# Chapter 1

# Introduction to AutoCAD LT

## Learning Objectives

***After completing this chapter, you will be able to:***

- *Start AutoCAD LT and start a drawing in AutoCAD LT.*
- *Understand the various components of the initial AutoCAD LT screen.*
- *Invoke AutoCAD LT commands from the keyboard, menu, toolbar, shortcut menu, and* ***TOOL PALETTES****.*
- *Understand the functioning of dialog boxes in AutoCAD LT.*
- *Start a new drawing using the* ***QNEW*** *command and the* ***Startup*** *dialog box.*
- *Save the work using various file-saving commands.*
- *Close a drawing.*
- *Open an existing drawing.*
- *Understand the concept of Multiple Document Environment.*
- *Quit AutoCAD LT.*
- *Use the various options of AutoCAD LT's help.*
- *Understand the use of Active Assistance, Learning Assistance, and other interactive help topics.*

# STARTING AutoCAD LT

When you turn on your computer, the operating system (Windows 95, Windows NT, Windows 98, Windows XP, and so on) is automatically loaded. This will display the Windows screen with various application icons. You can start AutoCAD LT by double-clicking on the AutoCAD LT 2006 icon on the desktop of your computer. You can also load AutoCAD LT from the Windows taskbar by choosing the **Start** button at the bottom left corner of the screen (default position) to display the menu. Choose **Programs** to display the program folders. Now, choose the **Autodesk > AutoCAD LT 2006** folder to display the AutoCAD LT programs and then choose **AutoCAD LT 2006** to start AutoCAD LT, see Figure 1-1.

***Figure 1-1*** *Windows screen with taskbar and application icons*

# AutoCAD LT SCREEN COMPONENTS

The various components of the initial AutoCAD LT screen are the drawing area, the command window, menu bar, several toolbars, model and layout tabs, and the status bar (Figure 1-2). A title bar that has the AutoCAD LT symbol and the current drawing name is displayed on top of the screen.

## Drawing Area

The drawing area covers the major portion of the screen. Here you can draw the various objects and use the various commands. To draw the objects you need to define the coordinate points, which can be selected by using your pointing device. The position of the pointing device is

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***Figure 1-2*** *AutoCAD LT screen components*

represented on the screen by the cursor. There is a coordinate system icon at the lower left corner of the drawing area. The window also has the standard Windows buttons such as close, minimize, scroll bar, and so on, available on the top right corner. These buttons have the same functions as for any other standard window.

## Command Window

The command window is present at the bottom of the drawing area and has the Command prompt where you can enter the commands. It also displays the subsequent prompt sequences and the messages. You can change the size of the window by placing the cursor on the top edge (double line bar known as the grab bar) and then dragging it. This way you can increase its size to see all the previous commands you have used. By default, the command window displays only three lines. You can also press the F2 key to display the **AutoCAD LT Text window**, which displays the previous commands and prompts.

**Tip**

*You can hide all the toolbars displayed on the screen by either pressing the CTRL+0 keys or by choosing the* ***Clean Screen*** *option from the* ***View*** *menu. To turn on the display of the toolbars again, press the CTRL+0 keys on the keyboard.*

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## Status Bar

The status bar is displayed at the bottom of the screen, as shown in Figure 1-3. This bar contains some useful information and buttons that will make it easy to change the status of some AutoCAD LT functions. To change the status, you must choose the buttons that toggle between on and off.

***Figure 1-3*** *Default status bar display*

### Coordinate Display

The coordinates information is displayed in the left corner of the status bar. You can select this coordinate button to toggle between on and off. The **COORDS** system variable controls the type of display of the coordinates. If the value of the **COORDS** variable is set to 0, the coordinate display is static, that is, the coordinate values displayed in the status bar change only when you specify a point. If the value of the **COORDS** variable is set to 1 or 2, the coordinate display is dynamic. When the variable is set to 1, AutoCAD LT constantly displays the absolute coordinates of the graphics cursor with respect to the UCS origin. The polar coordinates (length<angle) are displayed if you are in an AutoCAD LT command and the **COORDS** variable is set to 2. You can use the key F6 to turn the coordinate display on or off.

### SNAP

The snap mode allows you to move the cursor in fixed increments. If the **Snap** mode is on, the **SNAP** button is displayed as pressed in the status bar; otherwise, it is not displayed. You can also use the function key F9 as a toggle key to turn snap off or on.

### GRID

The grid lines are used as reference lines to draw objects in AutoCAD LT. If the grid display is on, the **GRID** button is displayed as pressed and the grid lines are displayed on the screen. The function key F7 can be used to turn the grid display on or off.

### ORTHO

The **Ortho** mode allows you to draw lines at right angles only. If this mode is on, the **ORTHO** button is pressed in the status bar. You can use the F8 key to turn ortho on or off.

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### POLAR

If you turn the polar tracking on, the movement of the cursor is restricted along a path based on the angle set as the polar angle settings. Choosing the **POLAR** button in the status bar turns the polar tracking on. You can also use the function key F10. Remember that turning the polar tracking on, automatically turns off the **Ortho** mode.

### OSNAP

When the object snap is on, you can use the running object snaps to snap on to a point. If the object snap is on, the **OSNAP** button is displayed as pressed in the status bar. You can also use the F3 key to turn the object snap on or off. If **OSNAP** is off, the running object snaps are temporarily disabled. The status of **OSNAP** (off or on) does not prevent you from using the immediate mode object snaps.

### DYN*

The **DYN** button is used to turn the **Dynamic Input** on or off. Dynamic input is a new concept in AutoCAD LT 2006 that provides heads-up display for the commands and the prompt sequences. The commands, prompts, and dimensional inputs will now be displayed on the drawing area and you do not need to look at the command prompt all the time. This saves the design time and also increases the efficiency of the user. If the **Dynamic Input** mode is turned on, you will be allowed to enter the commands through the pointer input, enter the numerical values through the dimensional input, and select options through the dynamic prompt in the graphics window. To turn the **Dynamic Input** on or off, you can also use the **F12** function key. Working with the dynamic mode is discussed later in detail in this chapter.

### LWT

Choosing this button in the status bar allows you to turn on or off the display of lineweights in the drawing. If the **LWT** button is not pressed, the display of Lineweight is turned off.

### MODEL

The **MODEL** button is displayed in the status bar when you are working in the model space to create drawings. You can choose this button to shift to the layouts (paper space) where you can create drawing views. Once you switch to layouts, this button is replaced by the **PAPER** button. You can choose the **PAPER** button to shift back to the model space.

**Note**

*All the buttons in the status bar are discussed in detail in Chapter 4, Working With Drawing Aids.*

*The menu bar and toolbar are discussed in the following section. The model and layout tabs are discussed in Chapter 11, Model Space Viewports, Paper Space Viewports, and Layouts.*

## Status Bar Tray Options

The status bar tray options are displayed at the lower-right corner of the screen. These options are used to access the frequently used commands in AutoCAD LT.

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## Communication Center

The **Communication Center** displays a message and an alert whenever Autodesk provides the latest information regarding software updates and their other products. You can configure the settings of the **Communication Center** by clicking on the **Communication Center** icon. The **Communication Center** dialog box is displayed. Choose the **Settings** button to display the **Configuration Settings** dialog box. The **Please Select Country** drop-down list available in the **Country** area allows you to select the name of your country. The **Check for New Content** area allows the **Communication Center** to update you daily, weekly, monthly, or on demand on the latest information provided by Autodesk.

