Chapter 7

Controlling the Drawing Display and Creating Text

Learning Objectives

After completing this chapter, you will be able to:

- *Use the **REDRAW** and **REGEN** commands.*
- *Use the **ZOOM** command and its options.*
- *Understand the **PAN** and **VIEW** commands.*
- *Understand the use of the **Aerial View** window.*
- *Draw text using the **TEXT** command.*
- *Create paragraph text using the **MTEXT** command.*
- *Edit text using the **DDEDIT** command.*
- *Use the **PROPERTIES** palette to change the properties of the text.*
- *Substitute fonts and specify alternate default fonts.*
- *Create text styles using the **STYLE** command.*
- *Determine text height.*
- *Check spellings and find and replace text.*

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BASIC DISPLAY OPTIONS

Drawing in AutoCAD LT is much simpler than manual drafting in many ways. Sometimes while drawing, it is very difficult to see and alter minute details. In AutoCAD LT, you can overcome this problem by viewing only a specific portion of the drawing. For example, if you want to display a part of the drawing on a larger area, you can use the **ZOOM** command, which lets you enlarge or reduce the size of the drawing displayed on the screen. Similarly, you can use the **REGEN** command to regenerate the drawing and **REDRAW** to refresh the screen. In this chapter, you will learn some of the drawing display commands, such as **REDRAW**, **REGEN**, **PAN**, **ZOOM**, and **VIEW**. These commands can also be used in the transparent mode. Transparent commands are commands that can be used while another command is in progress. Once you have completed the process involved with a transparent command, AutoCAD LT automatically returns you to the command with which you were working before you invoked the transparent command.

REDRAWING THE SCREEN

Menu:	View > Redraw
Command:	REDRAW

The **REDRAW** command redraws the screen and is used to remove the small cross marks (blips) that appear when a point is specified on the screen when BLIPMODE is set to on. The blip mark is not treated as an element of the drawing. The **REDRAW command** also redraws the objects that do not display on the screen as a result of editing some other object. In AutoCAD LT, several commands redraw the screen automatically (for example, when a grid is turned off), but it is sometimes useful to redraw the screen explicitly. In AutoCAD LT, the **REDRAW** command can also be used in the transparent mode. This can be done by typing an apostrophe in front of the command. The apostrophe appended to a command indicates that the command is to be used as a transparent command (Command: **'REDRAW**). Use of the **REDRAW** command does not involve a prompt sequence; instead, the redrawing process takes place without any prompting for information.

The **REDRAW** command affects only the current viewport. If you have more than one viewport you can use the **REDRAWALL** command to redraw all the viewports. **Redraw** in the **View** menu is the **REDRAWALL** command.

Tip

*While working on complex drawings, it may be better to set the **BLIPMODE** variable as **Off** (default) instead of using the **REDRAW** command to clear blips.*

REGENERATING THE DRAWING

Menu:	View > Regen
Command:	REGEN

The **REGEN** command makes AutoCAD LT regenerate the entire drawing to update it. The need for regeneration usually occurs when you change certain aspects of the drawing. All objects

in the drawing are recalculated and redrawn in the current viewport. One of the advantages of this command is that the drawing is refined by smoothing out circles and arcs. To use this command, enter **REGEN** at the Command prompt. AutoCAD LT displays the message **Regenerating model** while it regenerates the drawing. The **REGEN** command affects only the current viewport. If you have more than one viewport, you can use the **REGENALL** command to regenerate all of them. The **REGEN** command can be aborted by pressing ESC. This saves time if you are going to use another command that causes automatic regeneration.

Tip
*Under certain conditions, the **ZOOM** and **PAN** commands automatically regenerate the drawing. Some other commands also perform regenerations under certain conditions.*

ZOOMING DRAWINGS

Toolbar:	Zoom toolbar or Standard > Zoom flyout
Menu:	View > Zoom
Command:	ZOOM

Creating drawings on the screen would not be of much use if you can not magnify the drawing view to work on the minute details. Getting close to or away from the drawing is the function of the **ZOOM** command. In other words, this command enlarges or reduces the view of the drawing on the screen, but it does not affect the actual size of the objects. In this way, the **ZOOM** command functions like the zoom lens on a camera. When you magnify the apparent size of a section of the drawing, you see that area in greater detail. On the other hand, if you reduce the apparent size of the drawing, you see a larger area. With this release of AutoCAD LT, zooming of the entities is animated. You can see the transition of entities from the initial state to the zoomed state.

The ability to zoom in, or magnify, has been helpful in creating the minuscule circuits used in the electronics and computer industries. This is one of the most frequently used commands. Also, this command can be used transparently, which means that it can be used while working in other commands. The **ZOOM** command can be invoked transparently from the **Zoom** toolbar, as shown in Figure 7-1 or by right-clicking in the drawing area and choosing **Zoom** from the menu. This option is also available in the shortcut menu that is displayed even when you are inside some other command, as shown in Figure 7-2.

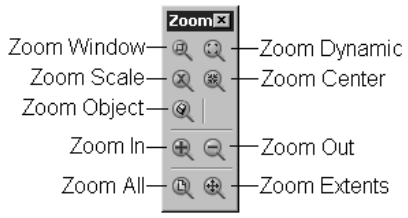

Figure 7-1 *Selecting the **Zoom** options from the **Zoom** toolbar*

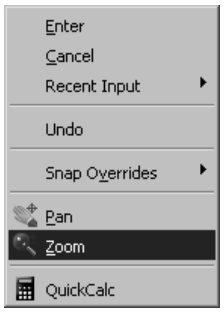

Figure 7-2 *Invoking the **ZOOM** command from the **LINE** command shortcut menu*

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This command has several options and can be used in a number of ways. The following is the prompt sequence that is displayed when you invoke this command.

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:

Realtime Zooming

You can use the **Realtime Zoom** to zoom in and zoom out interactively. To zoom in, invoke the command, then hold the pick button down and move the cursor up. If you want to zoom in further, release the pick button and bring the cursor down. Specify a point and move the cursor up again. Similarly, to zoom out, hold the pick button down and move the cursor down. If you move the cursor vertically up from the midpoint of the screen to the top of the window, the drawing is magnified by 100% (zoom in 2x magnification). Similarly, if you move the cursor vertically down from the midpoint of the screen to the bottom of the window, the drawing display is reduced 100% (zoom out 0.5x magnification). Realtime zoom is the default setting for the **ZOOM** command. Pressing ENTER after entering the **ZOOM** command automatically invokes the realtime zoom.

When you use the realtime zoom, the cursor becomes a magnifying glass and displays a plus sign (+) and a minus sign (–). When you reach the zoom out limit, AutoCAD LT does not display the minus sign (–) while dragging the cursor. Similarly, when you reach the zoom in limit, AutoCAD LT does not display the plus sign (+) while dragging the cursor. To exit the realtime zoom, press ENTER or ESC, or select **Exit** from the shortcut menu.

All Option

This option of the **ZOOM** command adjusts the display area on the basis of the drawing limits (Figure 7-3) or extents of the object, whichever is greater. Even if the objects are not within the limits, they are still included in the display. Hence, with the help of the **All** option, you can view the entire drawing in the current viewport (Figure 7-4).

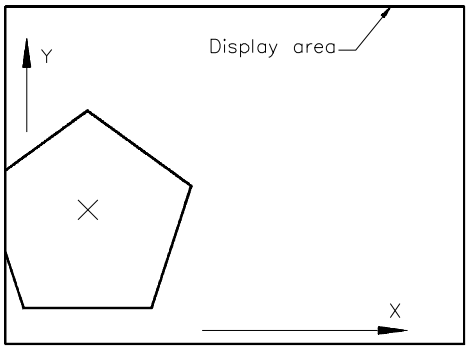

Figure 7-3 *Drawing showing limits*

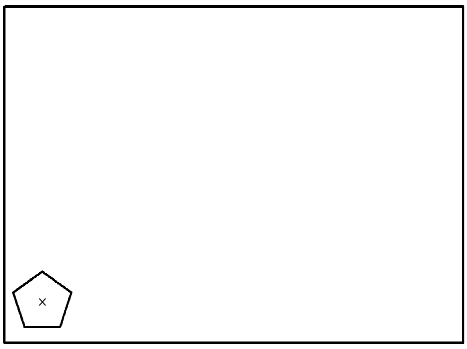

Figure 7-4 *The* ***Zoom All*** *option*

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Center Option

This option lets you define a new display window by specifying its center point (Figures 7-5 and 7-6) and the magnification or height. Here, you are required to enter the **center** and the **height** of the subsequent screen display. If you press ENTER instead of entering a new center point, the center of the view will remain unchanged. Instead of entering a height, you can enter the **magnification factor** by typing a number. If you press ENTER at the height prompt, or if the height you enter is the same as the current height, magnification does not take place. For example, if the current height is 2.7645 and you press ENTER at the **magnification or height <current>** prompt, magnification will not take place. The smaller the value, the greater the enlargement of the image. You can also enter a number followed by **X**. This indicates the change in magnification, not as an absolute value, but as a value relative to the current screen.

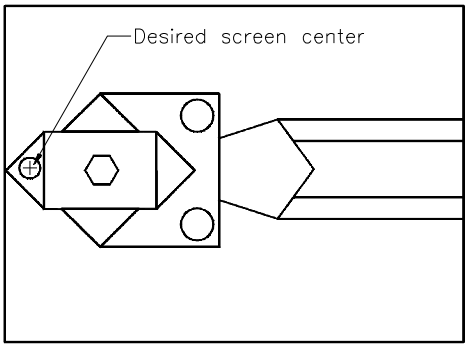

Figure 7-5 *Drawing before using the* ***ZOOM Center*** *option*

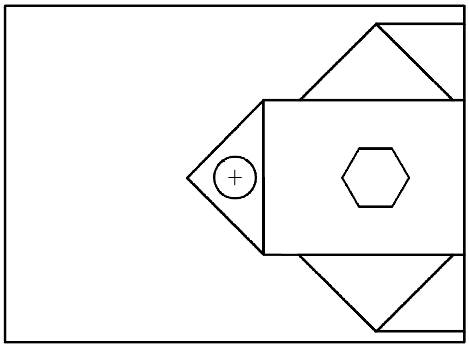

Figure 7-6 *Drawing after using the* ***ZOOM Center*** *option*

The prompt sequence is given next.

Command : **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **C** Enter
Specify center point : *Specify a center point.*
Enter magnification or height <current>: 5X Enter

In Figure 7-6, the current magnification height is 5X, which magnifies the display five times. If you enter a value of 2, the size (height and width) of the zoom area changes to 2 X 2 around the specified center. In Figure 7-7, if you enter .12 as the height after specifying the center point as the circle's center, the circle will zoom to fit in the display area since its diameter is 0.12. The prompt sequence is given next.

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: **C** Enter
Center point: *Select the center of the circle.*
Magnification or Height <5.0>: **0.12** Enter

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Extents Option

As the name indicates, this option lets you zoom to the extents of the biggest object in the drawing. The extents of the drawing comprise the area that has the drawings in it. The rest of the empty area is neglected. With this option, all objects in the drawing are magnified to the largest possible display, see Figure 7-8.

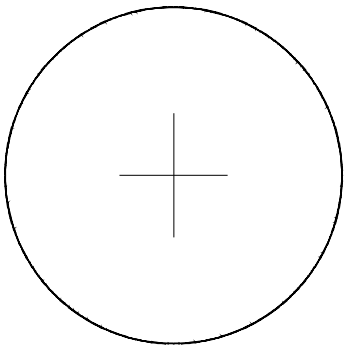

Figure 7-7 *Drawing after using the* ***ZOOM Center*** *option*

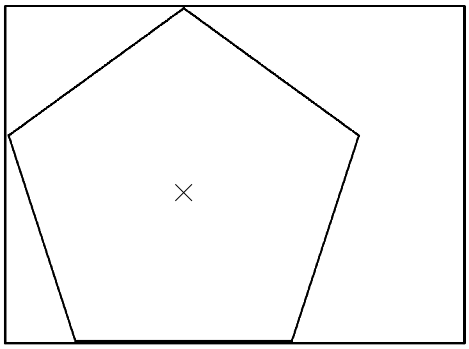

Figure 7-8 *The* ***ZOOM Extents*** *option*

Dynamic Option

This option displays the portion of the drawing that you have already specified. The prompt sequence for using this option of the **ZOOM** command is given next.

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: **D** Enter

You can then specify the area you want to be displayed by manipulating a view box representing your viewport. This option lets you enlarge or shrink the view box and move it around. When you have the view box in the proper position and size, the current viewport is cleared by AutoCAD LT and a special view selection screen is displayed. This special screen comprises information regarding the current view as well as the available views. In a color display, the different viewing windows are very easy to distinguish because of their different colors, but in a monochrome monitor, they can be distinguished by their shape.

Blue dashed box representing drawing extents

Drawing extents are represented by a dashed blue box (Figure 7-9), which constitutes the larger of the drawing limits or the actual area occupied by the drawing.

Green dashed box representing the current view

A green dashed box is formed to represent the area that the current viewport comprises when the **Dynamic** option of the **ZOOM** command is invoked (Figure 7-10).

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Panning view box (X in the center)

A view box initially of the same size as the current view box is displayed with an X in the center (Figure 7-11). You can move this box with the help of your pointing device. This box, known as the **panning view box**, helps you to find the center point of the zoomed display you want. When you have found the center, press the pick button to make the zooming view box appear.

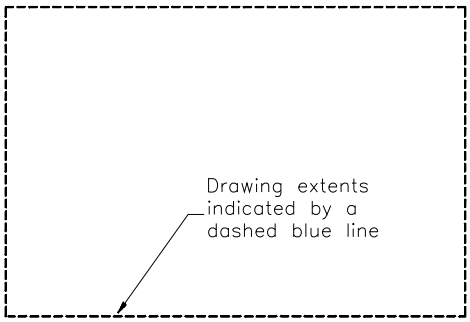

Figure 7-9 *Box representing drawing extents*

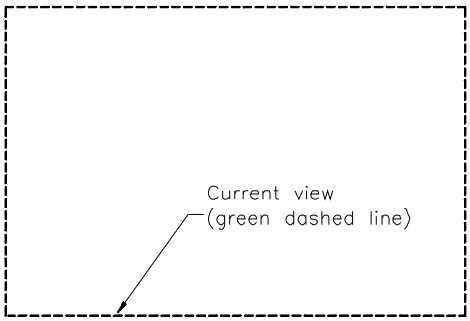

Figure 7-10 *Representation of the current view*

Zooming view box (arrow on the right side)

After you press the pick button in the center of the panning view box, the X in the center of the view box is replaced by an arrow pointing to the right edge of the box. This **zooming view box** (Figure 7-12) indicates the ZOOM mode. You can now increase or decrease the area of this box according to the area you want to zoom into. To shrink the box, move the pointer to the left; to increase it, move the pointer to the right. The top, right, and bottom sides of the zooming view box move as you move the pointer, but the left side remains fixed, with the zoom base point at the midpoint of the left side. You can slide it up or down along the left side. When you have the zooming view box in the desired size for your zoom display, press ENTER to complete the command and zoom into the desired area of the drawing. Before pressing ENTER, if you want to change the position of the zooming view box, click the pick button of your pointing device to make the panning view box reappear. After repositioning, press ENTER.

Previous Option

While working on a complex drawing, you may need to zoom in on a portion of the drawing to edit some minute details. Once the editing is over you may want to return to the previous view. This can be done using the **Previous** option of the **ZOOM** command. Without this option, it would be very tedious to zoom back to the previous views. AutoCAD LT saves the view specification of the current viewport whenever it is being altered by any of the ZOOM options or by the PAN, VIEW Restore, **DVIEW**, or **PLAN** commands (which are discussed later). Up to ten views are saved for each viewport. The prompt sequence for this option is given next.

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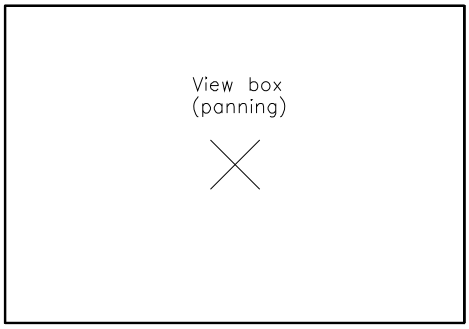

Figure 7-11 *The panning view box*

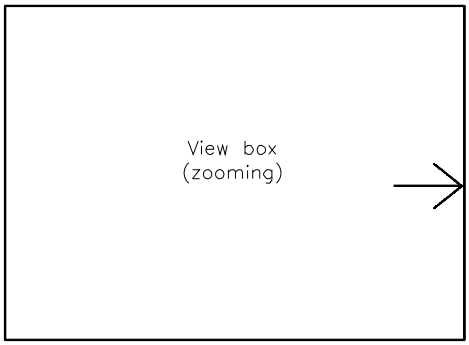

Figure 7-12 *The zooming view box*

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: **P** Enter

Successive **ZOOM > P** commands can restore up to ten previous views. **VIEW** here refers to the area of the drawing defined by its display extents. If you erase some objects and then issue a **ZOOM > Previous** command, the previous view is restored, but the erased objects are not.

Window Option

This is the most commonly used option of the **ZOOM** command. It lets you specify the area you want to zoom in on, by letting you specify two opposite corners of a rectangular window. The center of the specified window becomes the center of the new display screen. The area inside the window is magnified or reduced in size to fill the display as completely as possible. The points can be specified either by selecting them with the help of the pointing device or by entering their coordinates. The prompt sequence is given next.

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: *Specify a point.*
Specify opposite corner: *Specify another point.*

Whenever the **ZOOM** command is invoked, the window method is one of two default options. This is illustrated by the previous prompt sequence where you can specify the two corner points of the window without invoking any option of the **ZOOM** command. The **Window** option can also be used by entering **W**. In this case the prompt sequence is given next.

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: **W** Enter
Specify first corner: *Specify a point.*
Specify opposite corner: *Specify another point.*

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Scale Option

The **Scale** option of the **ZOOM** command is a very versatile option. It can be used in the following ways.

Scale: Relative to full view

This option of the **ZOOM** command lets you magnify or reduce the size of a drawing according to a scale factor (Figure 7-13). A scale factor equal to 1 displays an area equal in size to the area defined by the established limits. This may not display the entire drawing if the previous view was not centered on the limits or if you have drawn outside the limits. To get a magnification relative to the full view, you can enter any other number. For example, you can type 4 if you want the displayed image to be enlarged four times. If you want to decrease the magnification relative to the full view, you need to enter a number that is less than 1. In Figure 7-14, the image size decreased because the scale factor is less than 1. In other words, the image size is half of the full view because the scale factor is 0.5.
The prompt sequence is given next.

