Chapter 10

Dimension Styles and Dimensioning System Variables

Learning Objectives

After completing this chapter, you will be able to:

- *Use styles and variables to control dimensions.*
- *Create dimensioning styles.*
- *Set dimension variables using the various tabs of the* ***New****,* ***Modify****, and* ***Override Dimension Style*** *dialog boxes.*
- *Set other dimension variables that are not in dialog boxes.*
- *Understand dimension style families and how to apply them in dimensioning.*
- *Use dimension style overrides.*
- *Compare and list dimension styles.*
- *Import externally referenced dimension styles.*

USING STYLES AND VARIABLES TO CONTROL DIMENSIONS

In AutoCAD LT, the appearance of dimensions on the drawing screen and the manner in which they are saved in the drawing database are controlled by a set of dimension variables. The dimensioning commands use these variables as arguments. The variables that control the appearance of the dimensions can be managed with dimension styles. You can use the **Dimension Style Manager** dialog box to control the dimension styles and dimension variables through a set of dialog boxes.

CREATING AND RESTORING DIMENSION STYLES

Toolbar:	Dimension > Dimension Style
Menu:	Dimension > Dimension Style or Format > Dimension style
Command:	DIMSTYLE

The dimension styles control the appearance and positioning of dimensions and leaders in the drawing. If the default dimensioning style (STANDARD) does not meet your requirements, you can select another existing dimensioning style or create a new one that does. The default dimension style file name is **STANDARD**. Dimension styles can be created by using the **Dimension Style Manager** dialog box. Choose **Format > Dimension Style** or **Dimension > Dimension Style** from the menu bar to invoke the **Dimension Style Manager** dialog box (Figure 10-1).

*Figure 10-1 The **Dimension Style Manager** dialog box*

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In the **Dimension Style Manager** dialog box, choose the **New** button to display the **Create New Dimension Style** dialog box (Figure 10-2). Enter the dimension style name in the **New Style Name** text box and then select a style on which you want to base your style from the **Start With** drop-down list. The **Use for** drop-down list allows you to select the dimension type to which you want to apply the new dimension style. For example if you wish to use the new style for only the diameter dimension, you can select **Diameter dimension** from the **Use for** drop-down list. Choose the **Continue** button to display the **New Dimension Style** dialog box where you can define the new style.

Figure 10-2 *The* ***Create New Dimension Style*** *dialog box*

In the **Dimension Style Manager** dialog box, the current dimension style name is shown in front of **Current dimstyle** and is also shown highlighted in the **Styles** list box. A brief description of the current style (its differences from the default settings) is also displayed in the **Description** area. The **Dimension Style Manager** dialog box also has a **Preview o**f the window that displays a preview of the current dimension style. A style can be made current (restored) by selecting the name of the dimension style you want to make current from the list of defined dimension styles and choosing the **Set Current** button. You can also make a style current by double-clicking on the style name in the **Styles** list box. The drop-down list in the **Dimension** toolbar also displays the dimension styles. Selecting a dimension style from this list also sets it current. The list of dimension styles displayed in the **Styles** list box is dependent on the option selected from the **List** drop down-list. If you select the **Styles in use** option, only the dimension styles in use will be listed in the **Style** list box. If you right-click on a style in the **Style** list box, a shortcut menu is displayed that provides you with the options to **Set current**, **Rename**, or **Delete** a dimension style. Selecting the **Don't list styles in Xrefs** check box does not list the names of Xref styles in the **Styles** list box. Choosing the **Modify** button displays the **Modify Dimension Style** dialog box where you can modify an existing style. Choosing the **Override** button displays the **Override Current Style** dialog box where you can define overrides to an existing style (discussed later in this chapter). Both these dialog boxes along with the **New Dimension Style** dialog box have identical properties. Choosing the **Compare** button displays the **Compare Dimension Styles** dialog box (also discussed later in this chapter) that allows you to compare two existing styles.

NEW DIMENSION STYLE DIALOG BOX

The **New Dimension Style** dialog box can be used to specify the dimensioning attributes (variables) that affect the various properties of the dimensions. The various tabs provided under the **New Dimension Style** dialog box are discussed next.

Lines Tab

The options in the **Lines** tab (Figure 10-3) of the **New Dimension Style** dialog box are used to specify the dimensioning attributes (variables) that affect the format of the dimension lines . For example, the appearance and behavior of the dimension lines and extension lines can be changed

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Figure 10-3 *The* ***Lines*** *tab of the* ***New Dimension Style*** *dialog box*

with this tab. If the settings of the dimension variables have not been altered in the current editing session, the settings displayed in the dialog box are the default settings.

Dimension Lines Area

This area provides you with the options of controlling the display of the dimension lines and leader lines. These options are discussed next.

Color. This drop-down list is used to set the colors for the dimension lines and arrowheads. Its dimension arrowheads have the same color as the dimension line because arrows constitute a part of the dimension line. The color you set here will also be assigned to the leader lines and arrows. The default color for the dimension lines and arrows is BYBLOCK. You can specify the color of the dimension line by selecting it from the **Color** drop-down list. You can also select **Other** from the **Color** drop-down list to display the **Select Color** dialog box where you can choose a specific color. The color number or the special color label is stored in the **DIMCLRD** variable.

Linetype. This drop-down list is used to set the linetype for the dimension lines.

Lineweight. This drop-down list is used to specify the lineweight for the dimension line. You can select the required lineweight by selecting it from this drop-down list. This value is also stored in the **DIMLWD** variable. The default value is BYBLOCK. Keep in mind that you cannot assign the lineweight to the arrowheads using this drop-down list.

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Extend beyond ticks. The **Extend beyond ticks** (Oblique tick extension) spinner will be available only when you select the oblique, Architectural tick, or any such arrowhead type in the **1st** and **2nd** drop-down lists in the **Arrowheads** area. This spinner is used to specify the distance by which the dimension line will extend beyond the extension line. The extension value, entered in the **Extend beyond ticks** edit box, gets stored in the **DIMDLE** variable. By default, this edit box is disabled because the oblique arrowhead type is not selected.

Baseline spacing. The **Baseline spacing** (baseline increment) spinner is used to control the spacing between successive dimension lines drawn using the baseline dimensioning, see Figure 10-4. You can specify the dimension line increment to your requirement by specifying the desired value using the **Baseline spacing** spinner. The default value displayed in the **Baseline spacing** spinner is 0.38 units. This spacing value is also stored in the **DIMDLI** variable.

Figure 10-4 *Baseline increment*

Suppress. The **Suppress** check boxes control the display of the first and second dimension lines. By default, both dimension lines will be drawn. You can suppress one or both the dimension lines by selecting their corresponding check boxes. The values of these check boxes are stored in the **DIMSD1** and **DIMSD2** variables.

Note

The first and second dimension lines are determined by how you select the extension line origins. If the first extension line origin is on the right, the first dimension line is also on the right.

Extension Lines Area

Color. This drop-down list is used to control the color of the extension lines. The default extension line color is BYBLOCK. You can assign a new color to the extension lines by selecting it from this drop-down list. The color number or the color label is saved in the **DIMCLRE** variable.

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Linetype ext line 1. The options in this drop-down list are used to specify the linetype of extension line 1. By default, the line type for the extension line 1 is set to BYBLOCK. You can change the linetype by selecting a new value of linetype from the drop-down list.

Linetype ext line 2. The options in this drop-down list are same as for the Linetype ext line 1. The options are used to specify the line type of extension line 2.