## Toolbar/Window Positions Unlocked*

The **Toolbar/Window Positions Unlocked** icon is used to lock and unlock the positions of toolbars and windows. When you click on this icon, a shortcut menu is displayed. Choosing the **Floating toolbar** option allows you to lock the current position of the floating toolbars. A checkmark will be displayed in the shortcut menu on the type of toolbars that are currently locked. Choosing the **Docked Toolbars** option from the shortcut menu allows you to lock the current position of all the docked toolbars. Similarly, you can lock or unlock the position of floating and docked windows, such as the **PROPERTIES** window or the **TOOL PALETTES**. If you move the cursor on the **All** option, a cascading menu is displayed that provides the option to lock and unlock all the toolbars and windows.

**Note**

*LOCKUI system variable is responsible for the locking and unlocking of the toolbars and windows. The following are the values of system variable :*

*Lockui<0> No toolbar or window locked*
*Lockui<1> Locks all docked toolbars*
*Lockui<2> Locks all docked windows*
*Lockui<4> Locks all floating toolbars*
*Lockui<8> Locks all floating windows*

## Plot/Publish Details Report Available

This icon is displayed when some plotting or a publishing activity was performed in the background. When you click on this icon, the **Plot and Publish Details** dialog box will be displayed that provides the details about the plotting or the publishing activity. You can copy this report to the Clipboard by choosing the **Copy to Clipboard** button from the dialog box.

## Manage Xrefs

The **Manage Xrefs** icon is displayed whenever an external reference drawing is attached to the selected drawing. This icon displays a message and an alert whenever the Xreffed drawing needs to be reloaded. To know about the detailed information regarding the status of each xref in the drawing and the relation between the various Xrefs, click on the **Manage Xrefs** icon; the **XRef Manager** dialog box will be displayed. The Xrefs are discussed in detail in Chapter 16, Understanding External References.

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### Validate Digital Signatures

The **Validate Digital Signatures** icon is displayed whenever AutoCAD LT drawing has a valid digital signature. You can validate the digital signature by clicking on this icon.

# INVOKING COMMANDS IN AutoCAD LT

When you start AutoCAD LT and you are in the drawing area, you need to invoke AutoCAD LT commands to perform any operation. For example, if you want to draw a line, first you have to invoke the **LINE** command, and then you define the start point and endpoint of the line. Similarly, if you want to erase objects, you must enter the **ERASE** command, and then select the objects for erasing. AutoCAD LT has provided the following methods to invoke commands:

| | | |
|---|---|---|
| **Keyboard** | **Menu** | **Toolbar** |
| **Shortcut menu** | **TOOL PALETTES** | |

## Keyboard

You can invoke any AutoCAD LT command from the keyboard by typing the command name and then pressing the **ENTER** Key. If the **Dynamic Input** is on and the cursor is in the drawing area, by default, the command will be entered through the **Pointer Input** box. The **Pointer Input** box is a small box displayed on the right of the cursor, as shown in Figure 1-4. However, if the cursor is currently placed on any toolbar or menu bar, or if the **Dynamic Input** is turned off, the command will be entered through the Command prompt. Before you enter a command, the Command prompt is displayed as the last line in the command window area. If the Command prompt is not displayed, you must cancel the existing command by pressing the ESC (Escape) key. The following example shows how to invoke the **LINE** command from the keyboard:

***Figure 1-4*** *The* ***Pointer Input*** *box displayed when the* ***Dynamic Mode*** *is on*

Command: **LINE** or **L** Enter (L is command alias)

## Menu

You can also select commands from the menu. The menu bar that displays the menu bar titles is at the top of the screen. As you move the cursor over the menu bar, different titles are highlighted. You can choose the desired item by pressing the pick button of your pointing device. Once the item is selected, the corresponding menu is displayed directly under the title. You can invoke a command from the menu by pressing the pick button of your pointing device. Some of the menu items in the menu display an arrow on the right side, which indicates that the menu item has a cascading menu. The cascading menu provides various options to execute the same AutoCAD LT command. You can display the cascading menu by choosing the menu item or by just moving the arrow pointer to the right of that item. You can then choose any item in the cascading menu by highlighting the item or command and pressing the pick button of your pointing device. For example, if you want to draw an ellipse using the Center option, choose **Draw** from the menu

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bar, then choose **Ellipse** from the **Draw** menu, and finally choose **Center** from the cascading menu, as shown in Figure 1-5. In this text, this command selection sequence will be referenced as **Draw** > **Ellipse** > **Center**.

**Note**

*The menus and toolbars tend to vary in every release of AutoCAD LT.*

## Toolbar

In Windows, the toolbar is an easy and convenient way to invoke a command. Each toolbar contains a group of buttons representing various AutoCAD LT commands. When you move the cursor over the buttons of a toolbar, the button gets lifted and a three-dimensional (3D) box encloses the button on which the cursor is resting. The tooltip (name of the button) is also displayed below the button. Once you locate the desired button, the command associated with that button can be invoked by choosing the button. For example, you can invoke the **LINE** command by choosing the **Line** button from the **Draw** toolbar, see Figure 1-6.

*Figure 1-5 Invoking the **ELLIPSE** command from the **Draw** menu*

***Figure 1-6** The **Draw** toolbar*

Some of the buttons in a toolbar have a small triangular arrow at the lower right corner. This indicates that the button has a flyout attached to it. If you hold the button down, the flyout is displayed. The flyout contains the various options for the command. When you choose a command from the toolbar, the command prompts are displayed in the command window. By default the **Standard**, **Styles**, **Layers**, **Properties**, **Draw**, and **Modify** toolbars are displayed on the screen and are docked to the top and the left side edges of the drawing area.

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## Displaying Toolbars

Most of the toolbars that are frequently used are available by default when you start new session of AutoCAD LT. If any additional toolbar is required, right-click on any toolbar to display a shortcut menu and choose the desired toolbar.

## Moving and Resizing Toolbars

The toolbars can be moved anywhere on the screen by placing the cursor on the title bar area and then dragging it to the desired location. You must hold the pick button down while dragging. While moving the toolbars you can dock them to the top or sides of the screen by dropping them in the docking area. You may also prevent docking by holding the CTRL key when moving the toolbar to a desired location. You can also change the size of the toolbars by placing the cursor anywhere on the border of the toolbar where it takes the shape of a double arrow (Figure 1-7), and then pulling it in the desired direction (Figure 1-8). You can also customize toolbars to meet your requirements, which is discussed in the Customizing portion of this textbook.

***Figure 1-7*** *Reshaping the* ***Draw*** *toolbar*

***Figure 1-8*** *The* ***Draw*** *toolbar reshaped*

## Shortcut Menu

AutoCAD LT has provided shortcut menus as an easy and convenient way of invoking commands. These menus are context-sensitive, which means that the commands present in them are dependent on the place/object for which they are displayed. This menu is invoked by right-clicking and is displayed at the cursor location. You can right-click anywhere on the drawing area to display the general shortcut menu. It generally contains an option to select the previously invoked command again, apart from the common commands for Windows, as shown in Figure 1-9.

***Figure 1-9*** *Shortcut menu with no active command*

If you right-click in the drawing area while a command is active, the shortcut menu that is displayed contains the options of that particular command. Figure 1-10 shows the shortcut menu when the **POLYLINE** command is active.

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If you right-click on the layout tabs, the shortcut menu displayed contains the options for the layouts (Figure 1-11).