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: **S** Enter
Enter a scale factor (nX or nXP) : **0.5** Enter

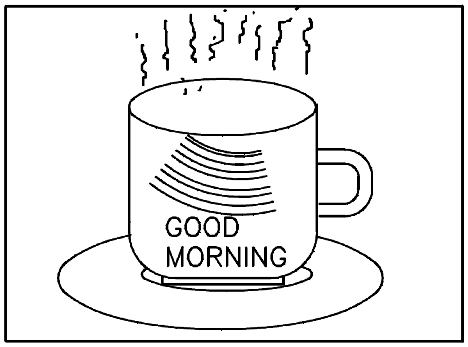

Figure 7-13 *Drawing before the* ***ZOOM Scale*** *option*

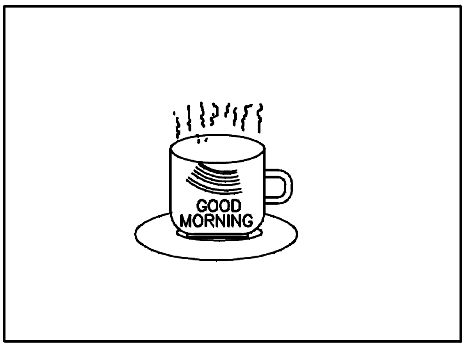

Figure 7-14 *Drawing after the* ***ZOOM Scale*** *option*

Scale: Relative to current view

The second way to scale is with respect to the current view (Figure 7-15). In this case, instead of entering only a number, enter a number followed by an **X**. The scale is calculated with reference to the current view. For example, if you enter **0.25X**, each object in the drawing will be displayed at one-fourth (¼) of its current size. The following example increases the display magnification by a factor of 2 relative to its current value (Figure 7-16).

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: **2X** Enter

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Scale: Relative to paper space units

The third method of scaling is with respect to paper space. You can use paper space in a variety of ways and for various reasons. For example, you can array and plot various views of your model in the paper space. To scale each view relative to paper space units, you can use the **ZOOM XP** option. Each view can have an individual scale. The drawing view can be at any scale of your choice in a model space viewport. For example, to display a model space at one-fourth (¼) the size of the paper space units, the prompt sequence is given next.

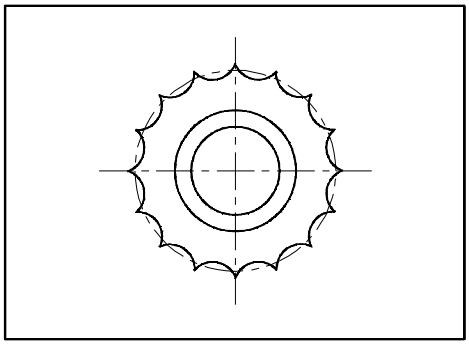

Figure 7-15 *Drawing, before the* ***ZOOM Scale*** *(X) option*

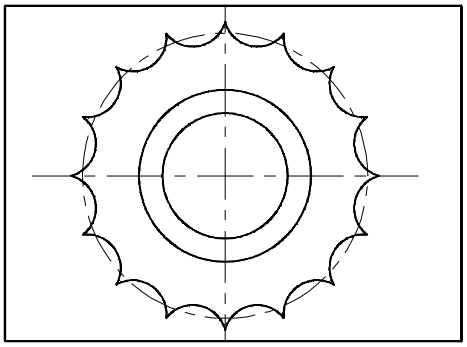

Figure 7-16 *Drawing, after the* ***ZOOM Scale*** *(X) option*

Command: **ZOOM** Enter
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object]<real time>: **1/4XP** Enter

Note

For a better understanding of this topic, refer to "Model Space Viewports, Paper Space Viewports and Layouts" in Chapter 11.

Object Option

The **Object** option of the **ZOOM** command is used to select one or more than one objects and display them at the at the center of the screen in the largest possible size.

Zoom In and Out

You can also zoom into the drawing using the **In** option, which doubles the image size.

Similarly, you can use the **Out** option to decrease the size of the image by half. To invoke these options from the command line, enter **ZOOM 2X** for the **In** option or **ZOOM .5X** for the **Out** option at the Command prompt. The center of the screen is taken as the reference point for enlarging or reducing the view of the drawing.

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Exercise 1

Mechanical

Draw Figure 7-17 according to the given dimensions. Use the **ZOOM** command to get a bigger view of the drawing. Do not dimension the drawing.

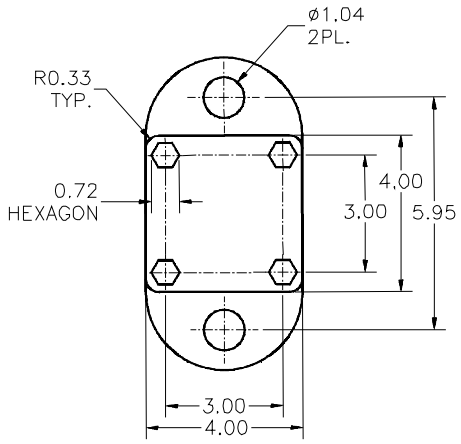

Figure 7-17 *Drawing for Exercise 1*

PANNING DRAWINGS

Toolbar:	Standard Toolbar > Pan Realtime
Menu:	View > Pan
Command:	PAN

You may want to view or draw on a particular area outside the current display. You can do this using the **PAN** command. If done manually, this would be like holding one corner of the drawing and dragging it across the screen. The **PAN** command allows you to bring into view the portion of the drawing that is outside the current display area. This is done without changing the magnification of the drawing. The effect of this command can be illustrated by imagining that you are looking at a big drawing through a window (**display window**) that allows you to slide the drawing right, left, up, and down to bring the part you want to view inside this window. You can invoke the **PAN** command from the shortcut menu also.

Panning in Realtime

You can use the **Realtime Pan** to pan the drawing interactively. To pan a drawing, invoke the command and then hold the pick button down and move the cursor in any direction. When you select the realtime pan, AutoCAD LT displays an image of a hand indicating that you are in the **PAN** mode. **Realtime pan** is the default setting for the **PAN** command. Choosing the **Pan Realtime** button in the toolbar and entering **PAN** at the Command prompt automatically invokes the realtime pan. To exit the realtime pan, press ENTER or ESC, or choose **Exit** from the shortcut menu.

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For the **PAN** command, there are various options that can be used to pan the drawing in a particular direction. These items can be invoked only from the menu shown in Figure 7-18.

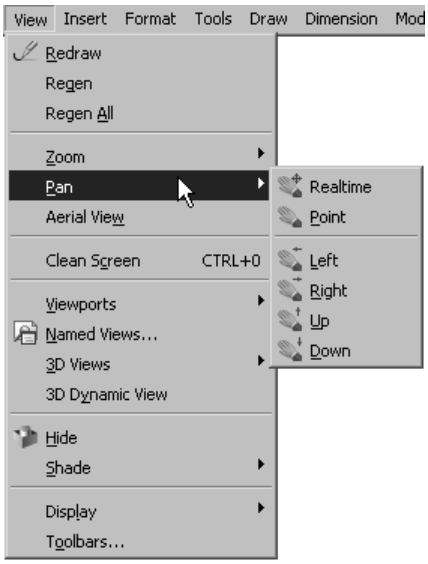

Figure 7-18 *Invoking the* ***PAN*** *command options from the* ***View*** *menu*

Point

In this option, you are required to specify the displacement. To do this, you need to specify in what direction to move the drawing and by what distance. You can give the displacement either by entering the coordinates of the points or by specifying the coordinates by using a pointing device. The coordinates can be entered in two ways. One way is to specify a single coordinate pair. In this case, AutoCAD LT takes it as a relative displacement of the drawing with respect to the screen. For example, in the following case, the **PAN** command would shift the displayed portion of the drawing 2 units to the right and 2 units up.

Command: *Select the* ***Point*** *option from the* ***View*** *menu.*
Specify base point or displacement: **2,2** Enter
Specify second point: Enter

In the second case, you can specify two coordinate pairs. AutoCAD LT computes the displacement from the first point to the second. Here, the displacement is calculated between point (3,3) and point (5,5).

Command: *Specify the Point item from the View menu.*
Specify base point or displacement: **3,3** Enter *(Or specify a point.)*
Specify second point: **5,5** Enter *(Or specify a point.)*

Left

Moves the drawing left so that some of the right portion of the drawing is brought into view.

Right

Moves the drawing right so that some of the left portion of the drawing is brought into view.

Up

Moves the drawing up so that some of the bottom portion of the drawing is brought into view.

Down

Moves the drawing down so that some of the top portion of the drawing is brought into view.

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Tip

*You can use the scroll bars to pan the drawing vertically or horizontally. The scroll bars are located at the right side and the bottom of the drawing area. You can control the display of the scroll bars in the **Display** tab of the **Options** dialog box.*

CREATING VIEWS

Toolbar:	View > Named Views
Menu:	View > Named Views
Command:	VIEW

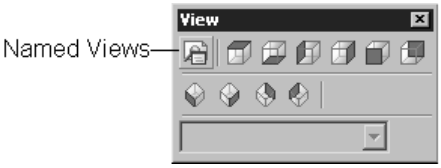

Figure 7-19 *Invoking **the Named Views** from the **View** toolbar*

While working on a drawing, you may frequently be working with the **ZOOM** and **PAN** commands, and you may need to work on a particular drawing view (some portion of the drawing) more often than others. Instead of wasting time by recalling your zooms and pans and selecting the same area from the screen over and over again, you can store the view under a name and restore it using the name you have given. Choose the **Named Views** button on the **View** toolbar (Figure 7-19) to invoke the **View** dialog box. This dialog box is used to save the current view under a name so that you can restore (display) it later. It does not save any drawing object data, only the view parameters needed to redisplay that portion of the drawing.

View Dialog Box

You can save and restore the views from the **View** dialog box shown in Figure 7-20. This dialog box is very useful when you are saving and restoring many view names. With this dialog box, you can name the current view or restore some other view. The **Named Views** tab lists all the created named views. The **Orthographic and Isometric Views** tab lists all the preset views of the drawing, and allows you to set current any of those views. The following are the various options in the **Named Views** tab of the **View** dialog box.

Current View

The **View** list box displays a list of the named views in the drawing. The list appears with the names of all saved views and the space, in which each was defined (Model space and paper space are discussed in Chapter 11).

New

The **New** button allows you to create a new view and save it by giving it a name. When you choose the **New** button, the **New View** dialog box is displayed, as shown in Figure 7-21.

View name. You can enter the name for the view in the **View name** edit box.

View category. You can specify the category of the view from the **View category** edit box. The categories include the front view, top view, and so on. If there are some existing categories, you can select them from this drop-down list also.

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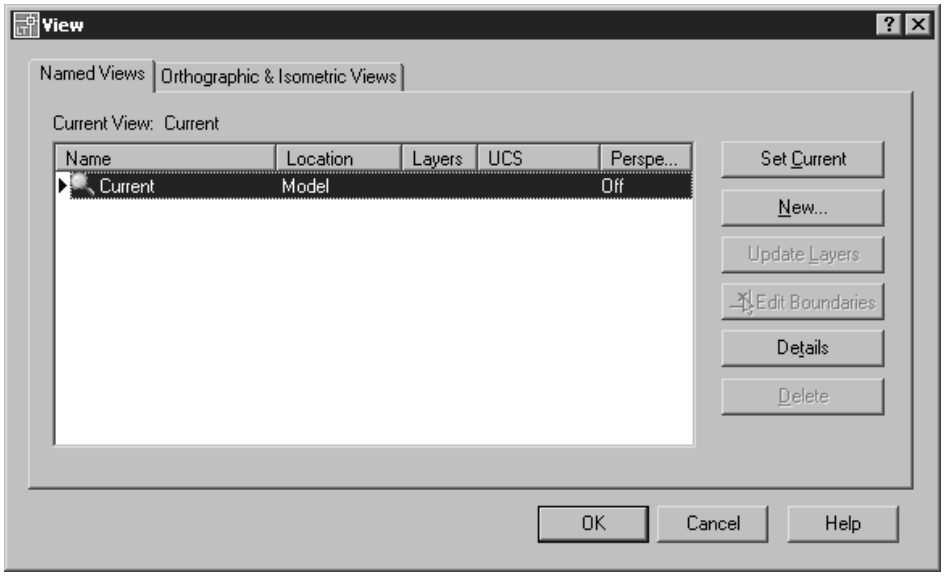

Figure 7-20 *The* ***View*** *dialog box*

Boundary Area. This area is used to specify the boundary of the view. If you want to save the current display as the view, select the **Current display** radio button. If you want to define a window that will specify the new view (without first zooming in on that area), select the **Define window** radio button. As soon as you select this radio button, the dialog boxes will be temporarily

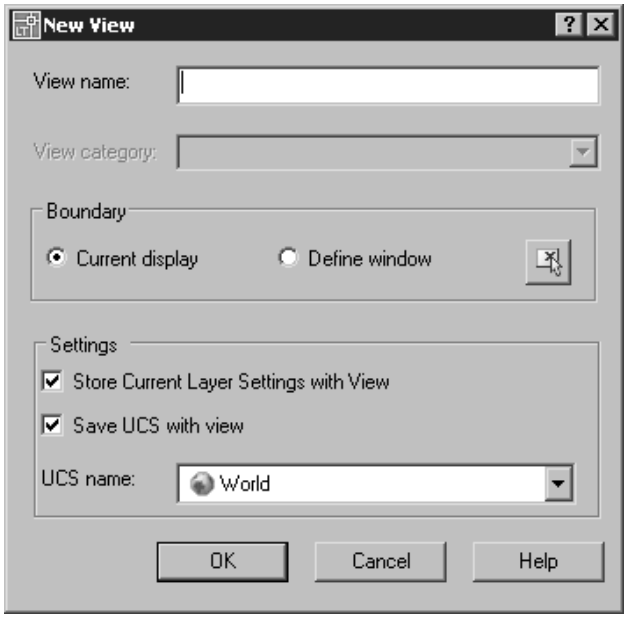

Figure 7-21 *The* ***New View*** *dialog box*

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closed and you will be prompted to define two corners of the window. You can modify the window by choosing the **Define View Window** button. You can also enter the *X* and *Y* coordinates in the **Specify first corner** and **Specify other corner** in the command lines.

Settings Area. This area allows you to save the layers and the UCS with the new view. If you want to save the settings of the visibility of the current layers with the view, select the **Store Current Layer Settings with View** radio button. You can select the **Save UCS with view** radio button to save a UCS with the view. The UCS to be saved with the view can be selected from the **UCS name** drop-down list.

Set Current

The **Set Current** button allows you to replace the current viewport by the view you specify. AutoCAD LT uses the center point and magnification of each saved view and executes a **ZOOM Center** with this information when a view is restored.

Update Layers

The **Update Layers** button is chosen to update the layer information saved with an existing view.

Edit Boundaries

The **Edit Boundaries** button is chosen to edit the boundary that was defined using the **New View** dialog box while creating the new view.

Details

You can also see the description of the general parameters of a view by selecting the particular view and then choosing the **Details** button. When you choose this button, the **View Details** dialog box is displayed.

Tip

You can use the shortcut menu to rename or delete any named view in the dialog box. You can also update the layer information or edit the boundary of a view using this shortcut menu.

Using the Command prompt

You can also use the **-VIEW** command to work with views at the Command prompt.

Command: **-VIEW** Enter
Enter an option [?/lAyer state/Orthographic/Delete/Restore/Save/Ucs/Window]:

You can use the various options to specify the view category, layer state, save, restore, delete, or list the views. If you try to restore a Model space view while working in layouts, AutoCAD LT will prompt you to select a viewport and then automatically switch to the floating model space using that viewport. In this case, AutoCAD LT will prompt you further as follows.

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Enter view name to restore: *Specify the name of the Model space view*
Restoring Model space view.
Select Viewport for view: *Select the viewport in which the Model space view should be restored.*

You can select the viewport you want by selecting its border. This particular viewport must be on and active. The restored viewport also becomes the current one.

You can use the Window option to define the view. The prompt sequence is given next.

Command: **-VIEW** Enter
Enter an option [?/lAyer state/Orthographic/Delete/Restore/Save/Ucs/Window]:
W Enter
Enter view name to save: *Enter the name.*
Specify first corner: *Specify a point.*
Specify opposite corner: *Specify another point.*

Tip
*The options of the **VIEW** command can be used transparently by entering '**-VIEW** at the Command prompt.*

ARIEL VIEW

Menu:	View > Aerial View
Command:	DSVIEWER

As a navigational tool, AutoCAD LT provides you with the option of opening another drawing display window along with the graphics screen window you are working on. This window, called the **Aerial View** window, can be used to view the entire drawing and select those portions you want to quickly zoom or pan. The AutoCAD LT graphics screen window resets itself to display the portion of the drawing you have selected for Zoom or Pan in the **Aerial View** window. You can keep the **Aerial View** window open as you work on the graphics screen, or minimize it so that it stays on the screen as a button which can be restored when required (Figure 7-22). As with all windows, you can resize or move it. The **Aerial View** window can also be invoked when you are in the midst of any command other than the **DVIEW** command. The **Aerial View** window has two menus, **View** and **Options**. It also has a toolbar containing zoom in, zoom out, and zoom global options. The following is a description of the available options in the **Aerial View**.

Toolbar Buttons

The **Aerial View** window has three buttons: **Zoom In**, **Zoom Out**, and **Global** (Figure 7-23).

Zoom In

This option leads to the magnification by a factor of **2** centered on the current view box.

Zoom Out

This option leads to the reduction by **half** centered on the current view box.

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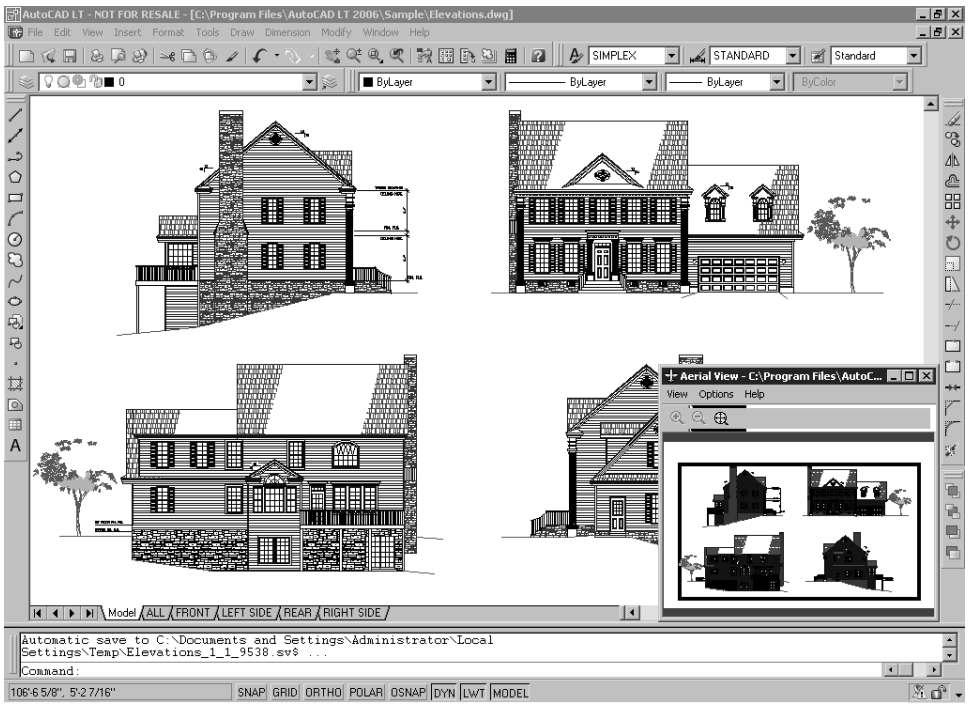

Figure 7-22 *AutoCAD LT graphics screen with the* ***Aerial View*** *window*

Global

The **Global** option displays the complete drawing in the **Aerial View** window.

Menus

The menus in the menu title bar are **View** and **Options**. The **View** menu contains **Zoom In**, **Zoom Out**, and **Global** options, which are the same as their respective buttons in the toolbar. The options in the **Options** menu are discussed next.

