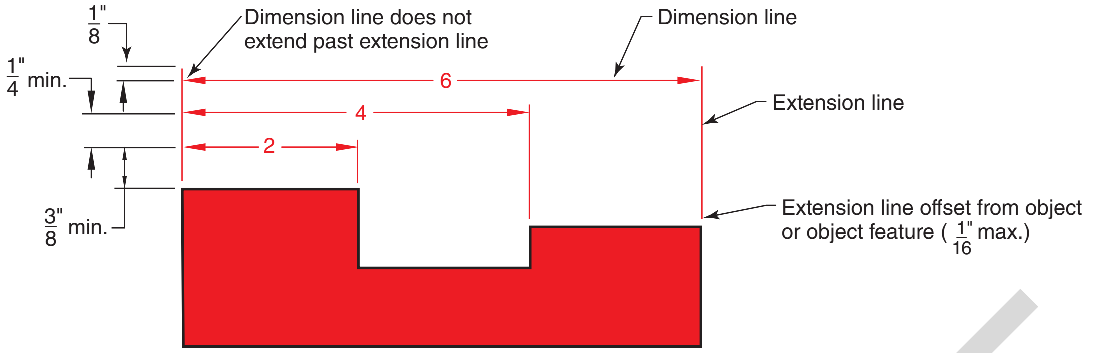
The dimension line is a thin black line used to indicate linear distance as it relates to a given object. It is drawn to the same weight as a centerline. It should be thin enough to contrast with the visible lines. It is usually broken near the center for the insertion of the actual lettered dimension. Although this is the most common practice, placement of the lettered dimension may vary. In some cases, the lettered dimension may be placed outside the extension lines. Refer to **Figure 10-1**.

The dimension line is capped at each end with arrowheads or some other type of terminator. The type of terminator used generally depends on the drafting discipline. For example, tick marks are used as dimension line terminators on architectural drawings.



Goodheart-Willcox Publisher

Figure 10-2. The two accepted methods for dimensioning drawings. The unidirectional method is preferred.



Goodheart-Willcox Publisher

Figure 10-3. Lines used for linear dimensioning. Note how dimension lines and extension lines contrast in thickness with visible lines.

Extension lines are drawn to the same weight as dimension lines. They extend the dimension beyond the outline of the view so that the dimension can be read easily. Extension lines indicate the beginning and ending points of linear distances. An extension line begins 1/16″ (1.5 mm) away from the point, edge, intersection, or feature of the object and extends about 1/8″ (3 mm) past the last dimension line it is used for. An extension line may originate within a view and extend across a view. However, it should not start or stop at a visible line. Extension lines may cross any kind of a line except a dimension line. Hence, it is recommended that smaller dimensions (size or location dimensions) should be placed nearest the view, while the larger dimensions (overall dimensions) should be placed farther from the view.

Spacing between dimensions in a view may vary depending on the drawing. Generally, to avoid crowding, the dimension line for a dimension should be placed at least 3/8″ (10 mm) from the object. The spacing between dimensions should be at least 1/4″ (6 mm).

A ***leader*** is another type of dimension line. It is an angular line used to point out special characteristics of objects. Leaders are commonly used to specify sizes of circles and arcs. They are also used with notes. A leader does not require extension lines because it usually does not reference a linear distance. The arrowed portion of the leader is always angular. It is inclined at an angle ranging from 15° to 75° (30°, 45°, and 60° lines are common). A leader is never vertical or horizontal, **Figure 10-4**. When associated with a circle or an arc, the leader always points at or intersects the primary center of the round feature. **Figure 10-4A** shows the correct orientation for leader-directed dimensions applied to circles or arcs. In most cases, a leader is drawn with a small horizontal tail called a shoulder. When the leader is to the left of the note, the shoulder connects to the beginning of the first line of information associated with the leader. When the leader is to the right of the note, the shoulder connects to the end of the last line of information associated with the leader. The shoulder typically extends 1/4″ (6 mm) from the leader line. See **Figure 10-5**.

Arrowheads for leaders and dimension lines are drawn freehand and should be carefully made, **Figure 10-6**. The solid arrowhead is generally preferred. It is made narrower and slightly longer than the open arrowhead. For most applications, 1/8″ (3 mm) long arrowheads are satisfactory. To save time, preprinted (dry transfer) arrowheads may be applied by the drafter instead of manually constructing them. Also, when dimensioning linear distances, make certain that the tips of the arrows on the dimension lines touch the extension lines but do not extend past them. Refer to **Figure 10-3**.

Types of Dimensions

As previously discussed, the drafter's primary concern when dimensioning an object is to convey the basic information needed to manufacture the part. For the beginning drafter, a very basic and easy-to-understand

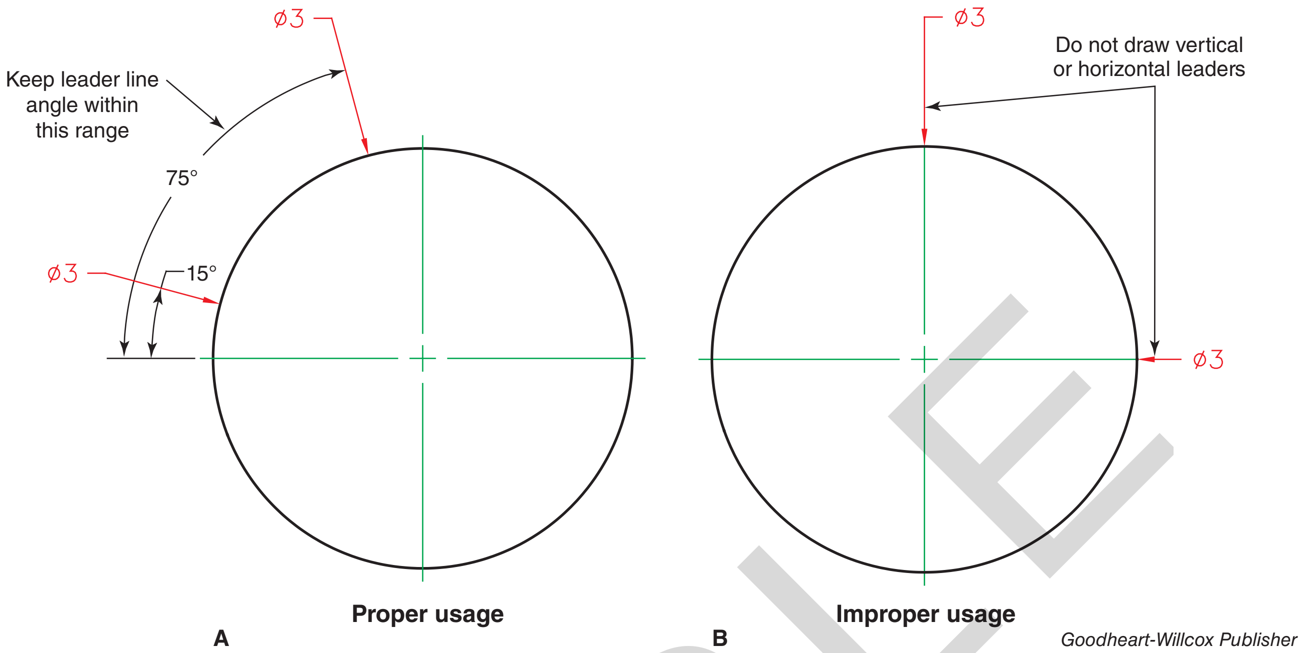


Figure 10-4. Leaders are used to connect dimensioning information to object features. A—Properly drawn leaders. Leader lines are drawn at an angle ranging from 15° to 75°. B—Leader lines should never be vertical or horizontal.

way to determine this information is to identify all object distances as one of three types—overall, location, and size. These object descriptions are known as overall, location, and size dimensions.

Overall dimensions provide the overall object size and tell the manufacturer how large a piece of material to use to make the object. ***Location dimensions*** indicate where particular features lie along or within the object. ***Size dimensions*** specify how large the particular features are, **Figure 10-7**. As objects become more complex, identifying these dimensions becomes more difficult. However, if you can visualize an object, analyze its features, and think in these terms, you can learn to become very proficient and consistent in applying dimensions to drawings.

Simply put, when dimensioning an object, apply the following steps:

1. To identify the overall dimensions, tell how large the object is overall. Describe the positive mass. Provide the overall width, height, and depth dimensions.

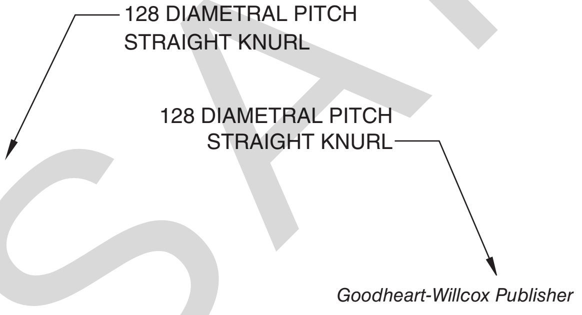


Figure 10-5. The leader shoulder connects to the first or last line of information depending on the placement of the leader with the note.

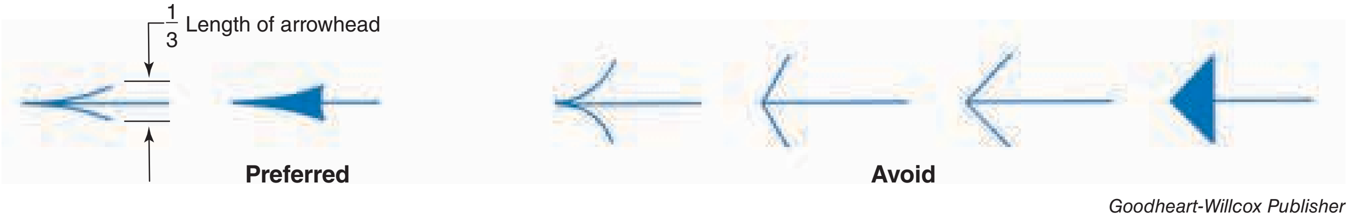


Figure 10-6. Arrowheads are drawn 1/8″ long. The solid arrowhead is generally preferred.

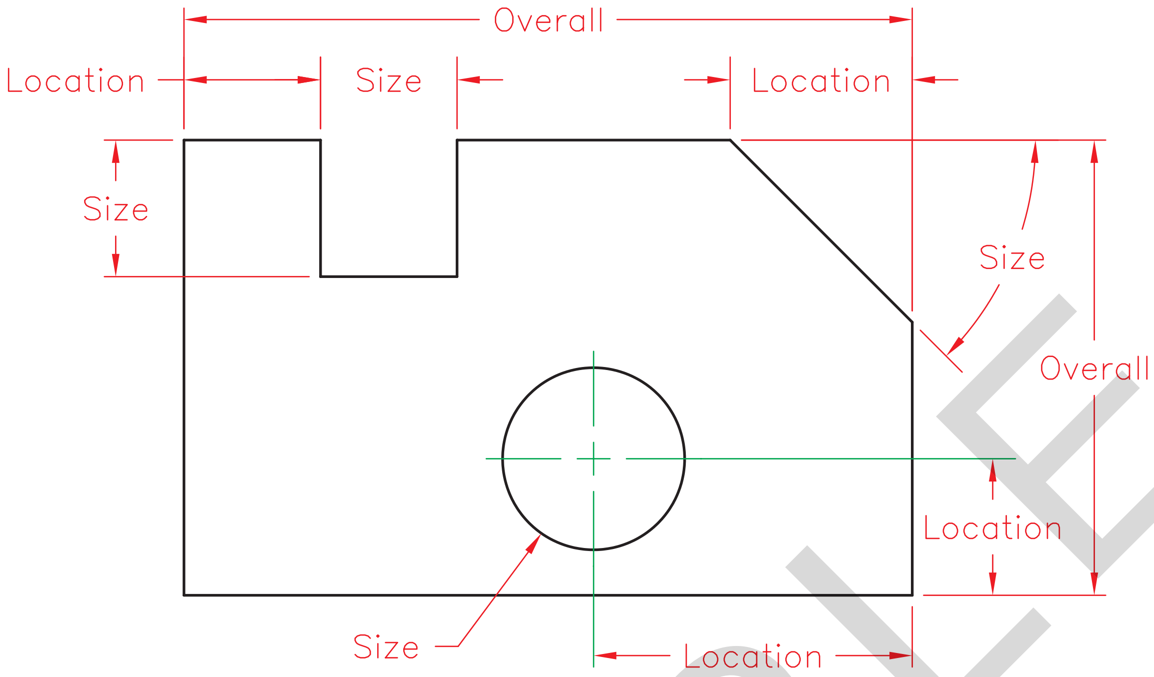


Figure 10-7. Object dimensions can be classified as overall, location, or size dimensions. The three features of this object are described by the size dimensions.

2. To identify the location dimensions, tell where the features of the object are. Locate the negative masses.
3. To identify the size dimensions, tell how large the features (negative masses) are.

While following the previous steps, make accurate and consistent use of several fundamental dimensioning rules. These are discussed next.

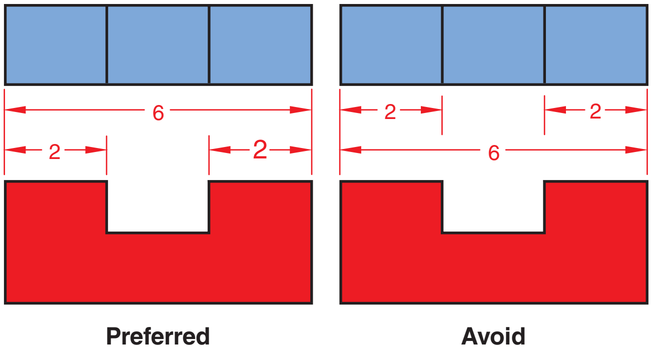
Note

When identifying location dimensions, as much as possible, try to locate the features with respect to the original edges of the original mass of the object. Try *not to* locate them with respect to other features. Doing this has a tendency to either dictate the order of procedure for manufacturing or cause the person making the part to perform mental mathematics during the layout procedure.

Fundamental Rules for Dimensioning

For clarity, dimensions should conform to the following general rules:

1. Place location and size dimensions on the views that show the true shape of the feature being dimensioned (the primary views). See **Figure 10-8**. The true shape of the feature appears on the view that shows the feature as negative mass. Do not locate or give the size of features in secondary views. A secondary view is a view in which the feature is not visible (in other words, where it is hidden). In most cases, dimensioning to hidden lines is not an acceptable practice. Of course, as with some rules, there are exceptions. Your instructor will point these out when appropriate.

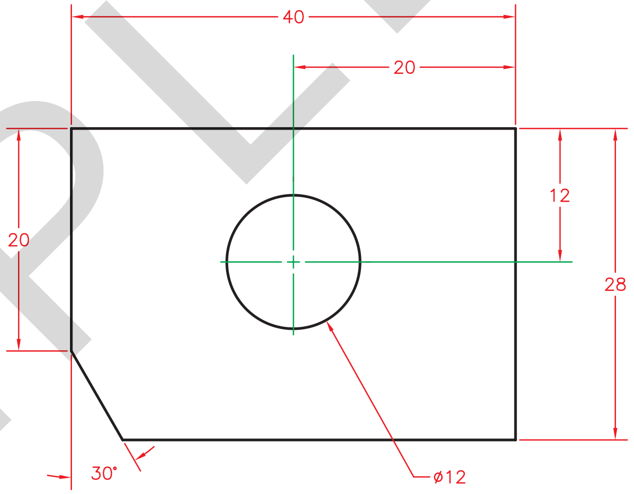


Goodheart-Willcox Publisher

Figure 10-8. Location and size dimensions are placed in views that show features in their true shape.

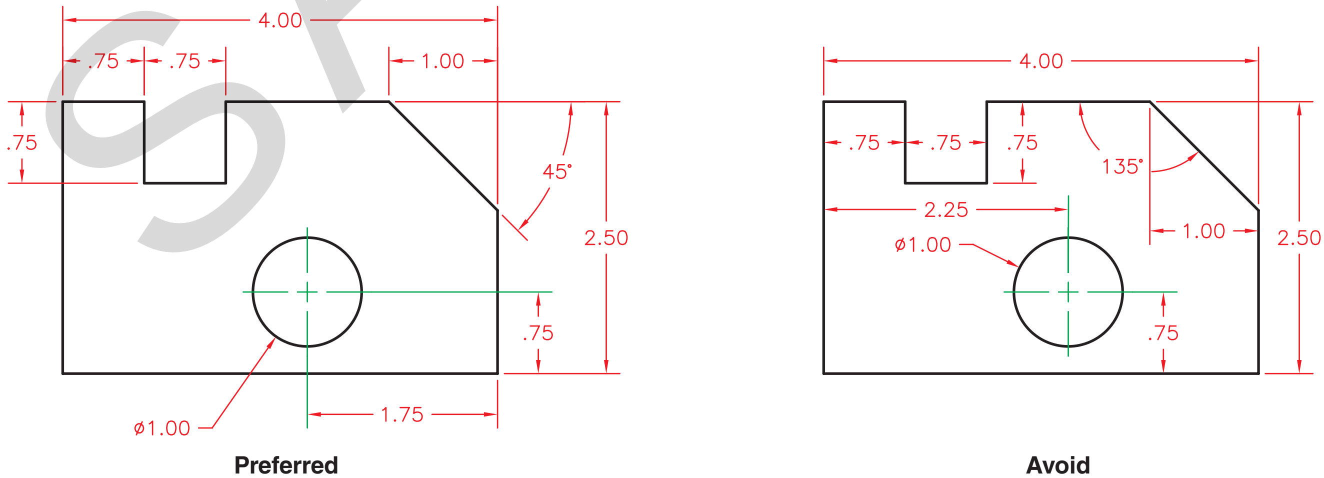
2. Unless absolutely necessary, dimensions should not be placed within the outlines (outer visible lines) of the views, **Figure 10-9**. Instead, extension lines should be used to extend the profiles and features of the object to around the perimeter for the purpose of linear dimensioning. Centerlines may be extended and used as extension lines since they are the same line weight, **Figure 10-10**. Dimensioning within the visible lines of views becomes acceptable when objects are very complex. However, the focus of this text is to teach basic dimensioning. More complex examples will be part of your future drafting studies.

3. If possible, dimensions should be grouped together rather than scattered about the drawing. See **Figure 10-11**. The illustration shown is a good example of grouping linear distances between views or on two sides of a view. Grouping also applies to sizes and information attached to leaders. When grouping dimensions, try not to extend extension lines for linear distances more than halfway across any given view, **Figure 10-12**. If this is required, it is probably best to locate or give the size of the feature along the nearer side of the object.



Goodheart-Willcox Publisher

Figure 10-10. Notice how the centerlines are extended and used as extension lines for dimensioning.



Goodheart-Willcox Publisher

Figure 10-9. Dimensions should not be placed within the visible lines of views. They should be placed with extension lines around the perimeter.