Note

Now you can specify different linetypes for the dimensions and the extensions lines. Use ***Linetype*** *drop-down list in* ***Dimension Lines*** *are to specify the linetype for dimensions. Use* ***Linetype ext line 1*** *and* ***Linetype ext line 2*** *drop-down list to specify the linetype for the extension lines.*

Lineweight. This drop-down list is used to modify the lineweight of the extension lines. The default value is BYBLOCK. You can change the lineweight value by selecting a new value from this drop-down list. The value for lineweight is stored in the **DIMLWE** variable.

Extend beyond dim lines. It is the distance by which the extension lines extend past the dimension lines, see Figure 10-5. You can change the extension line offset using the **Extend beyond dim lines** spinner. This value is also stored in the **DIMEXE** variable. The default value for the extension distance is 0.1800 units.

Offset from origin. It is the distance by which the extension line is offsetted from the point you specify as the origin of the extension line, see Figure 10-6. You may need to override this setting for specific dimensions when dimensioning curves and angled lines. You can specify an offset distance of your choice using this spinner. AutoCAD LT stores this value in the **DIMEXO** variable. The default value for this distance is 0.0625.

Figure 10-5 Extension beyond dimension lines

Figure 10-6 The origin offset

Fixed length extension lines. With the **Fixed length extension line** checkbox selected, you can specify a fixed length for the extension lines in the **Length** edit box. By default, the check box is cleared and the length spinner is not available. Once the checkbox is selected, you can specify the length of the extension line either by entering a numerical value in the **Length** edit box or by using a spinner.

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Suppress. The **Suppress** check boxes are used to control the display of the extension lines. By default, both extension lines will be drawn. You can suppress one or both of them by selecting the corresponding check boxes (Figure 10-7). The values of these check boxes are stored in the **DIMSE1** and **DIMSE2** variables.

Figure 10-7 *Suppressing the extension lines*

Note

The first and second extension lines are determined by how you select the extension line origins. If the first extension line origin is on the right, the first extension line is also on the right.

Symbols and Arrows Tab

The options in the **Symbols and Arrow** tab (Figure-10-8) of the **New Dimension Style** dialog box are used to specify the variables and attributes that affect the format of the symbols and arrows. You can change the appearance of symbols and arrows.

Arrowheads Area

First/Second. When you create a dimension, AutoCAD LT draws the terminator symbols at the two ends of the dimension line. These terminator symbols, generally referred to as **arrowheads**, represent the beginning and end of a dimension. AutoCAD LT has provided nineteen standard termination symbols you can apply at each end of the dimension line. In addition to these, you can create your own arrows or terminator symbols. By default, the same arrowhead type is applied at both ends of the dimension line. If you select the first arrowhead, it is automatically applied to the second end by default. However, if you want to specify a different arrowhead at the second dimension line endpoint, you must select the desired arrowhead type from the **2nd** drop-down list. The first endpoint of the dimension line is the intersection point of the first extension line and the dimension line. The first extension line is determined by the first extension line origin. However, in angular dimensioning the second endpoint is located in a counterclockwise direction from the first point, regardless of how the points were selected when creating the angular dimension. The specified arrowhead types are selected from the **1st** and **2nd** drop-down

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Figure 10-8 *The* ***Symbols and Arrows*** *tab of the* ***New Dimension Style*** *dialog box*

lists. The first arrowhead type is saved in the **DIMBLK1** system variable, and the second arrowhead type is saved in the **DIMBLK2** system variable.

AutoCAD LT provides you with an option of specifying a user-defined arrowhead. To define a user-defined arrow, you must create one as a block. (See Chapter 14 for information regarding blocks.) Now, from the **1st** or the **2nd** drop-down list, select **User Arrow**. The **Select Custom Arrow Block** dialog box will be displayed, see Figure 10-9.

Figure 10-9 *The* ***Select Custom Arrow Block*** *dialog box*

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All the blocks in the current drawing will be available in the **Select from Drawing Blocks** drop-down list. You can select the desired block and it will become the current arrowhead.

Creating an Arrowhead Block

1. To create a block for an arrowhead, you will use a 1 X 1 box, see Figure 10-10. AutoCAD LT automatically scales the block's X and Y scale factors to the arrowhead size multiplied by the overall scale. The **DIMASZ** variable controls the length of the arrowhead. For example, if **DIMASZ** is set to 0.25, the length of the arrow will be 0.25 units. Also, if the length of the arrow is not 1 unit, it will leave a gap between the dimension line and the arrowhead block.

2. The arrowhead must be drawn as it would appear on the right side of the dimension line. Choose the **Make Block** button from the **Draw** toolbar to convert it into a block.

3. The insertion point of the arrowhead block must be the point that will coincide with the extension line, see Figure 10-10.

Leader. The **Leader** drop-down list displays the arrowhead types for the Leader arrow. Here, also, you can either select the standard arrowheads from the drop-down list or select **User Arrow** that allows you to define and use a user-defined arrowhead type.

Arrow size. This spinner is used to define the size of the arrowhead, see Figure 10-11. The default value is 0.18 units, which is stored in the **DIMASZ** system variable.

Figure 10-10 *Creating user-defined arrows*

Figure 10-11 *Defining arrow sizes*

Center Marks Area

This area deals with the options that control the appearance of the center marks and centerlines in the radius and diameter dimensioning. However, keep in mind that the center marks or the centerlines will be drawn only when dimensions are placed outside the circle.

None. If you select **None radio button**, no mark or line will be drawn at the center of the circle.

Mark. If you select the **Mark** radio button, a mark will be drawn at the center of the circle.

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Line. If you select the **Line** radio button, the centerlines will be drawn at the center of the circles.

Note

*If you use the **DIMCEN** command, a positive value will create a center mark, and a negative value will create a centerline. If the value is 0, AutoCAD LT does not create center marks or centerlines.*

Size. The **Size** spinner is used to set the size of the center marks or centerlines. This value is stored in the **DIMCEN** variable. The default value of the **DIMCEN** variable is 0.09.

Arc Length Symbol Area

The radio buttons in this area are used to specify the position of arc when applying the arc length dimension.

Preceding dimension text. If you select the **Preceding dimension text** radio button, the arc symbol will appear before the dimension text while applying the arc length dimension.

Above dimension text. With the **Above dimension text** radio button selected, the arc symbol in the arc length dimension will appear above the dimension text.

None. Select the **None** radio button, if you don't want the arc symbol to appear with the arc length dimension.

Radius Dimension Jog

The **Jog angle** edit box is used to specify the angle of jog which appears while applying the jogged dimension. By default the value is 45-degrees.

Note

*Unlike specifying a negative value for the **DIMCEN** variable, you cannot enter a negative value in the **Size** spinner. Selecting **Line** from the **Type** drop-down list automatically treats the value in the **Size** spinner as the size for the centerlines and sets **DIMCEN** to the negative of the value shown.*

Exercise 1 *Mechanical*

Draw Figure 10-12 and then set the values in the **Lines and Arrows** tab of the **New Dimension Style** dialog box to dimension the drawing as shown in this figure. (Baseline spacing = 0.25, Extension beyond dimension line = 0.10, Offset from origin = 0.05, Arrowhead size= 0.09.) Assume the missing dimensions.

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Figure 10-12 *Drawing for Exercise 1*

CONTROLLING DIMENSION TEXT FORMAT

Text tab

You can control the dimension text format through the **Text** tab of the **New Dimension Style** dialog box (Figure 10-13). In the **Text** tab, you can control the parameters such as the placement, appearance, horizontal and vertical alignment of the dimension text, and so on. For example, you can force AutoCAD LT to align the dimension text along the dimension line. You can also force the dimension text to be displayed at the top of the dimension line. You can save the settings in a dimension style file for future use. The **New Dimension Style** dialog box has a **Preview** window that updates dynamically to display the text placement as the settings are changed. Individual items of the **Text** tab and the related dimension variables are described next.