Auto Viewport

When you are working with viewports, you may need to change the view in the **Aerial View** window to display the current viewport view. You can achieve this by selecting the **Auto Viewport** option from the **Options** menu. If you are working in multiple viewports and you zoom or pan in the **Aerial View** window, only the view in the current viewport is affected.

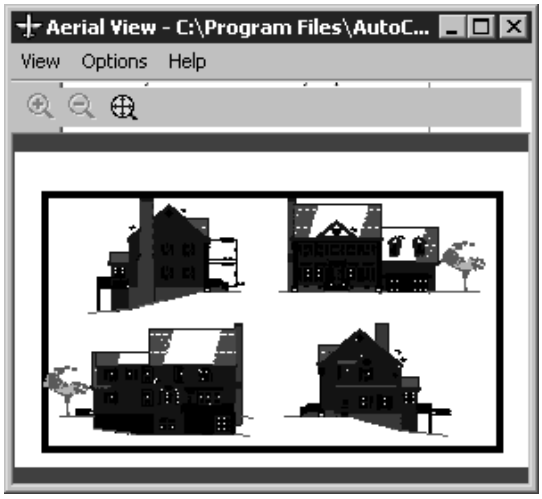

Figure 7-23 *The* ***Aerial View*** *window*

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Dynamic Update

When you make any changes in the current drawing, the view box is updated simultaneously in the **Aerial View** window if you have selected the **Dynamic Update** option. If **Dynamic Update** is unselected, the drawing is not updated simultaneously. The drawing in the **Aerial View** window is updated only when you move the cursor in the **Aerial View** window or invoke any of the menus or the toolbar buttons.

Realtime Zoom

This option dynamically updates the view in the drawing area when you are zooming in the **Aerial View** window in realtime.

CREATING TEXT

In manual drafting, lettering is accomplished by hand using a lettering device, pen, or pencil. This is a very time-consuming and tedious job. Computer-aided drafting has made this process extremely simple. Engineering drawings invoke certain standards to be followed in connection with the placement of a text in a drawing. In this section, you will learn how text can be added in a drawing by using **TEXT** and **MTEXT** commands.

Writing Single Line Text

Toolbar:	Text > Single Line Text
Menu:	Draw > Text > Single Line Text
Command:	TEXT

The **TEXT** command lets you write text on a drawing and allows you to delete, what has been typed, by using the BACKSPACE key. You can enter multiple lines in one command. The **TEXT** command displays a line in the drawing area where you specified the start point after entering the height and the rotation angle. This line identifies the start point and the size of the text height entered. The characters appear on the screen as you enter them. When you press ENTER after typing a line, the cursor automatically places itself at the start of the next line and you can enter the next line of the text. You can end the command by pressing the ENTER key or ESC key at the **Enter text** prompt.

The screen crosshairs can be moved irrespective of the cursor line for the text. If you specify a point, this command will complete the current line of the text and move the cursor line to the point you selected. This cursor line can be moved and placed anywhere on the screen; therefore, multiple lines of text can be entered at any desired location in the drawing area with a single **TEXT** command. By pressing BACKSPACE you can delete one character to the left of the current position of the cursor box. Even if you have entered several lines of text, you can use BACKSPACE and continue deleting, until you reach the start point of the first line.

This command can be used with most of the text alignment modes, although it is most useful in the case of left-justified texts. In the case of aligned texts, this command assigns a height appropriate for the width of the first line to every line of the text. Irrespective of the **Justify** option chosen, the text is first left-aligned at the selected point. The prompt sequence that follows when you choose this command is given next.

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Specify start point of text or [Justify/Style]: *Specify the start point.*
Specify height <0.2000>: **0.15** Enter
Specify rotation angle of text <0>: Enter
Enter text: *Enter first line of the text*
Enter text: *Enter first line of the text*
Enter text: Enter

Start Point Option

This is the default and the most commonly used option in the **TEXT** command. By specifying a start point, the text is left-justified along its baseline starting from the location of the starting point. Before AutoCAD LT draws the text, it needs the text height, the rotation angle for the baseline, and the text string to be drawn. It prompts you for all this information. The prompt sequence is given next.

Specify height <default>:
Specify rotation angle of text <default>
Enter text:

The **Specify height** prompt determines the distance by which the text extends above the baseline, measured by the capital letters. This distance is specified in drawing units. You can specify the text height by specifying two points or entering a value. In the case of a null response, the default height, that is, the height used for the previous text drawn in the same style, will be used.

The **Specify rotation angle of text** prompt determines the angle at which the text line will be drawn. The default value of the rotation angle is **0-degree** (3 o'clock, or east), and in this case the text is drawn horizontally from the specified start point. The rotation angle is measured in a counterclockwise direction. The last angle specified becomes the current rotation angle, and if you give a null response, the last angle specified will be used as default. You can also specify the rotation angle by specifying two points. The text is drawn upside down if a point is specified at a location to the left of the start point.

As a response to the **Enter text** prompt, enter the text string. Spaces are allowed between words. After entering the text, press ENTER.

Justify Option

AutoCAD LT offers various options to align the text. **Alignment** refers to the layout of the text. The main text alignment modes are **left, center,** and **right**. You can align a text using a combination of modes; for example, top/middle/baseline/bottom and left/center/right (Figure 7-24). **Top** refers to the line along which lie the top points of the capital letters; **Baseline** refers to the line along which lie their bases. Letters with descenders (such as p, g, y) dip below the baseline to the bottom.

When the **Justify** option is invoked, the user can place the text in one of the fourteen various alignment types by selecting the desired alignment option using the command prompt or the dynamic preview (Figure 7-25). The orientation of the text style determines the command interaction for Text Justify. (Text styles and fonts are discussed later in this chapter). For now,

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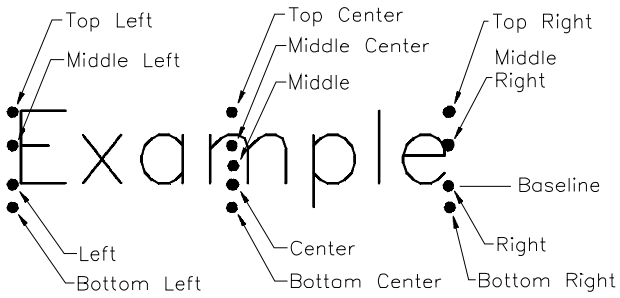

Figure 7-24 *Text alignment positions*

assume that the text style orientation is horizontal. The prompt sequence that will follow when you choose the **Single Line Text** button to use this option is given next.

Specify start point of text or [Justify/style]: **J** Enter
Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/ BR]: *Select any of these options.*

If the text style is vertically oriented (refer to the "Creating Text Styles" section later in this chapter), only four alignment options are available. The prompt sequence is as follows.

Specify start point of text or [Justify/Style]: **J** Enter
Enter an option [Align/Center/Middle/Right]:

If you know what justification you want, you can enter it directly at the **Specify start point of text or [Justify/Style]** prompt instead of first entering **J** to display the justification prompt. If you need to specify a style as well as a justification, you must specify the style first. The various alignment options are as follows.

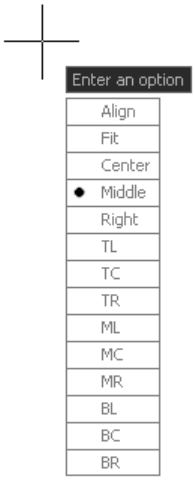

Figure 7-25 *Dynamic preview for the* ***Justify*** *option*

Align Option

In this option, the text string is written between two points (Figure 7-25). You must specify the two points that act as the endpoints of the baseline. The two points may be specified horizontally or at an angle. AutoCAD LT adjusts the text width (compresses or expands) so that it fits between the two points. The text height is also changed, depending on the distance between points and the number of letters.

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Specify start point of text or [Justify/Style]: **J** Enter
Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]: **A** Enter
Specify first endpoint of text baseline: *Specify a point.*
Specify second endpoint of text baseline: *Specify a point.*
Enter text: *Enter the text string.*

Fit Option

This option is very similar to the previous one. The only difference is that in this case, you select the text height, and it does not vary according to the distance between the two points. AutoCAD LT adjusts the letter width to fit the text between the two given points, but the height remains constant (Figure 7-27). The **Fit** option is not accessible for vertically oriented text styles. If you try the **Fit** option on the vertical text style, you will notice that the text string does not appear in the prompt. The prompt sequence is given next.

Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]: **F**
Specify first endpoint of text baseline: *Specify a point.*
Specify second endpoint of text baseline: *Specify a point.*
Specify height<current>: *Enter the height.*
Enter text: *Enter the text.*

Note

*You do not need to select the **Justify** option (J) for selecting the text justification. You can enter the text justification by directly entering justification when AutoCAD LT prompts "Specify start point of text or [Justify/Style]:"*

Center Option

You can use this option to select the midpoint of the baseline for the text. This option can be invoked by entering **Justify** and then **Center** or **C**. After you select or specify the center point, you must enter the letter height and the rotation angle (Figure 7-25).

Specify start point of text or [Justify/Style]: C Enter
Specify center point of text: *Specify a point.*
Specify height<current>: **0.15** Enter
Specify rotation angle of text<0>: Enter
Enter text: **CENTER JUSTIFIED TEXT** Enter

Middle Option

Using this option, you can center text not only horizontally, as with the previous option, but also vertically. In other words, you can specify the middle point of the text string (Figure 7-25). You can alter the text height and the angle of rotation to your requirement. The prompt sequence that will follow when you choose the **Single Line Text** button from the **Text** toolbar is given next.

Specify start point of text or [Justify/Style]: **M** Enter

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Specify middle point of text : *Specify a point.*
Specify height<current>: **0.15** Enter
Specify rotation angle of text<0>: Enter
Enter text: **MIDDLE JUSTIFIED TEXT** Enter

Right Option

This option is similar to the default left-justified start point option. The only difference is that the text string is aligned to the lower right corner (the endpoint you specify); that is, the text is **right-justified** (Figure 7-26). The prompt sequence that will follow when you choose this button is given next.

Specify start point of text or [Justify/Style]: R Enter
Specify right endpoint of text baseline: *Specify a point.*
Specify height<current>: **0.15** Enter
Specify rotation angle of text<0>: Enter
Enter text: **RIGHT JUSTIFIED TEXT** Enter

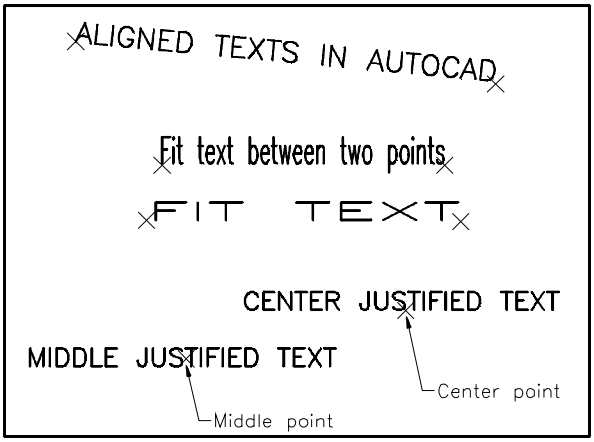

Figure 7-26 *Writing the text using the* ***Align****,* ***Fit****,* ***Center****, and* ***Middle*** *options*

TL Option

In this option, the text string is justified from the **top left** (Figure 7-27). The prompt sequence is given next.

Specify start point of text or [Justify/Style]: **TL** Enter
Specify top-left point of text: *Specify a point.*
Specify height<current>: **0.15** Enter
Specify rotation angle of text <0>: Enter
Enter text: **TOP/LEFT JUSTIFIED TEXT** Enter

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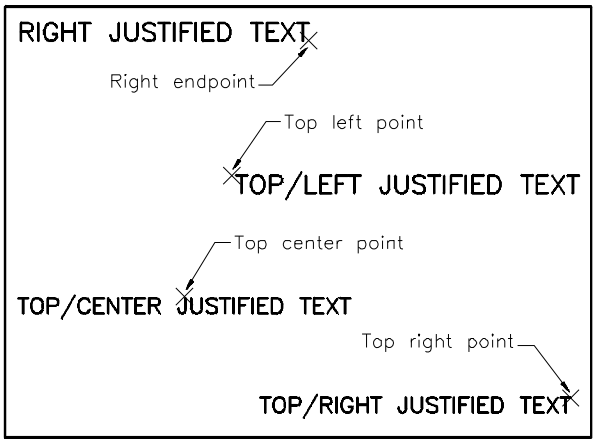

Figure 7-27 *Writing text using the Right, Top-Left, Top-Center, and Top-Right options*

Note

The rest of the text alignment options are similar to those just discussed, and you can try them on your own. The prompt sequence is almost the same as those given for the previous examples.

Style Option

With this option, you can specify another existing text style. Different text styles can have different text fonts, heights, obliquing angles, and other features. This option can be invoked by entering **TEXT** and then S at the next prompt. The prompt sequence that will follow when you choose the **Single Line Text** button for using this option is given next.

Specify start point of text or [Justify/Style]: **S** Enter
Enter style name or [?] <current>:

If you want to work in the previous text style, just press ENTER at the last prompt. If you want to activate another text style, enter the name of the style at the last prompt. You can also choose from a list of available text styles, which can be displayed by entering **?**. After you enter **?**, the next prompt is given next.

Enter text style(s) to list <*>:

Press ENTER to display the available text style names and the details of the current styles and commands in the **AutoCAD LT Text Window**.

Note

*With the help of the **Style** option of the **TEXT** command, you can select a text style from an existing list. If you want to create a new style, use the **STYLE** command, which is explained later in this chapter.*

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ENTERING SPECIAL CHARACTERS

In almost all drafting applications, you need to draw **special characters** (symbols) in the normal text and in the dimension text. For example, you may want to draw the degree symbol (°) or the diameter symbol (ø), or you may want to underscore or overscore some text. This can be achieved with the appropriate sequence of control characters (control code). For each symbol, the control sequence starts with a percent sign written twice (%%). The character immediately following the double percent sign depicts the symbol. The control sequences for some of the symbols are given next.

Control sequence	Special character
%%c	Diameter symbol (ø)
%%d	Degree symbol (°)
%%p	Plus/minus tolerance symbol (±)
%%o	Toggle for overscore mode on/off
%%u	Toggle for underscore mode on/off
%%%	Single percent sign (%)

For example, if you want to draw **25° Celsius**, you need to enter 25%%dCelsius. If you enter **43.0%%c**, you get **43.0ø** on the drawing screen. To underscore (underline) text, use the **%%u** control sequence followed by the text to be underscored. For example, to underscore the text: **UNDERSCORED TEXT IN AUTOCAD LT**, enter **%%uUNDERSCORED TEXT IN AUTOCAD LT** at the prompt asking for text to be entered. To underscore and overscore a text string, include **%%u%%o** at the text string.

Note

*The special characters %%o and %%u act as toggles. For example, if you enter "**This %%utoggles%%u the underscore**", the word **toggles** will be underscored (toggles).*

None of these codes will be translated in the **TEXT** command until this command is complete. For example, to draw the degree symbol, you can enter **%%d**. As you are entering these symbols, they will appear as %%d on the screen. After you have completed the command and pressed ENTER, the code %%d will be replaced by the degree symbol (°).

You may be wondering why a percent sign should have a control sequence when a percent sign can easily be entered at the keyboard by pressing the percent (%) key. The reason is that sometimes a percent symbol is immediately followed (without a space) by a control sequence. In this case, the **%%%** control sequence is needed to draw a single percent symbol. To make the concept clear, assume you want to draw **67%±3.5.** Try drawing this text string by entering **67%%%p3.5.** The result will be **67%p3.5,** which is wrong. Now enter **67%%%%%p3.5** and notice the result on the screen. Here you obtain the correct text string on the screen, that is, **67%±3.5**. If there were a space between 67% and ±3.5, you could enter 67% %%p3.5 and the result would be **67% ±3.5**.

In addition to the control sequences shown earlier, you can use the %%nnn control sequence to draw special characters. The nnn can take a value in the range of 1 to 126. For example, to draw the & symbol, enter the text string **%%038**.

CREATING MULTILINE TEXT*

Toolbar:	Draw, Text > Multiline Text
Tool Palette:	Command Tools > MText
Menu:	Draw > Text > Multiline Text
Command:	MTEXT

You can use the **MTEXT** command (Figure 7-28) to write a multiline text whose width can be specified by defining two corners of the text boundary or by entering a width, using the coordinate entry. The text created by the **MTEXT** command is a single object regardless of the number of lines it contains.

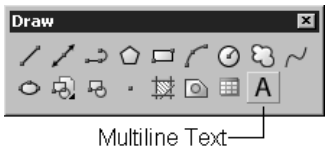

Figure 7-28 *Invoking the* ***Multiline Text*** *from the* ***Draw*** *toolbar*

Note

When you invoke the MTEXT command, a sample text is attached to the cursor. By default, the text "abc" is attached to the cursor. The MTJIGSTRING system variable stores the default contents of the sample text. You can specify a string of ten alphanumeric characters as the default sample text.

After specifying the width, you have to enter the text in the **In-Place Text Editor**. The prompt sequence is given next.

Command: **MTEXT** Enter
Current text style: "Standard". Text height: 0.2000
Specify first corner: *Select a point to specify first corner.*
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: *Select an option or select a point to specify other corner.*

After selecting the first corner, you can move the pointing device so that a box that shows the location and size of the paragraph text is formed. An arrow is displayed within the boundary, that indicates the direction of the text flow. When you define the text boundary, it does not mean that the text paragraph will fit within the defined boundary. AutoCAD LT only uses the width of the defined boundary as the width of the text paragraph. The height of the text boundary has no effect on the text paragraph. Once you have defined the boundary of the paragraph text, AutoCAD LT displays the **In-Place Text Editor**, as shown in Figure 7-29.

Note

Although the box boundary that you specify controls the width of the paragraph, a single word is not broken to adjust inside the boundary limits. This means that if you write a single word whose width is more than the box boundary specified, AutoCAD LT will write the word irrespective of the box width, and therefore, will exceed the boundary limits.

The **In-Place Text Editor** consists of the **Text formatting toolbar**, **Text window** (with a ruler at the top), and **Options shortcut menu**. The following is the description of the options in the **Text formatting** toolbar and the **Options shortcut menu**.

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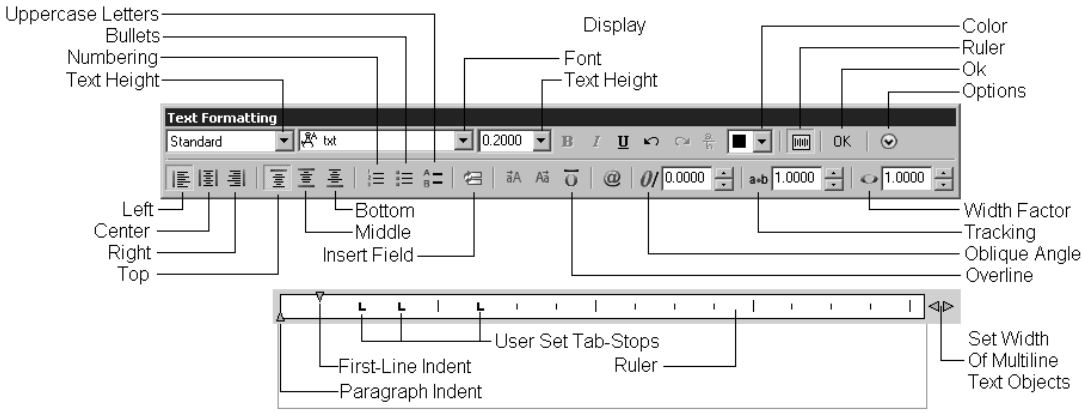

Figure 7-29 *The* ***In-Place Text Editor***

Text Formatting Toolbar

The options under this toolbar are as follows.

