Chapter 8

Basic Dimensioning, Geometric Dimensioning, and Tolerancing

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the need for dimensioning in drawings.*
- *Understand the fundamental dimensioning terms.*
- *Understand associative dimensioning.*
- *Create various types of dimensions in the drawing.*
- *Create center marks and centerlines.*
- *Attach leaders to objects.*
- *Use geometric tolerancing, feature control frames, and characteristics symbols.*
- *Combine geometric characteristics and create composite position tolerancing.*
- *Use the projected tolerance zone.*
- *Use feature control frames with leaders.*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

NEED FOR DIMENSIONING

To make designs more informative and practical, the drawing must convey more than just the graphic picture of the product. To manufacture an object, the drawing must contain size descriptions such as the length, width, height, angle, radius, diameter, and location of features. All this information is added to the drawing with the help of **dimensioning**. Some drawings also require information about tolerances with the size of features. This information conveyed through dimensioning is vital and often just as important as the drawing itself. With the advances in computer-aided design/drafting and computer-aided manufacturing, it has become mandatory to draw the part to actual size so that the dimensions reflect the actual size of the features. At times, it may not be necessary to draw the object of the same size as the actual object would be when manufactured, but it is absolutely essential that the dimensions be accurate. Incorrect dimensions will lead to manufacturing errors.

By dimensioning, you not only give the size of a part, you also give a series of instructions to a machinist, an engineer, or an architect. The way the part is positioned in a machine, the sequence of machining operations, and the location of various features of the part depend on how you dimension the part. For example, the number of decimal places in a dimension (2.000) determines the type of machine that will be used to do that machining operation. The machining cost of such an operation is significantly higher than for a dimension that has only one digit after the decimal (2.0). Similarly, whether a part is to be forged or cast, the radii of the edges, and the tolerance you provide to these dimensions determine the cost of the product, the number of defective parts, and the number of parts you get from a single die.

DIMENSIONING IN AutoCAD LT

The objects that can be dimensioned in AutoCAD LT range from straight lines to arcs. The dimensioning commands provided by AutoCAD LT can be classified into four categories:

Dimension Drawing Commands
Dimension Style Commands
Dimension Editing Commands
Dimension Utility Commands

While dimensioning an object, AutoCAD LT automatically calculates the length of the object or the distance between two specified points. Also, settings such as the gap between the dimension text and the dimension line, the space between two consecutive dimension lines, arrow size, and text size are maintained and used when the dimensions are being generated for a particular drawing. The generation of arrows, lines (dimension lines, extension lines), and other objects that form a dimension is automatically performed by AutoCAD LT to save the user's time. This also results in uniform drawings. However, you can override the default measurements computed by AutoCAD LT and change the settings of various standard values. The modification of dimensioning standards can be achieved through the dimension variables.

The dimensioning functions offered by AutoCAD LT provide you with extreme flexibility in dimensioning by letting you dimension various objects in a variety of ways. This is of great help

because different industries, such as architectural, mechanical, civil, or electrical, have different standards for the placement of dimensions.

FUNDAMENTAL DIMENSIONING TERMS

Before studying AutoCAD's dimensioning commands, it is important to know and understand various dimensioning terms that are common to linear, angular, radius, diameter, and ordinate dimensioning. Figures 8-1 and 8-2 show various dimensioning parameters.

Figure 8-1 *Various dimension parameters*

Figure 8-2 *Various dimension parameters*

Dimension Line

The **dimension line** indicates which distance or angle is being measured. Usually, this line has arrows at both ends, and the dimension text is placed along the dimension line. By default the dimension line is drawn between the extension lines (Figure 8-1). If the dimension line does not fit inside, two short lines with arrows pointing inward are drawn outside the extension lines. The dimension line for angular dimensions (which are used to dimension angles) is an arc. You can control the positioning and various other features of the dimension lines by setting the parameters in the dimension styles. (The dimension styles are discussed in Chapter 10.)

Dimension Text

Dimension text is a text string that reflects the actual measurement (dimension value) between the selected points as calculated by AutoCAD. You can accept the value that AutoCAD LT returns or enter your own value. In case you use the default text, AutoCAD LT can be supplied with instructions to append the tolerances to it. Also, you can attach prefixes or suffixes of your choice to the dimension text.

Arrowheads

An **arrowhead** is a symbol used at the end of a dimension line (where dimension lines meet the extension lines). Arrowheads are also called **terminators** because they signify the end of the dimension line. Since the drafting standards differ from company to company, AutoCAD LT allows you to draw arrows, tick marks, closed arrows, open arrows, dots, right angle arrows, or user-defined blocks (Figure 8-3). The user-defined blocks at the two ends of the dimension line

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

can be customized to your requirements. The size of the arrows, tick marks, user blocks, and so on can be regulated by using the dimension variables.

Figure 8-3 Using arrows, tick marks, and user-defined blocks

Extension Lines

Extension lines are drawn from the object measured to the dimension line (Figure 8-4). These lines are also called **witness lines.** Extension lines are used in linear and angular dimensioning. Generally, extension lines are drawn perpendicular to the dimension line. However, you can make extension lines inclined at an angle by using the **DIMEDIT** command (Oblique option) or by selecting **Dimension Edit** from the **Dimension** toolbar. AutoCAD LT also allows you to suppress either one or both extension lines in a dimension (Figure 8-5). Other aspects of the extension line can be controlled by using the dimension variables (these variables are discussed in Chapter 10).

Figure 8-4 Extension lines

Figure 8-5 Extension line suppression

Leader

A **leader** is a line that stretches from the dimension text to the object being dimensioned. Sometimes the text for dimensioning and other annotations do not adjust properly near the object. In such cases, you can use a leader and place the text at the end of the leader line. For example, the circle shown in Figure 8-6 has a keyway slot that is too small to be dimensioned. In this situation, a leader can be drawn from the text to the

Figure 8-6 Leader used to attach annotation

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

keyway feature. Also, a leader can be used to attach annotations such as part numbers, notes, and instructions to an object.

Center Mark and Centerlines

The **center mark** is a cross mark that identifies the center point of a circle or an arc. Centerlines are mutually perpendicular lines passing through the center of the circle/arc and intersecting the circumference of the circle/arc. A center mark or the centerlines are automatically drawn when you dimension a circle or arc (see Figure 8-7). The length of the center mark and the extension of the centerline beyond the circumference of the circle is determined by the value assigned to the **DIMCEN** dimension variable or the value assigned to **Center Marks for Circles** in the **Dimension Style Manager** dialog box.

Alternate Units

With the help of **alternate units**, you can generate dimensions for two systems of measurement at the same time (Figure 8-8). For example, if the dimensions are in inches, you can use the alternate units dimensioning facility to append metric dimensions to the dimensions (controlling the alternate units through the dimension variables is discussed in Chapter 10).

Figure 8-7 *Center mark and centerlines*

Figure 8-8 *Using alternate units dimensioning*

Tolerances

Tolerance is the amount by which the actual dimension can vary (Figure 8-9). AutoCAD LT can attach the plus/minus tolerances to the dimension text (actual measurement computed by AutoCAD LT). This is also known as **deviation tolerance**. The plus and minus tolerance that you specify can be the same or different. You can use the dimension variables to control the tolerance feature (these variables are discussed in Chapter 10).

Limits

Instead of appending the tolerances to the dimension text, you can apply the tolerances to the measurement itself (Figure 8-10). Once you define tolerances, AutoCAD LT will automatically calculate the upper and lower **limits** of the dimension. These values are then displayed as a dimension text.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

For example, if the actual dimension as computed by AutoCAD LT is 2.6105 units and the tolerance values are +0.025 and -0.015, the upper and lower limits are 2.6355 and 2.5955. After calculating the limits, AutoCAD LT will display them as dimension text, as shown in Figure 8-10. The dimension variables that control the limits are discussed in Chapter 10.

Figure 8-9 *Using tolerances with dimensions*

Figure 8-10 *Using limits dimensioning*

ASSOCIATIVE DIMENSIONS

Associative dimensioning is a method of dimensioning, in which the dimension is associated with the object that is dimensioned. In other words, the dimension is influenced by the changes in the size of the object. In the previous releases of AutoCAD, the dimensions were not truly associative, but were related to the objects being dimensioned by definition points on the DEFPOINTS layer. To cause the dimension to be associatively modified, these definition points had to be adjusted along with the object being changed. If, for example, you use the **SCALE** command to change an object's size and select the object, the dimensions will not be modified. If you select the object and its defpoints (using the Crossing selection method), then the dimension will be modified. However, in AutoCAD LT 2002, a new concept called **true associative dimensioning** was introduced. This concept ensures that if the dimensions are associated to the object and the object changes its size, the dimensions will also change automatically. With the introduction of the true associative dimensions, there is no need to select the definition points along with the object. This eliminates the use of definition points for updating the dimensions.