***Figure 1-10*** *Shortcut menu with the* ***POLYLINE*** *command active*

***Figure 1-11*** *Shortcut menu for the Layout tab*

You can also right-click on the command window to display the shortcut menu. This menu displays the six most recently used commands and some of the window options like Copy and Paste (Figure 1-12). The commands and their prompt entries are displayed in the History window (previous command lines not visible) and can be selected, copied, and pasted in the command line using the shortcut menu. As you press the up arrow key, the previously entered commands are displayed in the command window. Once the desired command is displayed at the Command prompt you can execute the command by simply pressing the ENTER key. You can also copy and edit any previously invoked command by locating it in the History window and then selecting the lines. Right-click in the command window to display the shortcut menu (Figure 1-12); select copy, and then paste the selected lines in the command line. After the lines are pasted, you can edit them.

You can right-click on the coordinate display area of the status bar to display the shortcut menu. This menu contains the options to modify the display of coordinates, as shown in Figure 1-13. You can also right-click on any of the toolbars to display the shortcut menu from where you can choose any toolbar to be displayed.

***Figure 1-12*** *Command line window shortcut menu*

***Figure 1-13*** *Status bar shortcut menu*

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**Tip**

*A shortcut menu is available for any situation while working in AutoCAD LT. You should try to make use of it frequently by right-clicking at various positions.*

## TOOL PALETTES

AutoCAD LT has provided **TOOL PALETTES** (Figure 1-14) as an easy and convenient way of placing and sharing hatch patterns and blocks in the current drawing. By default, AutoCAD LT displays the Tool Palettes as a window on the right of the drawing area. Also, the **TOOL PALETTES** window can be turned on or off by either choosing the **Tool Palettes** button available on the **Standard** toolbar or by pressing the CTRL+3 keys. You can resize the **TOOL PALETTES** using the resizing cursor that is displayed when you place the cursor on the top or bottom extremity of the **TOOL PALETTES.** The **TOOL PALETTES** are discussed in detail in Chapter 13, Hatching Drawings.

***Figure 1-14*** *The* ***TOOL PALETTES*** *window*

## AutoCAD LT DIALOG BOXES

There are certain commands which when invoked display a dialog box. A dialog box is a convenient method of user interface. In the menus, the menu item with the ellipses [...] displays the dialog box when you choose that item. For example, **Options** in the **Tools** menu displays the **Options** dialog box. Any dialog box contains a number of parts like the dialog label, radio buttons, text or edit boxes, check boxes, slider bars, image boxes, and command buttons. These components are also referred to as **TILES**. Some of the components of a dialog box are shown in Figure 1-15.

You can select the desired tile using the pointing device, which is represented by an arrow when a dialog box is invoked. The title bar displays the name of the dialog box. The **tabs** specify the various sections with a group of related options under them. The **check boxes** are toggle buttons for making the particular option available or unavailable. The **drop-down list** displays an item and an arrow on the right which when selected displays a list of items to choose from. You can make a selection in the **radio buttons.** Only one can be selected at a time. The **image box** displays the preview image of the item selected. The **text box** is an area where you can enter a text like a file name. It is also called an **edit box** because you can make any change to the text entered. In some dialog boxes there is the **[...] button**, which displays another related dialog box. There are certain **command buttons** (OK, Cancel, Help) at the bottom of the dialog box. The name implies their functions. The button with a dark border is the default button. The dialog box has a **Help** button for getting help on the various features of the dialog box. There is also a **question mark** (?) button near the top right corner of the dialog box meant for feature-specific help. If you want help on a particular feature of a dialog box, select the ? button. The ? gets attached with the cursor. You can then select the feature to display its help.

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***Figure 1-15*** *Components of a dialog box*

# STARTING A NEW DRAWING

| | |
|---|---|
| **Toolbar:** | Standard toolbar > QNew |
| **Menu:** | File > New |
| **Command:** | NEW or QNEW |

You can open a new drawing using the **QNEW** command. When you invoke the **QNEW** command, by default AutoCAD LT will display the **Select template** dialog box, as shown in Figure 1-16. This dialog box displays a list of the default templates available in AutoCAD LT 2006. You can select the desired template to open a new drawing, which will use the settings of the selected template.

By default, when you invoke the **QNEW** command, AutoCAD LT displays the **Select template** dialog box. You can also open a new drawing using the **Use a Wizard** and **Start from Scratch** options available in the **Create New Drawing** dialog box. By default, the display of this dialog box is turned off. To turn on the display of the **Create New Drawing** dialog box, choose **Tools > Options** from the menu bar. The **Options** dialog box is invoked; choose the **System** tab. Under the **General Options** area, select the **Show Startup dialog box** from the **Startup** drop-down list. Choose **Apply** and then choose the **OK** button. Next, whenever you invoke the **QNEW** command, the **Create New Drawing** dialog box will be displayed, as shown in Figure 1-17. The options provided in this dialog box are discussed next.

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***Figure 1-16*** *The* ***Select template*** *dialog box*

***Figure 1-17*** *The* ***Create New Drawing*** *dialog box*

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## Open a Drawing

By default, this option is not available.

## Start from Scratch

When you choose the **Start from Scratch** button (Figure 1-18), AutoCAD LT provides you with options to start a new drawing that contains the default AutoCAD LT setup for Imperial (*acad.dwt*) or Metric drawings (*acadiso.dwt*). If you select the Imperial default setting, the limits are 12X9, text height is 0.20, and dimensions and linetype scale factors are 1.

***Figure 1-18*** *Options that are displayed to start a new drawing when you choose the* ***Start from Scratch*** *button*

## Use a Template

When you choose the **Use a Template** button in the **Create New Drawing** dialog box, AutoCAD LT displays a list of templates supplied with AutoCAD LT, see Figure 1-19.

The default template file is *acad.dwt* or *acadiso.dwt*, depending on the installation. If you use a template file, the new drawing will have the same settings as specified in the template file. All the drawing parameters of the new drawing such as the units, limits, and other settings are already set according to the template file used. The preview of the template file selected is displayed in the dialog box. You can also define your own template files that are customized to your requirements (see Chapter 29, Template Drawings). To differentiate the template files from the drawing files, the template files have *.dwt* extension whereas the drawing files have *.dwg* extension. Any drawing file can be saved as a template file. You can use the **Browse** button to select other template files. When you choose the **Browse** button, the **Select a template file** dialog box is displayed with the **Template** folder open, displaying all the template files.

## Use a Wizard

The **Use a Wizard** option allows you to set the initial drawing settings before actually starting a new drawing. When you choose the **Use a Wizard** button, AutoCAD LT provides you the option of using the **Quick Setup** or **Advanced Setup**, see Figure 1-20.

In the **Quick Setup**, you can specify the units and the limits of the work area. In the **Advanced Setup**, you can set the units, limits, and the different types of settings for a drawing.

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***Figure 1-19*** *The default templates that are displayed when you choose the* ***Use a Template*** *button*

***Figure 1-20*** *The wizard options that are displayed when you choose the* ***Use a Wizard*** *button*

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## Advanced Setup

This option allows you to preselect various parameters of a new drawing such as the units of linear and angular measurements, type and direction of angular measurements, approximate area desired for the drawing, precision for displaying the units after decimal, and so on.

When you select the **Advanced Setup** wizard option from the **Create New Drawing** dialog box and choose the **OK** button, the **Advanced Setup** dialog box is displayed. The **Units** page is displayed by default, as shown in Figure 1-21.