Style

The **Style** drop-down list is the first drop-down list on the left in the **Text Formatting** toolbar. This drop-down list contains a list of all the text styles created in the current drawing. You can select the desired text style from this drop-down list. You can create a new text style using the **STYLE** command, which is explained later in this chapter.

Font

The **Font** drop-down list displays all the fonts in AutoCAD LT. You can select the desired font from this drop-down list, see Figure 7-30.

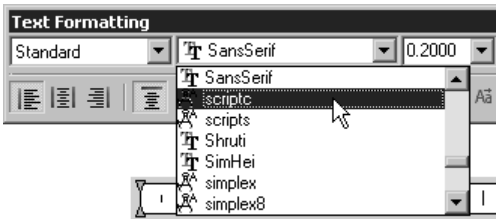

Figure 7-30 *The* ***Font*** *drop-down list*

Note

Irrespective of the font assigned to a text style, you can assign a different font to that style for the current multiline text using the ***Font*** *drop-down list.*

Text Height

The **Text Height** edit box is used to specify the text height of the multiline text. The default value in this edit box is 0.2000. Once you modify the height, AutoCAD LT retains that value unless you change it. Remember that the **MTEXT** height does not affect the height specified for the **TEXT** command.

Bold, Italic, Underline

You can use the appropriate tool buttons on the text box to make the selected text boldface, or italics, or create underlined text. Boldface and italics are not supported by SHX fonts and hence will not be available for the particular fonts. These three buttons toggle between on and off.

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Undo

The **Undo** button allows you to undo the actions in the **In-Place Text Editor**. You can also press the CTRL+Z keys to undo the previous actions.

Redo

The **Redo** button allows you to redo the actions in the **In-Place Text Editor**. You can also press the CTRL+Y keys to redo the previous actions.

Stack

To create a fraction text, you must use the stack button with special characters /, ^, and #. The character / stacks the text vertically with a line, and the character ^ stacks the text vertically without a line (tolerance stack). The character # stacks the text with a diagonal line. After you enter the text with the required special character between them, select the text, and then select the **Stack/Unstack** button. The stacked text is displayed equal to 70 percent of the actual height. If you enter two numbers separated by / or ^ and then press the ENTER key or the SPACEBAR, AutoCAD LT displays the **AutoStack Properties** dialog box, as shown in Figure 7-31. You can use this dialog box to control the stacking properties.

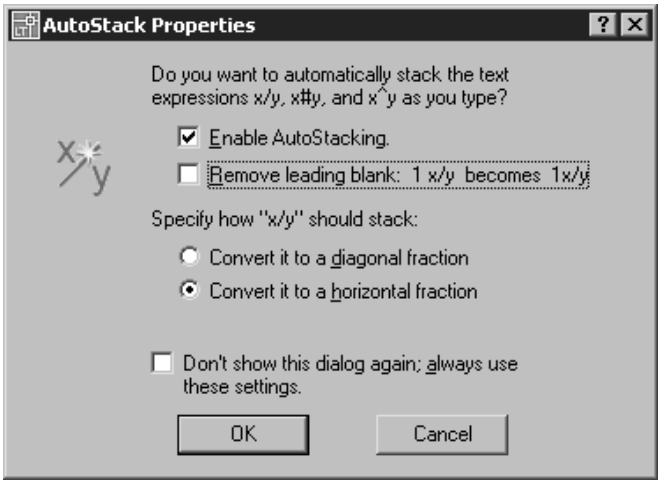

Figure 7-31 *The* ***AutoStack Properties*** *dialog box*

Color

The **Color** drop-down list is used to set the color for the multiline text. You can also select the color from the **Select Color** dialog box that is displayed by selecting **Select Color** in the drop-down list.

Ruler

This button is used to turn on or off the display of ruler in the **Text Window**.

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Left
This button is used to left align the text written in the **Text Window**.

Center
This button is used to center align the text written in the **Text Window**.

Right
This button is used to right align the text written in the **Text Window**.

Top
This button is used to top align the first row of the text written in the **Text Window**.

Middle
This button is used to middle align the text written in the **Text Window**.

Bottom
This button is used to bottom align the last row of the text written in the **Text Window**.

Numbering
This button is used to assign numbers (numerics like 1,2,3,....) to each row of the multiline text.

Bullets
This button is used to assign bullets to each row of the multiline text.

Uppercase Letters
This button is used to assign upper case alphabets to each row of the multiline text.

Uppercase
When you select text in the text editor and choose this button, the alphabets written in the lowercase are converted to the uppercase.

Insert Field
AutoCAD allows you to insert a field in the multiline text. The field contains data that is associative to the property that defines the field. For example, you can insert a field that has the name of the author of the current drawing. If you have already defined the author of the current drawing in the **Drawing Properties** dialog box, it will be automatically displayed in the field. If you modify the author and update the field, the changes are automatically made in the text. When you choose this option, the **Field** dialog box is displayed, as shown in Figure 7-32. You can select the field to be added from the **Field names** list box and define the format of the field using the **Format** list box. Choose **OK** after selecting the field and format. If the data in the selected field is already defined, it will be displayed in the **Text window**. If not, the field will display dashes (----).

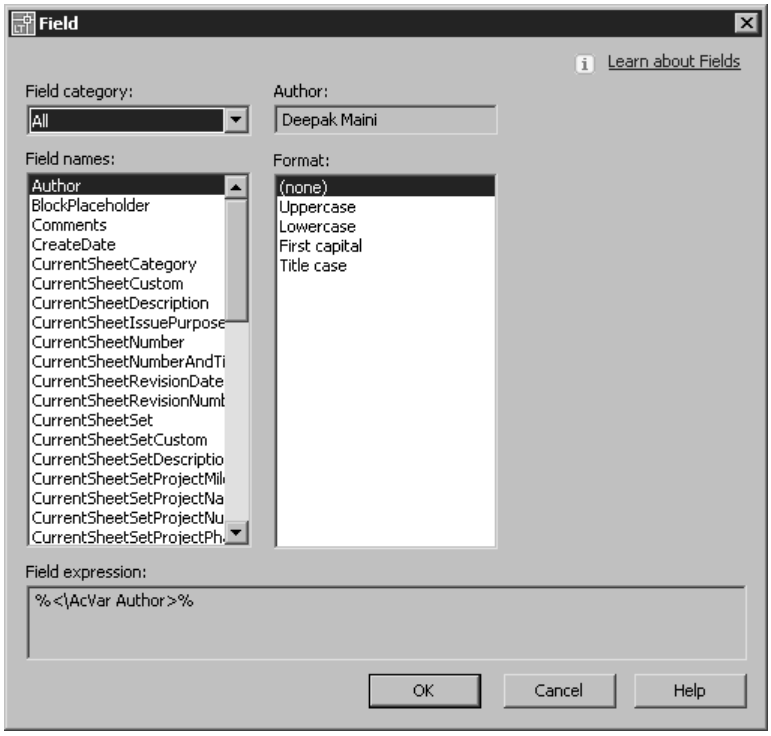

Figure 7-32 *The* ***Field*** *dialog box*

Note

To update a field after the text of that field is modified, double-click on the multiline text to display the ***In-Place Text Editor****. Click on the field in the* ***Text window*** *to select it and then right-click to display the shortcut menu. Choose* ***Update Field*** *from this menu; the field will be updated. You can also edit the field or convert it into text using the same shortcut menu.*

Lowercase

Choosing this button converts the alphabets written in the uppercase to the lowercase.

Overline

Choosing this button forces an overline on the top of text. You can also select the existing text and then choose this button to place a line on top of it.

Symbol

This option is used to insert the special characters in the text. When you choose **Symbol** button from the **Text Formatting** toolbar, a shortcut menu (Figure 7-33) appears that displays some predefined special characters. You can also choose **Other** from the shortcut menu to display the **Character Map** dialog box. This dialog box has a number of other special characters that you can insert in multiline text. To insert the characters from the dialog box, select the character you want to copy and then choose the **Select** button. Once you have selected all the required special characters, choose the **Copy** button and then close the dialog box. Now, in the **Text**

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window, position the cursor where you want to insert the special characters and right-click to display the shortcut menu. Choose **Paste** to insert the selected special character in the **In-Place Text Editor**.

Oblique Angle

This option is used to specify the slant angle for the text. Enter the angle in the **Oblique Angle** edit box or use spinner to specify the slant angle, which is measured from the positive direction of the X-axis, in the clockwise direction.

Tracking

This option is used to control the spacing between the selected characters. Select the characters and specify the value of the spacing in the **Tracking** edit box. You can also use the **Tracking** spinner to specify this value. The default tracking value is 1.000. You can specify values ranging from 0.7500 to 4.0000.

WIdth Factor

This option is used to control the width of characters. By default, the value is set to 1.000. Select the characters and use the **Width Factor** edit box or the **Width Factor** spinner to specify the width of the characters.

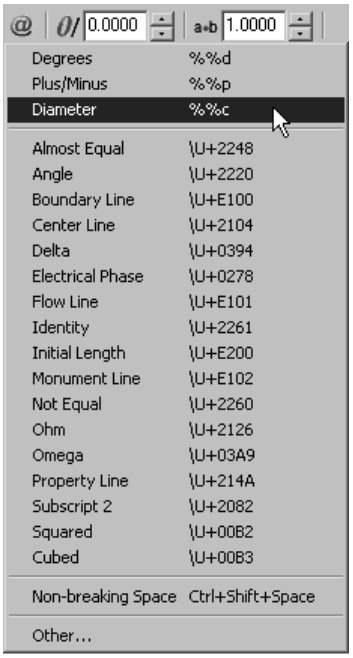

Figure 7-33 *Shortcut menu invoked when you choose the* ***Symbol*** *button from the* ***Text Formatting*** *toolbar*

Text Window

The **Text window** is used to enter the multiline text. The width of the active text area is determined by the width of the window that you specify when you invoke the **MTEXT** command. You can increase the size of the dialog box by dragging the right or the bottom edge of the text window. You can also use the scroll bar to move up or down to display the text.

The ruler on the top of the text window is used to specify the indentation of the current paragraph. The top slider of the ruler specifies the indentation of the first line of the paragraph while the bottom slider specifies the indentation of the other lines of the paragraph.

Options Shortcut Menu

The **Options** shortcut menu provides various options to edit the multiline text. To edit the text, select it and then choose the **Options** button form the **Text Formatting** toolbar. The shortcut menu (Figure 7-34) is displayed. The various options are discussed next.

Show Toolbar

This option is used to control the visibility of the **Text Formatting** toolbar.

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Show Options

This option is used to control the visibility of the advanced options in the **Text Formatting** toolbar. If this option is not selected, the second row of buttons will not be available in the **Text Formatting** toolbar.

Show Ruler

The **Show Ruler** option is used to control the visibility of the **Ruler** in the **Text Formatting Window**.

Opaque Background

Select this option to make the background of the **Text Window** opaque. By default, the background is transparent. The effect of the background (opaque/transparent) will be more visible if the text window is invoked with some entities existing in the drawing window.

Import Text

When you choose this option, AutoCAD LT displays the **Select File** dialog box. In this dialog box, you can select any text file you want to import as the multiline text. The imported text is displayed in the text area. Note that only the ASCII or RTF files are interpreted properly.

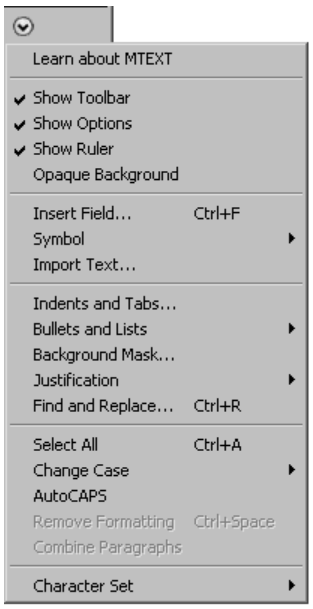

Figure 7-34 *The* ***Options*** *shortcut menu*

Indents and Tabs

The **Indents and Tabs** option allows you to set the indentation of the multiline text and the tab position. Note that as mentioned earlier, these options can also be set using the ruler and the sliders available on top of the drawing window. When you choose **Indents and Tabs** from the shortcut menu, the **Indents and Tabs** dialog box is displayed, as shown in Figure 7-35. The various options in the dialog box are discussed next.

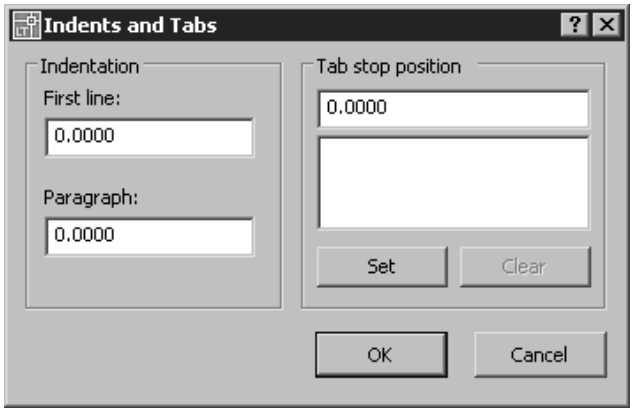

Figure 7-35 *The* ***Indents and Tabs*** *dialog* box

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Indentation Area. You can set the indentation for the first line and the following lines of the multiline text by entering the values in the **First Line** and the **Paragraph** edit boxes.

Tab stop position Area. The **Tab stop position** area is used to set the position up to which the cursor will move when you press the TAB key once at the starting of a new paragraph. This position is called the tab position. To set the tab position, enter the value in the edit box in this area and then choose the **Set** button. You will notice that the value appears in the list box below this area. You can choose the **Clear** button to delete the selected tab position value. Notice that a mark appears in the ruler that defines the tab stop position.

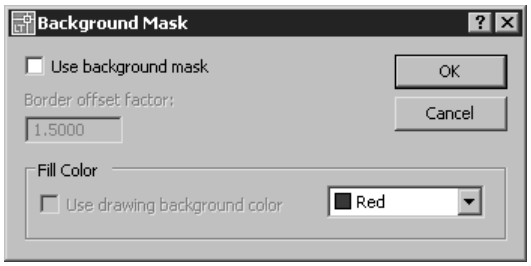

*Figure 7-36 The **Background Mask** dialog box*

Background Mask

This option is used to define a background color for the multiline text. When you choose this option, the **Background Mask** dialog box is displayed, as shown in Figure 7-36. To add a background mask, select the **Use background mask** check box and select the required color from the drop-down list in the **Fill Color** area. You can set the size of the colored background behind the text using the **Border offset factor** edit box. The box that defines the background color will be offset from the text by the value you define in this edit box. The value in this edit box is based on the height of the text. If you enter 1 as the value, the height of the colored background will be equal to the height of the text and will extend through the length of the window defined to write the multiline text. Similarly, if you enter 2 as the value, the height of the colored box will be twice the height of the text and will be equally offset above and below the text. However, the length of the colored box will still be equal to the length of the window defined to write the multiline text. You can select the **Use background** check box in the **Fill Color** area to use the color of the background of the drawing area to add the background mask.

Justification

In large complicated technical drawings, the **Justification** option is used to fit the text matter with a specified justification and alignment. For example, if the text justification is bottom-right (BR), the text paragraph will spill to the left and above the insertion point, regardless of how you define the width of the paragraph. When you select **Justification** from the shortcut menu, a cascading menu appears that displays the predefined text justifications. By default, the text is Top Left justified. You can choose the new justification from the cascading menu. The various justifications are TL, ML, BL, TC, MC, BC, TR, MR, BR. Figure 7-37 shows various text justifications for multiline text.

Find and Replace

When you choose this option, the **Find and Replace** dialog box is displayed, as shown in Figure 7-38. The options available in this dialog box are discussed next.

Find what. You can define the text string that you need to find in this text box. You can locate a part of a word or the complete word mentioned in this text box.

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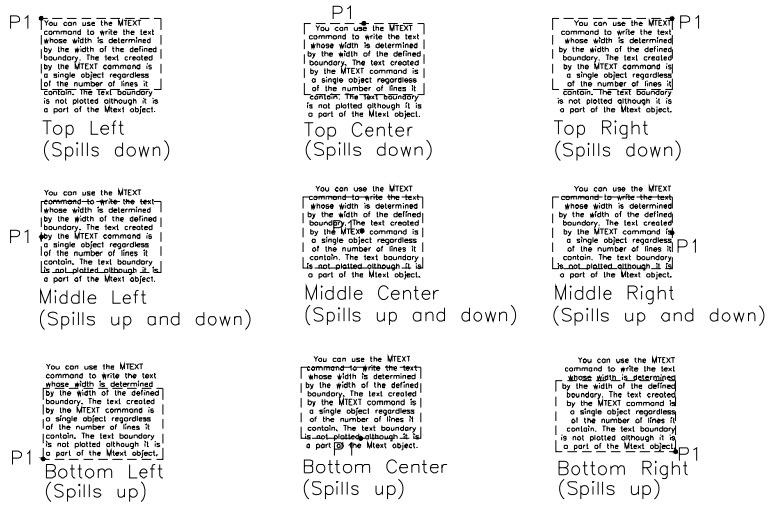

Figure 7-37 *Text justifications for* ***MTEXT*** *(P1 is the text insertion point)*

Replace with. If you want some text to be replaced, enter new text for replacement in this text box.

Match whole word only. If this check box is selected, AutoCAD LT will match the word in the text only if it is a single word identical to that mentioned in the **Find what** text box. For example, if you enter **and** in the **Find what** text box and select the **Match whole word only** check box, AutoCAD LT will not match the string **and** in words like sand, land, band, and so on.

Match case. If this check box is selected, AutoCAD LT will find the word only if the cases of all the characters in the word are identical to that of the word mentioned in the **Find what** text box.

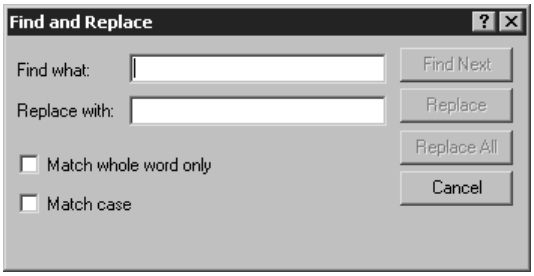

Figure 7-38 *The* ***Find and Replace*** *dialog box*

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Find Next. Choose this button to continue the search for the text entered in the **Find what** box.

Replace. Choose this button to replace the highlighted text with the text entered in the **Replace with** text box.

Replace All. If you choose this button, all the words in the current multiline text that match the word specified in the **Find what** text box will be replaced with the word entered in the **Replace with** box.

Select All

This option is used to select the complete text entered in the **Text window**.

Change Case

This option is used to change the case of the selected text to uppercase or lowercase.

AutoCAPS

If you choose this option, the case of all the text written or imported after choosing this option will be changed to uppercase. However, the case of the text written before choosing this option is not changed.

Remove Formatting

This option is used to remove the formatting such as bold, italics, or underline from the selected text. To use this option, select the text whose formatting you need to change and then right-click to display the shortcut menu. In the menu, choose the **Remove Formatting** option. The formatting of the selected text will be removed.

