9.1 DIMENSIONING

Geometrics is the science of specifying and tolerancing the shapes and locations of features on objects. Once the shape of a part is defined with an orthographic drawings, the size information is added also in the form of **dimensions**. Dimensioning a drawing also identifies the tolerance (or accuracy) required for each dimension.

9.1 If a part is dimensioned properly, then the intent of the designer is clear to both the person making the part and the inspector checking the part.

9.2 A fully defined part has three elements: graphics, dimensions, and words (notes).

9.2 SIZE AND LOCATION DIMENSIONS

A well dimensioned part will *communicate* the size and location requirements for each feature. Communications is the fundamental purpose of dimensions. Parts are dimensioned based on two criteria:

N Basic size and locations of the features.

N Details of a part's construction, for manufacturing.

On a drawing used in American industry, all dimensions are in inches, unless otherwise stated. Most countries outside of the United States use the metric system of measure, or the international system of units (SI), which is based on the meter. The SI system is being used more in the United States because of global trade and multinational company affiliations

Occasionally, a company will used dual dimensioning, that is, both metric and English measurements on a drawing.

Angular dimensions are shown either in decimal degrees or in degrees, minutes, and seconds.

9.2.1 TERMINOLOGY

There are a number of terms important to the understanding of dimensioning practices.

A **dimension** is the numerical value that defines the size or geometric characteristic of a feature.

A **basic dimension** is the numerical value defining the theoretically exact size of a feature.

A **reference dimension** is the numerical value enclosed in parentheses provided for information only and is not used in the fabrication of the part.

A **dimension line** is the thin solid line which shows the extent and direction of a dimension. Dimension lines are broken for insertion of dimension numbers.

Arrows are placed at the ends of dimension lines to show the limits of the dimension. Arrows are uniform in size and style no matter what the size of the drawing.

An **extension line** is the thin solid line perpendicular to a dimension line indicating which feature is associated with the dimension. There is a visible gap between the feature and the end of an extension line.

A **leader line** is the thin solid line used to indicate the feature with which a dimension, note, or symbol is associated.

A **tolerance** is the amount a particular dimension is allowed to vary.

Limits of size is the largest acceptable size and the minimum acceptable size of a feature. The largest acceptable size is expressed as the maximum material condition (MMC) whereas the smallest acceptable size is expressed as the least material condition (LMC).

Plus and minus dimensioning is the allowable positive and negative variance from the dimension specified

Diameter symbol is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the diameter of a circle. The symbol used is the Greek letter *phi*.

Radius symbol is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the radius of a circle. The radius symbol used is the capital letter *R*.

The **tolerance** is the amount a particular dimension is allowed to vary. All dimensions have either an explicit or implicit tolerance associated with it; that is, the tolerance may be noted directly on the dimesion or implied through a general note.

The **datum** is the theoretically exact point used as a reference for tabular dimensioning.

9.2.2 BASIC CONCEPTS

Dimensions are used to describe the size and location of features on parts for manufacture. The basic criterion is, "What information is necessary to make the object?" Dimensions should not be excessive, either through duplication or dimensioning a feature more than one way.

9.2.3 SIZE DIMENSIONS

A size dimension might be the overall width of the part or the diameter of a drilled hole. A location dimension might be length from the edge of the object to the center of the drilled hole.

The location and orientation of dimensions are based on the three positions: horizontal, vertical, and angles.

Horizontal—the left to the right distance relative to the drawing sheet.

Vertical—the up and down distance relative to the drawing sheet

Diameter—the full distance across a circle, measured through the center.

Radius—the distance from the center of an arc to any point on the arc, usually used on arcs less than half circles.

9.2.4 LOCATION AND ORIENTATION DIMENSIONS

In **rectangular coordinate dimensioning,** a base line (or datum line) is established for each coordinate direction, and all dimensions specified with respect to these baselines. This is also known as **datum dimensioning**, or **baseline dimensioning**. All dimensions are calculated as X and Y distances from an *origin point*, usually placed at the lower left corner of the part.