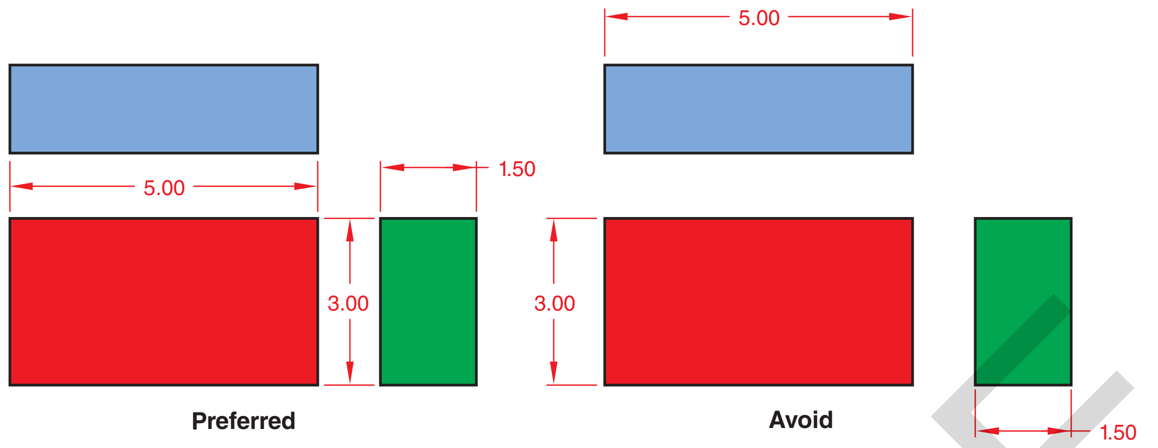


Figure 10-11. Keep dimensions grouped for easier understanding of the drawing.

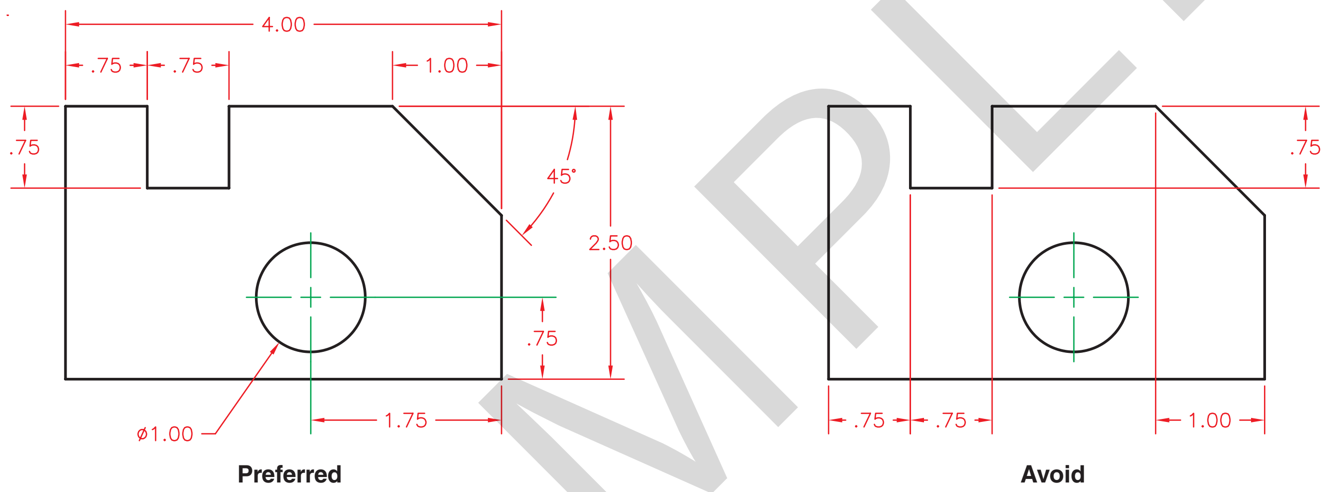
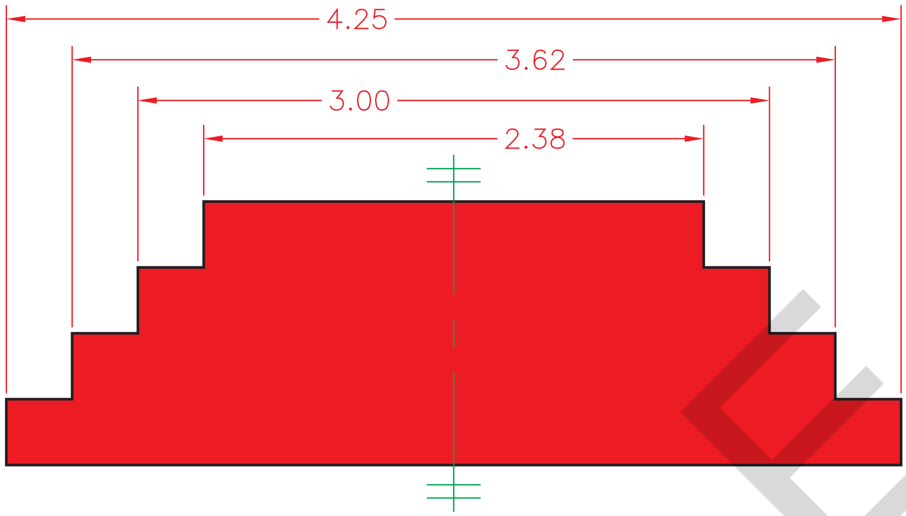


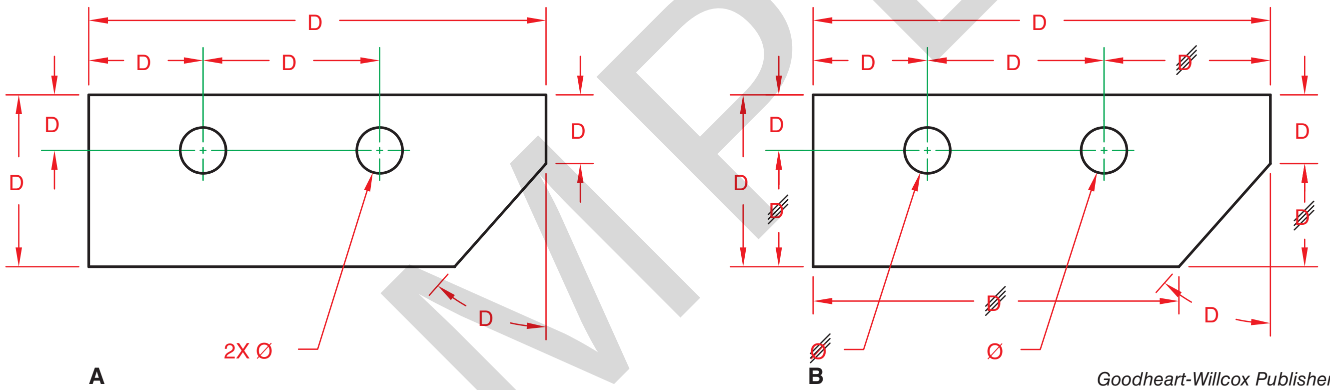
Figure 10-12. Do not extend extension lines more than halfway across a view.

4. Dimensions must be complete. No measuring or scaling of the drawing should be necessary for manufacturing the object. It should be possible to determine sizes and shapes without assuming any measurements.
5. Draw dimension lines parallel to the direction of measurement. If there are several parallel dimension lines, the numerals should be staggered to make them easier to read, **Figure 10-13**.
6. Dimensions should not be duplicated unless they are absolutely necessary to the understanding of the drawing. Omit all unnecessary dimensions, **Figure 10-14**. In mechanical drafting, it is common to provide an overall dimension and only enough location and size dimensions to fully describe the object. When several continuous dimensions are drawn in a straight line or row, the least important distance is typically omitted. Placing several dimensions in a straight line along an object to locate successive features is known as ***chain dimensioning***. Refer to **Figure 10-14**. In the example shown, three location dimensions are omitted from dimensional chains. Many times, an omitted dimension is a distance that relates one feature to another. If the information is not useful for the manufacture of the product, it is omitted.



Goodheart-Willcox Publisher

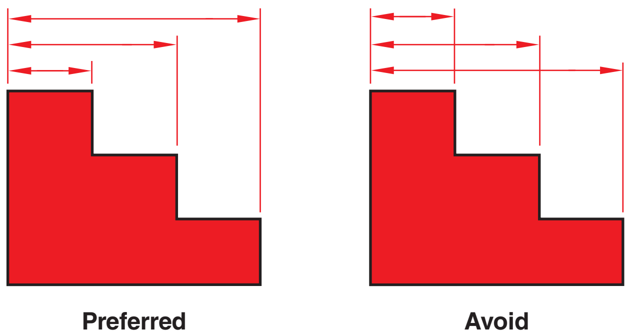
Figure 10-13. When there are several parallel dimension lines, numerals should be staggered for easier reading. The symmetry centerline indicates that the dimensions are centered with respect to each other.



Goodheart-Willcox Publisher

Figure 10-14. Avoid duplicating dimensions unless they are necessary to understand the drawing. A—Properly dimensioned object. B—Unnecessary dimensions are removed.

7. Plan your work carefully so that the extension lines do not cross dimension lines, **Figure 10-15**. Smaller (size or location) dimensions should be placed nearest the view, while the larger (overall) dimensions should be placed farther from the view.
8. When all dimensions on a drawing are in inches, the inch symbol (″) should not be used. However, the drawing should have a note stating “UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN INCHES” (or “MILLIMETERS,” as applicable). Dimensions on metric drawings are in millimeters (mm) unless otherwise noted. Special rules for dimensioning metric drawings are discussed later in this chapter.



Goodheart-Willcox Publisher

Figure 10-15. Crossing lines can be kept to a minimum if the shortest dimensions (size or location dimensions) are placed next to the object outline. Longer dimensions (overall dimensions) are placed farther from the drawing.

9. Numerals and fractions must be drawn to the proper height in relation to one another, **Figure 10-16**. Proper sizes for lettering fractional numbers are discussed in Chapter 8.

General Rules for Inch Dimensioning

As previously discussed, mechanical drawings based on the US Customary system are most commonly dimensioned in decimal inches. Decimal inch dimensioning should conform to the following rules:

1. A zero should not be used before the decimal point for values less than 1″. For example:
   .6, not 0.6
2. In drawings with geometric dimensioning and tolerancing, a dimension is expressed to the same number of decimal places as its tolerance. Geometric dimensioning and tolerancing is discussed later in this chapter. In inch tolerance dimensioning, zeros are added to the right of the decimal point for decimal inch values where necessary. Hence, if the tolerance is in three-place decimals, all dimensions will be in three places:
   .375 and .500, not .375 and .5

Fractional inch dimensioning is less common than decimal inch dimensioning in mechanical drafting. However, this type of dimensioning should conform to the following rule:

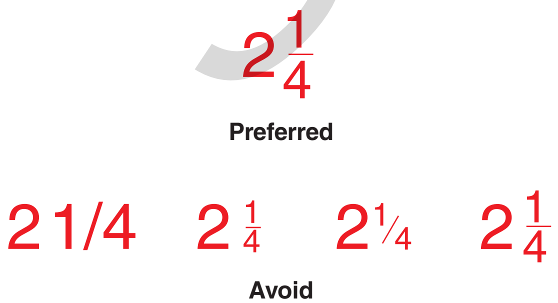
1. All fractions should be reduced to their lowest common denominator:
   1/2, not 4/8 or 8/16
   3/4, not 6/8 or 12/16

Dimensioning Circles, Holes, and Arcs

Many products are manufactured from parts that contain circles, cylindrical solids, round holes, or arcs in their design, **Figure 10-17**. These shapes are usually created using one or more manufacturing processes. Processes such as cutting, reaming, boring, turning, drilling, spotfacing, counterboring, and countersinking are used to produce round shapes, **Figure 10-18**.

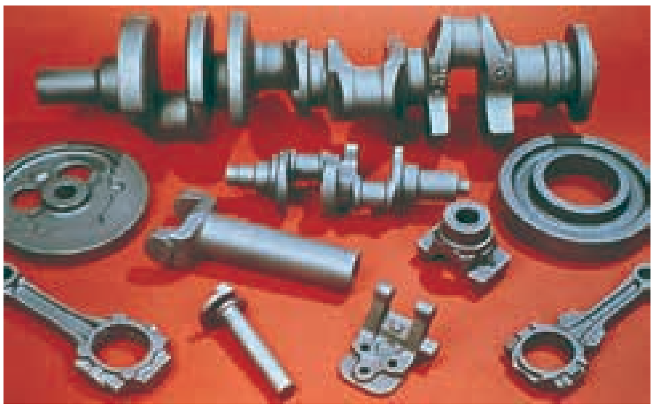
When dimensioning circles, holes, and arcs, it is important that the information given conveys the proper manufacturing procedure. The information may describe either a drilling or cutting procedure. If certain guidelines are followed, the correct procedure will be conveyed, and the correct tool will be chosen for the job.

When dimensioning an arc greater than 180°, dimension it as a diameter. This conveys a drilling procedure. When the drawing is used to make the part, the correct size and type of tool will be selected for manufacture. When dimensioning an arc 180° or smaller,



Goodheart-Willcox Publisher

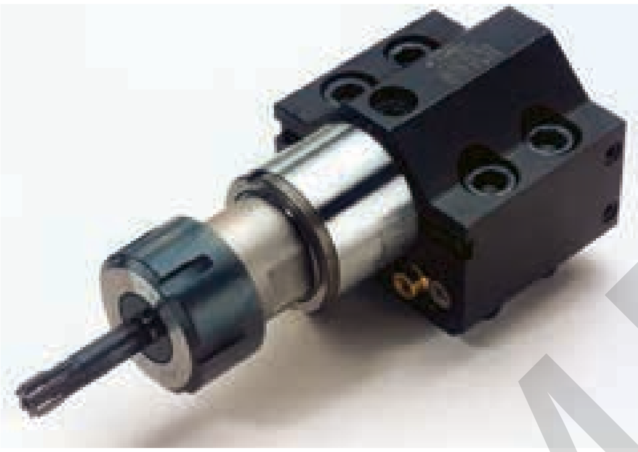
Figure 10-16. Numerals and fractions must be drawn to the proper size and orientation.



American Foundry Society

Figure 10-17. These automobile castings are typical of manufactured parts that contain circles, round holes, and arcs in their design.

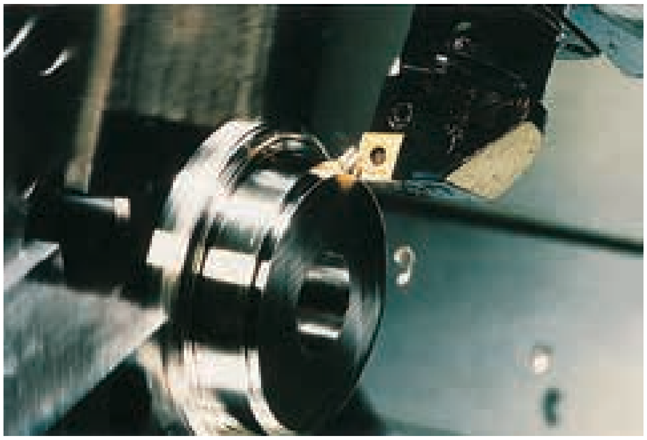
A *Komet of America, Inc.*



B *Komet of America, Inc.*

C *Hougen Manufacturing, Inc.*

D *Komet of America, Inc.*



E *Sandvik Coromant Co.*

Figure 10-18. Machine tools and operations used to produce hole features. A—A cutting tool with a drill bit. B—A cutting tool with a reamer bit. Reaming produces a very accurately sized hole. The hole is first drilled slightly undersize before reaming. C—Countersinking prepares a hole to receive a flathead screw. D—Boring is an internal machining operation. E—Turning work on a lathe.

Academic Link

Standardization in Industry

The need for standardization in manufacturing, making drawings, and dimensioning dates back many years. The Industrial Revolution created a demand for standards because of the need to manufacture interchangeable parts and also because of the existence of many different manufacturers. Standards were adopted and used in Europe before they came into use in the United States.

In the late 1700s, the SI Metric system was developed by French scientists and instituted as the world standard. The system was based on the meter, which was made equal to one-ten millionth of the distance from the North Pole to the equator. Other units of measure in the metric system, such as the liter and gram, were based on the meter.

The SI Metric system was specifically developed to address the need for standardization. By contrast, the US Customary system had evolved from the English system of weights and measures. This system standardized traditional units of measure, such as the inch, foot, and pound.

Most of the world today uses the metric system. Although it has yet to gain widespread acceptance in the United States, there have always been efforts to bring metric standards into practice in the US. The SI Metric system was legalized as the weights and measures standard by the US in the late 1800s. Also, the Metric Conversion Act of 1975 was intended to facilitate the conversion of inch-based standards to metric standards.

The metric system is used today by the US in international commerce. Industry uses drawings in metric units, and automobile mechanics and other technicians must be able to use metric tools. In all US automobiles, speedometers have readouts in both miles per hour and kilometers per hour. Can you think of other examples where the metric system is used?

dimension it as a radius. This conveys a cutting procedure. When the drawing is used to make the part, the correct layout will be completed and the appropriate cutting tool will be selected for the job.

The ASME drafting standard sets guidelines on how to “call out,” or specify, a diameter or radius dimension. The symbol ∅ indicates that the dimension is a diameter. The symbol is placed *before* the size.

Circles and round holes are dimensioned by giving the diameter. They should be dimensioned using one of the methods shown in **Figure 10-19**. The methods shown are recommendations based on the size of the feature. Note the different conventions used for placing the leader with the dimension. Each method corresponds to a specific diameter range for the feature. While dimensioning standards vary, these practices are common (the guidelines shown are not ASME standards).

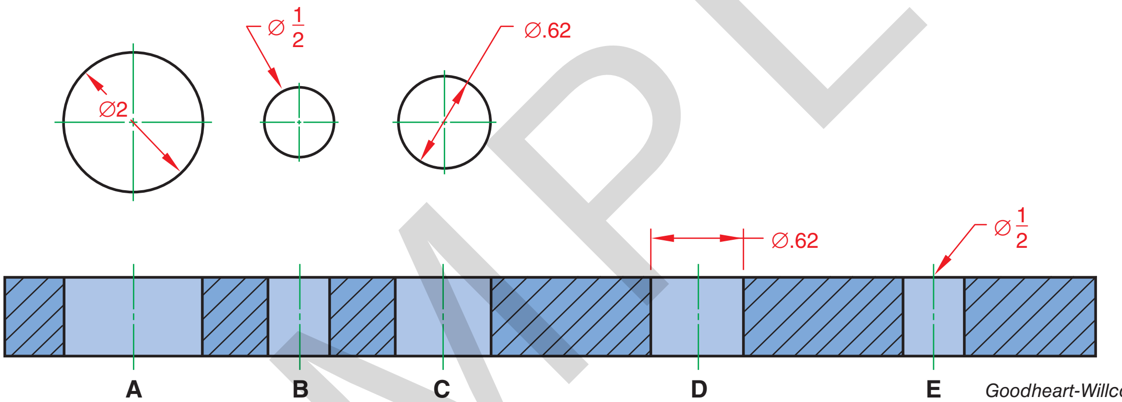
The dimensioning method shown in **Figure 10-19A** is commonly used for diameters of 1-1/2″ or greater. The method shown in **Figure 10-19B** is commonly used for diameters smaller than 3/4″. The method shown in **Figure 10-19C** is commonly used for diameters ranging from 3/4″ to 1-1/2″. The methods shown in **Figures 10-19D** and **10-19E** are for use only when dimensioning sectional views. Sectional views are discussed in Chapter 11.