Text Appearance Area

Text style. The **Text Style** drop-down list displays the names of the predefined text styles. From this list, you can select the style name that you want to use for dimensioning. You must define the text style before you can use it in dimensioning (see "**Creating Text Styles**" in Chapter 7). Choosing the [**...**] button displays the **Text Style** dialog box that allows you to create a new or modify an existing text style. The value of this setting is stored in the **DIMTXSTY** system variable. The change in the dimension text style does not affect the text style you are using to draw the other text in the drawing.

Text color. This drop-down list is used to modify the color of the dimension text. The default color is BYBLOCK. If you choose the **Other** option from the **Text Color** drop-down list, the **Select Color** dialog box is displayed, where you can choose a specific color. This color or color number is stored in the **DIMCLRT** variable.

Fill color. This drop-down list is used to set the fill color of the dimension text. A box of the selected color will be placed around the dimension text.

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Figure 10-13 *The* ***Text*** *tab of the* ***New Dimension Style*** *dialog box*

Text height. This spinner is used to modify the height of the dimension text, see Figure 10-14. You can change the dimension text height only when the current text style does not have a fixed height. In other words, the text height specified in the **STYLE** command should be zero. This is because a predefined text height (specified in the **STYLE** command) overrides any other setting for the dimension text height. This value is stored in the **DIMTXT** variable. The default text height is 0.1800 units.

Fraction height scale. This spinner is used to set the scale of the fractional units in relation to the dimension text height. This spinner will be available only when you select a format for the primary units, in which you can define the values in fractions, such as architectural or fractional. This value is stored in the **DIMTFAC** variable.

Draw frame around text. When selected, this check box draws a frame around the dimension text, see Figure 10-15. This value is stored as a negative value in the **DIMGAP** system variable.

Text Placement Area

Vertical. The **Vertical** drop-down list displays the options that control the vertical placement of the dimension text. The current setting is highlighted. Controlling the vertical placement of the dimension text is possible only when the dimension text is drawn in its normal (default) location. This setting is stored in the **DIMTAD** system variable. The options in this drop-down list are discussed next.

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Figure 10-14 *Changing the dimension height*

Figure 10-15 *Dimension text inside the frame*

Centered. If this option is selected, the dimension text gets positioned on the dimension line in such a way that the dimension line is split to allow for the placement of the text, see Figure 10-17. If the **1st** or **2nd Extension Line** option is selected in the **Horizontal** drop-down list, this centered setting will position the text on the extension line, not on the dimension line.

Above. If this option is selected, the dimension text is placed above the dimension line, except when the dimension line is not horizontal and the dimension text inside the extension lines is horizontal. The distance of the dimension text from the dimension line is controlled by the **DIMGAP** variable. This results in an unbroken solid dimension line being drawn under the dimension text, see Figure 10-16.

Outside. This option places the dimension text on the side of the dimension line.

JIS. This option lets you place the dimension text to conform to the **JIS** (Japanese Industrial Standards) representation.

Note

The horizontal and vertical placement options selected are reflected in the dimensions shown in the ***Preview*** *window.*

Horizontal. This drop-down list is used to control the horizontal placement of the dimension text. You can select the required horizontal placement from this list. However, remember that these options will be useful only when the **Place text manually when dimensioning** check box in the **Fine Tuning** area of the **Fit** tab is cleared. The options in this drop-down list are discussed next.

Centered. This option places the dimension text between the extension lines. This is the default option.

At Ext Line 1. This option is selected to place the text near the first extension line along the dimension line, see Figure 10-17.

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At Ext Line 2. This option is selected to place the text near the second extension line along the dimension line, see Figure 10-17.

Over Ext Line 1. This option is selected to place the text over the first extension line and also along the first extension line, see Figure 10-17.

Over Ext Line 2. This option is selected to place the text over the second extension line and also along the second extension line, see Figure 10-17.

Figure 10-16 *Vertical text placement*

Figure 10-17 *Horizontal text placement*

Offset from dim line. This spinner is used to specify the distance between the dimension line and the dimension text (Figure 10-18). You can set the text gap you need using this spinner. The text gap value is also used as the measure of minimum length for the segments of the dimension line and in basic tolerance. The default value specified in this box is 0.09 units. The value of this setting is stored in the **DIMGAP** system variable.

Text Alignment Area

Horizontal. This is the default option and if selected, the dimension text is drawn horizontally with respect to the current UCS (user coordinate system). The alignment of the dimension line does not affect the text alignment. Selecting this option turns both the **DIMTIH** and **DIMTOH** system variables **on**. The text is drawn horizontally even if the dimension line is at an angle.

Aligned with dimension line. Selecting this radio button aligns the text with the dimension line (Figure 10-19) and both the system variables **DIMTIH** and **DIMTOH** are off.

ISO Standard. If you select the **ISO Standard** radio button, the dimension text is aligned with the dimension line, only when the dimension text is inside the extension lines. Selecting this option turns the system variable **DIMTOH** on; that is, the dimension text outside the extension line is horizontal, regardless of the angle of the dimension line.

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Figure 10-18 *Offset from the dimension line*

Figure 10-19 *Specifying the test alignment*

Exercise 2

Mechanical

Draw Figure 10-20 and then set the values in the **Lines and Arrows** and **Text** tabs of the **New Dimension Style** dialog box to dimension the drawing, as shown in the figure. (Baseline spacing = 0.25, Extension beyond dimension lines = 0.10, Offset from origin = 0.05, Arrow size = 0.09, Text height = 0.08.)

Figure 10-20 *Drawing for Exercise 2*

FITTING DIMENSION TEXT AND ARROWHEADS

Fit Tab

The **Fit** tab provides you with the options that control the placement of dimension lines, arrowheads, leader lines, text, and the overall dimension scale (Figure 10-21).

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Figure 10-21 *The* ***Fit*** *tab of the* ***New Dimension Style*** *dialog box*

Fit Options Area

These options are used to set the priorities for moving the text and arrowheads outside the extension lines, if the space between the extension lines is not enough to fit both of them.

Either the text or the arrows, whichever fits best. This is the default option. In this option, AutoCAD LT places the dimension where it fits best between the extension lines.

Arrows. When you select this option, AutoCAD LT places the text and arrowheads inside the extension lines if there is enough space to fit both. If the space is not available , the arrows are moved outside the extension lines. If there is not enough space for text, both text and arrowheads are placed outside the extension lines.

Text. When you select this option, AutoCAD LT places the text and arrowheads inside the extension lines, if there is enough space to fit both. If there is enough space to fit the arrows, the arrows will be placed inside and the dimension text moves outside the extension lines. If there is not enough space for either the text or the arrowheads, both are placed outside the extension lines.

Both text and arrows. If you select this option, AutoCAD LT will place the arrows and dimension text between the extension lines, if there is enough space available to fit both. Otherwise, both the text and arrowheads are placed outside the extension lines.

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Always keep text between ext lines. This option always keeps the text between the extension lines even in cases where AutoCAD LT would not do so. Selecting this radio button does not affect the radius and diameter dimensions. The value is stored in the **DIMTIX** variable and the default value is **off**.

Suppress arrows if they don't fit inside the extension lines. If you select this check box, the arrowheads are suppressed if the space between the extension lines is not enough to adjust them. The value is stored in the **DIMSOXD** variable and the default value is **off**.

Text Placement Area

This area provides you with the options to position the dimension text when it is moved from the default position. The value is stored in the **DIMTMOVE** variable. The options in this area are as follows.