***Figure 1-21*** *The* ***Units*** *page of the* ***Advanced Setup*** *dialog box*

This page is used to set the units for measurement in the current drawing. You can select the required unit of measurement by selecting its respective radio button. You will notice that the preview image is modified accordingly. The different units of measurement you can choose from are Decimal, Engineering, Architectural, Fractional, and Scientific. You can also set the precision for measurement units by selecting it from the **Precision** drop-down list.

Choose the **Next** button to open the **Angle** page, as shown in Figure 1-22. You will notice that an arrow appears on the left of **Angle** in the **Advanced Setup** dialog box. This suggests that this page is current.

This page is used to set the units for angular measurements and the precision for it. The units for angle measurement can be set by selecting its radio button. The units for angular measurements that you can select are Decimal Degrees, Deg/Min/Sec, Grads, Radians, and Surveyor. The preview of the selected angular unit is displayed on the right of the radio buttons. The precision format changes automatically in the **Precision** drop-down list depending on the angle measuring system selected. You can then select the precision from the drop-down list.

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***Figure 1-22*** *The* ***Angle*** *page of the* ***Advanced Setup*** *dialog box*

The next page is the **Angle Measure** page, as shown in Figure 1-23. This page is used to select the direction of the base angle from which the angles will be measured. You can also set your own direction by selecting the **Other** radio button and then entering the value in its edit box. This edit box is available when you select the **Other** radio button.

***Figure 1-23*** *The* ***Angle Measure*** *page of the* ***Advanced Setup*** *dialog box*

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Choose **Next** to display the **Angle Direction** page (Figure 1-24) to set the orientation for the angle measurement. By default, the angles are positive if measured in a counterclockwise direction. This is because the **Counter-Clockwise** radio button is selected. If you select the **Clockwise** radio button, the angles will be considered positive when measured in the clockwise direction.

***Figure 1-24*** *The* ***Angle Direction*** *page of the* ***Advanced Setup*** *dialog box*

To set the limits of the drawing, choose the **Next** button. The **Area** page is displayed, as shown in Figure 1-25. You can enter the width and length of the drawing area in the respective edit boxes.

**Note**

*Even after you increase the limits of the drawing, the drawing display area is not increased. You need to invoke the* ***ZOOM*** *command and then invoke the* ***All*** *option to increase the drawing display area.*

## Quick Setup

When you select **Quick Setup** and choose the **OK** button, the **Quick Setup** dialog box is displayed. This dialog box has just two pages: **Units** and **Area**. The **Units** page is opened by default, as shown in Figure 1-26. The options in the **Units** page are similar to those in the **Units** page of the **Advanced Setup** dialog box. The only difference is that you cannot set the precision for the units in this dialog box.

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***Figure 1-25*** *The* ***Area*** *page of the* ***Advanced Setup*** *dialog box*

***Figure 1-26*** *The* ***Units*** *page of the* ***QuickSetup*** *dialog box*

Choose **Next** to display the **Area** page, as shown in Figure 1-27. The **Area** page is similar to that of the **Advanced Setup** dialog box where you can set the drawing limits.

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***Figure 1-27*** *The* ***Area*** *page of the* ***QuickSetup*** *dialog box*

**Tip**

*By default, when you open an AutoCAD LT session, a drawing opens automatically. But you can open a new drawing using various options such as* ***Start from Scratch*** *and* ***Wizards*** *before entering into the AutoCAD LT environment using the* ***Startup*** *dialog box. As mentioned earlier, the display of the* ***Startup*** *dialog box is turned off by default. Refer to the section of* ***Starting a New Drawing*** *to know how to turn on the display of this dialog box.*

## SAVING YOUR WORK

| | |
|---|---|
| **Toolbar:** | Standard > Save |
| **Menu:** | File >Save or Save As |
| **Command:** | QSAVE , SAVEAS , SAVE |

In AutoCAD LT or any computer system, you must save your work before you exit from the drawing editor or turn the system off. Also, it is recommended that you save your drawings after regular time intervals. In case of a power failure, an editing error, or other problems, all work saved before the problem started will be retained.

AutoCAD LT has provided the following commands that allow you to save your work on the hard disk of the computer or on the floppy diskette.

**QSAVE** **SAVEAS** **SAVE**

The **QSAVE**, **SAVEAS**, and **SAVE** commands allow you to save your drawing by writing it to a permanent storage device, such as a hard drive, or on a diskette in any removable drive.

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When you choose **Save** from the **File** menu , or choose the **Save** button in the **Standard** toolbar, the **QSAVE** command is invoked. If the current drawing is unnamed and you are saving the drawing for the first time in the present session, the **QSAVE** command will prompt you to enter the file name in the **Save Drawing As** dialog box (Figure 1-28). You can enter the name for the drawing and then choose the **Save** button in the dialog box. Once the drawing is saved and you make some changes to it, you can use the **QSAVE** command to save the drawing with the current name without prompting you to enter a file name. This allows you to do a quick save.

***Figure 1-28*** *The* ***Save Drawing As*** *dialog box*

When you invoke the **SAVEAS** command, the **Save Drawing As** dialog box is displayed, as shown in Figure 1-28. Even if the drawing has been saved with a file name, this command gives you an option to save it with a different file name. In addition to saving the drawing, it sets the name of the current drawing to the file name you specify, which is displayed in the title bar. This command is used when you want to save a previously saved drawing under a different file name. You can also use this command when you make certain changes to a template and want to save the changed template drawing but leave the original template unchanged.

The **SAVE** command is the most rarely used command and can be invoked only from the command line by entering **SAVE** at the Command prompt. It is similar to the **SAVEAS** command and displays the **Save Drawing As** dialog box always. With this command, you can save a previously saved drawing under a different file name, but this command does not set it as the current drawing.

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## Save Drawing As Dialog Box

The **Save Drawing As** dialog box displays the information related to the drawing files on your system. The various components of the dialog box are described next.

### Places list

A column of icons is displayed on the left side of the dialog box. These icons contain the shortcuts to the folders that are frequently used. You can quickly save your drawings in one of these folders. The **History** folder displays the list of most recently saved drawings. You can save your personal drawings in the **My Documents** or the **Favorites** folder. The **FTP** folder displays the list of the various FTP sites that are available for saving the drawing. By default, no FTP sites are shown in the dialog box. To add a FTP site to the dialog box, choose the **Tools** button available on the upper-right corner of the dialog box. When you choose this button, a shortcut menu is displayed. Select **Add/Modify FTP Locations**. The **Desktop** folder displays the list of contents available on the desktop. The **Buzzsaw** icons connect you to their respective pages on the Web. You can add any new folder in this list for easy access by simply dragging this folder on to the **Places list** area and then leaving it. You can rearrange all these folders by dragging them and then placing them at the desired locations. It is also possible to remove the folders when not in frequent use. Right-click on the particular folder and then select **Remove** from the shortcut menu.

### File name edit box

To save your work, enter the name of the drawing in the **File Name** edit box. This can be done by typing the file name or selecting it from the drop-down list. If you select the file name you want from the drop-down list, the name you select automatically appears in the **File name** edit box. If you have already assigned a name to the drawing, the current drawing name is taken as default. If the drawing is unnamed, the default name *Drawing1* is displayed in the **File Name** edit box. You can also choose the down arrow at the right of the edit box to display the names of the previously saved drawings and choose a name here.