Different conventions may be used for dimensioning diameters depending on the situation. For instance, if several concentric circles are being dimensioned with diameters greater than 1-1/2″ but increasing in size in relatively small increments, the drafter may choose not to use the common method for large diameters (diameters 1-1/2″ and larger) but rather one of the others to improve clarity. The key is to select the convention that most clearly does the job in any given situation.

Where it is not clear that a hole goes through the part, the abbreviation THRU follows the dimension. See **Figure 10-20A**. In cases where the hole does not go through the entire part, the depth of the hole is provided. See **Figure 10-20B**. A special symbol is placed before the depth size.

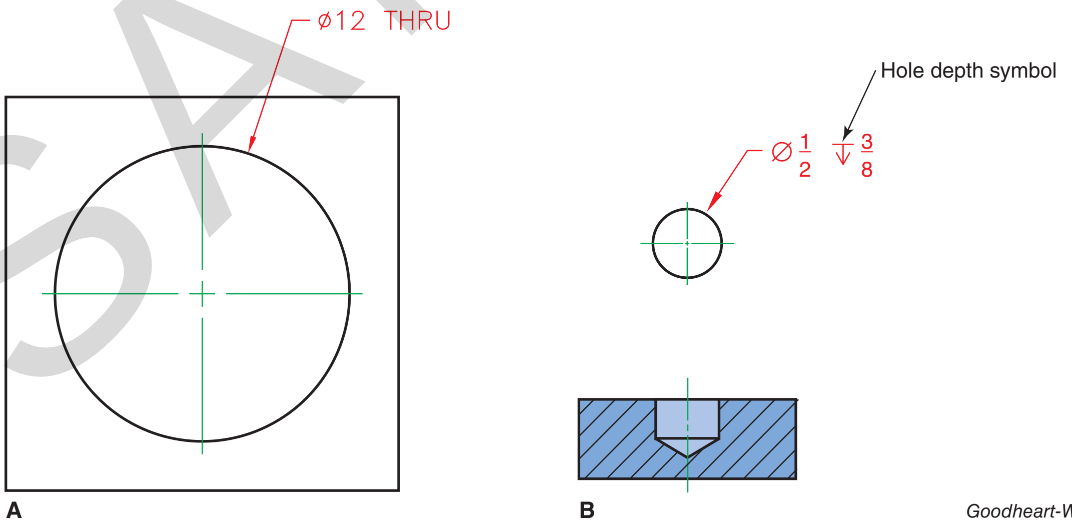
Symbols are also used to show whether the hole is to be countersunk, spotfaced, or counterbored in manufacturing. See **Figure 10-21**. A ***countersink*** is a chamfered recess at the end of a smaller hole used to receive the head of a fastener. A ***spotface*** is a recess at the end of a smaller hole used to provide a bearing surface for the head of a bolt or nut. A ***counterbore*** is similar to a spotface, but the depth of the recess is larger. See **Figure 10-22**.

When dimensioning a multiview drawing of a cylinder, the diameter and length of the cylinder should be dimensioned on the same view (the secondary view), **Figure 10-23A**. In other words, the diameter and length of the cylinder should both be represented as linear distances. If the diameters of several concentric circles must be dimensioned on a drawing, it may be more convenient to show them on the longitudinal view, **Figure 10-23B**.



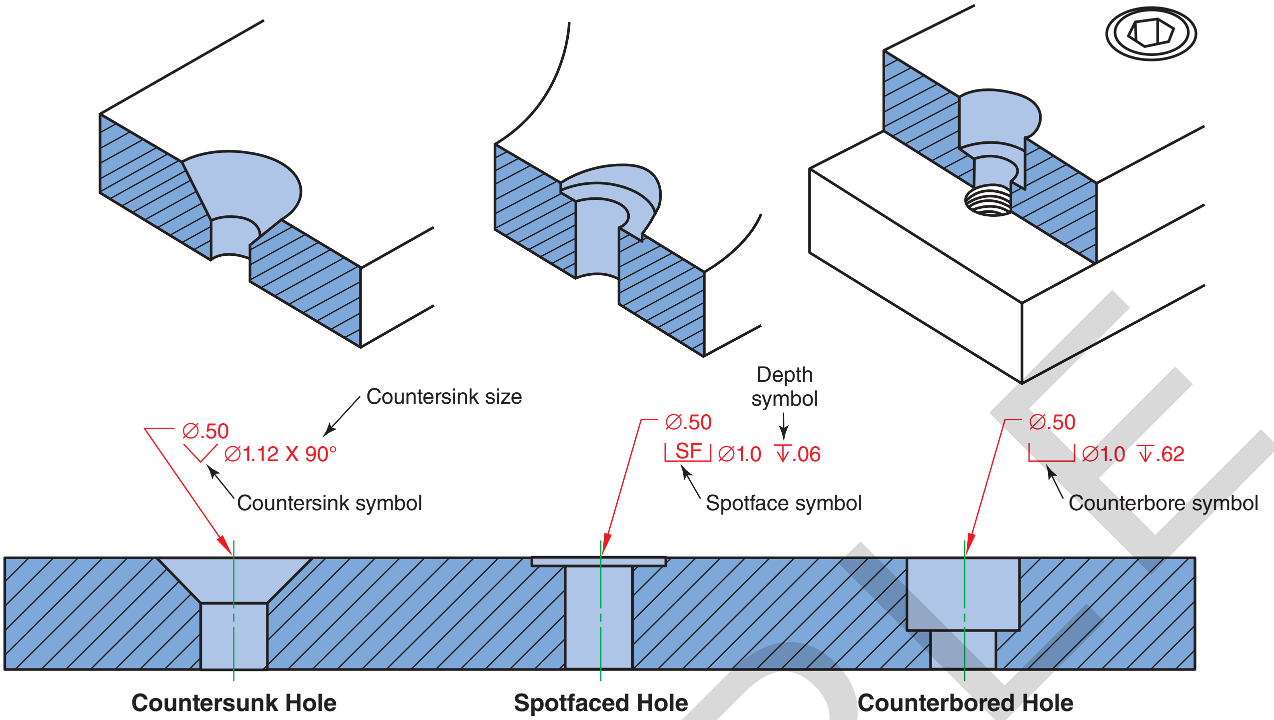
Goodheart-Willcox Publisher

Figure 10-19. Circles and holes are dimensioned by giving the diameter. Different conventions are shown.



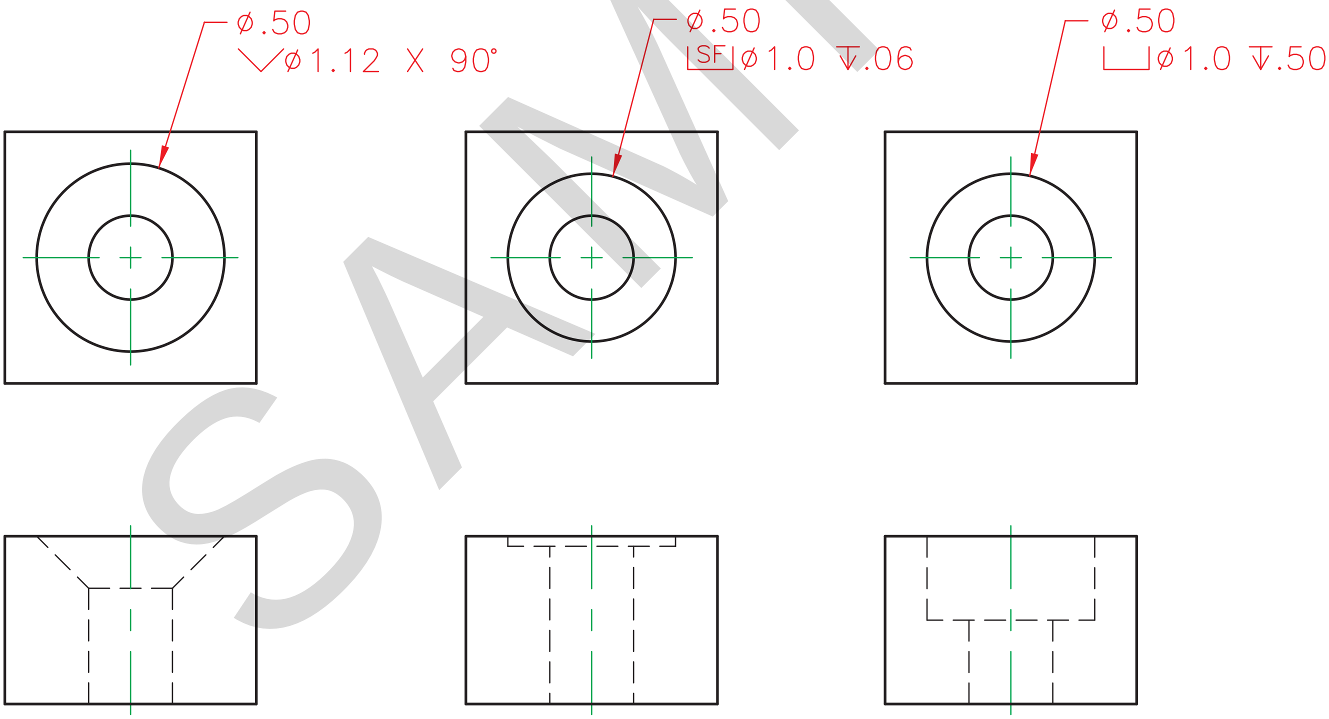
Goodheart-Willcox Publisher

Figure 10-20. Conventions for dimensioning holes. A—The abbreviation THRU is used with the hole size in views where it is not clear that the hole goes through the part. B—The hole depth size is provided in cases where the hole does not go through the entire part.



Goodheart-Willcox Publisher

Figure 10-21. Symbols and notes used to identify countersunk, spotfaced, and counterbored holes.



Goodheart-Willcox Publisher

Figure 10-22. Multiview drawings of hole features for manufacturing processes.

The correct way to use a leader to indicate a diameter is shown in **Figure 10-24**. The leader always points to the primary center of the diameter. When the leader is placed "inside" the circular feature, it intersects the primary center of the diameter. Refer to **Figure 10-19**. The tip of an arrowhead for any leader never *touches* the primary center point, **Figure 10-25**.

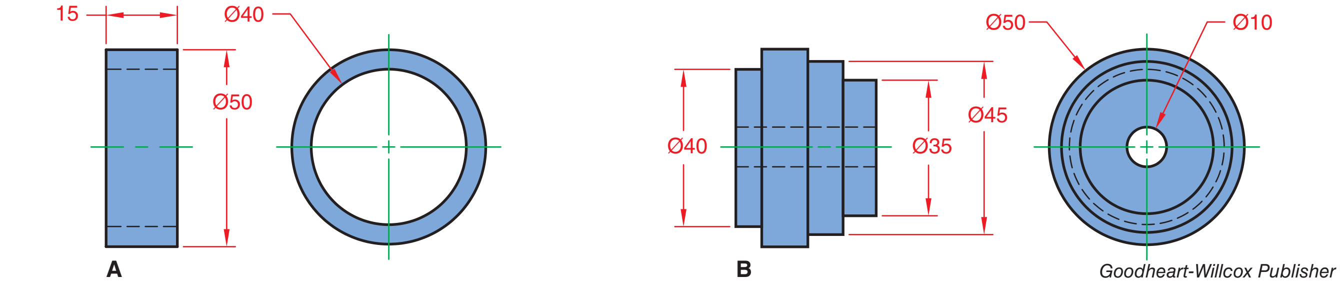


Figure 10-23. Recommended ways to dimension cylindrical objects and concentric circles. A—Dimensions for a cylinder are placed in the side view. B—Placing dimensions for a cylindrical object with several concentric circles.

When dimensioning arcs, the radius of the arc is given, **Figure 10-26**. The capital letter *R* indicates that the dimension is a radius. It is placed *before* the dimension. The methods shown in **Figure 10-26** are common. Each method corresponds to a specific range of radii for the feature. While dimensioning standards vary, these practices are typical (the guidelines shown are not ASME standards).

The dimensioning method shown in **Figure 10-26A** is commonly used for radii of 1″ or greater. The method shown in **Figure 10-26B** is commonly used for radii ranging from 3/8″ to 1″. The methods shown in **Figure 10-26C** are commonly used for radii smaller than 3/8″.

Different conventions may be used for dimensioning radii depending on the situation. For instance, if several concentric arcs are

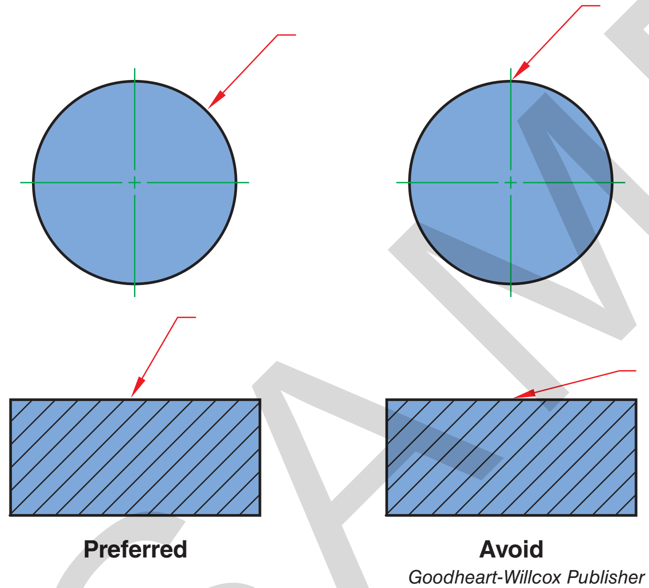


Figure 10-24. Placing leaders to dimension circles. The leader should radiate from the center of the object. Avoid using leaders oriented at lower angular values.

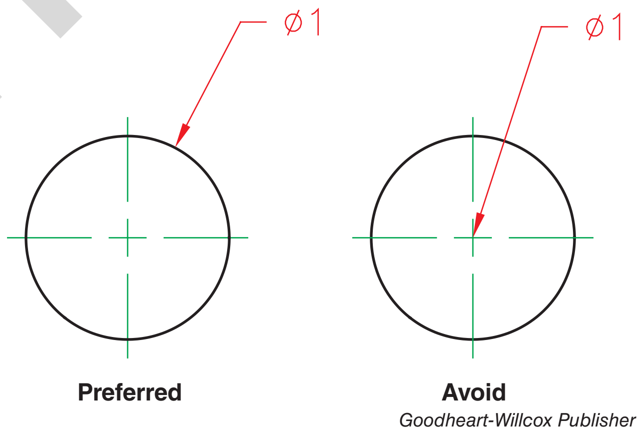


Figure 10-25. The tip of an arrowhead for any leader never touches a primary center point.

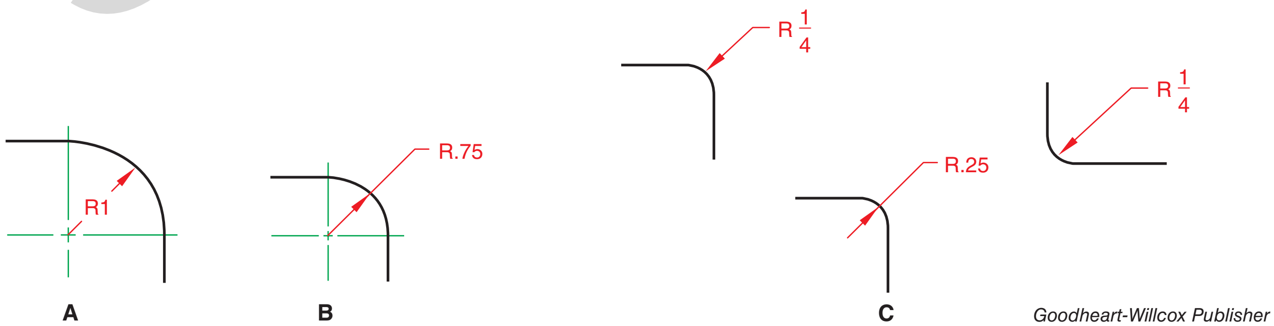


Figure 10-26. Recommended methods for dimensioning arcs.

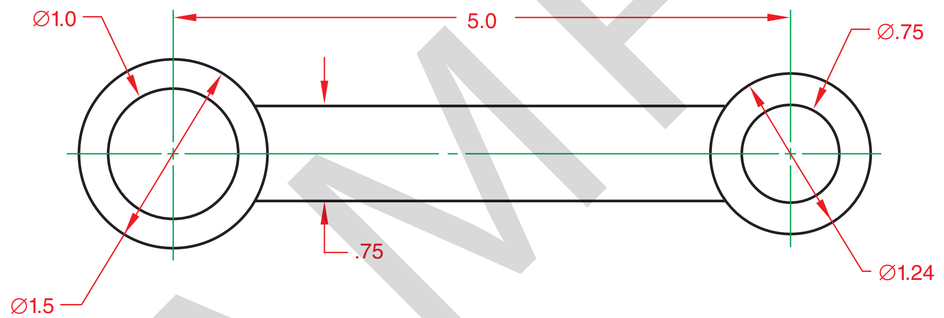
being dimensioned with radii greater than 1″ but increasing in size in relatively small increments, the drafter may choose not to use the common method for large radii (radii 1″ and larger) but rather one of the others to improve clarity. As with diameter dimensioning, the key is to select the style that most clearly does the job in any given situation. There are many acceptable conventions for radius dimensioning. The conventions in **Figure 10-26** are not the only ones that can be used. Consult your instructor for advice on whether one you have created is acceptable.

When placing location and size dimensions, the sizes of round holes and cylindrical parts are dimensioned with respect to primary center points, *never* with respect to the edges of the round feature. See **Figure 10-27**. However, as previously discussed, it is best to locate the center points of round features with respect to the original edges of the original mass of the object.

When it is necessary to dimension a series of holes that lie in a radial (circular) pattern, a special note (grouping statement) is given to designate the number of holes and their size, and the diameter of the circle on which they are located is dimensioned. See **Figure 10-28**. If the holes are equally spaced in the circular pattern, the information can be specified with an angular dimension. Refer to **Figure 10-28A**. Conventions for angular dimensions are discussed next.