9.2.5 COORDINATE DIMENSIONS

Tabular coordinate dimensioning involves labeling each feature with a letter, and then providing information on size and location in a table.

9.2.6 STANDARD PRACTICES

The guiding principal for dimensioning a drawing is clarity. To promote clarity, ANSI developed standard practices for showing dimensions on drawings.

Dimension placement depends on the space available between extension lines. When space permits, dimensions and arrows are placed between the extension lines.

When there is room for the numerical value but not both the arrows and the numerical value, the value is placed between the extension lines and the arrows are placed outside the extension lines.

When there is room for the arrows but not the numerical value, the arrows are placed between the extension lines with the value outside the extension lines adjacent to a leader.

When the space is too small for either arrows or the numerical value, both are placed outside of the extension lines as shown.

The minimum distance from the object to the first dimension is 10mm (3/8 inch). The minimum spacing between dimensions is 6mm (1/4 inch). Note that these are minimum values and may be increased where appropriate. There should be a visible gap between an extension line and the feature to which it refers. Extension lines should extend about 1mm (1/32 inch) beyond the last dimension line.

Dimensions should be *grouped* for uniform appearance as shown. As a general rule do not use object lines as part of your dimension

Where there are several parallel dimensions, the values should be *staggered*.

Extension lines are used to refer a dimension to a particular feature and are usually drawn perpendicular to the associated dimension line. Where space is limited, extension lines may be drawn at an angle. Where angled extension lines are used, they must be parallel and the associated dimension lines are drawn in the direction to which they apply.

Extension lines should not cross dimension lines, and should avoid crossing other extension lines whenever possible. When extension lines cross object lines or other extension lines, they are not broken. When extension lines cross or are close to arrowheads, they are broken for the arrowhead.

When the location of the center of a feature is being dimensioned, the center line of the feature is used as an extension line.

When a point is being located by extension lines only, the extensions lines must pass through the point.

When it is necessary to define a limited length or area that is to receive additional treatment (such as the knurled portion of a shaft), the extent of the limits may be shown by a chain line. The chain line is drawn parallel to the surface being defined. If the chain line applies to a surface of revolution, only one side need be shown.

When the limited area is being defined in a normal view of the surface, the area within the chain line boundary is section lined. Dimensions are added for length and location unless the chain line clearly indicates the location and extent of the surface area.

All dimension and note text must be oriented to be read from the bottom of the drawing (relative to the drawing format). Placement of all text to be read from the bottom of the drawing is called **unidirectional dimensioning**. **Aligned dimensions** have text placed parallel to the dimension line with vertical dimensions read from the right of the drawing sheet.

Dimensions are to be kept outside of the boundaries of views of objects wherever practical. Dimensions may be place within the boundaries of objects in cases where extension or leader lines would be too long, or where clarity would be improved.

If it is necessary to include a dimension which is out of scale, the out of scale dimension text must be underlined.

The symbol X is used to indicate the number of times a feature is to be repeated. The number of repetitions, followed by the symbol X and a space precedes the dimension text.

9.3 DETAIL DIMENSIONING

Holes are typically dimensioned in a view which best describes the shape of the hole. Diameters must be dimensioned with the diameter symbol preceding the numerical value. When holes are dimensioned with a leader line, the line must be radial. A **radial lin***e* is one that passes through the center of a circle or arc if extended. When it is not otherwise clear that a hole extends completely through a part, the word THRU shall follow the numerical value.

Symbols may be used for spotface, counterbore, and countersunk holes. These symbols always precede the diameter symbol. The depth symbol may be used to indicate the depth of a hole. The depth symbol is placed preceding the numerical value.

When the depth of a **blind hole** is specified, it refers to the depth of the full diameter of the hole.

When a chamfer or countersink is placed in a curved surface, the diameter given refers to the minimum diameter of the chamfer or countersink. If the depth or remaining thickness of material for a spotface is not given, the spotface depth is the smallest amount required to clean up the material surface to the specified diameter. Chamfers are dimensioned by providing either an angle and a linear dimension or by providing two linear dimensions. Chamfers of 45 degrees may be specified in a note.