### Files of type drop-down list

The **Files of type** drop-down list (Figure 1-29) is used to specify the drawing format, in which you want to save the file. For example, to save the file as AutoCAD LT 2000 drawing file, select **AutoCAD 2000/LT 2000 Drawing (*.dwg)** from the drop-down list.

***Figure 1-29*** *The* ***Files of type*** *drop-down list*

### Save in drop-down list

The current drive and path information is listed in the **Save in** drop-down list. AutoCAD LT will initially save the drawing in the default folder, but if you want to save the drawing in a different folder, you have to specify the path. For example, if you want to save the present drawing under the file name *house* in the *C1* folder, choose the arrow button in the **Save in** drop-down list to

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display the drop-down list and select C:. When you select C: all folders in C drive will be listed in the **File** list box. Double-click on 2006 or select 2006 and choose the **Open** button to display its folders. Again double-click on *C1* or select *C1* and choose the **Open** button to display drawing names in the **File** list box. Select *house* from the list, if it is already listed there, or enter it in the **File name** edit box and then choose the **Save** button. Your drawing (*house*) will be saved in the *C1* folder (*C:\2006\C1\house.dwg*). If you want to save the drawing on the A drive, select A: in the **Save in** drop-down list.

**Tip**

*The file name you enter to save a drawing should match the contents of the drawing. This helps you to remember the drawing details and makes it easier to refer to them later. Also the file name can be 255 characters long and can contain spaces and punctuation marks.*

**Note**

*If you want to save a drawing on the A or B drive, make sure the diskette you are using to save the drawings is formatted.*

## Views List

The **Views** drop-down list has options for the type of listing of files and displaying the preview images (Figure 1-30).

***Figure 1-30*** *The* ***Views*** *list*

### List, Details, Thumbnails, and Preview options

If you choose the **Details** option, it will display detailed information about the files (size, type, date, and time of modification) in the **Files** list box. In the detailed information, if you click on the **Name** label, the files are listed with the names in alphabetical order. If you double-click on the **Name** label, the files will be listed in reverse order. Similarly, if you click on the **Size** label the files are listed according to the size in ascending order. Double-clicking on the **Size** label will list the files descending by order of size. Similarly, you can click on the **Type** label or the **Modified** label to list the files accordingly. If you choose the **List** option, all files present in the current folder will be listed in the **File** list box. If you select the **Preview** option, the list box displays the **Preview image** box wherein the bitmap image of the file chosen is displayed. If cleared, the **Preview image** box is not displayed. If you select the **Thumbnails** option, the list box displays the preview of the all the drawings along with their names displayed at the bottom of the drawing preview.

## Create a New Folder Button

If you choose the **Create New Folder** button, AutoCAD LT creates a new folder under the name **New Folder**. The new folder is displayed in the **File** list box. You can accept the name or change it to your requirement.

## Up one level Button

The **Up one level** button displays the folders that are up by one level. For example, if you are in the *Sample* subfolder of the *AutoCAD LT 2006* folder, then choosing the **Up one level** button will take you to the *AutoCAD LT 2006* folder.

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## Search the Web

It displays the **Browse the Web** dialog box that enables you to access and store AutoCAD LT files on the Internet. You can also use ALT+3 keys to browse the Web when this dialog box is available on the screen.

## Tools List

The **Tools** drop-down list (Figure 1-31) has an option for adding or modifying the FTP sites. These sites can then be browsed from the FTP shortcut in the **Places** list. The **Add Current Folder to Places** and **Add to Favorites** options add the folder displayed in the **Save in** edit box to the **Places** list or to the **Favorites** folder. The **Options** button displays the **Saveas Options** dialog box where you can save the proxy images of custom objects. It has the **DWG Options** and **DXF Options** tabs. The **Security Options** button displays the **Security Options** dialog box, which is used to configure the security options of the drawing.

***Figure 1-31*** *The* ***Tools*** *list*

# AUTOMATIC TIMED SAVE

AutoCAD LT allows you to save your work automatically at specific intervals. To change the time intervals you can enter the time intervals in minutes in the **Minutes between saves** text box in the **File Safety Precautions** area available in the **Options** dialog box (**Open and Save** tab). This dialog box can be invoked from the **Tools** menu. Depending on the power supply, hardware, and type of drawings, you should decide on an appropriate time and assign that time to this variable. AutoCAD LT saves the drawing under the file name *auto.sv$*. The extension of the auto-save file is *.sv$*. You can also change the time interval by using the **SAVETIME** system variable.

# CREATION OF BACKUP FILES

If the drawing file already exists and you use **SAVE** or **SAVEAS** commands to update the current drawing, AutoCAD LT creates a backup file. AutoCAD LT takes the previous copy of the drawing and changes it from a file type *.dwg* to *.bak*, and the updated drawing is saved as a drawing file with the *.dwg* extension. For example, if the name of the drawing is *myproj.dwg*, AutoCAD LT will change it to *myproj.bak* and save the current drawing as *myproj.dwg*.

**Tip**

*Although the automatic save saves your drawing after a certain time interval, you should not completely depend on it because the procedure for converting the sv$ file into a drawing file is cumbersome. Therefore, it is recommended that you save your files regularly using the* ***QSAVE*** *or* ***SAVEAS*** *commands.*

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## Changing Automatic Timed Saved and Backup Files into AutoCAD LT Format

Sometimes, you may need to change the automatic timed saved and backup files into the AutoCAD LT format. To change the backup file into an AutoCAD LT format, open the folder in which you have saved the backup or the automatic timed saved drawing using **My Computer** or **Windows Explorer**. Choose the **Tools > Folder Options** from the menu bar to invoke the **Folder Option**s dialog box. Choose the **View** tab and under the **Advanced settings** area, clear the **Hide file extensions for known file types** text box, if selected. Exit the dialog box. Rename the automatic saved drawing or the backup file with a different name and also change the extension of the drawing from *.sv$* or *.bak* to *.dwg*. After you rename the drawing, you will notice that the icon of the automatic saved drawing or the backup file is replaced by the AutoCAD LT icon. This indicates that the automatic saved drawing or the backup file is changed into an AutoCAD LT drawing.

## Using the Drawing Recovery Manager to Recover Files*

The automatically saved files can also be retrieved using the **Drawing Recovery Manager**. If the automatic save operation is performed in the drawing and the system crashes accidently, the next time you run the AutoCAD LT, the **Drawing Recovery Dialog** box will be displayed, as shown in Figure 1-32.

***Figure 1-32*** *The* ***Drawing Recovery*** *dialog box*

The dialog box informs you that the program was unexpectedly failed and you can open the most suitable among the backup files created by AutoCAD LT. Choose the **Ok** button from the **Drawing Recovery** dialog box; the **Drawing Recovery Manager** is displayed, as shown in Figure 1-33, on the left of the drawing area. The **Backup Files** rollout lists the original files, the backup files, and the automatically saved files. Select the file; its preview will be displayed in the **Preview** rollout. Also, the information corresponding to the selected file is displayed in the **Details** rollout. To open the backup file, double-click on its name in the **Backup Files** rollout. Alternatively, right-click on the file name and choose **Open** from the shortcut menu. It is recommended that you save the backup file at the desired location before you start working on the file.