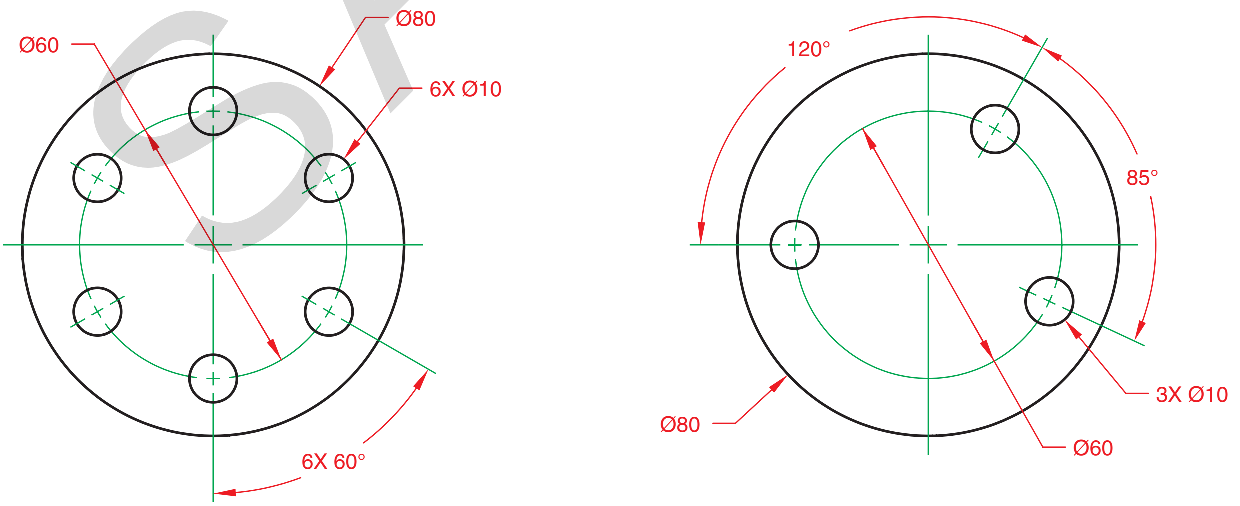
Dimensioning Angles

Angular dimensions are expressed either in degrees and decimal parts of a degree or



Goodheart-Willcox Publisher

Figure 10-27. Round holes and cylindrical parts are dimensioned from the centers when placing location dimensions.



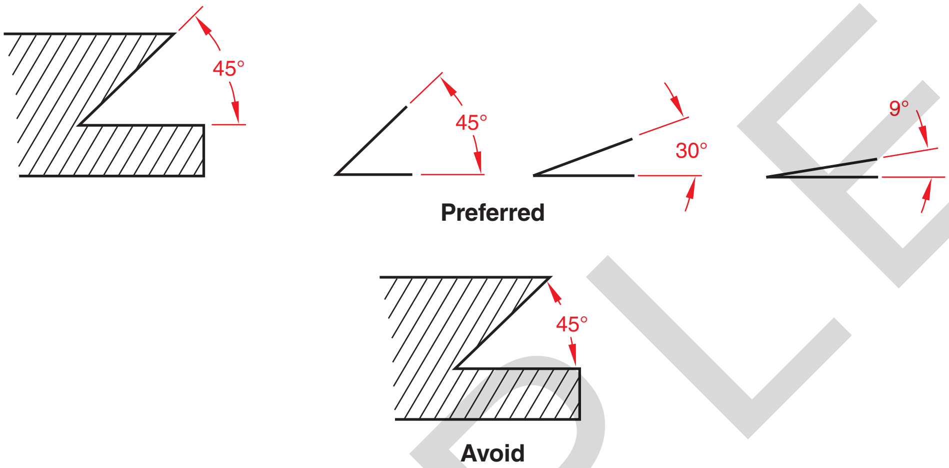
A B

Goodheart-Willcox Publisher

Figure 10-28. A—The correct way to dimension equally spaced holes in a radial pattern. B—Holes that are not equally spaced are dimensioned as shown.

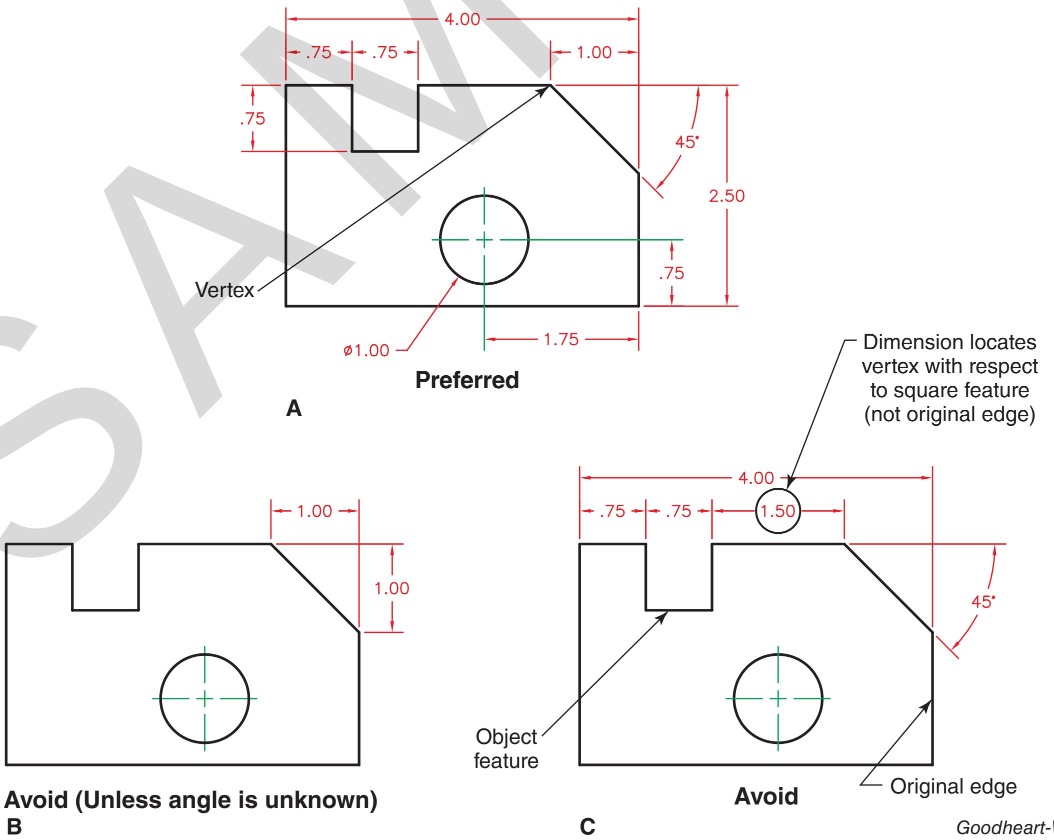
in degrees (°), minutes (′), and seconds (″). Where degrees are indicated alone, the numerical value is followed by the degree symbol. Where only minutes or seconds are specified in an angular dimension, 0° should precede the number of minutes or seconds.

Angles are dimensioned as shown in **Figure 10-29**. The origin (vertex) of the angle must be located and the size given. Whenever possible, locate the vertex with respect to an original edge of the original mass, not with respect to another feature. See **Figure 10-30**.



Goodheart-Willcox Publisher

Figure 10-29. Conventions for dimensioning angles. As with dimensioning linear distances, dimension angles to extension lines, not visible lines.



Goodheart-Willcox Publisher

Figure 10-30. Dimensioning angles. A—Locate the vertex of the angle and give its size. B—If the angle is unknown, providing a linear distance (as shown) is acceptable. C—Locate angles with respect to the original edges of the object, not with respect to other features.

Dimensioning Small Portions of an Object

There are many cases where the size of object features limits the space available to place dimensions. When the space between extension lines is too small for both the numbers of the dimension and the arrowheads, dimensions are placed as shown in **Figure 10-31**. These are just several alternatives. There are other options that are also acceptable. See your instructor for recommendations.

Adapting US Customary Conventions to Metric Dimensioning

Many companies use metric measurement extensively. Some companies have started changing to the SI Metric system to compete successfully in the international market.

It is important that industries working together on projects use the same measurement system. As previously discussed, metric drawings are dimensioned in millimeters. On drawings made with US Customary units, the standard practice is to use decimal inches. One problem encountered in the transition from the US Customary system to the SI Metric system has been at the interface of a part (a surface where parts come together). It is difficult to interface an object designed to metric standards to fit an existing object that was designed to inch standards.

Five methods have been devised for drafters to provide the information necessary to make metric parts that must interface with existing parts dimensioned using conventional measurement. These methods are listed as follows and discussed in the next sections:

- Dual dimensioning.
- Dimensioning with a tabular chart.
- Metric dimensioning with a conversion chart.
- Metric dimensioning (metric units only).
- Undimensioned master drawings.

Dual Dimensioning

Dual dimensioning was the first system devised to dimension engineering drawings with both inch and metric units. See **Figure 10-32**.

Dual dimensions are presented using one of two methods, the position method or the bracket method. See **Figure 10-33**. The inch dimension is placed first on drawings of products to be made in the United States. The metric dimension is placed first on drawings of products to be made where the SI Metric system is the basic form of measurement.

Dual dimensioning is the most complicated dimensioning system. It is seldom used today. However, large numbers of dual-dimensioned drawings are still in use. This method is presented in this text so that you will be aware of its existence.

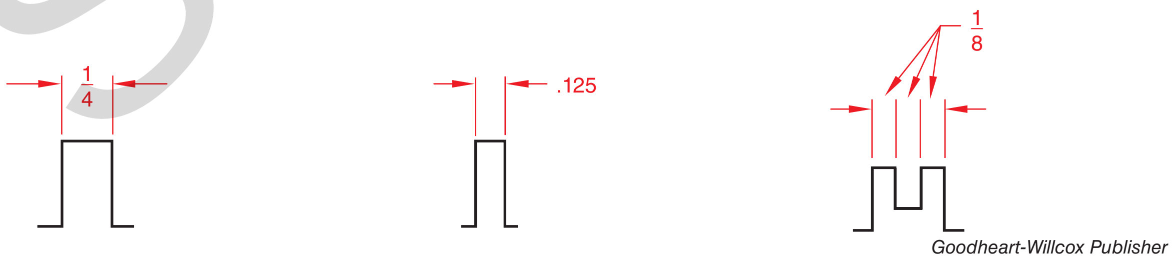
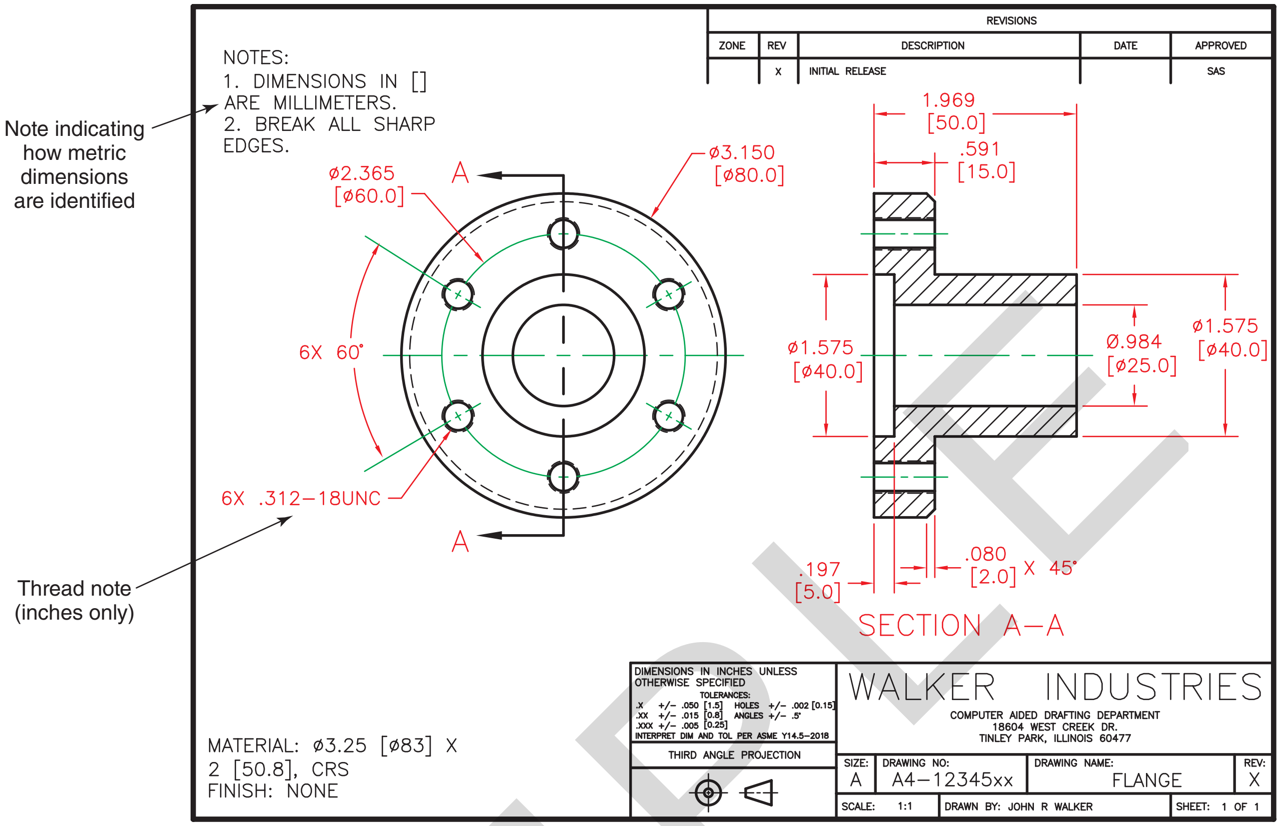
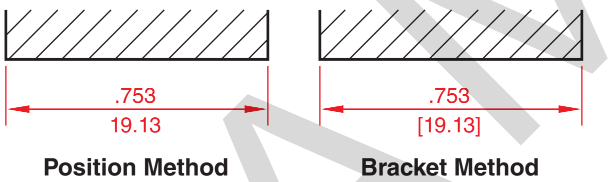


Figure 10-31. Depending on the space available between extension lines, the arrowheads, dimension numeral, or entire dimension may be placed outside the extension lines.



Goodheart-Willcox Publisher

Figure 10-32. A dual-dimensioned drawing. Inch dimensions are given first. The thread size is not given in metric units because there is no metric thread this size.



Goodheart-Willcox Publisher

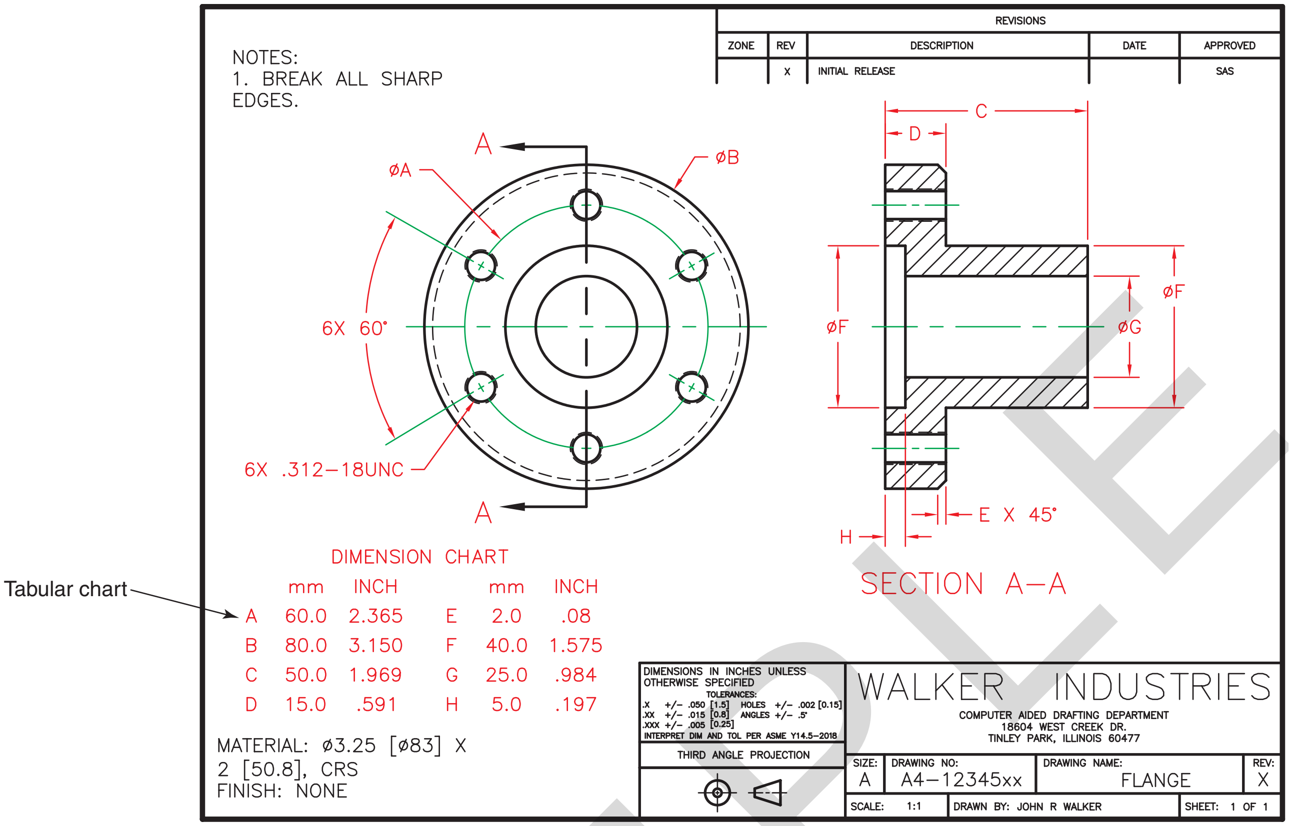
Figure 10-33. Methods of indicating inches and millimeters on a dual-dimensioned drawing.

Dimensioning with a Tabular Chart

When a drawing is shown with both inch and metric dimensions, a tabular chart is sometimes used. See **Figure 10-34**. With this technique, the dimensions are labeled with letters (A, B, C, etc.) rather than sizes. The letters are listed in the chart, and the chart shows the metric and inch equivalents of each dimension.