Combine Paragraphs

This option is used to combine the selected paragraphs into a single paragraph. AutoCAD LT replaces the returns between all the paragraphs by a space. As a result, the lines in the resultant paragraph are in continuation.

Character Set

This option is used to define the character set for the current font. You can select the desired character set from the cascading menu that is displayed when you choose this option.

Stack

Note that this option is available only if you select stack characters from the text. The **Stack** option is used to stack the selected text if there are any stack characters (characters separated by /, #, or ^) available in the multiline text.

Unstack

This option is also available only if there are some stacked characters in the text. The **Unstack** option is used to unstack the selected stacked text.

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Properties

This option is available only when you select a stacked text. When you choose this option, the **Stack Properties** dialog box is displayed, as shown in Figure 7-39. This dialog box is used to edit the text and the appearance of the selected stacked text. The options in this dialog box are discussed next.

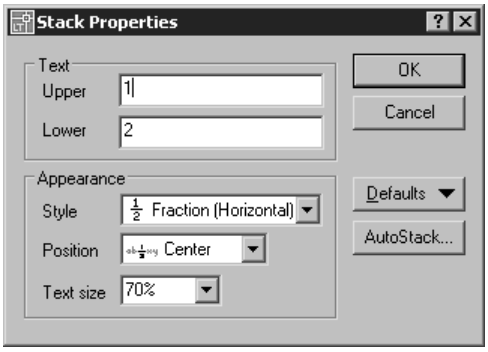

Figure 7-39 *The* ***Stack Properties*** *dialog box*

Text Area. You can change the upper and the lower values of the stacked text by entering their values in the **Upper** and **Lower** text boxes, respectively.

Appearance Area. You can change the style, position, and size of the stacked text by entering their values in the **Style**, **Position**, and **Text size** text boxes, respectively.

Default. This option allows you to restore the default values or save the new settings as the default settings for the selected stacked text.

AutoStack. When you choose this button, the **AutoStack Properties** dialog box is displayed, same as that shown earlier in Figure 7-31. The options in this dialog box were discussed under the **Stack** heading.

Tip

The options in the ***Options*** *shortcut menu are also available in the shortcut menu that is displayed when you select the text in the* ***Text window*** *and right-click. The shortcut menu is shown in Figure 7-40.*

Note

The ***Undo*** *and the* ***Redo*** *options are used to undo or redo the last actions done in the* ***In-Place Text Editor****. The* ***Cut*** *and* ***Copy*** *options can be used to move or copy the text from the text editor to any other application. Similarly, using the* ***Paste*** *option, you can paste text from any windows text-based application to the* ***In-Place Text Editor****.*

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Figure 7-40 *Shortcut menu*

Example 1 — *General*

In this example, you will use the **In-Place Text Editor** dialog box to write the following text in the drawing window.

`For long, complex entries, create multiline text using the MTEXT option. The angle is 10-degree. Dia = 1/2″ and length = 32 1/2″.`

The font of the text is **Swis721 BT,** text height is 0.20, color red, and written at an angle of 10-degree with Middle-Left justification. Make the word "multiline" bold, underline the text "multiline text", and make the word "angle" italic. The line spacing type and line spacing between the lines are **At least** and **1.5x** respectively. Use the symbol for degrees. After writing the text in the **Text window**, replace the word "option" with "command".

1. Choose the **Multiline Text** button from the **Draw** toolbar. After invoking the command, specify the first corner on the screen to define the first corner of the paragraph text boundary. You need to specify the rotation angle of the text before specifying the second corner of the paragraph text boundary. The prompt sequence is given next.

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Current text style: STANDARD. Text height: 0.2000
Specify first corner: *Select a point to specify first corner.*
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: **R** Enter
Specify rotation angle <0>: 10.
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: **L** Enter
Enter line spacing type [At least/Exactly] <At least>: Enter
Enter line spacing factor or distance <1x>: **1.5x**
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: *Select another point to specify the other corner.*

The **In-Place Text Editor** is displayed.

2. Select **Swis721 BT** true type font from the **Font** drop-down list.

3. Enter **0.20** in the **Text height** edit box, if the value in this edit box is not 0.2.

4. Select **Red** from the **Color** drop-down list.

5. Now enter the text in the **In-Place Text Editor**, as shown in Figure 7-41. To add the degrees symbol, choose the **Symbol** button from the **Text Formatting** toolbar and select **Degrees** from the shortcut menu. When you type 1/2 after Dia = and then press the ” key, AutoCAD LT displays the **AutoStack Properties** dialog box. Select **Convert it to a horizontal fraction** radio button, if it is not already selected. Also, make sure the **Enable AutoStacking** check box is selected. Now, close the dialog box.

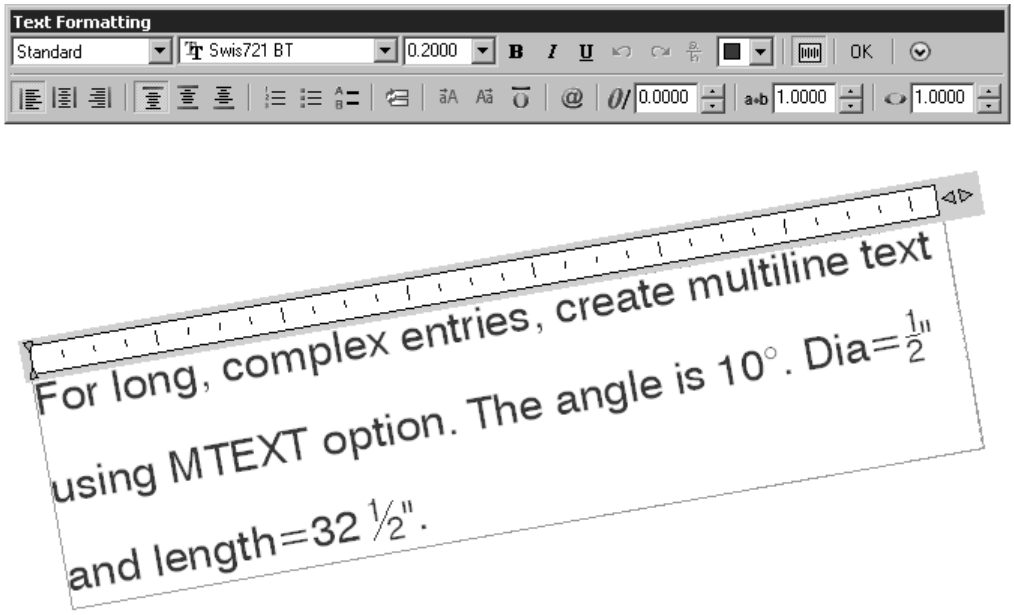

Figure 7-41 *The* ***In-Place Text Editor***

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Similarly, when you type 1/2 after length = 32 1/2 and then press the ” key, AutoCAD LT displays the **AutoStack Properties** dialog box. Select the **Convert it to a diagonal fraction** radio button. Close the dialog box.

6. Double-click on the word “multiline” to select it (or pick and drag to select the text) and then choose the **Bold** button to make it boldface and the **Underline** button to underline it.

7. Similarly, highlight the word “text” and choose the **Underline** button to underline it.

8. Highlight the word “angle” by double-clicking on it and then choose the **Italic** button.

9. Right-click in the **Text window** and choose **Justification > Middle Left** from the shortcut menu.

10. Click at the start of the multiline to move the cursor to the start. Now, right-click on the text window and choose **Find and Replace** from the shortcut menu. Alternatively, you can use CTRL+R keys or choose **Find and Replace** from the **Options** shortcut menu. On doing so, the **Find and Replace** dialog box is displayed.

11. In the **Find what** edit box, enter **option** and in the **Replace with** edit box, enter **command**.

12. Choose the **Find Next** button. AutoCAD LT finds the word “option” and highlights it . Choose the **Replace** button to replace **option** by **command**. The **AutoCAD LT** information box is displayed informing you that AutoCAD LT has finished searching for the word. Choose **OK** to close the information box.

Note

To set the width of multiline text objects, hold and drag the arrowhead on the right side of the ruler. In this example, the text has been accomodated in two lines.

13. Now, choose **Cancel** to close the **Find and Replace** dialog box and return to the **In-Place Text Editor**. Choose the **OK** button to exit the **In-Place Text Editor**. The text is displayed on the screen as shown in Figure 7-42.

Tip

*The text in the **In-Place Text Editor** can be selected by double-clicking on the word, by holding down the left mouse button of the pointing device and then dragging the cursor, or by triple-clicking on the text to select the entire line or paragraph.*

For long, complex entries, create **multiline text** using MTEXT command. The angle is 10°. Dia =½" and length =32 ½".

Figure 7-42 *Multiline text for Example 1*

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Exercise 2

General

Write the text on the screen, as shown in Figures 7-43 and 7-44. Use the special characters and text justification options shown in the drawing, using the **TEXT** and **MTEXT** commands. The text height is 0.1 and 0.15 for Figures 7-43 and 7-44, respectively.

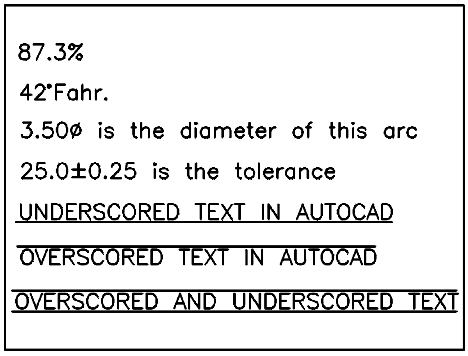

Figure 7-43 *Drawing with special characters*

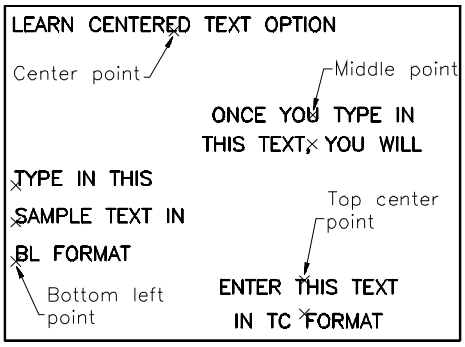

Figure 7-44 *Drawing for Exercise 2*

EDITING TEXT

The contents of **MTEXT** and **TEXT** object can be edited by using the **DDEDIT** and **PROPERTIES** commands. You can also use the AutoCAD LT editing commands, such as **MOVE**, **ERASE**, **ROTATE**, **COPY**, **MIRROR**, and **GRIPS** with any text object.

In addition to editing, you can also modify the text in AutoCAD LT. The modification that you can perform on the text include changing its scale and justification. The various editing and modifying operations are discussed next.

Editing Text Using the DDEDIT Command

Toolbar:	Text > Edit
Menu:	Modify > Object > Text > Edit
Command:	DDEDIT

You can use the **DDEDIT** command to edit the text. The most convenient way of invoking this command is by double-clicking on the text. If you double-click on the single line text written using the **TEXT** command, AutoCAD LT displays the **In-Place Text Editor** for single line text, in which the selected text is displayed. You can modify the text string in this edit box. The size of the bounding box increases or decreases, as you add more text or remove the existing text. Note that for the text object, you cannot modify any of its properties in the bounding box. However, if you double-click on the multiline text written using the **MTEXT** command, AutoCAD LT displays the text in the **In-Place Text Editor** for multiline text. You can make the changes using the various options in the editor. Apart from changing the text string, you are also allowed to change the properties of the paragraph text.

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You can also invoke the **Edit Text** dialog box and the **In-Place Text Editor** from the shortcut menu. Select the text for editing and then right-click in the drawing area. A shortcut menu is displayed and depending on the text object you have selected, the **Text Edit** or **Mtext Edit** options are available in the menu. Selecting these options displays the respective dialog boxes.

Editing Text Using the PROPERTIES Palette

Using the **DDEDIT** command with the text object, you can only change the text string and not its properties such as the height, angle, and so on. In this case, you can use the **PROPERTIES** palette for changing the properties. Select the text and choose the **Properties** button in the **Standard** toolbar. AutoCAD LT displays the **PROPERTIES** palette with all the properties of the selected text, as shown in Figure 7-45. Here you can change any value and also the text string. To edit a single line text, you can change the text string in the window. But for a multiline text, you must choose the **Full editor** button in the **Content** edit box. AutoCAD LT automatically switches to the **In-Place Text Editor**, where you can make changes to paragraph text.

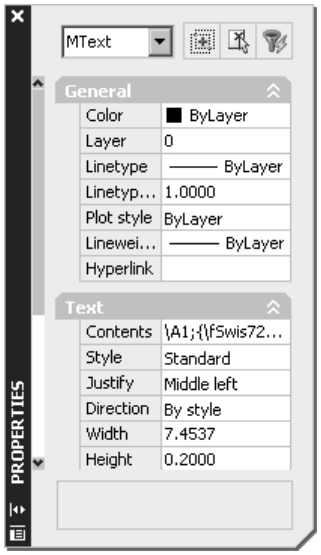

Figure 7-45 *The* ***PROPERTIES*** *palette*

Modifying the Scale of the Text

Toolbar:	Text > Scale
Menu:	Modify > Object > Text > Scale
Command:	SCALETEXT

You can modify the scale factor of the text using the **Scale Text** button in the **Text** toolbar. AutoCAD LT uses one of the justification options as the base point to scale the text. The prompt sequence that follows when you invoke this tool is given next.

Select objects: *Select the text object to scale*
Select objects: Enter
Enter a base point option for scaling
[Existing/Left/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <current>: *Specify an option that will be used as the base point to scale the text.*
Specify new height or [Match object/Scale factor] <0.2000>: *Enter the new height or select an option*

Match object

You can use the **Match object** option to select an existing text whose height will be used to scale the selected text.

Scale factor

You can use the **Scale factor** option to specify a scale factor to scale the text. You can also use the **Reference** option to specify the scale factor for the text.

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Modifying the Justification of the Text

Toolbar:	Text > Justify
Menu:	Modify > Object > Text > Justify
Command:	JUSTIFYTEXT

You can modify the justification of the text using the **Justify Text** button in the **Text** toolbar. Note that even after modifying the justification using this command, the location of the text is not changed. The prompt sequence that follows when you invoke this tool is given next.

Select objects: *Select the text object whose justification needs to be changed*
Select objects: Enter
Enter a justification option
[Left/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Left>: *Specify the new justification for the text.*

INSERTING TABLE IN THE DRAWING

Toolbar:	Draw > Table
Menu:	Draw > Table
Command:	TABLE

A number of mechanical, architectural, electric, or civil drawings require a table in whch some information about the drawing is displayed.For example, a drawing of an assembly needs the Bill of Material, which is a table providing details such as the number of parts in the drawing, their names, their material, and so on. To enter this information, AutoCAD LT allows you to create tables using the **TABLE** command. When you invoke this command, the **Insert Table** dialog box is displayed, as shown in Figure 7-46. The options in this dialog box are discussed next.

Table Style Settings Area

The options in this area are used to define the settings for the table style. These options are discussed next.

Table Style name

This drop-down list displays the names of the various table styles in the current drawing. By default, it displays only **Standard**. This is the default table style in a drawing.

Table Style dialog

This button is chosen to display the **Table Style** dialog box that can be used to create a new table style, or modify and delete an existing table style. You can also use this dialog box to set a table style current. You will learn more about creating a new table style in the next section.

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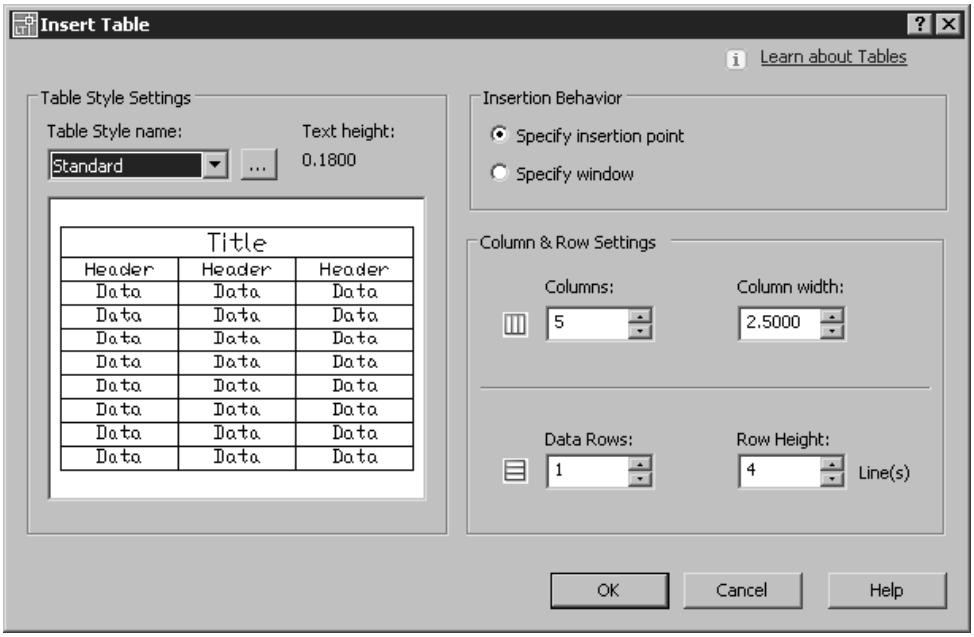

*Figure 7-46 The **Insert Table** dialog box*

Insertion Behavior Area

The options in this area are used to specify the method of placing the table in the drawing. These options are discussed next.

Specify insertion point

This radio button is selected to place the table using the upper left corner of the table. If this radio button is selected and you choose **OK** from the **Insert Table** dialog box, you will be prompted to select the insertion point, which is by default the upper left corner of the table. By creating a different table style, you can change the point using which the table is inserted.

Specify window

If this radio button is selected and you choose **OK** from the **Insert Table** dialog box, you will be prompted to specify two corners for placing the table. The number of rows and columns in the table will depend on the size of the window you define. When you choose **OK** from the **Insert Table** dialog box after selecting this radio button, you will be prompted to select the first and the second corner. Depending on the size of the window defined by the two corner, the number and size of rows and columns are defined.

Column and Row Settings Area

The options in this area are used to specify the number and size of rows and columns. The availability of these options depend on the option selected from the **Insertion Behavior** area. These options are discussed next.

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Columns
This spinner is used to specify the number of columns in the table.

Column width
This spinner is used to specify the width of columns in the table.

Data Rows
This spinner is used to specify the number of rows in the table.

Row Height
This spinner is used to specify the height of rows in the table. The height is defined in terms of lines and the minimum value is one line.

After setting the parameters in the **Insert Table** dialog box, choose the **OK** button. Depending on the type of insertion behavior selected, you will be prompted to insert the table. As soon as you complete the insertion procedure, the **In-Place Text editor** is displayed and you are allowed to enter the parameters in the first row of the table. By default, the first row is the title of the table. After entering the data, press ENTER. The first field of the first column is highlighted, which is the column head, and you are allowed to enter the data in it.

AutoCAD LT allows you to use the arrow keys on the keyboard to move to the other cells in the table. You can enter the data in the field and then press the arrow key to move to the other cells in the table. After entering the data in all the fields, press ENTER to exit the **Text Formatting** toolbar.