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**Tip**

*You can open the **Drawing Recovery Manager** again by choosing **File > Drawing Utilities > Drawing Recovery Manager** from the menu bar or by entering **DRAWINGRECOVERY** at the Command prompt.*

***Figure 1-33** The **Drawing Recovery Manager***

## CLOSING A DRAWING

You can use the **CLOSE** command to close the current drawing file without actually quitting AutoCAD LT. If you choose **Close** from the **File** menu or enter **CLOSE** at the Command prompt, the current drawing file is closed. If you have not saved the drawing after making the last changes to it and you invoke the **CLOSE** command, AutoCAD LT displays a dialog box that allows you to save the drawing before closing. This box gives you an option to discard the current drawing or changes made to it. It also gives you an option to cancel the command. After closing the drawing, you are still in AutoCAD LT from where you can open a new or an already saved drawing file. You can also use the close button (**X**) of the drawing window to close the drawing.

**Note**

*You can close a drawing even if a command is active.*

## OPENING AN EXISTING DRAWING

You can open an existing drawing file that has been saved previously. There are three methods that can be used to open a drawing file, the **Select File** dialog box, **Startup** dialog box, and by **Dragging and Dropping**.

### Opening an Existing Drawing Using the Select File Dialog Box

| | |
|---|---|
| **Toolbar:** | Standard > Open |
| **Menu:** | File > Open |
| **Command:** | OPEN |

If you are already in the drawing editor, and you want to open a drawing file you can use the **OPEN** command. The **OPEN** command also displays the **Select File** dialog box, see Figure 1-34. You can select the drawing to open using this dialog box. This dialog box is similar to the standard dialog boxes. You can choose the particular file you want to open from the particular folder. You can change the folder from the **Look in** drop-down list. You can then select the name of the drawing from the list box or you can enter the name of the drawing file you want to open in the **File name** edit box. After selecting the drawing file, you can select the **Open** button to open the file. Here, you can choose *Drawing1* from the list and then choose the **Open** button to open the drawing.

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***Figure 1-34*** *The* ***Select File*** *dialog box*

When you select a file name, its image is displayed in the **Preview** box. If you are not sure about the file name of a particular drawing but know the contents, you can select the file names and look for the particular drawing in the **Preview** box. You can also change the file type by selecting it in the **Files of type** drop-down list. Apart from the *dwg* files, you can open the *dwt* (template) files or the *dxf* files. You have all the standard icons in the **Places** list, which can be used to open the drawing files from different locations. The **Open** button has a drop-down list, as shown in Figure 1-35. You can choose a method for opening the file using this drop-down list. These methods are discussed next.

***Figure 1-35*** *The* ***Open*** *drop-down list*

## Open Read-Only

If you want to view a drawing without altering it, you must select the **Open Read-Only** option from the drop-down list. In other words, read only protects the drawing file from changes. AutoCAD LT does not prevent you from editing the drawing, but if you try to save the opened drawing to the original file name, AutoCAD LT warns you that the drawing file is **write protected**. However, you can save the edited drawing to a file with a different file name using the **SAVEAS** command. This way you can preserve your drawing.

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### Select Initial View

A view is defined as the way you look at an object. The **Select Initial View** option allows you to specify the view you want to load initially when AutoCAD LT loads the drawing. This option will work if the drawing has saved views. This option is generally used while working on a large complicated drawing in which you want to work on a particular portion of the drawing. You can save that particular portion as a view and then select it to open the drawing next time. You can save a desired view by using AutoCAD LT's **VIEW** command (see "**Creating Views**", Chapter 7). If the drawing has no saved views, selecting this option will load the last view. If you select the **Select Initial View** check box and then the **OK** button, AutoCAD LT will display the **Select Initial View** dialog box, see Figure 1-36. You can select the view name from this dialog box, and AutoCAD LT will load the drawing with the selected view displayed.

***Figure 1-36*** *The* ***Select Initial View*** *dialog box*

**Tip**

*Apart from opening a drawing from the* ***Startup*** *dialog box or the* ***Select File*** *dialog box, you can also open a drawing from the* ***File*** *menu, which displays the four most recently opened drawings, by simply choosing the desired file name.*

*It is possible to open an AutoCAD LT 2000 drawing in AutoCAD LT 2006. When you save this drawing it is automatically converted and saved as an AutoCAD LT 2006 drawing file.*

## Opening an Existing Drawing Using the Startup Dialog Box

If you have configured the settings to show the **Startup** dialog box from the **Options** dialog box, the **Startup** dialog box is displayed every time you start a new AutoCAD LT session. The first button in this dialog box is the **Open a Drawing** button. When you choose this button, a list of the most recently opened drawings is displayed for you to select from, see Figure 1-37. The **Browse** button displays the **Select File** dialog box, which allows you to browse another file.

**Note**

*The display of the dialog boxes related to opening and saving drawings is disabled if the* ***FILEDIA*** *system variable is set to 0. The initial value of this variable is 1.*

## Opening an Existing Drawing Using the Drag and Drop Method

You can also open an existing drawing in AutoCAD LT by dragging it from the Window Explorer and dropping it into AutoCAD LT. If you drop the selected drawing in the drawing area, the drawing will be inserted as a block and as a result it cannot be modified. But, if you drag the drawing from the Window Explorer and drop it anywhere other than the drawing area AutoCAD LT opens the selected drawing.

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***Figure 1-37*** *List of recently opened drawings*

## QUITTING AutoCAD LT

You can exit the AutoCAD LT program by using the **EXIT** or **QUIT** commands. Even if you have a command active, you can choose **File > Exit** from the menu bar to quit the AutoCAD LT program. In case the drawing has not been saved, it allows you to save the work first through a dialog box. Note that if you choose **No** in this dialog box, all the changes made in the current list till the last save will be lost. You can also use the close button (**X**) of the main AutoCAD LT window (present in the title bar) to end the AutoCAD LT session.

## DYNAMIC INPUT MODE*

As mentioned earlier, turning the **Dynamic Input** mode on allows you to enter the commands through a pointer input and the dimensions using the dimensional input. When this mode is turned on, all the prompts are available at the tooltip as dynamic prompts and you can select the command options through dynamic prompt. The settings for **Dynamic Input** are done through the **Dynamic Input** tab of **Drafting Settings** dialog box. Invoke the **Drafting Settings** dialog box by selecting the **Drafting Settings** option from the **Tools** drop-down list. Choose the **Dynamic Input** tab in the **Drafting Settings** dialog box, as shown in Figure 1-38. The options available in this tab are discussed next.

### Enable Pointer Input

With the **Enable Pointer Input** check box selected, you can enter the Commands through the pointer input. In Figure 1-39, the **CIRCLE** command is being entered through the pointer input. If this check box is cleared, the **Dynamic Input** will be turned off and the commands will be entered through the Command prompt, similar to the previous releases of AutoCAD LT.

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***Figure 1-38*** *The* ***Dynamic Input*** *tab of the* ***Drafting Settings*** *dialog box*

Choosing the **Settings** button from the **Pointer Input** area displays the **Pointer Input Settings** dialog box, as shown in Figure 1-40. The radio buttons available in the **Format** area of this dialog box are used to set the default settings for specifying the other points after specifying the first point. By default, the **Polar format** and **Relative coordinates** radio buttons are selected. As a result, the coordinates will be specified in the polar form and with respect to the relative coordinates system. You can select the **Cartesian format** radio button to enter the coordinates in the cartesian form. Likewise, if you select the **Absolute coordinates** radio button, the numerical entries will be measured with respect to the absolute coordinate system.

The **Visibility** area, in the **Pointer Input Settings** dialog box, is used to set the visibility of the coordinates tool

***Figure 1-39*** *Entering a command using the pointer input*

***Figure 1-40*** *The* ***Pointer Input Settings*** *dialog box*

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tips. By default, the **When a command ask for a point** radio button is selected. You can select the other radio buttons to modify this display.