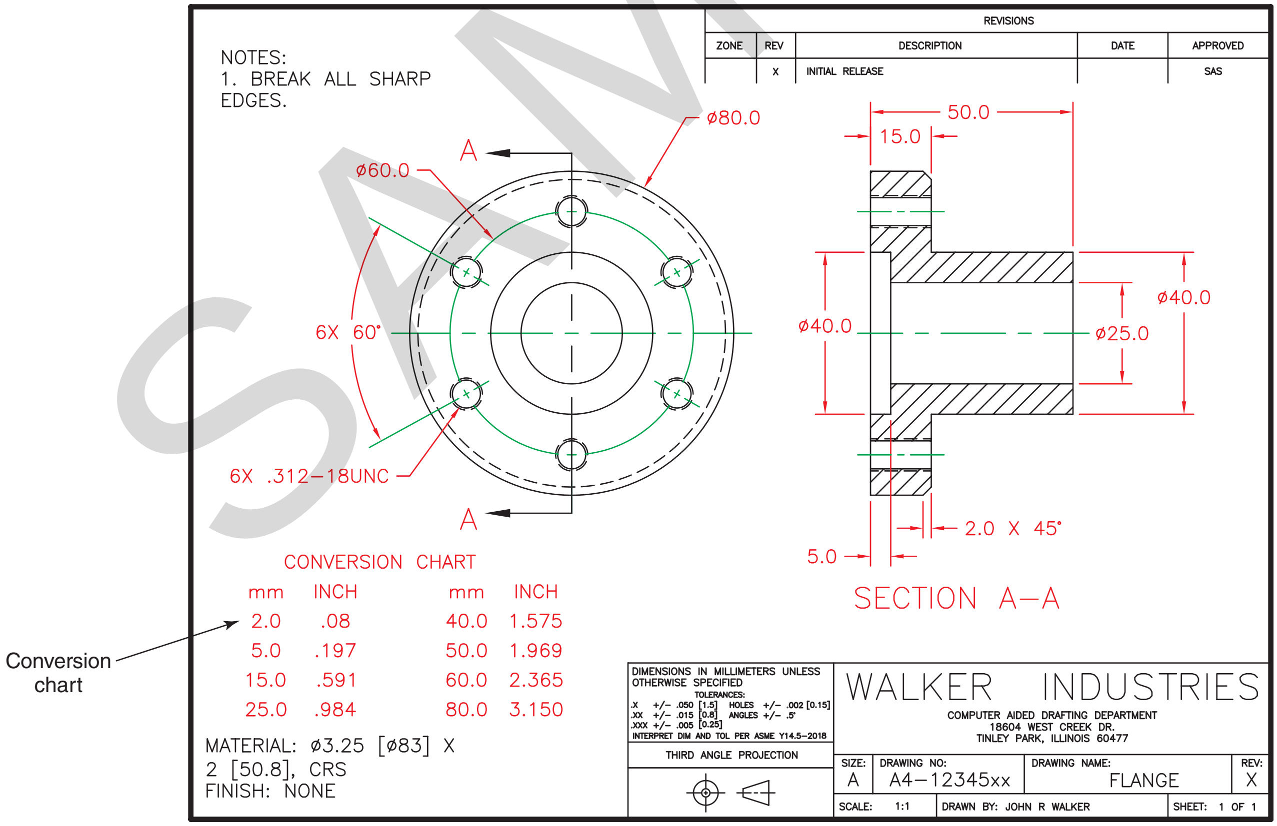
Metric Dimensioning with a Conversion Chart

When a part is designed to metric standards, the engineering drawing is made with metric dimensions. A note on the drawing informs the person reading the drawing that all dimensions are in metric units. Some metric drawings provide a conversion chart, **Figure 10-35**. The chart shows metric dimensions in the left column and the inch equivalents in the right column. This method permits a comparison of values.



Goodheart-Willcox Publisher

Figure 10-34. Dimensioning with a tabular chart. The chart shows the inch and millimeter equivalents for each dimension.



Goodheart-Willcox Publisher

Figure 10-35. A metric drawing with a conversion chart.

Metric Dimensioning (Metric Units Only)

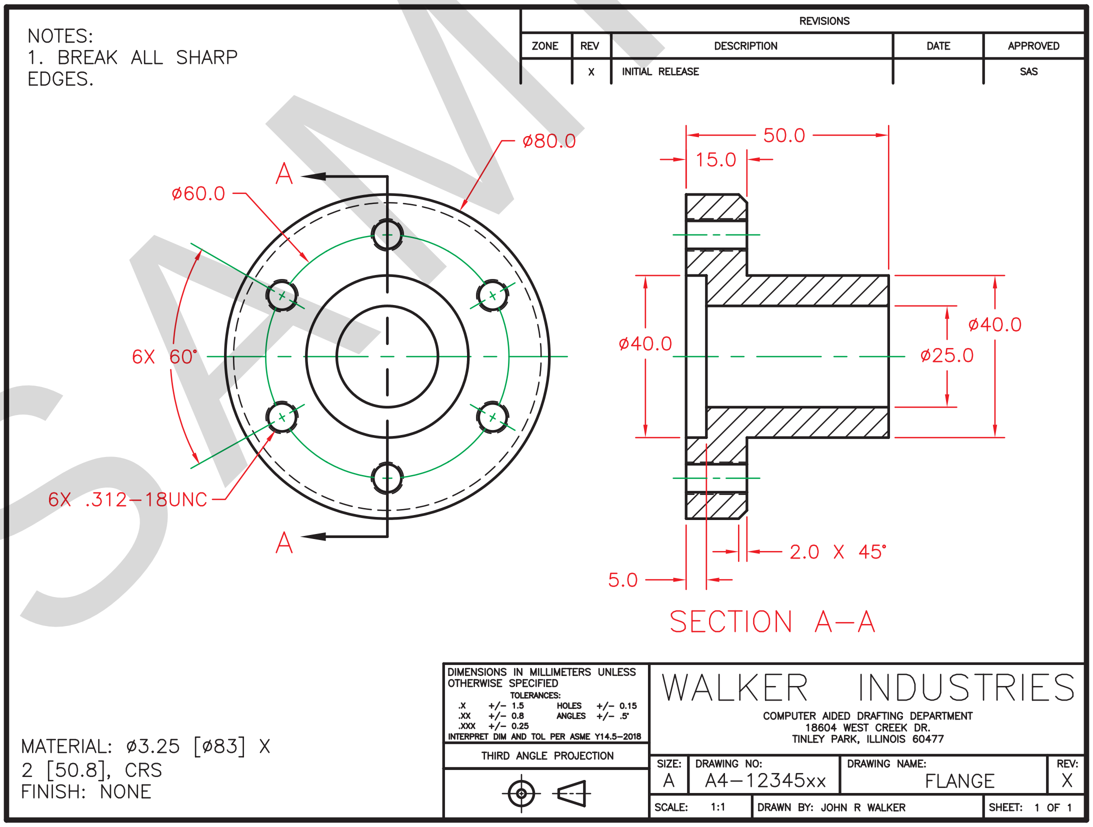
Some metric drawings provide metric dimensions only, **Figure 10-36**. This method is the quickest way to get engineers, designers, drafters, and craft workers to "think metric." The drawing has a note stating "UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN MILLIMETERS" and no conversion values are given.

Undimensioned Master Drawings

When designing parts to meet different measurement standards, drawings may be made without dimensions so that the dimensions can be added later in the different formats. When this is the case, a master drawing is first made without dimensions, **Figure 10-37**. Next, prints are made. Metric dimensions may be added to one print, and inch dimensions to another print. Notes and details can be added in whatever language is needed to produce the part—German, French, English, Japanese, etc.

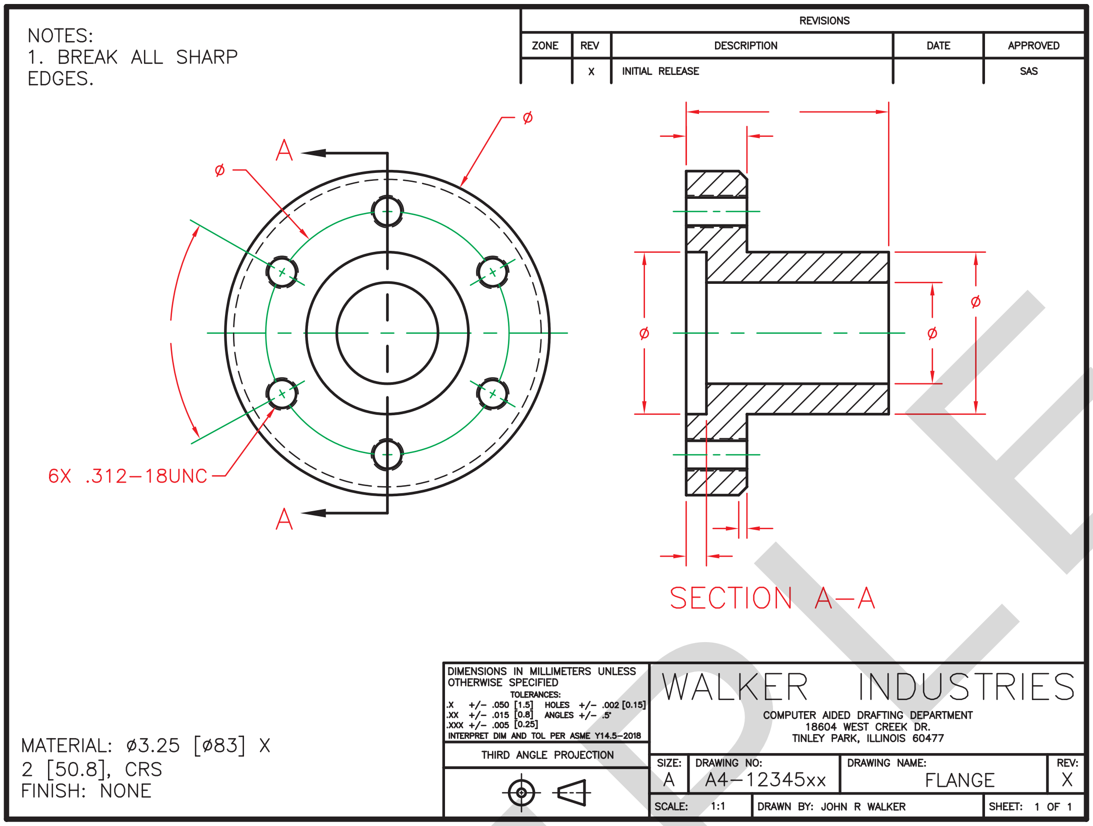
Whatever dimensioning method is used, drafting personnel must have both scales and templates on hand. They must also have a thorough knowledge of the metric system.

In drafting or design work, you will find that you cannot specify a 9/16 bolt by merely listing the metric equivalent of 9/16″ (14.29 mm). There is no metric bolt that corresponds to this diameter. The same problem will occur if a 1.0″ diameter is specified as a 25.4 mm diameter. There is no metric-sized shaft 1.0″ in diameter. The closest size would be 25.0 mm. To get the 25.4 mm diameter shaft, an expensive machining operation would be necessary to turn a larger shaft to the required 25.4 mm diameter.



Goodheart-Willcox Publisher

Figure 10-36. A metric drawing dimensioned in metric units only. Since there is no metric equivalent for the thread indicated on the drawing, the thread size is given in decimal inch units.



Goodheart-Willcox Publisher

Figure 10-37. When parts are designed to conform to different standards of measurement, the drawing may be completed without dimensions. Except for the thread size, no dimensions are shown on this master drawing. After the print is made, dimensions in either inch or metric units are added.

General Rules for Metric Dimensioning

As previously discussed, most metric drawings are made in millimeters. This is standard practice. Millimeter dimensioning should conform to the following rules:

1. When a millimeter dimension is a whole number, the decimal point and following zero are not shown. This holds true unless tolerances are shown.

   125, not 125.0
2. When a dimension is less than one millimeter, a zero is shown to the left of the decimal point:

   0.5, not .5
3. All metric drawings should be clearly identified as such. The symbol for millimeter (mm) should not be added to every dimension. Instead, the drawing should have a note stating "UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN MILLIMETERS."
4. A space (or comma) is not used to separate digits for dimensions into groups:

   20500, not 20 500
5. When the dimension exceeds a whole number by a decimal fraction of one millimeter, a zero does not follow the last digit to the right of the decimal point.

   125.5, not 125.50

Note

Where some millimeter dimensions are shown on an inch-dimensioned drawing, the symbol "mm" follows the millimeter values.

Tolerancing

The foundation of modern manufacturing is the ability to produce numerous parts, all of which meet the exact specifications for assembly or use. In reality, it is impossible to make a part to its exact dimensions. Tolerancing allows manufacturers to control the precision and quality of the parts they make. A ***tolerance*** is an allowable variance from the original dimension. Tolerances permit allowances in size from the specified dimension that occur during manufacturing.

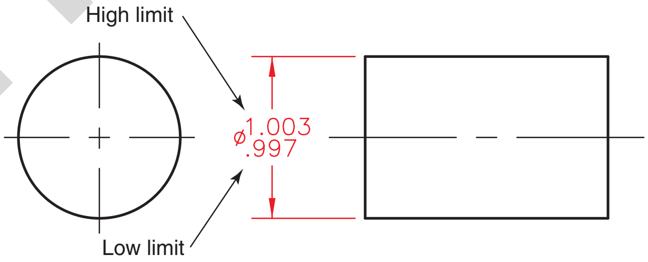
Tolerances are added to the ***basic size***, the theoretically exact size represented by the dimension. Tolerances can be applied to all feature dimensions, including size and location dimensions. The unit of measure for a tolerance is the same as the unit of measure for the related dimension.

There are two major methods for applying tolerances directly to the dimensions: limit tolerancing and plus-and-minus tolerancing.

Limit Tolerancing

Limit tolerancing uses limits to define the allowable sizes of a dimension. ***Limits*** are the absolute maximum and minimum sizes allowed, **Figure 10-38**. The high limit is the largest acceptable dimension allowed and the low limit is the smallest acceptable dimension allowed. The difference between the high limit and the low limit is the tolerance.

When specifying a limit tolerance, the high limit appears above the low limit. If the limits are displayed next to each other, the low limit appears first, followed by the high limit.



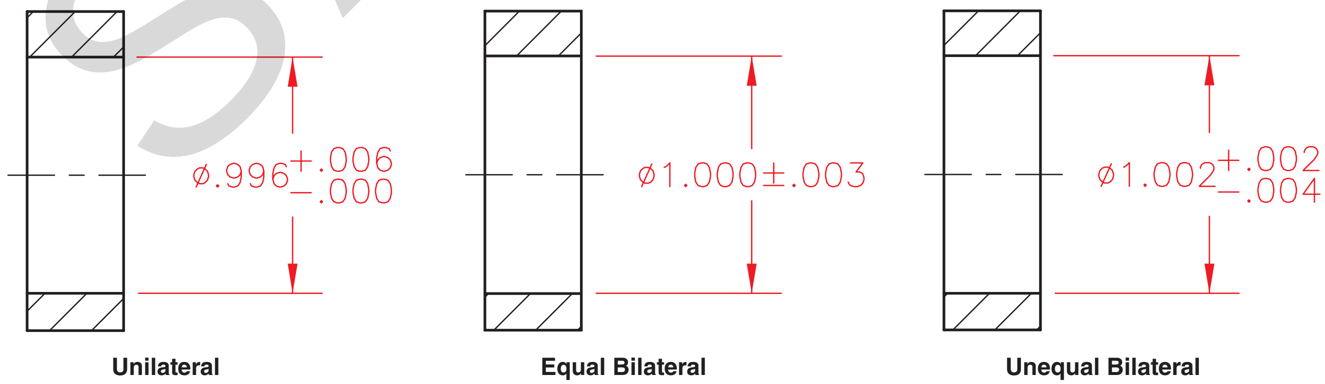
Goodheart-Willcox Publisher

Figure 10-38. Limit tolerancing uses limits to define the maximum and minimum acceptable values for a dimension.

Plus-and-Minus Tolerancing

The ***plus-and-minus tolerancing*** method specifies the permissible variation of a dimension in the positive and negative directions. A plus symbol (+) is used to indicate variation in the positive direction. A minus symbol (–) indicates variation in the negative direction. The plus-and-minus tolerances are placed next to the basic size, with the plus tolerance above the minus tolerance.

A ***unilateral tolerance*** allows for variation in only one direction. A ***bilateral tolerance*** allows for variation in both directions. A bilateral tolerance can have equal or unequal values. If the values are equal, a plus-and-minus symbol (±) is used. See **Figure 10-39**. The total tolerance for an equal bilateral tolerance is



Goodheart-Willcox Publisher

Figure 10-39. Plus-and-minus tolerancing defines the basic size of a dimension and the allowable variation in the positive and negative direction. The unilateral, equal bilateral, and unequal bilateral tolerances shown all permit the same variance: .006″.

double the expressed value. For example, the total tolerance expressed by ±.003″ is *not* .003″. The tolerance is actually .003″ + .003″, or .006″.

General Tolerances

Not every dimension needs to be toleranced directly. ***General tolerances*** allow the drafter to apply a tolerance for all dimensions that do not have specified tolerances. General tolerances can be included in the title block or in a note, table, or separate document. Unless otherwise specified, general tolerances are applied to all applicable dimensions.

General tolerances can be provided for fractional, decimal, angular, and metric tolerances, **Figure 10-40**. Multiple decimal tolerances can be specified depending on the precision requirements of the drawing. One- and two-place decimal tolerances are used for standard precision requirements. Three- and four-place decimal tolerances are used for higher precision applications. Angular tolerances can be specified in degrees or decimal degrees. Metric tolerances only apply to metric dimensions.

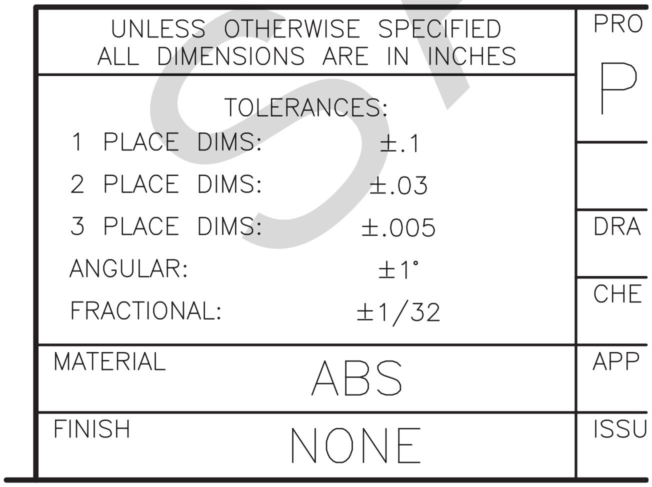
Geometric Dimensioning and Tolerancing (GD&T)

The design of parts for many complex applications requires a highly precise system of specifying dimensions and tolerances on drawings. Machining processes have improved to the point where highly accurate definitions of tolerances for *form* (shape), *orientation* (angle), and *position* (location) are needed to design products.

Geometric dimensioning and tolerancing (GD&T) is a standard system devised to control interpretation of the form, profile, orientation, position, and runout of features on drawings. This type of tolerancing provides the necessary precision for the most economical manufacture of parts. A common drawing language must be applied because different dimensioning standards exist and because parts for specialized products are typically manufactured in a number of locations, often in different countries. The GD&T system provides an international *language* that standardizes the dimensioning and tolerancing process. It is used in industries where parts are produced in mass quantities and interchangeability of parts is essential in production work.

The GD&T system uses standardized symbols to specify and explain geometric tolerances, **Figure 10-41**. ***Geometric characteristic symbols*** are used to identify specific geometric characteristics and feature controls. There are 12 geometric characteristic symbols in the GD&T system. These symbols relate to such variables as the form of an object, the profile of an object, the orientation of features, the position of features, and the runout of surfaces relative to an axis. Each symbol communicates a specific type of control in a geometric tolerance specification.

Geometric tolerances are often used to specify the location or relationship of features relative to a datum. A ***datum*** is a theoretically exact point, axis, or plane from which features of a part are located. With respect to geometric tolerancing, a datum is a referenced entity from which geometric controls are established. Datums are theoretical entities established by datum features. A ***datum feature*** is an actual feature on a part, such as a part surface. Datum features are identified by datum feature symbols. The datum feature symbol consists of a reference letter enclosed in a box attached to a



Goodheart-Willcox Publisher

Figure 10-40. General tolerances can be found in the tolerance block section of the title block.