The associative dimensions automatically update their values and location if the value or location of the object is modified. For example, if you edit an object using simple editing operations such as breaking, using the **BREAK** command, then the true associative dimension will be modified automatically. The dimensions can be converted into the true associative dimensions using the **DIMREASSOCIATE** command. The association of the dimensions with the objects can be removed using the **DIMDISASSOCIATE** command. Both these commands will be discussed later in this chapter.

The dimensioning variable **DIMASSOC** controls associativity of dimensions. The default value of this variable is **1**, which means turned on. When **DIMASSOC** is turned off, then the dimension will be placed in the exploded format. This means that the dimensions will now be placed as a

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

combination of individual arrowheads, dimension lines, extension lines, and text. Also, note that the exploded dimensions cannot be associated to any object.

DEFINITION POINTS

Definition points are the points drawn at the positions used to generate a dimension object. The definition points are used by the dimensions to control their updating and rescaling. AutoCAD LT draws these points on a special layer called **DEFPOINTS.** These points are not plotted by the plotter because AutoCAD LT does not plot any object on the **DEFPOINTS** layer. If you explode a dimension (which is as good as turning **DIMASSOC** off), the definition points are converted to point objects on the **DEFPOINTS** layer. In Figure 8-11, the small circles indicate the definition points for different objects.

Figure 8-11 *Definition points of linear, angular, and ordinate dimensions*

The definition points for linear dimensions are the points used to specify the extension lines and the point of intersection of the first extension line and the dimension line. The definition points for the angular dimension are the endpoints of the lines used to specify the dimension and the point used to specify the dimension line arc. For example, for a three-point angular dimension, the definition points are the extension line endpoints, angle vertex, and the point used to specify the dimension line arc.

The definition points for the radius dimension are the center point of the circle or arc, and the point where the arrow touches the object. The definition points for the diameter dimension are the points where the arrows touch the circle. The definition points for the ordinate dimension are the UCS origin, feature location, and leader endpoint.

Note

In addition to the definition points just mentioned, the middle point of the dimension text serves as the definition point for all types of dimensions.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

SELECTING DIMENSIONING COMMANDS

Using the Toolbar and the Dimension Menu

You can select the dimension commands from the **Dimension** toolbar by choosing the desired dimension button (Figure 8-12), or from the **Dimension** menu (Figure 8-13). The **Dimension** toolbar can also be displayed by right-clicking on any toolbar and choosing **Dimensions** from the shortcut menu.

Figure 8-12 *The* ***Dimension*** *toolbar*

Using the Pointer Input

You can directly enter a dimensioning command at the pointer input or use the **DIM** or the **DIM1** commands to invoke the dimensioning commands. Both these methods of using the command line are discussed next.

Using Dimensioning Commands

The first method of using the pointer input is to directly enter the dimensioning command at the pointer input. For example, if you want to draw the linear dimension, the **DIMLINEAR** command can be entered directly at the pointer input.

Command: **DIMLINEAR** Enter
Specify first extension line origin or <select object>: *Select a point or press ENTER.*
Specify second extension line origin: *Select second point.*
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *Select a point to locate the position of the dimension.*
Command: *(After you have finished dimensioning, AutoCAD LT returns to the Command prompt.)*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

DIM and DIM1 Commands

Since dimensioning has several options, it also has its own command mode. The **DIM** command keeps you in the dimension mode, and the **Dim:** prompt is repeated after each dimensioning command until you exit the dimension mode to return to the normal AutoCAD LT Command prompt. To exit the dimension mode, enter EXIT (or just E) at the **Dim:** prompt. You can also exit by pressing ESC. The previous command will be repeated if you press the SPACEBAR or ENTER at the **Dim:** prompt. In the dimension mode, it is not possible to execute the normal set of AutoCAD LT commands, except function keys, object snap overrides, control key combinations, transparent commands, dialog boxes, and menus.

Figure 8-13 *Selecting dimensions from the* ***Dimension*** *menu*

Command: **DIM**
Dim: **Hor**
Specify first extension line origin or <select object>: *Select a point or press ENTER.*
Specify second extension line origin: *Select the second point.*
Specify dimension line location or [MText/Text/Angle]: *Select a point to locate the position of the dimension.*
Enter dimension text <default>: *Press ENTER to accept the default dimension.*
Dim: *(After you have finished dimensioning, AutoCAD LT returns to the* ***Dim:*** *prompt.)*

The **DIM1** command is similar to the **DIM** command. The only difference is that **DIM1** lets you execute a single dimension command and then automatically takes you back to the normal Command prompt.

Command: **DIM1**
Dim: **Hor**
Specify first extension line origin or <select object>: *Select a point or press ENTER.*
Specify second extension line origin: *Select the second point.*
Specify dimension line location or [MText/Text/Angle]: *Select a point to locate the position of the dimension.*
Enter dimension text <default>: *Press ENTER to accept the default dimension.*
Command: *(After you are done dimensioning, AutoCAD LT returns Command prompt.)*

AutoCAD LT has provided the following seven fundamental dimensioning types.

Quick dimensioning	**Linear dimensioning**	**Diameter dimensioning**
Radius dimensioning	**Angular dimensioning**	**Ordinate dimensioning**
Arc Length dimensioning		

Figures 8-14 and 8-15 show the various fundamental dimension types.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Note

*The **DIMDEC** variable sets the number of decimal places for the value of primary dimension and the **DIMADEC** variable for angular dimensions. For example, if **DIMDEC** is set to 3, AutoCAD LT will display the decimal dimension up to three decimal places (2.037).*

seTtings] <Continuous>: *Press D for diameter dimensioning and select a point where you want to position the radial dimension.*

Figure 8-14 *Linear and angular dimensions*

Figure 8-15 *Radius, diameter, and ordinate dimensions*

CREATING LINEAR DIMENSIONS

Toolbar:	Dimension > Linear
Menu:	Dimension > Linear
Command:	DIMLIN or DIMLINEAR

Linear dimensioning applies to those dimensioning commands that measure the shortest distance between two points. You can directly select the object to dimension or select two points. The points can be any two points in the space, endpoints of an arc or line, or any set of points that can be identified. To achieve accuracy, points must be selected with the help of object snaps or by selecting an object to dimension. In case, the object selected is aligned, then the linear dimensions will add **Horizontal** or **Vertical** dimensions to the object. The prompt sequence that will follow when you choose this button is given next.

Specify first extension line origin or <select object>: Enter
Select object to dimension: *Select the object.*
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *Select a point to locate the position of the dimension.*

Instead of selecting the object, you can also select the two endpoints of the line that you want to dimension (Figure 8-16). Usually the points on the object are selected by using the **object snaps** (endpoints, intersection, center, etc.). The prompt sequence is as follows.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Specify first extension line origin or <select object>: *Select a point.*
Specify second extension line origin: *Select second point.*
Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *Select a point to locate the position of the dimension.*

Figure 8-16 *Drawing linear dimension*

When using the **DIMLINEAR** command, you can obtain the horizontal or vertical dimension by simply defining the appropriate dimension location point. If you select a point above or below the dimension, AutoCAD LT creates a horizontal dimension. If you select a point that is on the left or right of the dimension, AutoCAD LT creates a vertical dimension through that point.

DIMLINEAR Command Options

The options under this command are discussed next.

Mtext Option

The **Mtext** option allows you to override the default dimension text and also change the font, height, and so on, using the **In-Place Text Editor** dialog box. When you enter **M** at the **Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]** prompt, the **In-Place Text Editor** is displayed. By default, it includes the <> code to represent the measured dimension text. You can change the text by entering a new text and deleting the <> code. You can also use the various options of the text editor (explained in Chapter 7) and then choose **OK**. However, if you override the default dimensions, the dimensional associativity of the **dimension text** is lost. This means that if you modify the object using the definition points, AutoCAD LT will not recalculate the dimension text. Even if the dimension is a true associative dimension, the text will not be recalculated when the object is modified. The prompt sequence to invoke this option is given next.

Specify first extension line origin or <select object>: *Specify a point.*
Specify second extension line origin: *Specify second point.*
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: **M** *(Enter dimension text in **In-Place Text Editor** dialog box and then choose **OK**.)*
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *Specify the dimension location.*

Text Option

This option also allows you to override the default dimension. However, this option will prompt you to specify the new text value in the Command prompt itself, see Figure 8-18. The prompt

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

sequence to invoke this option is given next.

Specify first extension line origin or <select object>: *Select a point.*
Specify second extension line origin: *Select second point.*
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: **T**
Enter dimension text <Current>: *Enter new text.*
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *Specify the dimension location.*

Angle Option

This option lets you change the angle of the dimension text, see Figure 8-17.