Beside the dimension line. This option places the dimension text beside the dimension line.

Over the dimension line, with a leader. Selecting this option places the dimension text away from the dimension line and a leader line is created, which connects the text to the dimension line. But, if the dimension line is too close to the text, a leader is not drawn. The Horizontal placement decides whether the text is placed to the right or left of the leader.

Over the dimension line, without a leader. In this option, AutoCAD LT does not create a leader line, if there is insufficient space to fit the dimension text between the extension lines. The dimension text can be moved freely, independent of the dimension line.

Scale for Dimension Features Area

The options under this area set the value for the overall Dimension scale or scaling to the paper space.

Use overall scale of. The current general scaling factor that pertains to all of the size-related dimension variables, such as text size, center mark size, and arrowhead size, is displayed in the **Use overall scale of** spinner. You can alter the scaling factor to your requirement by entering the scaling factor of your choice in this spinner. Altering the contents of this box alters the value of the **DIMSCALE** variable, since the current scaling factor is stored in it. The overall scale (**DIMSCALE**) is not applied to the measured lengths, coordinates, angles, or tolerance. The default value for this variable is 1.0. In this condition, the dimensioning variables assume their preset values and the drawing is plotted at a full scale. The scale factor is the reciprocal of the drawing size and so the drawing is to be plotted at the half size. The overall scale factor (**DIMSCALE**) will be the reciprocal of ½, which is 2.

Note

*If you are in the middle of the dimensioning process and you change the **DIMSCALE** value and save the changed setting in a dimension style file, the dimensions with that style will be updated.*

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Tip
When you increase the limits of the drawing, increase the overall scale of the drawing using the ***Use overall scale of*** *spinner before dimensioning. This will save the time required in changing the individual scale factors of all the dimension parameters.*

Scale dimensions to layout (paperspace). If you select the **Scale dimensions to layout (paper space)** radio button, the scale factor between the current model space viewport and the floating viewport (paper space) is computed automatically. Also, by selecting this radio button, you disable the **Use overall scale of** spinner (it is disabled in the dialog box) and the overall scale factor is set to 0. When the overall scale factor is assigned a value of 0, AutoCAD LT calculates an acceptable default value based on the scaling between the current model space viewport and the paper space. If you are in the paper space (**TILEMODE=0**), or are not using the **Scale dimensions to layout (paper space)** feature, AutoCAD LT sets the overall scale factor to 1; otherwise, AutoCAD LT calculates a scale factor that makes it possible to plot text sizes, arrow sizes, and other scaled distances at the values, in which they have been previously set. (For further details regarding model space and layouts, refer to Chapter 11.)

Fine Tuning Area

The **Fine Tuning** area provides additional options governing placement of the dimension text. The options are as follows.

Place text manually when dimensioning. When you dimension, AutoCAD LT places the dimension text in the middle of the dimension line (if there is enough space). If you select the **Place text manually when dimensioning** check box, you can position the dimension text anywhere along the dimension line. You will also notice that when you select this check box, the **Horizontal Justification** is ignored. This setting is saved in the **DIMUPT** system variable. The default value of this variable is **off**. Selecting this check box enables you to position the dimension text anywhere along the dimension line.

Always draw dim line between ext lines. This check box is selected when you want the dimension line to appear between the extension lines, even if the text and dimension lines are placed outside the extension lines. When you select this option in the radius and diameter dimensions (when default text placement is horizontal), the dimension line and arrows are drawn inside the circle or arc, and the text and leader are drawn outside. When you select the **Always draw dimension line between extension lines** check box, the **DIMTOFL** variable is set to on by AutoCAD LT. The default setting is off.

FORMATTING PRIMARY DIMENSION UNITS

Primary Units Tab

You can use the **Primary Units** tab of the **New Dimension Style** dialog box to control the dimension text format and precision values (Figure 10-22). You can use the options under this tab to control Units, Dimension Precision, and Zero Suppression for dimension measurements. AutoCAD LT lets you attach a user-defined prefix or suffix to the dimension text. For example, you can define the diameter symbol as a prefix by entering %%C in the **Prefix** edit box;

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AutoCAD LT will automatically attach the diameter symbol in front of the dimension text. Similarly, you can define a unit type, such as **mm**, as a suffix; AutoCAD LT will then attach **mm** at the end of every dimension text. This tab also enables you to define zero suppression, precision, and dimension text format.

Figure 10-22 *The* ***Primary Units*** *tab of the* ***New Dimension Style*** *dialog box*

Linear Dimensions Area

Unit format. This drop-down list provides you with the options of specifying the units for the primary dimensions. The formats include Decimal, Scientific, Architectural, Engineering, Fraction, and Windows Desktop. Keep in mind that by selecting a dimension unit format, the drawing units (which you might have selected by using the **UNITS** command) are not affected. The unit setting for linear dimensions is stored in the **DIMLUNIT** system variable.

Precision. This drop-down list is used to control the number of decimal places for the primary units. The setting for precision (number of decimal places) is saved in the **DIMDEC** variable.

Fraction format. This drop-down list is used to set the fraction format. The options are Diagonal, Horizontal, and not stacked. This drop-down list will be available only when you select **Architectural** or **Fractional** from the **Unit format** drop-down list. The value is stored in the **DIMFRAC** variable.

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Decimal separator. This drop-down list is used to select an option that will be used as the decimal separator. For example, Period [.], Comma [,] or Space [ ]. If you have selected Windows desktop units in the **Unit Format:** drop-down list, AutoCAD LT uses the Decimal symbol settings. The value is stored in the **DIMDSEP** variable.

Round off. The **Round off** spinner sets the value for rounding off the dimension values. For example, if the **Round off** spinner is set to 0.05, a value of 0.06 will round off to 0.10. The number of decimal places of the round off value you enter in the edit box should be less than or equal to the value in the **Precision:** edit box. The value is stored in the **DIMRND** variable and the default value in the **Round Off:** edit box is 0. Also see Figure 10-23.

Figure 10-23 *Rounding off the dimension measurements*

Prefix. You can append a prefix to the dimension measurement by entering it in this edit box. The dimension text is converted into **Prefix<dimension measurement>** format. For example, if you enter the text "Abs" in the Prefix edit box, "Abs" will be placed in front of the dimension text (Figure 10-24). The prefix string is saved in the **DIMPOST** system variable.

Note

*Once you specify a prefix, default prefixes such as **R** in radius dimensioning and Ø in diameter dimensioning are cancelled.*

Suffix. Just like appending a prefix, you can append a suffix to the dimension measurement by entering the desired suffix in this edit box. For example, if you enter the text mm in the **Suffix** edit box, the dimension text will have <dimension measurement>mm format, see Figure 10-24. AutoCAD LT stores the suffix string in the **DIMPOST** variable.

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Figure 10-24 *Adding prefix and suffix to the dimensions*

Tip

*The **DIMPOST** variable is used to define both prefix and suffix to the dimension text. This variable takes a string value as its argument. For example, if you want to have a suffix for centimeters, set **DIMPOST** to cm. To establish a prefix to a dimension text, type the prefix text string and then "<>".*

Measurement Scale Area

Scale factor. You can specify a global scale factor for the linear dimension measurements by setting the desired scale factor in the **Scale factor** spinner. All the linear distances measured by dimensions, which include radii, diameters, and coordinates, are multiplied by the existing value in this spinner. For example, if the value of the **Scale factor** spinner is set to 2, two unit segments will be dimensioned as 4 units (2 X 2). However, the angular dimensions are not affected. In this manner, the value of the linear scaling factor affects the contents of the default (original) dimension text (Figure 10-25). The default value for linear scaling is 1. With the default value, the dimension text generated is the actual measurement of the object being dimensioned. The linear scaling value is saved in the **DIMLFAC** variable.