Type of Tolerance	Geometric Characteristic	Symbol
Form	Straightness	⏤
	Flatness	⏥
	Circularity	○
	Cylindricity	⌭
Profile	Profile of a line	⌒
	Profile of a surface	⌓
Orientation	Angularity	∠
	Perpendicularity	⊥
	Parallelism	//
Position	Position	⌖
Runout	Circular runout	↗ or ↗
	Total runout	⌰ or ⌰

A

Geometric Tolerance Specification	Symbol
At maximum material condition (*or* maximum material boundary)	Ⓜ
At least material condition (*or* least material boundary)	Ⓛ
Diameter	Ø
Radius	R
Basic dimension	[30]
Reference dimension	(30)
Counterbore	⌴
Countersink	⌵
Depth/deep	↧
Square (shape)	□
Dimension not to scale	25
Number of times/places	6X
Datum feature	[A]—◀
Between	◀—▶

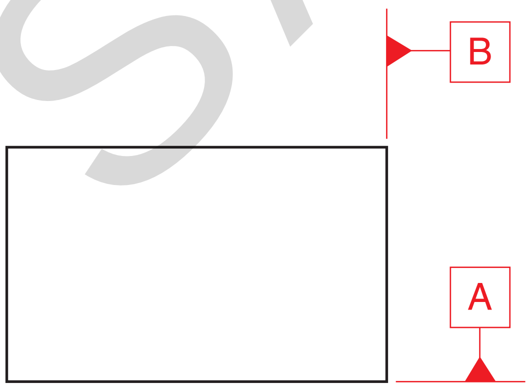
B

Goodheart-Willcox Publisher

Figure 10-41. Standard symbols used in geometric dimensioning and tolerancing. A—Geometric characteristic symbols. B—Additional symbols used in the GD&T system.

triangle, **Figure 10-42**. Datum feature symbols are placed on a feature surface, on an extension line, on a leader shoulder, or on a dimension line.

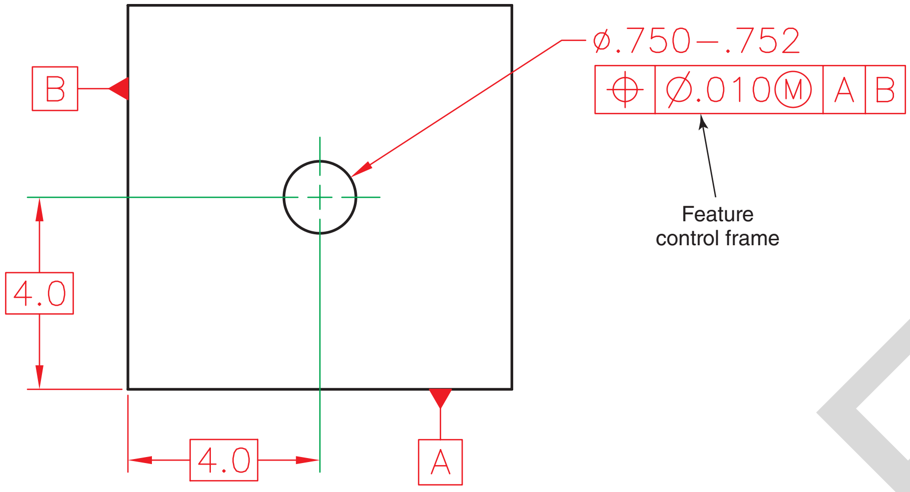
Geometric characteristic symbols, datum feature references, and other geometric tolerance requirements are specified in a feature control frame. A ***feature control frame*** is a rectangular compartment that contains a divided series of symbols identifying geometric tolerance. See **Figure 10-43**. The geometric characteristic symbol is given first and is followed by the allowable tolerance and one or more datum feature reference letters. Feature control frames may be shown along with the dimension or attached to a dimension line or extension line.



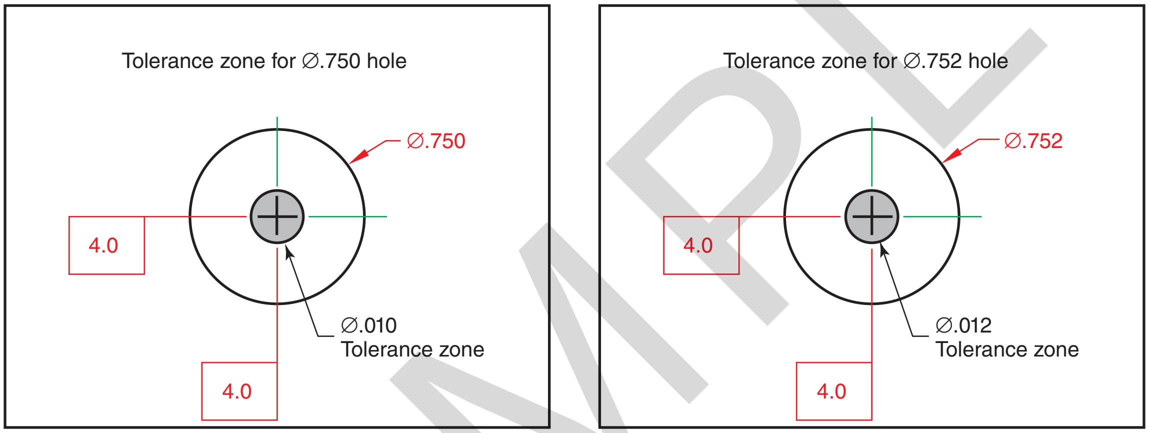
Goodheart-Willcox Publisher

Figure 10-42. A datum feature is identified by a datum feature symbol. The symbol consists of a reference letter inside a box attached to a solid triangle.

Figure 10-43 shows an example of positional tolerancing with the maximum material condition (MMC) modifier applied and the corresponding interpretation. **Figure 10-44** shows an example of positional tolerancing with the least material condition (LMC) modifier applied and the corresponding interpretation. In both figures, the table at the bottom indicates the tolerance at each produced hole size and the amount of bonus tolerance resulting from the application of the material condition modifier. The values in the table are diameter values.



Drawing

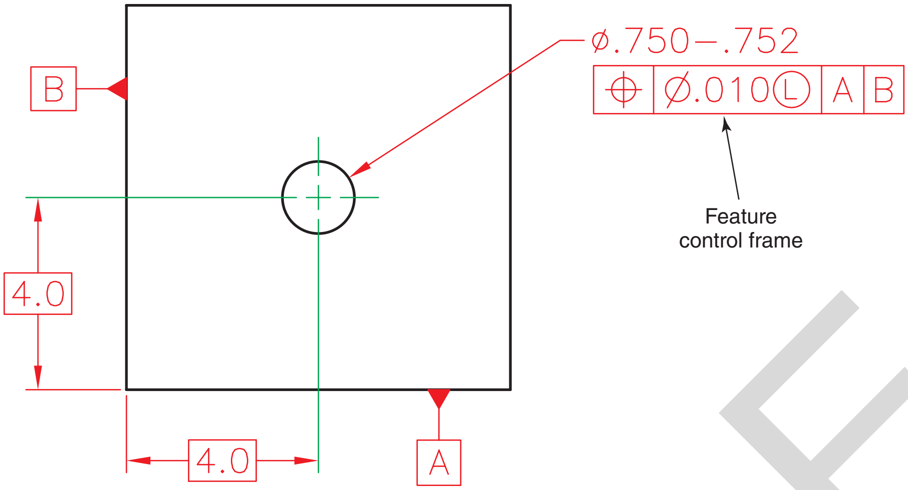


Produced Hole Size	Specified Positional Tolerance	Bonus Tolerance	Total Positional Tolerance
MMC .750	.010	.000	.010
.751	.010	.001	.011
LMC .752	.010	.002	.012

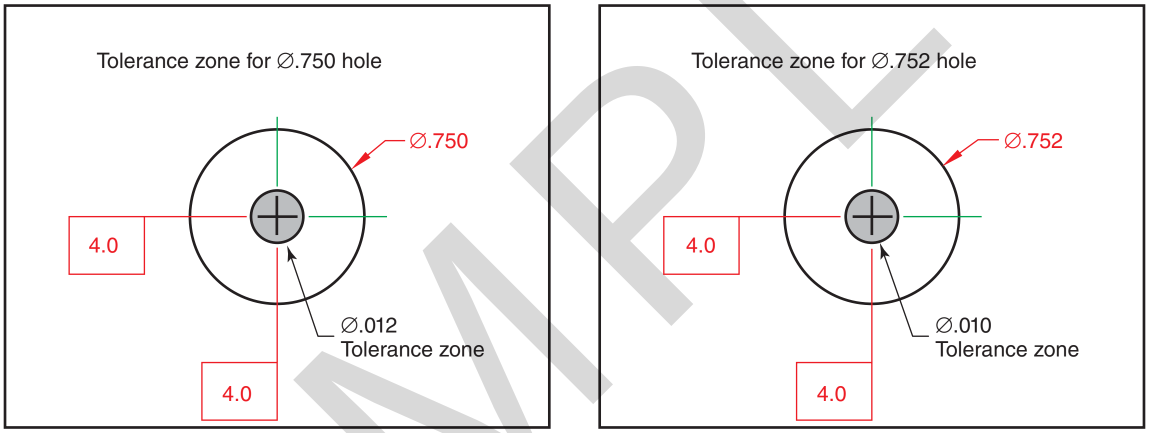
Interpretation

Goodheart-Willcox Publisher

Figure 10-43. The feature control frame on this drawing provides precise information to manufacture the component part. It specifies the tolerance for the position of the hole at maximum material condition. The interpretation of the information is shown below the drawing. The values in the table are diameter values. By learning the GD&T system, you will be able to interpret the exact location and form of features.



Drawing



Produced Hole Size	Specified Positional Tolerance	Bonus Tolerance	Total Positional Tolerance
MMC .750	.010	.002	.012
.751	.010	.001	.011
LMC .752	.010	.000	.010

Interpretation

Goodheart-Willcox Publisher

Figure 10-44. The feature control frame on this drawing specifies the tolerance for the position of the hole at least material condition. The interpretation shows the tolerance values when the hole is produced at different sizes.

CAD Application

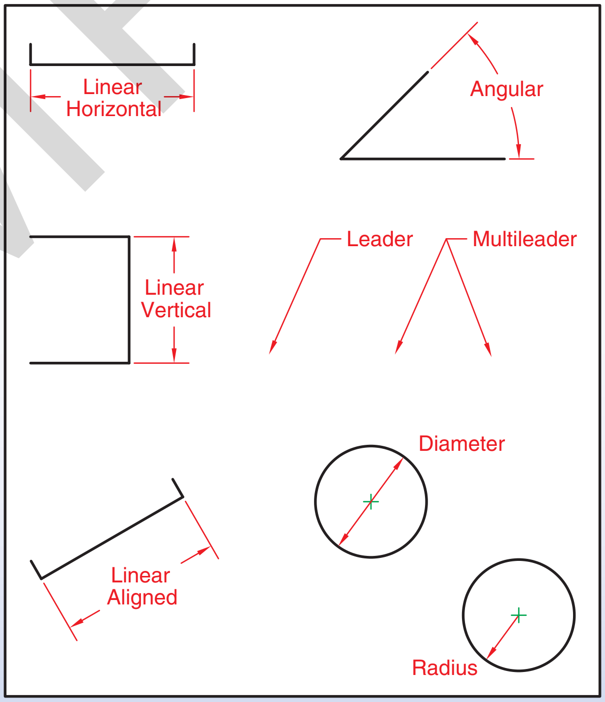
Dimensioning in CAD

Dimensioning a drawing manually can be a very time-consuming process. In CAD drafting, dimensions can be created quickly and accurately with commands and drawing aids such as object snaps. A number of common dimensioning commands are typically available.

As discussed in Chapter 8, lettering is known as *text* in CAD. Text is created with text styles. In similar fashion, dimensions in CAD are drawn with dimension styles. Typically, each dimension style has a text style assigned to it. There are also settings for the unit format. A dimension style can be created for decimal inch, fractional, metric, or dual dimensioning. When creating a dimension style, it is important to set the correct scale for dimensions in relation to the drawing scale. This ensures that the dimensions will be drawn to the proper size in relation to object features.

Dimension styles also provide controls for the extension line and dimension line format, dimension spacing, and text placement. Dimensions can be placed in a number of ways with CAD, depending on the size of the feature and proximity to other dimensions. For example, the dimension text and dimension line can be placed inside or outside the extension lines. Also, as with manual drafting, dimensions may be placed using unidirectional, aligned, or chain dimensioning.

Dimensions are usually placed on a separate layer from other layers in the drawing. After creating a dimension style, it can be set current and used with various dimensioning commands. There are normally different commands for linear, aligned, angular, diameter, radius, and leader dimensioning. See **Figure 10-45**. In linear dimensioning, object points (such as corners or center points) are selected for the extension line origins. The placement of the dimension line and text is then specified. In angular dimensioning, the lines making up the angle are selected for the extension line origins. When placing a diameter or radius dimension, the circle or arc is first selected, followed by locations for the leader line and text. The diameter or radius symbol is generated automatically, and the dimension value is calculated automatically by the program. Leaders are drawn in a similar manner. Points are selected for the placement of the arrowhead, line, and text. When creating a note, the text is entered manually. The text can include a variety of symbols, such as the counterbore or countersink symbol. A special type of leader called a *multileader* can be used when it is necessary to create a leader object with more than one leader segment. Refer to **Figure 10-45**. Commands associated with multileaders provide additional functions for changing the appearance of the leader line and aligning or grouping multiple leaders attached to notes.



Goodheart-Willcox Publisher

Figure 10-45. Different types of dimensioning are associated with commands in CAD programs.

CAD Application (continued)

Some programs provide a single dimensioning command for quickly applying the correct type of dimension based on the object selected. The command recognizes the object and generates the appropriate dimension. For instance, selecting a line creates a linear dimension with extension lines originating from the endpoints.

Special editing functions are commonly available for modifying dimensions. Editing commands may be used to reposition dimension text, specify different text, or change the dimension style.

Some CAD programs have special commands for geometric dimensioning and tolerancing applications. When this is the case, commands are normally available for drawing datum feature symbols, feature control frames, and geometric characteristic symbols. When placing tolerance dimensions, there are options for setting the tolerance method, limits, and precision.

The same rules used in manual dimensioning apply to dimensioning in CAD. To save drawing time, create different dimension styles based on the drafting discipline used. Dimension styles should observe ASME, school, or company standards.

Chapter 10 Review

Summary

- Dimensions and notes communicate the information necessary to make products.
- The two most common systems of measurement are the US Customary system and the SI Metric system. Drawings made with the US Customary system are normally dimensioned in decimal inches. Metric drawings are normally dimensioned in millimeters.
- Unidirectional dimensions are placed horizontally, whereas aligned dimensions are parallel to the dimension line.
- Lines used in dimensioning include dimension lines, extension lines, and leader lines.
- Overall dimensions define the overall size of an object, location dimensions indicate where features lie, and size dimensions define the size of particular features.
- Dimensions should conform to the general rules of dimensioning.
- Arcs 180° or smaller are dimensioned by giving the radius. Circles and round holes are dimensioned by giving the diameter.
- Tolerances define the allowable variation from the original dimension.
- Limit tolerancing and plus-and-minus tolerancing are the two major methods of applying tolerances directly to dimensions.
- Geometric dimensioning and tolerancing (GD&T) is used to control interpretation of the form, profile, orientation, position, and runout of features on drawings.
- Dimensions can be created quickly and accurately in CAD drafting using dimension styles and the appropriate dimensioning commands.

Test Your Knowledge

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

1. Explain why it is important to provide the most pertinent information in the clearest way possible when dimensioning a drawing.
2. Mechanical drawings dimensioned in inches are most commonly dimensioned in _____ inches.
3. Metric drawings are most commonly dimensioned in _____.
4. In unidirectional dimensioning, dimensions are placed _____ so that they are read from the bottom of the drawing.
5. The dimension line is a _____.
   A. heavy black line
   B. thin black line
   C. heavy dashed line
   D. thin dashed line
6. An extension line begins 1/16″ away from the point, edge, intersection, or feature of the object and extends about _____ past the last dimension line it is used for.
7. A(n) _____ is an angular dimension line used to specify sizes of circles and arcs or to point out special characteristics of an object.
8. For most applications, _____ long arrowheads are satisfactory for dimensioning.
   A. 1/8″
   B. 1/4″
   C. 1/2″
   D. 3/4″
9. The three basic types of dimensions are overall, _____, and size dimensions.
10. Dimensions are placed on the views that show the true _____ of the feature being dimensioned.

11. Which of the following is true of dimensions?
    A. Dimensions should be placed within the outlines of the views.
    B. Overall dimensions should be placed nearest the view, while the smaller dimensions should be placed farther from the view.
    C. Dimensions should be placed so that no scaling of the drawing is necessary.
    D. Dimensions should be duplicated across views.
12. Define *chain dimensioning*.
13. Give the symbols used to indicate the diameter of a circle and the radius of an arc.
14. On your answer sheet, sketch the following hole symbols.
    A. Hole depth
    B. Countersunk hole
    C. Counterbored or spotfaced hole
15. When associated with the diameter of a circle, a leader always points to what?
16. When dimensioning the multiview drawing of a cylinder, the diameter and length should be dimensioned in which view?
17. Explain the different types of units used to express angular dimensions.
18. Define *dual dimensioning*.
19. What is the tolerance of the dimension 1.45±.05"?
20. Rewrite the limit tolerance 4.23–4.27 as an equal bilateral tolerance.
21. A(n) _____ is a theoretically exact point, axis, or plane from which features of a part are located.
22. Briefly describe some of the factors to consider when creating a dimension style for use in a CAD drawing.

Applying Your Knowledge

1. Contact several industrial drafting departments. If possible, obtain copies of the standards they use for dimensioning. Compare how similar or dissimilar they are. Do they vary by industry?
2. Obtain copies of several industrial prints. Study how they are dimensioned and how notes are used. Are there any provisions for foreign components to be used with the items on the prints? Will the parts be exported?

STEM Activities

1. **Engineering.** Go to the school library. Find a shelf of books that are written on a topic that interests you. Measure the height, width, and thickness of each book and calculate the volume in cubic inches. Design a three-shelf bookcase large enough to hold just these books with no room to spare. Create a multiview drawing of the bookcase. Fully dimension the drawing using the rules of dimensioning discussed in this chapter. Take care to create the drawing and dimensions with the proper line weights.
2. **Engineering.** Design a pencil holder with a circular base. Create a multiview drawing of your design. Fully dimension the drawing using the rules of dimensioning discussed in this chapter.