Horizontal Option

This option lets you create a horizontal dimension regardless of where you specify the dimension location, see Figure 8-17.

Vertical Option

This option lets you create a vertical dimension regardless of where you specify the dimension location, see Figure 8-17.

Rotated Option

This option lets you create a dimension that is rotated at a specified angle, see Figure 8-17.

Note

If you override the default dimensions, the dimensional associativity of the dimension text is lost and AutoCAD LT will not recalculate the dimension when the object is scaled.

Figure 8-17 *Text, Angle, Horizontal, Vertical, and Rotated options*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Example 1 *General*

In this example, you will use linear dimensioning to dimension a horizontal line of 4 units length. The dimensioning will be done by selecting the object and by specifying the first and second extension line origins. Using the **In-Place Text Editor**, modify the default text such that the dimension is underlined.

Selecting the Object

1. Choose the **Linear** button from the **Dimension** toolbar. The prompt sequence is as follows.

   Specify first extension line origin or <select object>: Enter
   Select object to dimension: *Select the line.*
   Specify dimension line location or
   [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: **M**

   *The **In-Place Text Editor** will be displayed, as shown in Figure 8-18. The default dimension value will be displayed in the **<>** code. Select this code and then choose the **Underline** button to underline the text.*

Figure 8-18 *The **In-Place Text Editor***

   Specify dimension line location or
   [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *Place the dimension.*
   Dimension text = 4.00

Specifying Extension Line Origins

2. Choose the **Linear** button from the **Dimension** toolbar. The prompt sequence is as follows.

   Specify first extension line origin or <select object>: *Select the first endpoint of the line using the **Endpoint** object snap, see Figure 8-19.*
   Specify second extension line origin: *Select the second endpoint of the line using the **Endpoint** object snap, see Figure 8-19.*
   Specify dimension line location or
   [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: **M**

   *Select the code and then choose the **Underline** button to underline the text in the **In-Place Text Editor**.*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *Place the dimension.*
Dimension text = 4.00

Figure 8-19 *Line for Example 1*

CREATING ALIGNED DIMENSIONS

Toolbar:	Dimension > Aligned
Menu:	Dimension > Aligned
Command:	DIMALIGNED

Generally, the drawing consists of various objects that are neither parallel to the *X* axis nor to the *Y* axis. Dimensioning of such objects can be done using **aligned dimensioning**. In horizontal or vertical dimensioning, you can only measure the shortest distance from the first extension line origin to the second extension line origin along the horizontal or vertical axis, respectively, whereas, with the help of aligned dimensioning, you can measure the true aligned distance between the two points. The working of the **ALIGNED** dimension command is similar to that of the other linear dimensioning commands. The dimension created with the **ALIGNED** command is **parallel to the object being dimensioned**. The prompt sequence that will follow when you choose this button is given next.

Specify first extension line origin or <select object>: *Specify the first point or press ENTER to select the object.*
Specify second extension line origin: *Specify second point.*
Specify dimension line location or
[Mtext/Text/Angle]: *Specify the location for the dimension line.*
Dimension text = Current.

The options provided under this command are similar to those under the **DIMLINEAR** command. Figure 8-20 illustrates aligned dimensioning.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-20 *Aligned dimensioning*

Exercise 1

Mechanical

Draw the object shown in Figure 8-21 and then use linear and aligned dimensioning to dimension the part. The distance between the dotted lines is 0.5 units. The dimensions should be up to 2 decimal places. To get dimensions up to 2 decimal places, enter DIMDEC at the Command prompt and then enter 2. (There will be more information about dimension variable in Chapter 10.)

Figure 8-21 *Drawing for Exercise 1*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

CREATING THE ARC LENGTH DIMENSION*

The **Arc Length** dimensioning is used to dimension the length of an arc or the polyline arc segment. You are required to select an arc or a Polyline arc segment and the dimension location. Figure 8-22 shows the **Arc Length** dimensioning of an arc. You can invoke this command by choosing the **Arc Length** button from the dimension toolbar. The prompt sequence that will follow is given next.

Select arc or polyline arc segment: *Select arc or polyline arc segment to dimension*
Specify arc length dimension location, or [Mtext/Text/Angle/Partial]: *Specify the location for the dimension line.*
Dimension text = *Current*

Using the **Partial** option, you can dimension a selected portion of the arc, as shown in Figure 8-23. The prompt sequence for the **Partial** option is given next.

Select arc or polyline arc segment: *Select arc or polyline arc segment to dimension*
Specify arc length dimension location, or [Mtext/Text/Angle/Partial]: **P** Enter
Specify first point for arc length dimension: *Specify the first point on arc*
Specify second point for arc length dimension: *Specify the second point on arc*
Specify arc length dimension location, or [Mtext/Text/Angle/Partial]: *Specify the location for the dimension line.*
Dimension text = *Current*

Figure 8-22 *Arc Length dimensioning*

Figure 8-23 *Partial Arc Length dimensioning*

CREATING ROTATED DIMENSIONS

Rotated dimensioning is used when you want to place the dimension line at an angle (if you do not want to align the dimension line with the extension line origins selected), as shown in Figure 8-24. The **ROTATED** dimension option will prompt you to specify the dimension line angle. You can invoke this command by using **DIMLINEAR** (Rotate option) or by entering **ROTATED** at the **Dim:** Command prompt. The prompt sequence is given next.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Dim: **ROTATED**
Specify angle of dimension line <0>: **110**
Specify first extension line origin or <select object>: *Select the lower right corner of the triangle.*
Specify second extension line origin: *Select the top corner.*
Specify dimension line location or [MText/Text/Angle]: *Select the location for the dimension line.*
Enter dimension text <2.0597>: Enter

Figure 8-24 *Rotated dimensioning*

Tip
You can draw horizontal and vertical dimensioning by specifying the rotation angle of 0-degree for horizontal dimensioning and 90-degree for vertical dimensioning.

CREATING BASELINE DIMENSIONS

Toolbar:	Dimension > Baseline
Menu:	Dimension > Baseline
Command:	DIMBASE or DIMBASELINE

Sometimes in manufacturing, you may want to locate different points and features of a part with reference to a fixed point (base point or reference point). This can be accomplished by using **BASELINE dimensioning** (Figure 8-25). With this command you can continue a linear dimension from the first extension line origin of the first dimension. The new dimension line is automatically offset by a fixed amount to avoid overlapping of the dimension lines. This has to be kept in mind that **there must already exist a linear, ordinate, or angular associative dimension to use the Baseline dimensions.** When you choose the **Baseline Dimension** button, the last linear, ordinate, or angular dimension that was created will be selected and used as the baseline. The prompt sequence that will follow when you choose this button is given next.

Specify a second extension line origin or [Undo/Select] <Select>: *Select the origin of the second extension line.*
Dimension text = current
Specify a second extension line origin or [Undo/Select] <Select>: *Select the origin of the second extension line.*
Dimension text = current
Specify a second extension line origin or [Undo/Select] <Select>: *Select the origin of the second extension line or press ENTER.*
Select base dimension: Enter

Figure 8-25 *Baseline dimensioning*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

When you use the **DIMBASELINE** command, you cannot change the default dimension text. However, the **DIM** mode commands allows you to override the default dimension text.

Command: **DIM**
Dim: **HOR**
Specify first extension line origin or <select object>: *Select left corner (P1, Figure 8-24). (Use Endpoint object snap.)*
Specify second extension line origin: *Select the origin of the second extension line (P2).*
Specify dimension line location or [MText/Text/Angle]: **T**
Enter dimension text <1.0000>: **1.0**
Specify dimension line location or [MText/Text/Angle]: *Select the dimension line location.*
Dim: **BASELINE (or BAS)**
Specify a second extension line origin or [Select] <Select>: *Select the origin of the next extension line (P3).*
Enter dimension text <2.0000>: **2.0**
Dim: **BAS**
Specify a second extension line origin or [Select] <Select>: *Select the origin of the next extension line (P4).*
Enter dimension text <3.000>: **2.75**

The next dimension line is automatically spaced and drawn by AutoCAD LT.

CREATING CONTINUED DIMENSIONS

Toolbar:	Dimension > Continue
Menu:	Dimension > Continue
Command:	DIMCONT or DIMCONTINUE

With this command, you can continue a linear dimension from the second extension line of the previous dimension. This is also called as **chained** or **incremental dimensioning**. Note that there must exist linear, ordinate, or angular associative dimension to use the Continue dimensions. The prompt sequence that will follow when you choose this button is given next.