Tip
*You can also right-click while entering the data in the table to display the shortcut menu. This shortcut menu is similar to that shown in the **In-Place Text Editor** and can be used to insert field, symbols, text, and so on.*

CREATING A NEW TABLE STYLE

Toolbar:	Styles > Table Style
Menu:	Format > Table Style
Command:	TABLESTYLE

To create a new table style, choose the **Table Style Manager** button from the **Styles** toolbar; the **Table Style** dialog box is displayed, as shown in Figure 7-47. You can also invoke this dialog box by choosing the **Table Style Dialog** button [**...**] from the **Insert Table** dialog box.

To create a new table style, choose the **New** button, the **Create New Table Style** dialog box will be displayed, as shown in Figure 7-48.

Enter the name of the table style in the **New Style Name** edit box. Select the style on which you

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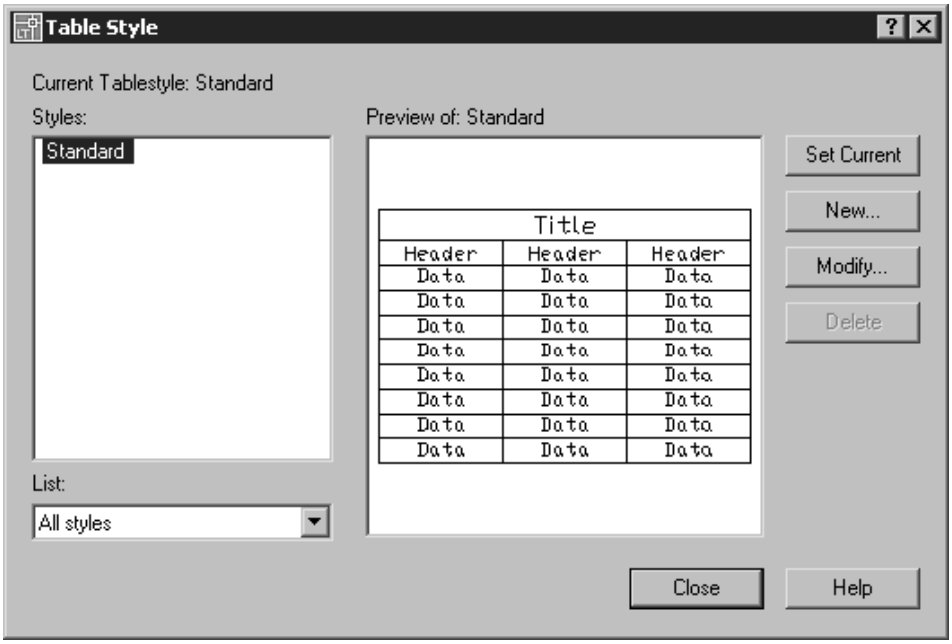

Figure 7-47 *The* ***Table Style*** *dialog box*

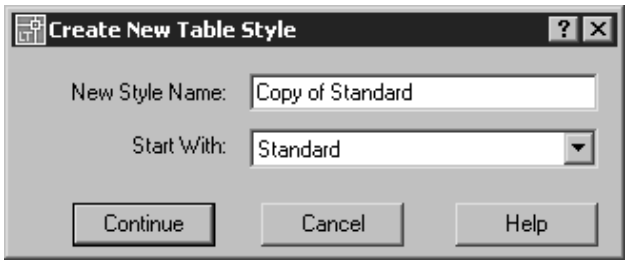

Figure 7-48 *The* ***Create New Table Style*** *dialog box*

want to base the new style from the **Start With** drop-down list. By default, this drop-down list shows only **Standard**. After specifying the settings, choose **Continue**; the **New Table Style** dialog box will be displayed. This dialog box has three tabs, which are discussed next.

Data Tab

The options available in the **Data** tab, shown in Figure 7-49, are used to specify the settings for the data to be entered in the cells of the table, cell border properties, table direction, and the margins between the data and border of the cells. These options are discussed next.

Cell properties Area

The options in this area are used to set the properties for the style, color, height, and alignment of the text in the cells. These options are discussed next.

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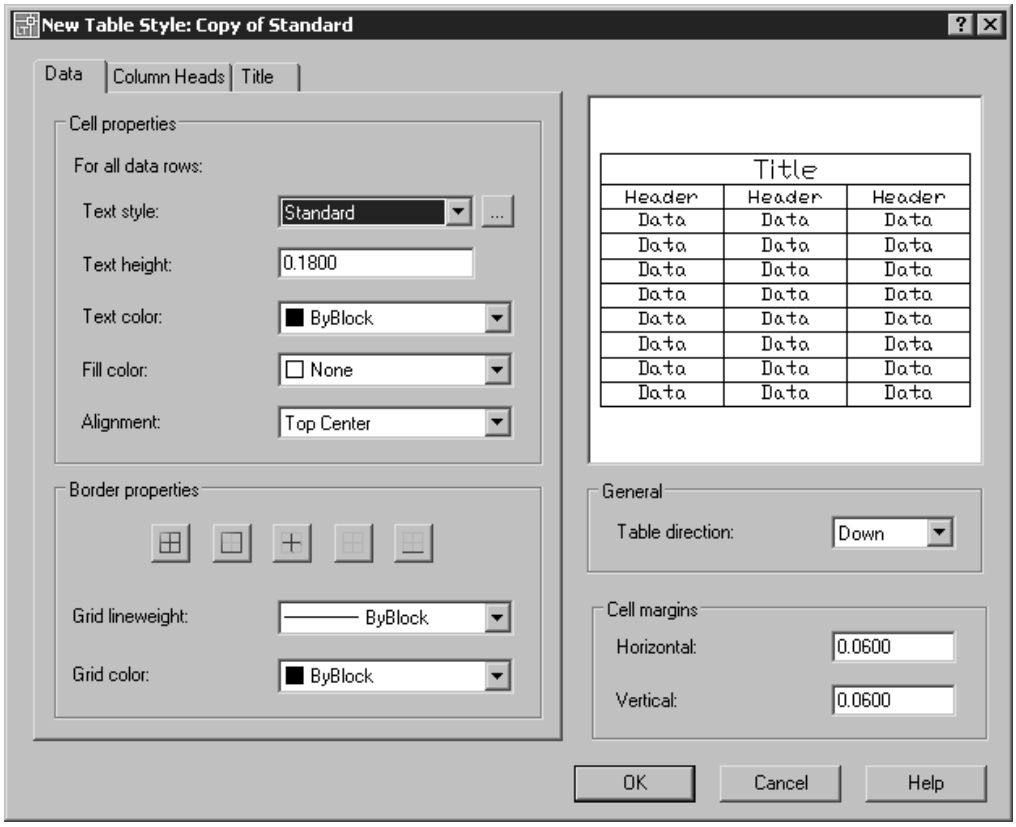

Figure 7-49 *The* ***Data*** *tab of the* ***New Table Style*** *dialog box*

Text style. This drop-down list is used to select the text style that will be used to enter the text in the cells. By default, it shows only **Standard**, which is the default text style. You will learn to create more text styles later in this chapter.
Text height. This edit box is used to specify the height of the text to be entered in the cells.

Text color. This drop-down list is used to specify the color of the text that will be entered in the cells. If you select the **Select Color** option, the **Select Color** dialog box is displayed, which can be used to select from index color, true color, or from the color book.
Fill color. This drop-down list is used to specify the fill color for the cells.

Alignment. This drop-down list is used to specify the alignment of the text that will be entered in the cells. The default alignment is top center.

Border properties Area

The options in this area are used to set the properties of the border of the table. The lineweight and color settings that you specify using this area will be applied to all borders, outside borders, inside borders, no border, or bottom border, depending on which button is chosen from this area. The two drop-down lists in this area are discussed next.

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Grid lineweight. This drop-down list is used to specify the lineweight of the border that you specify using the buttons in this area.

Grid color. This drop-down list is used to specify the color of the border that you specify using the buttons in this area.

Column Heads Tab

The options available in the **Column Heads** tab, shown in Figure 7-50, are used to specify the settings for the data to be entered in the column heads of the table and the border properties of the column heads. These options will be available only if the **Include Header row** check box in the **Cell properties** area is selected. These options are similar to those discussed in the **Data** tab.

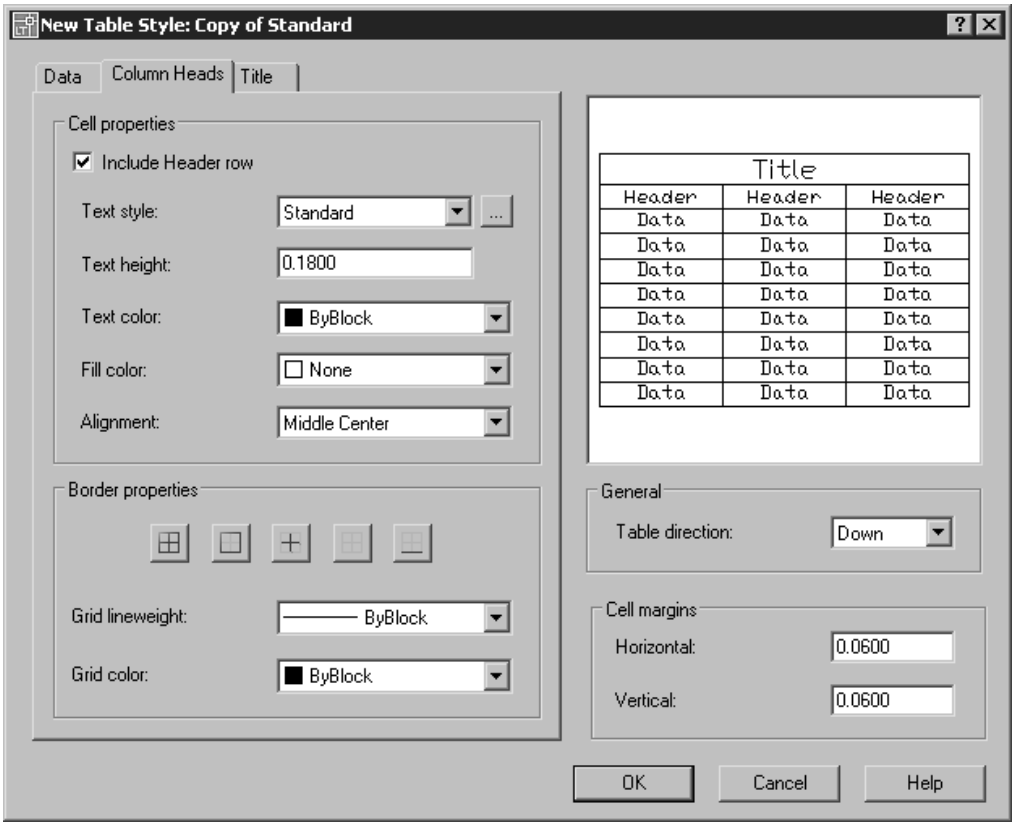

Figure 7-50 *The* ***Column Heads*** *tab of the* ***New Table Style*** *dialog box*

Title Tab

The options in the **Title** tab, shown in Figure 7-51, are used to specify the settings for the data to be entered in the title of the table and the border properties of the table. These options will be available only if the **Include Title row** check box in the **Cell properties** area is selected. These options are similar to those discussed in the **Data** tab.

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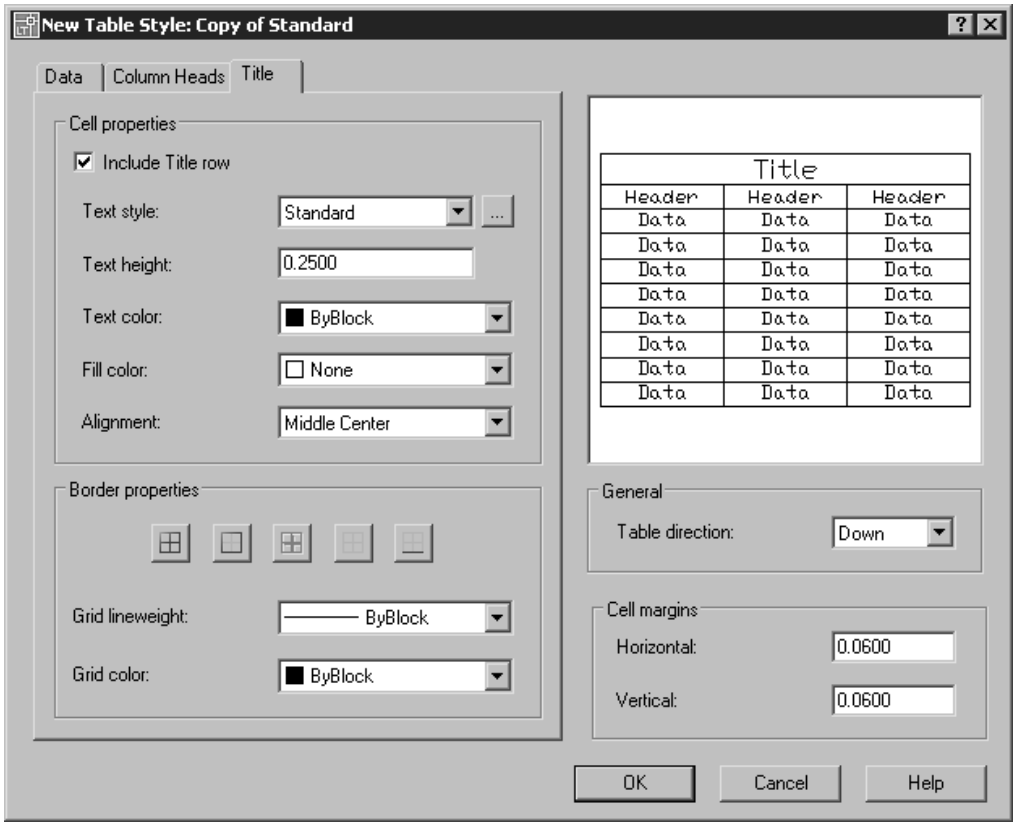

Figure 7-51 *The* ***Title*** *tab of the* ***New Table Style*** *dialog box*

General Area

This area has the **Table direction** drop-down list, which is used to specify the direction of the table. The default direction is down. As a result, the title and headers are at the top and the data fields are below them. If you select **Up** from the **Table direction** drop-down list, the title and headers will be at the bottom and the data fields will be on the top.

Cell margins Area

The options in this area are used to set the margins between the data in the cells and the horizontal and vertical borders. These options are discussed next.

Horizontal

This edit box is used to specify the minimum spacing between the data entered in the cells and the left and right border lines of the cells.

Vertical

This edit box is used to specify the minimum spacing between the data entered in the cells and the top and bottom border lines of the cells.

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SETTING A TABLE STYLE CURRENT

To set a table style current so that it is used to create all the new tables, invoke the **Table Style** dialog box by choosing the **Table Style Manager** button from the **Styles** toolbar. Select the table style from the **Styles** list box in the **Table Style** dialog box and choose the **Set Current** button. The current table style on top of the **Table Style** dialog box now displays the name of the table style you made current. You can also set a table style current by selecting it from the **Table Style Control** drop-down list in the **Styles** toolbar. This is a more convenient method of setting a table style current.

MODIFYING A TABLE STYLE

To modify a table style, invoke the **Table Style** dialog box by choosing the **Table Style Manager** button from the **Styles** toolbar. Select the table style from the **Styles** list box in the **Table Style** dialog box and choose the **Modify** button. The **Modify Table Style** dialog box is displayed. This dialog box is similar to the **New Table Style** dialog box. Modify the options in the various tabs and areas of this dialog box and then choose **OK**.

MODIFYING TABLES

Select any cell of a table and right-click to display a shortcut menu, as shown in Figure 7-52. The options in the shortcut menu are used to **Modify the table**, **insert block**, **add formulas** and perform other operations. The **Cut** and **Copy** options can be used to move or copy the contents from one cell to the other. Using the **Paste** option, you can paste the contents that you have cut or copied from one cell to the other. Place the cursor on the **Recent Input** option; a flyout, that contains the recent command inputs, is displayed. The options available in the shortcut menu are discussed next.

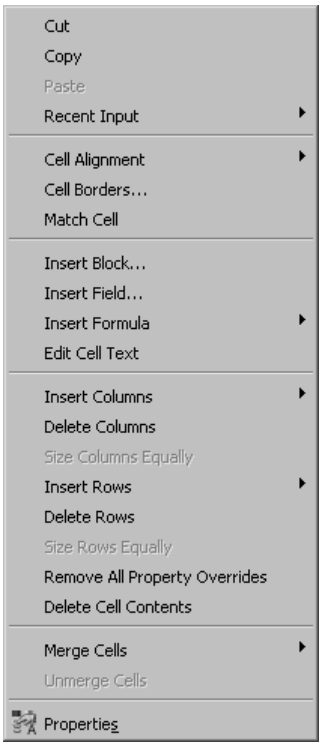

Figure 7-52 *The shortcut menu*

Cell Alignment

Choose the **Cell Alignment** option from the shortcut menu to display a flyout. The options in the flyout are used to align the contents of cells. These options are similar to the justification options of multiline texts.

Cell Borders

Choose the **Cell Borders** option to display the **Cell Border Properties** dialog box, see Figure 7-53. The options in this dialog box are similar to those in the **Border Properties Area** of **Data Tab** in the **New Table Style** dialog box.

Match Cells

This option is used to inherit the properties of one cell into the other. For example, if you have specified **Top Left Cell Alignment** to the source cell, then using the **Match Cells** option, you

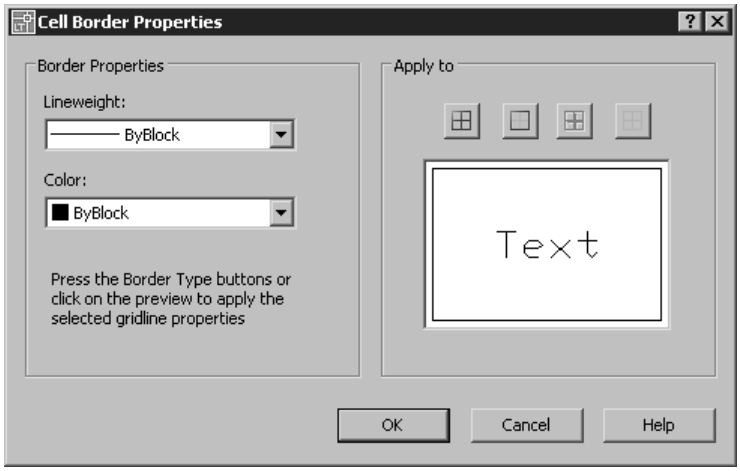

*Figure 7-53 The **Cell Border Properties** dialog box*

can inherit this property to the destination cell. This option is useful if you have assigned a number of properties to one cell and you want to inherit these properties in some specified number of cells. Choose the **Match Cell** option from the shortcut menu; the cursor is changed to the match properties cursor and you are prompted to select the destination cell. Select the cells, in which you want the properties to be inherited and then press ENTER.

Insert Block

This option is used to insert a block in the selected cell. Choose the **Insert Block** option; **Insert a Block in a Tabel Cell** dialog box is displayed, as shown in Figure 7-54. Enter the name of the block in the **Name** edit box or choose the **Browse** button to locate the destination file of the block. If you have browsed the file path, it will be displayed in the **Path** area. The options available in the **Insert a Block in a Tabel Cell** dialog box are discussed next.