Slotted holes may be dimensioned any of several ways depending on which is most appropriate for the application.

Keyseats are dimensioned in a particular way, because they present some unusual problems.

Summary of current (Y14.5-1994) and previous dimensioning standards used for various features.

The diameter is specified for holes and blind holes. Blind holes are ones that do not go through the part. If the hole does not go through, the depth is specified, preceded by the depth symbol. Holes with no depth call out are assumed to go through.

A **counterbore** symbol is placed before the diameter callout, and the depth of the counterbore is added with a depth symbol. If a depth is stated, it is a counterbore. If not, then it is a spotface. The full note shows the diameter of the through hole followed by the diameter of the counterbore then the depth of the counterbore. The spotface has the same specification as the counterbore, except that the depth is not specified.

A **countersink** symbol is placed with a diameter of the finished countersink, followed by the angle specification. The reason the depth is not given is that the resultant diameter is much easier to measure.

If a full circle or an arc of more than half of a circle is being dimensioned, the diameter is specified, preceded by the diameter symbol which is the Greek letter phi. If the arc is less than half of a circle, then the radius is specified and it is preceded by an R. Concentric circles should be dimensioned in the longitudinal view whenever practical.

Radii are dimensioned with the radius symbol preceding the numerical value. The dimension line for radii shall have a single arrowhead touching the arc. When there is adequate room the dimension is placed between the center of the radius and the arrowhead. When space is limited, a radial leader line is used. When the center of an arc is not clearly defined by being tangent to other dimensioned features on the object, the center of the arc is noted with a small cross.

The center of the radius is not noted if the radius is shown tangent to defined surfaces. If the center of an arc interferes with another view or is outside of the drawing area, foreshortened dimension lines may be used.

When a radius is dimensioned in a view where it does not appear true shape the word TRUE appears preceding the radius symbol.

There are standards that apply directly to each size thread. ANSI Y14.6 is a complete definition of all of the inch series threads. Local notes are used to identify thread types and dimensions. For threaded holes, the note should be placed on the circular view. For external threads, the note is placed on the longitudinal view of the thread.

Two dimensions are necessary for a groove, the width and the depth or diameter.

There are many manufacturers standards that have been devised over the years which define the sizes of certain commodities. Figure 9.32 shows a gage table for sheet metal.

9.4 DIMENSIONING TECHNIQUES

Dimensioning is accomplished by adding size and location information. One dimensioning technique is called **contour dimensioning,** because the contours or shapes of the object are dimensioned in their most descriptive view. For example, the radius of a arc would be dimensioned where it appears as an arc and not as a hidden feature.

A second method of dimensioning a part is to break the part into its geometric configurations. This method is called **geometric breakdown** and is used on objects made of geometric primitives, such as prisms, cylinders, and spheres, or their derivatives such as half spheres or negative cylinders (holes).

The importance of accurate and unambiguous dimensioning cannot be overemphasized. The primary guideline is that of clarity and whenever two guidelines appear to conflict, the method which most clearly communicates the size information shall prevail. Every dimension must have an associated tolerance, and that tolerance must be clearly shown on the drawing.

- Avoid over-dimensioning a part. Double dimensioning of a feature is not permitted.

- Dimensions should be placed in the view which most clearly describes the feature being dimensioned.

- A minimum spacing between the object and dimensions and between dimensions must be maintained.

- A visible gap shall be placed between the end of extension lines and the feature to which they refer.

- Manufacturing methods should not be specified as part of the dimension unless no other method of manufacturing is acceptable.

- Placing dimensions within the boundaries of a view should be avoided whenever practicable.

- Dimensions for materials typically manufactured to gages or code numbers shall be specified by numerical values.

- Unless otherwise specified, angles shown on drawings are assumed to be 90 degrees.