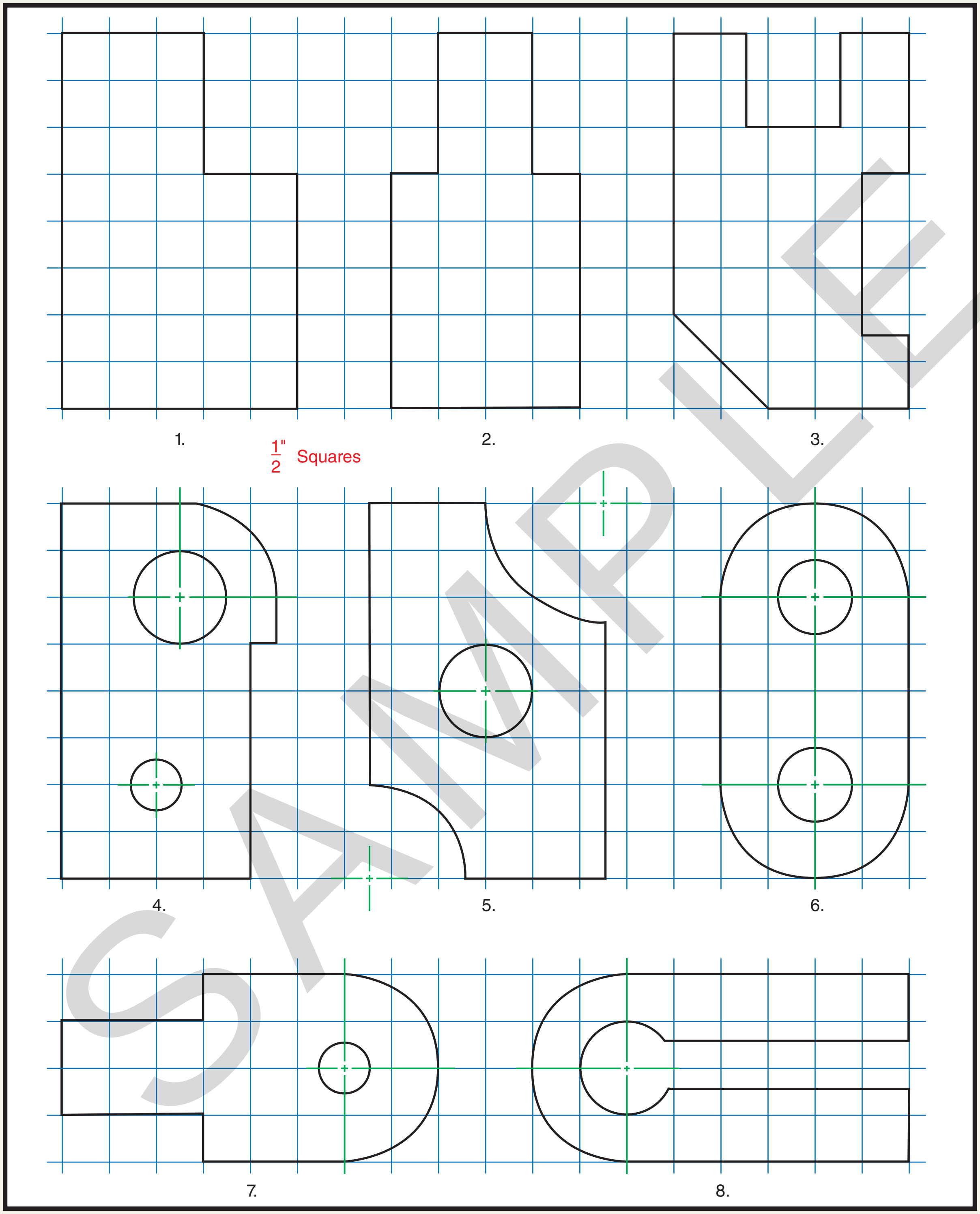
Communicating about Drafting

Working with a partner, create a multiview drawing of a smartphone. Use the information in this chapter and do any research needed to dimension your drawing. Share your drawing with the class. Discuss the importance of accurate dimensions.

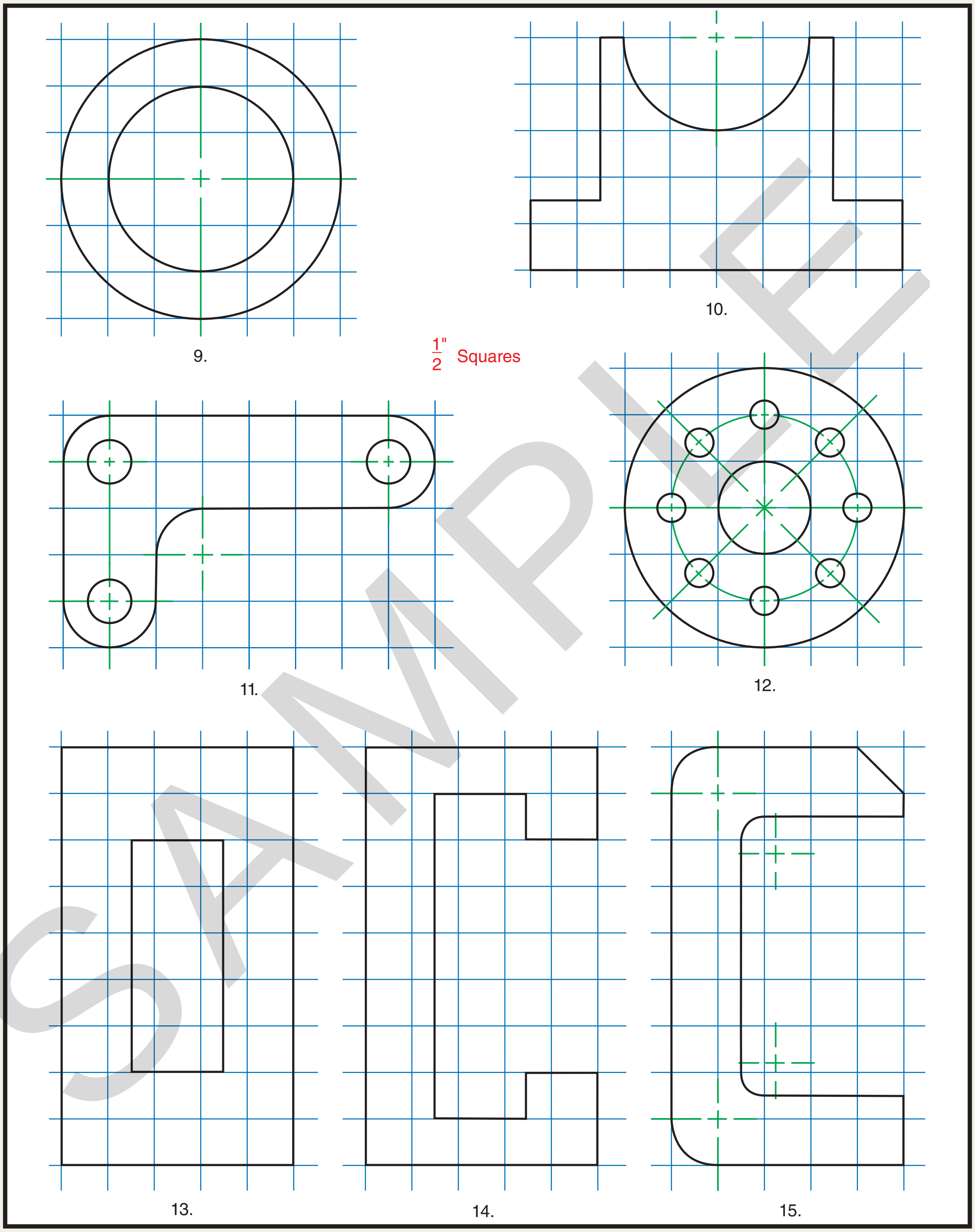
Drawing Problems

Draw the problems shown on the following pages. Follow the directions in each problem.

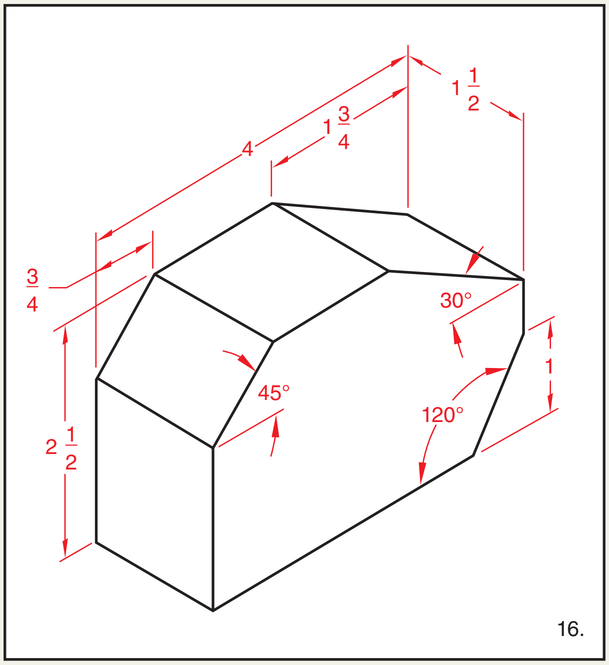
Drawing Problems



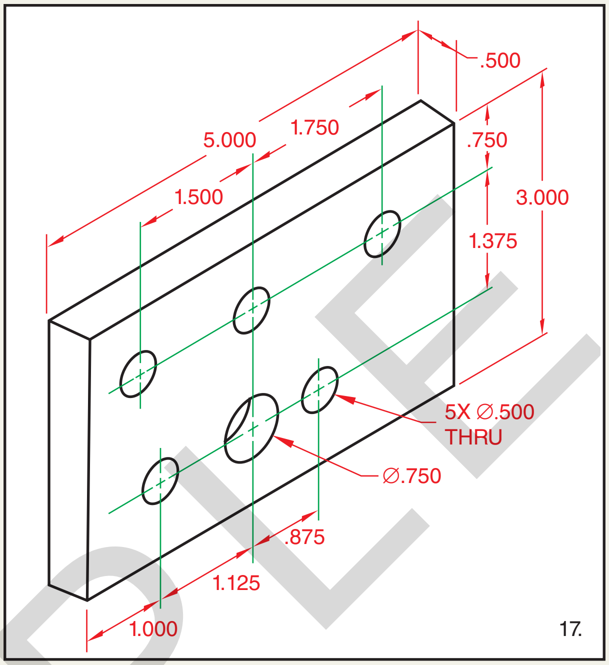
Problem Sheet 10-1. ★ Redraw and dimension. Draw only the view shown. Draw two problems per sheet. Grid squares are 1/2″.



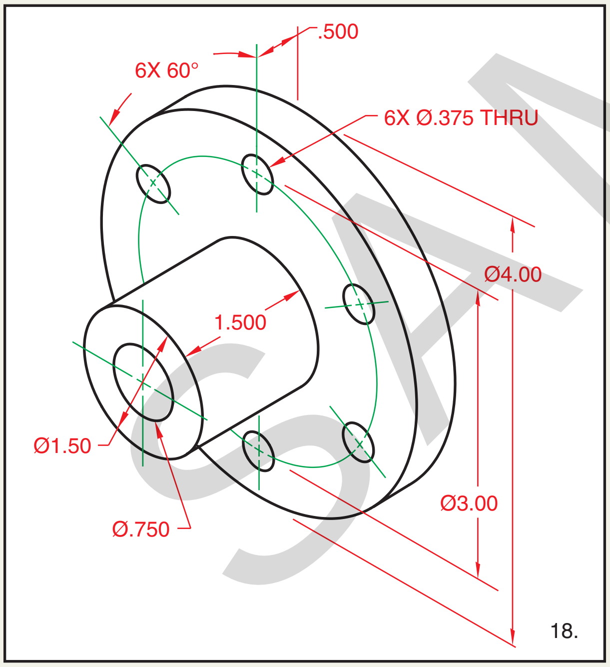
Problem Sheet 10-2. ★ Redraw and dimension. Draw only the view shown. Grid squares are 1/2″.



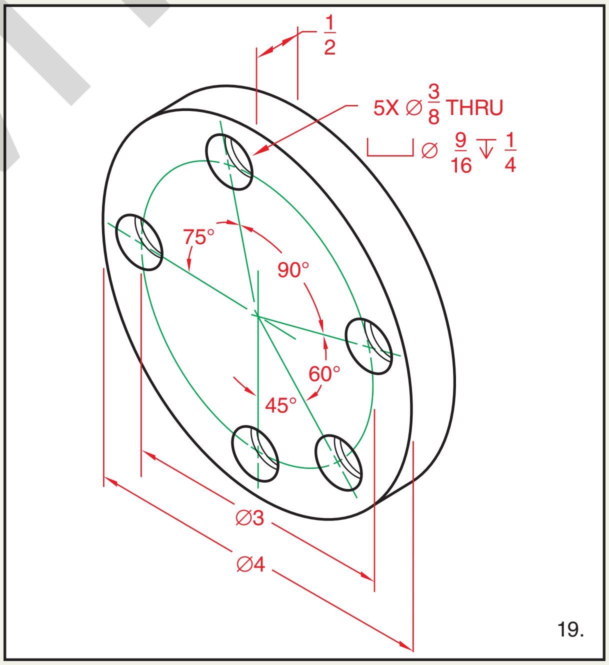
Problem 10-16. ★ Shim. Draw the necessary views and correctly dimension them.



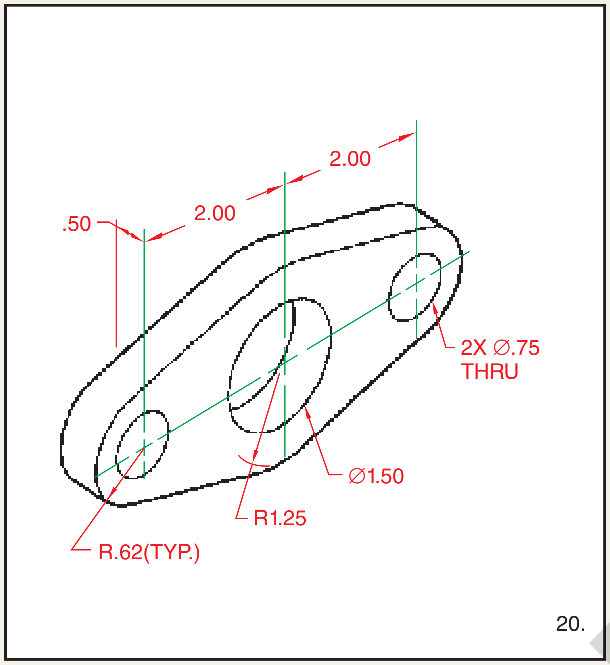
Problem 10-17. ★★ Alignment Plate. Draw the necessary views and correctly dimension them.



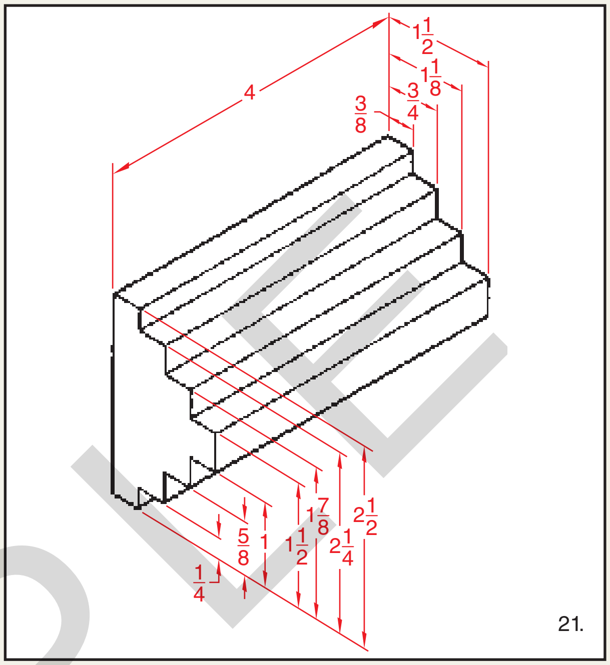
Problem 10-18. ★★ Face Plate. Draw a two-view drawing and dimension correctly.



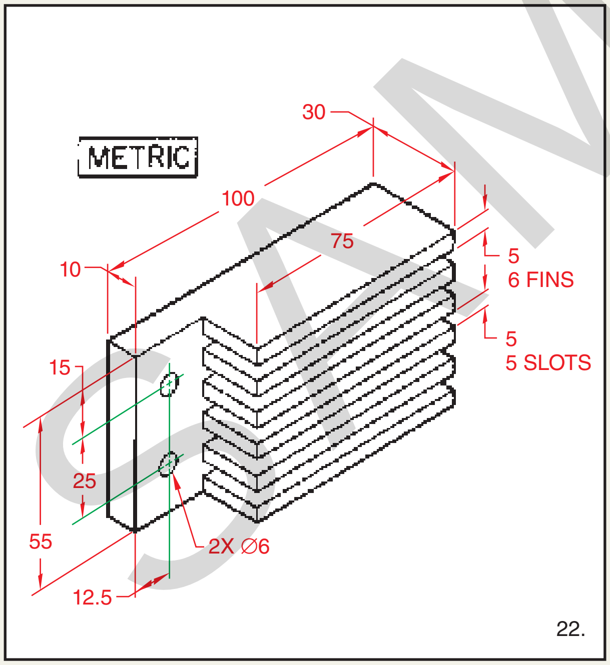
Problem 10-19. ★★ Cover Plate. Draw the necessary views and correctly dimension them.



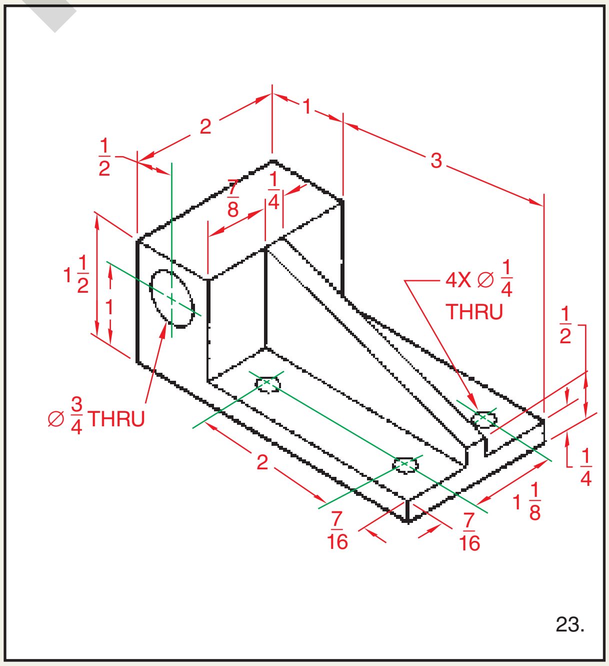
Problem 10-20. ★★ Gasket. Draw a two-view drawing and dimension correctly.