## Enable Dimensional Input where possible

This check box is selected by default. As a result, the dimension input field is displayed in the graphics area showing the preview of that dimension. Figure 1-41 displays the dimension input field. The dotted lines shows the geometric parameters like length, radius or diameter corresponding to that dimension. In Figure 1-42, a line is being drawn using **Pline** command. The two dimensional inputs that are shown are for the length of the line and the angle with the positive direction of X axis.

Using the TAB key, you can toggle between the dimension input fields. As soon as you have specified one dimension and moved to the other dimension the previous dimension will be locked. If the Enable **Dimensional Input where possible** checkbox is cleared, the preview of the dimensions will not be displayed. You can only enter the dimensions at the dimensions in the dimensional input field available below the cursor, as shown in Figure 1-42.

***Figure 1-41*** *Dimensional input and dynamic prompt with* ***Dynamic Input*** *on*

***Figure 1-42*** *Dimensional input and dynamic prompt with* ***Dynamic Input*** *on*

Choose **Settings** button from the **Dimension Input** area to display the **Dimension Input Settings** dialog box, as shown in Figure 1-43.

By default, the **Show 2 dimension input field at a time** radio button is selected. As a result, two dimension input fields will be displayed in the drawing area while stretching a sketched entity. The two input fields will depend on the entity that is being stretched. For example, if you are stretching a line using one of its endpoints, the input field will show the total length of the line and the change in its length. Similarly, while stretching a circle using a grip on its circumference, the input fields will show the total radius and the change in it. You can set the priority to display only one input field or various input fields simultaneously by selecting their respective check boxes.

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***Figure 1-43*** *The* ***Dimension Input Settings*** *dialog box*

**Tip**

*If multiple dimension input fields are available, use the TAB key to switch between the dimension input fields*

## Show command prompting and command input near the crosshairs

If this checkbox is selected, the prompt sequences will be dynamically displayed near the crosshairs. Whenever a blue arrow appears at the pointer input, it suggests that the access options are available. To access these options, press the down arrow key to see the dynamic prompts, listing all option. In the dynamic prompt, you can use the cursor or the down arrow key to jog through the options. A black dot will appear before the option that is currently active. In Figure 1-40, the **Length** option is currently active, press ENTER from the keyboard to confirm the Polyline creation with Length option.

## Drafting Tooltip Appearance

When you choose the **Drafting Tooltip Appearance** button, the **Tooltip Appearance** dialog box is displayed, as shown in Figure 1-44.

This dialog box contains the options to customize the tooltip appearance. The color area contains two buttons that you can choose to change the color of the tooltip in model space or layouts.

The edit box in the **Size** area is used to specify the size of the tooltip. You can also use slider to control the size of tool tip. The preview is displayed in the **Model Preview** area and the **Layout Preview** area as soon as the value is changed in the Size edit box. Likewise, the transparency of the tooltip can be controlled using the edit box or the slider available in the **Transparency** area.

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***Figure 1-44*** *The* ***Tooltip Appearance*** *dialog box*

Selecting the **Override OS settings for all drafting tooltips** radio button in the **Apply to** area ensures that the changes made in the **Tooltip Appearance** dialog box will be applied to all drafting tooltips. If you select the **Use settings only for Dynamic Input tooltips** radio button is selected, the changes will be applied only to the **Dynamic Input** tooltips. For example, if you change any of the parameters using the **Tooltip Appearance** dialog box and select the **Use settings only for Dynamic Input tooltips** radio button, the tooltips for dynamic input will be modified, but for the polar tracking it will consider the original values. On the other hand, if you select the **Override OS settings for all drafting tooltips** radio button, the tooltips displayed for the polar tracking will also be modified, based on the values in the **Tooltip Appearance** dialog box.

## CREATING AND MANAGING WORKSPACES*

A workspace is defined as a customized arrangement the toolbars, menus, and window palettes in the AutoCAD LT environment. Workspaces are required when you need to customize the AutoCAD LT environment for a specific use, in which you need only a certain sets of toolbars and menus. For such requirements, you can create your own workspaces, in which only specified toolbars, menus and palettes will be available. By default, the **AutoCAD LT Default** workspace is current when you start AutoCAD LT. You can select any other predefined workspace from the drop-down list in the **Workspaces** toolbar, see Figure 1-45, or by choosing **Window > Workspaces > Name of the Workspace** from the menu bar.

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## Creating a New Workspace

To create a new workspace, invoke the toolbars and the window palettes that you want to be displayed in the new workspace. Next, select the **Save Current As** option from the drop-down list available in the **Workspaces** toolbar; the **Save Workspace** dialog box will be displayed, as shown in Figure 1-46. Enter the name of the new workspace in the **Name** edit box and choose the **Save** button.

*Figure 1-45 The **Workspaces** toolbar*

*Figure 1-46 The **Save Workspace** dialog box*

The new workspace is now current in the drop-down list in the **Workspaces** toolbar. Also, when you choose **Window > Workspaces** from the menu bar, a check mark is displayed on the left of the current workspace in the cascading menu. Likewise, you can create workspaces as per your requirement and can switch from one workspace to the other by selecting the name from the drop-down list in the **Workspaces** toolbar.

## Modifying the Workspace Settings

AutoCAD LT allows you to modify the workspace settings. To do so, choose the **Workspace Settings** button in the **Workspaces** toolbar; the **Workspace Settings** dialog box will be displayed, as shown in Figure 1-47. All **Workspaces** that are created are listed in the **My Workspace** drop-down list. You can make any of the available workspaces as My Workspace by selecting it in the **My Workspace** drop-down list. You can choose the **My Workspace** button from the **Workspaces** toolbar to change the current workspace to the one that was set as My Workspace in the **Workspace Settings** dialog box. The other options available in this toolbar are discussed next.

*Figure 1-47 The **Workspace Settings** dialog box*

### Menu Display and Order Area

The options in this area are used to set the order of display of workspaces in the drop-down list of the **Workspaces** toolbar or in the **Window > Workspaces** menu. By default, the workspaces are listed in the sequence of their creation. To change the order, select the workspace and choose the **Move Up** or **Move Down button**. You can also add a separator between the workspaces by

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choosing the **AddSeparator** button. A separator is a line that is placed between the two Workspaces in the drop-down list of the **Workspaces** toolbar or in the **Window > Workspaces** menu. The separator will be added above the selected workspace. Figure 1-48 shows seperators added between all workspaces in the **Workspaces** toolbar.

***Figure 1-48*** *The* ***Workspaces*** *drop-down list after adding seperators*

### When Switching Workspaces Area

By default, the **Do not save changes to workspace** radio button is selected in this area. This ensures that while switching the workspaces, the changes made in current workspace will not be saved. If you select the **Automatically save workspace changes** radio button, the changes made in the current workspace will be automatically saved when you switch to other workspace.

## The AutoCAD LT HELP

| | |
|---|---|
| **Toolbar:** | Standard > Help |
| **Menu:** | Help > Help |
| **Command:** | HELP or ? |

***Figure 1-49*** *The* ***Help*** *menu*

You can get online help and documentation on the working of AutoCAD LT 2006 commands from the **Help** menu, see Figure 1-49, or by pressing the F1 key. The various options available in the **Help** menu are discussed next.

## Help

Choosing the **Help** option displays the **AutoCAD LT 2006 Help** dialog box, as shown in Figure 1-50.