Specify a second extension line origin or [Undo/Select] <Select>: *Select the origin of the second extension line or press ENTER to select the existing dimension.*
Select continued dimension: *Select the dimension.*
Specify a second extension line origin or [Undo/Select] <Select>: *Specify the point on the origin of the second extension line.*
Dimension text = current
Specify a second extension line origin or [Undo/Select] <Select>: *Specify the point on the origin of the second extension line.*
Dimension text = current
Specify a second extension line origin or [Undo/Select] <Select>: Enter
Select continued dimension: Enter

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

In this case also, the **DIM** command should be used if you want to change the default dimension text.

Command: **DIM**
Dim: **HOR**
Specify first extension line origin or <select object>: *Select left corner (P1, see Figure 8-27). (Use Endpoint object snap.)*
Specify second extension line origin: *Select the origin of the second extension line (P2, see Figure 8-27).*
Specify dimension line location or [MText/Text/Angle]: **T**
Enter dimension text <current>: **0.75**
Specify dimension line location or [MText/Text/Angle]: *Select the dimension line location.*
Dim: **CONTINUE**
Specify a second extension line origin or [Select] <Select>: *Select the origin of the next extension line (P3, see Figure 8-26).*
Enter dimension text <current>: Enter
Dim: **CONTINUE**
Specify a second extension line origin or [Select] <Select>: *Select the origin of next extension line (P4, see Figure 8-27).*
Enter dimension text <current>: Enter

Figure 8-26 *Continue dimensioning*

The default base (first extension line) for the dimensions created with the **CONTINUE** command is the previous dimension's second extension line. You can override the default by pressing ENTER at the **Specify a second extension line origin or [Select] <Select>** prompt, and then specifying the other dimension. The extension line origin nearest to the selection point is used as the origin for the first extension line.

Tip
*You can use the **Select** option of the **DIMBASELINE** or the **DIMCONTINUE** command to select any other existing dimension to be used as the baseline or continuous dimension.*

Exercise 2 — *Mechanical*

Draw the object shown in Figure 8-27 and then use the **baseline** dimensioning to dimension the top half and **continue dimensioning** to dimension the bottom half. The distance between the dotted lines is 0.5 units.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Figure 8-27 *Drawing for Exercise 2*

CREATING ANGULAR DIMENSIONS

Toolbar:	Dimension > Angular
Menu:	Dimension > Angular
Command:	DIMANGULAR or DIMANG

Angular dimensioning is used when you want to dimension an angle. This command generates a dimension arc (dimension line in the shape of an arc with arrowheads at both ends) to indicate the angle between two nonparallel lines. This command can also be used to dimension the vertex and two other points, a circle with another point, or the angle of an arc. For every set of points there exists one acute angle and one obtuse angle (inner and outer angles). If you specify the dimension arc location between the two points, you will get the acute angle; if you specify it outside the two points, you will get the obtuse angle. Figure 8-28 shows the four ways to dimension two nonparallel lines. The prompt sequence that will follow when you choose this button is given next.

Select arc, circle, line, or <specify vertex>: *Select the object or press ENTER to select a vertex point where two segments meet.*
Select second line: *Select the second object.*
Specify dimension arc line location or [Mtext/Text/Angle]: *Place the dimension or select an option.*
Dimension text = current

The methods of dimensioning various entities using this command are discussed next.

Dimensioning the Angle Between Two Nonparallel Lines

The angle between two nonparallel lines or two straight line segments of a polyline can be dimensioned with the **DIMANGULAR** dimensioning command. The vertex of the angle is taken as the point of intersection of the two lines. The location of the extension lines and dimension arc is determined by how you specify the dimension arc location.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-28 *Angular dimensioning between two nonparallel lines*

The following example illustrates the dimensioning of two nonparallel lines using the **DIMANGULAR** command invoked using the **Angular Dimension** button.

Select arc, circle, line, or <specify vertex>: *Select the first line.*
Select second line: *Select the second line.*
Specify dimension arc line location or [Mtext/Text/Angle]: **M** *(Enter the new value in the* ***In-Place Text Editor*** *dialog box.)*
Specify dimension arc line location or [Mtext/Text/Angle]: *Specify the dimension arc location.*

Dimensioning the Angle of an Arc

Angular dimensioning can also be used to dimension the angle of an arc. In this case, the center point of the arc is taken as the vertex and the two endpoints of the arc are used as the extension line origin points for the extension lines (Figure 8-29). The following example illustrates the dimensioning of an arc using the **DIMANGULAR** command.

Select arc, circle, line, or <specify vertex>: *Select the arc.*
Specify dimension arc line location or [Mtext/Text/Angle]: *Specify a location for the arc line or select an option.*

Angular Dimensioning of Circles

The angular feature associated with the circle can be dimensioned by selecting a circle object at the **Select arc, circle, line, or <specify vertex>** prompt. The center of the selected circle is used as the vertex of the angle. The first point selected (when the circle is selected for angular dimensioning) is used as the origin of the first extension line. In a similar manner, the second point selected is taken as the origin of the second extension line (Figure 8-30). The following is the prompt sequence for dimensioning a circle.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Select arc, circle, line, or <specify vertex>: *Select the circle at the point where you want the first extension line.*
Specify second angle endpoint: *Select the second point on or away from the circle.*
Specify dimension arc line location or [Mtext/Text/Angle]: *Select the location for the dimension line.*

Figure 8-29 *Angular dimensioning of arcs*

Figure 8-30 *Angular dimensioning of circles*

Angular Dimensioning Based on Three Points

If you press ENTER at the **Select arc, circle, line, or <specify vertex>** prompt, AutoCAD LT allows you to select three points to create an angular dimension. The first point is the vertex point, and the other two points are the first and second angle endpoints of the angle (Figure 8-31). The coordinate specifications of the first and the second angle endpoints must not be identical. However, the angle vertex coordinates and one of the angle endpoint coordinates can be identical. The following example illustrates angular dimensioning by defining three points.

Select arc, circle, line, or <specify vertex>: Enter
Specify angle vertex: *Specify the first point, vertex. This is the point where the two segments meet. If the two segments do not meet actually, use the* ***Apparent Intersection*** *object snap.*
Specify first angle endpoint: *Specify the second point. This point will be the origin of the first extension line.*
Specify second angle endpoint: *Specify the third point. This point will be the origin of the second extension line.*
Specify dimension arc line location or [Mtext/Text/Angle]: *Select the location for the dimension line.*

Note

If you use the ***DIMANGULAR*** *command, you cannot specify the text location. With the* ***DIMANGULAR*** *command, AutoCAD LT positions the dimensioning text automatically. If you want to manually define the position of the dimension text, use the* ***ANGULAR*** *option of the* ***DIM*** *command.*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-31 *Angular dimensioning for three points*

Exercise 3

Mechanical

Make the drawing shown in Figure 8-32 and then use **angular dimensioning to** dimension all angles of the part. The distance between the dotted lines is 0.5 units.

Figure 8-32 *Drawing for Exercise 3*

CREATING DIAMETER DIMENSIONS

Toolbar:	Dimension > Diameter
Menu:	Dimension > Diameter
Command:	DIMDIAMETER or DIMDIA

Diameter dimensioning is used to dimension a circle or an arc. Here, the measurement is done between two diametrically opposite points on the circumference of the circle or arc (Figure 8-33). The dimension text generated by AutoCAD LT commences with the Ø

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

symbol, to indicate a diameter dimension. The prompt sequence that will follow when you choose this button is given next.

Select arc or circle: *Select an arc or circle by selecting a point anywhere on its circumference.*
Dimension text = Current
Specify dimension line location or [Mtext/Text/Angle]: *Specify a point to position the dimension.*

Figure 8-33 *Diameter dimensioning*

If you want to override the default value of the dimension text, use the Mtext or Text option. The control sequence %%C is used to obtain the diameter symbol ø. It is followed by the dimension text that should appear in the diameter dimension. For example, if you want to write a text that displays a value ø20, then enter %%c20 at the text prompt.

Tip
The control sequence %%d can be used to generate the degree symbol " ° " (45°).

CREATING JOGGED DIMENSIONS

Toolbar:	Dimension > Jogged
Menu:	Dimension > Jogged
Command:	DIMJOGGED

The necessity of the jogged dimension is due to the space constraint in certain drawings and to avoid merging of the dimension line with other dimensions. Also, there are instances when it is not possible to show the center of the circle in the sheet. In such situations, the jogged dimensions are used, as shown in Figure 8-34. The jogged dimensions can be added using the **Jogged** tool. Note that you can add only jogged radius dimensions. To add jogged dimensions, choose the **Jogged** button from the **Dimension** toolbar and select an arc or a circle; you are now prompted to select the center location override. The center location override specifies the start point of the dimension line. Next, specify the dimension line location and the jog location. The command prompt when you invoke the **Jogged** tool is given next.