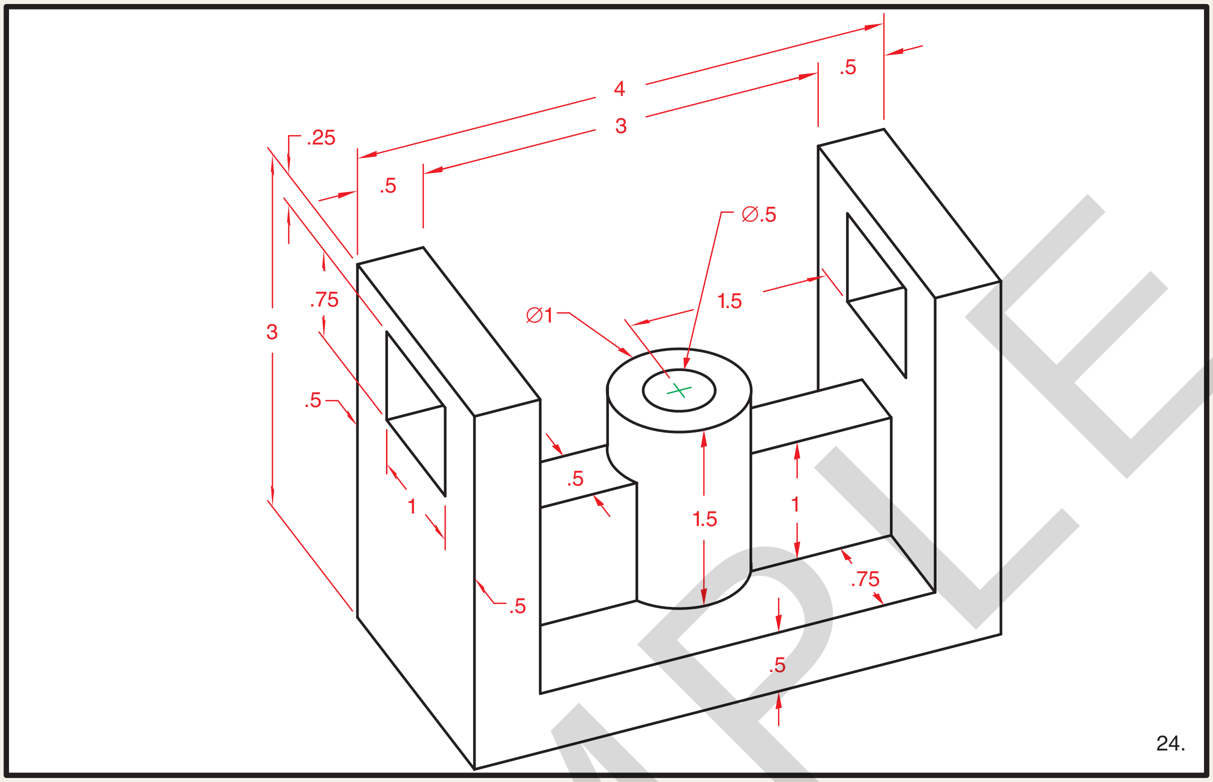
Problem 10-21. ★★ Step Block. Draw the necessary views and correctly dimension them.



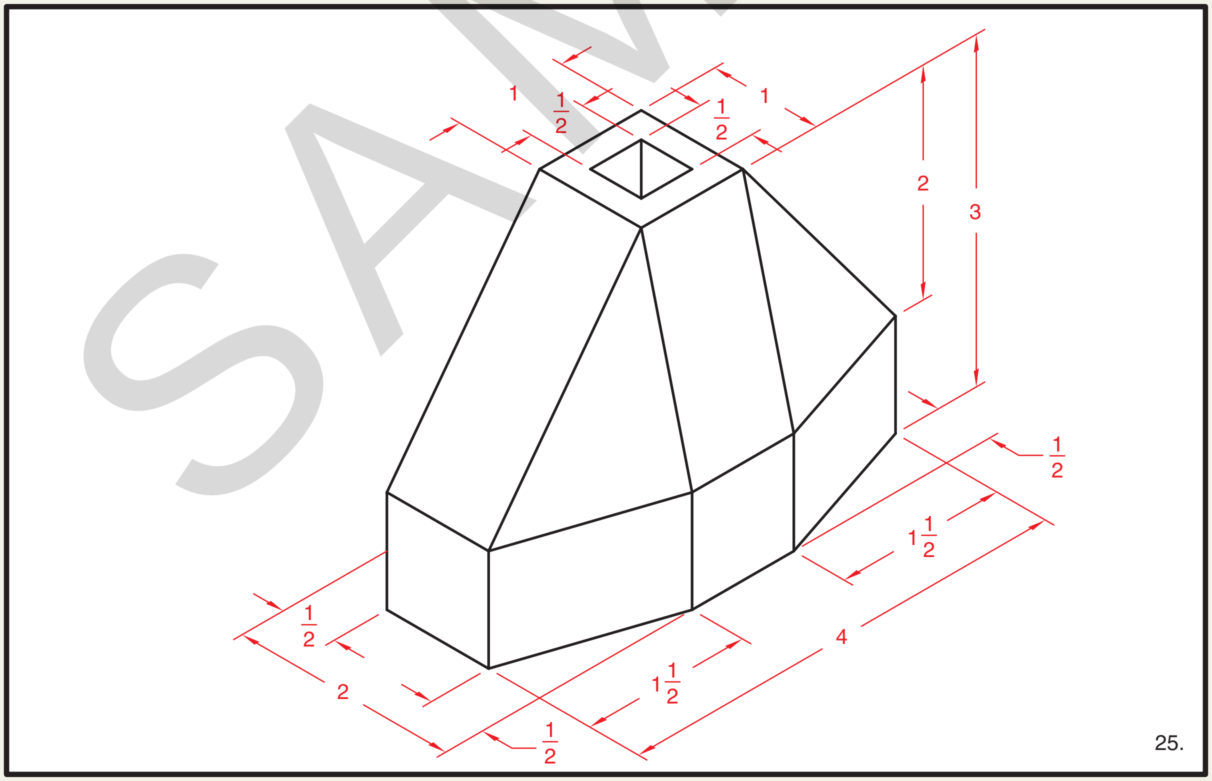
Problem 10-22. ★★★ Heat Sink. Draw the necessary views and correctly dimension them.



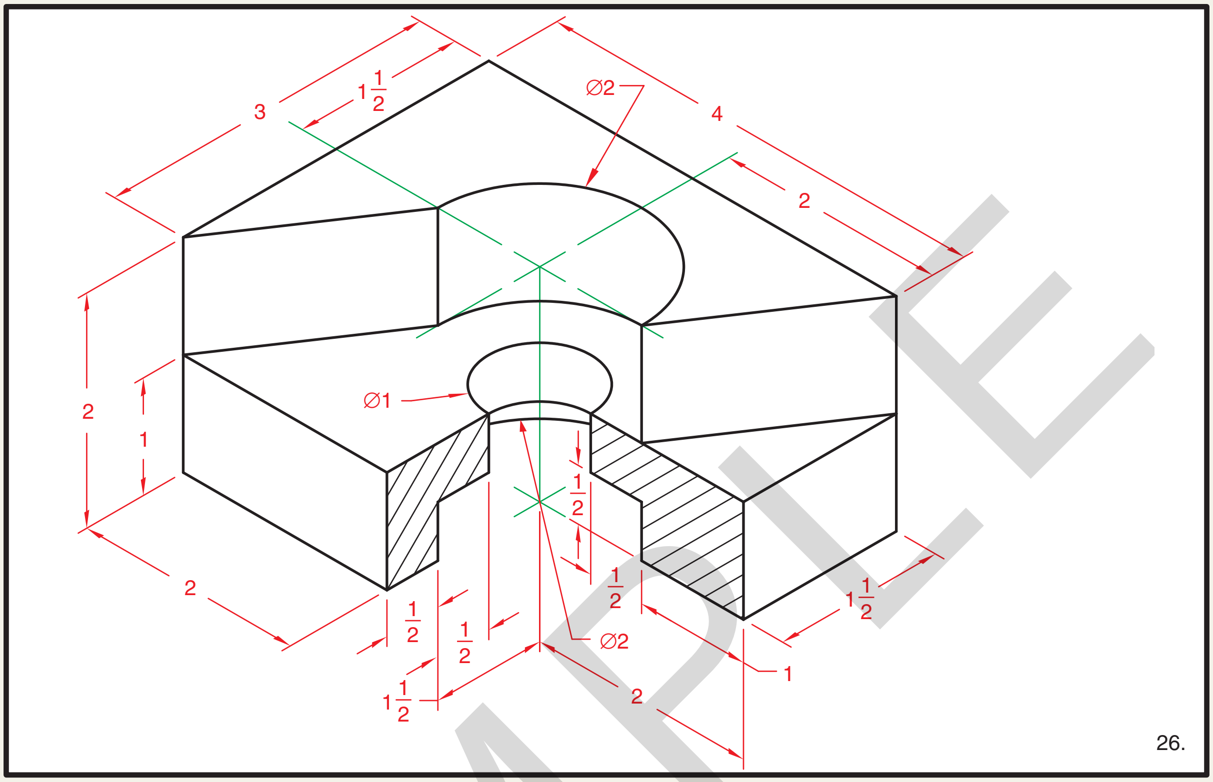
Problem 10-23. ★★★ Motor Bracket. Draw the necessary views and correctly dimension them.



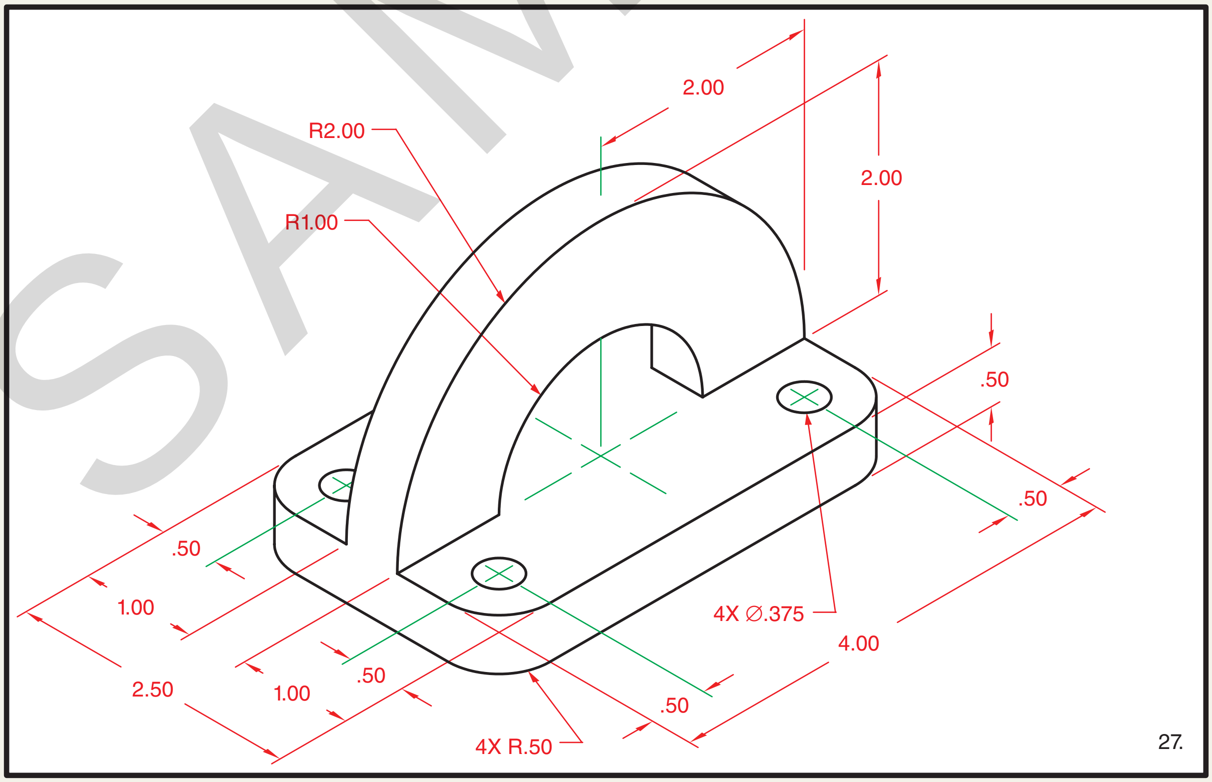
Problem 10-24. ★★★ Bearing Cradle. Draw the necessary views and correctly dimension them.



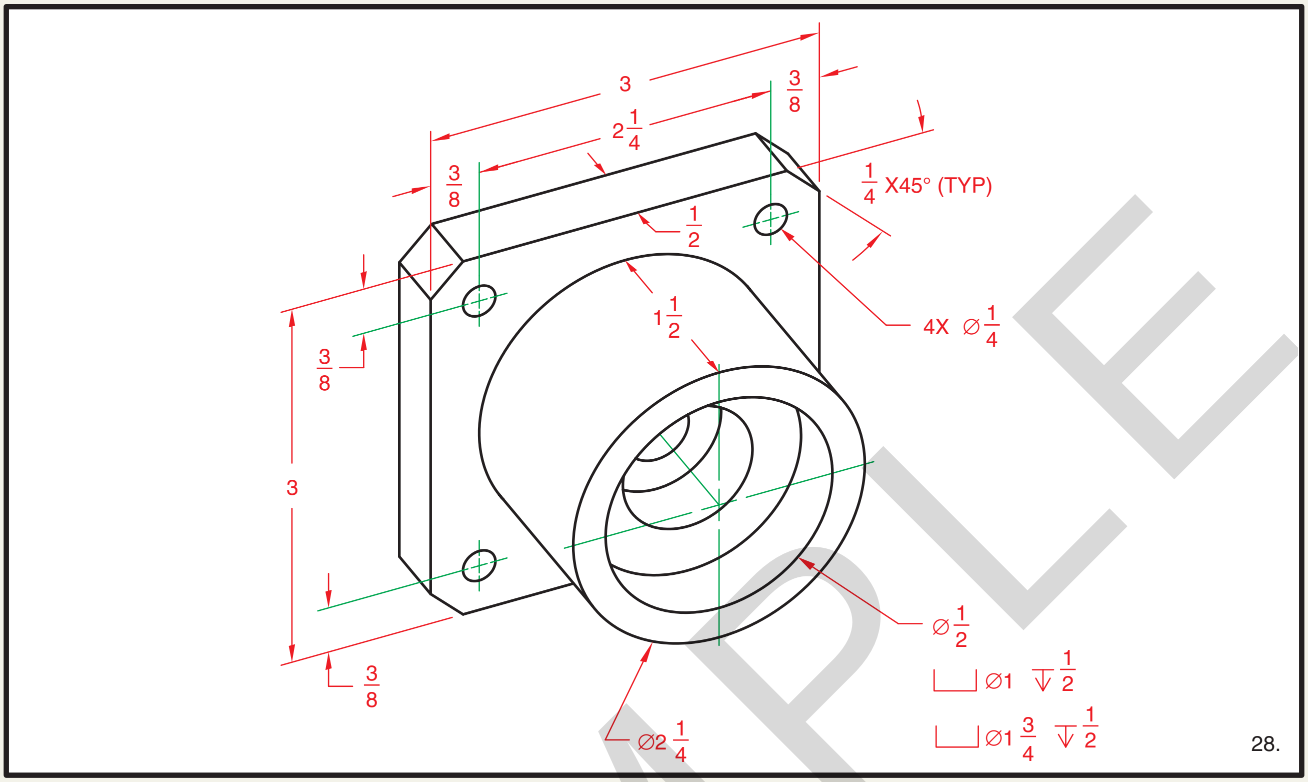
Problem 10-25. ★★★ Pedestal Base. Draw the necessary views and correctly dimension them.



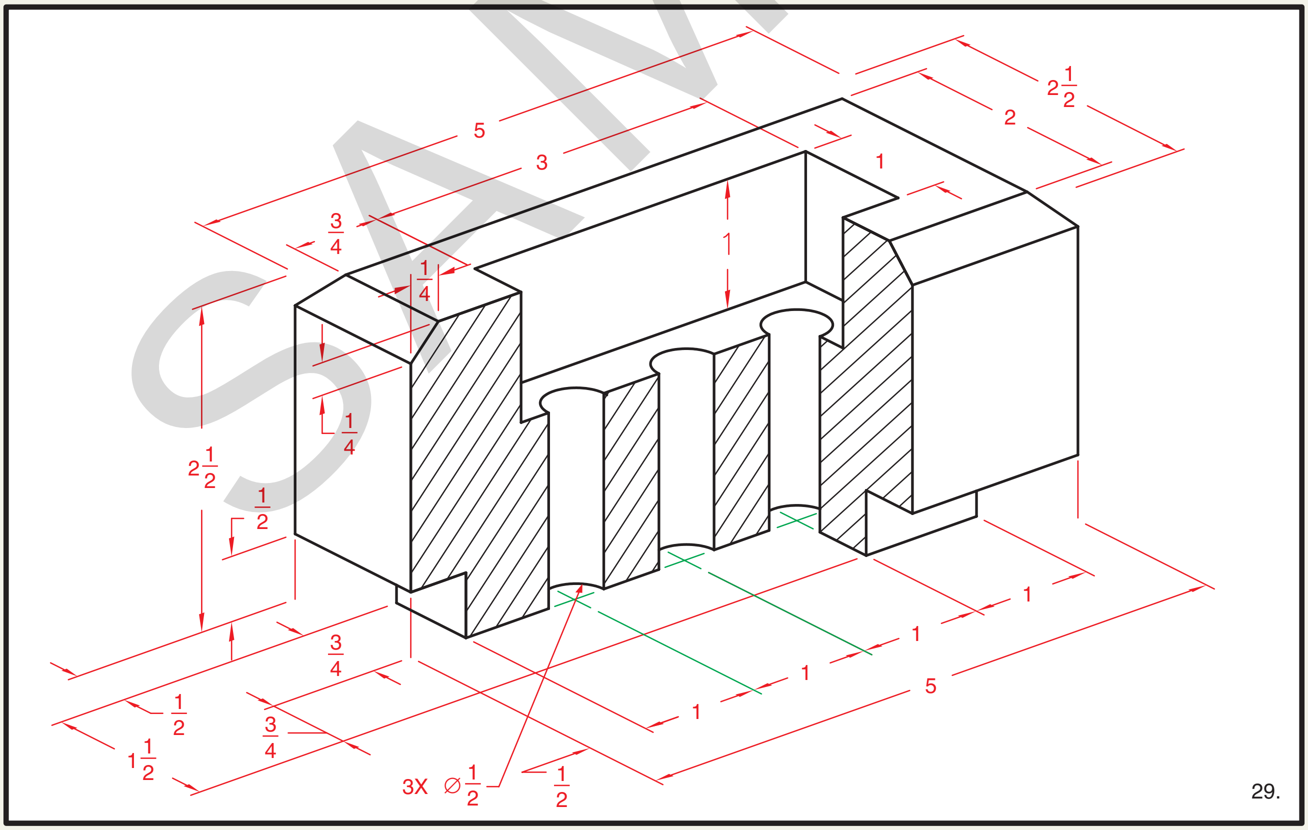
Problem 10-26. ★★★ Switch Base. Draw the necessary views and correctly dimension them.



Problem 10-27. ★★★ Latch Bracket. Draw the necessary views and correctly dimension them.



Problem 10-28. ★★★ Bearing Plate. Draw the necessary views and correctly dimension them.



Problem 10-29. ★★★ Slotted Insert. Draw the necessary views and correctly dimension them.

Design Problems

Design and prepare drawings for the following.

A. Storage cabinet for video games
B. Carrying case for gaming accessories
C. Wall hanger for skateboard or snowboard
D. Portable storage case for DVDs
E. Snowboard or sled
F. Ice skating blade
G. Snooker or pool equipment rack

30.

Problem 10-30. ★★★ Design Problems. Complete each design problem. Draw and dimension as many views as necessary to fully describe the object.