- Dimensioning to hidden lines should be avoided whenever possible. Hidden lines are less clear than visible lines.

- The depth of blind, counterbored, or countersunk holes may be specified in a note along with the diameter.

- Diameters, radii, squares, counterbores, spotfaces, countersinks, and depth should be specified with the appropriate symbol preceding the numerical value.

- Leader lines for diameters and radii should be radial lines.

9.5 TOLERANCING

Tolerances are used to control the amount of variation inherent in all manufactured parts. In particular, tolerances are assigned to mating parts in an assembly. For example, the slot in the part must accommodate another part. One of the great advantages of using tolerances is that it allows for **interchangeable parts,** thus permitting the replacement of individual parts.

Tolerance is the total amount a dimension may vary and is the difference between the **upper** (maximum) and **lower** (minimum) limits.

Tolerances are expressed as:

9.6 TOLERANCE REPRESENTATION

Direct limits or as tolerance values applied directly to a dimension.

Geometric tolerances contain the following:

- Notes referring to specific conditions.

- A general tolerance note in the title block.

If a dimension has a tolerance added directly to it, that tolerance supersedes the general tolerance note. *A tolerance added to a dimension always supersedes the standard tolerance, even if the added tolerance is larger than the standard tolerance.*

Tolerances can be applied directly to dimensioned features, using limit dimensioning This is the ANSI preferred method; the maximum and minimum sizes are specified as part of the dimension. Either the upper limit is placed above the lower limit, or when the dimension is written in a single line, the lower limit precedes the upper limit and they are separated by a dash.

The basic size is given, followed by a plus/minus sign and the tolerance value. Tolerance can be unilateral or bilateral. A unilateral tolerance varies in only one direction, while a bilateral varies in both directions. If the variation is equal in both directions, then the variation is preceded by a ± symbol. The plus and minus approach can only be used when the two variations are equal.

A system of two parts that have toleranced dimensions. These two parts are used as an example to define ANSI Y14.5M-1982 terms:

- **Nominal size** a dimension used to describe the general size usually expressed in common fractions.

- **Basic size** the theoretical size used as a starting point for the application of tolerances.

- **Actual size** the measured size of the finished part after machining.

- **Limits** the maximum and minimum sizes shown by the toleranced dimension.

- **Allowance** is the minimum clearance or maximum interference between parts.

- **Tolerance** is the total variance in a dimension which is the difference between the upper and lower limits. The tolerance of the slot in Figure 14.50 is .004" and the tolerance of the mating part is .002".

- **Maximum material condition (MMC)** is the condition of a part when it contains the most amount of material. The MMC of an external feature such as a shaft is the upper limit. The MMC of an internal feature such as a hole is the lower limit.

- **Least material condition (LMC)** is the condition of a part when it contains the least amount of material possible. The LMC of an external feature is the lower limit of the part. The LMC of an internal feature is the upper limit of the part.

The degree of tightness between mating parts is called the **fit**. **Clearance fit** occurs when two toleranced mating parts will always leave a space or clearance when assembled. **Interference fit** occurs when two toleranced mating parts will always interfere when assembled. **Transition fit** occurs when two toleranced mating parts will sometimes be an interference fit and sometimes be a clearance fit when assembled.

The **loosest fit** is the difference between the smallest feature A and the largest feature B. The tightest fit is the difference between the largest feature A and the smallest feature B.

When dimensioning a part it is critical to start out by identifying the functional features first. Many times these features are holes. Any other features that come in contact with other parts, especially moving parts, are considered functional. Dimension these features first, then all other non-functional features can be considered.

The additive rule for tolerances is that tolerances taken in the same direction from one point of reference are additive. The corollary is that tolerances to the same point taken from different directions become additive. The effect is called **tolerance stack-up**.

The most desirable, and the most recommended, procedure is to relate the two holes directly to each other, not to either side of the part, not to the overall width of the part.