Figure 8-34 *Jogged dimensioning*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Select arc or circle: *Select arc or circle to be dimensioned*
Specify center location override: *specify a point which will currently override the actual center point location*
Dimension text = *Current*
Specify dimension line location or [Mtext/Text/Angle]: *Specify a point position dimension line*
Specify jog location: *Specify a point for the positioning of jog.*

CREATING RADIUS DIMENSIONS

Toolbar:	Dimension > Radius
Menu:	Dimension > Radius
Command:	DIMRADIUS or DIMRAD

Radius dimensioning is used to dimension a circle or an arc (Figure 8-35). Radius and diameter dimensioning are similar; the only difference is that instead of the diameter line, a radius line is drawn (half of the diameter line), which is measured from the center to any point on the circumference. The dimension text generated by AutoCAD LT is preceded by the letter **R** to indicate a radius dimension. If you want to use the default dimension text (dimension text generated automatically by AutoCAD), simply specify a point to position the dimension at the **Specify dimension line location or [Mtext/Text/Angle]** prompt. You can also enter a new value or specify a prefix or suffix, or suppress the entire text by entering a blank space following the **Enter dimension text <current>** prompt. A center mark for the circle/arc is drawn automatically, provided the center mark value controlled by the **DIMCEN** variable is not 0. The prompt sequence that will follow when you choose this button is given next.

Figure 8-35 *Radius dimensioning*

Select arc or circle: *Select the object you want to dimension.*
Dimension text = Current
Specify dimension line location or [Mtext/Text/Angle]: *Specify the dimension location.*

If you want to override the default value of the dimension text, use the **Text** or the **Mtext** option.

GENERATING CENTER MARKS AND CENTERLINES

Toolbar:	Dimension > Center Mark
Menu:	Dimension > Center Mark
Command:	DIMCENTER

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

When circles or arcs are dimensioned with the **DIMRADIUS or DIMDIAMETER** command, a small mark known as a center mark, or line known as centerline, may be drawn at the center of the circle/arc. Sometimes, you may want to mark the center of a circle or an arc without using these dimensioning commands. This can be achieved with the help of the **DIMCENTER** command. You can invoke this command by choosing the **Center Mark** button from the **Dimension** toolbar or by entering **CENTER** (or **CEN**) at the **Dim:** prompt. When you invoke this command, you will be prompted to select the arc or the circle. The result of this command will depend upon the value of the **DIMCEN** variable. If the value of this variable is positive, center marks are drawn, see Figure 8-36 and if the value is negative, centerlines are drawn, see Figure 8-37.

Figure 8-36 *Using a positive value for* ***DIMCEN***

Figure 8-37 *Using a negative value for* ***DIMCEN***

Note

The center marks created by ***DIMCENTER*** *or* ***DIM, CENTER*** *are lines, not associative dimensioning objects, and they have an explicit linetype.*

Exercise 4 *Mechanical*

Draw the model shown in Figure 8-38 and then use the radius and diameter dimensioning commands to dimension the part. Use the **DIMCENTER** command to draw the centerlines through the circles. The distance between the dotted lines is 0.5 units.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-38 *Figure for Exercise 4*

CREATING ORDINATE DIMENSIONS

Toolbar:	Dimension > Ordinate
Menu:	Dimension > Ordinate
Command:	DIMORDINATE or DIMORD

Ordinate dimensioning is used to dimension the *X* and *Y* coordinates of the selected point. This type of dimensioning is also known as **arrowless** dimensioning because no arrowheads are drawn in it. Ordinate dimensioning is also called **datum dimensioning** because all dimensions are related to a common base point. The current UCS (user coordinate system) origin becomes the reference or the base point for ordinate dimensioning. With ordinate dimensioning, you can determine the X or Y displacement of a selected point from the current UCS origin.

In ordinate dimensioning, AutoCAD LT automatically places the dimension text (*X* or *Y* coordinate value) and the leader line along the *X* or *Y* axis (Figure 8-39). Since ordinate dimensioning pertains to either the *X* coordinate or the *Y* coordinate, you should keep ORTHO on. When ORTHO is off, the leader line is automatically given a bend when you select the second leader line point that is offset from the first point. This allows you to generate offsets and avoid overlapping text on closely spaced dimensions. In ordinate dimensioning, only one extension line (leader line) is drawn.

The leader line for an *X* coordinate value will be drawn perpendicular to the *X* axis, and the leader line for a *Y* coordinate value will be drawn perpendicular to the *Y* axis. Since you cannot override this, the leader line drawn perpendicular to the *X* axis will have the dimension text aligned with the leader line. The dimension text is the X datum of the selected point. The leader line drawn perpendicular to the *Y* axis will have the dimension text, which is the Y datum of the selected point, aligned with the leader line. Any other alignment specification for the dimension text is nullified. Hence, changes in the Text Alignment in the **Dimension Style Manager** dialog box (**DIMTIH** and **DIMTOH** variables) have no effect on the alignment of the

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Figure 8-39 *Ordinate dimensioning*

dimension text. You can specify the coordinate value you want to dimension at the **Specify leader endpoint or [Xdatum/Ydatum/MText/Text/Angle]** prompt.

If you select or enter a point, AutoCAD LT checks the difference between the feature location and the leader endpoint. If the difference between the *X* coordinates is greater, the dimension measures the *Y* coordinate; otherwise, the *X* coordinate is measured. In this manner AutoCAD LT determines whether it is an X or Y type of ordinate dimension. However, if you enter Y instead of specifying a point, AutoCAD LT will dimension the *Y* coordinate of the selected feature. Similarly, if you enter X, AutoCAD LT will dimension the *X* coordinate of the selected point. The prompt sequence that follows when you choose this button is given next.

Specify feature location: *Select a point on an object*.
Specify leader endpoint or [Xdatum/Ydatum/Mtext/Text/Angle]: *Enter the endpoint of the leader.*

You can override the default text with the help of the **Mtext** or the **Text** option. If you use the **Mtext** option, the **In-Place Text Editor** will be displayed. If you use the **Text** option, you will be prompted to specify the new text in the Command line itself.

Exercise 5 *General*

Draw the model shown in Figure 8-40 and then use ordinate dimensioning to dimension the part. The distance between the dotted lines is 0.5 units.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-40 *Using the* ***DIMORDINATE*** *command to dimension the part*

WORKING WITH TRUE ASSOCIATIVE DIMENSIONS

The true associative dimensions are the dimensions that are automatically modified when the objects to which they are associated are modified. By default, all dimensions in AutoCAD LT are true associative dimensions. If the dimension attached to the object is true associative, then it will be modified automatically when the object is modified. In this case, you do not have to select the definition points of the dimensions. Any dimension in AutoCAD LT can be converted into a disassociated dimension and then back to the true associative dimension. This is discussed next.

Removing the Dimension Associativity

Command:	DIMDISASSOCIATE

The **DIMDISASSOCIATE** command is used to remove the associativity of the dimensions from the object to which they are associated. When you invoke this command, you will be prompted to select the dimensions to be disassociated. The true association of the selected dimensions is automatically removed once you exit this command. The number of dimensions disassociated is displayed in the Command prompt.

Converting a Dimension into a True Associative Dimension

Menu:	Dimension > Reassociate Dimensions
Command:	DIMREASSOCIATE

The **DIMREASSOCIATE** command is used to create a true associative dimension by associating the selected dimension to the specified object. When you invoke this command, you will be prompted to select the objects. These objects are the dimensions to be associated. Once you

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

select the dimensions to be associated, a cross is displayed and you are prompted to select the feature location. This cross implies that the dimension is not associated. You can define a new association point for the dimensions by selecting the objects or by using the object snaps. If you select a dimension that has already been associated to an object, the cross will be displayed inside a box. The prompt sequence that is displayed varies depending upon the type of dimension selected. In case of **linear**, **aligned**, **radius**, and **diameter** dimensions, you can directly select the object to associate the dimension. If the arcs or circles are assigned angular dimensions using three points, then also you can select these arcs or circles directly for associating the dimensions. For rest of the dimension types, you can use the object snaps to specify the point to associate the dimensions.

DRAWING LEADERS

Toolbar:	Dimension > Quick Leader
Menu:	Dimension > Leader
Command:	QLEADER

The leader line is used to attach annotations to an object or when the user wants to show a dimension without using another dimensioning command. Sometimes, leaders of the dimensions of circles or arcs are so complicated that you need to construct a leader of your own. The leaders can be created using the **QLEADER** command. The leaders drawn by using this command create the arrow and the leader lines as a single object. The text is created as a separate object. This command can create multiline annotations and offer several options such as copying existing annotations and so on. You can customize the leader and annotation by selecting the **Settings** option at the **Specify first leader point, or [Settings]<Settings>** prompt. The prompt sequence that will follow when you choose this button is given next.