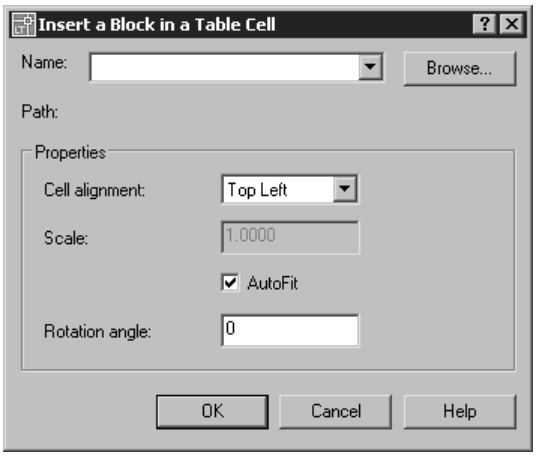

*Figure 7-54 The **Insert a Block in a Tabel Cell** dialog box*

Properties Area

The options in this area are discussed next.

Cell Alignment. This drop-down list is used to define the block alignment in the selected cell.

Scale. This drop-down list is used to specify the scale of the block. By default, this edit box is not available because the **AutoFit** check box is selected below this drop-down list. Selecting the **AutoFit** check box ensures that the block is scaled such that it fits in the selected cell.

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Rotation angle. The **Rotation angle** edit box is used to specify the angle by which the block will be rotated before being placed in the cell.

Insert Formula

Place the cursor on the **Insert Formula** option to display the **Insert Formula** flyout. The flyout is shown in Figure 7-55. This flyout contains the formulas that can be applied to a given cell. The formula calculates the values for that cell using the values of other cell. In a table, the columns are named with letters (like A, B, C, ...) and rows are named with numbers (like 1, 2, 3, ...). The **TABLEINDICATOR** system variable controls the display of column letters and row numbers. By default, **TABLEINDICATOR** system variable is set to 1, that means the row numbers and column letters will be displayed when **In-Place Text Editor** is invoked. Set the system variable to zero to turn off the visibility of row numbers and column letters. The nomenclature of cells is done using the column letters and row numbers. For example, the cell corresponding to column A and row 2 is A2. For a better understanding, some of the cells have been labeled accordingly in Figure 7-56.

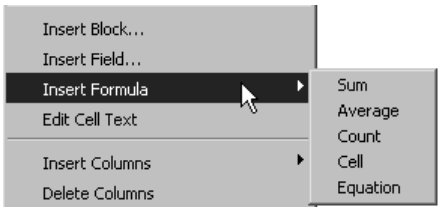

Figure 7-55 *The* ***Insert Formula*** *flyout*

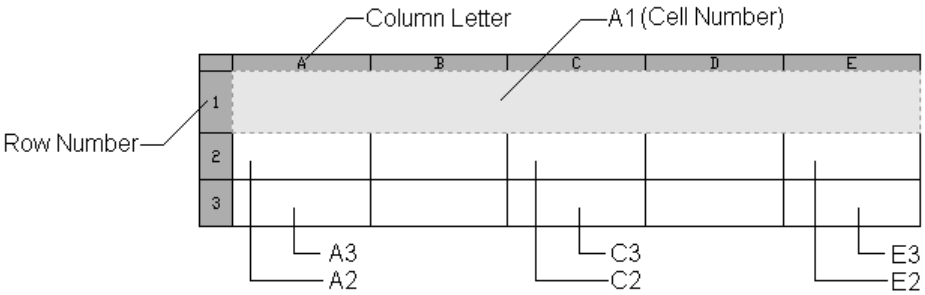

Figure 7-56 *The* ***Table*** *showing nomenclature for* ***Columns***, ***Rows***, *and* ***Cells***

Formulas are defined by the range of cells. The range of cells is specified by writing the first and the last cell of the range, separated by a colon (:). The range takes all the cells falling between specified cells. For example, if you write A2 : C3, this means all the cells falling in 2nd and 3rd rows, Column A and B will be taken into account. To insert a formula, double click on the cell; **In-Place Text Editor** is invoked. You can now write the syntax of the formula in the cell. The syntax for different formulas are discussed later while explaining different formulas. Formulas can also be inserted by using the **Insert Formula** flyout, shown in Figure 7-55. Different formulas available in the **Insert Formula** flyout are discussed next.

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Sum

The **Sum** option gives the output for a given cell as the sum of the numerical values entered in a specified range of cells. Invoke the shortcut menu by selecting the cell, in which you want to insert the formula and then right click. Choose the **Sum** option from the shortcut menu. You are prompted to select the first corner of the tabel cell range and then the second corner. The sum of values of all the cells that fall between the selected range will be displayed as the output. As soon as you specify the second corner, **In-Place Text Editor** is displayed and also the formula is displayed in the cell. In addition to the formula, you can also write multilne text in the cell. Choose the **Ok** button to exit the editor. When you exit the text editor, the formula is replaced by a hash (#). Now, if you enter numerical values in the cells included in the range, the hash (#) is replaced according to the addition of those numerical values. The prompt sequence, when you select the **Sum** option, is given next.

Select first corner of table cell range: *Specify a point in the first cell of the cell range.*
Select second corner of table cell range: *Specify a point in the last cell of the cell range*

Note

*Syntax for **Sum** option is: =Sum(Number of first cell of cell range (for example: A2):Number of last cell of cell range (for example: C5))*

Average

This option is used to insert a formula, that calculates the average of values of the cells falling in the cell range. Prompt sequence is the same as for the **Sum** option.

Note

*Syntax for **Average** option is: =Average(Number of first cell of cell range (for example: A2):Number of last cell of cell range (for example: C5))*

Count

This option is used to insert a formula that calculates the number of cells falling under the cell range. The prompt sequence is the same as for the **Sum** option.

Note

*Syntax for **Count** option is: =Count(Number of first cell of cell range (for example: A2):Number of last cell of cell range (for example: C5))*

Cell

This option equates the current cell with the selected cell. Whenever there is a change in the value of the selected cell, the change is automatically updated in the other cell. Select the **Cell** option from the shortcut menu; you are prompted to select a table cell. Select a cell with which you want to equate the current cell. The prompt sequence for the **Cell** option is given next.

Select table cell: *Select a cell to equate with the current cell*

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Note
*Syntax for **Cell** option is: =Number of the cell*

Equation

Using this option, you can manually write equations. The syntax for writing the equations should be the same as explained earlier.

Edit Cell Text

This option is used to edit cell text using **In-Place Text Editor**.

Insert Columns

This option is used to insert a column in the table. Choose the **Insert Columns** option to display the **Insert Column** flyout. Two options are available in the flyout. Choose the **Right** option to add a new column on the right side of the selected cell. Choose the **Left** option to add a column on the left side of the selected cell.

Delete Columns

Choose this option to delete the column corresponding to the selected cell.

Size Columns Equally

This option is highlighted only when you have done multiple selections of cells. Use the **SHIFT** key to select multiple cells and then choose the **Size Columns Equally** option from the shortcut menu to make the size of all the selected columns equal. To do so, select the table by clicking anywhere on the table boundaries. Now invoke the shortcut menu and choose the **Size Columns Equally** option.

Insert Rows

This option is used to insert a row in the table. When you choose this option, the **Insert Rows** flyout is displayed. Two options are available in the flyout. Choose the **Above** option; a new row will be added above the selected cell. Choose the **Below** option to add a row below the selected cell.

Delete Rows

Select this option to delete the row corresponding to the selected cell.

Size Rows Equally

This option is used to make the selected rows equal in size. The procedure for making rows equal in size is the same as that for **Size Columns Equally**.

Delete Cell Contents

Choose this option to delete all the contents including formulas from the selected cell. You can also select the cell and choose **DELETE** from the keyboard to delete the contents of the cell.

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Merge Cells

This option is used to merge the cells. Place the cursor on the **Merge Cells** option; the **Merge Cells** flyout is displayed. Three options are available in the flyout. Select multiple cells using the **SHIFT** key and then select **All** from the flyout to merge all the selected cells. To merge all the cells in the row of the selected cell, select the **By Row** option from the flyout. To merge all the cells in the column of the selected cell, select the **By Row** option from the flyout.

SUBSTITUTING FONTS

AutoCAD LT provides you the facility to designate the fonts that you want to substitute for the other fonts used in the drawing. The information about font mapping is specified in the font mapping file (*acad.fmp*). The font mapping has the following advantages.

1. You can specify a font to be used when AutoCAD LT cannot find a font used in the drawing.

2. You can enforce the use of a particular font in the drawings. If you load a drawing that uses different fonts, you can use font mapping to substitute the desired font for the fonts used in the drawing.

3. You can use *.shx* fonts while creating or editing a drawing. When you are done and ready to plot the drawing, you can substitute other fonts for *.shx* fonts.

The font mapping file is an ASCII file with *FMP* extension containing one font mapping per line. The format on the line is given next.

Base name of the font file;**Name of the substitute font with extension** (ttf, shx, etc.)

For example, if you want to substitute the ROMANC font for *SWISS.TTF*; the entry is given next.

SWISS;ROMANC.SHX

You can enter this line in the *acad.fmp* file or create a new file. To create a new font mapping file, you need to specify this new file. You can use the **Options** dialog box to specify the new font mapping file. To specify a font mapping table in the **Options** dialog box, choose **Tools > Options** from the menu bar to display the **Options** dialog box. Choose the **Files** tab and click on the **plus** sign next to **Text Editor, Dictionary, and Font File Names**. Now, click on the plus sign next to **Font Mapping File** to display the path and the name of the font mapping file. Double-click on the file to display the **Select a file** dialog box. Select the new font mapping file and exit the **Options** dialog box. At the Command prompt, enter **REGEN** to convert the existing text font to the font as specified in the new font mapping file. You can also use the **FONTMAP** system variable to specify the new font map file.

Command: **FONTMAP** Enter
Enter new value for FONTMAP, or . for none <"path and name of the current font mapping file">: *Enter the name of the new font mapping file.*

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The following file is a partial listing of *acad.fmp* file with the new font mapping line added (swiss;romanc.shx).

swiss;romanc.shx
cibt;CITYB___.TTF
cobt;COUNB___.TTF
eur;EURR____.TTF
euro;EURRO___.TTF
par;PANROMAN.TTF
rom;ROMANTIC.TTF
romb;ROMAB___.TTF
romi;ROMAI___.TTF
sas;SANSS___.TTF
sasb;SANSSB__.TTF
sasbo;SANSSBO_.TTF
saso;SANSSO__.TTF

Note
The text styles that were created using the PostScript fonts are substituted with an equivalent TrueType font and plotted using the substituted font.

Specifying an Alternate Default Font

When you open a drawing file that specifies a font file that is not on your system or is not specified in the font mapping file, AutoCAD LT, by default, substitutes the *simplex.shx* font file. You can specify a different font file in the **Options** dialog box or do so by changing the **FONTALT** system variable.

Command: **FONTALT** Enter
New value for FONTALT, or . for none <"simplex.shx">: *Enter the font file name.*

CREATING TEXT STYLES

Toolbar:	Text > Text Style
	Styles > Text Style
Menu:	Format > Text Style
Command:	STYLE

By default, text in AutoCAD LT is written using the default text style called **Standard**. This text style is assigned a default text font (*TXT.SHX*) and the default formatting. However, if you need to write a text using some other font and other parameters, you need to use the **In-Place Text Editor**. This is because only using this command, you can change the formatting and font of the text.

However, it is a very tedious job to use the **In-Place Text Editor** every time to write the text and change its properties. This is the reason AutoCAD LT provides you with an option of modifying

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the default text style or creating a new text style. After creating a new text, you can make it current. All the text written after making the new style current will use this style.

To create a new text style or to modify the default style, choose the **Text Style Manager** button from the **Styles** toolbar. The **Text Style** dialog box is displayed, as shown in Figure 7-57.

In the **Style Name** edit box the default style (**Standard**) will be displayed. For creating a new style, choose the **New** button to display the **New Text Style** dialog box, as shown in Figure 7-58, and enter the name of the style you want to create.

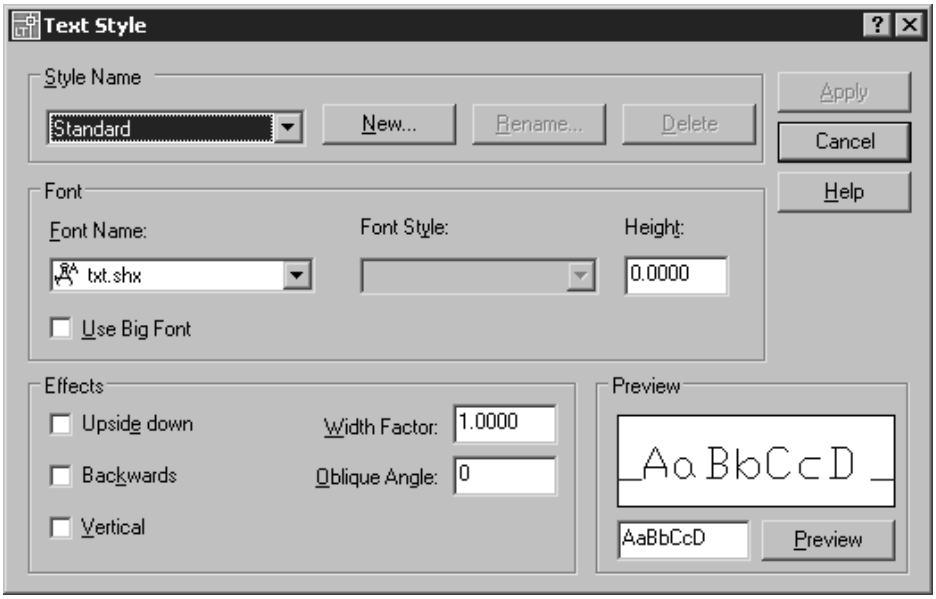

Figure 7-57 *The* ***Text Style*** *dialog box*

A new style having the entered name and the properties present in the **Text Style** dialog box will be created. To modify this style, select the style name from the list box and then change the different settings by entering new values in the appropriate boxes. You can change the font by selecting a new font from the **Font Name** drop-down list. Similarly, you can change the text height, width, and oblique angle.

Figure 7-58 *The* ***New Text Style*** *dialog box*

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Remember that if you have already specified the height of the text in the **Text Style** dialog box, AutoCAD LT will not prompt you to enter the text height while writing the text using the **TEXT** command. The text will be created using the height specified in the text style. If you want AutoCAD LT to prompt you for the text height, specify 0 text height in the dialog box. For **Width Factor,** 1 is the default. If you want the letters expanded, enter a width factor greater than 1. For compressed letters, enter a width factor less than 1. Similarly, for the **Oblique Angle**, 0 is the default. If you want the slant of the letters toward the right, the value should be greater than 0; to slant the letters toward the left, the value should be less than 0. You can also force the text to be written upside down, backwards, and vertically by checking their respective check boxes. As you make the changes, you can see their effect in the **Preview** box. After making the desired changes, choose the **Apply** button and the **Close** button to exit the dialog box. Figure 7-59

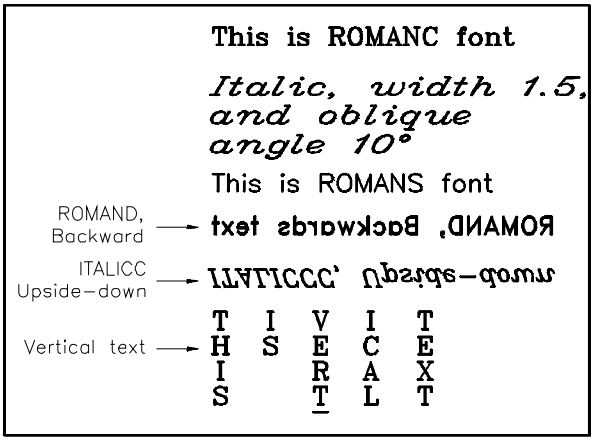

Figure 7-59 *Specifying different features to text style files*

shows text objects with all these settings.

DETERMINING TEXT HEIGHT

The actual text height is equal to the product of the **scale factor** and the **plotted text height**. Therefore, scale factors are important numbers for plotting the text at the correct height. This factor is a reciprocal of the drawing plot scale. For example, if you plot a drawing at a scale of ¼ = 1, you calculate the scale factor for text height as follows.

¼" = 1" (i.e., the scale factor is 4)

The scale factor for an architectural drawing that is to be plotted at a scale of ¼" = 1'0" is calculated as given next

¼" = 1'0", or ¼" = 12", or 1 = 48

Therefore, in this case, the scale factor is 48.

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For a civil engineering drawing with a scale 1"= 50', the scale factor is shown next.

1" = 50', or 1" = 50X12", or 1 = 600

Therefore, the scale factor is 600.

Next, calculate the height of the AutoCAD LT text. If it is a full-scale drawing (1=1) and the text is to be plotted at 1/8" (0.125), it should be drawn at that height. However, in a civil engineering drawing, a text drawn 1/8" high will look like a dot. This is because the scale for a civil engineering drawing is 1"= 50', which means that the drawing you are working on is 600 times larger. To draw a normal text height, multiply the text height by 600. Now, the height will be as calculated below.

0.125" x 600 = 75

Similarly, in an architectural drawing, which has a scale factor of 48, a text that is to be 1/8" high on paper must be drawn 6 units high, as shown in the following calculation:

0.125 x 48 = 6.0

It is very important to evaluate scale factors and text heights before you begin a drawing. It would be even better to include the text height in your prototype drawing by assigning the value to the **TEXTSIZE** system variable.

Menu:	Tools > Spelling
Command:	SPELL

CHECKING SPELLING

You can check the spelling of text (text generated by the **TEXT** or **MTEXT** commands) by using the **SPELL** command. The prompt sequence is given next.

Command: **SPELL**
Select object: *Select the text for spell check or enter ALL to select all text objects.*

If no misspelled words are found in the selected text, then AutoCAD LT displays a message. If the spelling is incorrect for any word in the selected text, AutoCAD LT displays the **Check Spelling** dialog box, as shown in Figure 7-60. The misspelled word is displayed under **Current word**, and correctly spelled alternate words are listed in the **Suggestions** box. The dialog box also displays the misspelled word with the surrounding text in the **Context** box. You may select a word from the list, ignore the correction, and continue with the spell check, or accept the

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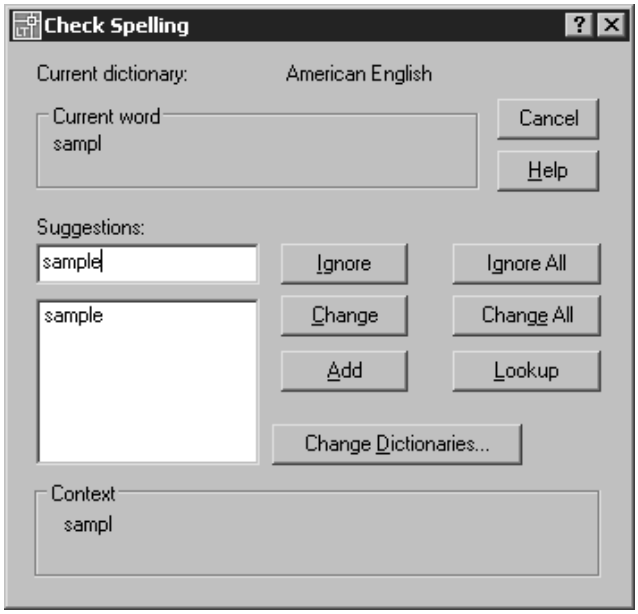

Figure 7-60 *The* ***Check Spelling*** *dialog box*

change. To add the listed word in the custom dictionary, choose the **Add** button. The dictionary can be changed from the **Check Spelling** dialog box or by specifying the name in the **DCTMAIN** or **DCTCUST** system variables. The dictionary must be specified for the spell check.