The terms used in metric tolerancing are as follows:

- **Basic Size** is the size to which limits of deviation are assigned and are the same for both parts.
- **Deviation** is the difference between the size of the part and the basic size.
- **Upper deviation** is the difference between the maximum limit of size and the basic size.
- **Lower deviation** is the difference between the minimum limit of size and the basic size.
- **Fundamental deviation** is the deviation closest to the basic size.
- **Tolerance** is the difference between the maximum and minimum size limits on a part.
- **Tolerance zone** represents the tolerance and its position in relation to the basic size.
- **International tolerance grade (IT)** a group of tolerances which vary depending upon the basic size, but have the same level of accuracy with a given grade.
- **Hole basis** is the system of fits where the minimum hole size is the basic size.
- **Shaft basis** is the system of fits where the minimum shaft size is the basic size.

Some methods of designating metric tolerances on drawings.

The *hole basis system* is for clearance, interference, and transition fits.

The *shaft basis system* is for clearance, interference, and transition fits.

A description of preferred metric fits.

The line and note form for representing tolerances on drawings. The line form gives the actual tolerance values, and the note form uses the basic size and letters that refer to standard tables to determine the size.

Techniques for applying metric tolerances to a technical drawing.

A special group of English unit tolerance relationships, called **preferred precision fits**, have been developed over many years and work well in certain circumstances. Tables in the Appendices list each type of fit.

When a shaft and a hole are the same size, it is referred to as a **line-to-line fit**.

The *basic size* is the exact theoretical size from which the limits of mating parts are assigned when tolerancing. In the basic hole system, used to apply tolerances to a hole and shaft assembly, the smallest hole is assigned the basic diameter from which the tolerance and allowance is applied.

Creating a clearance fit using the basic hole system.

Applying tolerances for an interference fit.

The basic shaft system, a less popular method of applying tolerances to a shaft and a hole, can be applied when using shafts that are produced in standard sizes. For this system, the largest diameter of the shaft is assigned the basic diameter from which all tolerances are applied.

9.7 TOLERANCES IN CAD

Rather an interpreting dimensions on a drawing, a machinist can extract geometric information directly from an electronic database created by a CAD system. For that reason, it is important that the geometry be as accurate as possible when working in CAD. For that reason, it is important to alter geometry and updated the dimension value from it rather than hand editing the dimension value.

CAD drawings, then, can be considered *geometry files* rather than simply drawings.

Since a CAD system can typically only represent geometry at one size, a feature is constructed at its basic size with its tolerance range note in the dimension or as a general note.

Do not use coordinate positions from a readout to locate features because readouts are accurate only to the number of decimals displayed. For example, a reading of 4.0000 may appear to be 4 inches, but if the readout is changed to five decimals, it may read 4.00004.

Avoid the temptation to type over a dimension when it is not exactly correct; *change the geometry, not the dimension value.*

9.8 SURFACE TEXTURE SYMBOLS

The surface texture of a finished part is critical for many products, such as automobiles and aircraft, to reduce friction between parts or aerodynamic drag caused by the friction of air passing over the surface. Standard drawing practices relate directly to the grinding process, which is used to produce finished surfaces. ANSI Y14.36–1978 (R1987) is the standard used for designating surface texture and material.

Finish marks are shown in every view in which the finished surface appears as a visible or hidden line

9.9 SUMMARY

Dimensioning is a method of accurately communicating size information for objects and structures so that they can be produced. Dimensioning of mechanical devices follows standards established by ANSI. These standards include the proper use and placement of dimensional information on engineering drawings. Many parts need to be dimensioned using toleranced values. Tolerances allow a dimension to vary within limits. Toleranced dimensions are especially useful in the accurate manufacture of assembled parts.

Just as the clear communication about the shape of an object is accomplished by following the standard principles and conventions of orthographic projection, the clear communication about the size of an object is accomplished through adherence to standard dimensioning practices. Unless both shape and size information are communicated clearly, it is not possible to move design ideas to reality. Tables 9.1 and 9.2 summarize the basics of dimensioning and tolerancing.