Specify first leader point, or [Settings]<Settings>: *Specify the start point of the leader.*
Specify next point: *Specify endpoint of the leader.*
Specify next point: *Specify next point*.
Specify text width <current>: *Enter text width of multiline text.*
Enter first line of annotation text <MText>: *Press ENTER; AutoCAD LT displays the* ***In-Place Text Editor*** *dialog box. Enter text in the dialog box and then choose* ***OK*** *to exit.*

If you press ENTER at the **Specify first leader point, or [Settings]<Settings>** prompt then the **Leader Settings** dialog box is displayed. The **Leader Settings** dialog box gives you a number of options for the leader line and the text attached to it. It has the following tabs.

Annotation Tab

This tab provides you with various options to control annotation features, see Figure 8-41.

Annotation Type Area

Mtext. When selected, AutoCAD LT uses the **In-Place Text Editor** to create annotation. When you select this radio button, the options in the **Mtext options** area are available.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-41 *The* ***Annotation*** *tab of the* ***Leader Settings*** *dialog box*

Copy an Object. This option allows you to copy an existing annotation object (like multiline text, single line text, tolerance, or block) and attach it at the end of the leader. For example, if you have a text string in the drawing that you want to place at the end of the leader, you can use the **Copy an Object** option to place it at the end of the leader.

Tolerance. When you select the **Tolerance** option, AutoCAD LT displays the **Geometric Tolerance** dialog box on the screen. Specify the tolerance in the dialog box and choose **OK** to exit. AutoCAD LT will place the specified Geometric Tolerance with the feature control frame at the end of the leader (Figure 8-42).

Block Reference. The **Block Reference** option allows you to insert a predefined block at the end of the leader. When you select this option, AutoCAD LT will prompt you to enter the block name and insertion point.

Figure 8-42 *Leader with MText, Block, Tolerance, and MText with Frame annotations*

None. This option creates a leader without placing any annotation at the end of the leader.

MText options Area

The options under this area will be available only if the **MText** radio button is selected from the **Annotation Type** area. This area provides you with the following options.

Prompt for width. Selecting this check box allows you to specify the width of the multiline text annotation.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Always left justify. This option left justifies the multiline text annotation in all situations. Selecting this check box makes the Prompt for the width option unavailable.

Frame text. Selecting this check box draws a box around the multiline text annotation.

Annotation Reuse Area

The options under this area allow you to reuse the annotation.

None. When selected, AutoCAD LT does not reuse the leader annotation.

Reuse Next. This option allows you to reuse the annotation that you are going to create next for all subsequent leaders.

Reuse Current. This option allows you to reuse the current annotation for all subsequent leaders.

Leader Line & Arrow Tab

The options under this area are related to the leader parameters, see Figure 8-43.

Figure 8-43 *The* ***Leader Settings*** *dialog box,* ***Leader Line & Arrow*** *tab*

Leader Line Area

This area gives the options for the **leader line type** such as straight or spline. The **Spline** option draws a spline through the specified leader points and the **Straight** option draws straight lines. Figure 8-44 shows straight and spline leader lines.

Number of Points Area

No Limit. If this check box is selected, you can define as many number of points as you want in

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

the leader line. AutoCAD LT will keep prompting you for the next point until you press ENTER at this prompt.

Maximum. This spinner is used to specify the maximum number of points on the leader line. The default value is 3, which means that by default, there will be only three points in the leader line. You can specify the maximum number of points using this spinner. This has to be kept in mind that the start point of the leader is the first leader point. This spinner will be available only if the **No Limit** check box is cleared. Figure 8-44 shows a leader line with five number of points.

Figure 8-44 *Splined and straight leaders*

Arrowhead Area

The drop-down list under this area allows you to define a leader arrowhead. The arrowhead is the same as the one for dimensioning. You can also use the user-defined arrows by selecting **User Arrow** from the drop-down list.

Angle Constraints Area

The options provided under this area are used to define the angle for the segments of the leader lines.

First Segment. This drop-down list is used to specify the angle at which the first leader line segment will be drawn. You can select the predefined values from this drop-down list.

Second Segment. This drop-down list is used to specify the angle at which the second leader line segment will be drawn.

Attachment Tab

The **Attachment** tab (Figure 8-45) will be available only if you have selected **MText** from the **Annotation Type** area of the **Annotation** tab. The options provided under this tab are used for attaching the multiline text to the leader. It has two columns: **Text on left side** and **Text on right side**. Both these columns have five radio buttons below them. Each radio button corresponds to the option of attaching the multiline text. If you draw a leader from the right to the left, AutoCAD LT uses the settings under **Text on left side**. Similarly, if you draw a leader from the left to the right, AutoCAD LT uses the settings as specified under **Text on right side**. This area also provides you with the **Underline bottom line** check box. If this check box is selected, then the last line of the multiline text will be underlined.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Figure 8-45 *The* ***Leader Settings*** *dialog box, Attachment tab*

Exercise 6

Mechanical

Make the drawing shown in Figure 8-46 and then use the **QLEADER** command to dimension the part, as shown. The distance between the dotted lines is 0.5 units.

Figure 8-46 *Drawing for Exercise 6*

Note

You can use the Command line to create the leaders with the help of the ***LEADER*** *command. The options under this command are similar to those under the* ***QLEADER*** *command. The only difference is that the* ***LEADER*** *command uses the Command line.*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

USING LEADER WITH THE DIM COMMAND

You can also draw a leader by using the **Leader** option of the **DIM** command. The **DIM**, **LEADER** command creates the arrow, leader lines, and the text as separate objects. This command has the feature of defaulting to the most recently measured dimension. Once you invoke the **DIM**, **LEADER** command and specify the first point, the prompt sequence is similar to that of the **LINE** command. The start point of the leader should be specified at the point closest to the object being dimensioned. After drawing the leader, enter a new dimension text or keep the default one.

Command: **DIM**
Leader (or **L**)
Leader start: *Specify the starting point of the leader.*
To point: *Specify the endpoint of the leader.*
To point: *Specify the next point.*
To point: Enter
Dimension text <current>: *Enter dimension text.*

The value between the angle brackets is the current value, that is, the measurement of the most recently dimensioned object. If you want to retain the default text, press ENTER at the **Dimension text <current>** prompt. You can enter text of your choice, specify a prefix/suffix, or suppress the text. The text can be suppressed by pressing the SPACEBAR, and then pressing ENTER at the **Dimension text <current>** prompt. An arrow is drawn at the start point of the leader segment if the length of the segment is greater than two arrow lengths. If the length of the line segment is less than or equal to two arrow lengths, only a line is drawn.

GEOMETRIC DIMENSIONING AND TOLERANCING

One of the most important parts of the design process is giving the dimensions and tolerances, since every part is manufactured from the dimensions given in the drawing. Therefore, every designer must understand and have a thorough knowledge of the standard practices used in the industry to make sure that the information given on the drawing is correct and can be understood by other people. Tolerancing is equally important, especially in the assembled parts. Tolerances and fits determine how the parts will fit. Incorrect tolerances could result in a product that is not usable. In addition to dimensioning and tolerancing, the function and the relationship that exists between the mating parts is important if the part is to perform the way it was designed. This aspect of the design process is addressed by **geometric dimensioning and tolerancing**, generally known as **GDT**.

Geometric dimensioning and tolerancing is a means to design and manufacture parts with respect to the actual function and relationship that exists between different features of the same part or the features of the mating parts. Therefore, a good design is not achieved by just giving dimensions and tolerances. The designer has to go beyond dimensioning and think of the intended function of the part and how the features of the part are going to affect its function. For example, Figure 8-47 shows a part with the required dimensions and tolerances. In this drawing, there is no mention of the relationship between the pin and the plate. Is the pin perpendicular to the plate? If it is, to what degree should it be perpendicular? Also, it does not mention on

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

which surface the perpendicularity of the pin is to be measured. A design like this is open to individual interpretation based on intuition and experience. This is where geometric dimensioning and tolerancing play an important part in the product design process.

Figure 8-48 has been dimensioned using geometric dimensioning and tolerancing. The feature symbols define the datum (reference plane) and the permissible deviation in the perpendicularity of the pin with respect to the bottom surface. From a drawing like this, chances of making a mistake are minimized.

Figure 8-47 *Traditional dimensioning and tolerancing technique*

Figure 8-48 *Geometric dimensioning and tolerancing*

GEOMETRIC CHARACTERISTICS AND SYMBOLS

Before discussing the application of AutoCAD LT commands in geometric dimensioning and tolerancing, you need to understand the following feature symbols and tolerancing components. Figure 8-49 shows the geometric characteristics and symbols used in geometric dimensioning and tolerancing.