Figure 10-25 *Identical figures dimensioned using different scale factors*

Note

The linear scaling value is not exercised on rounding a value or on plus or minus tolerance values. Therefore, changing the linear scaling factor will not affect the tolerance values.

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Apply to layout dimensions only. When you select the **Apply to layout dimensions only** check box, the scale factor value is applied only to the dimensions in the layout. The value is stored as a negative value in the **DIMLFAC** variable. If you change the **DIMLFAC** variable from the **Dim:** prompt, AutoCAD LT displays the viewport option to calculate the **DIMLFAC** variable. First, set the **TILEMODE** to 0 (paper space), and then invoke the **MVIEW** command to get the **Viewport** option.

Zero Suppression Area. The options in this area are used to suppress the leading or trailing zeros in the dimensioning. This area provides you with four check boxes. These check boxes can be selected to suppress the leading or trailing zeros or zeros in the feet and inches. The **0 Feet** and the **0 Inches** check boxes will be available only when you select **Engineering** or **Architectural** from the **Unit format** drop-down list. When the Architectural units are being used, the **Leading** and **Trailing** check boxes are disabled. For example, if you select the **0 Feet** check box, the dimension text 0'-8 ¾" becomes 8 ¾". By default, the 0 Feet and 0 Inches value is suppressed. If you want to suppress the inches part of a feet-and-inches dimension when the distance in the feet portion is an integer value and the inches portion is zero, select the **0 Inches** check box. For example, if you select the **0 Inches** check box, the dimension text 3'-0" becomes 3'. Similarly, if you select the **Leading** check box, the dimension that was initially 0.53 will become .53. If you select the **Trailing** check box, the dimension that was initially 2.0 will become 2.

Angular Dimensions Area

This area provides you with the options to control the units format, precision, and zero suppression for the Angular units.

Units format. The **Units format** drop-down list displays a list of unit formats for the angular dimensions. The default value, in which the angular dimensions are displayed, is **Decimal Degrees**. The value governing the unit setting for the angular dimensions is stored in the **DIMAUNIT** variable.

Precision. You can select the number of decimal places for the angular dimensions from this drop-down list. This value is stored in the **DIMADEC** variable.

Zero Suppression. Similar to the linear dimensions, you can suppress the **leading**, **trailing**, neither, or both zeros in the angular dimensions by selecting the respective check boxes in this area. The value is stored in the **DIMAZIN** variable.

FORMATTING ALTERNATE DIMENSION UNITS

Alternate Units tab

By default, the **Alternate Units** tab of the **New Dimension Style** dialog box is disabled and the value of the **DIMALT** variable is turned off. If you want to perform alternate units dimensioning, select the **Display Alternate Units** check box. By doing so, AutoCAD LT activates various options in this area (Figure 10-26). This tab sets the format, precision, angles, placement, scale, and so on for the alternate units in use. In this tab, you can specify the values that will be applied to the alternate dimensions.

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Figure 10-26 *The* ***Alternate Units*** *tab of the* ***New Dimension Style*** *dialog box*

Alternate Units Area

The options in this area are identical to those under the **Linear Dimensions** area of the **Primary Units** tab. This area provides you with the options to set the format for all dimension types except Angular.

Unit format. You can select a unit format to apply to the alternate dimensions from this drop-down list. The options under this drop-down list include Scientific, Decimal, Engineering, Architectural stacked, Fractional stacked, Architectural, Fractional, and Windows Desktop. The value is stored in the **DIMALTU** variable. The relative size of fractions is governed by the **DIMTFAC** variable.

Precision. You can select the number of decimal places for the alternate units from the **Precision** drop-down list. The value is stored in the **DIMALTD** variable.

Multiplier for alt units. To generate a value in the alternate system of measurement, you need a factor with which all the linear dimensions will be multiplied. The value for this factor can be set using the **Multiplier for alt units** spinner. The default value of 25.4 is for dimensioning in inches with the alternate units in millimeters. This scaling value (contents of the **Multiplier for alt units** spinner) is stored in the **DIMALTF** variable.

Round distances to. This spinner is used to set a value to which you want all your measurements (made in alternate units) to be rounded off. This value is stored in the **DIMALTRND** system

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variable. For example, if you set the value of the **Round distances to** spinner to 0.25, all the alternate dimensions get rounded off to the nearest .25 unit.

Prefix/Suffix. The **Prefix** and **Suffix** edit boxes are similar to the edit boxes in the **Linear Dimensions** area of the **Primary Units** tab. You can enter the text or symbols that you want to precede or follow the alternate dimension text. The value is stored in the **DIMAPOST** variable. You can also use control codes and special characters to display special symbols.

Zero Suppression Area

This area allows you to suppress the leading or trailing zeros in decimal unit dimensions by selecting either, both, or none of the **Trailing** and **Leading** check boxes. Similarly, selecting the **0 Feet** check box suppresses the zeros in the feet area of the dimension, when the dimension value is less than a foot. Selecting the **0 inches** check box suppresses the zeros in the inches area of the dimension. For example, 1'-0" becomes 1'. The **DIMALTZ** variable controls the suppression of zeros for alternate unit dimension values. The values that are between 0 and 3 affect the feet-and-inch dimensions only.

Placement Area

This area provides the options that control the positioning of the Alternate units. The value is stored in the **DIMAPOST** variable.

After primary value. Selecting the **After primary value** radio button places the alternate units dimension text after the primary units. This is the default option, see Figure 10-27.

Below primary value. Selecting the **Below primary value** radio button places the alternate units dimension text below the primary units, see Figure 10-27.

Figure 10-27 illustrates the result of entering information in the **Alternate Units** tab. The decimal places get saved in the **DIMALTD** variable, the scaling value (contents of the **Multiplier for alt units** spinner) in the **DIMALTF** variable, and the suffix string (contents of the **Suffix** edit box) in the **DIMAPOST** variable. Similarly, the units format for alternate units are in **DIMALTU**, and suppression of zeros for alternate units decimal values are in **DIMALTZ**.

Figure 10-27 *Placements of alternate units*

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FORMATTING THE TOLERANCES

Tolerances Tab

The **Tolerances** tab (Figure 10-28) allows you to set the parameters for options that control the format and display of the tolerance dimension text. These include the alternate unit tolerance dimension text.

Figure 10-28 *The* ***Tolerances*** *tab of the* ***New Dimension Style*** *dialog box*

Tolerance Format Area

The **Tolerance Format** area of the **Tolerances** tab (Figure 10-29) lets you specify the tolerance method, tolerance value, position of tolerance text, and precision and height of the tolerance text. For example, if you do not want a dimension to deviate more than plus 0.01 and minus 0.02, you can specify this by selecting **Deviation** from the **Method** drop-down list and then specifying the plus and minus deviation in the **Upper Value** and the **Lower Value** edit boxes. When you dimension, AutoCAD LT will automatically append the tolerance to it. The **DIMTP** variable sets the maximum (or upper) tolerance limit for the dimension text and **DIMTM** variable sets the minimum (or lower) tolerance limit for the dimension text. Different settings and their effects on relevant dimension variables are explained in the following sections.

Method. The **Method** drop-down list lets you select the tolerance method. The tolerance methods supported by AutoCAD LT are **Symmetrical**, **Deviation**, **Limits**, and **Basic**. These tolerance methods are described next.

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None. Selecting the **None** option sets the **DIMTOL** variable to 0 and does not add tolerance values to the dimension text, that is, the **Tolerances** tab is disabled.