You can use this dialog box to access help on different topics and commands. It has five tabs: **Contents**, **Index**, **Search**, **Favorites**, and **Ask me**, which display the corresponding help topics. If you are in the middle of a command and require help regarding it, choosing the **Help** button displays information about that particular command in the dialog box.

### Contents

This tab displays the help topics, which are organized by categories, pertaining to different sections of AutoCAD LT such as the User's Guide, Command Reference, and so on. To select a category, double-click on the corresponding book icon or choose the plus sign on the left. The icon becomes an open book with a minus (-) sign and a list of headings associated with that category is displayed. Use the plus sign (+) to further open the headings until you reach the help topic, which has a question mark (?) displayed with it. Choose the topic to display information about the selected topic or command in the window present on the right side of the dialog box.

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***Figure 1-50 AutoCAD LT 2006 Help*** *dialog box (**Contents** tab)*

## Index

This tab displays the complete index (search keywords) in an alphabetical order. To display information about an item or a command, type the item (word) or command name in the edit box. With each letter entered, the listing keeps on changing in the list area, displaying the possible topics. When you enter the word and if AutoCAD LT finds that word, it is automatically highlighted in the list area. Choose the **Display** button to display information about it.

## Search

This tab creates a word list based on all the keywords present in the online help files. When you type a word you are looking for and then choose the **List Topics** button, a list of matching words is displayed in a window below to narrow down your search. This search is dependent on the option you have selected at the bottom of the dialog box, where you can search the previous results and also match words similar to those you searched. Use the scroll bar to scroll through the list, select the desired topic, and then choose the **Display** button to display its help.

## Ask Me

When you choose this tab, you will be prompted to enter a query in the edit box and then press ENTER. A list of topics related to the question follow. It also shows the book (category) from which it has been selected.

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## Info Palette

Choosing this option displays **INFO PALETTE**, which gives you an access to context-sensitive help. The **INFO PALETTE** can be started by entering **ASSIST** at the Command prompt. Whenever you invoke a command, the **INFO PALETTE** displays quick help related to that command.

## New Features Workshop

This option gives you an interactive list of all the new features in AutoCAD LT 2006. You can choose this option from the **Help** menu, which displays the **New Features Workshop** window with a list of various topics (Figure 1-51). When you choose a topic, a description of the feature improvement is displayed.

***Figure 1-51*** *The* ***New Features Workshop*** *window*

## Additional Resources

This utility connects you to the **Product Support, Training**, **Autodesk User Group International** Web pages and Web sites through the Microsoft Internet Explorer.

## About

This option gives you information about the Release, Serial Number, Licensed To, and also the legal description about AutoCAD LT.

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# ADDITIONAL HELP RESOURCES

1. You can get help for a command while working by pressing the F1 key. The **Help** dialog box containing information about the command is displayed. You can exit the dialog box and continue with the command.

2. You can get help about a dialog box by choosing the **Help** button in that dialog box.

3. Some of the dialog boxes have a **question mark** (**?**) button at the top right corner just adjacent to the **close** button. When you choose this button, the ? gets attached to the cursor. You can then drop it on any item in the dialog box to display information about that particular item.

4. Autodesk has provided several resources that you can use to get assistance with your AutoCAD LT questions. The following is a list of some of the resources:

   a. Autodesk Web site *http://www.autodesk.com*
   b. AutoCAD LT Technical Assistance Web site *http://www.autodesk.com/support*
   c. AutoCAD LT Discussion Groups Web site *http://discussion.autodesk.com/index.jspa*

5. You can also get help by contacting the author, Sham Tickoo, at *stickoo@calumet.purdue.edu*.

6. You can download AutoCAD LT drawings, programs, and special topics by accessing the author's Web site at *http://technology.calumet.purdue.edu/met/tickoo/students/students.htm*.

## Self-Evaluation Test

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

1. You can press the F3 key to display the **AutoCAD LT Text Window**, which displays the previous commands and prompts. (T/F)

2. If the value of the COORDS variable is set to 1, the coordinate display is static, that is, the coordinate values displayed in the status bar change only when you specify a previous point. (T/F)

3. In AutoCAD LT, by default, command is entered at the Command prompt . (T/F)

4. If the current drawing is unnamed and you are saving the drawing for the first time in the present session, the **QSAVE** command will prompt you to enter the file name in the **Save Drawing As** dialog box. (T/F)

5. A _____ is displayed on top of the AutoCAD LT screen that has the AutoCAD LT symbol and the current drawing name.

6. The _____displays a message and an alert whenever Autodesk provides the latest information regarding software updates and their other products.

7. If you want to work on a drawing without altering the original, you must select the _____ option from the **Open** drop-down list in the **Select File** dialog box.

8. The _____ window automatically appears whenever you invoke any command in AutoCAD LT

9. You can use the _____ command to close the current drawing file without actually quitting AutoCAD LT.

10. The _____system variable can be used to change the time interval for automatic save.

## Review Questions

**Answer the following questions.**

1. The shortcut menu invoked by right-clicking in the command window displays the six most recently used commands and some of the window options such as **Copy**, **Paste**, and so on. (T/F)

2. It is possible to open an AutoCAD LT 2002 drawing in AutoCAD LT 2006. (T/F)

3. The file name you enter to save a drawing in the **Save Drawing As** dialog box file name can be 255 characters long but cannot contain spaces and punctuation marks. (T/F)

4. You can close a drawing in AutoCAD LT 2006 even if a command is active. (T/F)

5. Which one of the following combination of keys is pressed to hide all the toolbars displayed on the screen?

   (a) CTRL+3 (b) CTRL+0
   (c) CTRL+5 (d) CTRL+2

6. Which one of the following combination of keys is pressed to turn on or off the display of the **TOOL PALETTES** window?

   (a) CTRL+3 (b) CTRL+0
   (c) CTRL+5 (d) CTRL+2

7. Which of the following commands is used to exit from the AutoCAD LT program?

   (a) **QUIT** (b) **END**
   (c) **CLOSE** (d) None

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8. Which of the following options available in the **Startup** dialog box is used to set the initial drawing settings before actually starting a new drawing?

   (a) **Start from Scratch** (b) **Use a Template**
   (c) **Use a Wizard** (d) None

9. When you choose **Save** from the **File** menu or choose the **Save** button in the **Standard** toolbar, which of the following commands is invoked?

   (a) **SAVE** (b) **LSAVE**
   (c) **QSAVE** (d) **SAVEAS**

10. AutoCAD LT has provided _____ as an easy and convenient way of placing and sharing hatch patterns and blocks in the current drawing.

11. By default the angles are positive if measured in a _____ direction.

12. You can change the size of the toolbars by placing the cursor anywhere on the _____ of the toolbar where it takes the shape of a double-sided arrow.

13. To differentiate the template files from the drawing files, the template files have _____ extension whereas the drawing files have _____ extension.

14. You can also use _____ and _____ instead of dragging and dropping the objects from one drawing to another while multiple drawings are opened.

15. The _____ tab of the **AutoCAD LT 2006 Help: User Documentation** dialog box displays the help topics that are organized by categories pertaining to different sections of AutoCAD LT.

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**Answers to Self-Evaluation Test**

**1** - F, **2** - F, **3** - F, **4** - T, **5** - **Archive**, **6** - **Open Read-Only**, **7** - **Communication Center**, **8** - Partial Open, **9** - **CLOSE**, **10** - **SAVETIME**