Kind of feature	Type of feature	Characteristics	
Related	Location	Position	⌖
		Concentricity or Coaxiality	◎
		Symmetry	⌯
	Orientation	Parallelism	//
		Perpendicularity	⊥
		Angularity	∠
Individual	Form	Cylindricity	⌭
		Flatness	▱
		Circularity or Roundness	○
		Straightness	—
Individual or related	Profile	Surface Profile	⌓
		Line Profile	⌒
Related	Runout	Circular Runout	↗
		Total Runout	⌰

Figure 8-49 *Characteristics and symbols used in GTOL*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Note
These symbols are the building blocks of geometric dimensioning and tolerancing.

ADDING GEOMETRIC TOLERANCE

Toolbar:	Dimension > Tolerance
Menu:	Dimension > Tolerance
Command:	TOLERANCE

Geometric tolerance displays the deviations of profile, orientation, form, location, and runout of a feature. In AutoCAD, geometrical tolerancing is displayed by feature control frames. The frames contain all the information about tolerances for a single dimension. To display feature control frames with the various tolerancing parameters, the specifications are entered in the **Geometric Tolerance** dialog box (Figure 8-50).

Figure 8-50 *The* ***Geometric Tolerance*** *dialog box*

The various components that constitute the GTOL are shown in Figures 8-51 and 8-52.

Figure 8-51 *Components of GTOL*

Figure 8-52 *Components of GTOL*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Feature Control Frame

The **feature control frame** is a rectangular box that contains the geometric characteristics symbols and tolerance definition. The box is automatically drawn to standard specifications; you do not need to specify its size. You can copy, move, erase, rotate, and scale the feature control frame. You can also snap to them using various Object snap modes. You can edit feature control frames using the **DDEDIT** command or you can also edit them using **GRIPS**. The system variable **DIMCLRD** controls the color of the feature control frame. The system variable **DIMGAP** controls the gap between the feature control frame and the text.

Geometric Characteristics Symbol

The geometric characteristics symbols indicate the characteristics of a feature like straightness, flatness, perpendicularity, and so on. You can select the required symbol from the **Symbol** dialog box (Figure 8-53). This dialog box is displayed by selecting the box provided in the **Sym** area of the **Geometric Tolerance** dialog box. To select the required symbol, just pick the symbol using the left mouse button. The symbol will now be displayed in the box under the **Sym** area.

Figure 8-53 *The* ***Symbol*** *dialog box*

Tolerance Value and Tolerance Zone Descriptor

The tolerance value specifies the tolerance on the feature as indicated by the tolerance zone descriptor. For example, a value of .003 indicates that the feature must be within a 0.003 tolerance zone. Similarly, ϕ.003 indicates that this feature must be located at a true position within a 0.003 diameter. The tolerance value can be entered in the edit box provided under the **Tolerance 1** or the **Tolerance 2** area of the **Geometric Tolerance** dialog box. The tolerance zone descriptor can be invoked by selecting the box located to the left of the edit box. The system variable **DIMCLRT** controls the color of the tolerance text, variable **DIMTXT** controls the tolerance text size, and variable **DIMTXSTY** controls the style of the tolerance text. Using the **Projected Tolerance Zone**, inserts a projected tolerance zone symbol, which is an encircled P, after the projected tolerance zone value.

Material Condition Modifier

The **material condition modifier** specifies the material condition when the tolerance value takes effect. For example, ϕ.003(M) indicates that this feature must be located at a true position within a 0.003 diameter at maximum material condition (MMC). The material condition modifier symbol can be selected from the **Material Condition** dialog box (Figure 8-54). This dialog box can be invoked by selecting the boxes located on the right side of the edit boxes under the **Tolerance 1**, **Tolerance 2**, **Datum 1**, **Datum 2**, and **Datum 3** areas of the **Geometric Tolerance** dialog box.

Figure 8-54 *The* ***Material Condition*** *dialog box*

Datum

The datum is the origin, surface, or feature from which the measurements are made. The datum

is also used to establish the geometric characteristics of a feature. The datum feature symbol consists of a reference character enclosed in a feature control frame. You can create the datum feature symbol by entering characters (like -A-) in the **Datum Identifier** edit box in the **Geometric Tolerance** dialog box and then selecting a point where you want to establish this datum.

You can also combine datum references with geometric characteristics. AutoCAD LT automatically positions the datum references on the right end of the feature control frame.

Example 2 *Mechanical*

In the following example, you will create a feature control frame to define perpendicularity specification, see Figure 8-55.

1. Chose the **Tolerance** button from the **Dimension** toolbar to display the **Geometric Tolerance** dialog box. Choose the upper box from the **Sym** area to display the **Symbol** dialog box. Select the **perpendicularity** symbol. It will now be displayed in the **Sym** area.

2. Select the box on the left of the upper edit box under the **Tolerance 1** area. A diameter symbol will appear to denote a cylindrical tolerance zone.

Figure 8-55 *Drawing for Example 2*

3. Enter **0.005** in the upper edit box under the **Tolerance 1** area.

4. Enter **A** in the edit box under the **Datum 1** area. Choose the **OK** button to accept the changes made in the **Geometric Tolerance** dialog box.

5. The **Enter tolerance location** prompt is displayed in the Command line area and the **Feature Control Frame** is attached to the cursor at its middle left point. Select a point to insert the frame.

6. To place the datum symbol, use the **TOLERANCE** command to display the **Geometric Tolerance** dialog box. In the **Datum Identifier** edit box, enter **A**.

7. Choose the **OK** button to accept the changes to the **Geometric Tolerance** dialog box, and then select a point to insert the frame.

COMPLEX FEATURE CONTROL FRAMES

Combining Geometric Characteristics

Sometimes, it is not possible to specify all geometric characteristics in one frame. For example, Figure 8-56 shows the drawing of a plate with a hole in the center.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Figure 8-56 *Combining feature control frames*

In this part, it is determined that surface C must be perpendicular to surfaces A and B within 0.002 and 0.004, respectively. Therefore, we need two frames to specify the geometric characteristics of surface C. The first frame specifies the allowable deviation in perpendicularity of surface C with respect to surface A. The second frame specifies the allowable deviation in perpendicularity of surface C with respect to surface B. In addition to these two frames, we need a third frame that identifies datum surface C.

All the three feature control frames can be defined in one instance of the **TOLERANCE** command.

1. Choose the **Tolerance** button to invoke the **Geometric Tolerance** dialog box. Select the box under the **Sym** area to display the **Symbol** dialog box. Select the **perpendicular** symbol. AutoCAD LT will display the selected symbol in the first row of the **Sym** area.

2. Enter **0.002** in the first row edit box under the **Tolerance 1** area and enter **A** in the first row edit box under the **Datum 1** area.

3. Select the second row box under the **Sym** area to display the **Symbol** dialog box. Select the **perpendicular** symbol. AutoCAD LT will display the selected symbol in the second row box of the **Sym** area.

4. Enter **.004** in the second row edit box under the **Tolerance 1** area and enter **B** in the second row edit box under the **Datum 1** area.

5. In the **Datum Identifier** edit box enter **C**, and then choose the **OK** button to exit the dialog box.

6. In the graphics screen, select the position to place the frame.

7. Similarly, create the remaining feature control frames.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Composite Position Tolerancing

Sometimes the accuracy required within a pattern is more important than the location of the pattern with respect to the datum surfaces. To specify such a condition, composite position tolerancing may be used. For example, Figure 8-57 shows four holes (pattern) of diameter 0.15. The design allows a maximum tolerance of 0.025 with respect to datums A, B, and C at the maximum material condition (holes are smallest). The designer wants to maintain a closer positional tolerance (0.010 at MMC) between the holes within the pattern. To specify this requirement, the designer must insert the second frame. This is generally known as composite position tolerancing. AutoCAD LT provides the facility to create two composite position tolerance frames by means of the **Geometric Tolerance** dialog box. The composite tolerance frames can be created as follows.

Figure 8-57 *Composite position tolerancing*

1. Invoke the **TOLERANCE** command to display the **Geometric Tolerance** dialog box. Select the box under the **Sym** area to display the **Symbol** dialog box. Select the **position** symbol. AutoCAD LT will display the selected symbol in the first row of the **Sym** area.

2. In the first row of the **Geometric Tolerance** dialog box, enter the geometric characteristics and the datum references required for the first position tolerance frame.

3. In the second row of the **Geometric Tolerance** dialog box, enter the geometric characteristics and the datum references required for the second position tolerance frame.