Symmetrical. This option is used to specify the symmetrical tolerances. When you select this option, the **Lower Value** spinner is disabled and the value specified in the **Upper Value** spinner is applied to both plus and minus tolerance. For example, if the value specified in the **Upper Value** spinner is 0.05, the tolerance appended to the dimension text is ±0.05, see Figure 10-29. The value of **DIMTOL** is set to 1 and the value of **DIMLIM** is set to 0.

Deviation. If you select the **Deviation** tolerance method, the values in the **Upper Value** and **Lower Value** spinners will be displayed as plus and minus dimension tolerances. If you enter values for the plus and minus tolerances, AutoCAD LT appends a plus sign (+) to the positive values of the tolerance and a negative sign (–) to the negative values of the tolerance. For example, if the upper value of the tolerance is 0.005 and the lower value of the tolerance is 0.002, the resulting dimension text generated will have a positive tolerance of 0.005 and a negative tolerance of 0.002 (Figure 10-29). Even if one of the tolerance values is 0, a sign is appended to it. On specifying the deviation tolerance, AutoCAD LT sets the **DIMTOL** variable value to 1 and the **DIMLIM** variable value to 0. The values in the **Upper Value** and **Lower Value** edit boxes are saved in the **DIMTP** and **DIMTM** system variables, respectively.

Limits. If you select the **Limits** tolerance method from the **Method** drop-down list, AutoCAD LT adds the upper value (contents of the **Upper Value** spinner) to the dimension text (actual measurement) and subtracts the lower value (contents of the **Lower Value** spinner) from the dimension text. The resulting values are displayed as the dimension text, see Figure 10-29. Selecting the **Limits** tolerance method results in setting the **DIMLIM** variable value to 1 and the **DIMTOL** variable value to 0. The numeral values in the **Upper Value** and **Lower Value** edit boxes are saved in the **DIMTP** and **DIMTM** system variables, respectively.

Figure 10-29 *Various tolerancing methods*

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Basic. A basic dimension text is a dimension text with a box drawn around it (Figure 10-29). The basic dimension is also called a reference dimension. Reference dimensions are used primarily in geometric dimensioning and tolerances. The basic dimension can be realized by selecting the basic tolerance method. The distance provided around the dimension text (distance between dimension text and the rectangular box) is stored as a negative value in the **DIMGAP** variable. The negative value signifies the basic dimension. The default setting is off, resulting in the generation of dimensions without the box around the dimension text.

Precision. The **Precision** drop-down list is used to select the number of decimal places for the tolerance dimension text. The value is stored in **DIMTDEC** variable.

Upper value/Lower value. In the **Upper value** spinner the positive upper or maximum value is specified. If the method of tolerances is symmetrical, the same value is used as the lower value also. The value is stored in the **DIMTP** variable. In the **Lower** spinner, the lower or minimum value is specified. The value is stored in the **DIMTM** variable.

Scaling for height. The **Scaling for height** spinner is used to specify the height of the dimension tolerance text relative to the dimension text height. The default value is 1, which means the height of the tolerance text is the same as the dimension text height. If you want the tolerance text to be 75 percent of the dimension height text, enter 0.75 in the **Scaling for height** edit box. The ratio of the tolerance height to the dimension text height is calculated by AutoCAD LT and then stored in the **DIMTFAC** variable. **DIMTFAC = Tolerance Height/Text Height.**

Vertical position. This drop-down list allows you to specify the location of the tolerance text for deviation and symmetrical methods only. The three alignments that are possible are with the **Bottom**, **Middle**, or **Top** of the main dimension text. The settings are saved in the **DIMTOLJ** system variable (Bottom=0, Middle=1, and Top=2).

Zero Suppression Area

This area controls the zero suppression in the tolerance text depending on which one of the check boxes is selected. Selecting the **Leading** check box suppresses the leading zeros in all the decimal tolerance text. For example, 0.2000 becomes .2000. Selecting the **Trailing** check box suppresses the trailing zeros in all the decimal tolerance text. For example, 0.5000 becomes 0.5. Similarly, selecting both the boxes suppresses both the trailing and leading zeros and selecting none, suppresses none. If you select **0 Feet** check box, the zeros in the feet portion of the tolerance dimension text are suppressed if the dimension value is less than one foot. Similarly, selecting the **0 Inches** check box suppresses the zeros in the inches portion of the dimension text. The value is stored in the **DIMTZIN** variable.

Alternate Unit Tolerance Area

The options in this area define the precision and zero suppression settings for the Alternate unit tolerance values. The options under this area will be available only when you display the alternate units along with the primary units.

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Precision. This drop-down list is used to set the number of decimal places to be displayed in the tolerance text of the alternate dimensions. This value is stored in the **DIMALTTD** variable.

Zero Suppression Area. Selecting the respective check boxes controls the suppression of the **Leading** and **Trailing** zeros in decimal values and the suppression of zeros in the Feet and Inches portions for dimensions in the feet and inches format. The value is stored in the **DIMALTTZ** variable.

Exercise 3 *Mechanical*

Draw Figure 10-30 and then set the values in the various tabs of the **New Dimension Style** dialog box to dimension it, as shown. (Baseline spacing = 0.25, Extension beyond dim lines = 0.10, Offset from origin = 0.05, Arrowhead size =0.07, Text height = 0.08.) Assume the missing dimensions.

Figure 10-30 *Drawing for Exercise 3*

OTHER DIMENSIONING VARIABLES

Positioning Dimension Text (DIMTVP Variable)

You can position the dimension text with respect to the dimension line by using the **DIMTVP** system variable (Figure 10-31). In certain cases, **DIMTVP** is used with **DIMTAD** to control the vertical position of the dimension text. The **DIMTVP** value applies only when the **DIMTAD** is off. To select the vertical position of the dimension text to meet your requirement (over or under the dimension line), you must first calculate the numerical value by which you want to offset the text from the dimension line. The vertical placing of the text is done by offsetting the dimension text. The magnitude of the offset of the dimension text is a product of text height and the **DIMTVP** value. If the value of **DIMTVP** is 1, **DIMTVP** acts as **DIMTAD**. For example, if you want to position the text 0.25 units from the dimension line, the value of **DIMTVP** is calculated as follows.

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DIMTVP=Relative Position value/Text Height value
DIMTVP=0.25/ 0.09=2.7778

The value 2.7778 is stored in the dimension variable **DIMTVP**. If the absolute value is less than 0.70, the dimension line is broken to accommodate the dimension text. Relative positioning is not effective on angular dimensions.

Figure 10-31 *Vertical positioning of the dimension text*

DIMENSION STYLE FAMILIES

The dimension style feature of AutoCAD LT lets the user define a dimension style with values that are common to all dimensions. For example, the arrow size, dimension text height, or color of the dimension line are generally the same in all types of dimensioning such as linear, radial, diameter, and angular. These dimensioning types belong to the same family because they have some characteristics in common. In AutoCAD LT, this is called a **dimension style family,** and the values assigned to the family are called **dimension style family values.**

After you have defined the dimension style family values, you can specify variations on it for other types of dimensions such as radial and diameter. For example, if you want to limit the number of decimal places to two in radial dimensioning, you can specify that value for radial dimensioning. The other values will stay the same as the family values to which this dimension type belongs. When you use the radial dimension, AutoCAD LT automatically uses the style that was defined for radial dimensioning; otherwise, it creates a radial dimension with the values as defined for the family. After you have created a dimension style family, any changes in the parent style are applied to family members, if the particular property is the same. Special suffix codes are appended to the dimension style family name that correspond to different dimension types. For example, if the dimension style family name is MYSTYLE and you define a diameter type of dimension, AutoCAD LT will append $4 at the end of the dimension style family name. The name of the diameter type of dimension will be MYSTYLE$4. The following are the suffix codes for different types of dimensioning.