Note

You can also rename and use the MS Word dictionary or any other dictionary. Choose the ***Change Dictionaries*** *button to display the* ***Change Dictionaries*** *dialog box and enter the new dictionary name with the .cus extension.*

Toolbar:	Text > Find
Menu:	Edit > Find
Command:	FIND

TEXT QUALITY AND TEXT FILL

AutoCAD LT supports **TrueType fonts**. You can use your own **TrueType fonts** by adding them to the Fonts directory. You can also keep your fonts in a separate directory, in which case you must specify the location of your fonts directory in the AutoCAD LT search path.

The resolution and text fill of the **TrueType font** text is controlled by the **TEXTFILL** and **TEXTQLTY** system variables. If **TEXTFILL** is set to 1, the text will be filled. If the value is set to 0, the text will not be filled. On the screen the text will appear filled, but when it is plotted the text will not be filled. The **TEXTQLTY** variable controls the quality of the **TrueType font** text.

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The value of this variable can range from 0 to 100. The default value is 50, which gives a resolution of 300 dpi (dots per inch). If the value is set to 100, the text will be drawn at 600 dpi. The higher the resolution, the more time it takes to regenerate or plot the drawing.

FINDING AND REPLACING TEXT

You can use the **FIND** command to find and replace the text. The text could be a line text

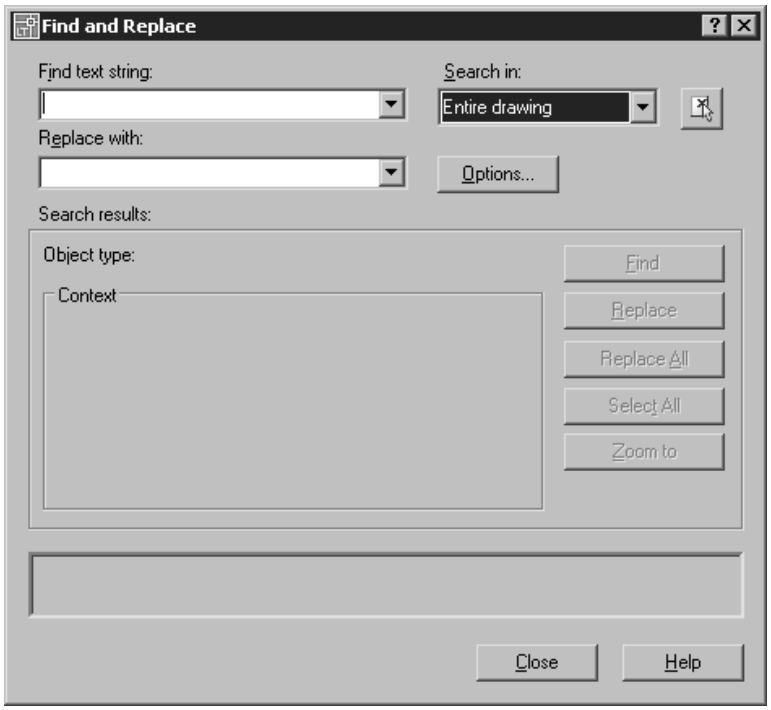

*Figure 7-61 The **Find and Replace** dialog box*

created by the **TEXT** command, paragraph text created by the **MTEXT** command, dimension annotation text, block attribute value, hyperlinks, or hyperlink description. When you invoke this command, AutoCAD LT displays the **Find and Replace** dialog box, as shown in Figure 7-61. You can use this dialog box to perform the following functions.

Find Text

To find the text, enter the text you want to find in the **Find text string** edit box. You can search the entire drawing or confine the search to the selected text. To select the text, choose the **Select Objects** button. The **Find and Replace** dialog box is temporarily closed and AutoCAD LT switches to the drawing window. Once you have selected the text, the **Find and Replace** dialog box is redisplayed. In the **Search in** drop-down list, you can specify if you want to search the entire drawing or the current selection. If you choose the **Options** button, AutoCAD LT displays the **Find and Replace Options** dialog box, as shown in Figure 7-62. In this dialog box, you can

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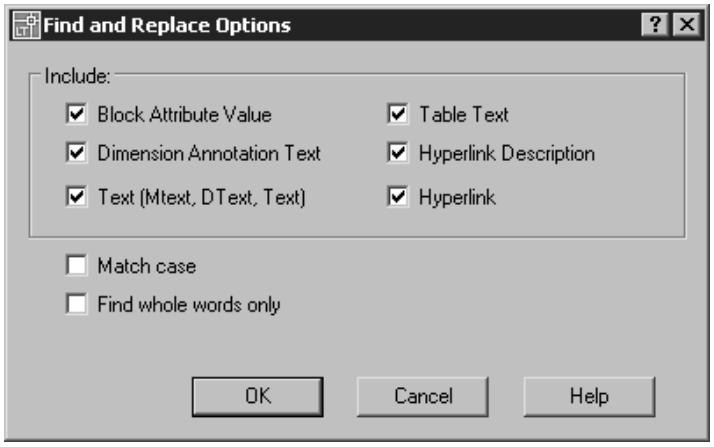

Figure 7-62 *The* ***Find and Replace Options*** *dialog box*

specify whether to find the whole word and whether to match the case of the specified text. To find the text, choose the **Find** button. The found text, with the surrounding text, is displayed in the **Context** area. To find the next occurrence of the text, choose the **Find Next** button.

Replacing Text

If you want to replace the specified text with the new text, enter the new text in the **Replace with** edit box. Now, if you choose the **Replace** button, only the found text will be replaced. If you choose the **Replace All** button, all occurrences of the specified text will be replaced with the new text.

Self-Evaluation Test

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

1. Tables in AutoCAD LT are created using the **TABLET** command. (T/F)

2. The **ALL** option of the **ZOOM** command displays the drawing limits or extents, whichever is greater. (T/F)

3. While using the ZOOM scale option, with respect to the current view, you have to enter a number followed by an **X**. (T/F)

4. You can use the scroll bars to pan the drawing in any direction. (T/F)

5. The **VIEW** command does not save any drawing object data. Only the _____ parameters required to redisplay that portion of the drawing are saved.

6. Multiple lines of text can be entered at any desired location in the drawing area with a single _____ command.

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7. With the _____ justification option of the **TEXT** command, AutoCAD LT adjusts the letter width to fit the text between the two given points, but the height remains constant.

8. In the **MTEXT** command, the height specified in the In-Place Text Editor does not affect the _____ system variable.

9. With the _____ command, to edit a single line text, you can change the text string in the window, but for a multiline text you must choose the **Full editor** button in the **Content** edit box.

10. You can use the _____ system variable to specify the new font mapping file.

Review Questions

Answer the following questions.

1. You cannot insert a field using the **MTEXT** command. (T/F)

2. If the **BLIPMODE** variable is set to On, blip marks do not appear on the screen. (T/F)

3. With the **ZOOM** command, the actual size of the object changes. (T/F)

4. The **REDRAW** command can be used as a transparent command. (T/F)

5. The **TEXT** command does not allow you to see the text on the screen as you type it. (T/F)

6. Which command recalculates all the objects in a drawing and redraws the current viewport only?

   (a) **REDRAW** (b) **REDRAWALL**
   (c) **REGEN** (d) **REGENALL**

7. Which of the following commands cannot be used transparently?

   (a) **ZOOM** (b) **PAN**
   (c) **VIEW** (d) **REDRAW**

8. How many views are saved with the **Previous** option of the **ZOOM** command?

   (a) 6 (b) 8
   (c) 10 (d) 12

9. In the **In-Place Text Editor**, which of the following characters is used to stack the text with a diagonal line without using the **Autostack Properties** dialog box?

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(a) ^ (b) /
(c) # (d) @

10. Which command can be used to create a new text style and modify the existing ones?

(a) **TEXT** (b) **MTEXT**
(c) **STYLE** (d) **SPELL**

11. The four main text alignment modes are _____, _____, _____, and _____.

12. You can use the _____ command to write a paragraph text whose width can be specified by defining the _____ of the text boundary.

13. When the **Justify** option is invoked, the user can place text in one of the _____ various alignment types by selecting the desired alignment option.

14. The text created by the _____ command is a single object regardless of the number of lines it contains.

15. Using the **MTEXT** command, the character _____ stacks the text vertically without a line (tolerance stack).

16. If you want to edit text, select it and then right-click such that the various editing options _____ in the menu become available.

17. You can view the entire drawing (even if it is beyond limits) with the help of the _____ option.

18. In the **ZOOM** Window option, the area inside the window is _____to completely _____ the current viewport.

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Exercises

Exercise 3

General

Write the text, shown in Figure 7-63, on the screen. Use the text justification that will produce the text as shown in the drawing. Assume a value for text height. Use the **PROPERTIES** palette to change the text, as shown in Figure 7-64.

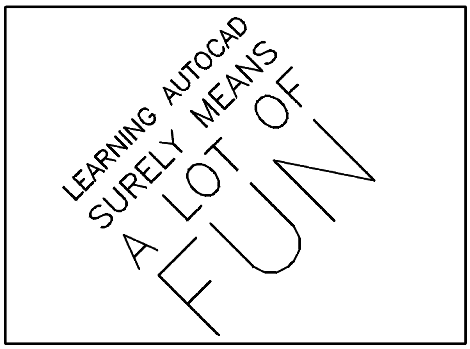

Figure 7-63 *Drawing for Exercise 3*

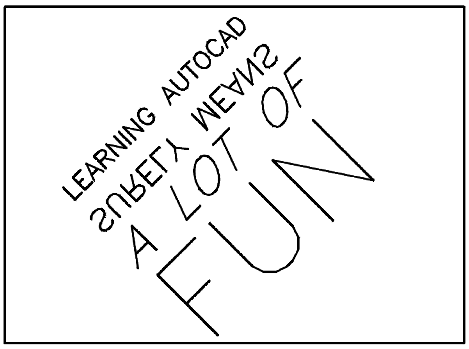

Figure 7-64 *Drawing for Exercise 3 (After changing the text)*

Exercise 4

General

Write the text on the screen, as shown in Figure 7-65. You must first define new text styles using the **STYLE** command with the attributes, as shown in the figure. The text height is 0.25 units.

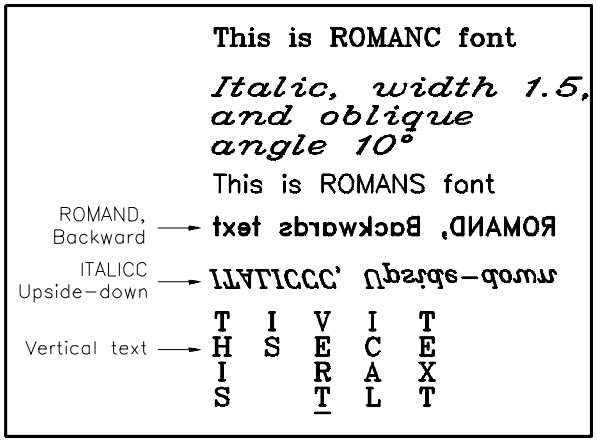

Figure 7-65 *Drawing for Exercise 4*

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Exercise 5

Mechanical

Draw the sketch shown in Figure 7-66 using the draw, edit, text, and display commands. Do not dimension the drawing.

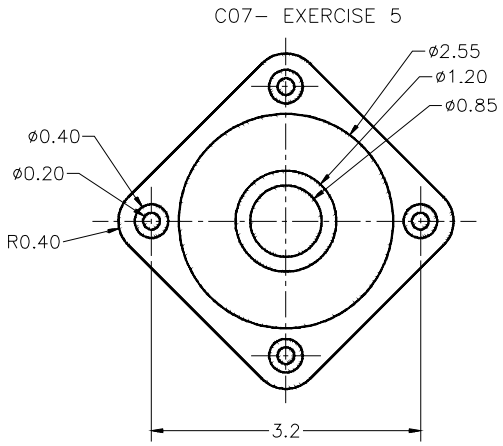

Figure 7-66 *Drawing for Exercise 5*

Exercise 6

Mechanical

Draw the sketch shown in Figure 7-67. Use the **MIRROR** command to duplicate the features that are identical. Also, add the text shown in the figure. Use the display commands to facilitate the process. Do not dimension the drawing.

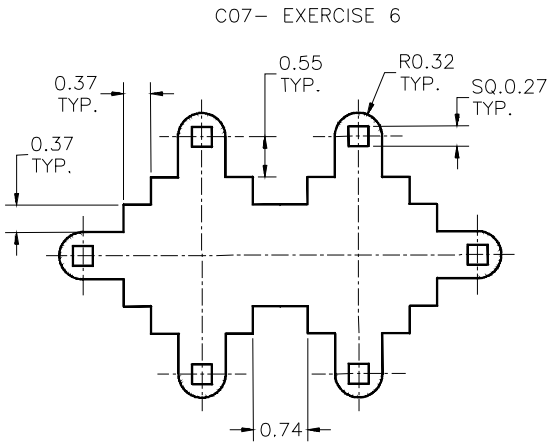

Figure 7-67 *Drawing for Exercise 6*

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Exercise 7

Mechanical

Draw Figure 7-68 and also add the text shown in the figure. Use the display commands to facilitate the process. Do not dimension the drawing.

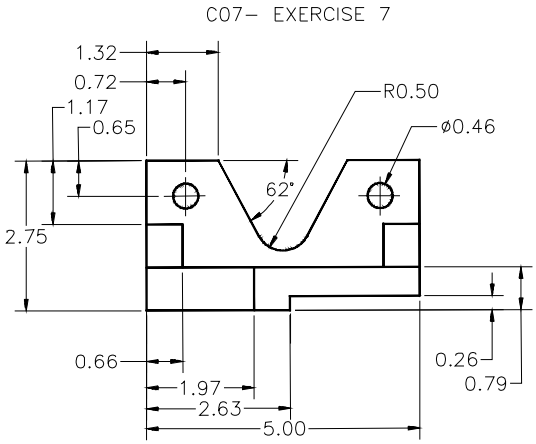

Figure 7-68 *Drawing for Exercise 7*

Exercise 8

Mechanical

Draw the sketch shown in Figure 7-69. Use the **MIRROR** command to duplicate the features that are identical. Use the display commands to facilitate the process. Add the text to the drawing but do not dimension the drawing.

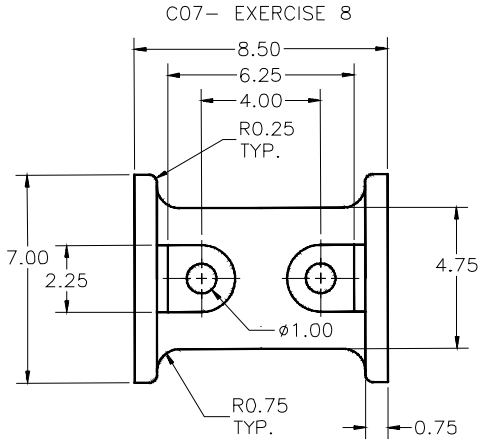

Figure 7-69 *Drawing for Exercise 8*

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Problem Solving Exercise 1

Architectural

Draw the sketch shown in Figure 7-70 using the draw, edit, and display commands. Also add the text to the drawing. Assume the missing dimensions. Do not dimension the drawing.

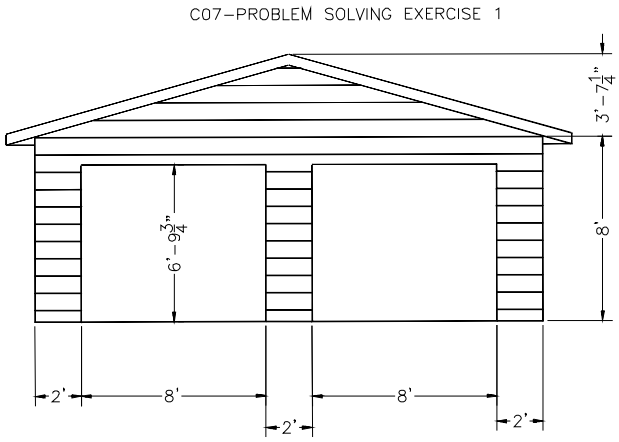

Figure 7-70 *Drawing for Problem Solving Exercise 1*

Problem Solving Exercise 2

Architectural

Draw Figure 7-71 using AutoCAD LT's draw, edit, and display commands. Also, add text to the drawing. Assume the missing dimensions. Do not dimension the drawing.

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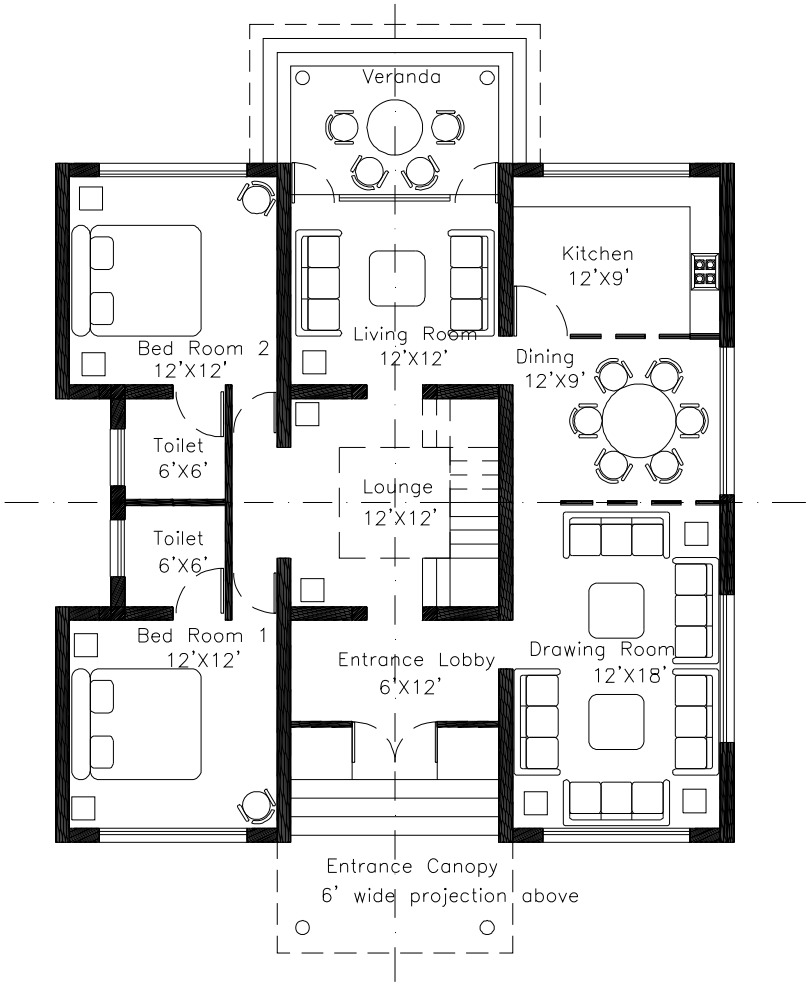

Figure 7-71 *Drawing for Problem Solving Exercise 2*

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Answers to Self-Evaluation Test

1 - F, **2** - T , **3** - T, 4 - T , **5** - view , **6 - TEXT**, **7** - **Fit**, **8** - **TEXTSIZE**, **9** - **PROPERTIES**, **10** - **FONTMAP**

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