4. When you have finished entering the values, choose the **OK** button in the **Geometric Tolerance** dialog box, and then select the point where you want to insert the frames. AutoCAD LT will create the two frames and automatically align them with the common position symbol, as shown in Figure 8-58.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

USING FEATURE CONTROL FRAMES WITH LEADERS

The **Leader Settings** dialog box invoked using the **QLEADER** command has the Tolerance option that allows you to create the feature control frame and attach it to the end of the leader extension line, see Figure 8-58. The following is the prompt sequence for using the **QLEADER** command with the Tolerance option:

Specify first leader point, or [Settings] <Settings>: *Press ENTER to display the **Leader Settings** dialog box. Choose the **Annotation** tab. Now, select the **Tolerance** radio button from the **Annotation Type** area. Choose **OK**.*
Specify first leader point, or [Settings] <Settings>: *Specify the start point of the leader line.*
Specify next point: *Specify the second point.*
Specify next point: *Specify the third point to display the **Geometric Tolerance** dialog box.*

Figure 8-58 *Using the feature control frame with leaders*

PROJECTED TOLERANCE ZONE

Figure 8-60 shows two parts joined with a bolt. The lower part is threaded, and the top part has a drilled hole. When these two parts are joined, the bolt that is threaded in the lower part will have the orientation error that exists in the threaded hole. In other words, the error in the threaded hole will extend beyond the part thickness, which might cause interference, and the parts may not assemble. To avoid this problem, projected tolerance is used. The projected tolerance establishes a tolerance zone that extends above the surface. In Figure 8-59, the position tolerance for the threaded hole is 0.010, which extends 0.1 above the surface (datum A). By using the projected tolerance, you can ensure that the bolt is within the tolerance zone up to the specified distance.

You can use the AutoCAD LT GDT feature to create feature control frames for the projected tolerance zone as follows:

1. Invoke the **QLEADER** command and then press ENTER at the **Specify first leader point, or [Settings] <Settings>** prompt to display the **Leader Settings** dialog box.

2. Choose the **Annotation** tab and then select the **Tolerance** radio button from the **Annotation Type** area. Choose **OK** to return to the Command line. Specify the first, second, and the third leader point, as shown in Figure 8-59. As soon as you specify the third point, the **Geometric Tolerance** dialog box will be displayed. Choose the position symbol from the **Symbol** dialog box.

3. In the first row of the **Geometric Tolerance** dialog box, enter the geometric characteristics and the datum references required for the first position tolerance frame.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-59 *Projected tolerance zone*

4. In the **Height** edit box, enter the height of the tolerance zone (0.1 for the given drawing) and select the box on the right of Projected Tolerance Zone. The projected tolerance zone symbol will be displayed in the box.

5. Choose the **OK** button in the **Geometric Tolerance** dialog box. AutoCAD LT will create the two frames and automatically align them, as shown in Figure 8-57.

6. Similarly, attach the datum symbol also.

Example 3 *Mechanical*

In the following example, you will create a leader with a combination feature control frame to control runout and cylindricity, see Figure 8-60.

1. Choose the **Quick Leader** button from the **Dimension** toolbar to invoke the **QLEADER** command. The prompt sequence is as follows.

   Specify first leader point, or [Settings] <Settings>: *Press ENTER to display the **Leader Settings** dialog box. Choose the **Annotation** tab. Select the **Tolerance** radio button from the **Annotation Type** area. Choose **OK**.*
   Specify first leader point, or [Settings] <Settings>: *Specify the leader start point as shown in Figure 8-60.*
   Specify next point: *Specify the second point of the leader.*
   Specify next point: *Specify the third point of the leader line to display the **Tolerance** dialog box.*

Figure 8-60 *Drawing for Example 3*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

2. Choose the runout symbol from the **Symbol** dialog box. The **runout** symbol will be displayed on the first row of the **Sym** area. Enter **0.15** in the first row edit box under the Tolerance 1 area.
3. Enter **C** in the edit box under the **Datum 1** area.
4. Select the edit box on the second row of the **Sym** area and select the **cylindricity** symbol. The **cylindricity** symbol will be displayed in the second row of the **Sym** area.
5. Enter **0.05** in the second row edit box of the Tolerance 1 area.
6. Enter **C** in the **Datum Identifier** edit box.
7. Choose the **OK** button to accept the changes to the **Geometric Tolerance** dialog box. The control frames will be automatically attached at the end of the leader.

Self-Evaluation Test

Answer the following questions, and then compare your answers to those given at the end of this chapter.

1. You can specify dimension text of your own or accept the measured value computed by AutoCAD LT. (T/F)
2. The rotated dimension can be specified to get the effect of horizontal or vertical dimension. (T/F)
3. The center point of an arc/circle is taken as the vertex angle in the angular dimensioning of the arc/circle. (T/F)
4. You cannot combine GTOL with the leaders. (T/F)
5. The _____ command is used to dimension only the *X* coordinates of the selected object.
6. _____ symbols are the building blocks of geometric dimensioning and tolerancing.
7. In the _____ dimensions, the dimension text is, by default, aligned with the object being dimensioned.
8. The dimensions that are automatically updated when the object to which they are assigned change are called _____.
9. The _____ point is taken as the vertex point of the angular dimensions while dimensioning an arc or circle.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

10. The feature control frame is _____ in shape.

Review Questions

Answer the following questions.

1. Only inner angles (acute angles) can be dimensioned with angular dimensioning. (T/F)

2. In addition to the most recently drawn dimension (the default base dimension), you can use any other linear dimension as the base dimension. (T/F)

3. In continued dimensions, the base for successive continued dimensions is the base dimension's first extension line. (T/F)

4. In rotated dimensioning, the dimension line is always aligned with the object being dimensioned. (T/F)

5. Using which command can the dimensions be converted into true associative dimensions?

   (a) **DIMREASSOCIATE** (b) **DIMASSOCIATE**
   (c) **DIMDISASSOCIATE** (d) **None**

6. Which command can be used to dimension the arc length?

   (a) **DIMLINEAR** (b) **DIMANGULAR**
   (c) **DIMARC** (d) **None**

7. Using which command can you dimension different points and features of a part with reference to a fixed point?

   (a) **DIMBASELINE** (b) **DIMANGULAR**
   (c) **DIMRADIUS** (d) **DIMALIGNED**

8. Using which command can you add geometric dimensions and tolerance to the current drawing?

   (a) **GTOL** (b) **TOLERANCE**
   (c) **TEXT** (d) **None**

9. Which option of the **QLEADER** command allows you to make the settings for attaching tolerances to the leader?

   (a) **Settings** (b) **Set**
   (c) **Undo** (d) **None**

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

10. Geometric dimensioning and tolerancing is generally known as _____.

11. Give three examples of geometric characteristics that indicate the characteristics of a feature. _____.

12. The three ways to return to the Command prompt from the **Dim:** prompt (dimensioning mode) are _____, _____, and _____.

13. The seven fundamental dimensioning types provided by AutoCAD LT are _____ _____.

14. Horizontal dimensions measure displacement along the _____.

15. Vertical dimensions measure displacement along the _____.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Exercises

Exercise 7 — *Mechanical*

Draw the object shown in Figure 8-61 and then dimension it. Save the drawing as a **DIMEXR7** drawing file.

Figure 8-61 *Drawing for Exercise 7*

Exercise 8 — *Mechanical*

Draw and dimension the object shown in Figure 8-62. Save the drawing as **DIMEXR8**.

Figure 8-62 *Drawing for Exercise 8*

Exercise 9 *Mechanical*

Draw the object shown in Figure 8-63 and then dimension it. Save the drawing as **DIMEXR9**.

Figure 8-63 *Drawing for Exercise 9*

Exercise 10 *Mechanical*

Draw the object shown in Figure 8-64 and then dimension it. Save the drawing as **DIMEXR10**.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Figure 8-64 *Drawing for Exercise 10*

Problem Solving Exercise 1

Mechanical

Draw Figure 8-65 and dimension it, as shown in the drawing. The **FILLET** command should be used where needed. Save the drawing as **DIMPSE1**.

Figure 8-65 *Drawing for Problem Solving Exercise 1*

Problem Solving Exercise 2

Mechanical

Draw Figure 8-66 and then dimension it, as shown in the drawing. Save the drawing as **DIMPSE2**.

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Figure 8-66 *Drawing for Problem Solving Exercise 2*

Problem Solving Exercise 3 — *Mechanical*

Draw Figure 8-67 and then dimension it, as shown in the drawing. Save the drawing as **DIMPSE3**.

Figure 8-67 *Drawing for Problem Solving Exercise 3*

Problem Solving Exercise 4 — *Mechanical*

Draw the three orthographic views of an object, as shown in Figure 8-68 and then dimension it, as shown in the drawing. Save the drawing as **DIMPSE4**.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Figure 8-68 *Drawing for Problem Solving Exercise 4*

© CADCIM Technologies, USA. For online training, contact sales@cadcim.com

Answers to Self-Evaluation Test

1 - T, **2** - T, **3** - T, **4** - F, **5** -**DIMORDINATE**, **6** - geometric characteristics, **7** - aligned, **8** - true associative dimensions, **9** - center, **10** - rectangular