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Suffix Code	Dimension Type	Suffix Code	Dimension Type
0	Linear	2	Angular
3	Radius	4	Diameter
6	Ordinate	7	Leader

Example 1 *Mechanical*

The following example illustrates the concepts of dimension style families, Figure 10-32.

1. Specify the values for the dimension style family.
2. Specify the values for the linear dimensions.
3. Specify the values for the diameter and radius dimensions.
4. After creating the dimension style, use it to dimension the given drawing.

Figure 10-32 *Drawing for Example 1*

1. Open a new file and then draw the object shown in Figure 10-32.

2. Invoke the **Dimension Style Manager** dialog box by choosing the **Dimension Style** button from the **Dimension** toolbar. AutoCAD LT will display **Standard** in the **Styles** list box. Select the S**tandard** style from the **Styles** list box.

3. Choose the **New** button to display the **Create New Dimension Style** dialog box. In this dialog box, enter **MyStyle** in the **New Style Name** edit box. Select **Standard** from the **Start With** drop-down list. Also select **All dimensions** from the **Use for** drop-down list. Now, choose the **Continue** button to display the **New Dimension Style: MyStyle** dialog box. In this dialog box, choose the **Lines and Arrows** tab and enter the following values.

Baseline Spacing: 0.15
Offset from origin: 0.03
Center Mark for circle, Size: 0.05
Extension beyond dim line: 0.07
Arrow size: 0.09
Center Mark for Circles Type: Line

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4. Choose the **Text** tab and change the following values:

   Text Height: 0.09 **Offset from dimension line: 0.03**

5. Choose the **Fit** tab and set the value of the **Use overall scale of** spinner in the **Scale for Dimension Features** area to **1**.

6. After entering the values, choose the **OK** button to return to the **Dimension Style Manager** dialog box. This dimension style contains the values that are common to all dimension types.

7. Now, choose the **New** button again in the **Dimension Style Manager** dialog box to display the **Create New Dimension Style** dialog box. AutoCAD LT displays **Copy of MyStyle** in the **New Style name** edit box. Select **MyStyle** from the **Start with** drop-down list if it is not already selected. From the **Use for** drop-down list, select **Linear dimensions**. Choose the **Continue** button to display the **New Dimension Style: MyStyle: Linear** dialog box and set the following values in the **Text** tab:

   a. Select the **Aligned with dimension line** radio button in the Text alignment area.
   b. In the **Text placement** area, from the **Vertical** drop-down list, select **Above**.

8. In the **Primary Units** tab, set the precision to two decimal places. Choose the **OK** button to return to the **Dimension Style Manager** dialog box.

9. Choose the **New** button again to display the **New Dimension Style** dialog box. Select **MyStyle** from the **Start with** drop-down list. Also, select **Diameter dimension** type from the **Use for** drop-down list. Choose the **Continue** button to display the **New dimension Style: MyStyle: Diameter** dialog box.

10. Choose the **Primary units** tab and set the **Precision** to two decimal places. In the **Lines and Arrows** tab, select **Line** from the **Type** drop-down list in the **Center mark for circle** area. Choose the **OK** button to return to the **Dimension Style Manager** dialog box.

11. In this dialog box, choose the **New** button to display the **Create New Dimension Style** dialog box. Select **MyStyle** from the **Start with** drop-down list and **Radius dimensions** from the **Use for** drop-down list. Choose the **Continue** button to display the **New Dimension Style: MyStyle: Radial** dialog box.

12. Choose the **Primary Units** tab and set the precision to two decimal places. Enter **Rad** in the **Prefix** edit box.

13. In the **Fit** tab, select the **Text** radio button in the **Fit Options** area. Choose the **OK** button to return to the **Dimension Style Manager** dialog box.

14. Select **MyStyle** from the **Styles** list box and choose the **Set current** button. Choose the **Close** button to exit the dialog box.

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15. Use the linear and baseline dimensioning to draw the linear dimensions as shown in Figure 10-33. You will notice that when you enter any linear dimensioning, AutoCAD LT automatically uses the values that were defined for the linear type of dimensioning.

16. Use the diameter dimensioning to dimension the circles as shown in Figure 10-32. Again, notice that the dimensions are drawn according to the values specified for the diameter type of dimensioning.

17. Now, use the radius dimensioning to dimension the fillet, as shown in Figure 10-32.

USING DIMENSION STYLE OVERRIDES

Most of the dimension characteristics are common in a production drawing. The values that are common to different dimensioning types can be defined in the dimension style family. However, at times, you might have different dimensions. For example, you may need two types of linear dimensioning: one with tolerance and one without. One way to draw these dimensions is to create two dimensioning styles. You can also use the dimension variable overrides to override the existing values. For example, you can define a dimension style (**MyStyle**) that draws dimensions without tolerance. Now, to draw a dimension with tolerance or update an existing dimension, you can override the previously defined value through the **Dimension Style Manager** dialog box or by setting the variable values at the Command prompt. The following example illustrates how to use the dimension style overrides.

Example 2 *Mechanical*

In this example, you will update the overall dimension (3.00) so that the tolerance is displayed with the dimension. You will also add two linear dimensions, as shown in Figure 10-33.

This problem can be solved by dimension style overrides as well as using the **Properties** palette. However, here only the dimension stvle overrides method will be discussed.

Figure 10-33 *Drawing for Example 2*

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1. Invoke the **Dimension Style Manager** dialog box. Select **MyStyle** from the **Styles** list box and choose the **Override** button to display the **Override Current Style: My Style** dialog box. The options in this dialog box are identical to the **New Dimension Style** dialog box discussed earlier in the chapter.

2. Choose the **Tolerance** tab and select **Symmetrical** from the **Method** drop-down list.

3. Set the value of the **Precision** spinner to two decimal places. Set the value of the **Upper value** spinner to **0.02**. Choose the **OK** button to exit the dialog box (this does not save the style). You will notice that **<style overrides>** is displayed under **MyStyle** in the **Style** list box, indicating that the style overrides the **MyStyle** dimension style.

4. This **<style overrides>** is displayed until you save it under a new name or under the style it is displayed under, or until you delete it. Select **<style overrides>** and right-click to display the shortcut menu. Choose the **Save to current Style** option from the shortcut menu to save the overrides to the current style. Choosing the **Rename** option, allows you to rename the style override and save it as a new style.

5. Choose **Update** from the **Dimension** menu and select the dimension that measures **3.00**. It will now display the symmetrical tolerance.

6. Draw the remaining two linear dimensions. They will automatically appear with the tolerances, see Figure 10-34.

Tip

*You can also use the **DIMOVERRIDE** command to apply the change to the existing dimensions. Apply the changes to the **DIMTOL**, **DIMTP**, and **DIMTM** variables.*

COMPARING AND LISTING DIMENSION STYLES

Choosing the **Compare** button in the **Dimension Style Manager** dialog box displays the **Compare Dimension Styles** dialog box where you can compare the settings of two dimensions styles or list all the settings of one of them (Figure 10-34).

The **Compare** and the **With** drop-down lists display the dimension styles in the current drawing. Selecting the dimension styles from the respective lists compare the two styles. In the **With** drop-down list, if you select **None** or the same style as selected from the **Compare** drop-down list, all the properties of the selected style are displayed. The comparison results are displayed under four headings: **Description** of the Dimension Style property, the **System Variable** controlling a particular setting, and the **values of the variable for both the dimension styles** which differ in the two styles in comparison. The number of differences between the selected dimension styles are displayed below the **With** drop-down list. The button provided in this dialog box prints the comparison results to the Windows clipboard from where they can be pasted to other Windows applications.

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Figure 10-34 *The* ***Compare Dimension Styles*** *dialog box*

USING EXTERNALLY REFERENCED DIMENSION STYLES

The externally referenced dimensions cannot be used directly in the current drawing. When you Xref a drawing, the drawing name is appended to the style name and the two are separated by the vertical bar (|) symbol. It uses the same syntax as the other externally dependent symbols. For example, if the drawing (FLOOR) has a dimension style called DECIMAL and you Xref this drawing in the current drawing, AutoCAD LT will rename the dimension style to FLOOR|DECIMAL. You cannot make this dimension style current, nor can you modify or override it. However, you can use it as a template to create a new style. To accomplish this, invoke the **Dimension Style Manager** dialog box. If the **Don't list styles in Xrefs** check box is selected, the styles in the Xref are not displayed. Clear this check box to display the Xref dimension styles and choose the **New** button. In the **New Style Name** edit box of the **New Dimension Style** dialog box, enter the name of the dimension style. AutoCAD LT will create a new dimension style with the same values as those of the externally referenced dimension style (FLOOR|DECIMAL).

Self-Evaluation Test

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

1. You can invoke the **Dimension Style Manager** dialog box using both the **Format** menu and the **Dimension** menu. (T/F)

2. The size of the arrow block is determined by the value stored in the **Arrow size** edit box. (T/F)

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3. The default dimension style file name is **Drawing**. (T/F)

4. The size of the tolerance text with respect to the dimensions can be defined. (T/F)

5. The **DIMTVP** variable is used to control the _____ position of the dimension text.

6. When you select the **Arrows** option, AutoCAD LT places the text and arrowheads _____.

7. A basic dimension text is dimension text with a _____drawn around it.

8. The **Suppress** check boxes in the **Dimension Lines** area control the display of _____ and _____.

9. You can specify the tolerancing using _____ methods.

10. The _____ button in the **Dimension Style Manager** dialog box is used to override the current dimension style.

Review Questions

Answer the following questions.

1. You cannot replace the default arrowheads at the end of the dimension lines. (T/F)

2. When the **DIMTVP** variable has a negative value, the dimension text is placed below the dimension line. (T/F)

3. A dimension style name cannot be changed. (T/F)

4. The named dimension style associated with the dimension being updated by overriding is not updated. (T/F)

5. To use a dimension style for dimensioning you will have to first make it active using which of the following buttons?

   (a) **Set Current** (b) **New**
   (c) **Override** (d) **Modify**

6. To add a suffix **mm** to the dimensions, which tab of the **Dimension Style Manager** dialog box will you use?

   (a) **Fit** (b) **Text**
   (c) **Primary Units** (d) **Alternate Units**

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7. If you want to place the dimension text manually every time you create a dimension, which tab of the **Dimension Style Manager** dialog box will you use?

   (a) **Fit** (b) **Text**
   (c) **Primary Units** (d) **Alternate Units**

8. The size of the _____ is determined by the value stored in the **Arrow size** edit box.

9. When **DIMSCALE** is assigned a value of _____, AutoCAD LT calculates an acceptable default value based on the scaling between the current model space viewport and the paper space.

10. If you use the **DIMCEN** command, a positive value will create a center mark, and a negative value will create a _____.

11. If you select the _____ check box, you can position the dimension text anywhere along the dimension line.

12. You can append a prefix to the dimension measurement by entering the desired prefix in the **Prefix** edit box of the _____ dialog box.

13. If you select the **Limits** tolerance method from the **Method** drop-down list, AutoCAD LT _____ the upper value to the dimension and _____ the lower value from the dimension text.

14. You can also use the _____ command to override a dimension value.

15. What is the dimension style family, and how does it help in dimensioning? _____ _____.

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Exercises

Exercises 4 through 9

Mechanical

Create the drawings shown in Figures 10-35 through 10-40. You must create dimension style files and specify the values for the different dimension types such as linear, radial, diameter, and ordinate. Assume the missing dimensions.

Figure 10-35 *Drawing for Exercise 4*

Figure 10-36 *Drawing for Exercise 5*

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Figure 10-37 *Drawing for Exercise 6*

HOLE	X	Y	Z
RAD.	0.125	0.15	0.20

Figure 10-38 *Drawing for Exercise 7*

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HOLE	X1	X2	X3	Y1	Y2	Z1
DIM.	R0.2	R0.2	R0.2	R0.1	R0.1	R0.15
QTY.	1	1	1	1	1	1
X	0.25	1.375	2.50	0.75	2.0	1.375
Y	0.25	0.25	0.25	0.75	0.75	1.125
Z	THRU	THRU	THRU	1.0	1.0	THRU

Figure 10-39 *Drawing for Exercise 8*

SPANNER NO.	A	B	C	D
S1	0.85	0.65	0.50	0.38
S2	1	0.75	0.59	0.44
S3	1.15	0.88	0.67	0.52
S4	1.25	0.95	0.74	0.56

Figure 10-40 *Drawing for Exercise 9*

Exercise 10

Mechanical

Draw the sketch shown in Figure 10-41. You must create the dimension style and specify different dimensioning parameters. Also, suppress the leading and trailing zeros in the dimension style. Assume the missing dimensions.

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Figure 10-41 *Drawing for Exercise 10*

Exercise 11

Mechanical

Draw the sketch shown in Figure 10-42. You must create the dimension style and specify different dimensioning parameters in the dimension style. Assume the missing dimensions.

Figure 10-42 *Drawing for Exercise 11*

Exercise 12

Mechanical

Draw the sketch shown in Figure 10-43. You must create the dimension style and specify different dimensioning parameters in the dimension style. Assume the missing dimensions.

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Figure 10-43 *Drawing for Exercise 12*

Problem Solving Exercise 1

Mechanical

Create the drawing shown in Figure 10-44. You must create the dimension style and specify different dimensioning parameters in the dimension style. Also, suppress the leading and trailing zeros in the dimension style. Assume the missing dimensions.

Figure 10-44 *Drawing for Problem Solving Exercise 1*

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Problem Solving Exercise 2

Mechanical

Draw the shaft shown in Figure 10-45. You must create the dimension style and specify the dimensioning parameters based on the given drawing. Assume the missing dimensions.

Figure 10-45 *Drawing for Problem Solving Exercise 2*

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Problem Solving Exercise 3

Mechanical

Draw the connecting rod shown in Figure 10-46. You must create the dimension style and specify the dimensioning parameters based on the given drawing. Assume the missing dimensions.

Figure 10-46 *Drawing for Problem Solving Exercise 3*

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Problem Solving Exercise 4

Architectural

Draw the elevation of the house shown in Figure 10-47. Assume the missing dimensions.

Figure 10-47 *Drawing for Problem Solving Exercise 4*

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Problem Solving Exercise 5

Architectural

Create the drawing shown in Figure 10-48. Assume the missing dimensions.

Figure 10-48 *Drawing for Problem Solving Exercise 5*

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Problem Solving Exercise 6

Architectural

Create the drawing shown in Figure 10-49. Assume the missing dimensions.

Figure 10-49 *Drawing for Problem Solving Exercise 6*

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Answers to Self-Evaluation Test

1 - T **2** - T, **3** - **F**, **4** - T, **5** - vertical, **6** - inside, **7** - frame, **8** - first, second dimension lines, **9** - four, **10** - **Override**